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

Common to Lathe System / Machining Center System

OPERATOR'S MANUAL

B-64304EN/02

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.

⚠ **WARNING**

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.

⚠ **CAUTION**

The liquid-crystal display is manufactured with very precise fabrication technology. Some pixels may not be turned on or may remain on. This phenomenon is a common attribute of LCDs and is not a defect.

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

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

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 T/M series) 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 T/M series) 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

III. OPERATION

IV. MAINTENANCE

APPENDIX

I. GENERAL

1 GENERAL

This manual consists of the following parts:

About this manual

I. GENERAL
Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, explanations, and limitations.

III. OPERATION
Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.

IV. MAINTENANCE
Describes procedures for daily maintenance and replacing batteries.

APPENDIX
Lists parameters, valid data ranges, and alarms.

NOTE
1 This manual describes the functions common to both the lathe system and the machining center system. For the functions specific to the lathe system or machining center system, refer to the Operator's Manual (For Lathe System) (B-64304EN-1) or the Operator's Manual (For Machining Center System) (B-64304EN-2).
2 Some functions described in this manual may not be applied to some products. For detail, refer to the Descriptions manual (B-64302EN).
3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-64310EN).
Parameters are used to set functions and operating conditions of a CNC machine tool, and frequently-used values in advance. Usually, the machine tool builder factory-sets parameters so that the user can use the machine tool easily.
4 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

This manual describes the following models that are 'Nano CNC'.
'Nano CNC system' which realizes high precision machining can be constructed by combining these models and high speed, high precision servo controls.
In the text, the abbreviations may be used in addition to Model name indicated below.

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 explanatory 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 Some functions described in this manual may not be applied to some products. For details, refer to the Descriptions (B-64302EN).
>
> 3 For the 0*i*-D / 0*i* Mate-D, parameters need to be set to enable or disable some basic functions.
> For these parameters, refer to Section 4.51, " PARAMETERS OF 0*i*-D / 0*i* Mate-D BASIC FUNCTIONS" in the PARAMETER MANUAL (B-64310EN).

Special symbols

This manual uses the following symbols:

- **M**

Indicates the description that are valid only for the M series.
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.
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 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 in PROGRAMMING.).

- **;**

Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.

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

The following table lists the manuals related to Series 0*i* -D and 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	

Manual name	Specification number	
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 Executor PROGRAMMING MANUAL	B-64303EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64393EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-64403EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64414EN	
DeviceNet Board CONNECTION MANUAL	B-64443EN	
FL-net Board CONNECTION MANUAL	B-64453EN	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64303EN-4	
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	
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

Manual name	Specification number
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.

1.1 NOTES ON READING THIS MANUAL

⚠ **CAUTION**

1. The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.
2. In the header field of each page of this manual, a chapter title is indicated so that the reader can reference necessary information easily.
   By finding a desired title first, the reader can reference necessary parts only.
3. This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
   If a particular combination of operations is not described, it should not be attempted.

1.2 NOTES ON VARIOUS KINDS OF DATA

⚠ **CAUTION**

Machining programs, parameters, offset data, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.

II. PROGRAMMING

1 GENERAL

Chapter 1, "GENERAL", consists of the following sections:

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION

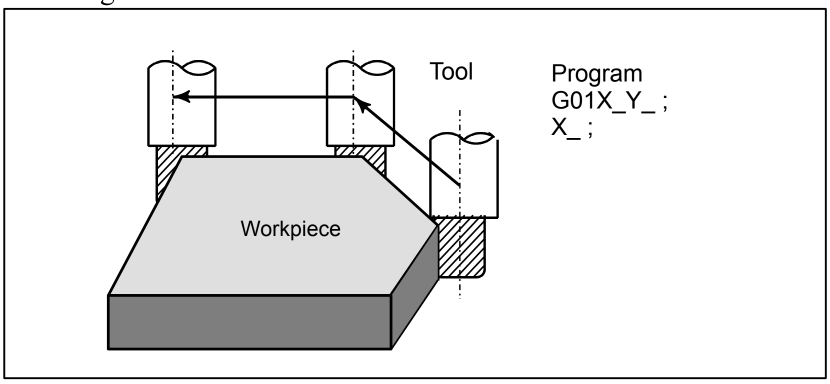
The tool moves along straight lines and arcs constituting the workpiece parts figure (See II-4).

Explanation

The function of moving the tool along straight lines and arcs is called the interpolation.

- Tool movement along a straight line

- For milling machining



- For lathe cutting

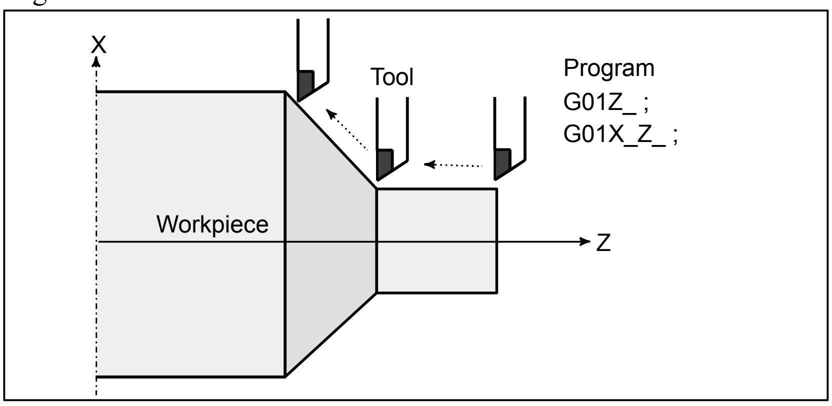
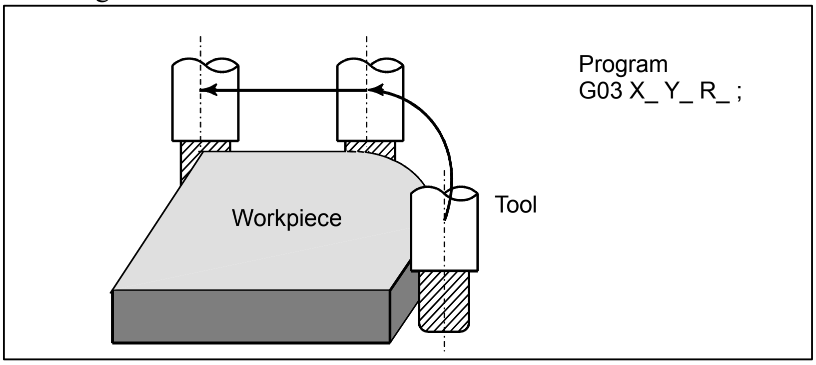


Fig. 1.1 (a) Tool movement along a straight line

- Tool movement along an arc

- For milling machining



- For lathe cutting

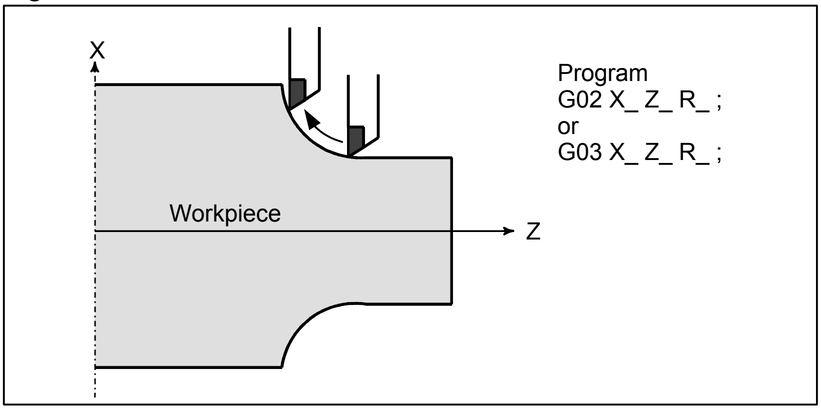


Fig. 1.1 (b) Tool movement along an arc

The term interpolation refers to an operation in which the tool moves along a straight line or arc in the way described above.

Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

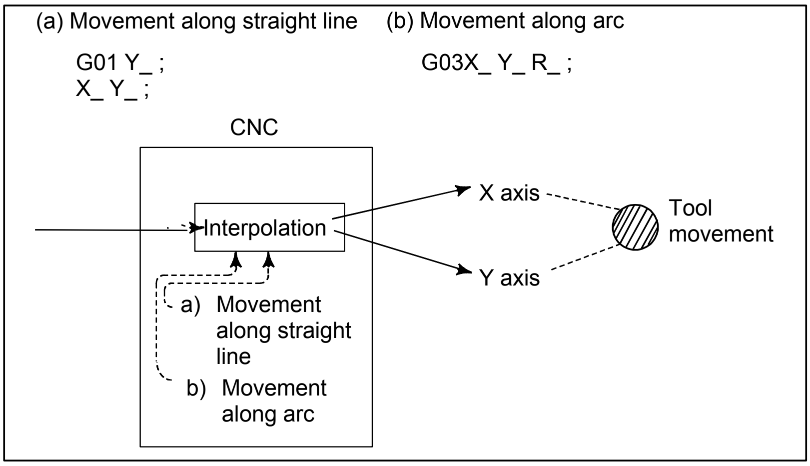


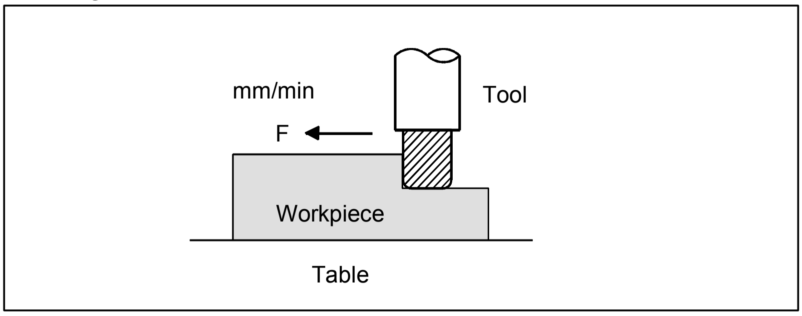
Fig. 1.1 (c) Interpolation function

> **NOTE**
> Some machines move workpieces instead of tools but this manual assumes that tools are moved against workpieces.

1.2 FEED-FEED FUNCTION

Movement of the tool at a specified speed for cutting a workpiece is called the feed.

- For milling machining



- For lathe cutting

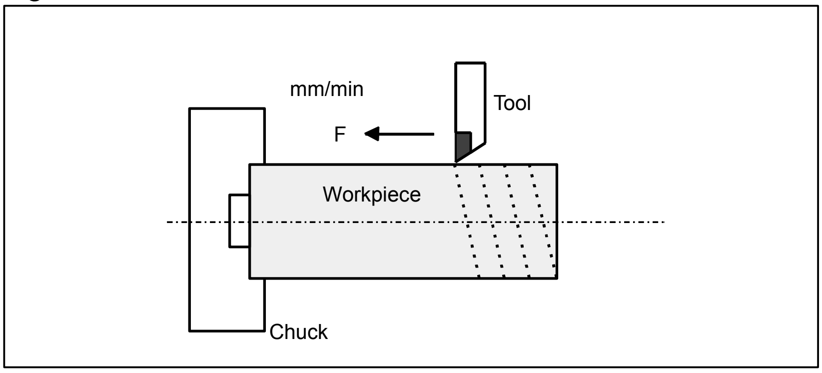


Fig. 1.2 (a) Feed function

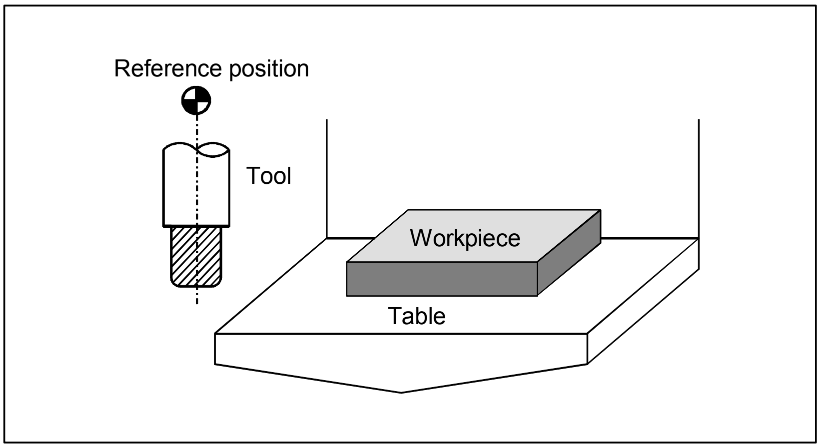
Feedrates can be specified by using actual numerics. For example, to feed the tool at a rate of 150 mm/min, specify the following in the program:
F150.0
The function of deciding the feed rate is called the feed function (See II-5).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine-specific Position)

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

- For milling machining



- For lathe cutting

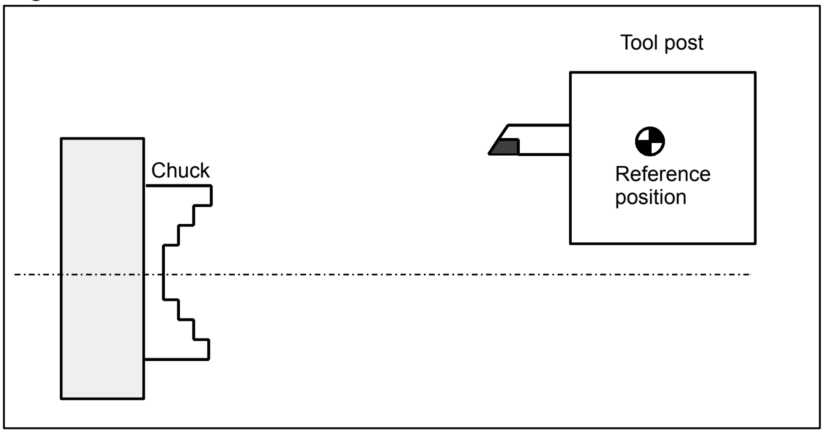


Fig. 1.3.1 (a) Reference position

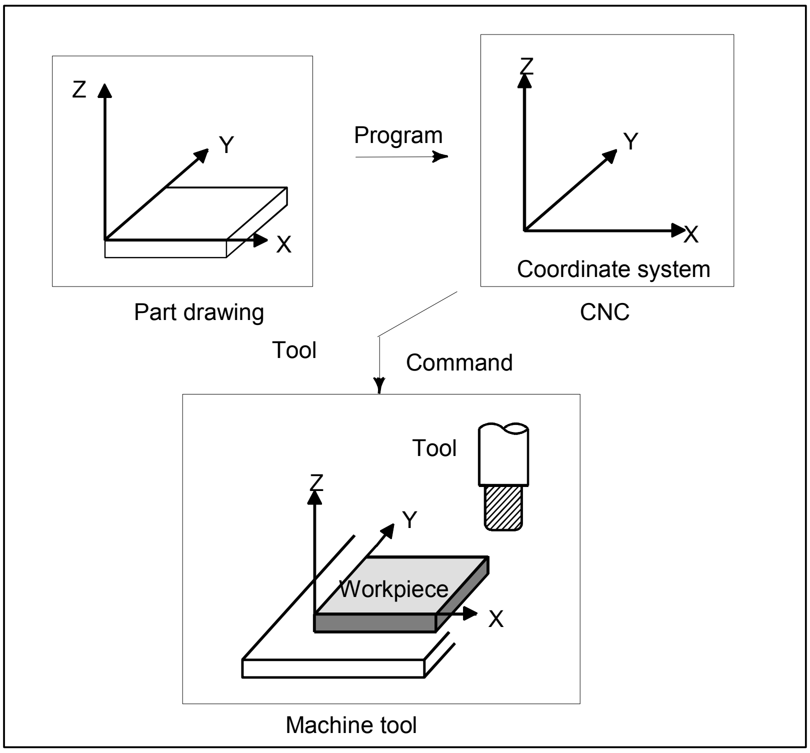
Explanation

The tool can be moved to the reference position in two ways:

1. Manual reference position return (See III-3.1)
   Reference position return is performed by manual button operation.
2. Automatic reference position return (See II-6)
   In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System

- For milling machining



- For lathe cutting

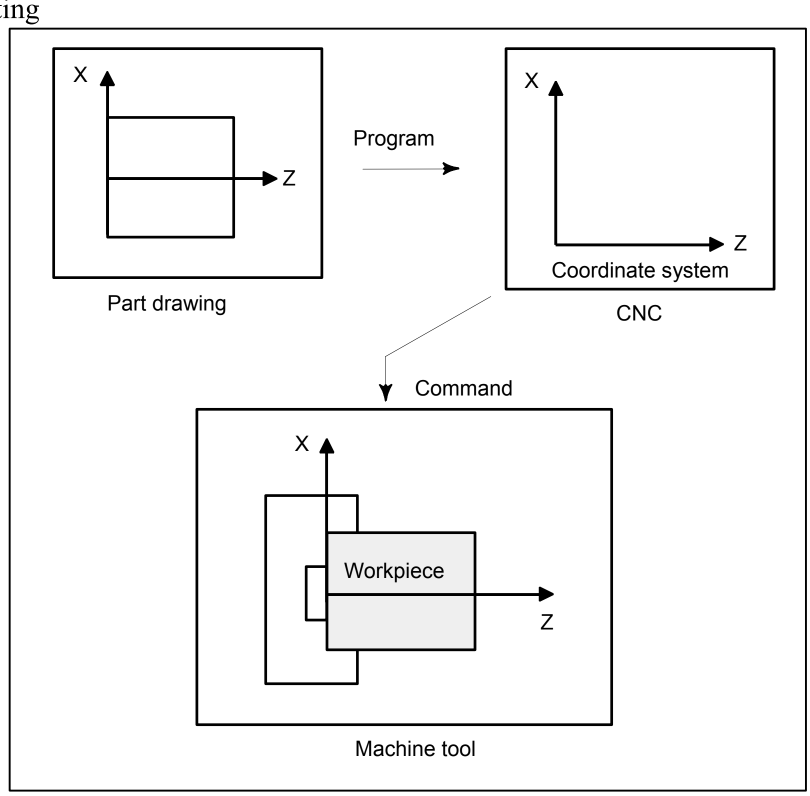


Fig. 1.3.2 (a) Coordinate system

Explanation

- Coordinate system

The following two coordinate systems are specified at different locations: (See II-7)

1 Coordinate system on part drawing
The coordinate system is written on the part drawing. As the program data, the coordinate values on this coordinate system are used.

2. Coordinate system specified by the CNC
The coordinate system is prepared on the actual machine tool table. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

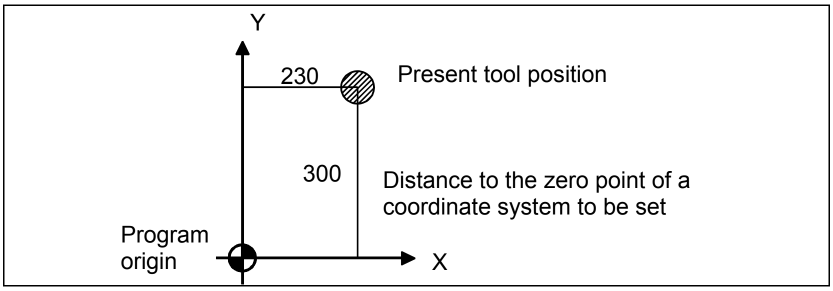
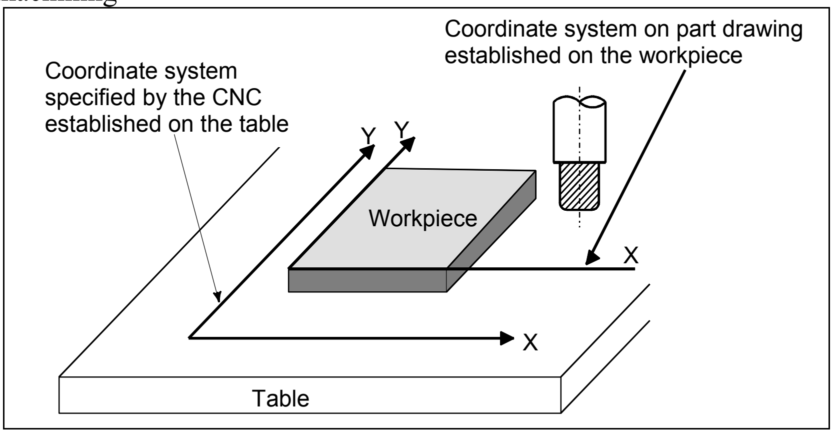


Fig. 1.3.2 (b) Coordinate system specified by the CNC

Concrete programming methods for setting coordinate systems specified by the CNC are explained in II-7, "COORDINATE SYSTEM".
The positional relation between these two coordinate systems is determined when a workpiece is set on the table.

- For milling machining



- For lathe cutting

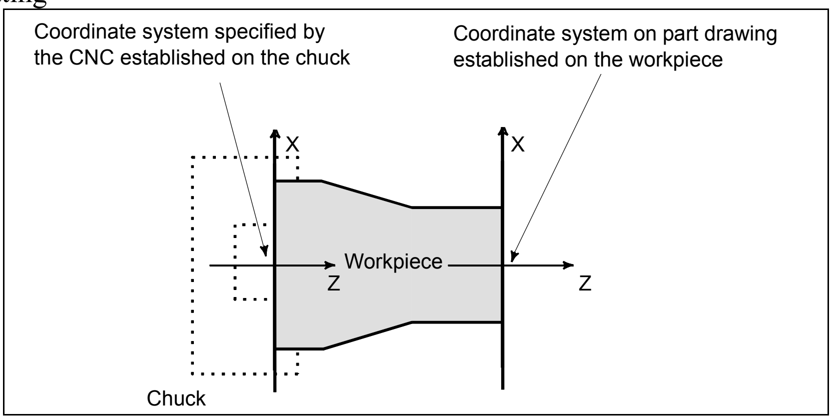


Fig. 1.3.2 (c) Coordinate system specified by CNC and coordinate system on part drawing

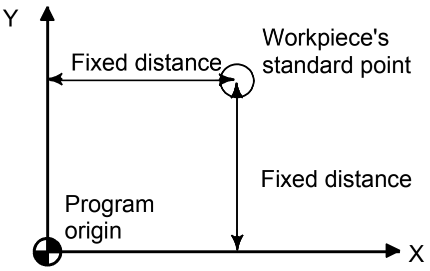
The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.
Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

- Methods of setting the two coordinate systems in the same position

M

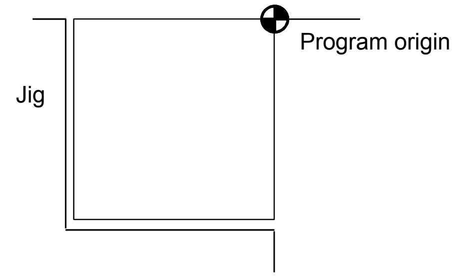
To set the two coordinate systems at the same position, simple methods shall be used according to workpiece shape, the number of machinings.

(1) Using a standard plane and point of the workpiece.



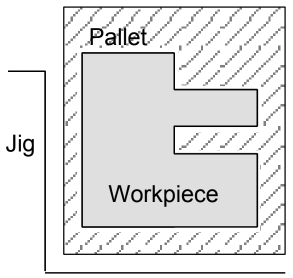
Bring the tool center to the workpiece standard point.
And set the coordinate system specified by CNC at this position.

(2) Mounting a workpiece directly against the jig



Meet the tool center to the reference position. And set the coordinate system specified by CNC at this position. (Jig shall be mounted on the predetermined point from the reference position.)

(3) Mounting a workpiece on a pallet, then mounting the workpiece and pallet on the jig

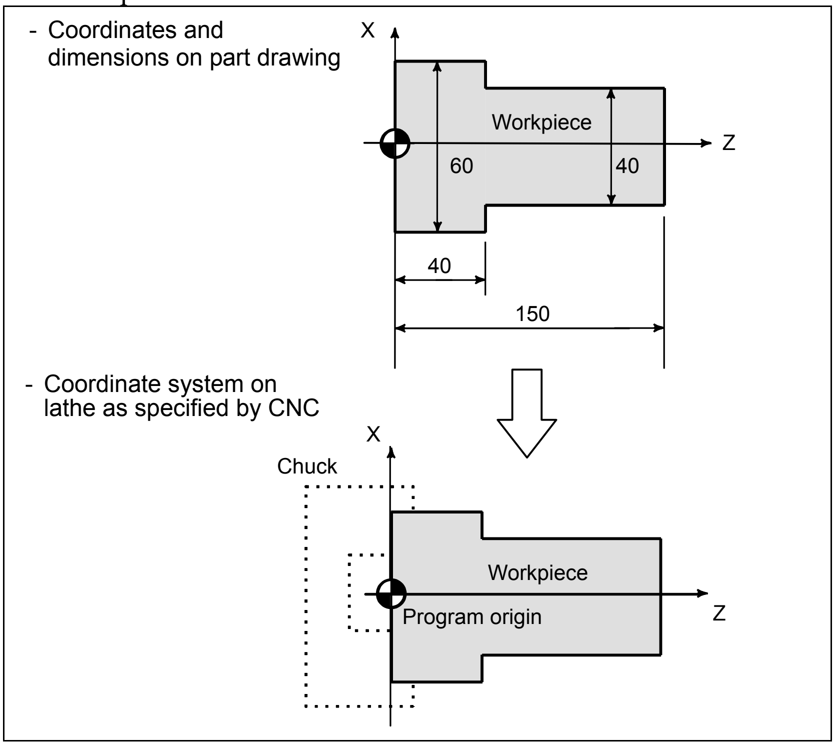


(Jig and coordinate system shall be specified by the same as (2)).

T

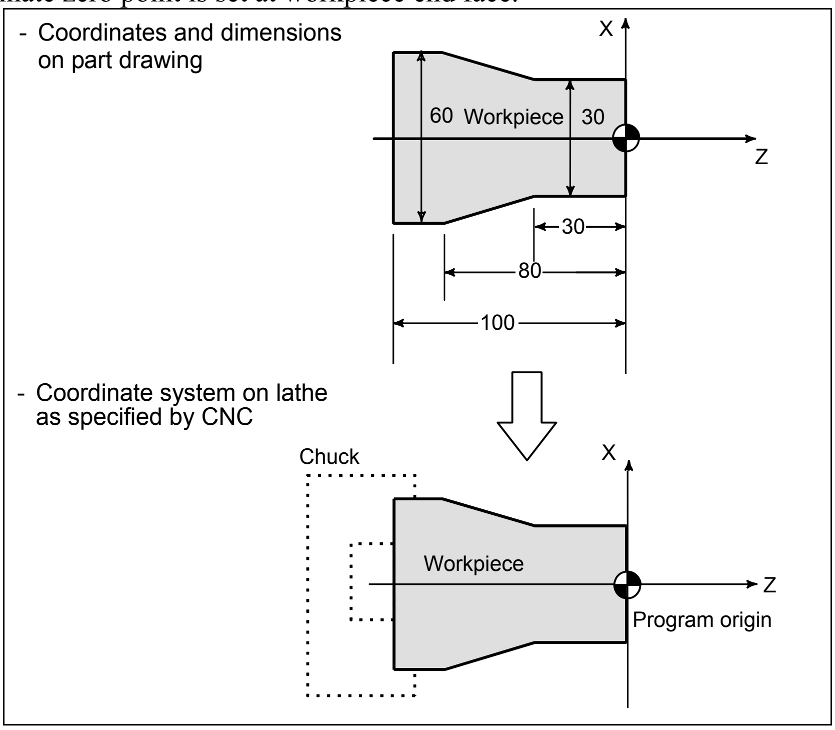
The following method is usually used to define two coordinate systems at the same location.

1 When coordinate zero point is set at chuck face



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the chuck face.

2. When coordinate zero point is set at workpiece end face.



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the end face of the workpiece.

1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute, Incremental Commands)

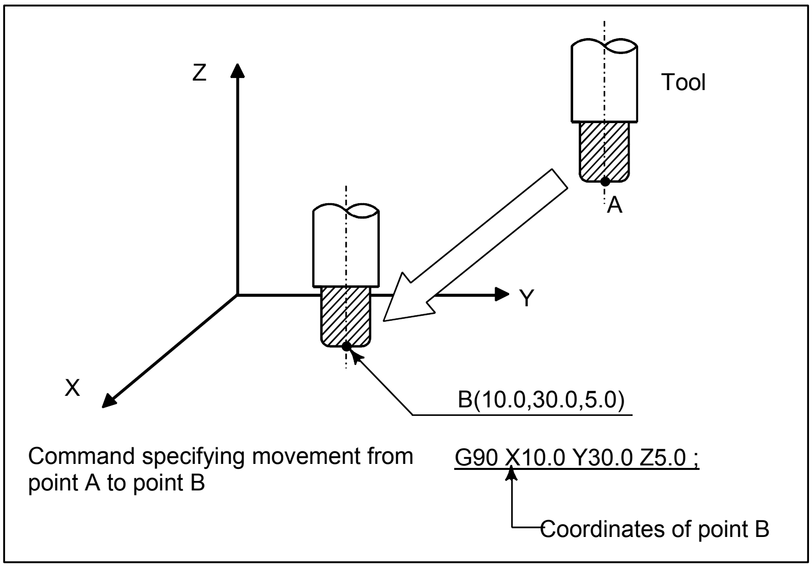
Explanation

Command for moving the tool can be indicated by absolute command or incremental command (See II-8.1).

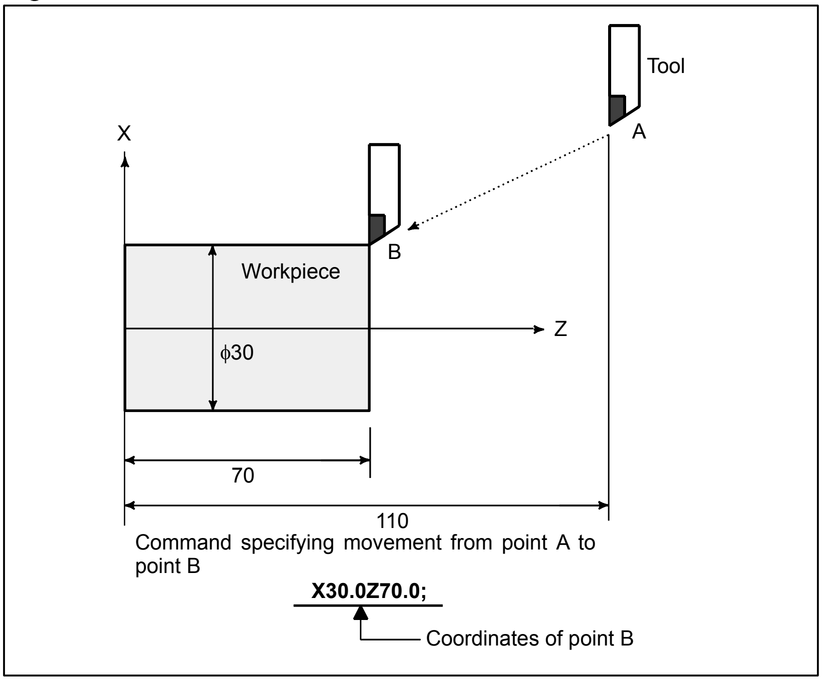
- Absolute command

The tool moves to a point at "the distance from zero point of the coordinate system" that is to the position of the coordinate values.

- For milling machining



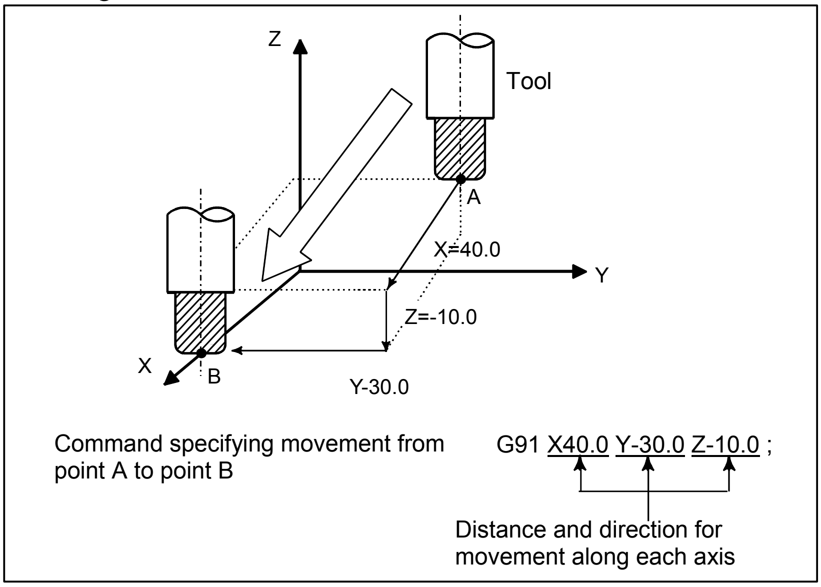
- For lathe cutting



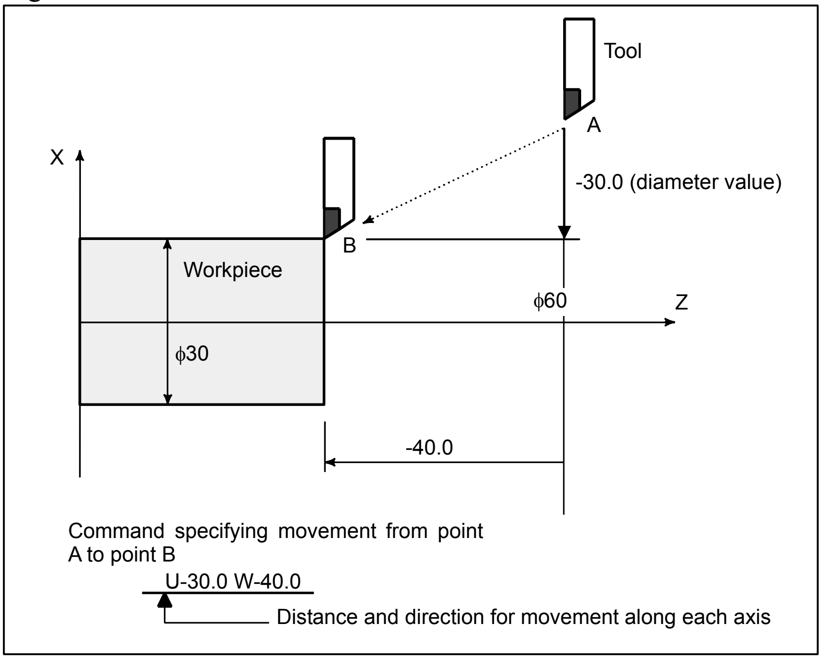
- Incremental command

Specify the distance from the previous tool position to the next tool position.

- For milling machining



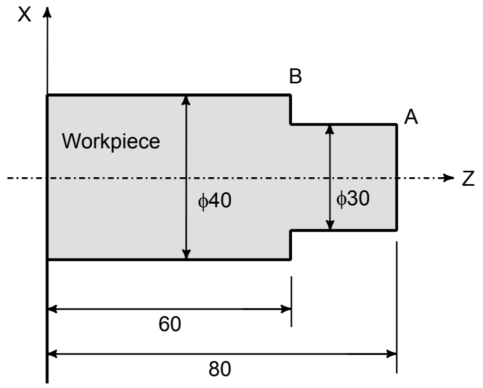
- For lathe cutting



- Diameter programming / radius programming

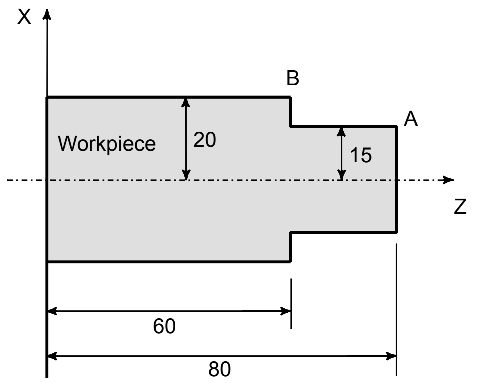
Dimensions of the X axis can be set in diameter or in radius. Diameter programming or radius programming is employed independently in each machine.

1. Diameter programming
   In diameter programming, specify the diameter value indicated on the drawing as the value of the X axis.



Coordinate values of points A and B **A(30.0, 80.0), B(40.0, 60.0)**

2. Radius programming
   In radius programming, specify the distance from the center of the workpiece, i.e. the radius value as the value of the X axis.

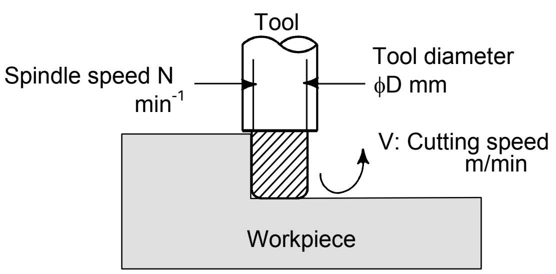


Coordinate values of points A and B **A(15.0, 80.0), B(20.0, 60.0)**

1.4 CUTTING SPEED - SPINDLE FUNCTION

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed. As for the CNC, the cutting speed can be specified by the spindle speed in min^{-1} unit.

- For milling machining

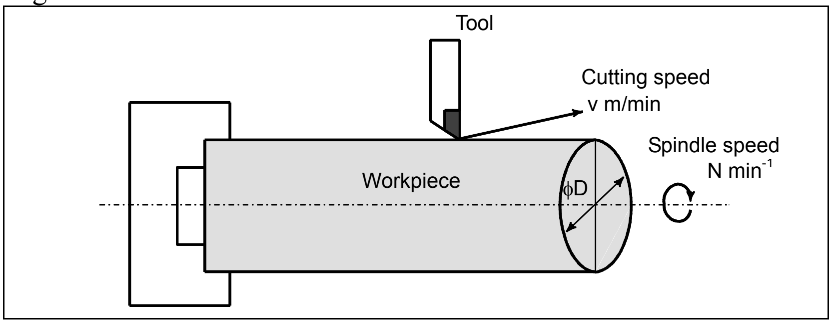


<When a workpiece should be machined with a tool 100 mm in diameter at a cutting speed of 80 m/min.>
The spindle speed is approximately 250 min^{-1}, which is obtained from N=1000v/πD. Hence the following command is required:

S250;

Commands related to the spindle speed are called the spindle speed function (See II-9).

- For lathe cutting



<When a workpiece 200 mm in diameter should be machined at a cutting speed of 300 m/min.>
The spindle speed is approximately 478 min^{-1}, which is obtained from N=1000v/πD. Hence the following command is required:

S478;

Commands related to the spindle speed are called the spindle speed function (See II-9).
The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.
This function is called the constant surface speed control function (See II-9.3).

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION

Overview

For each of various types of machining (such as drilling, tapping, boring, and milling for milling machining, or rough machining, semifinish machining, finish machining, threading, and grooving for turning), a necessary tool is to be selected. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

Examples

M

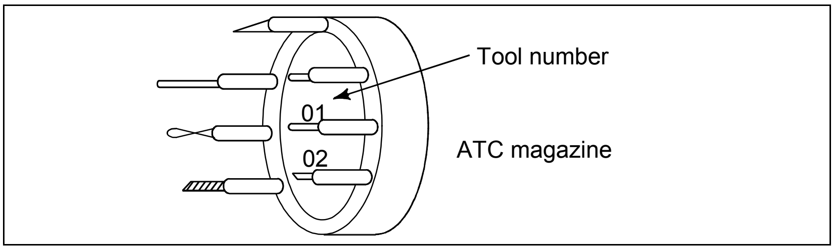


Fig. 1.5 (a) Tool used for various machining

<When No.01 is assigned to a drilling tool>
When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function (See II-10).

T

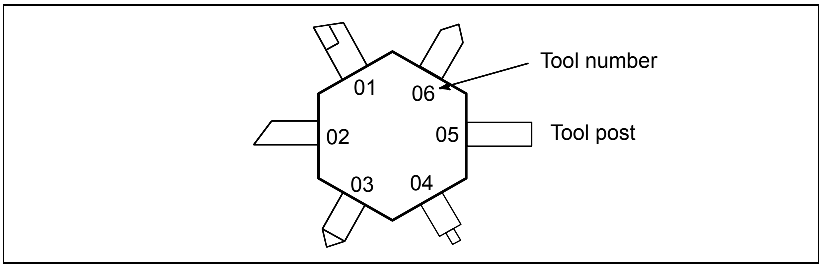


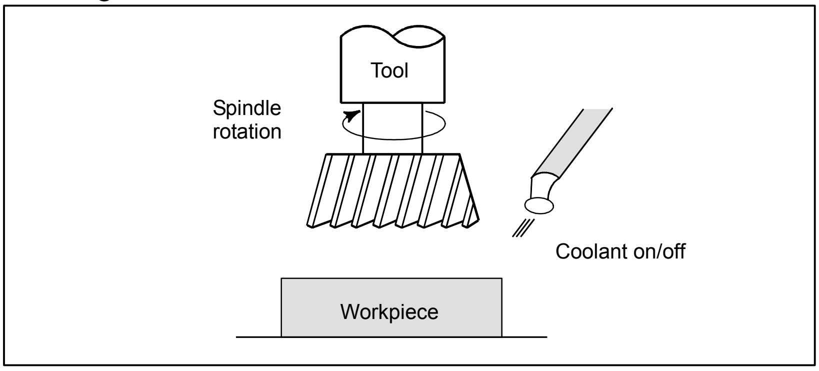
Fig. 1.5 (b) Tool used for various machining

<When No.01 is assigned to a roughing tool>
When the tool is stored at location 01 of the tool post, the tool can be selected by specifying T0101. This is called the tool function (See II-10).

1.6 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION

When a workpiece is actually machined with a tool, the spindle is rotated, coolant is supplied, and the chuck is opened/closed. So, control needs to be exercised on the spindle motor of the machine, coolant valve on/off operation, and chuck open/close operation.

- For milling machining



- For lathe cutting

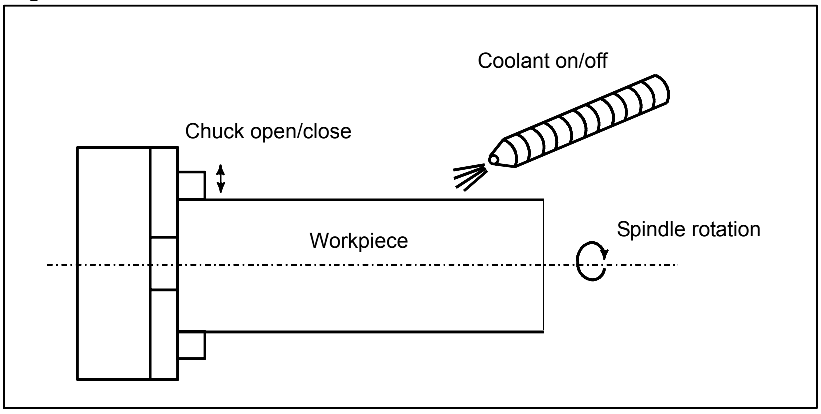


Fig. 1.6 (a) Auxiliary function

The function of specifying the on-off operations of the components of the machine is called the auxiliary function. In general, the function is specified by an M code (See II-11).
For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

1.7 PROGRAM CONFIGURATION

A group of commands given to the CNC for operating the machine is called the program. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In the program, specify the commands in the sequence of actual tool movements.

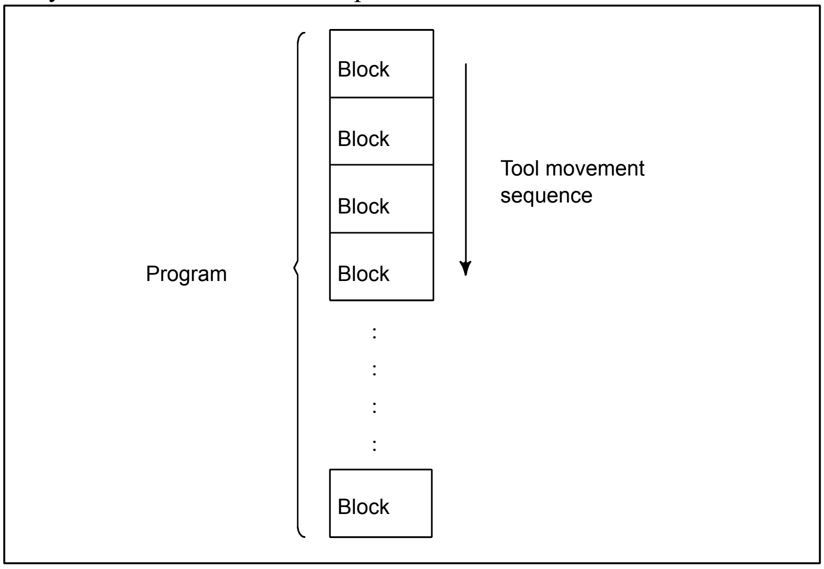


Fig. 1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See II-13).

Explanation

The block and the program have the following configurations.

- Block

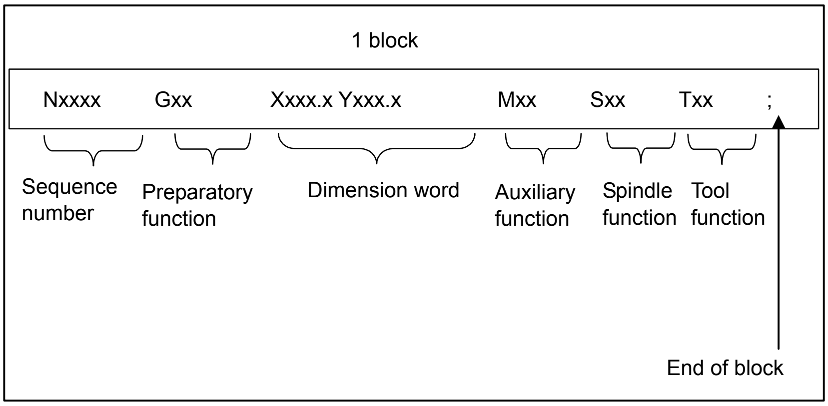


Fig. 1.7 (b) Block configuration

A block starts with a sequence number to identify the block and ends with an end-of-block code.
This manual indicates the end-of-block code by; (LF in the ISO code and CR in the EIA code).
The contents of the dimension word depend on the preparatory function. In this manual, the portion of the dimension word may be represent as IP_.

- Program

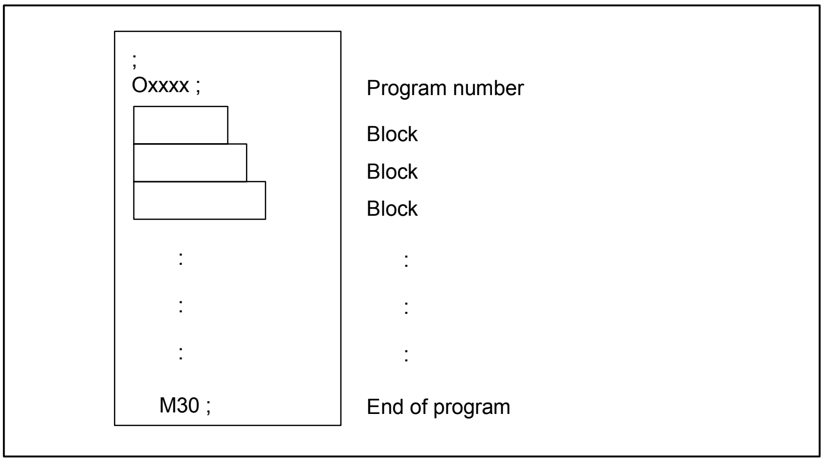


Fig. 1.7 (c) Program configuration

Normally, a program number is specified after the end-of-block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

- Main program and subprogram

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.

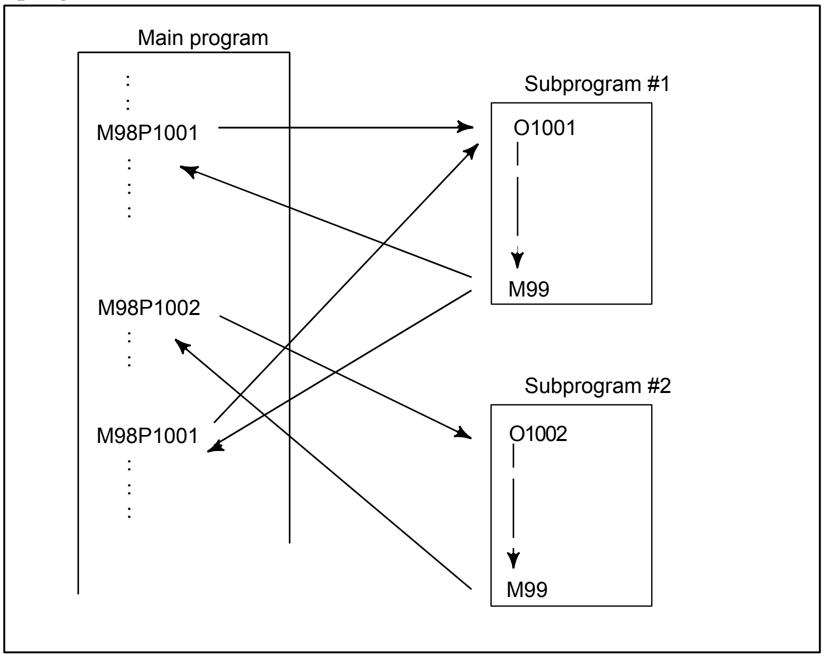
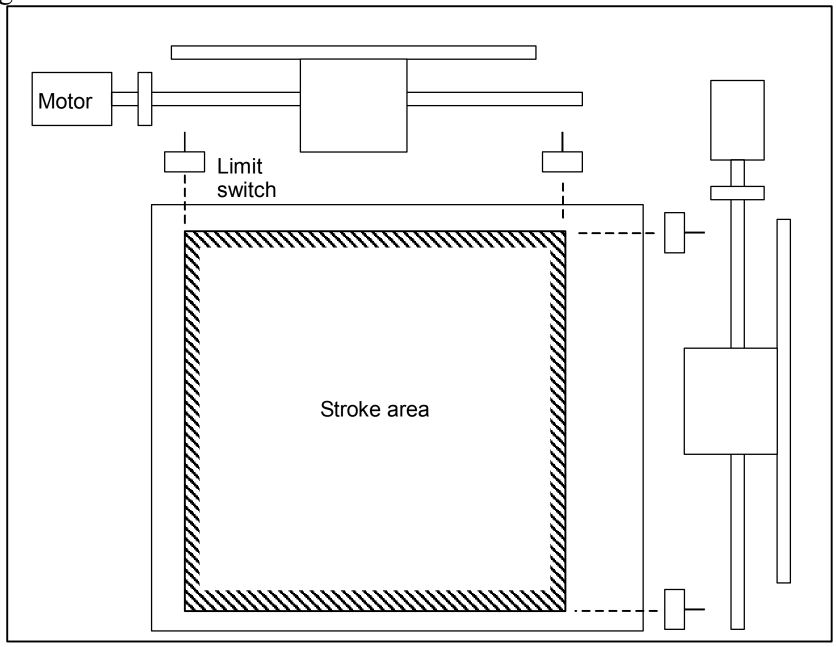


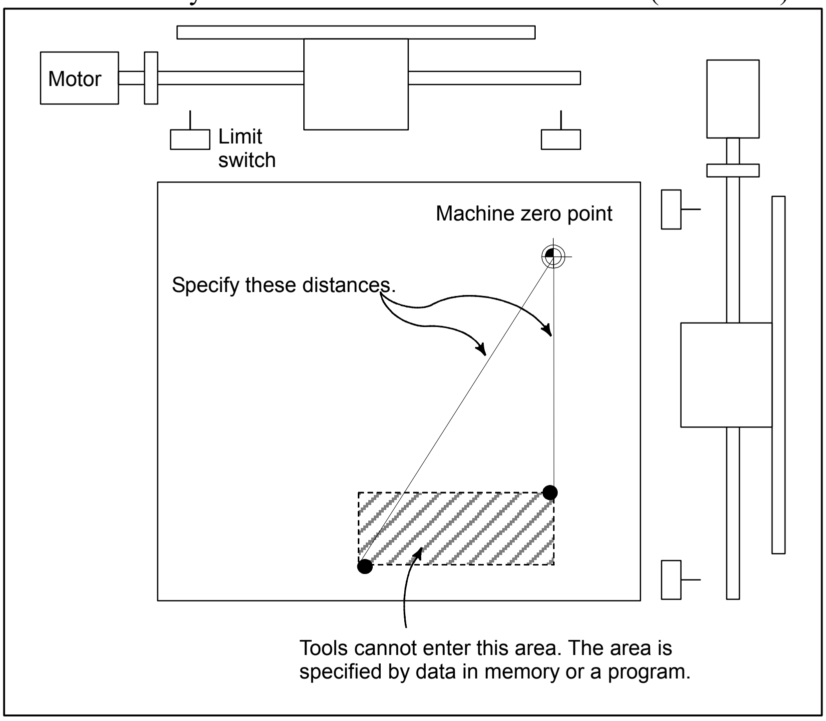
Fig. 1.7 (d) Subprogram execution

1.8 TOOL MOVEMENT RANGE - STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke.



Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory. This function is called stroke check (see III-6.3).



2 CONTROLLED AXES

Chapter 2, "CONTROLLED AXES", consists of the following sections:

2.1 NUMBER OF CONTROLLED AXES

Explanation

The number of controlled axes used with this NC system depends on the model and control type as indicated below.

Series 0*i* -D

Item	M series	T series	
Controlled path	1 path	1 path	2 paths
Total number of controlled axes (feed axes + spindle axes)	Max. 8 axes	Max. 8 axes	Max. 11 axes (Total of two path)
Total feed axes	Max. 7 axes	Max. 7 axes	Max. 9 axes (Total of two path)
Feed axes (for each path)	Max. 7 axes	Max. 7 axes	Max. 7 axes
Simultaneously controlled axes (for 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	Max. 3 axes	Max. 3 axes/4 axes
Cs contour control (each path/total)	2 axes	Max. 3 axes	Max. 3 axes/4 axes

Series 0*i* Mate -D

Item	M series	T series
Controlled path	1 path	1 path
Total number of controlled axes (feed axes + spindle axes)	Max. 5 axes	Max. 5 axes
Total feed axes	Max. 4 axes	Max. 3 axes
Feed axes (for each path)	Max. 4 axes	Max. 3 axes
Simultaneously controlled axes	Max. 3 axes	Max. 3 axes
Axis control by PMC	Max. 4 axes at a time	Max. 3 axes at a time (Not available on Cs axis)
Designation of spindle axes	1 axis	Max. 2 axes
Cs contour control	-	Max. 1 axis

NOTE
1 The maximum number of available controlled axes is limited according to the option configuration. Refer to the manual provided by the machine tool builder for details.
2 The number of simultaneously controllable axes for manual operation (jog feed, manual reference position return, or manual rapid traverse) is 1 or 3 (1 when bit 0 (JAX) of parameter No. 1002 is set to 0 and 3 when it is set to 1).

2.2 NAMES OF AXES

Explanation

The move axes of machine tools are assigned names. These names are referred to as addresses or axis names. Axis names are determined according to the machine tool. The naming rules comply with standards such as the ISO standards.

NOTE
Axis names are predetermined according to the machine used. Refer to the manual supplied by the machine tool builder.

2.3 INCREMENT SYSTEM

Explanation

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inches, or deg.
Three types of increment systems are available as indicated in Table 2.3 (a). For each axis, an increment system can be set using a bit from bit 0 and bit 1 (ISA or ISC) of parameter No. 1013.

Table 2.3 (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 (No.0100#0).
For selection between metric and inch for the least input increment, G code (G20 or G21) or a setting parameter selects it.
Combined use of the inch system and the metric system is not allowed. There are functions that cannot be used between axes with different unit systems (circular interpolation, cutter compensation (M series), etc.). For the increment system, see the machine tool builder's manual.

NOTE

An increment (in millimeters or inches) in the table indicates a diameter value when diameter specification is performed (bit 3 (DIA) of parameter No. 1006 is 1) or a radius value when radius specification is performed (bit 3 (DIA) of parameter No. 1006 is 0).

2.4 MAXIMUM STROKE

Explanation

The maximum stroke controlled by this CNC is shown in the table below:
Maximum stroke = Least command increment × 999999999
(99999999 for IS-A)
Commands that exceed the maximum stroke are not permitted.

Table 2.4 (a) Maximum strokes

Name of increment system	Least input increment	Maximum stroke
IS-A	0.01 mm	±999999.99 mm
	0.001 inch	±99999.999 inch
	0.01 deg	±999999.99 deg
IS-B	0.001 mm	±999999.999 mm
	0.0001 inch	±99999.9999 inch
	0.001 deg	±999999.999 deg
IS-C	0.0001 mm	±99999.9999 mm
	0.00001 inch	±9999.99999 inch
	0.0001 deg	±99999.9999 deg

NOTE

1 The actual stroke depends on the machine tool.
2 An increment (in millimeters or inches) in the table indicates a diameter value when diameter specification is performed (bit 3 (DIA) of parameter No. 1006 is 1) or a radius value when radius specification is performed (bit 3 (DIA) of parameter No. 1006 is 0).

3 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified.
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)

G01 and G00 are modal G codes in group 01.

```
G01 X_ ;
    Z_ ;  G01 is effective in this range.
    X_ ;
G00 Z_ ;  G00 is effective in this range.
    X_ ;
G01 X_ ;
   :
```

T

There are three G code systems in the T series: A, B, and C (Table 3.2(a)). Select a G code system using bits 6 (GSB) and 7 (GSC) of parameter No. 3401. Generally, OPERATOR'S MANUAL describes the use of G code system A, except when the described item can use only G code system B or C. In such cases, the use of G code system B or C is described.

Explanation

1. When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◤ as indicated in Table.
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by bit 7 (G23) of parameter No. 3402. However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting parameter G01 (No. 3402#0).
   (5) The user can select G90 or G91 by setting parameter G91 (No. 3402#3).
   When G code system B or C is used in the T series, setting bit 3 (G91) of parameter No. 3402 determines which code, either G90 or G91, is effective.
   (6) In the M series, the user can select G17, G18, or G19 by setting bits 1 (G18) and 2 (G19) of parameter No. 3401.
2. G codes other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010 occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle for drilling.
6. G codes are indicated by group.

M

7. The group of G60 (M series) is switched according to the setting of bit 0 (MDL) of parameter No. 5431. (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

T

8. For G code system A in the T series, the absolute/incremental command is identified by the address word (X/U, Z/W, C/H, Y/V) instead of the G code (G90/G91). Only the initial level is provided at the return point of the canned cycle for drilling..

3.1 G CODE LIST IN THE M SERIES

M

Table 3.1 (a) G code list

<table>
<tr><th>G code</th><th>Group</th><th colspan="2">Function</th></tr>
<tr><td>G00</td><td rowspan="4">01</td><td colspan="2">Positioning (rapid traverse)</td></tr>
<tr><td>G01</td><td colspan="2">Linear interpolation (cutting feed)</td></tr>
<tr><td>G02</td><td colspan="2">Circular interpolation CW or helical interpolation CW</td></tr>
<tr><td>G03</td><td colspan="2">Circular interpolation CCW or helical interpolation CCW</td></tr>
<tr><td>G04</td><td rowspan="7">00</td><td colspan="2">Dwell, Exact stop</td></tr>
<tr><td>G05.1</td><td colspan="2">AI advanced preview control / AI contour control</td></tr>
<tr><td>G05.4</td><td colspan="2">HRV3 on/off</td></tr>
<tr><td>G07.1 (G107)</td><td colspan="2">Cylindrical interpolation</td></tr>
<tr><td>G09</td><td colspan="2">Exact stop</td></tr>
<tr><td>G10</td><td colspan="2">Programmable data input</td></tr>
<tr><td>G11</td><td colspan="2">Programmable data input mode cancel</td></tr>
<tr><td>G15</td><td rowspan="2">17</td><td colspan="2">Polar coordinates command cancel</td></tr>
<tr><td>G16</td><td colspan="2">Polar coordinates command</td></tr>
<tr><td>G17</td><td rowspan="3">02</td><td>XpYp plane selection</td><td rowspan="3">Xp: X axis or its parallel axis
Yp: Y axis or its parallel axis
Zp: Z axis or its parallel axis</td></tr>
<tr><td>G18</td><td>ZpXp plane selection</td></tr>
<tr><td>G19</td><td>YpZp plane selection</td></tr>
<tr><td>G20</td><td rowspan="2">06</td><td colspan="2">Input in inch</td></tr>
<tr><td>G21</td><td colspan="2">Input in mm</td></tr>
<tr><td>G22</td><td rowspan="2">04</td><td colspan="2">Stored stroke check function on</td></tr>
<tr><td>G23</td><td colspan="2">Stored stroke check function off</td></tr>
<tr><td>G27</td><td rowspan="5">00</td><td colspan="2">Reference position return check</td></tr>
<tr><td>G28</td><td colspan="2">Automatic return to reference position</td></tr>
<tr><td>G29</td><td colspan="2">Movement from reference position</td></tr>
<tr><td>G30</td><td colspan="2">2nd, 3rd and 4th reference position return</td></tr>
<tr><td>G31</td><td colspan="2">Skip function</td></tr>
<tr><td>G33</td><td>01</td><td colspan="2">Threading</td></tr>
<tr><td>G37</td><td rowspan="2">00</td><td colspan="2">Automatic tool length measurement</td></tr>
<tr><td>G39</td><td colspan="2">Cutter compensation : corner circular interpolation</td></tr>
<tr><td>G40</td><td rowspan="3">07</td><td colspan="2">Cutter compensation : cancel</td></tr>
<tr><td>G41</td><td colspan="2">Cutter compensation : left</td></tr>
<tr><td>G42</td><td colspan="2">Cutter compensation : right</td></tr>
<tr><td>G40.1</td><td rowspan="3">19</td><td colspan="2">Normal direction control cancel mode</td></tr>
<tr><td>G41.1</td><td colspan="2">Normal direction control on : left</td></tr>
<tr><td>G42.1</td><td colspan="2">Normal direction control on : right</td></tr>
<tr><td>G43</td><td rowspan="2">08</td><td colspan="2">Tool length compensation +</td></tr>
<tr><td>G44</td><td colspan="2">Tool length compensation -</td></tr>
</table>

Table 3.1 (a) G code list

G code	Group	Function
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)
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

Table 3.1 (a) G code list

G code	Group	Function
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)

3.2 G CODE LIST IN THE T SERIES

T

Table 3.2 (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

Table 3.2 (a) G code list

G code system A	G code system B	G code system C	Group	Function
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)

Table 3.2 (a) G code list

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

4 INTERPOLATION FUNCTIONS

Interpolation functions specify the way to make an axis movement (in other words, a movement of the tool with respect to the workpiece or table).

Chapter 4, "INTERPOLATION FUNCTIONS", consists of the following sections:

4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental programming at a rapid traverse rate.
In the absolute programming, coordinate value of the end point is programmed.
In the incremental programming the distance the tool moves is programmed.

Format

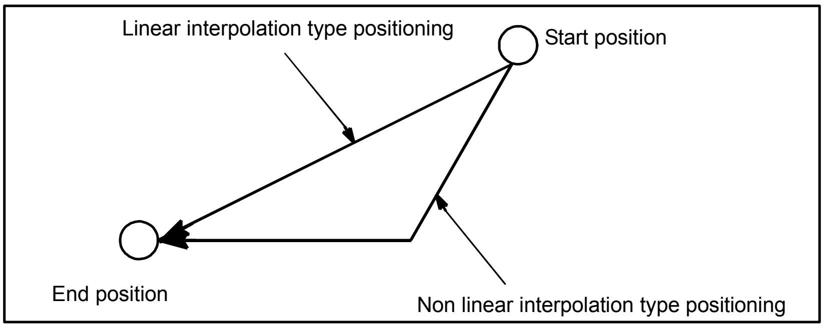
G00 IP_ ;

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

Explanation

Either of the following tool paths can be selected according to bit 1 (LRP) of parameter No. 1401.

- 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.
  However, the path is not identical to that of linear interpolation (G01).



The rapid traverse rate in G00 command is set to the parameter No. 1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position.
"In-position " means that the feed motor is within the specified range.
This range is determined by the machine tool builder by setting to parameter (No. 1826).

Limitation

The rapid traverse rate cannot be specified in the address F.
Even if linear interpolation type positioning is specified, nonlinear type interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

- G28 specifying positioning between the reference and intermediate positions.
- G53

4.2 LINEAR INTERPOLATION (G01)

Tools can move along a line.

Format

G01 IP_ **F_ ;**

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

Explanation

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.
The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.
The feedrate of each axis direction is as follows.

G01 $\alpha\underline{\alpha}$ $\beta\underline{\beta}$ $\gamma\underline{\gamma}$ $\zeta\underline{\zeta}$ **F**$\underline{f}$ **;**

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

Feed rate of β axis direction : $F\beta = \frac{\beta}{L} \times f$

Feed rate of γ axis direction : $F\gamma = \frac{\gamma}{L} \times f$

Feed rate of ζ axis direction : $F\zeta = \frac{\zeta}{L} \times f$

$$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$$

The feedrate of the rotary axis is commanded in the unit of deg/min (the unit is decimal point position).
When linear interpolation of linear axis α (such as X, Y, or Z) and rotation axis β (such as A, B, or C) is performed, the feedrate specified in F is the tangential feedrate in the α-β Cartesian coordinate system with A, B, or C represented in degrees and X, Y, or Z represented in millimeters or inches.
β-axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above formula, then the β-axis feedrate unit is changed to deg/min.

A calculation example is as follows.
G91 G01 X20.0B40.0 F300.0 ;
This changes the unit of the C axis from 40.0 deg to 40mm with metric input. The time required for distribution is calculated as follows:

$$\frac{\sqrt{20^2+40^2}}{300} \fallingdotseq 0.14907(\mathrm{min})$$

The feedrate for the C axis is

$$\frac{40}{0.14907} \fallingdotseq 268.3\,\mathrm{deg/min}$$

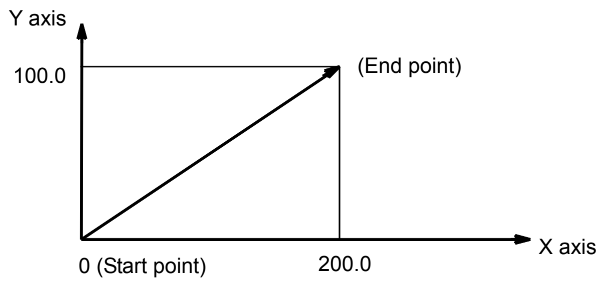
In simultaneous 3 axes control, the feedrate is calculated the same way as in 2 axes control.

Example

- Linear interpolation

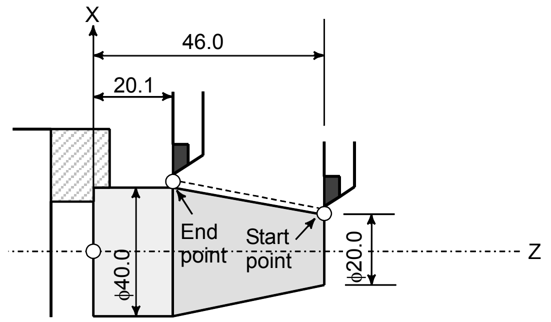
- For milling machining

(G91) G01X200.0Y100.0F200.0;



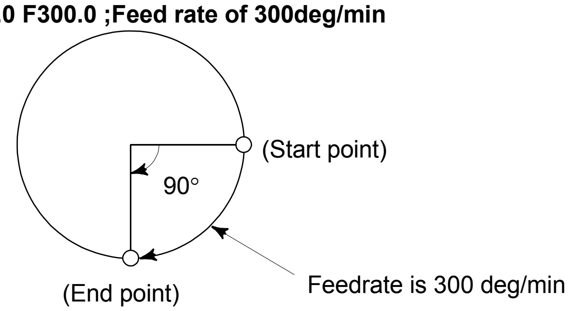
- For lathe cutting

(Diameter programming)
G01X40.0Z20.1F20; (Absolute programming)
or
G01U20.0W-25.9F20; (Incremental programming)



- Feedrate for the rotary axis

G91G01C-90.0 F300.0 ;Feed rate of 300deg/min



4.3 CIRCULAR INTERPOLATION (G02, G03)

The command below will move a tool along a circular arc.

Format

Arc in the XpYp plane

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

Arc in the ZpXp plane

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

Arc in the YpZp plane

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

Command	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular Interpolation : Clockwise direction (CW)
G03	Circular Interpolation : Counterclockwise direction (CCW)
Xp_	Command values of X axis or its parallel axis (set by parameter No. 1022)
Yp_	Command values of Y axis or its parallel axis (set by parameter No. 1022)
Zp_	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	Xp axis distance from the start point to the center of an arc with sign
J_	Yp axis distance from the start point to the center of an arc with sign
K_	Zp axis distance from the start point to the center of an arc with sign
R_	Arc radius (with sign, radius value for lathe cutting)
F_	Feedrate along the arc

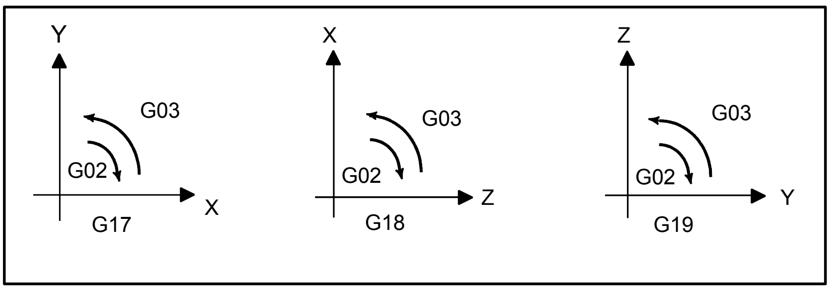
T

NOTE
The U-, V-, and W-axes can be used with G-codes B and C.

Explanation

- Direction of the circular interpolation

"Clockwise"(G02) and "counterclockwise"(G03) on the XpYp plane (ZpXp plane or YpZp plane) are defined when the XpYp plane is viewed in the positive-to-negative direction of the Zp axis (Yp axis or Xp axis, respectively) in the Cartesian coordinate system. See the figure below.



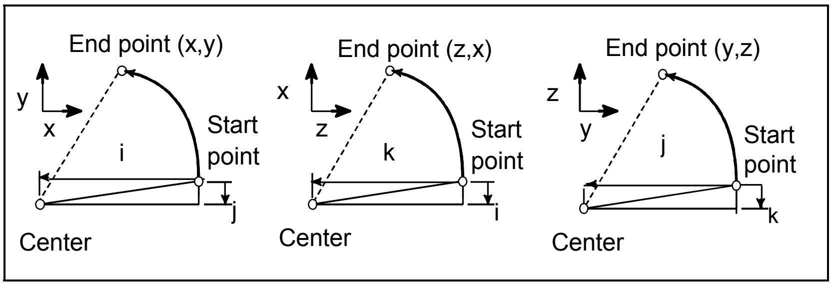
- Distance moved on an arc

The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified with sign.

- Distance from the start point to the center of arc

The arc center is specified by addresses I, J, and K for the Xp, Yp, and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.

I, J, and K must be signed according to the direction.



I0, J0, and K0 can be omitted.

If the difference between the radius at the start point and that at the end point exceeds the permitted value in a parameter (No.3410), an alarm PS0020 occurs.

- Command for a circle

When Xp, Yp, and Zp are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.

G02 I_ ; Command for a circle

- Arc radius

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.

M

When an arc exceeding 180° is commanded, the radius must be specified with a negative value.

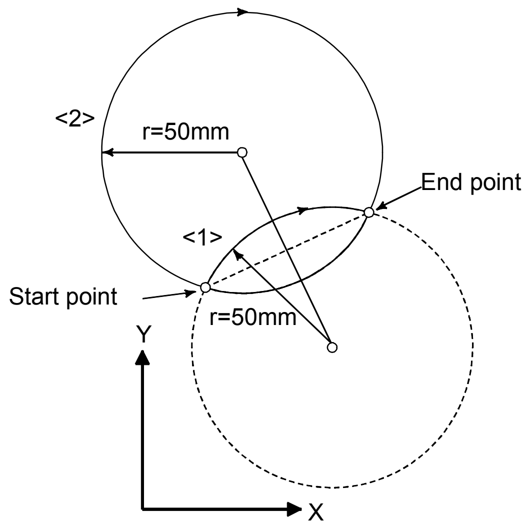
T

An arc exceeding 180° cannot be specified (a negative value cannot be used for the radius). If specified, alarm PS0023 is issued.

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.)

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 ;



- Feedrate

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.

The difference between the specified feedrate and the actual feedrate of the tool must fall within ±2%. However, this feedrate is measured along the arc after the tool radius compensation is applied

Limitation

- Simultaneously specifying R with I, J, and K

If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

- Specifying an axis that is not contained in the specified plane

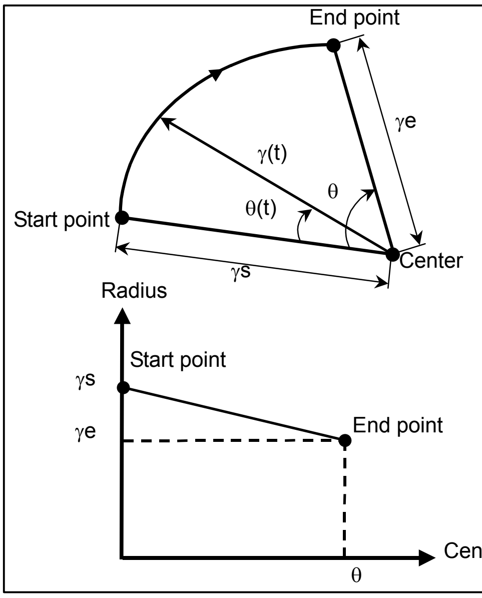
If an axis not comprising the specified plane is commanded, an alarm PS0028 occurs.
For example,
For milling machining:
If the X-axis and a U-axis parallel to the X-axis are specified when the XY plane is specified
For lathe cutting:
If the X-axis and a U-axis parallel to the X-axis are specified when the ZX plane is specified with G code system B or C

- Specifying a semicircle with R

When an arc having a center angle approaching 180° is specified, the calculated center coordinates may contain an error. In such a case, specify the center of the arc with I, J, and K.

- Difference in the radius between the start and end points

If the difference in the radius between the start and end points of the arc exceeds the value specified in parameter No. 3410, alarm PS0020 is generated.
When an end point does not lie on the arc, a spiral results, as shown below.

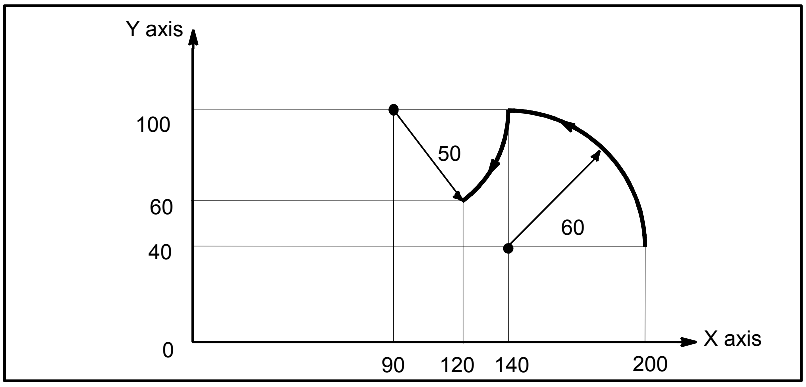


$$\gamma(t) = \gamma s + \frac{(\gamma e - \gamma s)\,\theta(t)}{\theta}$$

The arc radius changes linearly with the center angle θ(t). Spiral interpolation is performed using a circular command that specifies one arc radius for the start point and another arc radius for the end point. To use spiral interpolation, set a large value in parameter No. 3410, used to specify the limit on the arc radius error.

Example

M



The above tool path can be programmed as follows;

(1) In absolute programming

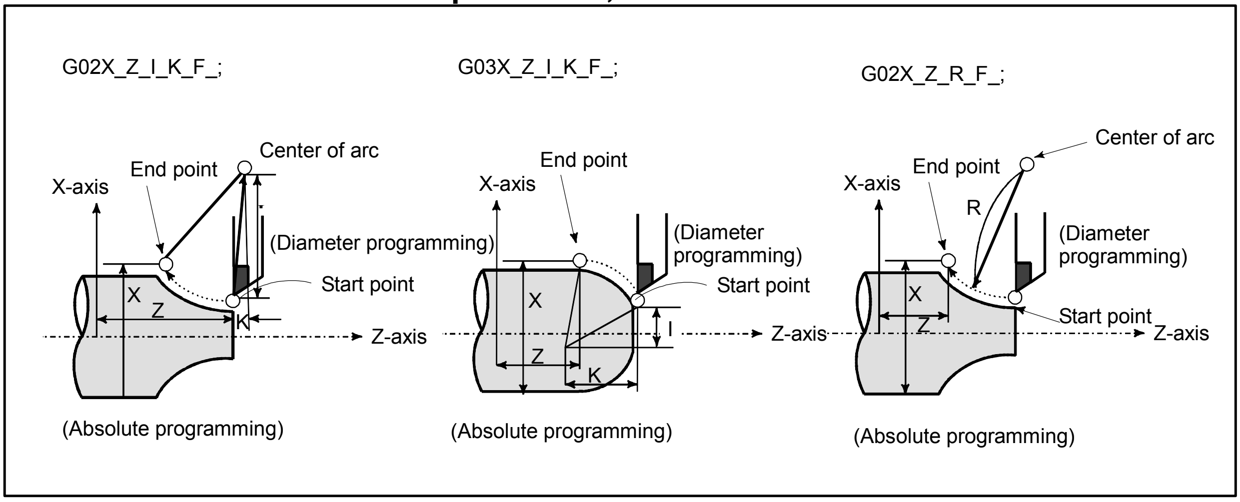
```
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300. ;
G02 X120.0 Y60.0 R50.0 ;
or
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 I-60.0 F300. ;
G02 X120.0 Y60.0 I-50.0 ;
```

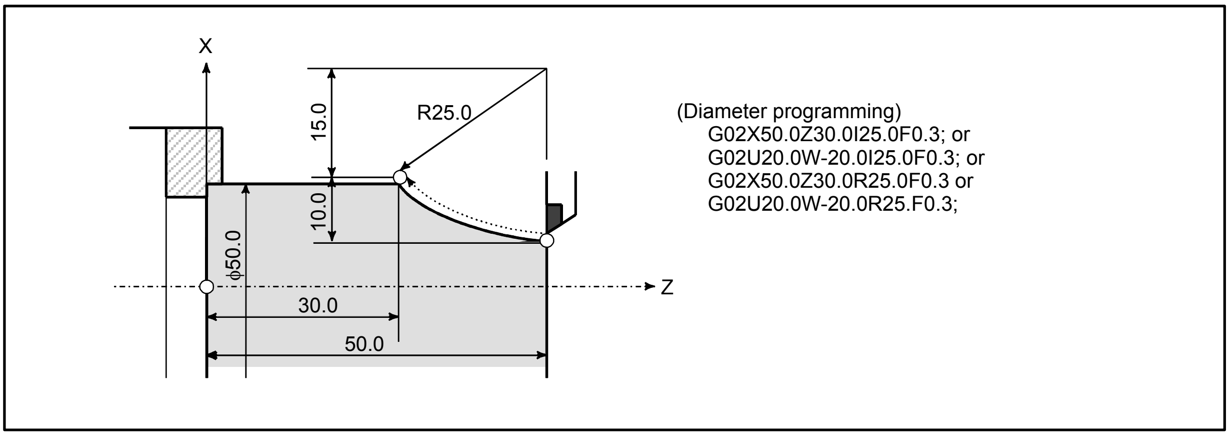
(2) In incremental programming

```
G91 G03 X-60.0 Y60.0 R60.0 F300. ;
G02 X-20.0 Y-40.0 R50.0 ;
or
G91 G03 X-60.0 Y60.0 I-60.0 F300. ;
G02 X-20.0 Y-40.0 I-50.0 ;
```

T

- **Command of circular interpolation X, Z**





4.4 HELICAL INTERPOLATION (G02, G03)

Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Format

Arc in the XpYp plane

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

Arc in the ZpXp plane

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

Arc in the YpZp plane

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

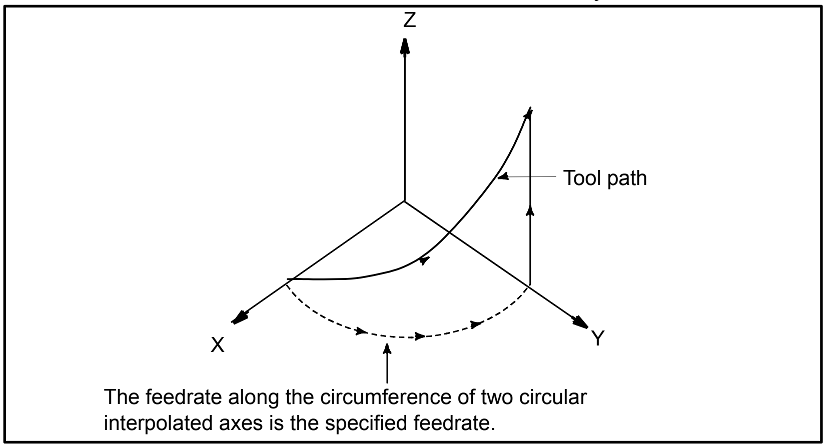
α, β : Any one axis where circular interpolation is not applied.
Up to two other axes can be specified.

Explanation

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 bit 5 (HTG) of parameter No.1403.
An F command specifies a feedrate along a circular arc, when HTG is specified to 0. 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.

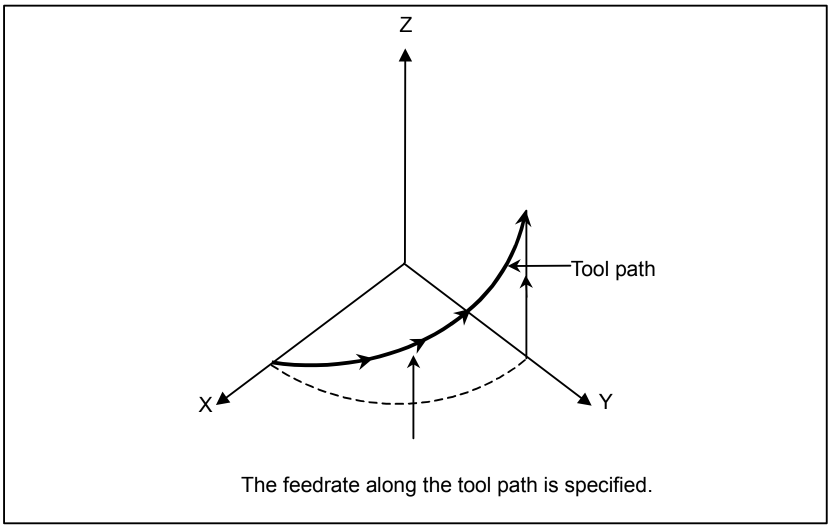


If HTG is set to 1, specify a feedrate along the tool path about the linear axis. Therefore, the tangential velocity of the arc is expressed as follows:

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

The velocity along the linear axis is expressed as follows:

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



Limitation

- Cutter compensation (M series) or tool nose radius compensation (T series) is applied only for a circular arc.
- Tool offset and tool length compensation (M series) cannot be used in a block in which a helical interpolation is commanded.

4.5 CYLINDRICAL INTERPOLATION (G07.1)

In cylindrical interpolation function, 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.
G107 can be used instead of G07.1.

Explanation

- Plane selection (G17, G18, G19)

To specify a G code for plane selection, set the rotary axis in parameter No. 1022 as a linear axis that is one of the basic three axes of the basic coordinate system or an axis parallel to one of the basic axes. For example, when rotary axis C-axis is assumed to be parallel to the X-axis, specifying G17, axis address C, and Y at the same time can select a plane formed by the C-axis and Y-axis (the Xp-Yp plane).

T

NOTE
The U-, V-, and W-axes can be used with G-codes B and C.

- Feedrate

A feedrate specified in the cylindrical interpolation mode is the feedrate on the circumference.

- Circular interpolation (G02, G03)

Circular interpolation can be performed between the rotary axis set for cylindrical interpolation and another linear axis. Radius R is used in commands in the same way as described.
The unit for a arc radius is not degrees but millimeters (for metric input) or inches (for inch input).

<Example Circular interpolation between the Z axis and C axis>
For the C axis of parameter (No.1022), 5 (axis parallel with the X axis) is to be set. In this case, the command for circular interpolation is
G18 Z_C_;
G02 (G03) Z_C_R_;
For the C axis of parameter (No.1022), 6 (axis parallel with the Y axis) may be specified instead. In this case, however, the command for circular interpolation is
G19 C_Z_;
G02 (G03) Z_C_R_;

- Tool radius/tool nose radius compensation

To perform tool radius/tool nose radius compensation in the cylindrical interpolation mode, cancel any ongoing tool radius/tool nose radius compensation mode before entering the cylindrical interpolation mode. Then, start and terminate tool radius/tool nose radius compensation within the cylindrical interpolation mode.

- Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment.
So when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative.
If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

$$\text{The actual amount of travel} = \left[\frac{\text{MOTION REV}}{2\times 2\pi\text{R}} \times \left[\text{Specified value} \times \frac{2\times 2\pi\text{R}}{\text{MOTION REV}}\right]\right]$$

MOTION REV : The amount of travel per rotation of the rotary axis (360°)
R : Workpiece radius
[ ] : Rounded to the least input increment

Limitation

- Arc radius specification in the circular interpolation

In the cylindrical interpolation mode, an arc radius cannot be specified with word address I, J, or K.

- Positioning

In the cylindrical interpolation mode, positioning operations (including those that produce rapid traverse cycles such as G28, G53, G73, G74, G76, G80 to G89) cannot be specified. Before positioning can be specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G07.1) cannot be performed in the positioning mode (G00).

- Cylindrical interpolation mode setting

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode must be cancelled before the cylindrical interpolation mode can be reset.

- Rotary axis

Only one rotary axis can be set for cylindrical interpolation. Therefore, it is impossible to specify more than one rotary axis in the G07.1 command.

- Rotary axis roll-over

If a rotary axis using the roll-over function is specified at the start of the cylindrical interpolation mode, the roll-over function is automatically disabled in the cylindrical interpolation mode. After the cylindrical interpolation mode is canceled, the roll-over function is enabled automatically.

- Tool radius/tool nose radius compensation

If the cylindrical interpolation mode is specified when tool radius/tool nose radius compensation is already being applied, correct compensation is not performed. Specify compensation in the cylindrical interpolation mode.

- Canned cycle for drilling

Canned cycles (G73, G74, and G81 to G89 for M series / G80 to G89 for T series) for drilling, cannot be specified during cylindrical interpolation mode.

- Tool offset

A tool offset must be specified before the cylindrical interpolation mode is set. No offset can be changed in the cylindrical interpolation mode.

M

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system (G92, G54 to G59) or local coordinate system (G52) cannot be specified.

- Index table indexing function

Cylindrical interpolation cannot be specified when the index table indexing function is being used.

T

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system (G50, G54 to G59) and local coordinate system (G52) cannot be specified.

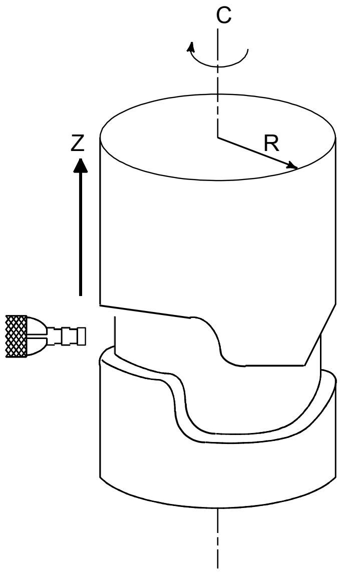
- Mirror image for double turret

Mirror image for double turret, G68 and G69, cannot be specified during cylindrical interpolation mode.

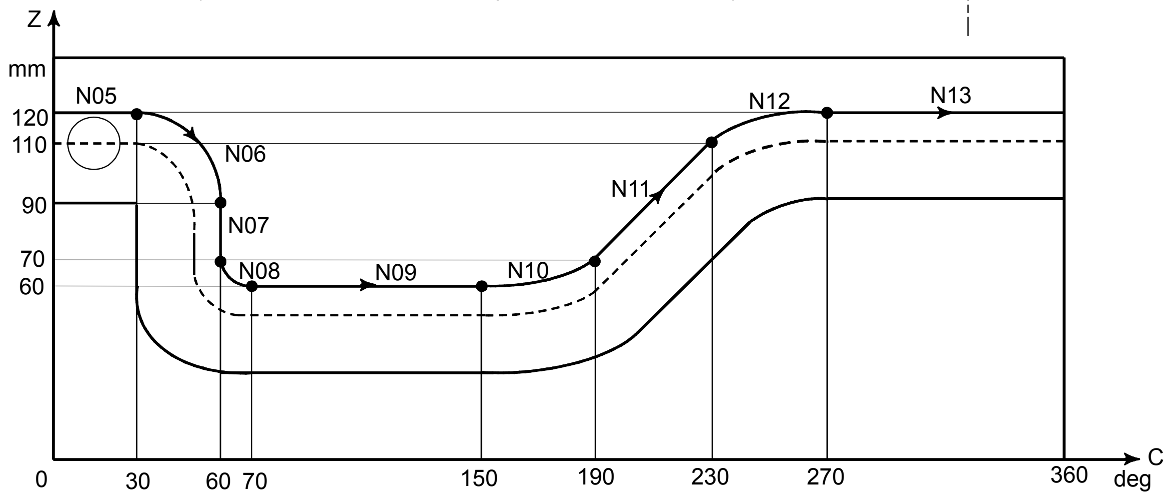
Example

Example of a Cylindrical Interpolation

```
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 ;
```



(* A command with a decimal point can also be used.)



4.6 SKIP FUNCTION (G31)

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)

Explanation

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5065, as follows.

#5061 : First axis coordinate value
#5062 : Second axis coordinate value
#5063 : Third axis coordinate value
#5064 : Fourth axis coordinate value

#5065 : Fifth axis coordinate value

> ⚠ **CAUTION**
> Disable feedrate override, dry run, and automatic acceleration/deceleration (however, these become available by setting bit 7 (SKF) of parameter No.6200 to 1.) when the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

> **NOTE**
> If G31 command is issued while tool radius/tool nose radius compensation is applied, an alarm PS0035 is displayed. Cancel the tool radius compensation with the G40 command before the G31 command is specified.

Example

- The next block to G31 is an incremental programming

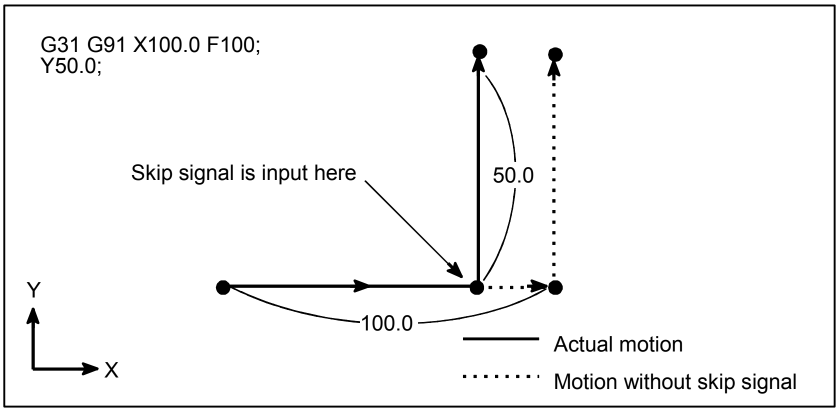


Fig. 4.6 (a) The next block is an incremental programming

- The next block to G31 is an absolute programming for 1 axis

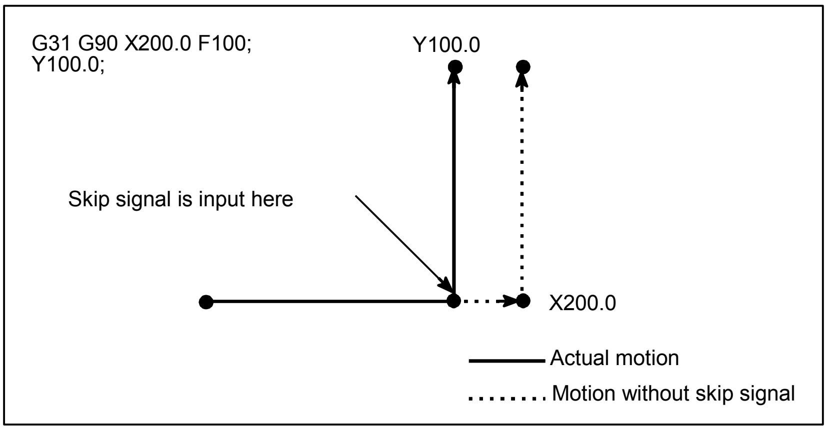


Fig. 4.6 (b) The next block is an absolute programming for 1 axis

- The next block to G31 is an absolute programming for 2 axes

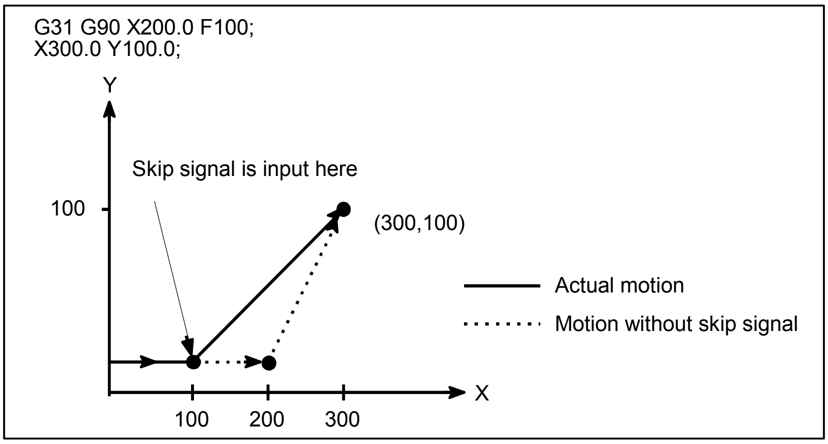


Fig. 4.6 (c) The next block is an absolute programming for 2 axes

4.7 MULTI-STEP SKIP (G31)

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 (four or eight signals, or four signals when high-speed skip signals are used) is turned on. In the block where Q1 to Q4 are specified after G04, dwell can be skipped when skip signals (four or eight signals, or four signals when high-speed skip signals are 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 IP_ F_ P_ ;
IP_ : End point
F_ : Feedrate
P_ : P1 to P4
Dwell
G04X(U,P)_ (Q_);
X(U,P)_ : Dwell time
Q_ : Q1 to Q4

Explanation

Multi-step skip is caused by specifying P1, P2, P3, or P4 in a G31 block. For an explanation of selecting (P1, P2, P3, or P4), refer to the manual supplied by the machine tool builder.
Specifying Q1, Q2, Q3, or Q4 in G04 (dwell command) enables dwell skip in a similar way to specifying G31. A skip may occur even if Q is not specified. For an explanation of selecting (Q1, Q2, Q3, or Q4), refer to the manual supplied by the machine tool builder.

- Correspondence to skip signals

Parameters Nos. 6202 to 6205 can be used to specify which signals are enabled of four or eight (four when high-speed skip signals are used) skip signals. Specification is not limited to one-to-one correspondence. It is possible to specify that one skip signal correspond to two or more Pn's or Qn's (n=1, 2, 3, 4). Also, bits 0 (DS1) and 7 (DS8) parameter No.6206 can be used to specify dwell.

> ⚠ **CAUTION**
> Dwell is not skipped when Qn is not specified and bits 0 (DS1) and 7 (DS8) parameter No.6206 are not set.

4.8 HIGH-SPEED SKIP SIGNAL (G31)

The skip function operates based on a high-speed skip signal (connected directly to the NC; 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 - 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.
For details, refer to the appropriate manual supplied from the machine tool builder.

Format

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

4.9 TORQUE LIMIT SKIP

Overview

Executing the move command following G31P99 (or G31P98) while overriding the torque limit[*1] on the servo motor enables cutting feed in to be performed in the same way as in linear interpolation (G01). If, during the movement with this command, the torque of the servo motor reaches the torque limit value (torque limit on the servo motor multiplied by the override) due to pressing or other causes or a skip signal (including a high-speed skip signal) is input, any remaining move commands will be canceled, and the next block is executed. (The operation of canceling any remaining move commands and executing the next block is called a skip operation in the remainder of this document.)
It is possible to override the torque limit on the servo motor with the following command methods:
(1) Execute the torque limit override command in the PMC window.

Execute the torque limit command in the PMC window in advance. If the torque limit override command is not set in advance, alarm PS0035 is issued. If the command falls outside the range, alarm PS0036 is issued.
*1 : The torque limit on the servo motor is automatically set to a value conforming to the motor type setting.

Format

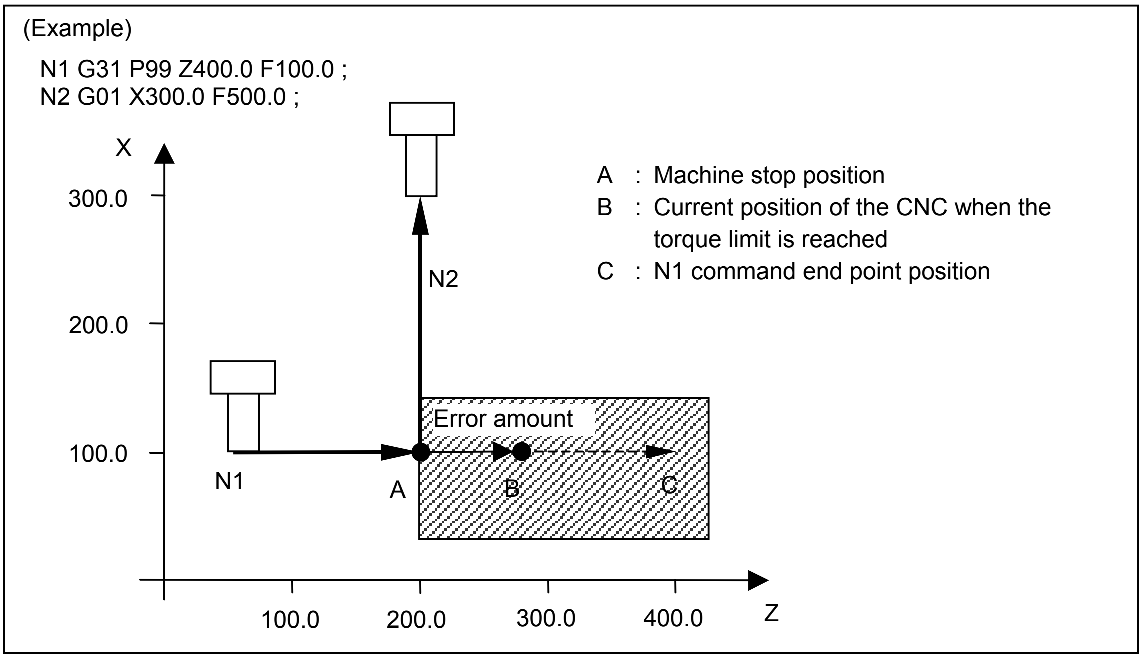
G31 P98 α_ F_ ;
G31 P99 α_ F_ ;

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

- Conditions for performing a skip operation

Condition	Command	
	G31P98	G31P99
The torque limit value is reached.	A skip operation is performed.	A skip operation is performed.
A skip signal is input.	No skip operation is performed.	A skip operation is performed.

- Operation during a torque limit skip



A torque limit skip presses a specified axis against a previously prepared part or other item while the torque limit command is being executed on the servo motor, and then performs a skip operation when the servo motor reaches the torque limit value. A skip operation is performed with the fact that the torque limit value as detected in the servo motor is reached. It is, therefore, not necessary to input a skip signal using a separate sensor or other device unlike with normal skip functions.

(1) At point A, the machine comes in contact with the object under measurement and stops. At this time, because the torque limit value is not reached, no skip operation is performed, move commands are continuously output, and the current position of the CNC is updated.

(2) Because move commands are output but the machine remains stopped, there occurs a difference (error amount) between the current position of the CNC and the machine position, and torque is applied to the servo motor.

(3) When the torque limit value is reached, a skip operation is performed at machine stop position, point A, and the N2 command is executed. Assuming that the current position of the CNC when the torque limit is reached is point B, the error amount during the torque limit skip is (A - B).

- Torque limit command

When no torque limit command is issued in the torque limit skip command in the PMC or other windows, alarm PS0035 is issued.

When no toque limit command is issued, the torque limit override value is either 0% or 100%.

The torque limit command is issued as shown in the following programming example.

(Program example)
O0012 ;
:
Mxx ; (Specify a torque limit from the PMC via the window)

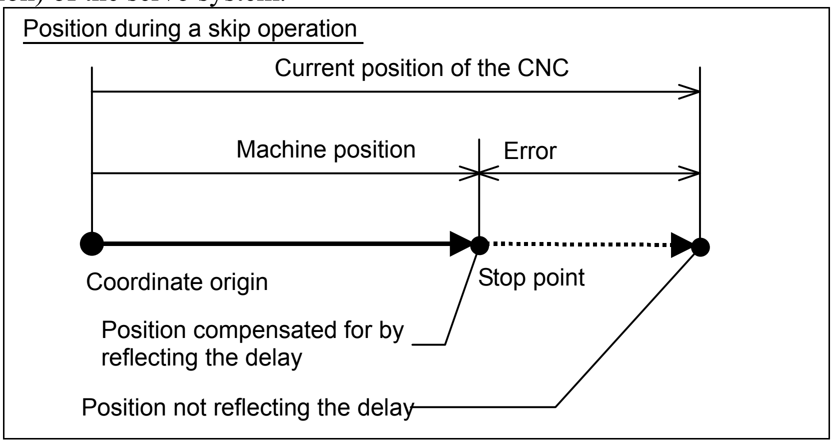
```
:
G31 P99 X200. F100. ; (Torque limit skip command)
:
G01 X100. F500. ; (Move command with the torque limit being still effective)
:
Myy ; (Cancel the torque limit from the PMC)
:
M30 ;
```

- Positional deviation limit during the torque limit command

While the torque limit skip command is being executed, the positional deviation limit check with the settings of parameters Nos. 1828 and 1829 is not performed. Instead, the positional deviation limit check with the settings of parameter No. 6287 is performed. If the positional deviation exceeds the limit, alarm SV0004 is issued and an instantaneous stop occurs.

- Custom macro variables

When the torque limit skip command is executed, the custom macro system variables #5061 to #5065 (skip signal position) store the coordinate position assumed at the end of the skip. In reality, there is a deviation due to the delay of the servo system between the machine position and the current position of the CNC when a skip operation is executed. This deviation can be determined from the positional deviation of the servo. By setting bit 2 (TSE) of parameter No. 6201, it is possible to select whether or not the skip signal position to be stored in system variables should be compensated for with the error (positional deviation) of the servo system.



> **NOTE**
> 1 Specify only a single axis with the torque limit skip command. If no axis is specified or an attempt is made to specify more than one, alarm PS0369 is issued.
> 2 Do not issue the torque limit skip command in G41 or G42 mode. Otherwise, alarm PS0035 is issued.
> 3 A torque limit arrival signal is output regardless of the torque limit skip command.
> 4 Do not issue the torque limit skip command for an axis being synchronized with synchronization control (such as synchronization control or electronic gear box (M series)).
> 5 Do not specify the torque limit skip command in a continuous block.
> 6 The higher the movement speed, the larger the error between the position at which the machine stops and the position at which a skip is actually detected. Also, the error increases as the speed is varied during movement. Do not vary the speed with override and so on.

5 FEED FUNCTIONS

Chapter 5, "FEED FUNCTIONS", consists of the following sections:

5.1 OVERVIEW

The feed functions control the feedrate of the tool. The following two feed functions are available:

- Feed functions

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

- Override

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

- Automatic acceleration/deceleration

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.1(a)).

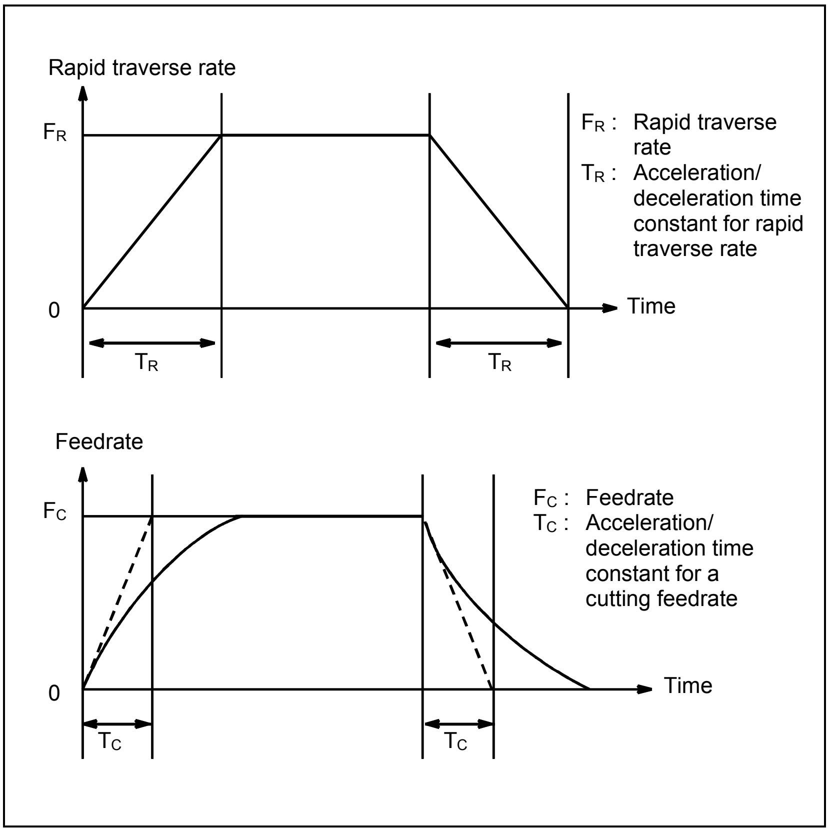


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- Tool path in a cutting feed

When the movement direction changes between a specified block and the next block during cutting feed, the tool path may be rounded because of the relationship between the time constant and feedrate (Fig. 5.1(b)).

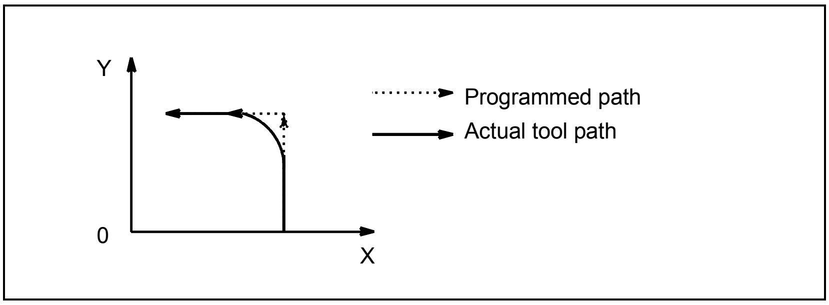


Fig. 5.1 (b) Example of tool path between two blocks

In circular interpolation, a radial error occurs (Fig. 5.1(c)).

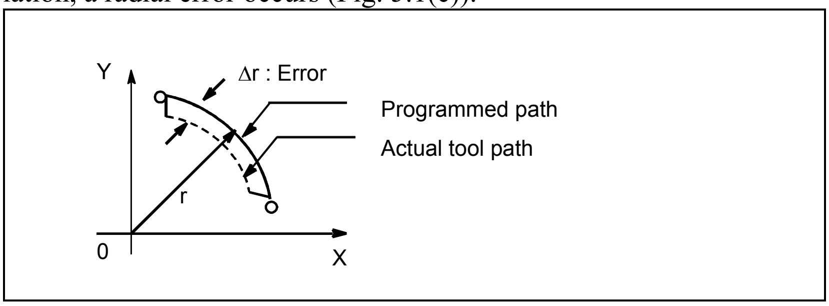


Fig. 5.1 (c) Example of radial error in circular interpolation

The rounded-corner path shown in Fig. 5.1(b) and the error shown in Fig. 5.1 (c) depend on the feedrate. So, the feedrate needs to be controlled for the tool to move as programmed.

5.2 RAPID TRAVERSE

Format

G00 IP_ ;

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

Explanation

The positioning command (G00) positions the tool by rapid traverse. 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 machine tool builder (in-position check).
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse feedrate need be programmed.
The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel: F0, 25%, 50%, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
It is also possible to select the rapid traverse override in steps of 1% or 0.1% in the range of 0% to 100%.
For detailed information, refer to the appropriate manual of the machine tool builder.

5.3 CUTTING FEED

Overview

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.
In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.

M

Four modes of specification are available:

1. Feed per minute (G94)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G95)
   After F, specify the amount of feed of the tool per spindle revolution.
3. Inverse time feed (G93)
   Specify the inverse time (FRN) after F.
4. One-digit F code feed
   Specify a desired one-digit number after F. Then, the feedrate set with the CNC for that number is set.

T

Two modes of specification are available:

1. Feed per minute (G98)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G99)
   After F, specify the amount of feed of the tool per spindle revolution.

Format

M

Feed per minute

G94 ; G code (group 05) for feed per minute

F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution

G95 ; G code (group 05) for feed per revolution

F_ ; Feedrate command (mm/rev or inch/rev)

Inverse time feed (G93)

G93 ; Inverse time feed command G code (05 group)

F_ ; Feedrate command (1/min)

One-digit F code feed

Fn ;

n : Number from 1 to 9

T

Feed per minute

G98 ; G code (group 05) for feed per minute

F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution

G99 ; G code (group 05) for feed per revolution

F_ ; Feedrate command (mm/rev or inch/rev)

Explanation

- Direction of the cutting feedrate

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

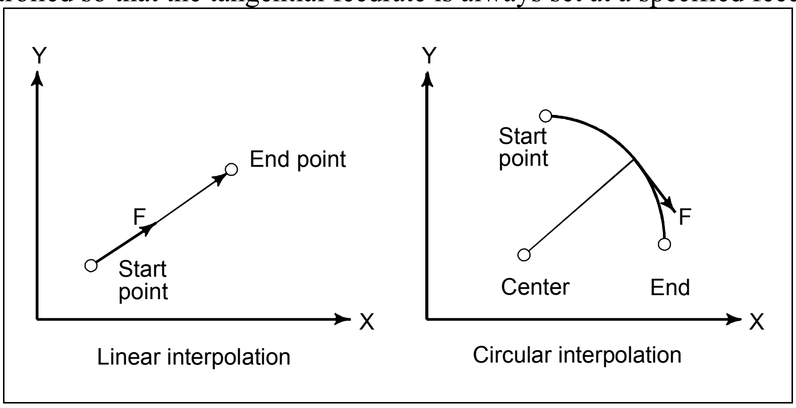


Fig. 5.3 (a) Tangential feedrate (F)

- Feed per minute

After specifying G code for feed per minute (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G code for feed per minute is a modal code. Once a G code for feed per minute is specified, it is valid until G code for feed per revolution (feed per revolution) is specified.

M

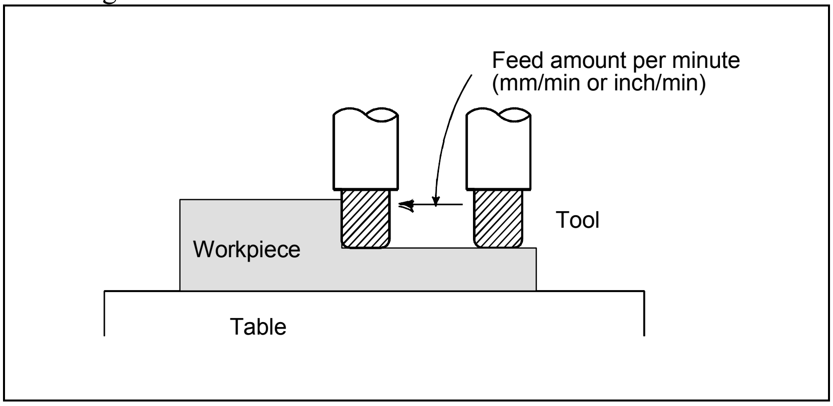
At power-on, the feed per minute mode is set.

T

Either the feed per minute mode or the feed per revolution mode is selected during power-on is determined by bit 4 (FPM) of parameter No. 3402.

An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

- For milling machining



- For lathe cutting

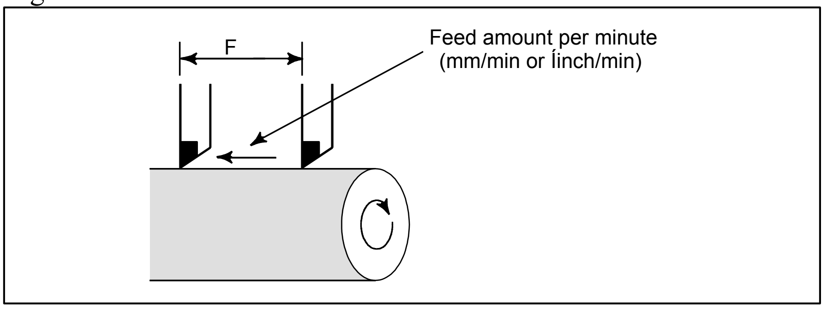


Fig. 5.3 (b) Feed per minute

> ⚠ **CAUTION**
> No override can be used for some commands such as for threading.

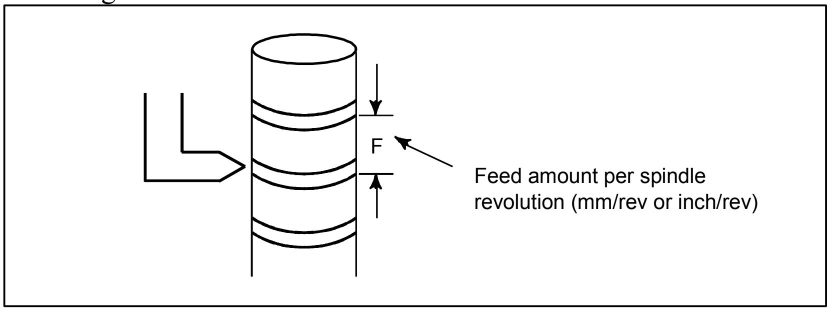
- Feed per revolution

After specifying G code for feed per revolution (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. G code for feed per revolution is a modal code. Once a G code for feed per revolution is specified, it is valid until G code for feed per minute (feed per minute) is specified.

An override from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

If bit 0 (NPC) of parameter No. 1402 has been set to 1, feed per revolution commands can be specified even when a position coder is not being used. (The CNC converts feed per revolution commands to feed per minute commands.)

- For milling machining



- For lathe cutting

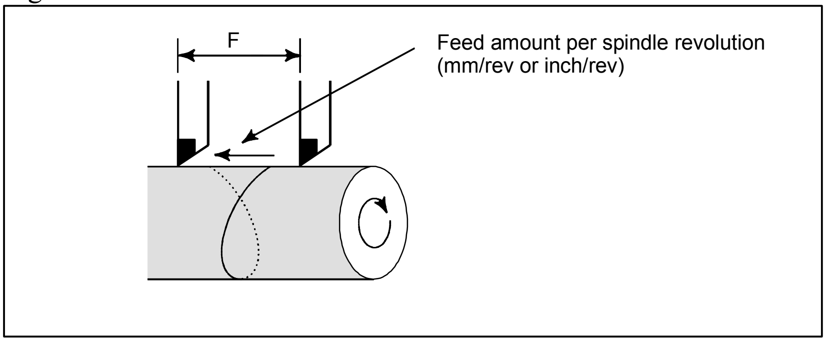


Fig. 5.3 (c) Feed per revolution

> ⚠ **CAUTION**
> When the speed of the spindle is low, feedrate fluctuation may occur.
> The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

M

- Inverse time feed

When G code for inverse time feed is specified, the inverse time specification mode (G93 mode) is set. Specify the inverse time (FRN) with an F code.
A value from 0.001 to 9999.999 can be specified as FRN, regardless of whether the input mode is inches or metric, or the increment system is IS-B or IS-C.

F code specification value	FRN
F1	0.001
F1 (*1)	1.000
F1.0	1.000
F9999999	9999.999
F9999 (*1)	9999.000
F9999.999	9999.999

> **NOTE**
> *1 Value specified in fixed-point format with bit 0 (DPI) of parameter No. 3401 set to 1

G code for inverse time feed is a modal G code and belongs to group 05 (includes G code for feed per revolution and G code for feed per minute).
When an F value is specified in inverse time specification mode and the feedrate exceeds the maximum cutting feedrate, the feedrate is clamped to the maximum cutting feedrate.
In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arc radius. This means that actual machining time is longer when the arc radius is longer than the arc distance and shorter when the arc radius is shorter than the arc distance. Inverse time feed can also be used for cutting feed in a canned cycle.

NOTE

1 In the inverse time specification mode, an F code is not handled as a modal code and therefore needs to be specified in each block. If an F code is not specified, alarm PS0011 (FEED ZERO (COMMAND)) is issued.
2 When F0 is specified in inverse time specification mode, alarm PS0011 (FEED ZERO (COMMAND)) is issued.
3 Inverse time feed cannot be used when PMC axis control is in effect.
4 If the calculated cutting feedrate is smaller than the allowable range, alarm PS0011 (FEED ZERO (COMMAND)) is issued.

Example

- **For linear interpolation (G01)**

$$FRN = \frac{1}{time(min)} = \frac{feedrate}{distance}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Distance: mm (for metric input)
inch (for inch input)

- To end a block in 1 (min)

$$FRN = \frac{1}{time(\text{min})} = \frac{1}{1(\text{min})} = 1$$

Specify F1.0.

- To end a block in 10 (sec)

$$FRN = \frac{1}{time(\text{sec})/60} = \frac{1}{10/60(\text{sec})} = 6$$

Specify F6.0.

- To find the movement time required when F0.5 is specified

$$TIME(\text{min}) = \frac{1}{FRN} = \frac{1}{0.5} = 2$$

2 (min) is required.

- To find the movement time required when F10.0 is specified

$$TIME(\text{min}) = \frac{1 \times 60}{FRN} = \frac{60}{10} = 6$$

6 (sec) is required.

- **For circular interpolation (G02, G03)**

$$FRN = \frac{1}{time(\text{min})} = \frac{feedrate}{arcradius}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Arc radius: mm (for metric input)
inch (for inch input)

NOTE

In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arcadias.

M

- One-digit F code feed

When a one-digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameter Nos. 1451 to 1459 is used. When F0 is specified, the rapid traverse rate is applied.
The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing one-digit F feedrate on the machine operator's panel, then by rotating the manual pulse generator.
The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{F\,\max}{100X}$$

Fmax : Feedrate upper limit for F1-F4 set by parameter (No.1460), or feedrate upper limit for F5-F9 set by parameter (No.1461)
X : Any value of 1-127 set by parameter No.1450
The feedrate set or altered is kept even while the power is off. The current feed rate is displayed on the LCD screen.

- Cutting feedrate clamp

Parameter No. 1430 can be used to specify the maximum cutting feedrate for each axis. When the cutting feedrate along an axis exceeds the maximum feedrate for the axis as a result of interpolation, the cutting feedrate is clamped to the maximum feedrate.

Reference

See Appendix D for range of feedrate command value.

5.4 CUTTING FEEDRATE CONTROL

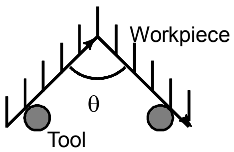
Cutting feedrate can be controlled, as indicated in Table 5.4 (a).

Table 5.4 (a) Cutting Feedrate Control

<table>
<tr><th colspan="2">Function name</th><th>G code</th><th>Validity of G code</th><th>Description</th></tr>
<tr><td colspan="2">Exact stop</td><td>G09</td><td>This function is valid for specified blocks only.</td><td>The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.</td></tr>
<tr><td colspan="2">Exact stop mode</td><td>G61</td><td>Once specified, this function is valid until G62 (M series), G63, or G64 is specified.</td><td>The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.</td></tr>
<tr><td colspan="2">Cutting mode</td><td>G64</td><td>Once specified, this function is valid until G61, G62 (M series), or G63 is specified.</td><td>The tool is not decelerated at the end point of a block, but the next block is executed.</td></tr>
<tr><td colspan="2">Tapping mode</td><td>G63</td><td>Once specified, this function is valid until G61, G62 (M series), or G64 is specified.</td><td>The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.</td></tr>
<tr><td rowspan="2">Automatic corner override (M series)</td><td>Automatic override for inner corners</td><td>G62 (M series)</td><td>Once specified, this function is valid until G61, G63, or G64 is specified.</td><td>When the tool moves along an inner corner during tool radius compensation, override is applied to the cutting feedrate to suppress the amount of cutting per unit of time so that a good surface finish can be produced.</td></tr>
<tr><td>Internal circular cutting feedrate change</td><td>- (M series)</td><td>This function is valid in the tool radius compensation mode, regardless of the G code.</td><td>The internal circular cutting feedrate is changed.</td></tr>
</table>

NOTE

1. The purpose of in-position check is to check that the servo motor has reached within a specified range (specified with a parameter by the machine tool builder). In-position check is not performed when bit 5 (NCI) of parameter No. 1601 is set to 1.
2. Inner corner angle θ: $2^\circ < \theta \leq \alpha \leq 178^\circ$
   (α is a set value)



Format

Exact stop	**G09 IP_ ;**
Exact stop mode	**G61 ;**
Cutting mode	**G64 ;**
Tapping mode	**G63 ;**
Automatic corner override	**G62 ;**

5.4.1 Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63)

Explanation

The inter-block paths followed by the tool in the exact stop mode, cutting mode, and tapping mode are different (Fig. 5.4.1 (a)).

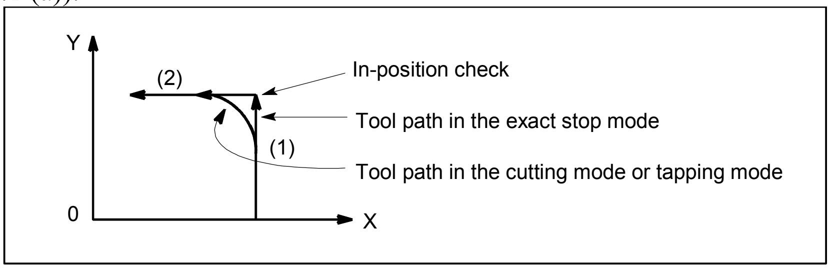


Fig. 5.4.1 (a) Example of tool paths from block (1) to block (2)

⚠ **CAUTION**

The cutting mode (G64 mode) is set at power-on or system clear.

5.4.2 Automatic Corner Override (M Series)

M

When tool radius compensation is performed, the movement of the tool is automatically decelerated at an inner corner and internal circular area. This reduces the load on the tool and produces a smoothly machined surface.

5.4.2.1 Automatic override for inner corners (G62)

M

Explanation

- Override condition

When G62 is specified, and the tool path with tool radius compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.
There are four types of inner corners (Fig. 5.4.2(a)).
2°≤θ≤θp≤178° in Fig. 5.4.2(a)qp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001° or less.

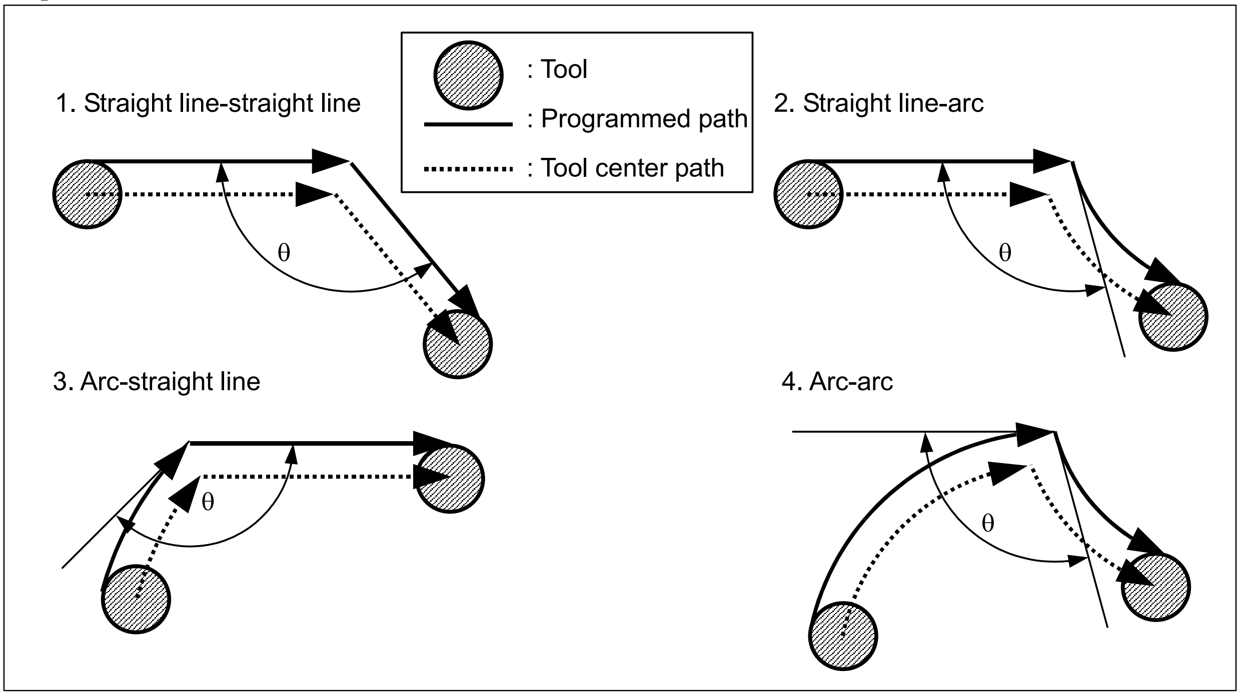


Fig. 5.4.2(a) Inner corner

- Override range

When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances Ls and Le, where the feedrate is overridden, are distances from points on the tool center path to the corner (Fig. 5.4.2(b), Fig. 5.4.2(c), Fig. 5.4.2(d)). Ls and Le are set with parameter Nos. 1713 and 1714.

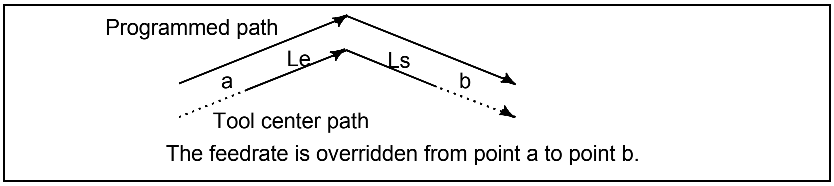


Fig. 5.4.2.1 (b) Override Range (Straight Line to Straight Line)

When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants (Fig. 5.4.2(c)).

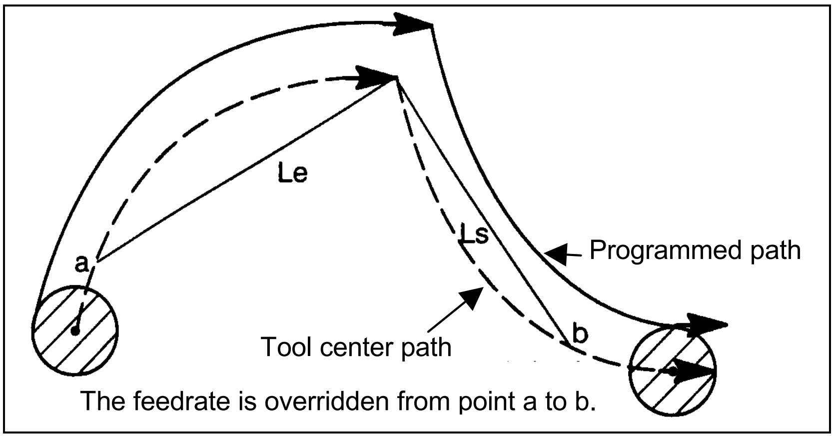


Fig. 5.4.2(c) Override Range (Arc to Arc)

Regarding program (2) of an arc, the feedrate is overridden from point a to point b and from point c to point d (Fig. 5.4.2(d)).

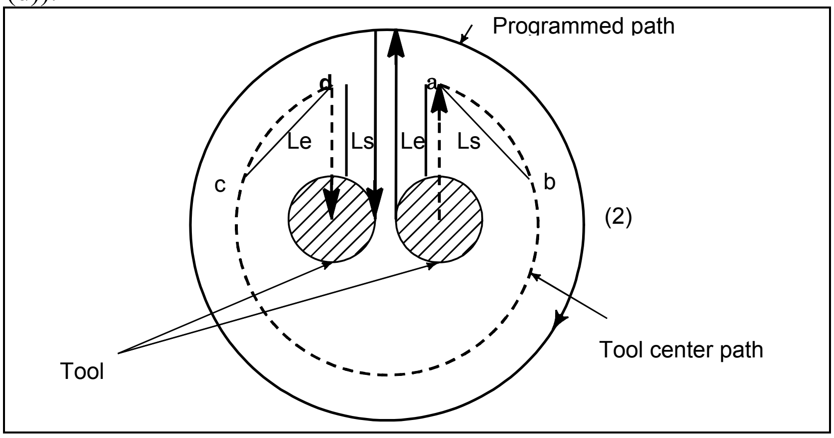


Fig. 5.4.2(d) Override Range (Straight Line to Arc, Arc to Straight Line)

- Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run and one-digit F code feed specification.
In the feed per minute mode, the actual feedrate is as follows:

$F = (\text{automatic override for inner corners}) \times (\text{feedrate override})$

Limitation

- Acceleration/deceleration before interpolation

Override for inner corners is disabled during acceleration/deceleration before interpolation.

- Start-up/G41, G42

Override for inner corners is disabled if the corner is preceded by a start-up block or followed by a block including G41 or G42.

- Offset

Override for inner corners is not performed if the offset is zero.

5.4.2.2 Internal circular cutting feedrate change

M

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below (Fig. 5.4.2(e)). This function is valid in the tool radius compensation mode, regardless of the G62 code.

$$F = \frac{Rc}{Rp}$$

Rc : Tool center path radius
Rp : Programmed radius
It is also valid for the dry run and the one-digit F code feed command.

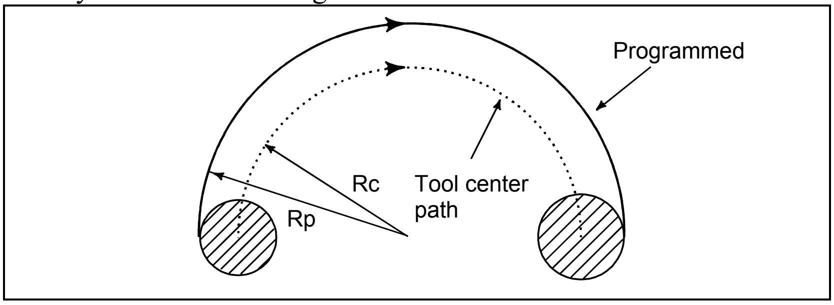


Fig. 5.4.2(e) Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp ≒ 0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp≤MDR, the feedrate of the tool is (F×MDR).
When parameter No. 1710 is 0, the minimum deceleration ratio (MDR) is 100%.

> ⚠ **CAUTION**
> When internal circular cutting must be performed together with override for inner corners, the feedrate of the tool is as follows:
> $$F \times \frac{Rc}{Rp} \times (\text{override for the inner corners}) \times (\text{feedrate override})$$

5.5 FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS

Overview

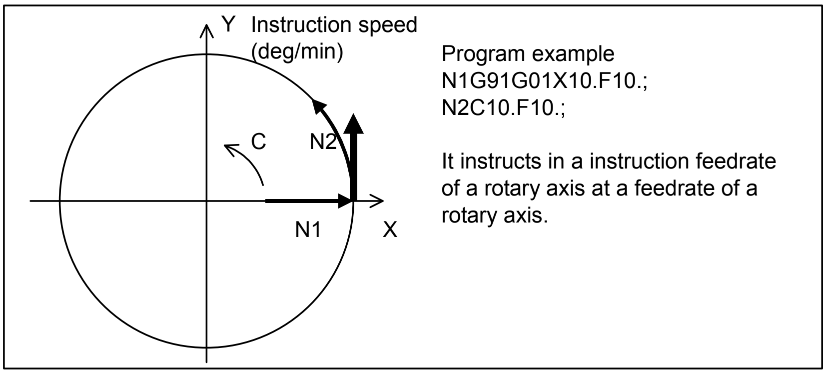
This function acquires movement feedrate on imaginary circle by synthetic movement distance is calculated from movement distance of a rotary axis by using instruction angle and the parameter of an imaginary radius (No.1465).
Then, movement feedrate on imaginary circle is feedrate of a rotary axis.

Explanation

Cutting feedrate

- Conventional method

In linear interpolation of a linear axis and a rotary axis, 1deg in movement angle of a rotary axis is interpolated as 1mm in movement distance (1inch at inch input).



Feedrate of liner axis(X axis) $F_X = F \times \dfrac{\Delta X}{L} \ (mm/min)$

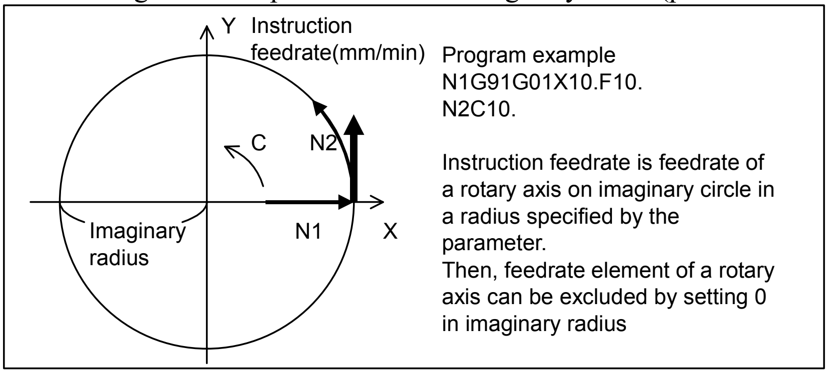
Feedrate of rotary axis(C axis) $F_C = F \times \dfrac{\Delta C}{L} \ (deg/min)$

Synthetic movement distance $L = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2 + \Delta B^2 + \Delta C^2} \ (mm)$

Movement time $T = \dfrac{L}{F} \ (min)$

- Feedrate instruction on imaginary circle of a rotary axis

In this function, synthetic movement distance is obtained based on movement distance of a rotary axis requested from instruction angle and the parameter of an imaginary radius(parameter No.1465).



Feedrate of liner axis(X axis) $F_X = F \times \dfrac{\Delta X}{L'} \ (mm/min)$

Feedrate of rotary axis(C axis) $F_C = F \times \dfrac{\Delta C}{L'} \ (deg/min)$

Synthetic movement distance $L' = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2 + \left(\dfrac{\pi \times l_B \times \Delta B}{180}\right)^2 + \left(\dfrac{\pi \times l_C \times \Delta C}{180}\right)^2} \ (mm)$

Movement time $T' = \dfrac{L'}{F} \ (min)$

l_B , l_C : imaginary radius(parameter No.1465)

It becomes feedrate from which movement feedrate on imaginary circle is instructed by this. <Example 1 reference>

In this function, feedrate of a axis becomes L/L' times for the feedrate displayed on the NC screen from difference of the method of obtaining movement distance. Especially, a movement of a axis quickens when small value is set to an imaginary radius. Note input of the parameter enough.

Cutting feedrate is clamped based on the maximum cutting feedrate parameter (No.1430) and feedrate of an actual axis (data before this function is converted). Therefore, it is possible to instruct at feedrate more than setting the maximum cutting feedrate by setting big value in an imaginary radius (parameter No.1465). When small value is set in an imaginary radius, it is clamped at the feedrate following setting the maximum cutting feedrate.
Moreover, dry run becomes effective as for this function, too.

Advanced preview control (T series) / AI advanced preview control (M series) / AI contour control (M series)

Advanced preview control (T series) / AI advanced preview control (M series) / AI contour control (M series) is done to movement feedrate on imaginary circle.
Therefore, It is likely not to become feedrate in the calculation in this function by feedrate control of advanced preview control (T series) / AI advanced preview control (M series) / AI contour control (M series). Then, feedrate of advanced preview control (T series) / AI advanced preview control (M series) / AI contour control (M series) is clamped at parameter(No.1432). Moreover, it is clamped at parameter(No.8465), When the parameter(No.8465) is not 0.

0mm in imaginary radius

When an imaginary radius is assumed 0mm, synthesized distance is as follows because the movement distance of a rotary axis becomes 0mm.

$$L' = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$$

A movement feedrate of a linear axis can be instruction feedrate F by excluding feedrate element of a rotary axis.<Example 2 reference>
Moreover, it moves at the maximum cutting feedrate in case of this setting and instruction only in a rotary axis.

Examples

When the following block is instructed on IS-B,
G91 G01 C10. F10. ;

(1) The calculation is as follows, when 10.000(10mm) is set in an imaginary radius(parameter No.1465).

$$L' = \sqrt{\left(\frac{\pi \times l_C \times \Delta B}{180}\right)^2} = \sqrt{\left(\frac{\pi \times 10_{(mm)} \times 10_{(deg)}}{180}\right)^2} = 1.7453292\cdots_{(mm)}$$

$$F_C = 10_{(mm/\min)} \times \frac{10_{(deg)}}{1.7453292\cdots_{(mm)}} = 57.2957795\cdots_{(deg/min)}$$

$$T' = \frac{L'}{F} = \frac{1.7453292\cdots_{(mm)}}{10_{(mm/min)}} = 0.17453292\cdots_{(min)} = 10.4719755\cdots_{(sec)}$$

Therefore, the movement time becomes about 10.472(sec), and the rotation feedrate becomes about 57.296(deg/min). The feedrate on 10.000mm in an imaginary radius becomes 10.000mm/min at instruction feedrate in Fig.5.5(a).

(2) The calculation is as follows, when 36.000(36mm) is set in an imaginary radius(parameter No.1465).

$$L' = \sqrt{\left(\frac{\pi \times l_C \times \Delta B}{180}\right)^2} = \sqrt{\left(\frac{\pi \times 36_{(mm)} \times 10_{(deg)}}{180}\right)^2} = 6.28318530\cdots_{(mm)}$$

$$F_C = 10_{(mm/min)} \times \frac{10_{(deg)}}{6.28318530\cdots_{(mm)}} = 15.9154943\cdots_{(deg/min)}$$

$$T' = \frac{L'}{F} = \frac{6.28318530\cdots_{(mm)}}{10_{(mm/min)}} = 0.628318530\cdots_{(min)} = 37.6991118\cdots_{(sec)}$$

Therefore, the movement time becomes about 37.700(sec), and the rotation feedrate becomes about 15.915(deg/min). The feedrate on 36.000mm in an imaginary radius becomes 10.000mm/min at instruction feedrate in Fig.5.5(a).

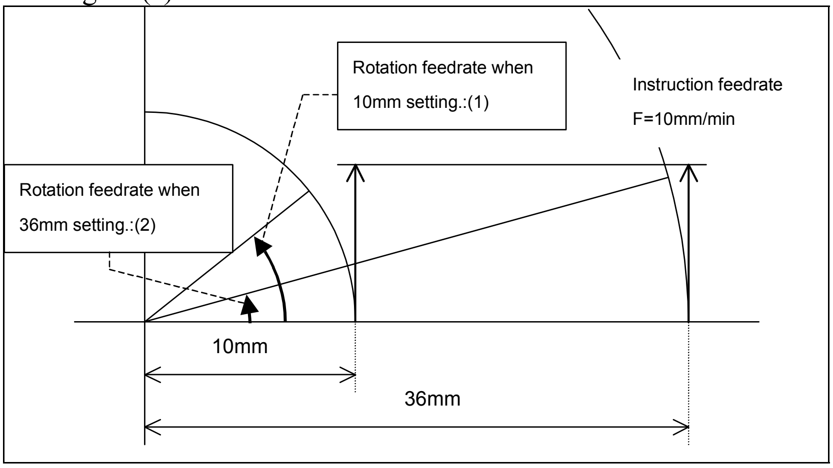


Fig. 5.5 (a)

Limitation

This function corresponds only the linear interpolation(G01).
However, it doesn't correspond to the following functions.

- Feed per revolution
- Cylindrical interpolation
- Axis control by PMC

M

- Inverse time feed
- Normal direction control

T

- Polar coordinate interpolation

NOTE

1. When the parameter ROTx (No.1006#0) and the parameter RFDx (No.1408#0) are 1, this function becomes effective.
2. The parameter RFDx (No.1408#0) and an imaginary radius (parameter No.1465) of this function can be rewriting by the programmable parameter input(G10).
3. It moves at the maximum cutting feedrate when this function effectively and sets 0 in an imaginary radius (parameter No.1465) and it instructs only a rotary axis.
4. Note setting RFDx (No.1408#0) and an imaginary radius (parameter No.1465) enough. Especially, a movement of axis quickens compared with this function unused when small value is set to an imaginary radius.
5. In this function, the same value as the parameter value (No.1408, No.1465) of a master axis is used with a slave axis, when it uses axis synchronous control.

5.6 DWELL

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)

Explanation

By specifying a dwell, the execution of the next block is delayed by the specified time. (Dwell per second)
By setting bit 1 (DWL) of parameter No. 3405 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)

Table 5.6 (a) Command value range of the dwell time (Command by X or U)

Increment system	Command value range	Dwell time unit
IS-A	0.01 to 999999.99	sec or rev
IS-B	0.001 to 99999.999	
IS-C	0.0001 to 9999.9999	

Table 5.6 (b) Command value range of the dwell time (Command by P)

Increment system	Command value range	Dwell time unit
IS-A	1 to 99999999	0.01 sec or rev
IS-B	1 to 99999999	0.001 sec or rev
IS-C	1 to 99999999	0.0001 sec or rev

In the case of dwell per second, the specification unit for dwell time specified with P can be fixed at 0.001 second by setting bit 7 (DWT) of parameter No. 1015 to 1.

NOTE

1 When X, U, or P is specified without a decimal point, the specification unit does not depend on inch/metric input. Depending on whether the X-axis is present, the following increment system is used:
- When the X-axis is present
  The increment system of the X-axis is used.
- When the X-axis is not present
  The increment system of the reference axis is used.

2 When P is specified, bit 7 (IPR) of parameter No. 1004 exercises no influence.

M

Specify dwell also to make an exact check in the cutting mode (G64 mode).
If the specification of P and X is omitted, an exact stop occurs.

6 REFERENCE POSITION

A CNC machine tool has a special position where, generally, the tool is exchanged or the coordinate system is set, as described later. This position is referred to as a reference position.

Chapter 6, "REFERENCE POSITION", consists of the following sections:

6.1 REFERENCE POSITION RETURN

Overview

- Reference position

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function.
For example, the reference position is used as a position at which tools are automatically changed. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters (No. 1240 to 1243).

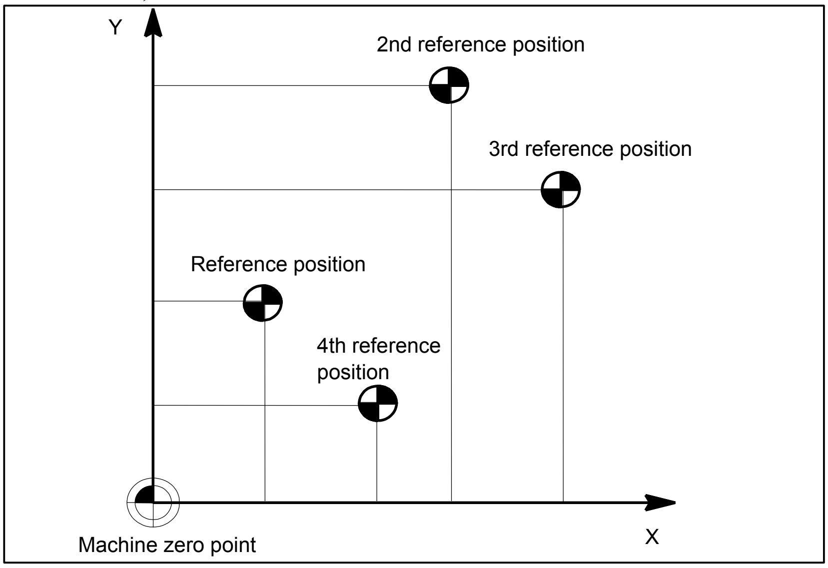


Fig. 6.1 (a) Machine zero point and reference positions

- Automatic reference position return (G28) and movement from the reference position (G29)

The automatic reference position return (G28) function returns tools automatically to the reference position via an intermediate position along a specified axis. When reference position return is completed, the lamp for indicating the completion of reference position return goes on.

M

The return from reference position (G29) function moves tools from the reference position to a specified position via an intermediate position along a specified axis.

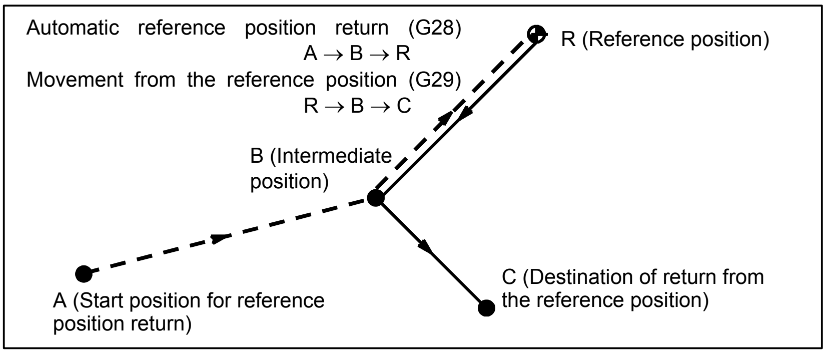


Fig. 6.1 (b) Reference position return and return form the reference position

> **NOTE**
> Movement from the reference position (G29) is enabled only for the M series.

- Reference position return check (G27)

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 (PS0092) "ZERO RETURN CHECK (G27) ERROR" is issued.
When no movement was made along the axis, whether the current position is the reference position is checked.

Format

- Automatic reference position return and 2nd/3rd/4th reference position return

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

- Reference position return check

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)

Explanation

- Automatic reference position return (G28)

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis. Therefore, for safety, the compensation functions, such as the cutter compensation, tool nose radius compensation, tool length compensation, and tool offset, should be cancelled before executing this command.
The coordinates for the intermediate position are stored in the CNC for the axes for which a value is specified in a G28 block. For the other axes, the previously specified coordinates are used.

(Example)

N1 G28 X40.0 ;
(The tool moves to the reference position along the X-axis and the intermediate position (X40.0) is stored.)
N2 G28 Y60.0 ;
(The tool moves to the reference position along the Y-axis and the intermediate position (Y60.0) is stored.)
N3 G29 X10.0 Y20.0 ;
(The tool moves to the position specified with G29 via the intermediate position (X40.0 Y60.0) previously specified with G28 along the X-axis and Y-axis.)

- 2nd, 3rd, and 4th reference position return (G30)

The 2nd, 3rd, and 4th reference position return (G30) function can be used after the reference positions are established.
The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.

M

- Movement from the reference position (G29)

This function is executed after the tool is returned to the reference position by G28 or G30.
For incremental programming, the command value specifies the incremental value from the intermediate point.
The tool moves to the intermediate and specified positions at the feedrate specified with a parameter.
When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to the commanded position through the intermediate point which has been shifted to the new coordinate system.
The same operations are performed also for G30 command.
After the power is turned on, an alarm (PS0305) is issued if an attempt is made to execute G29 (movement from the reference position) before G28 (automatic reference position return) or G30 (2nd, 3rd, and 4th reference position return) is executed.

- Reference position return check (G27)

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the lamp for indicating the completion of reference position return lights up.
When the tool returns to the reference position along only one axis, the lamp for the axis for indicating the completion of reference position return lights up.
After positioning, if the tool has not reached the reference position along a specified axis, an alarm (PS0092) "ZERO RETURN CHECK (G27) ERROR" is issued.
When no movement was made along the axis, whether the current position is the reference position is checked.

- Setting of the reference position return feedrate

Before a coordinate system is established with the first reference position return after power-on, the manual and automatic reference position return feedrates and automatic rapid traverse rate conform to the setting of parameter No. 1428 for each axis.
After a reference position is established upon the completion of reference position return, the manual reference position return feedrate conforms to the setting of the parameter No. 1428 for each axis.

NOTE

1 To this feedrate, a rapid traverse override (F0,25%,50%,100%) is applied, for which the setting is 100%.
2 After a reference position has been established upon the completion of reference position return, the automatic reference position return feedrate will conform to the ordinary rapid traverse rate.
3 When a value is set for parameter No. 1428, the feedrates conform to the parameter settings shown below.

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return (*1)	No.1428	No.1428 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

1420: Rapid traverse rate
1423: Jog feedrate
1424: Manual rapid traverse rate
1428: Reference position return feedrate

When parameter No. 1428 is set to 0, the feedrates conform to the parameter settings shown below.

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return (*1)	No.1424	No.1424 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

*1 By using JZR (bit 2 of parameter No. 1401), the manual reference position return feedrate can always be set as a jog feedrate.
*2 When RPD (bit 0 of parameter No. 1401) is 1, the setting of parameter No. 1424 (manual rapid traverse rate) is used.
When the setting of parameter No. 1424 (manual rapid traverse rate) is 0, parameter No. 1420 (rapid traverse rate) is used.
*3 When reference position return without dogs is performed in rapid traverse mode, or when manual reference position return is performed in rapid traverse mode regardless of deceleration dogs after a reference position is established, the reference position return feedrate for each of these functions (setting of DLF (bit 1 of parameter No. 1404)) is used.

Limitation

- Status the machine lock being turned on

The lamp for indicating the completion of reference position return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a reference position return check command is specified.

- When automatic reference position return (G28) is executed if no reference position is established

When automatic reference position return (G28) is executed if no reference position is established, movement from the intermediate position in a reference position direction is the same as that in manual reference position return.
(This movement is referred to as a low-speed type of automatic reference position return (G28).)
In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006). Therefore the specified intermediate position must be a position to which reference position return is possible.

> **NOTE**
> When automatic reference position return (G28) is executed after a reference position is established, positioning is performed from the intermediate position to the reference position. This movement is referred to as a high-speed type of automatic reference position return (G28).

- Reference position return check in an offset mode

In an offset mode, the position to be reached by the reference position return check is the position obtained by adding the offset value.
Therefore, if the position with the offset value added is not the reference position, the lamp for indicating the completion of reference position return does not light up, but an alarm is displayed instead. Accordingly, cancel compensation and specify G27 in ordinary cases.

- Lighting the lamp when the programmed position does not coincide with the reference position

When the machine tool system is an inch system with metric input, the lamp for indicating the completion of reference position return may also light up even if the programmed position is shifted from the reference position by the least setting increment. This is because the least setting increment of the machine tool system is smaller than its least command increment.

Example

G28G90X1000.0Y500.0 ;	(Programs movement from A to B. The tool moves to reference position R via intermediate position B.)
T111 ;	
M06 ;	(Changing the tool at the reference position)
G29X1300.0Y200.0 ;	(Programs movement from B to C. The tool moves from reference position R to C specified with G29 via intermediate position B.)

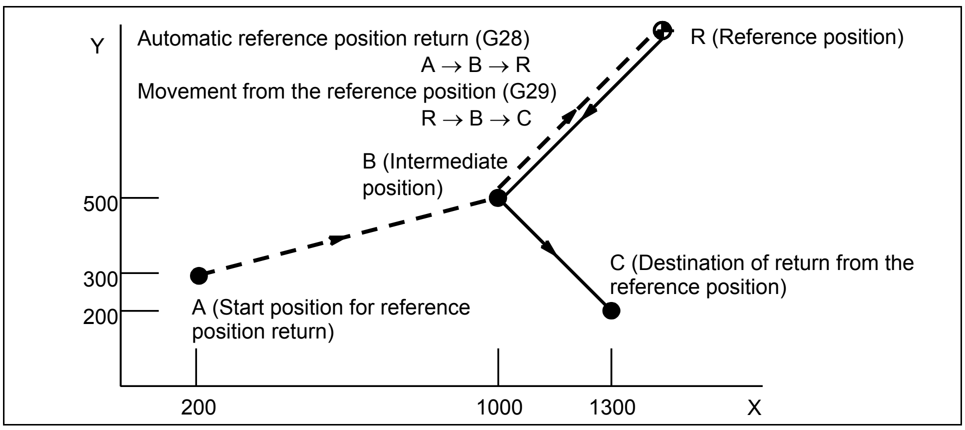
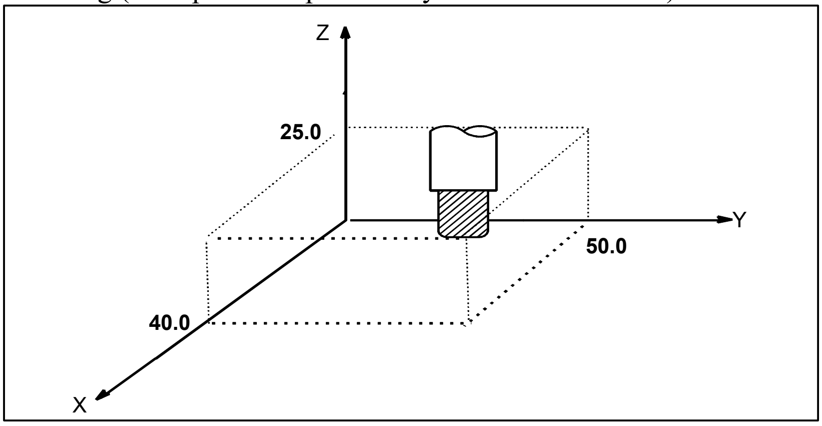


Fig. 6.1 (c) Reference position return and movement from the reference position

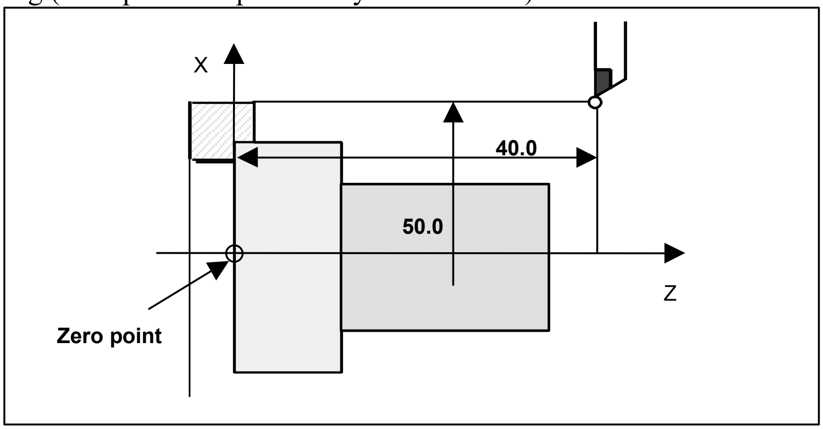
7 COORDINATE SYSTEM

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 using program axes.
When three program axes, the X-axis, Y-axis, and Z-axis, are used, coordinates are specified as follows:
X_Y_Z_
This command is referred to as a dimension word.

- For milling machining (Tool position specified by X40.0Y50.0Z25.0)



- For lathe cutting (Tool position specified by X50.0 Z40.0)



Coordinates are specified in one of following three coordinate systems:
(1) Machine coordinate system
(2) Workpiece coordinate system
(3) Local coordinate system
The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.
A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power-on (see III-3.1). A machine coordinate system, once set, remains unchanged until the power is turned off.
The reference position is not always the origin of the machine coordinate system.
(See "Setting a machine coordinate system" described later.)

Format

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

Explanation

- Selecting a machine coordinate system (G53)

When a command is specified the position on a machine coordinate system, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one-shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command for G53. When an incremental command is specified, the G53 command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

- High-speed G53 function

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.
Specifying P1 in a G53 block enables the high-speed G53 function.

Limitation

- Cancel of the compensation function

When the G53 command is specified, cancel the compensation functions such as the cutter compensation, tool length compensation, tool nose radius compensation, and tool offset.

- G53 specification immediately after power-on

Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute-position detector is attached.

- Blocks in which the high-speed G53 function is usable

The high-speed G53 function is usable in the following combinations of commands:

- G53 → G00
- G53 → G53

The high-speed G53 function is unusable in the following combination of commands:

- G00 → G53

- Specification in the same block

M

Commands G50/G51 (scaling), G50.1/G51.1 (programmable mirror image), and G68/G69 (coordinate system rotation) cannot be specified in the same block where the G53 command is specified.

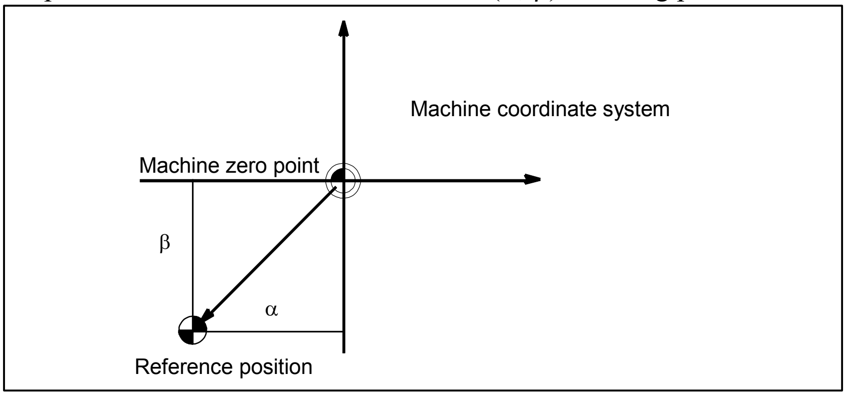
Note

NOTE
G53 is a G code for disabling buffering.

Reference

- Setting a machine coordinate system

When manual reference position return is performed after power-on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.



7.2 WORKPIECE COORDINATE SYSTEM

Overview

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).
A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).
A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

7.2.1 Setting a Workpiece Coordinate System

A workpiece coordinate system can be set using one of three methods:

(1) Method using a workpiece coordinate system setting G code
A workpiece coordinate system is set by specifying a value in the program after a workpiece coordinate system setting G code.
(2) Automatic setting
If bit 0 of parameter ZPR No. 1201 is set to 1, a workpiece coordinate system is automatically set when manual reference position return is performed (see III-3.1.).
When using the workpiece coordinate system function (bit 0 (NWZ) of parameter No. 8136 is 0), this method is disabled.
(3) Method using a workpiece coordinate system selection G code
Six workpiece coordinate systems can be set beforehand using the MDI panel. Program commands G54 to G59 can be used to select the workpiece axis to be used. (see III-12.3.4.)

When using an absolute command, establish the workpiece coordinate system in any of the above ways.

Format

- Setting a workpiece coordinate system

M

G92 IP_ ;

T

G50 IP_ ;

Explanation

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates.

M

If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set. Cutter compensation is cancelled temporarily with G92.

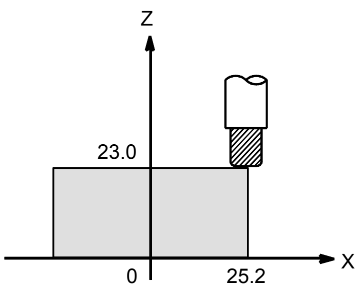
T

If IP_ is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

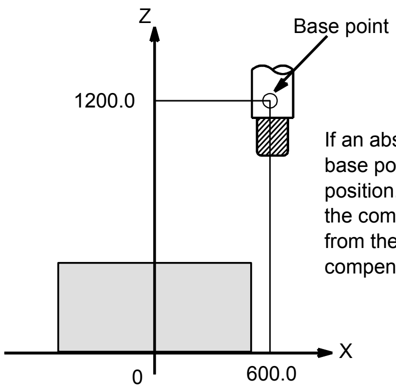
Example

M

(Example 1)
Setting the coordinate system by the G92X25.2Z23.0; command (The tool tip is the start point for the program.)



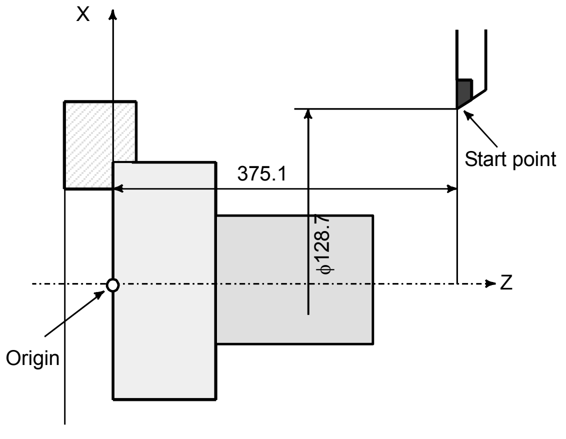
(Example 2)
Setting the coordinate system by the G92X600.0Z1200.0; command (The base point on the tool holder is the start point for the program.)



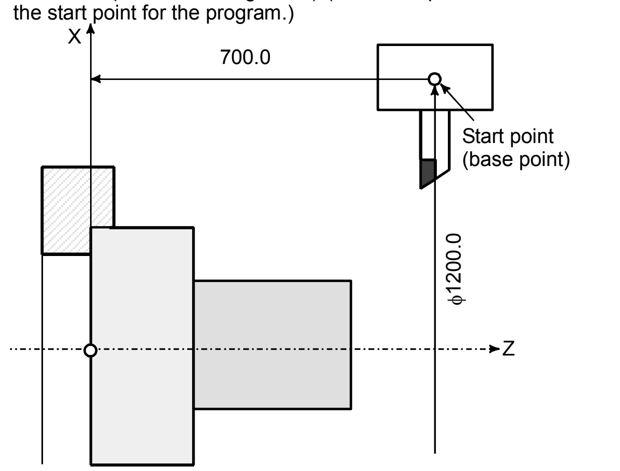
If an absolute command is issued, the base point moves to the commanded position. In order to move the tool tip to the commanded position, the difference from the tool tip to the base point is compensated by tool length offset.

T

(Example 1)
Setting the coordinate system by the G50X128.7Z375.1; command (Diameter designation) (The tool nose is the start point for the program.)



(Example 2)
Setting the coordinate system by the G50X1200.0Z700.0; command (Diameter designation) (The base point on the turret is the start point for the program.)



> ⚠ **CAUTION**
> The set workpiece coordinate system depends on diameter programming or radius programming.

Notes

- **Command for setting a workpiece coordinate system in the tool length compensation mode**

M

Executing a workpiece coordinate system setting G code command (G92) presets a coordinate system in such a way that the specified position will be a pre-compensation position.
However, this G code cannot be used together with a block where tool length compensation vectors vary. If it is used, alarm PS5391 is issued. For example, it cannot be used together with the following blocks.

> **EXAMPLE**
> 1 Block in which G43/G44 is issued
> 2 Block which is in the G43 or G44 mode and in which an H code is issued
> 3 Block which is in the G43 or G44 mode and in which G49 is issued
> 4 Block in which, in the G43 or G44 mode, compensation vectors are canceled using a G code such as G28 or G53 and then resumed again

When presetting a workpiece coordinate system, using the workpiece coordinate system setting G code, do not stop in the previous block to change a tool length compensation offset selected, for example, with the MDI.

7.2.2 Selecting a Workpiece Coordinate System

The user can choose from set workpiece coordinate systems as described below. (For information about the methods of setting, see II-7.2.1.)

(1) Once a workpiece coordinate system is set by a workpiece coordinate system setting G code or by automatic workpiece coordinate system setting, absolute commands indicate positions in the workpiece coordinate system.
(2) Choosing from six workpiece coordinate systems set using the MDI panel

By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

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

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

When bit 2 (G92) of parameter No. 1202 is set to 1, executing the workpiece coordinate system setting G92 code command results in the issue of an alarm PS0010. This is designed to prevent the user from confusing coordinate systems.

> ⚠ **CAUTION**
> The set workpiece origin offset value depends on diameter programming or radius programming.

Example

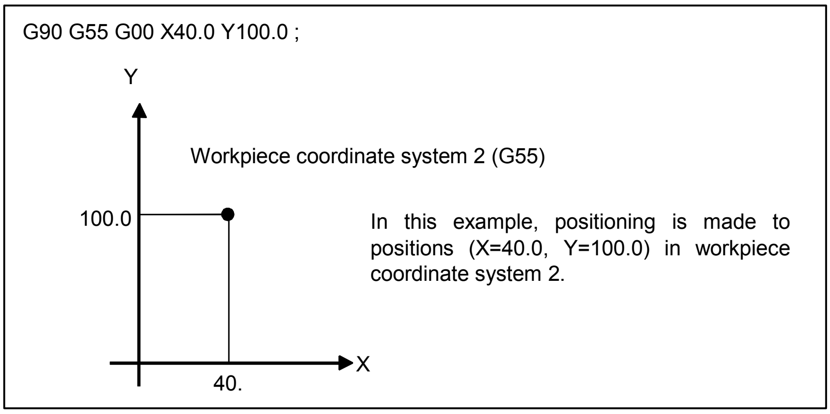


Fig. 7.2.2 (a)

7.2.3 Changing Workpiece Coordinate System

The six workpiece coordinate systems specified with G54 to G59 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 (see III-12.3.4)
(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. Refer to machine tool builder's manual for details.

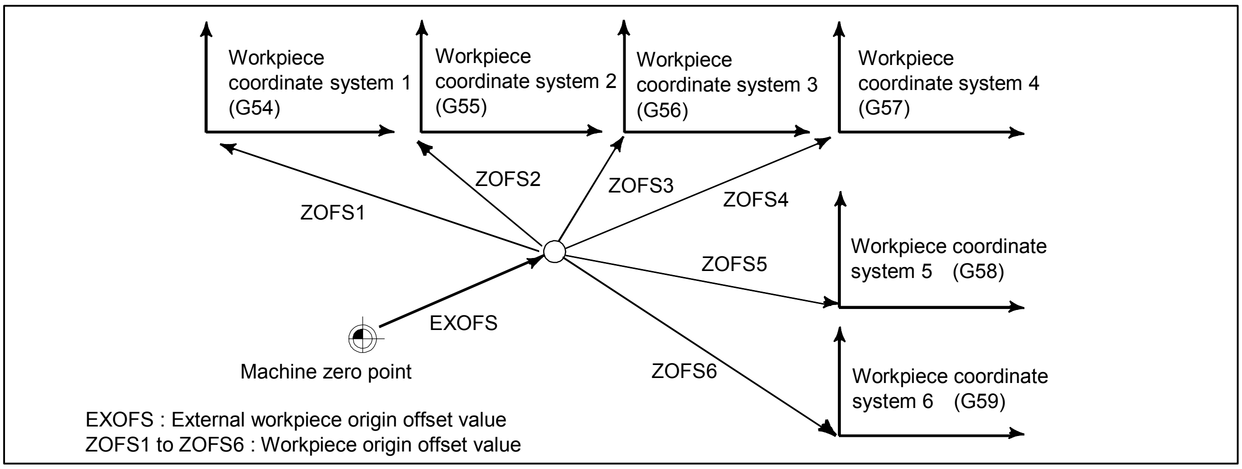


Fig. 7.2.3 (a) Changing an external workpiece origin offset value or workpiece origin offset value

Format

- Changing by inputting programmable data

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).

- Changing by setting a workpiece coordinate system

M

G92 IP_ ;

T

G50 IP_ ;

Explanation

- Changing by inputting programmable data

By specifying a programmable data input G code, the workpiece origin offset value can be changed for each workpiece coordinate system.

- Changing by setting a workpiece coordinate system

By specifying a workpiece coordinate system setting G code, the workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
Then, the amount of coordinate system shift is added to all the workpiece origin offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

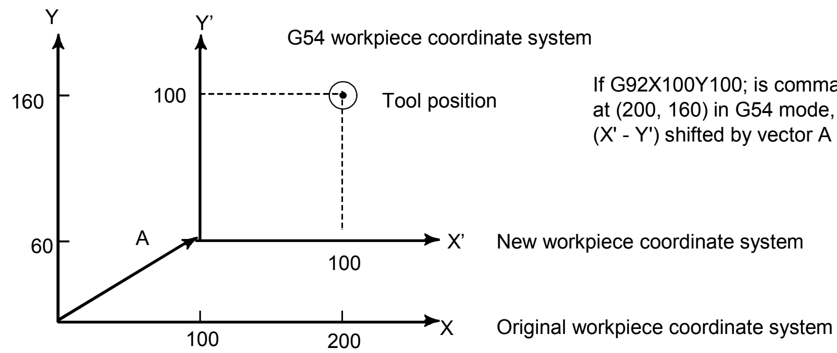
> ⚠ **CAUTION**
> When a coordinate system is set with workpiece coordinate system setting G92 code command after an external workpiece origin offset value is set, the coordinate system is not affected by the external workpiece origin offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Z = 80.0 is set.

T

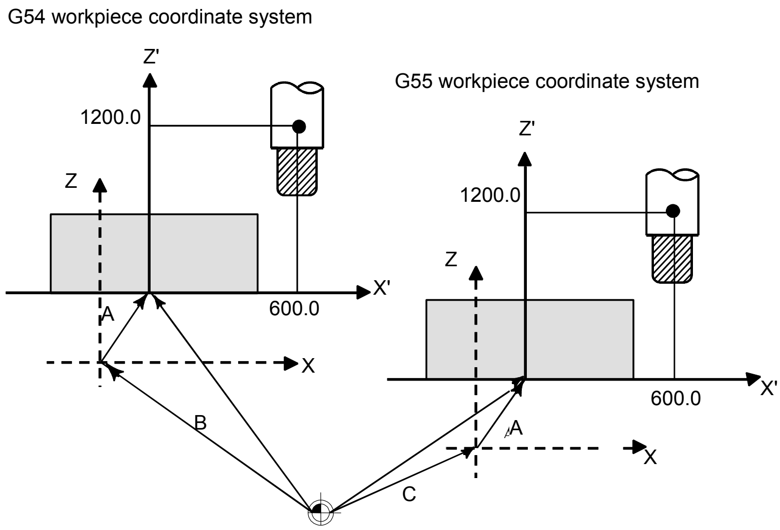
If IP is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. (Coordinate system shift)

Example

M



If G92X100Y100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' - Y') shifted by vector A is created.

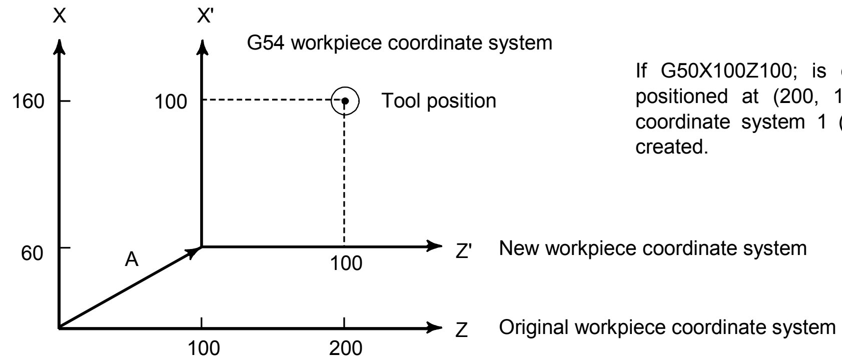


X' - Z' ....New workpiece coordinate system
X - Z ....Original workpiece coordinate system
A : Offset value created by G92
B : Workpiece origin offset value in the G54
C : Workpiece origin offset value in the G55

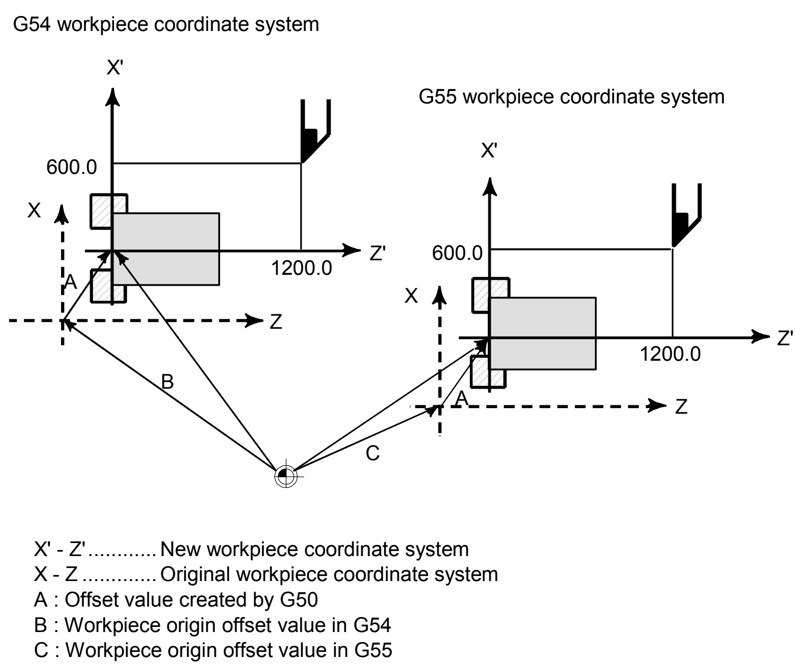
Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black circle on the tool (figure at the left) is at (600.0,1200.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:
G92X600.0Z1200.0;
Also, suppose that pallets are loaded at two different positions. If the relative relationship of the coordinate systems of the pallets at the two positions is correctly set by handling the coordinate systems as the G54 workpiece coordinate system and G55 workpiece coordinate system, a coordinate system shift with G92 in one pallet causes the same coordinate system shift in the other pallet. This means that workpieces on two pallets can be machined with the same program just by specifying G54 or G55.

Example

T



If G50X100Z100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' - Z') shifted by vector A is created.



Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black tool nose point on the tool (figure at the left) is at (600.0,1200.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:

G50X600.0Z1200.0;

Also, suppose that loading is performed at two different places and that the G54 and G55 workpiece coordinate systems are set for these two places. If the relative relationship between the coordinate systems in the two places is correctly set, a coordinate system shift with G50 in one loading place causes the same coordinate system shift in the other loading place. This means that workpieces on the two loading positions can be machined with the same program just by specifying G54 or G55.

7.2.4 Workpiece Coordinate System Preset (G92.1)

The workpiece coordinate system preset function presets a workpiece coordinate system shifted by manual intervention to the pre-shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece origin offset value.

There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command. The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (see III-12.1.4).

Format

M

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.

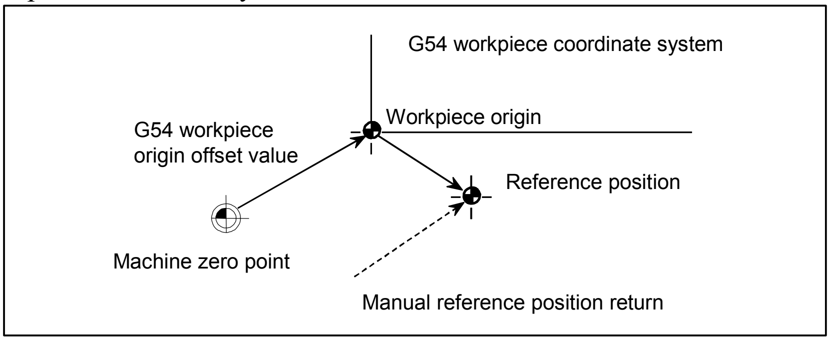
T

G50.3 IP 0 ; (G92.1 IP 0; for G code system B or C)

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.

Explanation

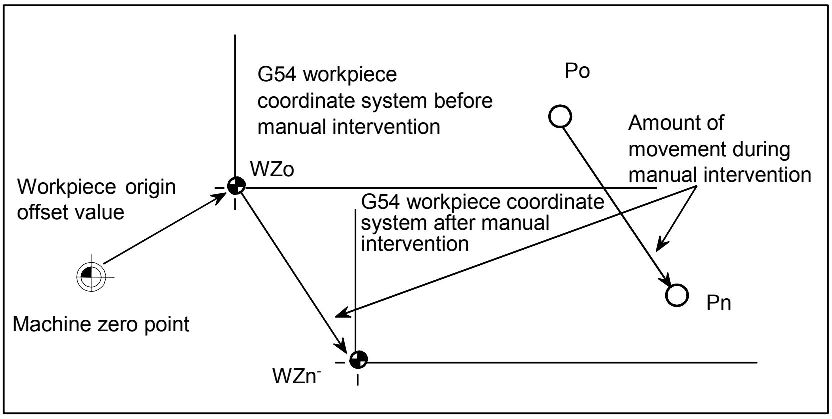
When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece origin offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its origin displaced from the machine zero point by the G54 workpiece origin offset value; the distance from the origin of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.



If an absolute position detector is provided, the workpiece coordinate system automatically set at power-up has its origin displaced from the machine zero point by the G54 workpiece origin offset value. The machine position at the time of power-up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece origin offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed below.

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

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.



In the operation above, a workpiece coordinate system once shifted can be preset using G code (G92.1) specification or MDI operation to a workpiece coordinate system displaced by a workpiece origin offset value from the machine zero point.
Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates as well as absolute coordinates.

Limitation

- Tool radius · tool nose radius compensation, tool length compensation, tool offset

When using the workpiece coordinate system preset function, cancel compensation modes: Tool radius · tool nose radius compensation, tool length compensation, and tool offset. If the function is executed without canceling these modes, compensation vectors are cancelled.

M

- Tool length compensation

When using the workpiece coordinate system preset function, cancel tool length compensation. If the function is executed without canceling these modes, compensation vectors are cancelled.

- Prohibited modes

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, or programmable image is set.

- Program restart

The workpiece coordinate system preset function is not executed during program restart.

7.2.5 Addition of Workpiece Coordinate System Pair (G54.1 or G54) (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.

Format

- Selecting the additional workpiece coordinate systems

G54.1 Pn ; or **G54 Pn ;**

Pn : Codes specifying the additional workpiece coordinate systems
n : 1 to 48

- Setting the workpiece origin offset value in the additional workpiece coordinate systems (G10)

G10 L20 Pn IP_ ;

Pn : Codes specifying the workpiece coordinate system for setting the workpiece origin offset value
n : 1 to 48
IP_ : Axis addresses and a value set as the workpiece origin offset

Explanation

- Selecting the additional workpiece coordinate systems

When a P code is specified together with G54.1 (G54), the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48).
A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power-on.

G54.1 P1 Additional workpiece coordinate system 1
G54.1 P2 Additional workpiece coordinate system 2
:
G54.1 P48 Additional workpiece coordinate system 48

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.

- Setting the workpiece origin offset value in the additional coordinate systems (G10)

When a workpiece origin offset value is specified using an absolute value, the specified value is the new offset value. When it is specified using an incremental value, the specified value is added to the current offset value to obtain a new offset value.

Limitation

- Specifying P codes

A P code must be specified after G54.1 (G54). If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1 (G54.1P1) is assumed.
If a value not within the specifiable range is specified in a P code, an alarm PS0030 is issued.
P codes other than workpiece offset numbers cannot be specified in a G54.1 (G54) block.

Example 1) G54.1 G04 P1000 ;
Example 2) G54.1 M98 P48 ;

7.2.6 Automatic Coordinate System Setting

When the workpiece coordinate system is not used (bit 0 (NWZ) of parameter No. 8136 is 1), if bit 0 (ZPR) of automatic coordinate system setting parameter No. 1201 is 1, a manual reference position return operation determines the coordinates automatically.
Once α, β, and γ are set with parameter No. 1250, a workpiece coordinate system is set upon reference position return so that the base point on the tool holder or the tip of the basic tool is positioned at X = α, Y = β, and Z = γ.
This processing occurs as if the following are specified at the reference position:

M

G92 X$\underline{\alpha}$ Y$\underline{\beta}$ Z$\underline{\gamma}$;

T

G50 X$\underline{\alpha}$ Z$\underline{\gamma}$;

When the workpiece coordinate system is used (bit 0 (NWZ) of parameter No. 8136 is 0), regardless of the setting of bit 0 (ZPR) of parameter No. 1201, a manual reference position return operation establishes the workpiece coordinate system based on the workpiece origin offset (parameters Nos. 1220 to 1226).

T

When the setting of a workpiece coordinate system shift amount is other than 0, a workpiece coordinate system shifted by the amount is set.

7.2.7 Workpiece Coordinate System Shift (T Series)

T

Explanation

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 (see III-3.1).
Set the desired shift amount in the workpiece coordinate system shift memory.

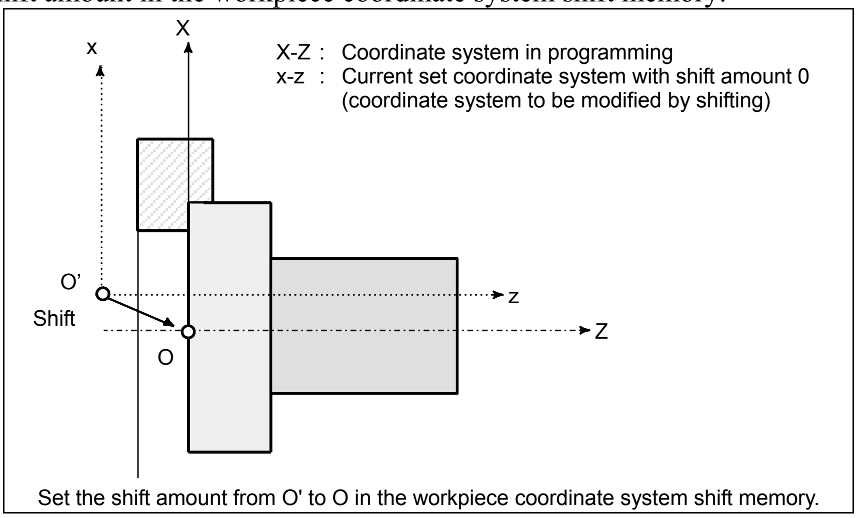


Fig. 7.2.7 (a) Workpiece coordinate system shift

Format

- Changing the workpiece coordinate system shift amount

G10 P0 IP_;

IP : Settings of an axis address and a workpiece coordinate system shift amount

> ⚠ **CAUTION**
> A single block can contain a combination of X, Y, Z, C, U, V, W, and H (in G code system A). In this case, if commands are specified for the same axis, whichever appears later becomes valid.

Limitation

- Shift amount and coordinate system setting command

Specifying a coordinate system setting command (G50 (for G code system A) or G92 (for G code system B/C)) invalidates the shift amount that has already been set.

Example)

When G50X100.0Z80.0; is specified, a coordinate system is set so that the current base position of the tool is at X =100.0 and Z = 80.0, regardless of which value has been set for the workpiece coordinate system shift amount.

- Shift amount and coordinate system setting

When the shift amount is already set, if an automatic coordinate system setting is made by a manual reference position return, the set coordinate system is immediately shifted by the shift amount.

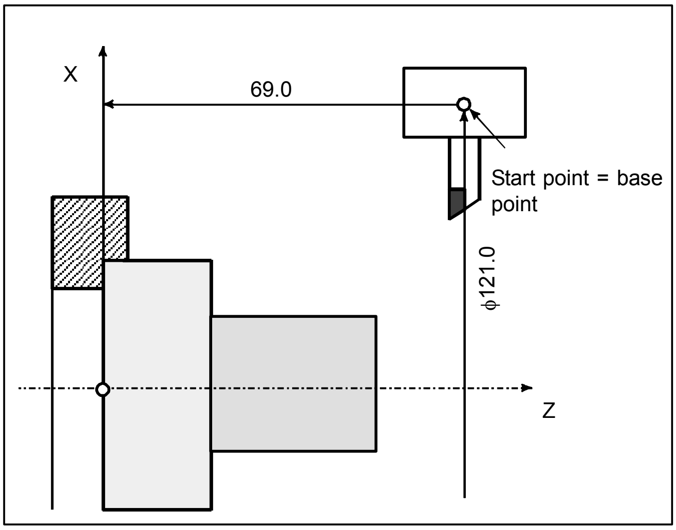
- Diameter and radius values

The workpiece coordinate system shift amount depends on diameter programming or radius programming.

Example)

To set the base point indicated by X = Φ120.0 (diameter value) and Z = 70.0 with reference to the workpiece origin, if the distance to the current base point is indicated by X = Φ121.0 and Z = 69.0, the shift amount is set as shown below.

X=1.0, Z=-1.0



7.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format

G52 IP_; Setting the local coordinate system

:

G52 IP 0 ; Canceling of the local coordinate system

IP_ : Origin of the local coordinate system

Explanation

By specifying G52 IP_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.
Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the origin of a new local coordinate system in the workpiece coordinate system.
To cancel the local coordinate system or specify the coordinate value in the workpiece coordinate system, match the origin of the local coordinate system with that of the workpiece coordinate system.

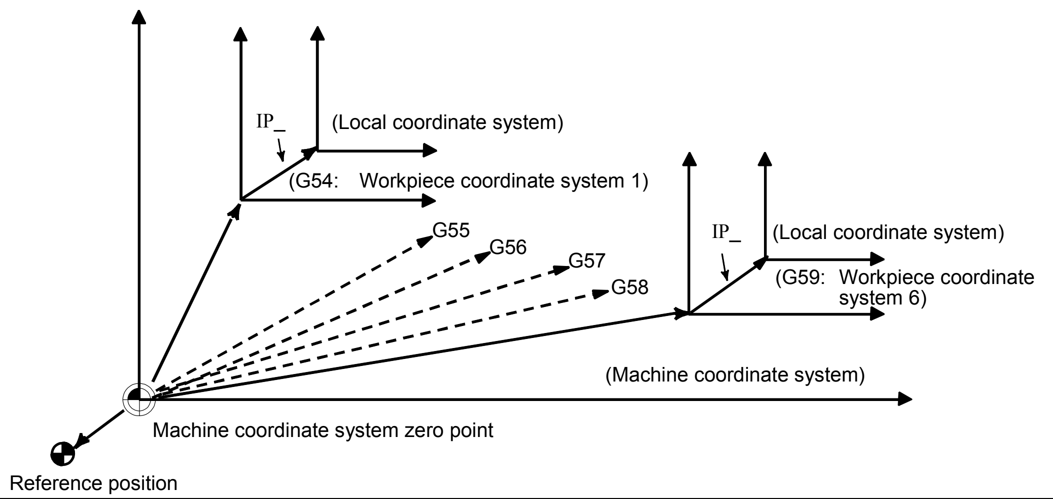


Fig. 7.3 (a) Setting the local coordinate system

⚠ **CAUTION**

1 When ZCL (bit 2 of parameter No.1201) is set to 1 and an axis returns to the reference position by the manual reference position return function, the origin of the local coordinate system of the axis matches that of the workpiece coordinate system. The same is true when the following command is issued:

G52 α0 ;

α: Axis which returns to the reference position

2 The local coordinate system setting does not change the workpiece and machine coordinate systems.

⚠ CAUTION

3 Whether the local coordinate system is canceled at reset depends on the parameter setting. The local coordinate system is canceled when either bit 3 (RLC) of parameter No.1202 is set to 1. The local coordinate system is canceled regardless of the setting of bit 3 (RLC) of parameter No. 1202 when bit 6 (CLR) of parameter No. 3402 is 0 and bit 7 (WZR) of parameter No. 1201 is 1 or when bit 6 (CLR) of parameter No. 3402 is 1 and bit 6 (C14) of parameter No. 3407 is 0.

4 When a workpiece coordinate system is set with the G92 (G50 for G code system A in the T series) command, the local coordinate system is canceled. However, the local coordinate system of an axis for which no coordinate system is specified in a G92 (G50 for G code system A in the T series) block remains unchanged.

5 G52 cancels the offset temporarily in tool radius · tool nose radius compensation.

6 Command a move command immediately after the G52 block in the absolute mode.

7.4 PLANE SELECTION

Select the planes for circular interpolation, cutter compensation (M series), coordinate system rotation (M series), and drilling by G-code.
The following table lists G-codes and the planes selected by them.

Explanation

Table 7.4 (a) Plane selected by G code

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			

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.
When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.
Parameter No. 1022 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 plane is unchanged in the block in which G17, G18 or G19 is not commanded.
The movement instruction is irrelevant to the plane selection.

M

When the power is turned on or the CNC is reset, G17 (XY plane), G18 (ZX plane), or G19 (YZ plane) is selected by bits 1 (G18) and 2 (G19) of parameter No. 3402).

T

When the power is turned on, G18 (ZX plane) is selected.

NOTE

U-, V-, and W-axes can be used with G-codes B and C.

Example

Plane selection when the X-axis is parallel with the U-axis.

G17 X_ Y_ ;   XY plane,

G17 U_ Y_ ;	UY plane
G18 X_ Z_ ;	ZX plane
X_Y_ ;	Plane is unchanged (ZX plane)
G17 ;	XY plane
G18 ;	ZX plane
G17 U_ ;	UY plane
G18 Y_ ;	ZX plane, Y axis moves regardless without any relation to the plane.

8 COORDINATE VALUE AND DIMENSION

Chapter 8, "COORDINATE VALUE AND DIMENSION", consists of the following sections:

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING

There are two ways to command travels of the tool; 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 a tool movement.

M

G90 and G91 are used to programming absolute or incremental programming, respectively.

T

Absolute programming or incremental programming is used depending on the programming used. See following tables.

G code system	A	B or C
Command method	Address word	G90, G91

Format

M

Absolute programming **G90** IP_ ;
Incremental programming **G91** IP_ ;

T

- G code system A

	Absolute programming	Incremental programming
X axis move command	X	U
Z axis move command	Z	W
Y axis move command	Y	V
C axis move command	C	H

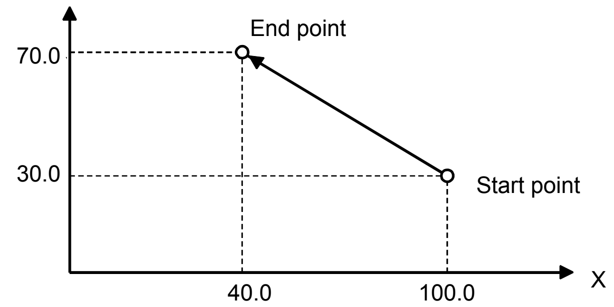
- G code system B or C

Absolute programming **G90** IP_ ;
Incremental programming **G91** IP_ ;

Example

M

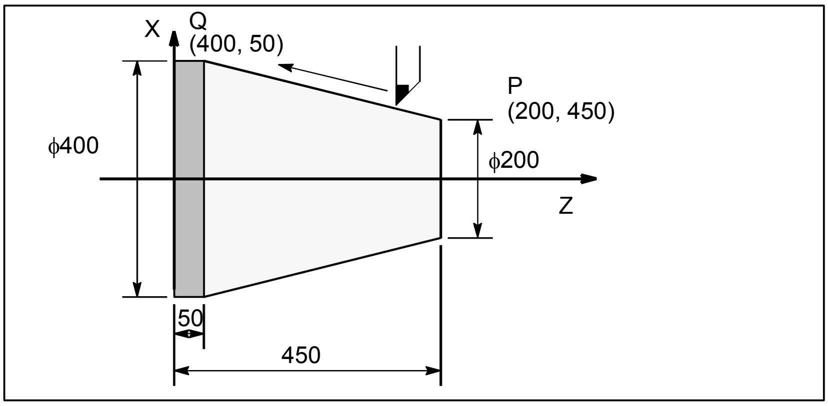
G90 X40.0 Y70.0 ; Absolute programming
G91 X-60.0 Y40.0 ; Incremental programming



T

Tool movement from point P to point Q (diameter programming is used for the X-axis)

	G code system A	G code system B or C
Absolute programming	X400.0 Z50.0 ;	G90 X400.0 Z50.0 ;
Incremental programming	U200.0 W-400.0 ;	G91 X200.0 Z-400.0 ;



NOTE

1. Absolute programming and incremental programming can be used together in a block.
   In the above example, the following command can be specified : X400.0 W-400.0 ; (in the G code system A)
2. When absolute programming and incremental programming for the same axis are used at the same time (for example, X and U, or Z and W) in one block in G code system A, absolute programming or incremental programming, whichever is specified later, is valid.
3. Incremental programming cannot be used when names of the axes are A and B during G code system A is selected.

8.2 INCH/METRIC CONVERSION (G20, G21)

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

Format

Inch input
Metric input

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. 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 (II-2.3). 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.

> ⚠ **WARNING**
> G20 and G21 must not be switched during a program.

> **NOTE**
> 1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
> 2 The inch and metric input can also be switched using settings (see III-12.3.1).
> 3 To disable the function of bit 2 (IRF) of parameter No. 14000 or bit 0 (NIM) of parameter No. 11222, perform inch/metric conversion at a point having a machine coordinate of 0.

Performing inch/metric conversion in the reference position (parameter No. 1240 is not 0)

Conventionally, inch/metric conversion must be performed at a point having a machine coordinate of 0. However, setting bit 2 (IRF) of parameter No. 14000 to 1 enables inch/metric conversion to be performed in the reference position (parameter No. 1240).

If an attempt is made to perform inch/metric conversion when an axis with this function enabled is not in the reference position, alarm PS5362 is issued to cancel the attempt.
Before trying to perform inch/metric conversion, be sure to set the axis of interest to the reference position, using the G28 command, for example.
About the axis by which bit 7 (IMAx) of parameter No.14000 is set to 1, even if inch/metric conversion is performed when the axis is not in the reference position, alarm PS5362 is not generated.
Therefore, be sure to set 1 in IMAx of an unrelated axis to inch/metric conversion such as the rotary axis.
As a result, an unrelated axis to inch/metric conversion need not be performed reference position return before inch/metric conversion is performed.

In addition, if the workpiece coordinate system has been shifted, using the following commands or operations, bit 1 (CIM) of parameter No. 11222 can be used to select whether to issue alarm PS1298 or to clear the offset.

- Manual intervention performed with the manual absolute signal being off
- Move command issued with the machine locked
- Move command issued using a manual handle interrupt
- Mirror image-based operation
- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting

Switching conditions

All of the following conditions must be met to perform inch/metric conversion in the reference position. Failing to satisfy any of the conditions results in alarm PS1298 being issued. For electronic gear box synchronization, alarm PS1595 is issued.

- Positioning or linear interpolation
- Polar coordinate interpolation cancel mode (T series)
- Polar coordinate command cancel (M series)
- Spindle Speed fluctuation detection off (T series)
- Tool radius · tool nose radius compensation cancel
- Normal direction control cancel (M series)
- Tool length offset cancel (M series)
- Scaling cancel (M series)
- Programmable mirror image cancel (M series)
- Polygon turning cancel (T series)
- Macro modal call cancel
- Coordinate system rotation mode off (M series)
- Mirror image for double turret or balanced cutting mode cancel (T series)
- Canned cycle cancel
- Electronic gear box synchronization cancel (M series)
- Constant surface speed control cancel

The following setting is necessary to perform inch/metric conversion in the reference position.

- Workpiece coordinate system (parameter NWZ(No.8136#0)=0)

Restrictions

The following operations need to be performed at a point having a machine coordinate of 0.

- Inch/metric conversion based on bit 2 (INI) of setting parameter No. 0
- Inch/metric conversion based on programmable parameter input (G10)
- Inch/metric conversion based on custom macro variable No. 3005

Performing inch/metric conversion in positions other than the reference position

Setting bit 0 (NIM) of parameter No. 11222 enables inch/metric conversion to be performed even in positions other than the reference position.

In addition, if the workpiece coordinate system has been shifted, using the following commands or operations, bit 1 (CIM) of parameter No. 11222 can be used to select whether to issue alarm PS1298 or to clear the offset.

- Manual intervention performed with the manual absolute signal being off
- Move command issued with the machine locked
- Move command issued using a manual handle interrupt
- Mirror image-based operation

- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting

If an axis is under any of the following controls, however, no automatic coordinate system conversion based on this function can be carried out for the axis.

- Axis control by PMC
- Axis synchronous control (for slave axes when the master axis is a PMC axis)
- Spindle control with servo motor

Switching conditions

Performing inch/metric conversion in any position other than the reference position requires satisfying all of the following conditions. Failing to satisfy any of the conditions results in alarm PS1298 being issued. For electronic gear box synchronization, alarm PS1595 is issued.

- Positioning or linear interpolation
- Polar coordinate interpolation cancel mode (T series)
- Polar coordinate command cancel (M series)
- Spindle Speed fluctuation detection off (T series)
- Tool radius · tool nose radius compensation cancel
- Normal direction control cancel (M series)
- Tool length offset cancel (M series)
- Scaling cancel (M series)
- Programmable mirror image cancel (M series)
- Polygon turning cancel (T series)
- Macro modal call cancel
- Coordinate system rotation mode off (M series)
- Mirror image for double turret or balanced cutting mode cancel (T series)
- Canned cycle cancel
- Electronic gear box synchronization cancel (M series)
- Constant surface speed control cancel

The following settings are necessary to perform inch/metric conversion at a point other than the reference position.

- Workpiece coordinate system (parameter NWZ(No.8136#0)=0)
- Workpiece coordinate system preset
  (parameter NWC(No.8136#1)=0)

Restrictions

The following operations need to be performed at a point having a machine coordinate of 0.

- Inch/metric conversion based on bit 2 (INI) of setting parameter No. 0
- Inch/metric conversion based on programmable parameter input (G10)
- Inch/metric conversion based on custom macro variable No. 3005

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:

M

X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F

T

X, Y, Z, U, V, W, A, B, C, I, J, K, R, F

Explanation

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters inch, or deg. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal notation by using the bit 0 (DPI) of parameter No.3401.Values can be specified both with and without decimal point in a single program.

Example

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

> ⚠ **CAUTION**
> When specifying a dimension word for a command G code in a block, be sure to place the dimension word after the command G code.

NOTE

1 A specified value less than the least increment is treated as shown below (rounded off to the right side).

Example 1)

When a value is specified directly at an address (in the case of IS-B)

X-0.0004 ; Treated as X0.000
X0.0004 ; Treated as X0.000
X-0.0005 ; Treated as X0.000
X0.0005 ; Treated as X0.001
X-0.0006 ; Treated as X-0.001
X0.0006 ; Treated as X0.001

As shown in the table below, commands including positive and negative fractions having the same absolute value with respect to zero are repeated. In right side rounding, the results are periodical. In general rounding, however, the results are not periodical due to rounding symmetry with respect to zero.

To prevent this, right side rounding is used for calculation.

	Right side rounding	General rounding
G90 G00 X0 ;	X 0.000	X 0.000
G91 X-0.0015 ;	X -0.001	X -0.002
G91 X0.0015 ;	X 0.000	X -0.001
G91 X-0.0015 ;	X -0.001	X -0.003
G91 X0.0015 ;	X 0.000	X -0.002
G91 X-0.0015 ;	X -0.001	X -0.004
G91 X0.0015 ;	X 0.000	X -0.003

Example 2)

When a value is assigned to a macro variable (in the case of IS-B)
Similarly, right side rounding is performed.

#100=1.2345 ;
X#100 ; Treated as X1.235
#100=-1.2345 ;
X#100 ; Treated as X-1.234

2 When more than nine digits are specified, an alarm occurs. If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.

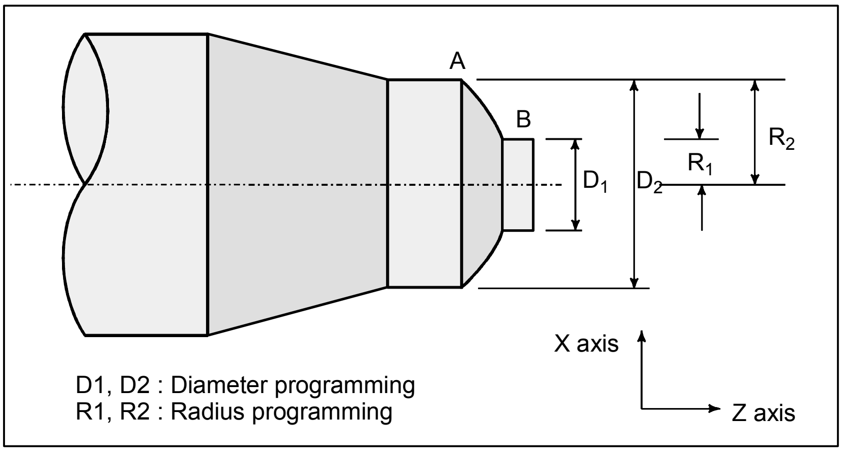
Examples:

X0.123456789 ;
Alarm PS0003 occurs because more than nine digits are specified.

X1234567.8 ;
If the least input increment is 0.001 mm, the value is converted to integer 1234567800. Because the integer has more than nine digits, an alarm occurs.

8.4 DIAMETER AND RADIUS PROGRAMMING

Since the workpiece cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways :
Diameter and Radius



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

Explanation

- Notes on diameter programming/radius programming for each command

Radius programming or diameter programming can be specified by bit 3 (DIA) of parameter No.1006. When using diameter programming, note the conditions listed in the Table 8.4 (a).

Table 8.4 (a) Notes on specifying diameter value

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D2 minus D1 for tool path B to A.
Coordinate system setting	Specifies a coordinate value with a diameter value
Component of tool offset value	Bit 1 of parameter No.5004 determines either diameter or radius value
Parameters in canned cycle (T series), such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.
Display of axis position	Displayed as diameter value

9 SPINDLE SPEED FUNCTION (S FUNCTION)

The spindle speed can be controlled by specifying a value following address S.
Chapter 9, "SPINDLE SPEED FUNCTION (S FUNCTION)", consists of the following sections:

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE

When a value is specified after address S, the code signal and strobe signal are sent to the machine to control the spindle rotation speed.
A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND)

The spindle speed can be specified directly by address S followed by a max. five-digit value (min^{-1}). The unit for specifying the spindle speed may vary depending on the machine tool builder. Refer to the appropriate manual provided by the machine tool builder for details.

9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.

Format

- Constant surface speed control command

G96 S<u>xxxxx</u> ;
↑ Surface speed (m/min or feet/min)
This surface speed unit may change according to machine tool builder's specification.

- Constant surface speed control cancel command

G97 S<u>xxxxx</u> ;
↑ Spindle speed (min^{-1})
This surface speed unit may change according to machine tool builder's specification.

- Constant surface speed controlled axis 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

T

NOTE
If multi-spindle control (spindle selecting based on address P) is enabled, axis specification based on address P is disabled. Use parameter No. 3770 for axis specification.

- Clamp of maximum spindle speed

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.

Explanation

- Constant surface speed control command (G96)

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G96 command must specify the axis along which constant surface speed control is applied. A G97 command cancels the G96 mode. When constant surface speed control is applied, a spindle speed higher than the value specified in G92 S_ ; or G50 S_ ; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped. S (surface speed) commands in the G96 mode are assumed as S = 0 (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

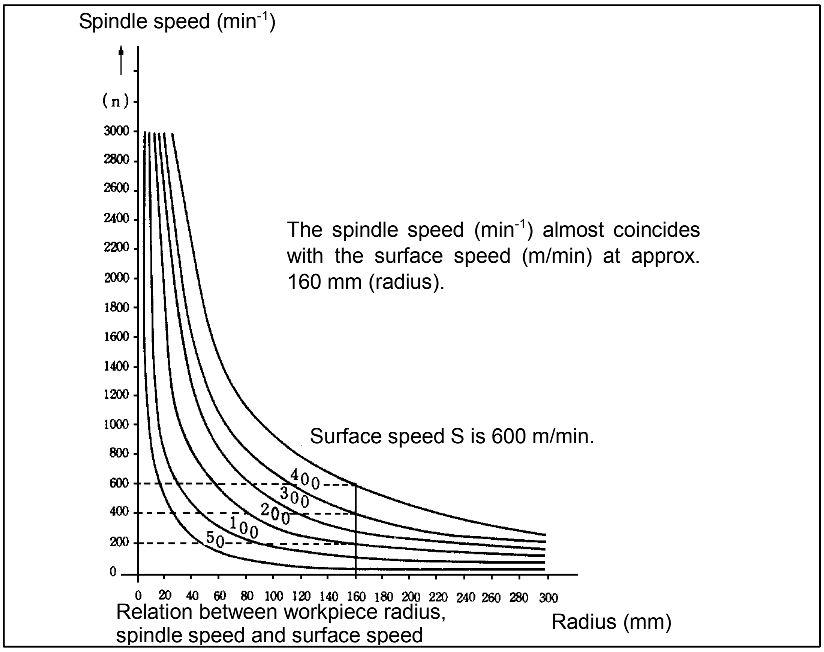


Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

- Setting the workpiece coordinate system for constant surface speed control

To execute the constant surface speed control, it is necessary to set the workpiece coordinate system , and so the coordinate value at the center of the rotary axis, for example, Z axis, (axis to which the constant surface speed control applies) becomes zero.

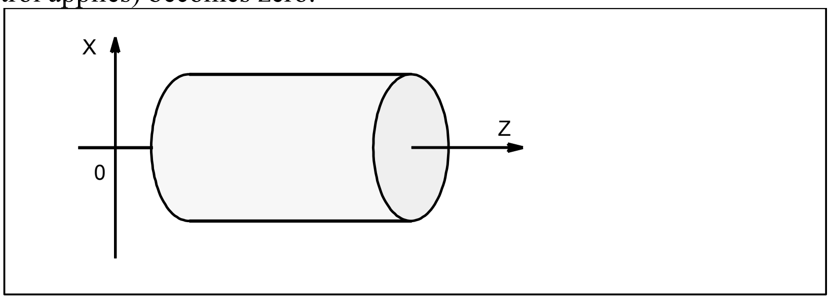
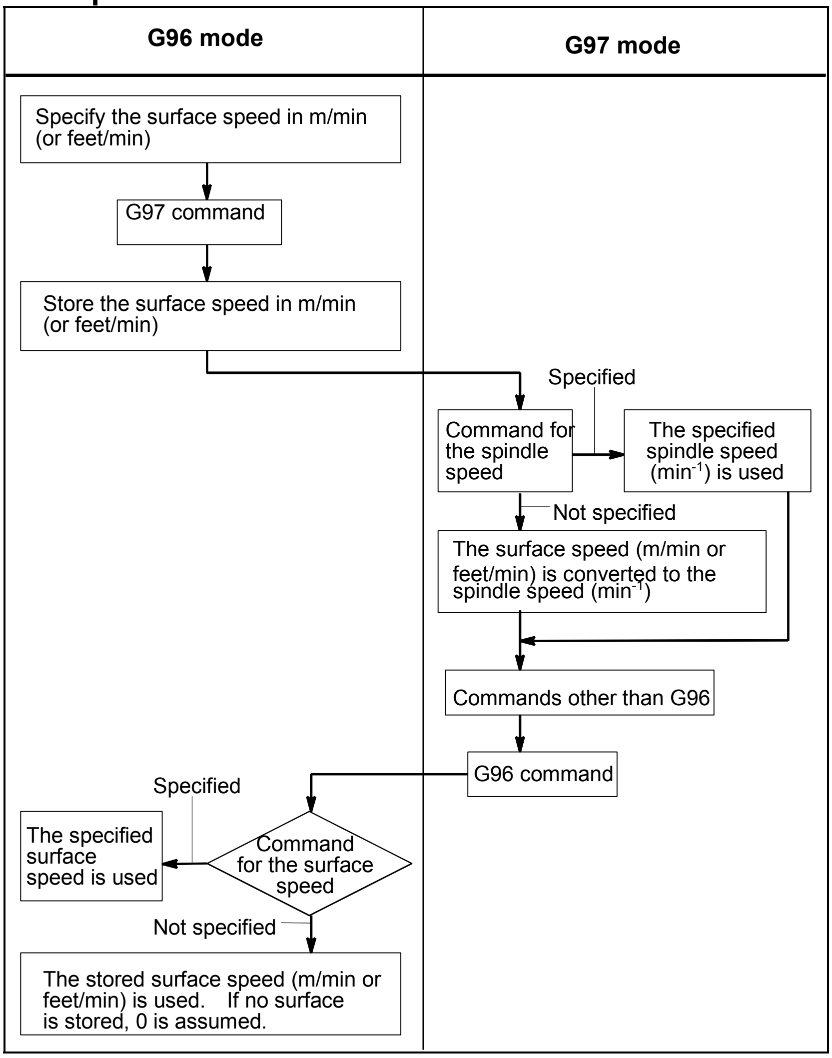


Fig. 9.3 (b) Example of the workpiece coordinate system for constant surface speed control

- Surface speed specified in the G96 mode



Limitation

- Constant surface speed control for threading

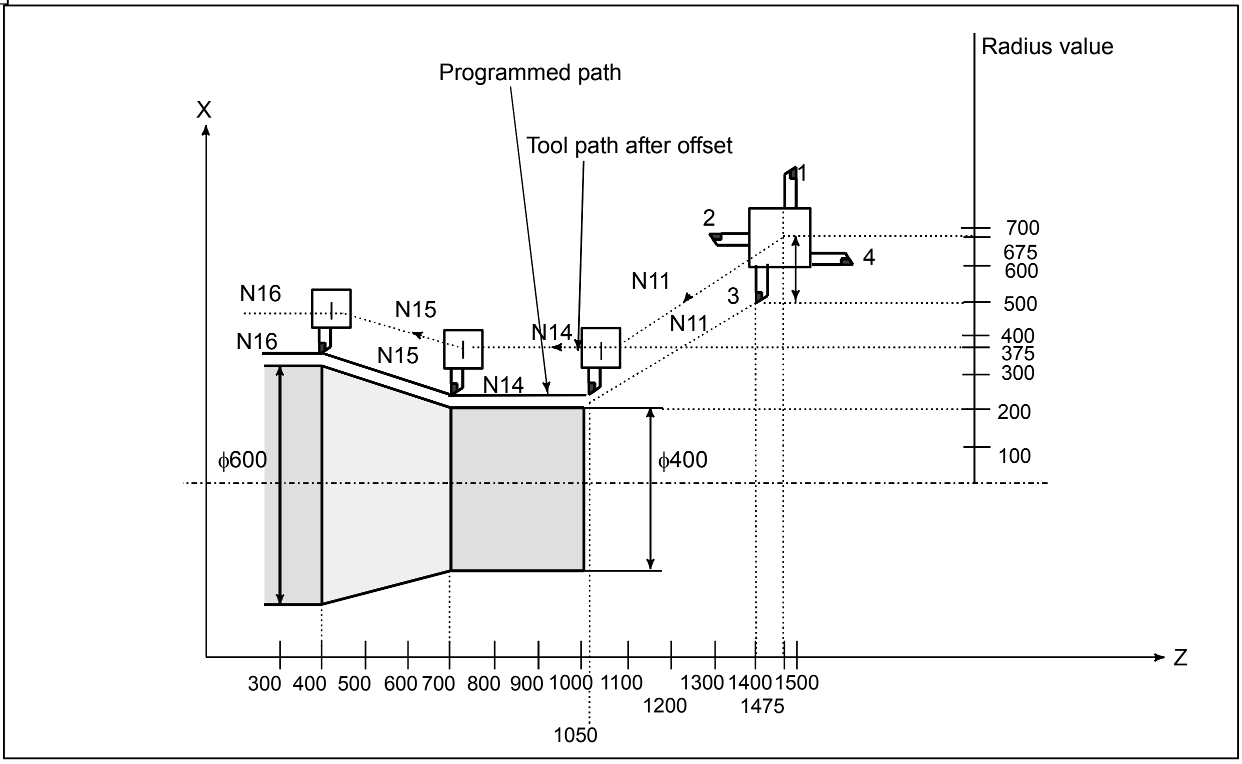
The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

- Constant surface speed control for rapid traverse (G00)

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not executed at rapid traverse.

Example

T



N8 G00 X1000.0 Z1400.0 ;
N9 T33;
N11 X400.0 Z1050.0 ;
N12 G50 S3000 ; (Designation of max. spindle speed)
N13 G96 S200 ; (Surface speed 200 m/min)
N14 G01 Z700.0 F1000 ;
N15 X600.0 Z 400.0;
N16 Z_ ;

The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid. At the end point N15 in the example above, the speed at 600 dia. (Which is not the turret center but the tool nose) is 200 m/min. If X axis coordinate value is negative, the CNC uses the absolute value.

9.4 SPINDLE POSITIONING FUNCTION

Overview

T

In turning, the spindle connected to the spindle motor is rotated at a certain speed to rotate the workpiece mounted on the spindle. This spindle control status is referred to as spindle rotation mode.
The spindle positioning function turns the spindle connected to the spindle motor by a certain angle to position the workpiece mounted on the spindle at a certain angle. This spindle control status is referred to as spindle positioning mode.

The spindle positioning function involves the following three operations:

1. Canceling the spindle rotation mode and entering the spindle positioning mode

Place the spindle in the spindle positioning mode and establish a reference position by specifying a given M code (set with a parameter). (Spindle orientation)

2. Positioning the spindle in the spindle positioning mode
   The spindle is positioned in either of the two methods:
   1) Positioning with an arbitrary angle by an axis address
   2) Positioning with a semi-fixed angle by a given M code (set with a parameter)
3. Canceling the spindle positioning mode, and entering the spindle rotation mode
   Place the spindle in the spindle rotation mode by specifying a given M code (set with a parameter).

The least command increment, least input increment, and maximum value for the spindle positioning axis are as follows:

- Least command increment
  $\frac{360}{4096} \fallingdotseq$ 0.088 deg (when the gear ratio of the spindle to the position coder
- Least input increment
  0.001 deg (IS-B)
- Maximum value
  ±999999.999 deg

NOTE

1 Be sure to set 1 in bit 1 (AXC) of parameter No.8133 and 0 in bit 2 (SCS) of parameter No.8133 to use the spindle positioning function.

2 Both serial spindle Cs contour control function and spindle positioning function cannot be made effective at the same time.
If both are specified as AXC=1 and SCS=1, both functions become invalid.
Therefore, when a negative value is set in parameter No.1023 with above specification, alarm (SV1026) is generated.

3 Be sure to set 0 in bit 1 (AXC) of parameter No.8133 and 1 in bit 2 (SCS) of parameter No.8133 to use the serial spindle Cs contour control function.

9.4.1 Spindle Orientation

When spindle positioning is first performed after the spindle motor is used for normal spindle operation, or when spindle positioning is interrupted, the spindle orientation is required.
Orientation permits the spindle to stop at a predetermined position.
Orientation is directed by the M code set in parameter No. 4960. The direction of orientation can be set with a parameter.
For the serial spindle, it is set in bit 4 (RETSV) of parameter No. 4000. With the grid shift function, the orientation position can be shifted in a range of 0 to 360 deg with parameter No. 4073 for a serial spindle.

- Feedrate during spindle orientation

An orientation feedrate for a serial spindle is determined by a spindle parameter setting.
In orientation, the serial spindle stops at the orientation position after several rotations of the spindle motor.

- Omission of orientation

By using bit 2 (ISZ) of parameter No. 4950, orientation upon switching to spindle positioning mode can be omitted if it is unnecessary (for example, when no start position is specified and incremental positioning from the current position is only required). More specifically, when an M code for switching to spindle positioning mode is specified, the spindle control mode is simply switched to spindle positioning mode and then the processing is completed without orientation.

- Program reference position

The position at which orientation is completed is assumed to be a program reference position. However, the program reference position can be changed through coordinate system setting (G92 or G50) or automatic coordinate system setting (bit 0 (ZPR) of parameter No. 1201).
When a setting is made to omit orientation, a program reference position is not established, and operation by an absolute command is unpredictable during spindle positioning with an axis address.

9.4.2 Spindle Positioning (T Series)

T

The spindle can be positioned with a semi-fixed angle or arbitrary angle.

- Positioning with a semi-fixed angle

Use an M code to specify a positioning angle. The specifiable M code value may be one of the six values from Mα to M (α+5). Value α must be set in parameter No. 4962 beforehand. The positioning angles corresponding to Mα to M (α+5) are listed below. Value β must be set in parameter No. 4963 beforehand.

M-code (Ex.) β = α + 5	Positioning angle	(Ex.) β = 30°
Mα	β	30°
M (α + 1)	2β	60°
M (α + 2)	3β	90°
M (α + 3)	4β	120°
M (α + 4)	5β	150°
M (α + 5)	6β	180°

When the number of M codes to be used, value γ, is specified in parameter No. 4964, a specifiable M code value may be in a range of values from Mα to M (α + (γ - 1)), up to 255 values from Mα to M (α + (255 - 1)).

M code (Ex.) γ = 11	Positioning angle	(Ex.) β = 30°
Mα	β	30°
M (α + 1)	2β	60°
M (α + 2)	3β	90°
M (α + 3)	4β	120°
...	...	...
M (α + 11 - 1)	11β	330°

The direction of rotation can be specified in IDM (bit 1 of parameter 4950).

- Positioning with an arbitrary angle

Specify the position with an arbitrary angle using the axis address followed by a signed numeric value or numeric values. The axis address must be specified in the G00 mode.
(The explanation below is given assuming that a C axis address is set.)

(Example) C-45000
C180.000

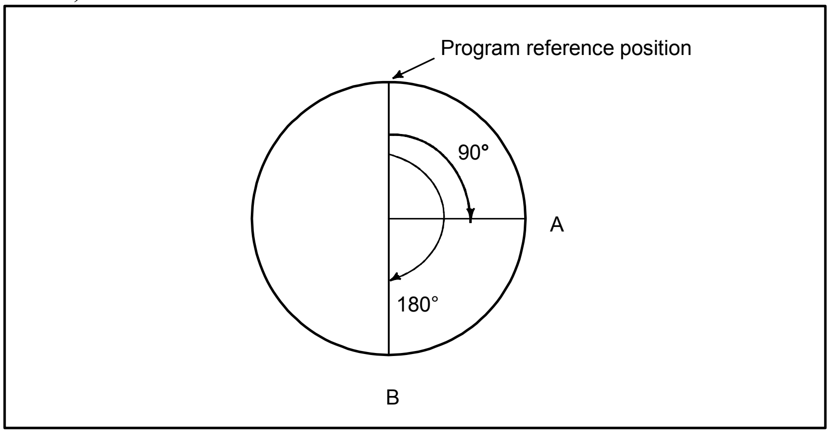
A numeric with the decimal point can be entered. The value must be specified in degrees.

(Example) C36.0=C36 degrees

- Absolute commands and incremental commands

Incremental commands are always used for positioning with a semi-fixed angle (using M codes).

The direction of rotation can be specified with bit 1 (IDM) of parameter No. 4950.
Absolute and incremental commands can be used for positioning with an arbitrary angle.
With absolute commands for positioning with an arbitrary angle, when the rotation axis rollover function is used (bit 0 (ROA) of parameter No. 1008 is 1), shortcut control is also enabled (bit 1 (RAB) of parameter No. 1008 is 0).



Command format		G-code system A		G-code system B or C	
		Address used	Command A-B in the above figure	Address used and G code	Command A-B in the above figure
Absolute command	Specify the end point with a distance from the program reference position.	C	C180. ;	G90,C	G90 C180. ;
Incremental command	Specify a distance from the start point to the end point.	H	H90. ;	G91,C	G91 C90. ;

- Feedrate during positioning

The feedrate during positioning equals the rapid traverse speed specified in parameter No. 1420.
For the specified speed, an override of 100%, 50%, 25%, and F0 (parameter No. 1421) can be applied.

9.4.3 Canceling Spindle Positioning (T Series)

T

When modes are to be switched from spindle positioning to normal spindle rotation, the M code set in parameter No. 4961 must be specified.

Also, the spindle positioning mode is canceled and the spindle rotation mode is set in the following cases:

<1> A reset operation (including an emergency stop) occurs when a servo alarm is issued.

<2> A reset operation (including an emergency stop) occurs when a spindle alarm is issued.

<3> An orientation operation in progress is stopped due to a reset or alarm, or for some other reason.

<4> A reset operation (including an emergency stop) occurs when bit (IOR) 0 of parameter No. 4950 is 1.

> ⚠ **CAUTION**
>
> 1 During execution of spindle positioning sequences (canceling the spindle rotation mode and entering the spindle positioning mode, positioning the spindle in the spindle positioning mode, and canceling the spindle positioning mode and entering the spindle rotation mode), the automatic operation stop signal *SP is invalid. This means that automatic operation does not stop until all the sequences are completed, even when the *SP signal becomes 0.

⚠ **CAUTION**

2 Dry run and machine lock cannot be performed during spindle positioning.
3 Auxiliary function lock is disabled for M codes for the spindle positioning function.
4 Both serial spindle Cs contour control function (bit 2 (SCS) of parameter No.8133) and spindle positioning function (bit 1 (AXC) of parameter No. 8133) cannot be made effective at the same time.
If both are specified as AXC=1 and SCS=1, both functions become invalid. Therefore, when a negative value is set in parameter No.1023 with above specification, alarm (SV1026) is generated.
Be sure to set AXC=1 and SCS=0 to use the spindle positioning function.
Be sure to set AXC=0 and SCS=1 to use the serial spindle Cs contour control function.
5 The spindle positioning axis is handled as a controlled axis. Therefore, controlled axis-related signals (such as the overtravel signal) must be set.
6 When the rigid tapping function (bit 3 (NRG) of parameter No. 8135) is used together with the spindle positioning function (bit 1 (AXC) of parameter No. 8133), rigid tapping cannot be specified in the spindle positioning mode or spindle positioning cannot be specified in the rigid tapping mode.

NOTE

1 M code commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. (Also, M code commands for positioning of another spindle must not be contained in the same block.)
Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.
2 Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.
3 Axis address commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. However, the following commands can be contained in the same block where axis address commands are specified:
G00, G90, G91, G92 (G-code systems B and C)
G00, G50 (G-code system A)
4 M code commands for spindle positioning specify M codes that are not buffered.
5 Spindle positioning cannot be performed by manual operation (in jog feed, manual handle feed, or other mode).
6 Spindle positioning cannot be performed by PMC axis control.
7 For spindle positioning, program restart operation cannot be performed. Use the MDI for these operations.
8 The stored stroke limit check is disabled for the spindle positioning axis.
9 The control axis detach function is disabled for the spindle positioning axis.
10 The pitch error compensation function is disabled for the spindle positioning axis.
11 When a setting is made to omit spindle orientation, the reference position return completion signal does not become 1.
12 In spindle orientation, all-axis interlock and axis-specific interlock are checked only when a block is started. A signal is ignored if input during the execution of the block.
13 A difference between a specified travel distance and an actual travel distance is maintained until spindle positioning mode is canceled.

9.5 SPINDLE SPEED FLUCTUATION DETECTION (T SERIES)

T

Overview

With this function, an overheat alarm (OH0704) 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.
G26 enables spindle speed fluctuation detection.
G25 disables spindle speed fluctuation detection.

Format

- Spindle fluctuation detection on

G26 Pp Qq Rr Ii ;

P: Time (in ms) from the issue of a new spindle rotation command (S command) to the start of checking whether the actual spindle speed is so fast that an overheat can occur. When a specified speed is reached within the time period of P, a check is started at that time.

Q: Tolerance (%) of a specified spindle speed

$$q = \left| 1 - \frac{actual\ spindle\ speed}{specified\ spndle\ speed} \right| \times 100$$

If a specified spindle speed lies within this range, it is regarded as having reached the specified value. Then, the checking of an actual spindle speed is started.

R: Spindle speed fluctuation (%) at which the actual spindle speed is so fast that an overheat can occur

$$r = \left| 1 - \frac{speed\ that\ can\ cause\ overheat}{speciofied\ spndle\ speed} \right| \times 100$$

If the fluctuation of the actual spindle speed to the specified spindle speed exceeds the spindle speed fluctuation of R, the actual spindle speed is regarded as being so fast that an overheat can occur.

I: Spindle speed fluctuation width at which the actual spindle speed (min^{-1}) is so fast that an overheat can occur
If the fluctuation (width) between the specified and actual spindle speeds exceeds the spindle speed fluctuation width of I, the actual spindle speed is regarded as being so fast that an overheat can occur.

G26 enables the spindle speed fluctuation detection function. The values specified for P, Q, R, and I are set in the following parameters: No. 4914, No. 4911, No. 4912, and No. 4913, respectively. Each command address corresponds to a parameter number as listed below.

Command address	Parameter number
Q	No.4911
R	No.4912
I	No.4913
P	No.4914

If the P, Q, R, or I command address is omitted, the function detects the fluctuation of the actual spindle speed according to the value set in the corresponding parameter (No. 4914, No. 4911, No. 4912, or No. 4913).

The parameters (No. 4914, No. 4911, No. 4912, and No. 4913) for the spindle on which the currently selected position coder is mounted are used for the setting and spindle speed fluctuation detection check.

- Spindle fluctuation detection off

G25;

G25 disables the spindle speed fluctuation detection function.
When G25 is specified, the parameters (No. 4914, No. 4911, No. 4912, and No. 4913) are unchanged. When the power is turned on or after a reset (clear state (bit 6 (CLR) of parameter No. 3402 = 1)) is executed, the spindle speed fluctuation detection function is disabled (G25). For the clear state, also check the setting of bit 0 (C08) of parameter No. 3407.

Explanation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Si or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm (OH0704) is activated when the actual spindle speed varies for the commanded speed (Sc) under the condition that the variation width exceeds the allowable variation width (Sm).

$|Sc - Sa| > Sm$

Sc : Specified spindle speed
Sa : Actual spindle speed
Si : The allowable constant variation width which is independent of the specified spindle speed (parameter (No.4913))
Sr : The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r = parameter (No.4912))

Parameter FLR(No.4900#0)= 0	Parameter FLR(No.4900#0)= 1
$Sr = Sc \times \frac{r}{100}$	$Sr = Sc \times \frac{r}{1000}$

Sm: Si or Sr, whichever is greater

- Conditions to start spindle speed fluctuation detection

If the specified spindle speed Sc changes, spindle speed fluctuation detection starts when one of the conditions below is met:

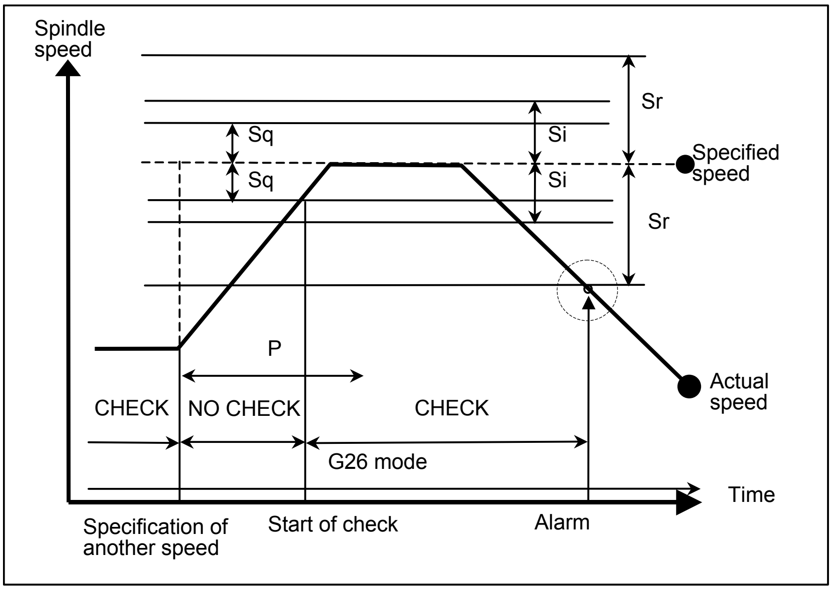
<1> The actual spindle speed falls in a range of (Sc - Sq) to (Sc + Sq)
Sc : Specified spindle speed
Sq : Tolerance within which the spindle is assumed to attain the programmed speed (parameter (No.4911))

Parameter FLR= 0	Parameter FLR= 1
$Sq = Sc \times \frac{q}{100}$	$Sq = Sc \times \frac{q}{1000}$

<2> When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

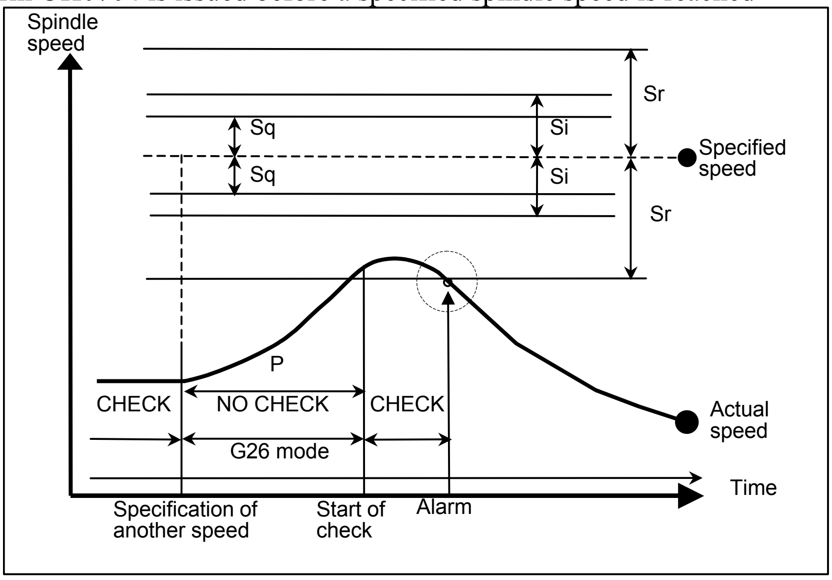
- Examples of spindle speed fluctuation detection

(Example 1)
When an alarm (OH0704) is issued after a specified spindle speed is reached



(Example 2)

When an alarm OH0704 is issued before a specified spindle speed is reached



Specified speed : (Speed specified by address S and five-digit value) × (spindle override)
Actual speed : Speed detected with a position coder
p : Period after a change occurs in the actual spindle speed until detection starts
Parameter No.4914, address P
Sq : (Specified spindle speed)× (Detection start tolerance (q))
Parameter No.4911, address Q

Parameter FLR = 0	Parameter FLR = 1
$\frac{q}{100}$	$\frac{q}{1000}$

Sr : (Specified spindle speed) × (Allowable variation (r))
Parameter No.4912, address R

Parameter FLR = 0	Parameter FLR = 1
$\frac{r}{100}$	$\frac{r}{1000}$

Si : Allowable variation width
Parameter No.4913, address I

If the difference between the specified speed and actual speed exceeds both Sr and Si, an alarm OH0704 is raised.

- Relationship between spindle speed control and each spindle

Function \ Spindle	Serial spindle	
	1st spindle	2nd spindle
Spindle speed fluctuation detection	Possible	Possible(*1)

> **NOTE**
> 1 It is necessary to enable multi spindle control (bit 3 (MSP) of parameter No. 8133 is 1).
> 2 The spindle speed fluctuation detection function is effective for a single spindle. The function cannot be executed for two or more spindles.
> The spindle speed fluctuation detection function is effective for a spindle on which the currently selected position coder is mounted. Just a single position coder can be selected. Multiple position coders cannot be selected. For the selection of a position coder, see the section of "Multi spindle."
> * Position coder selection signals (PC2SLC<Gn028.7>)
> 3 The parameters that become valid are the parameters of the spindle speed fluctuation detection function (No.4911, No.4912, No.4913, No.4914) for the spindle on which the currently selected position coder is mounted.

- Spindle for which to detect the spindle speed fluctuation

For the spindle for which to detect the spindle speed fluctuation, refer to the appropriate manual provided by the machine tool builder.

9.6 SPINDLE CONTROL WITH SERVO MOTOR

Overview

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.

- Spindle motors and supported functions

Spindle functions	Conventional spindle control	Spindle control with servo motor
Feed per revolution	O	O
Threading	O	O
Polygon machining	O	× *1
Spindle speed fluctuation detection	O	×
Spindle synchronous control	O	×
Simple spindle synchronous control	O	×
Polygon machining with two spindles	O	×
Spindle orientation Multi-point orientation Spindle output switching	O	×
Inter-path spindle control	O	O
Constant surface speed control	O	O
Multi-spindle control	O	O
Rigid tapping	O	O
Spindle output control with PMC	O	O
Actual spindle speed output	O	O
Spindle indexing	×	O

*1 Servo motor can be used as tool rotation axis.
It is not possible to use as a spindle which rotates workpiece.

Notes

(1) This function is optional.
(2) The available functions and spindle axis composition are predetermined according to the machine used. Refer to the manual supplied by the machine tool builder.
(3) If necessary, refer to "SPINDLE CONTROL WITH SERVO MOTOR" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).

9.6.1 Spindle Control with Servo Motor

Specification

- Command with a program

This function can be used to specify the S spindle rotation command for a servo motor spindle in the SV speed control mode. For rotation control of a servo motor, specify G96.4 to start the SV speed control mode. Once the SV speed control mode is specified, the S command is valid for the servo motor until the SV speed control mode is released. Do not specify a positioning command in the SV speed control mode. Otherwise, alarm PS0445 is issued.
Before positioning operation, release the SV speed control mode. To release the SV speed control mode, specify the spindle indexing command G96.1/G96.2. For details on spindle indexing, see "Spindle Indexing Function". When bit 0 (PCE) of parameter No. 11006 is 1, a positioning command can be specified. When bit 0 (PCE) of parameter No. 11006 is 0, a positioning command cannot be specified for a servo motor spindle. If specified, alarm PS0601 is issued.
In a mode other than the SV speed control mode, the S command is invalid. However, since information of the S command is recorded, if the SV speed control mode is entered, rotation starts at the specified rotation speed.

- Command with a signal

SV speed control mode signal <Gn521> can also be used to start and cancel the SV speed control mode. The SV speed control mode is started or canceled on a rising or falling edge of the SV speed control mode signal. Therefore, to start the SV speed control mode again after the SV speed control mode is started by using the signal then canceled by a programmed command, input the SV speed control mode signal again or specify G96.4.

The SV speed control mode status can be checked using the SV speed control mode in-progress signals <Fn521>.

When the signal is set to 0 during rotation, spindle indexing is performed, then the SV speed control mode is canceled. Spindle indexing performs positioning to R0 (absolute value 0). For spindle indexing, see Subsection II-9.6.2, "Spindle Indexing Function".

Format

G96.4 P_ ;	**Start SV speed control mode**
M03(M04) S_ P_ ;	Rotation command

S: Spindle speed [min^{-1}] (numeric value of up to five digits)
P: Spindle selection with multi-spindle control

- Starting the SV speed control mode

When multi-spindle control is enabled and a spindle is selected with address P (Bit 3 (MPP) of parameter No. 3703 = "1"), the SV speed control mode can be started by specifying G96.4 and spindle selection command P. To specify address P, use parameter No. 3781 (P code for selecting a spindle). When a spindle is selected with spindle selection signals SWS1 to SWS2 <Gn027.0 to Gn027.1>, it is possible to start the SV speed control mode if spindle selection signal is input when program is analyzed in the G96.4 command block. In any case, when a servo motor spindle has not been selected in the G96.4 command block, alarm PS0602 is issued.

When multi-spindle control is disabled, it is possible to start the SV speed control mode if G96.4 is independently commanded. When there is no servo motor spindle in the system, alarm PS0602 is issued.

- Canceling the SV speed control mode

When multi-spindle control is enabled and a spindle is selected with address P (Bit 3 (MPP) of parameter No. 3703 = "1"), the SV speed control mode can be canceled by specifying a spindle indexing command (G96.1/G96.2) and spindle selection command P. When a spindle is selected with spindle selection signals SWS1 to SWS2 <Gn027.0 to Gn027.1>, it is possible to cancel the SV speed control mode if spindle selection signal is input when program is analyzed in the spindle indexing command (G96.1/G96.2) block. In any case, when a servo motor spindle has not been selected, alarm PS0602 is issued.

When multi-spindle control is disabled, it is possible to cancel the SV speed control mode if G96.4 is independently commanded. When there is no servo motor spindle in the system, alarm PS0602 is issued.

For the command format of the spindle indexing command, see "Spindle Indexing Function".

- Notes

To specify “G96.4 P_ ;”, use an independent block.

However, it is possible to specify "G96.4 P_ S_ ;".

Moreover, in 2-path system, when it is specified from paths other than the path that a servo motor spindle belongs, alarm PS0602 is issued.

Explanation

- Command

(1) Spindle speed command output

Set up the spindle speed command in the same way as for the ordinary speed command (S command). Before specifying a speed command (S command), start the SV speed control mode. When performing positioning, cancel the SV speed control mode, and select the position control mode.

The sequence by the following input signals is unnecessary.

*ESPA, MRDYA, and SFRA

(2) Condition for stopping spindle speed output

Commands once output to a spindle become 0 if *SSTP becomes “0” or if a command (such as S0) that makes the spindle speed command output 0 is issued. Also, issuing a spindle indexing command (G96.1/G96.2) makes the spindle speed command output 0. In addition, an emergency stop condition and servo alarm bring the spindle to a stop.

With M05, the CNC does not make the command output to the spindle 0.

(3) Stopping rotation by spindle indexing

Issuing a command that specifies a position enables the rotating axis to stop at the specified position. For details, see Subsection, "Spindle Indexing Function."

(4) Maximum speed

The maximum speed that can be specified is usually 2777 min^{-1}.

However, setting bit 3 (IRC) of parameter No. 1408 to "1" may be able to increase the limit to about 27770 min^{-1} depending on the performance of the related motor and detector.

- Multi-spindle control and commands from another path

When a servo motor spindle and another spindle are present together in the same path, the multi-spindle control function is required to specify a rotation command. The path spindle control function can handle commands from another path. The address P-based spindle select function for multi-spindle control can be used to select spindles in another path.

Example program commands follow.

Example 1: (Spindle selection with address P)

Bit 3 (MPP) of parameter No. 3703 = "1": A spindle is selected with address P.

Spindle configuration (S1 = first spindle and S2 = second spindle)

Path 1	Path 2
S1 (spindle axis)	S3 (spindle axis)
S2 (servo motor spindle)	-

Setting address P for spindle selection in multi-spindle control

Parameter	Path 1	Path 2
3781	11 (S1)	21 (S3)
	12 (S2)	-

Program example

Command issuing path	Program	Operation
1	M03 S1000 P12 ;	S2 rotates in normal direction at 1000 min^{-1}.
1	M03 S1500 P21 ;	S3 rotates in normal direction at 1500 min^{-1}.
2	M04 S1500 P11 ;	S1 rotates in reverse direction at 1500 min^{-1}.

- Manual operation of a servo motor spindle

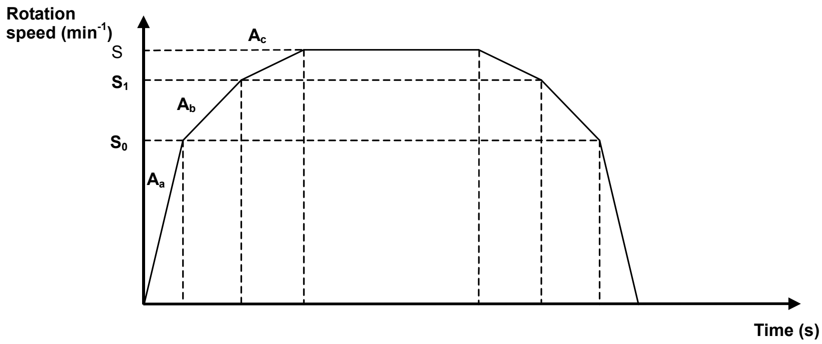
Before performing the manual operation and manual reference position return of a servo motor spindle, release the SV speed control mode. Manual operation and manual reference position return in the SV speed control mode are invalid.

When using an absolute position detector, manual reference position return is not required. If reference position return (G28) is performed in a program when position control is disabled (bit 0 (PCE) of parameter No. 11006 is 0), alarm PS0601 is issued.

> **NOTE**
> If a movement command is specified for a servo motor spindle when position control is disabled (bit 0 (PCE) of parameter No. 11006 is 0), alarm PS0601 is issued.

- Acceleration/deceleration (time constant)

It is possible to change the spindle acceleration/deceleration specified in a rotation command according to the speed of the spindle.
Acceleration/deceleration can be switched at two points of speed, using parameters S0 and S1 (switching speed). In addition, parameters Aa, Ab, and Ac are available to set up three acceleration/deceleration spans.



S_0 : Setting of parameter No. 11020 (acceleration/deceleration is switched at rotation speed S_0 (min^{-1})).
S_1 : Setting of parameter No. 11021 (acceleration/deceleration is switched at rotation speed S_1 (min^{-1})).
S : Specified rotation speed (min^{-1}).
Aa : Setting of parameter No. 11030 (acceleration/deceleration (min-1/s) used between rotation speeds 0 and S_0).
Ab : Setting of parameter No. 11031 (acceleration/deceleration (min-1/s) used between rotation speeds S_0 and S_1).
Ac : Setting of parameter No. 11032 (acceleration/deceleration (min-1/s) used between S1 and the specified speed S).

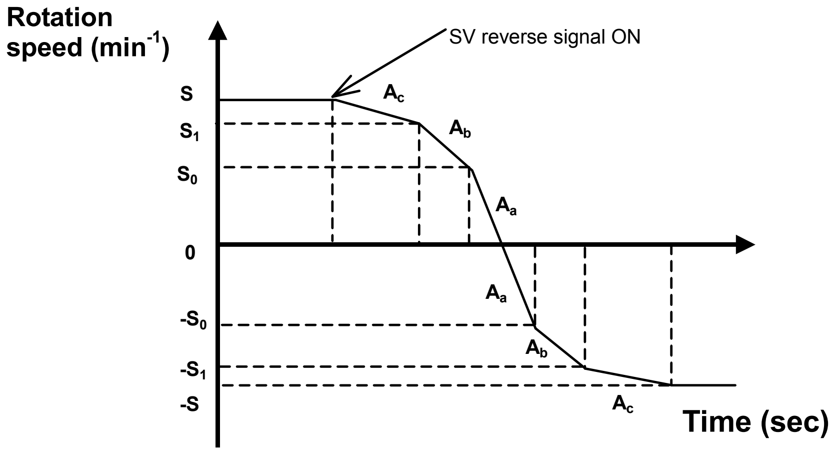
Determine the setting of each parameter according to the torque characteristic of the motor.

- Acceleration/deceleration after interpolation

Acceleration/deceleration after interpolation is available in the SV speed control mode. Bit 1 (TCR) of parameter No.11001 can be used to select a time constant type, that is, parameter No. 1622
(Time constant of acceleration/deceleration in cutting feed for each axis) or parameter No. 11016 (Time constant of acceleration/ deceleration in SV speed control mode for each axis).

- Direction of rotation

The voltage polarity at the time of spindle speed output can be changed using bit 6 (CWM) and bit 7 (TCM) of parameter No. 3706.
It is also possible to reverse the direction of spindle rotation, using the SV reverse signal <Gn523>.
These functions are usable in the rotation mode and during rigid tapping. Changing the signal to ON/OFF during rotation causes the spindle to decelerate, reverse, and then accelerate.



- Display

Bit 3 (NDF) of parameter No. 3115 can be used to specify whether to display the actual speed. This, however, is not considered in the SV speed control mode regardless of the setting of the NDF parameter bit. In addition, bits 0 (NDP) and 1 (NDA) of parameter No. 3115 can be used to specify whether to display, respectively, the current position and the remaining amount of movement.

Restrictions

(1) When multi-spindle control is enabled, in settings other than multi-spindle P type, specifying G96.* P is regarded as invalid.
(2) When starting or canceling the SV speed control mode during automatic operation, be sure to specify the path of the target axis of the SV speed control mode. When it is specified from the other path, alarm PS0602 is issued.
(3) Be sure to set M code which starts or cancels the SV speed control mode to M code preventing buffering (parameter No.3411 to 3420, No.3421 to 3432).
(4) When the SV speed control mode is started or cancelled except for automatic operation, be sure to perform manual reference position return before starting the automatic operation so that the target axis lose its reference position.
The axis may not move correctly to the commanded position in SV speed control cancel mode (position control mode) when the automatic operation is started without performing manual reference position return.

Notes

(1) This function is optional.
(2) The available functions and spindle axis composition are predetermined according to the machine used. Refer to the manual supplied by the machine tool builder.
(3) If necessary, refer to "Spindle Control with Servo Motor" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).
(4) Unlike spindle motors, a servo motor spindle stops rotating when a servo alarm, an emergency stop, or a machine lock occurs.
(5) The maximum motor speed is obtained by applying the feed gear to the maximum motor speed set in parameter No. 11015.
(6) If bit 3 (IRC) of parameter No. 1408 is set to 0, the error prevention function may work when the speed has reached around 2778 min^{-1}. When a speed of 2778 min^{-1} or more is required, set the IRC parameter bit to 1. If IRC is set to 1, it becomes possible to specify a speed of up to approximately 27770 min^{-1}.

9.6.2 Spindle Indexing Function

Format

G96.1 P_ R_ ;	After spindle indexing is completed, the operation of the next block is started.
G96.2 P_ R_ ;	Before spindle indexing is completed, the operation of the next block is started.
G96.3 P_ ;	After it is confirmed that spindle indexing is completed, the operation of the next block is started.

P: Spindle selection with multi-spindle control
R: Stoppage angle [deg] (0 to (parameter No. 1260))

When multi-spindle control is enabled, select a spindle according to the specification of multi-spindle control along with the G96.1/G96.2/ G96.3 command. When a spindle is selected with address P, to specify address P, use parameter No. 3781 (P code for selecting a spindle). In any case, when a servo motor spindle has not been selected, alarm PS0602 is issued.
When multi-spindle control is disabled, it is possible to command G96.4 independently. When there is no servo motor spindle in the system, alarm PS0602 is issued.
In 2-path system, when it is specified from paths other than the path that a servo motor spindle belongs, alarm PS0602 is issued.
To turn the position control mode ON without performing spindle indexing, do so after canceling the SV speed control mode by issuing G96.1 command with no R specified when the motor is at a halt.
When the motor is rotating, issuing a G96.1 (or G96.2) command with no R specified results in the motor coming to a halt by behaving in the same manner as for R0.

NOTE
1. When having specified G96.2, check the completion of the movement by specifying G96.3 before specifying the next move command for the spindle. If the next move command is specified without confirming the completion of the movement, alarm PS0601 is issued. Similarly, if rigid tapping is specified without confirming the completion of the movement, alarm PS0445 is issued.
2. A block specifying G96.1, G96.2, or G96.3 must not contain any other command. If an axis command is specified in the same block, alarm PS0446 is issued.

- Reference position return

Before executing the first command of spindle indexing to the tool rotation axis with a servo motor, be sure to perform a manual reference position return. When the absolute-position detector is used, however, a manual reference position return is not necessary. When the reference position return command (G28) is executed in a program, alarm PS0601 is issued.

Spindle indexing command

- Move command

(1) Command waiting for spindle indexing to be completed
If G96.1 is issued, the next block is executed after spindle indexing is completed.
(2) Command not waiting for spindle indexing to be completed
If G96.2 is issued, the next block can be executed before spindle indexing is completed.

- Movement completion check command

G96.3 is used to check to see if spindle indexing is completed. If it has not been completed, the next block waits for spindle indexing to be completed. If it is completed, the next block is executed.

- SV speed control mode cancellation

If G96.1 is used to perform spindle indexing, the SV speed control mode is canceled when spindle indexing is completed.
If G96.2 is used to perform spindle indexing, G96.3 can be used to check to see if spindle indexing is completed and, if completed, cancel the SV speed control mode. Issuing G96.2 not followed by G96.3 cannot cancel the SV speed control mode even if spindle indexing is completed.
If SV speed control mode cancel, it is necessary to be commanded in the path to which the live tool axis belongs.

Example for specifying the start/cancellation of the SV speed control mode (when bit 3 (MPP) of parameter No. 3703 = 1)

Spindle name	Spindle selection P code (parameter No. 3781)	Servo motor spindle address
S1	P1	C

Specification by programming (SV speed control mode in-progress signal <Fn521>)

Programmed command	Start/cancellation of speed control mode	Operation
G96.4 P1 ;	Start (SV speed control mode in-progress signal (C) = 1)	The SV speed control mode is started (C).
M03 S100 P1 ;	:	Servo motor spindle C turns in the forward direction at 100 [min^{-1}].
:	:	:
G96.1 P1 R0 ;	Cancel (SV speed control mode in-progress signal (C) = 0)	The servo motor spindle C stops when C = 0 (spindle indexing).

Mode switching by signal

Programmed command	Start/cancellation of speed control mode	Operation
M15 ;	Start (SV speed control mode in-progress signal (C) = 1)	The SV speed control mode is started by the M code (C).
M03 S100 P1 ;	:	Servo motor spindle C turns in the forward direction at 100 [min^{-1}].
:	:	:
G96.1 P1 R0 ;	Cancel (SV speed control mode in-progress signal (C) = 0)	The servo motor spindle C stops when C = 0 (spindle indexing).

- Spindle indexing command during spindle rotation

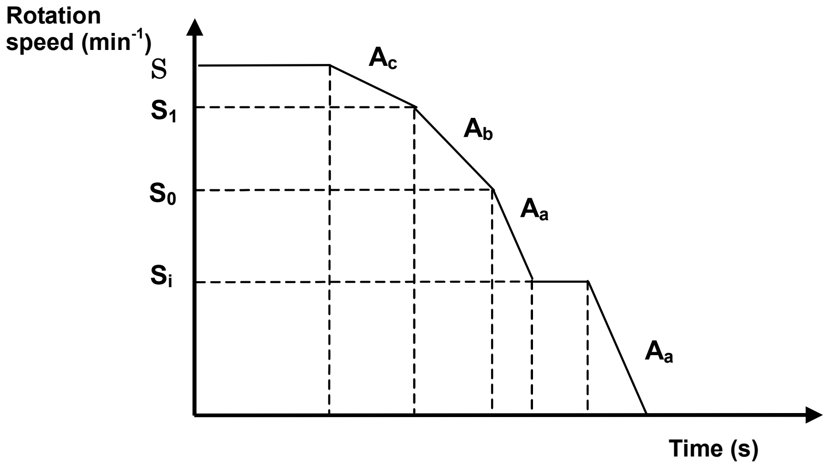
Issuing G96.1 or G96.2 with a position specified during spindle rotation causes the spindle to stop at the specified position.
Example)
M03 S1000 ;... Rotation at S1000
G96.1 P1 R180. ;.. Stoppage of rotation at the 180° position

- Spindle indexing speed

Issuing G96.1 or G96.2 causes a move speed to be dedicated to spindle indexing. Specify the move speed for spindle indexing, using parameter No. 11012.

- Spindle indexing acceleration/deceleration

Shown below is the acceleration/deceleration specified by G96.1/G96.2.



S_1 : Parameter No. 11020 setting (acceleration/deceleration is switched at a rotation speed of S_1 (min^{-1}).)
S_0 : Parameter No. 11021 setting (acceleration/deceleration is switched at a rotation speed of S_0 (min^{-1}).)
S : Command-specified rotation speed (min^{-1})
Si : Spindle indexing speed (min-1) parameter No. 11012
Aa : Parameter No. 11030 setting (acceleration/deceleration (min-1/s) used between rotation speeds 0 and S_0 (span 1))
Ab : Parameter No. 11031 setting (acceleration/deceleration (min-1/s) used between rotation speeds S_0 and S_1 (span 2))
Ac : Parameter No. 11032 setting (acceleration/deceleration (min-1/s) used between S1 and a specified speed S (span 3)))

- Example program commands

(1) A move command is issued to the spindle, using G96.2. The spindle starts moving, and the execution of the next block begins.
The spindle keeps moving even when any other block is being executed.
(The spindle indexing signal SPP<Fn522> is "1" during spindle indexing.)
(2) When another command is issued to the spindle, G96.3 is used to heck in advance whether the spindle has finished moving.
If the spindle is still moving (the spindle indexing signal is on), the CNC enters a wait state. If the spindle has finished moving, the command is issued to cause the spindle to start moving.

Example: Command not waiting for spindle indexing to finish and command checking whether spindle indexing has finished (parameter No. 3781 (S1) = "1")

Program command	Operation
G96.2 P1 R270.0 ;	Command not waiting for spindle indexing to finish. The first spindle S1 moves to 270.0.
G01 X10.0 Y20.0 F1000. ;	Starts cutting feed. No wait for spindle indexing to finish.
G02 X50.0 Y100.0 R50.0 ;	Starts circular interpolation. No wait for spindle indexing to finish.
G96.3 P1 ;	Checks whether spindle indexing has finished.
M29 S100 P1 ;	Starts rigid tapping if the spindle indexing signal is "0".
G84 X10.0 Y 20.0 R-5.0 Z-20.0 ;	Waits for rigid tapping command if the spindle indexing signal is "1".

Indexing by machine coordinates

In the function for spindle control with servo motor, the following operations can be selected:
(1) When the speed control mode is off, indexing to machine coordinate 0.000 is performed.
(2) When G code-based spindle indexing has been performed, an R command indicates a machine coordinate value.

Spindle indexing operation

Using bit 0 (SIC) of parameter No. 11005 can select which coordinate system, absolute or machine, is to be used in spindle indexing.

Example:

If the difference between the machine and absolute coordinates (machine coordinate – absolute coordinate) is 100.000:

- Spindle indexing performed by turning the SV speed control mode signal <Gn521> OFF (1 t□ 0) or issuing a spindle indexing
  G code (with no R specified)
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 0.000 and an absolute coordinate of 260.000.
- G code-based spindle indexing (with R specified)
  Assuming that spindle indexing is specified with R100.000:
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 200.000 and an absolute coordinate of 100.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.

Notes

(1) If G96.2 (not waiting for spindle indexing to finish) is issued, G96.3 must be issued to check whether spindle indexing has finished. Be sure to issue G96.3 after G96.2. If a movement along an axis is specified without checking (without issuing the G96.3 command), alarm PS0601 is issued. Similarly, if rigid tapping is specified, alarm PS0445 is issued.

(2) If it has not been checked, for example, because of a reset, whether spindle indexing has finished after G96.2 (not waiting for spindle indexing to finish) is issued, the SV speed control mode is not canceled.

(3) If the spindle stops rotating because of spindle indexing, the spindle speed command output becomes 0. To cause the spindle to restart rotating, place the spindle in the SV speed control mode, and then, issue an S command.

(4) Spindle indexing is enabled only in the SV speed control mode.

(5) When the spindle indexing speed (parameter No. 11012) is 0, the acceleration/deceleration switching speed (1st step) (parameter No. 11020) is the spindle indexing speed. When the acceleration/deceleration switching speed is also 0, the maximum speed (parameter No. 3741) of each spindle that corresponds to gear 1 is the spindle indexing speed.

(6) The G96.* P command is invalid for settings other than multi-spindle control type P.

(7) The SV speed control mode during automatic operation must be switched by a command from the path to which it belongs. When it is specified from the other path, alarm PS0602 is issued.

9.6.3 Rigid Tapping with Servo Motor

Format

The command format for this type of rigid tapping is the same as for the conventional type of rigid tapping.

For details, refer to the chapter of "RIGID TAPPING" in the OPERATOR’S MANUAL (Lathe System) (B-64304EN-1) or OPERATOR’S MANUAL (Machining Center System) (B-64304EN-2).

NOTE

Before rigid tapping can be specified, the SV speed control mode for the servo motor spindle must be canceled. If rotation is in progress, use G96.1/G96.2 to cancel the SV speed control mode. The mode of the servo motor spindle can be checked by checking the SV speed control mode in-progress signal (SVREV<Fn521>).
If rigid tapping is specified in the SV speed control mode, alarm PS0445 is issued.

Rigid tapping specification

- Feedrate

For rigid tapping, the feedrate of a drilling axis is one specified in an F command, and that of the spindle is S × the amount of movement per live tool axis (servo motor spindle) rotation [deg/min] (parameter No. 11011).

Feed per minute and feed per revolution are detailed later.
During rigid tapping, the spindle speed is limited by a parameter for specifying the maximum cutting feedrate for the axis used as a live tool axis, that is, parameter No. 1430 (or No. 1432 if acceleration/deceleration before interpolation is enabled).
Normally, the parameter of the maximum cutting feedrate (parameter No. 1430) (parameter No. 1432 when acceleration/deceleration before interpolation is enabled) can be set to up to 999999.999 [deg/min] (equivalent to S2778 [min^{-1}]). However, for an axis set as the live tool axis (bit 3 (IRC) of parameter No. 1408 = 1) that is used for rigid tapping, the maximum feedrate is limited to ten times as large as the setting of the maximum cutting feedrate parameter.
Example:
Maximum cutting feedrate parameter No. 1430 = 360000
Limit to the maximum spindle speed 360000×10 = 3600000 [deg/min] (S10000 [min^{-1}])

⚠ **CAUTION**

Make the tapper thread pitch equal to one specified by the program (F, S).
Otherwise, the tool or workpiece may be damaged.

Acceleration/deceleration control

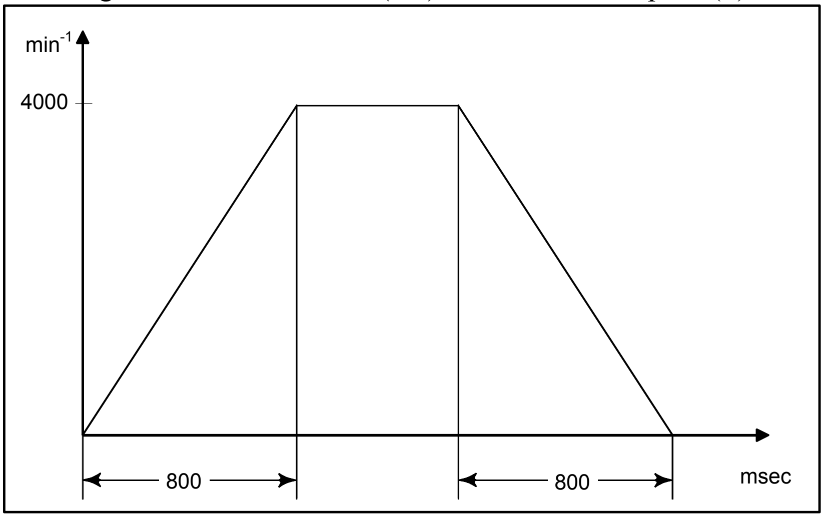
- Acceleration/deceleration after interpolation

Unlike conventional rigid tapping (with a spindle motor), rigid tapping with a servo motor allows the application of linear acceleration/deceleration with constant acceleration/deceleration time or bell-shaped acceleration/deceleration.
Resetting bit 0 (SRBx) of parameter No. 11001 to "0" makes it possible to apply linear acceleration/deceleration after interpolation of constant acceleration time type. Setting the bit to "1" makes it possible to apply bell-shape acceleration/deceleration after interpolation of constant acceleration time type.
The time constant for each gear is specified in parameter Nos. 11060 to 11063. If bit 2 (TDR) of parameter No. 5201 = "1", the tool extraction time constant for each gear is specified in parameter Nos. 11065 to 11068. Specify each of these parameters for the live tool axis (servo motor spindle) used in rigid tapping.
The acceleration/deceleration types and time constants used for drilling axes are set to the same values as for live tool axes (servo motor spindles).

NOTE

This type of rigid tapping and conventional rigid tapping (with a spindle motor) differ in time constant setting.

Example: Parameter settings are: Time constant (TC) = 800 msec and speed (S) = 4000 min^{-1}



- Acceleration/deceleration before interpolation

In this type of rigid tapping, when advanced preview control can be used, rigid tapping can be specified in the mode for acceleration/deceleration before look ahead interpolation to allow acceleration/deceleration before look ahead interpolation during rigid tapping. Acceleration/deceleration before look ahead interpolation is enabled when the advanced preview control mode is set to on. For the advanced preview control function, see "High-Speed High-Precision Function (Advanced Preview Control)".

Parameter No. 11050 is used to specify the maximum allowable acceleration value for acceleration/deceleration interpolation used in rigid tapping. Parameter No. 11051 is used to specify the acceleration change time for bell-shape acceleration/deceleration before interpolation. As for acceleration/deceleration before interpolation used in rigid tapping, the maximum allowable acceleration value is 100000 [deg/s^2].

It is possible to change a speed of up to S1000 [min^{-1}] (equivalent to 360000 [deg/min] in 60 [ms].

The time constant (parameter No. 11052) for cutting feed acceleration/deceleration after interpolation usable in the "acceleration/deceleration before look-ahead interpolation" mode is a constant-time type.

> **NOTE**
> Specify the same time constant for both drilling and live tool axes (servo motor spindle). Otherwise, it is likely that the machine may malfunction.

Spindle indexing

This function does not allow the spindle orientation function to be performed at the start of rigid tapping. Before specifying rigid tapping, perform spindle indexing to the position at which tapping is to be performed. For details, see "Spindle Indexing Function" described previously.

Notes

If the pitch is very small or the amount of travel along the drilling axis is large, the amount of travel along the rotation axis becomes large, possibly resulting in alarm PS0003.

9.6.4 Feed per Revolution

Overview

The function for spindle control with servo motor allows a feed per revolution to be specified.

From the separate detector connected to the spindle, the rate of feed per revolution is obtained. When the detector built into the servo motor is to be used, the feedrate is obtained based on the servo motor speed and gear ratio.
Which detector to use is specified by bit 1 (OPTx) of parameter No. 1815.

> **NOTE**
> If the machine is equipped with a live tool axis and spindle with servo motor, using rotation commands requires the multi-spindle control function.
> As for gear change, use T type gear change.
> See Section, “MULTI-SPINDLE CONTROL,” in Connection Manual (Function) (B-64303EN-1) for explanations about multi-spindle control.

Format

The command format for feed per revolution explained above is the same as for the ordinary types of feed per revolution.

- Examples

When the second spindle (C-axis) is used for rotation control and position control with the second spindle assumed to be a servo motor spindle
(The G code system A in lathe system is selected.)

Program command	Operation
M*** ;	The C axis rotation control mode is turned ON.
M03 S100 P2 ;	The servo motor spindle (C axis) rotates at 100 min^{-1}.
G99 G01 Z-100. F10. ;	The Z axis moves at a feed-per-revolution speed of 1000 mm/min.
:	:
M*** ;	The C axis position control mode is set to ON (rotation control mode OFF). The spindle stops with C = 0.000.

9.6.5 Spindle Output Control with PMC

Overview

The “spindle control with servo motor” function enables the PMC to be sued for spindle output control.

How to specify

After the start of the SV speed control mode, this function can be specified in the same way as in ordinary spindle control (with a spindle motor).
Refer to Section, “SPINDLE OUTPUT CONTROL BY THE PMC,” in this manual for detailed descriptions.

> **NOTE**
> For the “spindle control with servo motor” function, the maximum motor speed is one specified in parameter No.11015.

10 TOOL FUNCTION (T FUNCTION)

Chapter 10, "TOOL FUNCTION (T FUNCTION)", consists of the following sections:

10.1 TOOL SELECTION FUNCTION

By specifying an up to 8-digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool. This is used to select tools on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commendable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and T function commands.

(ii) Executing T function commands upon completion of move command execution.

The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

Explanations

T

The value after the T code indicates the desired tool. Part of the value is also used as a tool offset number that specifies the amount of tool offset or the like. The tool can be selected as follows according to the specification method and parameter settings.

<table>
<tr><th colspan="2">Description of a T code (Note 1)</th><th rowspan="2">How to specify the offset number for each parameter setting (Note 2)</th></tr>
<tr><th>LGN (No.5002#1) = 0</th><th>LGN (No.5002#1) = 1</th></tr>
<tr><td rowspan="2">TOOOOOOO O
↑ Tool selection
↑ Tool geometry tool wear offset</td><td rowspan="2">TOOOOOOO O
↑ Tool selection tool geometry offset
↑ Tool wear offset</td><td>The tool wear offset number is specified using the low-order one digit of a T code.</td></tr>
<tr><td>When parameter (No.5028) is set to 1</td></tr>
<tr><td rowspan="2">TOOOOOO OO
↑ Tool selection
↑ Tool geometry tool wear offset</td><td rowspan="2">TOOOOOO OO
↑ Tool selection tool geometry offset
↑ Tool wear offset</td><td>The tool wear offset number is specified using the low-order two digits of a T code.</td></tr>
<tr><td>When parameter (No.5028) is set to 2</td></tr>
<tr><td rowspan="2">TOOOOO OOO
↑ Tool selection
↑ Tool geometry tool wear offset</td><td rowspan="2">TOOOOO OOO
↑ Tool selection tool geometry offset
↑ Tool wear offset</td><td>The tool wear offset number is specified using the low-order three digits of a T code.</td></tr>
<tr><td>When parameter (No.5028) is set to 3</td></tr>
</table>

NOTE

1 The maximum number of digits of a T code can be specified by parameter (No.3032) as 1 to 8.

2 When parameter (No.5028) is set to 0, the number of digits used to specify the offset number in a T code depends on the number of tool offsets.

Example)

When the number of tool offsets is 1 to 9:	Low-order one digit
When the number of tool offsets is 10 to 99:	Low-order two digits
When the number of tool offsets is 100 to 200:	Low-order three digits

Refer to the machine tool builder's manual for correspondence between the T-code and the tool and the number of digit to specify tool selection.

Example (T2+2)

N1 G00 X1000 Z1400 ;
N2 T0313 ; (Select tool No. 3 and tool offset value No. 13)
N3 X400 Z1050 ;

10.2 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.

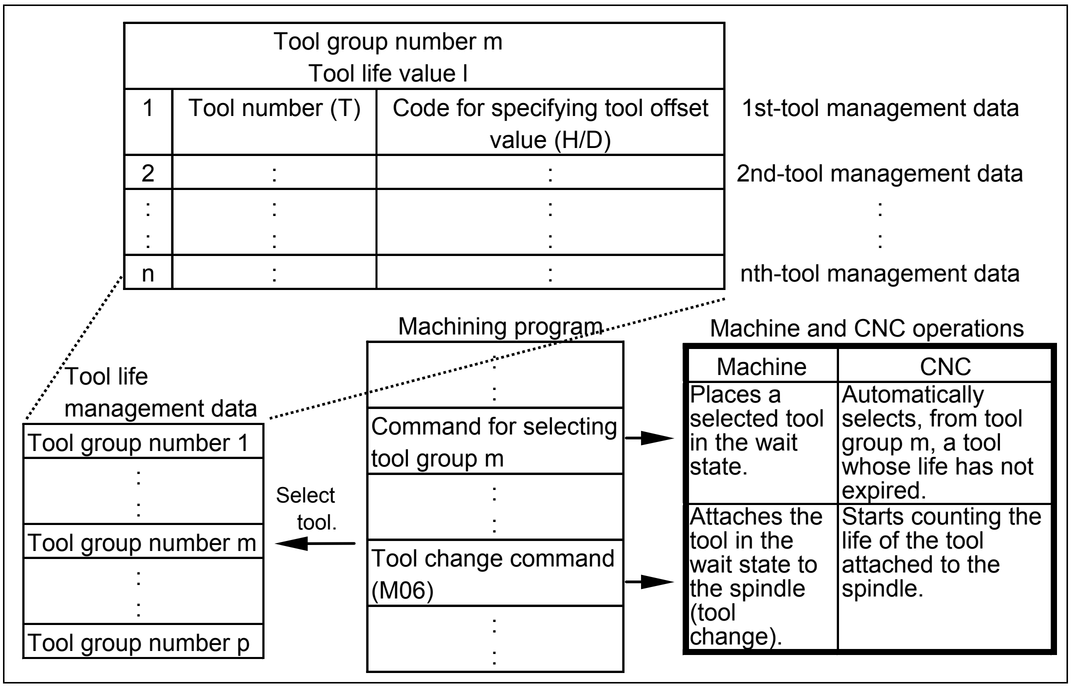


Fig. 10.2 (a) Tool selection from machining program

M

A group is selected by a T code, and tool life counting is started by the M06 command. (ATC type)

T

A group is selected, tool compensation is specified, and tool life counting is started only by a T code. (turret type)

- Maximum number of tool life management groups and 2-path system

A maximum of 128 tool life management groups can be used for each path.
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.

> ⚠ **CAUTION**
> When parameter No. 6813 is changed and the power is turned on, all data in the tool life management file is initialized. So, it is necessary to set life management data for all paths that use tool life management.

10.2.1 Tool Life Management Data

Tool life management data consists of tool group numbers, tool numbers, codes for specifying a tool offset value, and tool life values.

Explanation

- Tool group number

A maximum of 128 tool life management groups can be used for each path.
Set the maximum number of groups to be used in parameter No. 6813. Up to two tools can be registered for each of a maximum number of groups. If bit 0 (GS1) of parameter No. 6800 and bit 1 (GS2) of parameter No. 6800 is set, it is possible to change the combination of the number of registrable groups and the maximum number of tools.

Table 10.2.1 Maximum numbers of registrable groups and tools

GS2 (No. 6800#1)	GS1 (No. 6800#0)	Number of groups	Number of tools
0	0	1 to Maximum number of groups (parameter No.6813) x 1/8	1 to 16
0	1	1 to Maximum number of groups (parameter No.6813) x 1/4	1 to 8
1	0	1 to Maximum number of groups (parameter No.6813) x 1/2	1 to 4
1	1	1 to Maximum number of groups (parameter No.6813)	1 to 2

> ⚠ **CAUTION**
> After changing the setting of bit 0 (GS1) and bit 1 (GS2) of parameter No. 6800, re-register tool life management data by issuing the G10L3 (registration after deletion of data for all groups).
> Otherwise, the newly set combination does not become valid.

- Tool number

A tool number is specified with a T code. A number consisting of up to eight digits (99999999) can be specified.

> **NOTE**
> The maximum number of digits usable in the T code is specified in parameter No. 3032.

- Codes for specifying a tool offset value

M

Codes for specifying a tool offset value include an H code (for tool length offset) and a D code (for cutter compensation). Numbers up to 400 (up to three digits long) can be registered as codes for specifying tool offset values.

> **NOTE**
> If codes for specifying tool offset values are not used, the registration of these codes can be omitted.

T

Neither H code nor D code is used as a code for specifying tool offset values. The T code includes a compensation code.

- Tool life value

A tool life value can be registered in terms of use duration or use count. The maximum value is as follows:
Up to 4300 minutes can be registered if duration specification is selected, or up to 65535 times can be registered if use count specification is selected.

- Remaining life setting

Parameters Nos. 6844 and 6845 are used to set the life remaining until a new tool is selected.

10.2.2 Registering, Changing, and Deleting Tool Life Management Data

By programming, tool life management data can be registered in the CNC, and registered tool life management data can be changed or deleted.

Explanation

The program format varies depending on the following four types of operation:

- Registration after deletion of all groups

After all registered tool life management data is deleted, programmed tool life management data is registered.

- Change of tool life management data

Tool life management data can be set for a group for which no tool life management data is registered, and already registered tool life management data can be changed.

- Deletion of tool life management data

Tool life management data can be deleted.

- Setting of tool life count type

A count type (use duration or use count) can be set for each group separately.

Format

- Registration after deletion of all groups

M

Format	Meaning
G10 L3 ;	G10L3: Register data after deleting data of all groups.
P- L- ;	P-: Group number
T- H- D- ;	L-: Tool life value
T- H- D- ;	T-: Tool number
:	H-: Code for specifying tool offset value (H code)
P- L- ;	D-: Code for specifying tool offset value (D code)
T- H- D- ;	G11: End of registration
T- H- D- ;	
:	
G11 ;	
M02(M30) ;	

T

Format	Meaning
G10 L3 ;	G10L3: Register data after deleting data of all groups.
P- L- ;	P-: Group number
T- ;	L-: Tool life value
T- ;	T-: Tool number and tool offset number
:	G11: End of registration
P- L- ;	
T- ;	
T- ;	
:	
G11 ;	
M02(M30) ;	

If more than one offset value is to be used for the same tool within one process, specify the command as shown below (on the assumption that tool offset numbers are two digits long).

Format	Meaning
G10 L3 ;	
P- L- ;	
T0101 ;	Tool number 01, tool offset number 01
T0102 ;	Tool number 01, tool offset number 02
T0103 ;	Tool number 01, tool offset number 03
:	
G11 ;	
M02(M30) ;	

- Change of tool life management data

M

Format	Meaning
G10 L3 P1 ;	G10L3P1: Start changing group data.
P- L- ;	P-: Group number
T- H- D- ;	L-: Tool life value
T- H- D- ;	T-: Tool number
:	H-: Code for specifying tool offset value (H code)
P- L- ;	D-: Code for specifying tool offset value (D code)
T- H- D- ;	G11: End of change of group
T- H- D- ;	
:	
G11 ;	
M02(M30) ;	

T

Format	Meaning
G10 L3 P1 ; P- L- ; T- ; T- ; : P- L- ; T- ; T- ; : G11 ; M02(M30) ;	G10L3P1: Start changing group data. P-: Group number L-: Tool life value T-: Tool number and tool offset number G11: End of registration

- Deletion of tool life management data

Format	Meaning
G10 L3 P2 ; P- ; P- ; P- ; P- ; : G11 ; M02(M30) ;	G10L3P2: Start deleting group data. P-: Group number G11: End of deletion

- Setting of tool life count type

Format	Meaning
G10 L3 ; (or G10 L3 P1) ; P- L- Q- ; T- H- D- ; T- H- D- ; : G11 ; M02(M30) ;	Q: Life count type (1: Use count. 2: Duration.)

⚠ **CAUTION**

If the Q command is omitted, the life count type is set according to the setting of bit 2 (LTM) of parameter No. 6800.

Tool life value

A tool life value is registered as a duration or a use count according to the setting of bit 2 (LTM) of parameter No. 6800 or the setting of a count type (Q command). The maximum values are listed below.

Table 10.2.2 (a) Life count types and maximum life values

LFB (No.6805#4)	LTM (No.6800#2)	Life count type	Maximum life value
0	0	Use count specification	65535 times
	1	Duration specification	4300 minutes

If the count type is duration specification, the unit of a life value specified with address L in a program may be one minute or 0.1 second, which is determined by the setting of bit 1 (FGL) of parameter No. 6805.

Table 10.2.2 (b) Life value unit and maximum value in L command

LFB (No.6805#4)	FGL (No.6805#1)	Life value unit	Maximum value in L command	Example
0	0	1 minute	4300	L100: Life value is 100 minutes.
	1	0.1 second	2580000	L1000: Life value is 100 seconds.

10.2.3 Tool Life Management Commands in Machining Program

Explanation

M

- Commands

The following commands are used for tool life management:

T○○○○○○○○ ;

Specifies a tool group number.
The tool life management function selects, from a specified group, a tool whose life has not expired, and outputs its T code signal.
In ○○○○○○○○, specify the sum of the tool life management ignore number specified in parameter No. 6810 and a desired group number.
Example:
To specify tool group number 1 when the tool life management ignore number is 100, specify "T101;".

> **NOTE**
> If ○○○○○○○○ is not greater than a tool life management ignore number, the T code is treated as an ordinary T code.

M06;

Terminates tool life management for the previously used tools, and begins counting the life of a new tool selected with the T code.

> **NOTE**
> 1 M06 is treated as an M code not involving buffering.
> 2 If more than one M code is to be specified within the same block, specify M06 first among these M codes.

H99;

Selects the H code registered in tool life management data for the currently used tool to enable tool length offset. Parameter No. 13265 can be used to enable compensation according to an H code other than H99.

H00;

Cancels tool length offset.

D99;

Selects the D code registered in tool life management data for the currently used tool to perform cutter compensation.
Parameter No. 13266 can be used to enable compensation according to a D code other than D99.

D00;

Cancels cutter compensation.

> **NOTE**
> H99 and D99 must be specified after the M06 command. If a code other than the H/D code set in H99/D99 or parameters Nos. 13265 and 13266 is specified after M06, the H code or D code of tool life management data is not selected.

- Types

For tool life management, the four tool change types (types A to D) listed below are used. Which type to use varies from one machine to another. For details, refer to the relevant manual of each machine tool builder.

Table 10.2.3 Differences among tool change types

Tool change type	A		B		C		D	
Parameters M6T and M6E M6T (No.6800#7) M6E (No.6801#7)	M6T	M6E	M6T	M6E	M6T	M6E	M6T	M6E
	0	0	1	0	1	0		1
Tool group number specified in the same block as the tool change command (M06)	Tool group already used		Tool group to be used next					
Time when tool life is counted	Life counting is performed for a tool in the specified tool group if M06 is specified next.						Life counting is performed if a tool in the tool group specified in the same block as M06 is specified.	
Remarks	If the T command (retract tool group) following the M06 command is not the currently used tool group, alarm PS0155 is issued (if bit 6 (IGI) of parameter No. 6800 = 0).		Normally, if a tool group number command is specified alone, type B is used. However, even if a tool group number command is specified alone with type C, no alarm is issued. (This means that there is no difference in operation between types B and C.)				If M06 is specified alone, alarm PS0153 is raised.	

> **NOTE**
> If a tool group number is specified and a new tool is selected, the new tool selection signal is output.

T

- Commands

TOOOOOO99;

The tool life management function ends counting the life of the tool used so far, selects, from the group specified by OOOOOO, a tool whose life has not expired, outputs the T code signal for the tool, and starts counting the life of the tool.

Example:

Suppose that a T199 command (with a tool offset specified with the lower two digits) is issued to cause the tool life management function to select T10001 of tool group 1. Then, T code 100 is output, and the tool offset number 1 is selected.

If the condition that a new tool be selected is not met, and the second or subsequent selection of the same group is made since the entry of the control unit into the automatic operation start state from the reset state, the next to the currently selected T code is selected if more than one offset is registered.
If the third selection is made, for example, the third offset is selected from among multiple offsets registered for the same tool.

Example:

As shown below, suppose that two T codes (with a tool offset specified in the lower two digits) having the same tool number and multiple offset numbers are set in group 1.

T10001 T10002

The first T199 command issued since the entry of the control unit into the automatic operation start state from the reset state selects the first T code, T10001. Then, if T199 is issued again before the control unit is reset, the second T code, T10002, is selected. Furthermore, if T199 is issued again before the control unit is reset, the second T code, T10002, is selected because no third offset is present.
Setting bit 1 (TSM) of parameter No. 6801 set to 1 enables life counting to be performed for each T code separately even if T codes specifying multiple offsets for the same number are registered.

T○○○○○○88;

The offset of the tool whose life is currently counted by tool life management is canceled. The tool offset code is set to 00, and the tool number is output as a T code signal.

Example:

Suppose that the tool number of the tool currently used by the tool life management function is 100. Then, issuing a T188 command (with a tool offset specified with the lower two digits) outputs T code 100 and selects offset number 0, canceling the offset.

> **NOTE**
> If life counting is not performed, or if the specified tool does not belong to the group for which life counting is being performed, alarm PS0155 is issued.

The numbers of digits in ○○○○○○ and 99/88 vary as follows:

No.5028	99	88
1	T○○○○○○○ 9 ↑ ↑ Select group Start life counting	T○○○○○○○ 8 ↑ ↑ Select group Cancel tool offset
2	T○○○○○○ 99 ↑ ↑ Select group Start life counting	T○○○○○○ 88 ↑ ↑ Select group Cancel tool offset
3	T○○○○○ 999 ↑ ↑ Select group Start life counting	T○○○○○ 888 ↑ ↑ Select group Cancel tool offset

The maximum number of digits in T codes is set in parameter No. 3032.
The number of digits used for specifying an offset number is selected by parameter No. 5028. If 0 is selected, the number of digits depends on the number of tool offsets.
Example:

If there are one to nine tool offsets: Lowest digit
If there are 10 to 99 tool offsets: Lower two digits
If there are 100 to 200 tool offsets: Lower three digits

NOTE

Offset start and cancel operations involve compensation by moving a tool or by shifting the coordinate system. Using bit 6 (LWM) of parameter No. 5002 can select whether to perform a compensation operation if a T code is specified or if an axis move command is specified.
For details, refer to Subsection 5.1.5, “Offset Operation,” of the lathe system OPERATOR’S MANUAL (B-63944EN-1).

TOOOOOOΔΔ;

If the tool offset number in ΔΔ is neither 99 nor 88, the T code is treated as an ordinary T code. If life counting is being currently performed, it is ended.

Examples

M

- Tool change type A

If a block specifying a tool change command (M06) also contains a tool group command (T code), the T code is used as a command for returning the tool to its cartridge. By specifying a tool group number with a T code, the number of the tool used so far is output as a T code signal. If the specified tool number is not a tool number of the tool group of the tool being used, alarm PS0155 is issued. The alarm, however, can be suppressed by setting bit 6 (IGI) of parameter No. 6800 to 1.

Example: Suppose that the tool life management ignore number is 100.	
T101 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
M06 ;	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
T102 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
M06 ;	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
T101 ;	The number of the tool currently used (in group 1) is output with a T code signal. (Tool
:	number 010 is output.)
T103 ;	A tool whose life has not expired is selected from group 3.
:	(Suppose that tool number 200 is selected.)
M06 ;	The number of the tool currently used (in group 3) is output with a T code signal. (Tool
:	number 200 is output.)
T102 ;	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
G43 H99 ;	Tool length offset value for the tool selected from group 3 is used.
:	Cutter compensation value for the tool selected from group 3 is used.
G41 D99 ;	Cutter compensation is canceled.
:	
D00 ;	Tool length offset is canceled.
:	
H00 ;	

- Tool change types B and C

If a block specifying a tool change command (M06) also contains a tool group command (T code), the T code is used to specify a tool group number for which life counting is to be performed by the next tool change command.

Example: Suppose that the tool life management ignore number is 100.	
T101 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
M06 T102 ;	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
:	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
M06 T103 ;	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
:	A tool whose life has not expired is selected from group 3.
:	(Suppose that tool number 200 is selected.)
G43 H99 ;	Tool length offset value for the tool selected from group 2 is used.
:	Cutter compensation value for the tool selected from group 2 is used.
G41 D99 ;	Cutter compensation is canceled.
:	
D00 ;	Tool length offset is canceled.
:	
H00 ;	Tool life counting is performed for the tool in group 3.
:	(The life of tool number 200 is counted.)
M06 T104 ;	A tool whose life has not expired is selected from group 4.
:	
:	

- Tool change type D

For a tool selected by a tool group command (T code), life counting is performed by a tool change command (M06) specified in the same block as the tool group command. Specifying a T code alone does not results in an alarm; however, specifying an M06 command alone results in alarm PS0153. However, the alarm can be suppressed by setting bit 7 (TAD) of parameter No. 6805 to 1.

Example: Suppose that the tool life management ignore number is 100.	
T101 M06 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
:	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
T102 M06 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
:	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
G43 H99 ;	Tool length offset value for the tool selected from group 2 is used.
:	Cutter compensation value for the tool selected from group 2 is used.
G41 D99 ;	Cutter compensation value is canceled.
:	
D00 ;	Tool length offset is canceled.
:	
H00 ;	A tool whose life has not expired is selected from group 3.
:	(Suppose that tool number 200 is selected.)
T103 M06 ;	Tool life counting is performed for the tool in group 3.
:	(The life of tool number 200 is counted.)
:	
:	

T

Example: Suppose that offset numbers are two digits long.	
T0199 ; : : : : : :	A tool whose life has not expired is selected from group 1. (Suppose that T1001 is selected. The tool number is 10, and the offset number is 01.) Tool life counting is performed for the tool in group 1. (The life of tool number 10 is counted.) The offset of the tool being used in group 1 is canceled.
T0188 ; : : : :	(Since the tool being used is T1001, the tool number is 10, and the offset number is 00.) A tool whose life has not expired is selected from group 2. (Suppose that T2002 is selected. The tool number is 20, and the offset number is 02.)
T0299 ; : : : : :	Tool life counting is performed for the tool in group 2. (The life of tool number 20 is counted.) If more than one offset number is specified for the currently used tool in group 2, the next offset number is selected. (Suppose that T2002 and T2003 are registered with tool number 20. In this case, T2003
T0299 ; : : : : : :	is selected. The tool number is 20, and the offset number is 03.) Life counting for the tool in group 2 ends, and this command is treated as an ordinary T code. (The tool number is 03, and the offset number is 01.)
T0301 ; : :	

10.2.4 Tool Life Counting and Tool Selection

Either use count specification or duration specification is selected as the tool life count type according to the state of bit 2 (LTM) of parameter No. 6800. Life counting is performed for each group separately, and the life counter contents are preserved even after the power is turned off.

Table 10.2.4 Tool life management count types and intervals

Tool life count type	Use count specification	Time specification
Bit 2 (LTM) of parameter No. 6800	0	1
Life count interval	Incremented by one for tools used in one program Counting can be resumed by using tool life count restart M code (parameter No. 6811).	Bit 0 (FCO) of parameter No. 6805 0: Every second 1: Every 0.1 second Can be changed by override.

Explanation

M

- Use count specification (LTM=0)

If a tool group (T code) is specified, a tool whose life has not expired is selected from the specified tool group. The life counter for the selected tool is then incremented by one by a tool change command (M06). Unless a tool life count restart M code is specified, selection of a new tool and the increment operation can be performed only if a tool group number command and a tool change command are issued for the first time since the entry of the control unit into the automatic operation start state from the reset state.

> ⚠ **CAUTION**
> No matter how many times the same tool group number is specified in a program, the use count is not incremented, and no new tool is selected.

- Duration specification (LTM=1)

After all registered tool life management data is deleted, programmed tool life management data is registered.
If a tool group command (T code) is specified, a tool whose life has not expired is selected from the specified tool group. Then, life management for the selected tool is started by a tool change command (M06). Tool life management (counting) is performed by measuring the time during which the tool is actually used in the cutting mode at regular intervals (every second or 0.1 second). The tool life count interval is specified by bit 0 (FCO) of parameter No. 6805. The time required for single block stop, feed hold, rapid traverse, dwell, machine lock, and interlock operations is not counted.
Setting bit 2 (LFV) of parameter No. 6801 enables the life count to be overridden as directed by tool life count override signals. An override from 0 times to 99.9 times can be applied. If 0 times is specified, counting is not performed.

T

- Use count specification (LTM=0)

If a tool group command (T○○99 code) is issued, a tool whose life has not expired is selected from the specified tool group, and the life counter for the selected tool is incremented by one. Unless a tool life count restart M code is specified, selection of a new tool and the increment operation can be performed only if a tool group number command and a tool change command are issued for the first time since the entry of the control unit into the automatic operation start state from the reset state.

> ⚠ **CAUTION**
> No matter how many times the same tool group number is specified in a program, the use count is not incremented, and no new tool is selected.

- Duration specification (LTM=1)

If a tool group command (T○○99 code) is specified, a tool whose life has not expired is selected from the specified tool group, and tool life management for the selected tool starts.
Life management (counting) is performed by measuring the time during which the tool is actually used in the cutting mode at regular intervals (every second or 0.1 second). The life count interval is specified by bit 0 (FCO) of parameter No. 6805. The time required for single block stop, feed hold, rapid traverse, dwell, machine lock, and interlock operations is not counted.
Setting bit 2 (LFV) of parameter No. 6801 enables the life count to be overridden according tool life count override signals. An override from 0 times to 99.9 times can be applied. If 0 times is specified, counting is not performed.

> **NOTE**
> 1 When a tool is selected, tools are searched starting from the current tool toward the last tool to find a tool whose life has not expired. When the last tool is reached during this search, the search restarts from the first tool. If the search does find any tool whose life has not expired, the last tool is selected.
> When the tool currently used is changed by the tool skip signal, the next new tool is selected using the method described here.

NOTE

2 If tool life counting indicates that the life of the last tool in a group has expired, the tool change signal is output. If the life count type is duration specification, the signal is output as soon as the life of the last tool in the group has expired. If the life count type is use count specification, the signal is output when the CNC is reset by a command such as M02 or M30 or when the tool life count restart M code is specified after the life of the last tool in the group has expired.

3 If a T command is specified, a group and a tool in the group are selected while the T command is buffered. This means that if a block to be buffered contains a T command specifying a group while machining is being performed with that group selected, the next T command is already buffered even if the tool life expires during machining, so the next tool is not selected. To prevent this, if the life count type is duration specification and a T command is to be specified to select the same group successively, insert an M code for suppressing buffering into a place immediately before the T command.

4 When the tool life is counted, the remaining life of a group (the life value minus the life counter value) is compared with the remaining life setting, and the status of the tool life expiration prior notice signal is changed according to the result of the comparison.

- M99

If the life count is specified by use count and bit 0 (T99) of parameter No. 6802 is 1, the tool change signal TLCH<Fn064.0> is output and the automatic operation is stopped if the life of at least one tool group has expired when the M99 command is executed. If the life count type is duration specification, the tool change signal is output immediately when the life of at least one tool group has expired; when the M99 command is specified, the automatic operation is stopped, but no more tool change signal is output.

M

If the life count is specified by use count, a tool group command (T code) issued after the M99 command selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one.

T

If the life count is specified by use count, when a tool group command (T code) is specified after the M99 command is specified, a tool whose life has not expired is selected from a specified group, and the tool life counter is incremented by one.

10.2.5 Tool Life Count Restart M Code

Explanation

M

If the life count is specified by use count, the tool change signal is output if the life of at least one tool group has expired when a tool life count restart M code is issued. A tool group command (T code) issued after the tool life count restart M code selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one. This enables the tool life to be counted by a tool change command (M06) even if the command is not the first tool change command (M06) issued since the entry of the CNC into the automatic operation start state from the reset state. The tool life count restart M code is specified in parameter No. 6811.

Example:

Suppose that M16 is a tool life count restart M code and that the tool life management ignore number is 100.
Also suppose that the life count is specified by use count.

```
T101 ;  A tool whose life has not expired is selected from group 1.
   :
M06 ;   Tool life management is performed for group 1.
   :    (The tool life counter is incremented by one.)
T102 ;  A tool whose life has not expired is selected from group 2.
   :
M06 ;   Tool life management is performed for group 2.
   :    (The tool life counter is incremented by one.)
M16 ;   Tool life counting is restarted.
T101 ;  A tool whose life has not expired is selected from group 1.
   :
M06 ;   Tool life management is performed for group 1.
   :    (The tool life counter is incremented by one.)
```

T

If the life count is specified by use count, the tool change signal is output if the life of at least one tool group has expired when a tool life count restart M code is specified. A tool group command (T code) issued after the tool life count restart M code selects a tool whose life has not expired from a specified group, and the tool life counter is incremented by one. This enables the tool life to be counted by a tool group command (T code) even if the command is not the first tool group command issued since the entry of the CNC into the automatic operation start state from the reset state. The tool life count restart M code is specified in parameter No. 6811.

Example:

Suppose that M16 is a tool life count restart M code.
Also suppose that the life count is specified by use count.

```
T199 ;  A tool whose life has not expired is selected from group 1.
   :    Tool life management is performed for group 1.
   :    (The tool life counter is incremented by one.)
   :
T299 ;  A tool whose life has not expired is selected from group 2.
   :    Tool life management is performed for group 2.
   :    (The tool life counter is incremented by one.)
   :
M16 ;   Tool life counting is restarted.
T199 ;  A tool whose life has not expired is selected from group 1.
   :    Tool life management is performed for group 1.
   :    (The tool life counter is incremented by one.)
```

NOTE

1 The tool life count restart M code is treated as an M code not involved in buffering.
2 If the life count type is use count specification, the tool change signal is output if the life of at least one tool group has expired when the tool life count restart M code is specified. If the life count type is duration specification, specifying the tool life count restart M code causes nothing.

10.2.6 Disabling Life Count

Explanation

If bit 6 (LFI) of parameter No. 6804 is 1, the tool life count disable signal LFCIV can be used to select whether to cancel the tool life count.

If the tool life count disable signal LFCIV is 1, the tool life count disabled signal LFCIF becomes 1, and the tool life count is disabled.
If the tool life count disable signal LFCIV is 0, the tool life count disabled signal LFCIF becomes 0, and the tool life count is enabled.

NOTE

No buffering should occur when the state of the tool life count disable signal LFCIV is changed. So, use, for example, M codes not involved in buffering to change the signal state. If M06 (for the M series) or a tool change T code (for the T series) is issued in a block that follows directly a block in which an auxiliary function code with buffering enabled is used to turn on or off the tool life count disable signal LFCIV, it is likely that a command for specifying whether to count may become incorrect.

11 AUXILIARY FUNCTION

Overview

There are two types of auxiliary functions: the auxiliary function (M codes), which specifies the start and end of the spindle or the end of a program, and the second auxiliary function (B codes), which specifies the positioning or other operation of the indexing table.
When a move command and auxiliary function are specified in the same block, the commands are executed in one of the following two ways:
(1) Simultaneous execution of the move command and auxiliary function commands.
(2) Executing auxiliary function commands upon completion of move command execution.
The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

Chapter 11, "AUXILIARY FUNCTION", consists of the following sections:

11.1 AUXILIARY FUNCTION (M 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. Depending on the setting of bit 7 (M3B) of parameter No. 3404, up to three M codes can be specified.
Which M code corresponds to which machine function is determined by the machine tool builder. The machine processes all operations specified by M codes except those specified by M98, M99,M198 or called subprogram(Parameter No.6071 to 6079), or called custom macro (Parameter No.6080 to 6089). Refer to the machine tool builder's instruction manual for details.

Explanation

The following M codes have special meanings.

- M02,M30 (End of program)

This indicates the end of the main program.
Automatic operation is stopped and the CNC unit is reset. (This differs with the machine tool builder.)
After a block specifying the end of the program is executed, control returns to the start of the program.
Bits 5 (M02) and 4 (M30) of parameter No. 3404 can be used to disable M02, M30 from returning control to the start of the program.

- M00 (Program stop)

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. (This differs with the machine tool builder.)

- M01 (Optional stop)

Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel has been pressed.

- M98 (Calling of subprogram)

This code is used to call a subprogram. The code and strobe signals are not sent. See the subprogram II-13.3 for details.

- M99 (End of subprogram)

This code indicates the end of a subprogram.
M99 execution returns control to the main program. The code and strobe signals are not sent. See the subprogram section II-13.3 for details.

- M198 (Calling of external subprogram)

This code is used to call a subprogram in a file in an external input/output device. For details, refer to Section 4.5 "EXTERNAL SUBPROGRAM CALL (M198)" in Part III.

NOTE

The block next to M00, M01, M02, or M30 is not looked-ahead (not buffered).
Parameters Nos. 3411 to 3420 and Nos. 3421 to 3432 can be used to set the M codes that are not buffered similarly.
For the M codes that are not buffered, refer to the manual provided by the machine tool builder.

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

Usually, only one M code can be specified in one block. By setting bit 7 (M3B) of parameter No. 3404 to 1, however, up to three M codes can be specified simultaneously in one block.
Up to three M codes specified in one block are output to the machine at the same time. So, when compared with a case where a single M code is specified in one block, a reduced machining cycle time can be achieved.

Explanation

CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.
M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.
Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.
Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.
However, it is possible to specify multiple M codes that are sent to the machine in the same block unless they direct the CNC to perform internal operations. (Since the processing method depends on the machine, refer to the manual of the machine tool builder.)

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 ; : : :

11.3 SECOND AUXILIARY FUNCTIONS (B CODES)

Overview

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 are specified by parameter No.3033, an alarm is issued if the number of digits of a command exceeds the specified number.
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 No.3460.
However, the address used for the second auxiliary function cannot also be used as the address of the controlled axis. For details, refer to the manual available from the machine tool builder.

Explanation

- Range of specification

-99999999 to 99999999 (8 digits)

- Output value

The value specified after the address of the second auxiliary function is output on the code signals B00 to B31. Note the following about a output value.

1. When a command with a decimal point or a negative command is disabled
   (When bit 0 (AUP) of parameter No.3450 is set to 0)

   When the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is, regardless of the desktop calculator decimal point setting (bit 0 (DPI) of parameter No.3401).
   Example:

Specified value	Output value
B10	10

   When the second auxiliary function with a decimal point is specified, alarm PS0007 is issued.
   When the second auxiliary function is specified with a negative value, alarm PS0006 is issued.

2. When a command with a decimal point or a negative command is enabled
   (When bit 0 (AUP) of parameter No.3450 is set to 1)

   When the desktop calculator decimal point setting is not specified (when bit 0 (DPI) of parameter No.3401 is set to 0), if the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is.
   Example:

Specified value	Output value
B10	10

   When desktop calculator decimal point input is specified (when bit 0 (DPI) of parameter No.3401 is set to 1), if the second auxiliary function with no decimal point is specified, the specified value multiplied by a magnification is output on the code signals. (Magnifications are shown in Table 11.3 (a).)
   Example:
   Specified value Output value

B10 10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)

When the second auxiliary function with a decimal point is specified, the specified value multiplied by a magnification is output to the code signals. (Magnifications are shown in Table 11.3 (a).)
Example:
Specified value Output value
B10. 10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)
B0.123 1230 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

The magnification is determined as shown below according to the setting unit of the reference axis (specified by parameter No.1031) and bit 0 (AUX) of parameter No.3405.

Table 11.3 (a) Magnifications for an output value when the second auxiliary function with a decimal point is specified for desktop calculator decimal point input

Setting unit		Parameter AUX = 0	Parameter AUX = 1
Metric input system	Reference axis: IS-A	100×	100×
	Reference axis: IS-B	1000×	1000×
	Reference axis: IS-C	10000×	10000×
Inch input system	Reference axis: IS-A	100×	1000×
	Reference axis: IS-B	1000×	10000×
	Reference axis: IS-C	10000×	100000×

⚠ **CAUTION**

If a decimal fraction remains after multiplying the specified value with a decimal point by a magnitude in Table 11.3 (a), the fraction is truncated.
Example:
Specified value Output value
B0.12345 1234 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

NOTE

If the number of digits of the specified value exceeds the allowable number of digits (set by parameter No.3033), alarm PS0003 is issued.
When the specified value is multiplied by a magnitude in Table 11.3 (a), the allowable number of digits must be set for the resultant value.

Limitation

Addresses used for the second auxiliary functions (addresses specified with B or parameter No. 3460) cannot be used as the addresses used for controlled axis names.

12 PROGRAM MANAGEMENT

Chapter 12, "PROGRAM MANAGEMENT", consists of the following sections:

12.1 PROGRAM ATTRIBUTES

The following attributes can be set for programs:

- Change protection level/output protection level

- Change protection level/output protection level

With the 8-level data protection function, change and output protection can be provided for a specified program.
For details of the 8-level data protection function, see the description of the “Protection of data at eight levels” function.

12.2 RELATED PARAMETERS

This subsection lists the meanings of parameters related to program numbers and the folders and programs to be manipulated or executed.

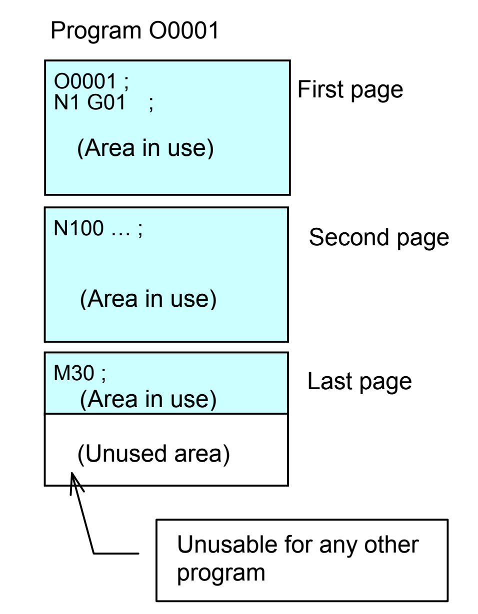
Parameter No.	Bit No.	Description
3202	0 (NE8)	Disables or enables editing of programs O8000 to O8999.
	4 (NE9)	Disables or enables editing of programs O9000 to O9999.
3210/3211	-	Password/keyword for protecting programs in the nine thousands
3404	2 (SBP)	In the subprogram call function, address P in the M198 block specifies a file/program number.
6001	5 (TCS)	Calls or does not call a custom macro by T code.
6050～6059	-	G code for calling a custom macro with program No. 9010 to 9019
6071～6079	-	M code for calling a subprogram with program No. 9001 to 9009
6080～6089	-	M code for calling a custom macro with program No. 9020 to 9029
6090/6091	-	ASCII code for calling a subprogram with program No. 9004/9005
8341/8343	-	Target program number and sequence number for sequence number comparison and stop

12.3 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	○2	○2	–	–
512Kbyte	400	○1	○1	○	○
1Mbyte	800	–	*2	–	–
2Mbyte	400	☆	–	–	–

○: Standard package (1/2) *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.

13 PROGRAM CONFIGURATION

Overview

- Main program and subprogram

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

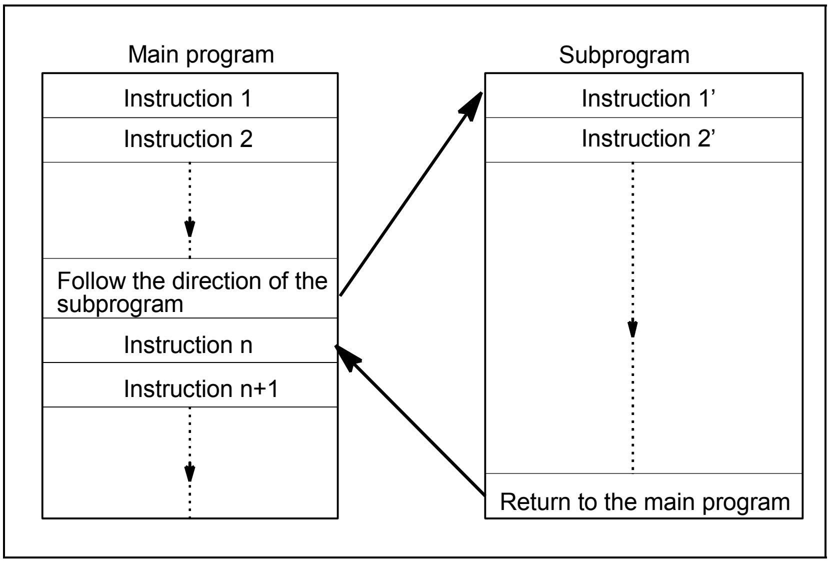


Fig. 13 (a) Main program and Subprogram

The CNC memory can hold 400 main programs and subprograms (800 main programs and subprograms for T series 2-path systems). A main program can be selected from the stored main programs to operate the machine. See III-9 or III-10.4 for the methods of registering and selecting programs.

- Program components

A program consists of the following components:

Table 13 (a) Program components

Components	Descriptions
Program code start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Program code end	Symbol indicating the end of a program file

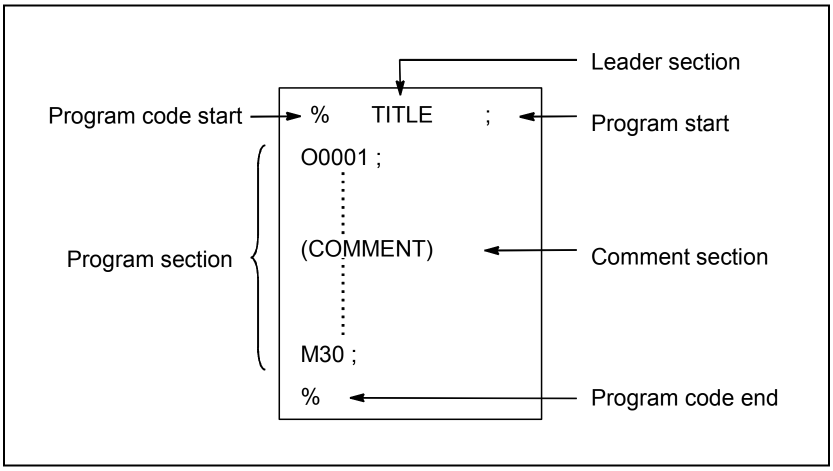


Fig. 13 (b) Program configuration

- Program section configuration

A program section consists of several blocks. A program section starts with a program number and ends with a program end code.

Program section configuration	**Program section**
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z-32.0 H01 ;
:	:
Block n	Nn Z0 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a slash (/) at the start of a block disables the execution of some blocks (see "optional block skip" in II-13.2).

13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See II-13.2 for a program section.

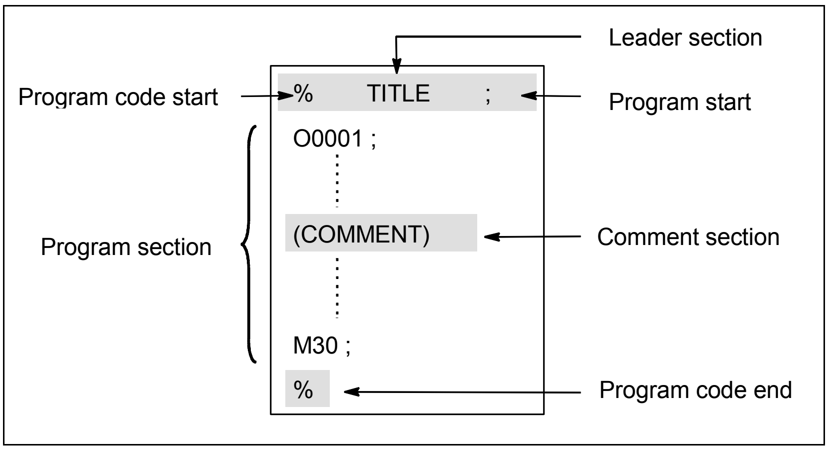


Fig. 13.1 (a) Program configuration

Explanation

- Program code start

The program code start indicates the start of a file that contains NC programs.
The mark is not required when programs are entered using ordinary personal computers. The mark is not displayed on the screen. However, if the file is output, the mark is automatically output at the start of the file.

Table 13.1 (a) Code of a program code start

Name	ISO code	EIA code	Notation in this manual
Program code start	%	ER	%

- Leader section

Data entered before the programs in a file constitutes a leader section.
When machining is started, the label skip state is usually set by turning on the power or resetting the system. In the label skip state, all information is ignored until the first end-of-block code is read. When a file is read into the CNC unit from an I/O device, leader sections are skipped by the label skip function.
A leader section generally contains information such as a file header. When skipping the leader section, it is possible to enter any code other than EOB because a TV parity check is not performed.

- Program start

The program start code is to be entered immediately after a leader section, that is, immediately before a program section.
This code indicates the start of a program, and is always required to disable the label skip function.
With ordinary personal computers, this code can be entered by pressing the return key.

Table 13.1 (b) Code of a program start

Name	ISO code	EIA code	Notation in this manual
Program start	LF	CR	;

NOTE
If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number.

- Comment section

Any information enclosed by the control-out and control-in codes is regarded as a comment.
The user can enter a header, comments, directions to the operator, etc. in a comment section.

Table 13.1 (c) Codes of a control-in and a control-out

Name	ISO code	EIA code	Notation in this manual	Meaning
Control-out	(	2-4-5	(	Start of comment section
Control-in	)	2-4-7	)	End of comment section

When a program is read into memory for memory operation, comment sections, if any, are not ignored but are also read into memory. Note, however, that codes other than those listed in the code table in Appendix A are ignored, and thus are not read into memory.
When data in memory is output on external I/O device (See III-8), the comment sections are also output.
When a program is displayed on the screen, its comment sections are also displayed. However, those codes that were ignored when read into memory are not output or displayed.
During memory operation or DNC operation, all comment sections are ignored.
The TV check function can be used for a comment section by setting bit 1 (CTV) of parameter No. 0100.

> ⚠ **CAUTION**
> If a long comment section appears in the middle of a program section, a move along an axis may be suspended for a long time because of such a comment section. So a comment section should be placed where movement suspension may occur or no movement is involved.

> **NOTE**
> 1 If only a control-in code is read with no matching control-out code, the read control-in code is ignored.
> 2 The following codes cannot be used in the comment section:
> - EOB
> - % (ER for EIA)

- Program code end

A program code end is to be placed at the end of a file containing NC programs.
If programs are entered using the automatic programming system, the mark need not be entered.
The mark is not displayed on the screen. However, when a file is output, the mark is automatically output at the end of the file.
If an attempt is made to execute % when M02 or M30 is not placed at the end of the program, the alarm PS5010 is occurred.

Table 13.1 (d) Code of a program code end

Name	ISO code	EIA code	Notation in this manual
Program code end	%	ER	%

13.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See II-13.1 for program components other than program sections.

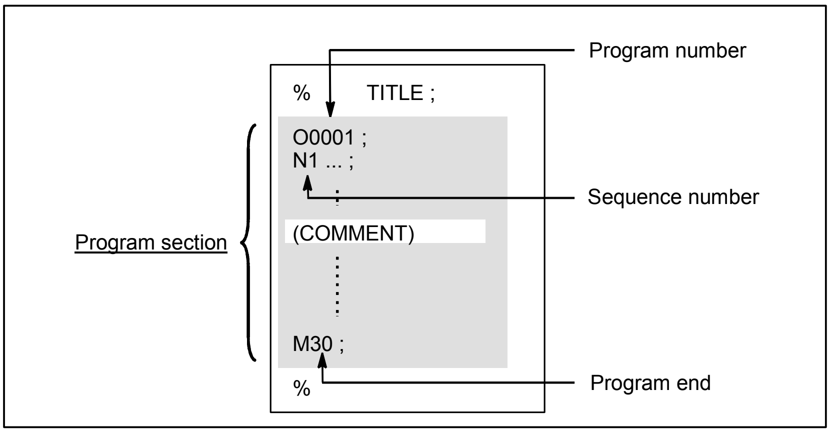


Fig. 13.2 (a) Program configuration

- Program number

A program number consisting of address O followed by a four-digit number is assigned to each program at the beginning registered in memory to identify the program. When the 8-digit number function is selected, the program number consists of eight digits.
In ISO code, the colon (:) can be used instead of O.
When no program number is specified at the start of a program, the sequence number (N....) at the start of the program is regarded as its program number.

If a five-digit sequence number is used, the lower four digits are registered as a program number. If the lower four digits are all 0, the program number registered immediately before added to 1 is registered as a program number. Note, however, that N0 cannot be used for a program number.
If there is no program number or sequence number at the start of a program, a program number must be specified using the MDI panel when the program is stored in memory (See III-8.2 or III-9.1)

> **NOTE**
> Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

- Sequence number and block

A program consists of several commands. One command unit is called a block. One block is separated from another with an EOB of end of block code.

Table 13.2 (a) EOB code

Name	ISO code	EIA code	Notation in this manual
End of block (EOB)	LF	CR	;

At the head of a block, a sequence number consisting of address N followed by a number not longer than five digits (1 to 99999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

```
N300 X200.0 Z300.0 ;    A sequence number is underlined.
```

Fig. 13.2 (b) Sequence number and block (example)

> **NOTE**
> N0 must not be used for the reason of file compatibility with other CNC systems.
> Program number 0 cannot be used. So 0 must not be used for a sequence number regarded as a program number.

- TV check (Vertical parity check)

A parity check is made for each block of input data. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, a P/S alarm (No.002) is output.
No TV check is made only for those parts that are skipped by the label skip function. Bit 1 (CTV) of parameter No. 0100 is used to specify whether comments enclosed in parentheses are counted as characters during TV check. The TV check function can be enabled or disabled by setting on the MDI unit (See III-12.3.1.).

- Block configuration (word and address)

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (-) may be prefixed to a number.)
For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address.

Word = Address + number (Example : X-1000)

Table 13.2 (b) indicates the usable addresses and their meanings.
The same address may have different meanings, depending on the preparatory function specification.

Table 13.2 (b) Major functions and addresses

Function	Address	Meaning
Program number	O(*)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Y, Z, U, V, W, A, B, C	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
	B	Table indexing, etc.
Program number designation	P	Subprogram number
Number of repetitions	P, L	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

M

Offset number	D, H	Offset number
Dwell	P, X	Dwell time

T

Dwell	P, X, U	Dwell time

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

N_	G_	X_ Y_	F_	S_	T_	M_	;
Sequence number	Preparatory function	Dimension word	Feed-function	Spindle speed function	Tool function	Auxiliary function	

Fig. 13.2 (c) 1 block (example)

- Major addresses and ranges of command values

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100 m (in millimeter input) along the X axis. However, an actual stroke along the X axis may be limited to 2 m for a specific machine tool.
Similarly, the CNC may be able to control a cutting feedrate of up to 240 m/min, but the machine tool may not allow more than 3 m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

Table 13.2 (c) 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

Function		Address	Input in mm	Input in inch
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.

- Optional block skip

When a slash followed by a number (/n (n=1 to 9)) is specified at the head of a block, and optional block skip signals BDT1 to BDT9 are set to 1 during automatic operation, the information (/n to the end of the block (EOB)) contained in the block for which /n corresponding to signal BDTn is specified is ignored.

Example 1)

/2 N123 X100.0 Y200.0 ;

Example 2)

//3 N123 X100.0 Y200.0 ; → Incorrect

/1 /3 N123 X100.0 Y200.0 ; → Correct

Input signal and program code

Input signal	Start code to be ignored
BDT1	/ or /1(NOTE)
BDT2	/2

Input signal	Start code to be ignored
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

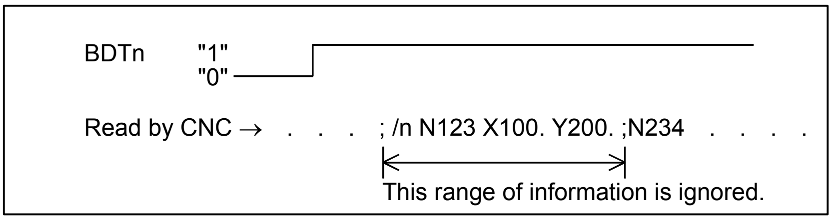
NOTE

1 Number 1 for /1 can be omitted. However, when two or more optional block skips are specified for one block, number 1 for /1 cannot be omitted.

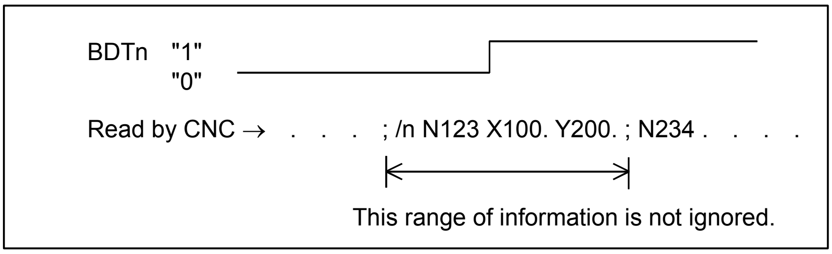
2 Depending on the machine tool, all optional block skip signals (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which switches are usable.

The following shows the relationship between the timing at which optional block skip signals BDT1 to BDT9 are set to 1 and the range of information to be ignored.

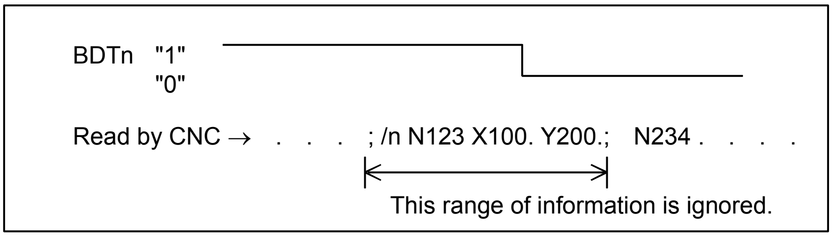
1. When the signal BDTn is set to 1 before the CNC starts reading a block that contains /n, the block is ignored.



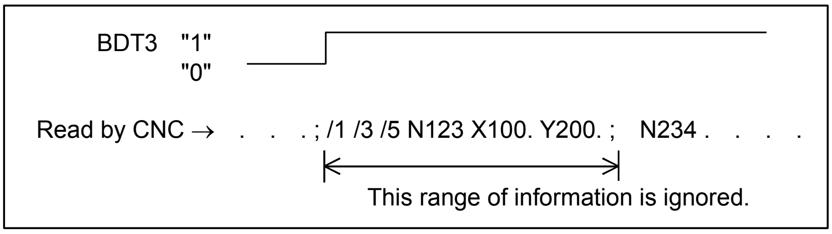
2. When the signal BDTn is set to 1 while the CNC is reading a block that contains /n, the block is not ignored.



3. When the signal BDTn is set to 0 while the CNC is reading a block that contains /n, the block is ignored.



4. Two or more optional block skips can be specified in one block. When the signal corresponding to any of the specified skips is set to 1, the block is ignored.



> **NOTE**
> 1 This function is not used when a program is registered in memory. A block containing / is registered in memory regardless of the statuses of optional block skip signals. When a program in memory is also output regardless of the statuses of optional block skip signals.
> In addition, the optional block skip function is enabled during a search for a sequence number.
> 2 Position of a slash
> A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.
> 3 TV and TH check
> When the optional block skip signal is set to 1, TH and TV checks are made for the skipped portions in the same way as when the optional block skip signal is set to 0.

- Program end

The end of a program is indicated by programming one of the following codes at the end of the program:

Table 13.2 (d) Code of a program end

Code	Meaning usage
M02	For main program
M30	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

> ⚠ **CAUTION**
> A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program. (see "Optional block skip".)

13.3 SUBPROGRAM (M98, M99)

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

- Subprogram configuration

One subprogram

Oxxxx ;	Subprogram number (or the colon (:) optionally in the case of ISO)
:	
M99;	Program end

M99 need not constitute a separate block as indicated below.

Example) X100.0 Y100.0 M99 ;

- Subprogram call

M98 Pxxxx xxxx ;

Subprogram number (the last four digits)

Number of times the subprogram is called repeatedly (the first four digits)

or

M98 Pxxxx Lxxxxxxxx ;

Number of times the subprogram is called repeatedly (Lxxxxxxxx)

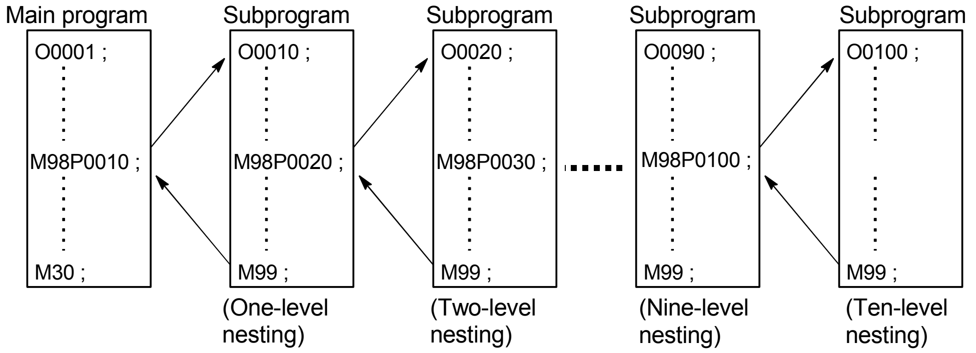
Subprogram number (Pxxxx)

NOTE

1. When a subprogram is call repeatedly (P8-digit number), the number of digits of the subprogram number is less than 4, pad the upper digit(s) with 0.
   Example)
   P100100: Call subprogram No. 100 ten times.
   P50001: Call subprogram No. 1 five times.
2. When the repeat count is omitted, a repeat count of 1 is assumed.
   In this case, it is not necessary to adjust the subprogram number length to 4 digits as described in Item 1 above.
3. When a subprogram is call repeatedly (P8-digit number), do not specify address L in the same block.

Explanation

When the main program calls a subprogram, it is regarded as a one-level subprogram call. Thus, subprogram calls can be nested up to ten levels as shown below.



A single call command can repeatedly call a subprogram up to 99999999 times. For compatibility with automatic programming systems, in the first block, Nxxxxx can be used instead of a subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

> **NOTE**
> 1 The M98 and M99 code signal and strobe signal are not output to the machine tool.
> 2 If the subprogram number specified by address P cannot be found, an alarm PS0078 is output.

Example

- M98 P51002 ;

  This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

- X1000.0 M98 P1200 ;

  This example calls the subprogram (number 1200) after an X axis movement.

- Execution sequence of subprograms called from a main program

Main program	1 2 3	Subprogram
N0010 . . . ;		O1010 . . . ;
N0020 . . . ;		N1020 . . . ;
N0030 M98 P21010 ;		N1030 . . . ;
N0040 . . . ;		N1040 . . . ;
N0050 M98 P1010 ;		N1050 . . . ;
N0060 . . . ;		N1060 . . . M99 ;

A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Special usage

- Specifying the sequence number for the return destination in the main program

If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. When P0 is specified, however, P is ignored. In addition, when the main program is running in a mode other than memory operation mode, P is ignored.

This method consumes a much longer time than the normal return method to return to the main program.

Main program	Subprogram
N0010 . . . ;	O1010 . . . ;
N0020 . . . ;	N1020 . . . ;
N0030 M98 P1010 ;	N1030 . . . ;
N0040 . . . ;	N1040 . . . ;
N0050 . . . ;	N1050 . . . ;
N0060 . . . ;	N1060 . . . M99 P0060 ;

- Using M99 in the main program

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.

Execution is repeated while the optional block skip function is set to off.

If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.
If/M99Pn ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.

```
                              N0010 . . . ;
                              N0020 . . . ;
                              N0030 . . . ;
Optional block skip           N0040 . . . ;
OFF                           N0050 . . . ;
                            / N0060 . . . M99 P0030 ;   Optional block skip
                              N0070 . . . ;             ON
                              N0080 M02 ;
```

- Using a subprogram only

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.
(See III-10.4 for information about search operation.)
In this case, if a block containing M99 is executed, control returns to the start of the subprogram for repeated execution. If a block containing M99Pn is executed, control returns to the block with sequence number n in the subprogram for repeated execution. To terminate this program, a block containing /M02 ; or /M30 ; must be placed at an appropriate location, and the optional block switch must be set to off ; this switch is to be set to on first.

```
  N1010 . . . ;
  N1020 . . . ;
  N1030 . . . ;
/ N1040 . . . M02 ;          Optional block skip ON
  N1050 M99 P1020 ;
```

- Subprogram call with sequence number

Setting bit 0 (SQC) of parameter No. 6005 to 1 can call a specified sequence number in the subprogram for execution.
In a subprogram call command, specify the letter Q followed by a sequence number to be called after the letter P for specifying a program number.

```
M98 Pxxxx Qxxxxx ;
    ↑     ↑
    |     └──── Sequence number
    └────────── Program number
```

This command causes program execution to start at the called sequence number in the subprogram. If a repetition count is specified, program execution is repeated from the specified sequence number.

Main program	**Sub program**
N0010···;	**O1010···;**
N0020···;	**N1020···;**
N0030 M98 P1010 Q1030 ;	**N1030···;**
N0040···;	**N1040···;**
N0050···;	**N1050···;**
N0060···;	**N1060···M99 ;**

This function enables a sequence number in the same program to be called for execution as shown below. This method, however, requests the programmer to be aware of an allowable call nesting level. If an attempt is made to exceed the allowable nesting level, alarm PS0077 meaning “TOO MANY SUB,MACRO NESTING” is issued.

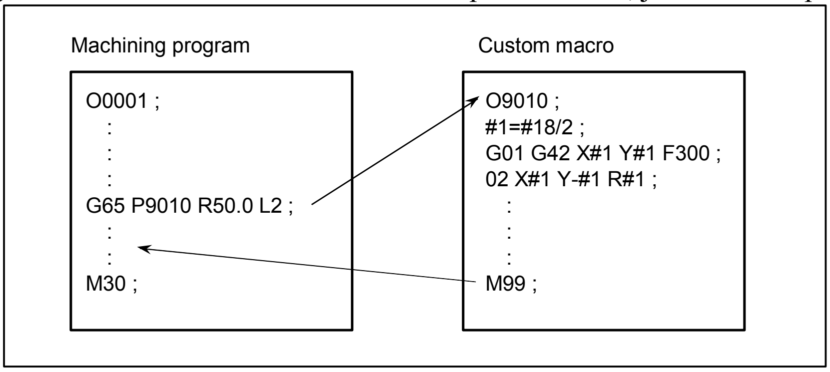
```
O0001 ;
N0010…;
N0020 M98 (P0001) Q0050 ;
N0030…;
N0040…;
N0050…;
N0060…;
N0070…M99;
```

For a call within the same program, specification of Pxxxx in a block can be omitted when the block includes M98.

This function is usable only for subprogram calls by M98; it is unusable for non-M98 calls, such as macro calls or external subprogram calls based on M198.

14 CUSTOM MACRO

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user-defined canned cycles.
A machining program can call a custom macro with a simple command, just like a subprogram.



14.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 and X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;
G01 X#1 F300 ;
```

Explanation

- Variable representation

When specifying a variable, specify a number sign (#) followed by a variable number.

#i (i = 1, 2, 3, 4, ...)

[Example] #5
#109
#1005

A variable can also be represented as follows using <expression> described in the section about arithmetic and logic operation commands.

#[<expression>]

[Example] #[#100]
#[#1001-1]
#[#6/2]

Variable #i shown in the following can be replaced with a variable of #[<expression>].

- Types of variables

Variables can be classified as local variables, common variables, and system variables according to the variable number. Each of those variables has its own usage and characteristics. Read-only system constants are also provided.

- Range of variable values
Local and common variables can have a value in the following ranges. If the result of calculation exceeds the range, an alarm PS0111 is issued.
When bit 0 (F0C) of parameter No.6008 = 0

Maximum value: approx. $\pm 10^{308}$
Minimum value: approx. $\pm 10^{-308}$
Numeric data handled by a custom macro conforms to the IEEE standard and is handled as a double-precision real number. An error resulting from operation depends on the precision.

When bit 0 (F0C) of parameter No.6008 = 1

Maximum value: approx. $\pm 10^{47}$
Minimum value: approx. $\pm 10^{-29}$

- Local variable (#1-#33)
A local variable is a variable that is used locally in a macro. That is, local variable #i used by a macro called at a certain time is different from that used by a macro called at another time, regardless of whether the two macros are the same. Therefore, for example, when macro A calls macro B during multiple calls or the like, it is impossible for macro B to corrupt a local variable used by macro A by erroneously using the variable.
A local variable is used to pass arguments. For information on correspondence between arguments and addresses, see the section about macro calling commands. The initial state of a local variable to which no arguments are passed is <null> and the user can freely use the variable. The attribute of a local variable is READ/WRITE enabled.

- Common variable (#100-#199, #500-#999)
A common variable is shared among the main program, subprograms called by the main program, and macros while a local variable is used locally in a macro. That is, #i used by a macro is the same as that used by another macro. Therefore, a resultant common variable obtained by using a macro can be used by another macro. The attribute of a common variable is basically READ/WRITE enabled. However, the common variable can be protected (its attribute is set to READ only) by specifying its variable number using parameters No.6031 and No.6032. A common variable can be freely used by the user even when its usage is not defined by the system. A total of 600 common variables (#100 to #199 and #500 to #999) can be used. Common variables #100 to #199 are cleared during power-off, but common variables #500 to #999 are not cleared during power-off.

- Write protection of a common variable
Multiple common variables (#500 to #999) can be protected (their attributes are set to READ only) by setting variable numbers in parameters No.6031 and No.6032. This protection is enabled for both Input/All Clear by MDI on the macro screen and write operation by a macro program. If the NC program specifies WRITE operation (used in the left side) for a common program in the set range, an alarm PS0116 is issued.

- System variable
A variable whose usage does not vary in the system. The attribute of a system variable is READ only, WRITE only, or READ/WRITE enabled depending on the nature of a system variable.

- System constant
A system constant can be referenced as with a variable even though its value is fixed. The attribute of a system constant is READ only.

- Omission of the decimal point
When a variable value is defined in a program, the decimal point can be omitted.

[Example]

When #1 = 123; is defined, the actual value of variable #1 is 123.000.

- Referencing variables

The value following an address can be replaced with a variable. When programming as <address>#i or <address>-#i, the variable value or the complement of it is used as the specified value of the address.

[Example] F#33 is the same as F1.5 when #33 = 1.5.
Z-#18 is the same as Z-20.0 when #18 = 20.0.
G#130 is the same as G3 when #130 = 3.0.

A variable cannot be referenced using address/, :, or O and N.

[Example] Programming such as O#27, N#1, or N[#1] is not allowed.
n (n = 1 to 9) in the optional block skip /n cannot be a variable.

A variable number cannot be specified by a direct variable.

[Example] When replacing 5 in #5 with #30, specify #[#30] instead of ##30. No values exceeding the maximum allowable value for each address can be specified.

[Example] When #140 = 120, G#140 exceeds the maximum allowable value.
When a variable is used as address data, the variable is automatically rounded off to the number of significant figures of each address or less.

[Example] For a machine with an increment system of 1/1000 mm (IS-B), when #1 = 12.3456, G00 X#1; becomes G00 X12.346;.

If <expression>, described later, is used, the value following an address can be replaced with <expression>.

<address>[<expression>] or <address>-[<expression>]

The program code shown above indicates the value of <expression> or the complement of the value is used as an address value. Note that a constant with no decimal point, enclosed in brackets ([ ]), is assumed to have a decimal point at the end.

[Example] X[#24+#18*COS[#1]]
Z-[#18+#26]

- Undefined variable

When the value of a variable is not defined, such a variable is referred to as a "null" variable. Variables #0 and #3100 are always null variables. They cannot be written to, but they can be read.

(a) Quotation
When an undefined variable is quotated, the address itself is also ignored.

Original command	G90 X100 Y#1
Equivalent command when #1 = <null>	G90 X100
Equivalent command when #1 = 0	G90 X100 Y0

(b) Definition/replacement, addition, multiplication
When a local variable or common variable is directly replaced with <null>, the result is <null>. When a system variable is directly replaced with <null> or the result of calculation including <null> is replaced, a variable value of 0 is assumed.

Original expression (local variable)	#2=#1	#2=#1*5	#2=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (common variable)	#100=#1	#100=#1*5	#100=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (system variable)	#2001=#1	#2001=#1*5	#2001=#1+#1
Replacement result (when #1 = <null>)	0	0	0
Replacement result (when #1 = 0)	0	0	0

(c) Comparison
<null> differs from 0 only for EQ and NE.
<null> is equal to 0 for GE, GT, LE, and LT.

- When <null> is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Established (true)	Established (true)	Established (true)	Not established (false)	Established (true)	Not established (false)

- When 0 is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Not established (false)	Not established (false)	Established (true)	Not established (false)	Established (true)	Not established (false)

- Specifying a system variable (constant) by its name

A system variable (constant) is specified by its variable number, but it can also be specified by its predetermined system variable (constant) name. A system variable (constant) name begins with an underscore (_), followed by up to seven uppercase letters, numerics, or underscores. For axis-dependent variables (such as coordinates) or variables having a lot of data of similar types (such as tool compensation), subscript [n] (n: integer) can be used to specify values. In this case, n can be specified in <expression> format (calculation format).
The command format must be specified in [#system-variable-name] format, as shown below.

[#_DATE]

[Example]

```
#101= [#_DATE] ;             : #3011 (year/month/date) is read off and assigned to #101.
#102= [#_TIME] ;             : #3012 (hour/minute/second) is read off and assigned to #102.
#103=[#_ABSMT[1]] ;          : #5021 (machine coordinate value of the 1st axis) is read off and assigned to #103.
#104=[#_ABSKP[#500*2]] ;     : #506x (skip position of [#500*2]th axis) is read off and assigned to #104.
```

If a value other than an integer is specified for subscript n, a variable value is referenced, assuming that the fractional portion is rounded off.

[Example]

```
[#_ABSIO[1.4999999]]   : This value is assumed to be [#_ABSIO[1]], that is, #5001.
[#_ABSIO[1.5000000]]   : This value is assumed to be [#_ABSIO[2]], that is, #5002.
```

NOTE
1 When the specified variable name is not registered, an alarm PS1098 is issued.
2 When a negative or other invalid subscript is specified, an alarm PS1099 is issued.

- System constant #0, #3100-#3102 (Attribute: R)

Constants used as fixed values in the system can be used as system variables. Such constants are called system constants. The system constants provided are shown below.

Constant number	Constant name	Description
#0, #3100	[#_EMPTY]	Null
#3101	[#_PI]	Circular constant π= 3.14159265358979323846
#3102	[#_E]	Base of natural logarithm e= 2.71828182845904523536

- Specifying a common variable by its name

Specifying a variable name set by the SETVN command described later allows reading from or writing to a common variable.
The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example]

```
X[#POS1] Y[#POS2] ;           : Specifying a position by the variable name
[#POS1] = #100+#101 ;         : Executing a assignment statement by the variable name
#[100+[#ABS]] = 500 ;         : Same as above (by a variable number)
#500 = [1000+[#POS2]*10] ;    : Reading a variable by a variable name
```

- Setting and specifying the name of a common variable (SETVN)

For the 50 common variables, #500 to #549, a name of up to eight characters can be specified by using a command as shown below.

SETVN n [VAR500, VAR501, VAR502,......] ;

n represents the starting number of a common variable for which the name is specified.
VAR500 is the variable name of variable n, VAR501 is the variable name of variable n+1, and VAR502 is the variable name of variable number n+2, and so on. Each string is delimited by a comma (,). All codes that can be used as meaningful information in a program except control in, control out, [,], EOB, EOR, and : (colon in a program number) can be used. However, each name must begin with an alphabetical character. Variable names are not cleared on switch-off.
Specifying a set variable name allows reading from or writing to the common variable. The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example] SETVN 510[TOOL_NO, WORK_NO, COUNTER1, COUNTER2];

The command above names the variables as follows.

Variable	Name
#510	#TOOL_NO
#511	#WORK_NO
#512	#COUNTER1
#513	#COUNTER2

The names specified by the command can be used in a program. For example, when 10 is assigned to #510, the expression [#TOOL_NO]=10; can be used instead #510=10;.

NOTE

If the same name was specified for different common variables, only the variable which has the smaller variable number can be referenced with the specified name.

14.2 SYSTEM VARIABLES

System variables can be used to read and write internal CNC data such as tool compensation values and current position data. System variables are essential for automation and general-purpose program development.

List of system variables and constants

n represents a subscript.
R, W, and R/W are attributes of a variable and indicate read-only, write-only, and read/write enabled, respectively.

- Interface signals

System variable number	System variable name	Attribute	Description
#1000-#1031	[#_UI[n]]	R	Interface input signals (BIT), UI000-UI031 NOTE) Subscript n represents a BIT position (0-31).
#1032-#1035	[#_UIL[n]]	R	Interface input signals (LONG), UI000-UI031/ UI100-UI131/ UI200-UI231/UI300-UI331 NOTE) Subscript n (0-3): 0 = UI000-UI031, 1 = UI100-UI131, 2 = UI200-231, 3 = UI300-UI331
#1100-#1131	[#_UO[n]]	R/W	Interface output signals (BIT), UO000-UO031 NOTE) Subscript n represents a BIT position (0-31).
#1132-#1135	[#_UOL[n]]	R/W	Interface output signals (LONG), UO000-UO031/ UO100-UO131/UO200-UO231/UO300-UO331 NOTE) Subscript n (0-3): 0 = UO000-UO031, 1 = UO100-UO131, 2 = UO200-231, 3 = UO300-UO331

- Tool compensation value

M

For tool compensation memory A (bit 6 (NGW) of parameter No. 8136 is 1)

System variable number	System variable name	Attribute	Description
#2001-#2200 #10001-#10400	[#_OFS[n]]	R/W	Tool compensation value Note)Subscript n represents a compensation number (1 to 200). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 400).

For tool compensation memory C (bit 6 (NGW) of parameter No. 8136 is 0)
when bit 3 (V10) of parameter No. 6000 is 0

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note)Subscript n represents a compensation number (1 to 200).
#10001-#10400			The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 400).
#2201-#2400	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note)Subscript n represents a compensation number (1 to 200).
#11001-#11400			The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 400).
#12001-#12400	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear) Note)Subscript n represents a compensation number (1 to 400).
#13001-#13400	[#_OFSDG[n]]	R/W	Tool compensation value (D code, geometry) Note)Subscript n represents a compensation number (1 to 400).

For tool compensation memory C (bit 6 (NGW) of parameter No. 8136 is 0)
when bit 3 (V10) of parameter No. 6000 is 1

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note)Subscript n represents a compensation number (1 to 200).
#10001-#10400			The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 400).

System variable number	System variable name	Attribute	Description
#2201-#2400	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note)Subscript n represents a compensation number (1 to 200).
#11001-#11400			The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 400).
#2401-#2600	[#_OFSDG[n]]	R/W	Tool compensation value (D code, geometry)(Note 1) Subscript n represents a compensation number (1 to 200). Note 1) Enabled when bit 5 (D10) of parameter No.6004 = 1.
#12001-#12400			The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 400).
#2601-#2800	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear)(Note 1) Subscript n represents a compensation number (1 to 200). Note 1) Enabled when bit 5 (D10) of parameter No.6004 = 1.
#13001-#13400			The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 400).

- Tool compensation value

T

Without tool geometry/wear compensation memory (bit 6 (NGW) of parameter No. 8136 is 1)

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10200	[#_OFSX[n]]	R/W	X-axis compensation value (*1) Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2101-#2164 #11001-#11200	[#_OFSZ[n]]	R/W	Z-axis compensation value (*1) Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2201-#2264 #12001-#12200	[#_OFSR[n]]	R/W	Tool nose radius compensation value Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2301-#2364 #13001-#13200	[#_OFST[n]]	R/W	Virtual tool tip T position Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2401-#2449 #14001-#14200	[#_OFSY[n]]	R/W	Y-axis compensation value (*1) Note)Subscript n represents a compensation number（1 to 49） The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

With tool geometry/wear compensation memory (bit 6 (NGW) of parameter No. 8136 is 0)

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10200	[#_OFSXW[n]]	R/W	X-axis compensation value（wear）(※1) Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).

System variable number	System variable name	Attribute	Description
#2101-#2164 #11001-#11200	[#_OFSZW[n]]	R/W	Z-axis compensation value（wear）(※1) Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2201-#2264 #12001-#12200	[#_OFSRW[n]]	R/W	Tool nose radius compensation value（wear） Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2301-#2364 #13001-#13200	[#_OFST[n]]	R/W	Virtual tool tip T position Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2401-#2449 #14001-#14200	[#_OFSYW[n]]	R/W	Y-axis compensation value（wear）(※1) Note)Subscript n represents a compensation number（1 to 49） The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2451-#2499 #19001-#19200	[#_OFSYG[n]]	R/W	Y-axis compensation value（geometry）(※1) Note)Subscript n represents a compensation number（1 to 49） The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2701-#2749 #15001-#15200	[#_OFSXG[n]]	R/W	X-axis compensation value（geometry）(※1) Note)Subscript n represents a compensation number（1 to 49） The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2801-#2849 #16001-#16200	[#_OFSZG[n]]	R/W	Z-axis compensation value（geometry）(※1) Note)Subscript n represents a compensation number（1 to 49） The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).
#2901-#2964 #17001-#17200	[#_OFSRG[n]]	R/W	Tool nose radius compensation value（geometry） Note)Subscript n represents a compensation number (1 to 64). The numbers on the left are also allowed. Note)Subscript n represents a compensation number (1 to 200).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Workpiece coordinate system shift amount

T

System variable number	System variable name	Attribute	Description
#2501	[#_WKSFTX]	R/W	X-axis workpiece shift amount
#2601	[#_WKSFTZ]	R/W	Z-axis workpiece shift amount

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes

- Automatic operation or the like

System variable number	System variable name	Attribute	Description
#3000	[#_ALM]	W	Macro alarm
#3001	[#_CLOCK1]	R/W	Clock 1 (ms)
#3002	[#_CLOCK2]	R/W	Clock 2 (hr)

System variable number	System variable name	Attribute	Description
#3003	[#_CNTL1]	R/W	Enable or disable the suppression of single block stop. Enable or disable the waiting of the auxiliary function completion signal.
#3003 bit0	[#_M_SBK]	R/W	Enable or disable the suppression of single block stop.
#3003 bit1	[#_M_FIN]	R/W	Enable or disable waiting for the auxiliary function completion signal.
#3004	[#_CNTL2]	R/W	Enable or disable feed hold. Enable or disable feedrate override. Enable or disable exact stop check.
#3004 bit0	[#_M_FHD]	R/W	Enable or disable feed hold.
#3004 bit1	[#_M_OV]	R/W	Enable or disable feedrate override.
#3004 bit2	[#_M_EST]	R/W	Enable or disable exact stop check.
#3005	[#_SETDT]	R/W	Read/write setting data.
#3006	[#_MSGSTP]	W	Stop with a message.
#3007	[#_MRIMG]	R	Status of a mirror image (DI and setting)
#3008	[#_PRSTR]	R	Restarting/not restarting a program

- Time

System variable number	System variable name	Attribute	Description
#3011	[#_DATE]	R	Year/Month/Date
#3012	[#_TIME]	R	Hour/Minute/Second

- Number of parts

System variable number	System variable name	Attribute	Description
#3901	[#_PRTSA]	R/W	Total number of parts
#3902	[#_PRTSN]	R/W	Number of required parts

- Tool compensation memory

M

System variable number	System variable name	Attribute	Description
#3980	[#_OFSMEM]	R	Tool compensation memory information

- Main program number

System variable number	System variable name	Attribute	Description
#4000	[#_MAINO]	R	Main program number

- Modal information

M

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note)Subscript n represents a G code group number.
#4102	[#_BUFB]	R	Modal information on blocks that have been specified by last minute (B code)

System variable number	System variable name	Attribute	Description
#4107	[#_BUFD]	R	Modal information on blocks that have been specified by last minute (D code)
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4111	[#_BUFH]	R	Modal information on blocks that have been specified by last minute (H code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4130	[#_BUFWZP]	R	Modal information on blocks that have been specified by last minute (additional workpiece coordinate system number)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note)Subscript n represents a G code group number.
#4302	[#_ACTB]	R	Modal information on the block currently being executed (B code)
#4307	[#_ACTD]	R	Modal information on the block currently being executed (D code)
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4311	[#_ACTH]	R	Modal information on the block currently being executed (H code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4330	[#_ACTWZP]	R	Modal information on the block currently being executed (additional workpiece coordinate system number)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note)Subscript n represents a G code group number.
#4502	[#_INTB]	R	Modal information on interrupted blocks (B code)
#4507	[#_INTD]	R	Modal information on interrupted blocks (D code)
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4511	[#_INTH]	R	Modal information on interrupted blocks (H code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)
#4530	[#_INTWZP]	R	Modal information on interrupted blocks (additional workpiece coordinate system number)

T

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note)Subscript n represents a G code group number.
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note)Subscript n represents a G code group number.
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note)Subscript n represents a G code group number.
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)

- Position information

System variable number	System variable name	Attribute	Description
#5001-#5005	[#_ABSIO[n]]	R	End point position of the previous block (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 5)
#5021-#5025	[#_ABSMT[n]]	R	Specified current position (machine coordinate system) Note) Subscript n represents an axis number (1 to 5).

System variable number	System variable name	Attribute	Description
#5041-#5045	[#_ABSOT[n]]	R	Specified current position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 5).
#5061-#5065	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 5).

- Tool length compensation value

M

System variable number	System variable name	Attribute	Description
#5081-#5085	[#_TOFS[n]]	R	Tool length compensation value Note) Subscript n represents an axis number (1 to 5).

- Tool offset value

T

System variable number	System variable name	Attribute	Description
#5081	[#_TOFSWX]	R	X-axis tool offset (wear)
#5082	[#_TOFSWZ]		Z-axis tool offset (wear)
#5083	[#_TOFSWY]		Y-axis tool offset (wear)
#5084 #5085	[#_TOFS[n]]		Tool offset (wear) for an arbitrary axis Note) Subscript n represents an axis number (4 or 5).
#5121	[#_TOFSGX]	R	X-axis tool offset (geometry)
#5122	[#_TOFSGZ]		Y-axis tool offset (geometry)
#5123	[#_TOFSGY]		Z-axis tool offset (geometry)
#5124 #5125	[#_TOFSG[n]]		Tool offset (geometry) for an arbitrary axis Note) Subscript n represents an axis number (4 or 5).

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Servo position deviation

System variable number	System variable name	Attribute	Description
#5101-#5105	[#_SVERR[n]]	R	Servo positional deviation Note) Subscript n represents an axis number (1 to 5).

- Manual handle interruption

System variable number	System variable name	Attribute	Description
#5121-#5125	[#_MIRTP[n]]	R	Manual handle interruption Note) Subscript n represents an axis number (1 to 5).

- Distance to go

System variable number	System variable name	Attribute	Description
#5181-#5185	[#_DIST[n]]	R	Distance to go Note) Subscript n represents an axis number (1 to 5).

- Workpiece origin offset value, extended workpiece origin offset value

M

System variable number	System variable name	Attribute	Description
#5201-#5205	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5221-#5225	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5241-#5245	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5261-#5265	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5281-#5285	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5301-#5305	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5321-#5325	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
Listed below are extended workpiece origin offset values.			
#7001-#7005	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#7021-#7025	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
:	:	:	:
#7941-#7945	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#14001-#14005	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#14021-#14025	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
:	:	:	:
#14941-#14945	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).

T

System variable number	System variable name	Attribute	Description
#5201-#5205	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5221-#5225	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5241-#5245	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5261-#5265	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5281-#5285	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5301-#5305	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).
#5321-#5325	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note)Subscript n represents an axis number (1 to 5).

- System constant

System constant number	System constant name	Attribute	Description
#0,#3100	[#_EMPTY]	R	Null
#3101	[#_PI]	R	Circular constant π = 3.14159265358979323846
#3102	[#_E]	R	Base of natural logarithm e = 2.71828182845904523536

Explanation

R, W, and R/W are attributes of a variable and represents read-only, write-only, and read/write enabled, respectively.

- Interface signal #1000-#1031, #1032, #1033-#1035 (Attribute: R) #1100-#1115, #1132, #1133-#1135 (Attribute: R/W)

[Input signal]

The status of interface input signals can be obtained by reading the value of system variables #1000 to #1032.

Variable number	Variable name	Point	Interface input signal
#1000	[#_UI[0]]	1	UI000 (2^{0})
#1001	[#_UI[1]]	1	UI001 (2^{1})
#1002	[#_UI[2]]	1	UI002 (2^{2})
#1003	[#_UI[3]]	1	UI003 (2^{3})
#1004	[#_UI[4]]	1	UI004 (2^{4})
#1005	[#_UI[5]]	1	UI005 (2^{5})
#1006	[#_UI[6]]	1	UI006 (2^{6})
#1007	[#_UI[7]]	1	UI007 (2^{7})
#1008	[#_UI[8]]	1	UI008 (2^{8})
#1009	[#_UI[9]]	1	UI009 (2^{9})
#1010	[#_UI[10]]	1	UI010 (2^{10})
#1011	[#_UI[11]]	1	UI011 (2^{11})
#1012	[#_UI[12]]	1	UI012 (2^{12})
#1013	[#_UI[13]]	1	UI013 (2^{13})
#1014	[#_UI[14]]	1	UI014 (2^{14})
#1015	[#_UI[15]]	1	UI015 (2^{15})
#1016	[#_UI[16]]	1	UI016 (2^{16})
#1017	[#_UI[17]]	1	UI017 (2^{17})
#1018	[#_UI[18]]	1	UI018 (2^{18})
#1019	[#_UI[19]]	1	UI019 (2^{19})
#1020	[#_UI[20]]	1	UI020 (2^{20})
#1021	[#_UI[21]]	1	UI021 (2^{21})
#1022	[#_UI[22]]	1	UI022 (2^{22})
#1023	[#_UI[23]]	1	UI023 (2^{23})
#1024	[#_UI[24]]	1	UI024 (2^{24})
#1025	[#_UI[25]]	1	UI025 (2^{25})
#1026	[#_UI[26]]	1	UI026 (2^{26})
#1027	[#_UI[27]]	1	UI027 (2^{27})
#1028	[#_UI[28]]	1	UI028 (2^{28})
#1029	[#_UI[29]]	1	UI029 (2^{29})
#1030	[#_UI[30]]	1	UI030 (2^{30})
#1031	[#_UI[31]]	1	UI031 (2^{31})
#1032	[#_UIL[0]]	32	UI000-UI031
#1033	[#_UIL[1]]	32	UI100-UI131
#1034	[#_UIL[2]]	32	UI200-UI231
#1035	[#_UIL[3]]	32	UI300-UI331

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

Since the read value is 1.0 or 0.0 regardless of the unit system, the unit system must be considered when a macro is created.
The input signals at 32 points can be read at a time by reading from system variables #1032 to #1035.

$$\#1032 = \sum_{i=0}^{30} \#[1000+i] \times 2^{i} - \#1031 \times 2^{31}$$

$$\#[1032+n] = \sum_{i=0}^{30} \left\{ 2^{i} \times V_{i} \right\} - 2^{31} \times V_{31}$$

When $UIn_i = 0$, $V_i = 0$.
When $UIn_i = 1$, $V_i = 1$.
n = 0-3

[Output signal]
Interface output signals can be sent by assigning values to system variables #1100 to #1132 for sending interface signals.

Variable number	Variable name	Point	Interface input signal
#1100	[#_UO[0]]	1	UO000 (2^0)
#1101	[#_UO[1]]	1	UO001 (2^1)
#1102	[#_UO[2]]	1	UO002 (2^2)
#1103	[#_UO[3]]	1	UO003 (2^3)
#1104	[#_UO[4]]	1	UO004 (2^4)
#1105	[#_UO[5]]	1	UO005 (2^5)
#1106	[#_UO[6]]	1	UO006 (2^6)
#1107	[#_UO[7]]	1	UO007 (2^7)
#1108	[#_UO[8]]	1	UO008 (2^8)
#1109	[#_UO[9]]	1	UO009 (2^9)
#1110	[#_UO[10]]	1	UO010 (2^{10})
#1111	[#_UO[11]]	1	UO011 (2^{11})
#1112	[#_UO[12]]	1	UO012 (2^{12})
#1113	[#_UO[13]]	1	UO013 (2^{13})
#1114	[#_UO[14]]	1	UO014 (2^{14})
#1115	[#_UO[15]]	1	UO015 (2^{15})
#1116	[#_UO[16]]	1	UO016 (2^{16})
#1117	[#_UO[17]]	1	UO017 (2^{17})
#1118	[#_UO[18]]	1	UO018 (2^{18})
#1119	[#_UO[19]]	1	UO019 (2^{19})
#1120	[#_UO[20]]	1	UO020 (2^{20})
#1121	[#_UO[21]]	1	UO021 (2^{21})
#1122	[#_UO[22]]	1	UO022 (2^{22})
#1123	[#_UO[23]]	1	UO023 (2^{23})
#1124	[#_UO[24]]	1	UO024 (2^{24})
#1125	[#_UO[25]]	1	UO025 (2^{25})
#1126	[#_UO[26]]	1	UO026 (2^{26})
#1127	[#_UO[27]]	1	UO027 (2^{27})
#1128	[#_UO[28]]	1	UO028 (2^{28})
#1129	[#_UO[29]]	1	UO029 (2^{29})

Variable number	Variable name	Point	Interface input signal
#1130	[#_UO[30]]	1	UO030 (2^{30})
#1131	[#_UO[31]]	1	UO031 (2^{31})
#1132	[#_UOL[0]]	32	UO000-UO031
#1133	[#_UOL[1]]	32	UO100-UO131
#1134	[#_UOL[2]]	32	UO200-UO231
#1135	[#_UOL[3]]	32	UO300-UO331

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

The output signals at 32 points can be written at a time by writing to system variables #1132 to #1135. The signals can also be read.

$$\#1132 = \sum_{i=0}^{30} \#[1100+i] \times 2^{i} - \#1131 \times 2^{31}$$

$$\#[1132+n] = \sum_{i=0}^{30} \{2^{i} \times V_{i}\} - 2^{31} \times V_{31}$$

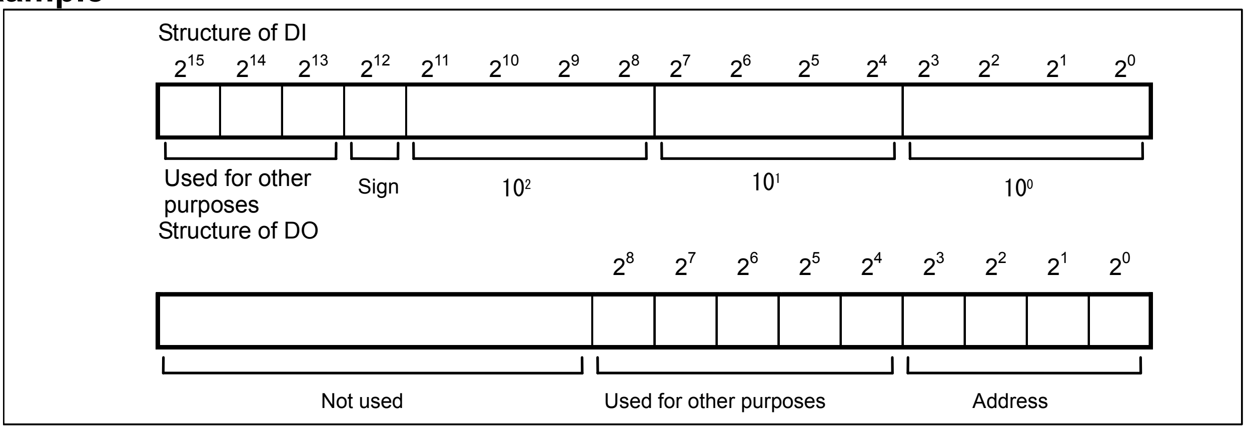
When $UIn_i = 0$, $V_i = 0$.
When $UIn_i = 1$, $V_i = 1$.
n = 0-3

NOTE

1 When a value other than 1.0 or 0.0 is assigned to variables #1100 to #1131, it is assumed as follows.
<null> is assumed to be 0.
A value other than <null> or 0 is assumed to be 1.
Where, a value less than 0.00000001 is undefined.

2 When any of UI016 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO016 to UO031, UO200 to UO231, and UO300 to UO331 are used, parameter MIF (No.6001#0) must be set to 1.

Example



<1> Address switching signed BCD 3 digits are read.

Macro calling instruction

```
G65 P9100 D (address);
```

A custom macro body is created as follows.

```
O9100 ;
#1132 = #1132 AND 496 OR #7 ;  :  Address sending
G65 P9101 T60 ;                :  Timer macro
#100 = BIN[#1032 AND 4095] ;   :  BCD 3 digits are read.
IF [#1012 EQ   0] GOTO   9100 ; :  A sign is attached.
#100 = -#100
N9100 M99 ;
```

<2> Eight types of address switching signed BCD 6 digits (3-digit integer part + 3-digit fractional part) are read into #101.

Structure on the machine side

When DO 2^0 = 0: Data with 3 decimal places
When DO 2^0 = 1: Data with 3-digit integer part
When DO 2^3 to 2^1 = 000: No1 data when #1 = 0
When DO 2^3 to 2^1 = 001: No2 data when #2 = 0
:
When DO 2^3 to 2^1 = 111: No8 data when #8 = 0

Macro calling instruction

```
G65     P9101     D (data number);
```

A custom macro body is created as follows.

```
O9101 ;
G65 P9101 D[#1*2+1] ;
#101 = #100   ;
G65 P9100 D[#1*2] ;
#101 = #101 + #100 / 1000 ;
M99 ;
```

- Tool compensation value #2001-#2800, #10001-#13400 (Attribute: R/W)

M

The compensation values can be obtained by reading system variables #2001 to #2800 or #10001 to #13400 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Tool compensation memory A (bit 6 (NGW) of parameter No.8136 = 1)

- When the number of compensations is 200 or less

Compensation number	Variable number	Variable name
1	#2001	[#_OFS[1]]
2	#2002	[#_OFS[2]]
:	:	:
199	#2199	[#_OFS[199]]
200	#2200	[#_OFS[200]]

- When the number of compensations is 400 (For compensation with a compensation number of 200 or less, #2001 to #2200 can also be used.)

Compensation number	Variable number	Variable name
1	#10001	[#_OFS[1]]
2	#10002	[#_OFS[2]]
:	:	:
399	#10399	[#_OFS[399]]
400	#10400	[#_OFS[400]]

<2> Tool compensation memory C (bit 6 (NGW) of parameter No.8136 = 0)

- When the number of compensations is 200 or less
  When bit 3 (V10) of parameter No.6000 = 0

Compensation number	H code			
	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2201	[#_OFSHG[1]]	#2001	[#_OFSHW[1]]
2	#2202	[#_OFSHG[2]]	#2002	[#_OFSHW[2]]
:	:	:	:	:
199	#2399	[#_OFSHG[199]]	#2199	[#_OFSHW[199]]
200	#2400	[#_OFSHG[200]]	#2200	[#_OFSHW[200]]

When bit 3 (V10) of parameter No.6000 = 1

Compensation number	H code			
	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2001	[#_OFSHG[1]]	#2201	[#_OFSHW[1]]
2	#2002	[#_OFSHG[2]]	#2202	[#_OFSHW[2]]
:	:	:	:	:
199	#2199	[#_OFSHG[199]]	#2399	[#_OFSHW[199]]
200	#2200	[#_OFSHG[200]]	#2400	[#_OFSHW[200]]

Compensation number	D code			
	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]]	#2601	[#_OFSDW[1]]
2	#2402	[#_OFSDG[2]]	#2602	[#_OFSDW[2]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]]	#2799	[#_OFSDW[199]]
200	#2600	[#_OFSDG[200]]	#2800	[#_OFSDW[200]]

NOTE

1. When #2401 to #2800 are used for reading or writing of D codes, bit 5 (D10) of parameter No.6004 must be set to 1.
2. When bit 5 (D10) of parameter No.6004 is set to 1, system variables #2500 to #2806 for workpiece origin offset cannot be used. Use system variables #5201 to #5324.

- When the number of compensations is 400 (For compensation with a compensation number of 200 or less, #2001 to #2800 can also be used.)

When bit 3 (V10) of parameter No.6000 = 0

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#11001	[#_OFSHG[1]]	#10001	[#_OFSHW[1]]
2	#11002	[#_OFSHG[2]]	#10002	[#_OFSHW[2]]
:	:	:	:	:
399	#11399	[#_OFSHG[399]]	#10399	[#_OFSHW[399]]
400	#11400	[#_OFSHG[400]]	#10400	[#_OFSHW[400]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#13001	[#_OFSDG[1]]	#12001	[#_OFSDW[1]]
2	#13002	[#_OFSDG[2]]	#12002	[#_OFSDW[2]]
:	:	:	:	:
399	#13399	[#_OFSDG[399]]	#12399	[#_OFSDW[399]]
400	#13400	[#_OFSDG[400]]	#12400	[#_OFSDW[400]]

When bit 3 (V10) of parameter No.6000 = 1

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#10001	[#_OFSHG[1]]	#11001	[#_OFSHW[1]]
2	#10002	[#_OFSHG[2]]	#11002	[#_OFSHW[2]]
:	:	:	:	:
399	#10399	[#_OFSHG[399]]	#11399	[#_OFSHW[399]]
400	#10400	[#_OFSHG[400]]	#11400	[#_OFSHW[400]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#12001	[#_OFSDG[1]]	#13001	[#_OFSDW[1]]
2	#12002	[#_OFSDG[2]]	#13002	[#_OFSDW[2]]
:	:	:	:	
399	#12399	[#_OFSDG[399]]	#13399	[#_OFSDW[399]]
400	#12400	[#_OFSDG[400]]	#13400	[#_OFSDW[400]]

- Tool compensation value #2001-#2964, #10001-#19200 (Attribute: R/W)

T

The compensation values can be obtained by reading system variables #2001 to #2964 or #10001 to #19200 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Without tool geometry/wear compensation memory (bit 6 (NGW) of parameter No.8136 = 1)

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#2002	[#_OFSX[2]]	
:	:	:	
63	#2063	[#_OFSX[63]]	
64	#2064	[#_OFSX[64]]	

Compensation number	Variable number	Variable name	Description
1	#2101	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#2102	[#_OFSZ[2]]	
:	:	:	
63	#2163	[#_OFSZ[63]]	
64	#2164	[#_OFSZ[64]]	
1	#2201	[#_OFSR[1]]	Tool nose radius compensation value
2	#2202	[#_OFSR[2]]	
:	:	:	
63	#2263	[#_OFSR[63]]	
64	#2264	[#_OFSR[64]]	
1	#2301	[#_OFST[1]]	Virtual tool tipT position
2	#2302	[#_OFST[2]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#2402	[#_OFSY[2]]	
:	:	:	
48	#2448	[#_OFSY[48]]	
49	#2449	[#_OFSY[49]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- When the number of compensations is 200 (For compensation with a compensation number of 64 or less, #2001 to #2449 can also be used.)

Compensation number	Variable number	Variable name	Description
1	#10001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#10002	[#_OFSX[2]]	
:	:	:	
199	#10199	[#_OFSX[199]]	
200	#10200	[#_OFSX[200]]	
1	#11001	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#11002	[#_OFSZ[2]]	
:	:	:	
199	#11199	[#_OFSZ[199]]	
200	#11200	[#_OFSZ[200]]	
1	#12001	[#_OFSR[1]]	Tool nose radius compensation value
2	#12002	[#_OFSR[2]]	
:	:	:	
199	#12199	[#_OFSR[199]]	
200	#12200	[#_OFSR[200]]	
1	#13001	[#_OFST[1]]	Virtual tool tip T position
2	#13002	[#_OFST[2]]	
:	:	:	
199	#13199	[#_OFST[199]]	
200	#13200	[#_OFST[200]]	
1	#14001	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#14002	[#_OFSY[2]]	
:	:	:	
199	#14199	[#_OFSY[199]]	
200	#14200	[#_OFSY[200]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

<2> With tool geometry/wear compensation memory (bit 6 (NGW) of parameter No.8136 = 0)

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSXW[1]]	X-axis compensation value (wear) (*1)
2	#2002	[#_OFSXW[2]]	
:	:	:	
63	#2063	[#_OFSXW[63]]	
64	#2064	[#_OFSXW[64]]	
1	#2101	[#_OFSZW[1]]	Z-axis compensation value (wear) (*1)
2	#2102	[#_OFSZW[2]]	
:	:	:	
63	#2163	[#_OFSZW[63]]	
64	#2164	[#_OFSZW[64]]	
1	#2201	[#_OFSRW[1]]	Tool nose radius compensation value (wear)
2	#2202	[#_OFSRW [2]]	
:	:	:	
63	#2263	[#_OFSRW [63]]	
64	#2264	[#_OFSRW [64]]	
1	#2301	[#_OFST[1]]	Virtual tool tip T position
2	#2302	[#_OFST[2]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSYW[1]]	Y-axis compensation value (wear) (*1)
2	#2402	[#_OFSYW [2]]	
:	:	:	
48	#2448	[#_OFSYW [48]]	
49	#2449	[#_OFSYW [49]]	
1	#2451	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#2452	[#_OFSYG [2]]	
:	:	:	
48	#2498	[#_OFSYG [48]]	
49	#2499	[#_OFSYG [49]]	
1	#2701	[#_OFSXG[1]]	X-axis compensation value (geometry) (*1)
2	#2702	[#_OFSXG[2]]	
:	:	:	
48	#2748	[#_OFSXG [48]]	
49	#2749	[#_OFSXG [49]]	
1	#2801	[#_OFSZG[1]]	Z-axis compensation value (geometry) (*1)
2	#2802	[#_OFSZG[2]]	
:	:	:	
48	#2848	[#_OFSZG[48]]	
49	#2849	[#_OFSZG[49]]	
1	#2901	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#2902	[#_OFSRG[2]]	
:	:	:	
63	#2963	[#_OFSRG[63]]	
64	#2964	[#_OFSRG[64]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- When the number of compensations is 200 (For compensation with a compensation number of 64 or less, #2001 to #2964 can also be used.)

Compensation number	Variable number	Variable name	Description
1	#10001	[#_OFSXW[1]]	X-axis compensation value (wear) (*1)
2	#10002	[#_OFSXW[2]]	
:	:	:	
199	#10199	[#_OFSXW[199]]	
200	#10200	[#_OFSXW[200]]	
1	#11001	[#_OFSZW[1]]	Z-axis compensation value (wear) (*1)
2	#11002	[#_OFSZW[2]]	
:	:	:	
199	#11199	[#_OFSZW[199]]	
200	#11200	[#_OFSZW[200]]	
1	#12001	[#_OFSRW[1]]	Tool nose radius compensation value (wear)
2	#12002	[#_OFSRW [2]]	
:	:	:	
199	#12199	[#_OFSRW[199]]	
200	#12200	[#_OFSRW[200]]	
1	#13001	[#_OFST[1]]	Virtual tool tip T position
2	#13002	[#_OFST[2]]	
:	:	:	
199	#13199	[#_OFST[199]]	
200	#13200	[#_OFST[200]]	
1	#14001	[#_OFSYW[1]]	Y-axis compensation value (wear) (*1)
2	#14002	[#_OFSYW [2]]	
:	:	:	
199	#14199	[#_OFSYW[199]]	
200	#14200	[#_OFSYW[200]]	
1	#15001	[#_OFSXG[1]]	X-axis compensation value (geometry) (*1)
2	#15002	[#_OFSXG[2]]	
:	:	:	
199	#15199	[#_OFSXG[199]]	
200	#15200	[#_OFSXG[200]]	
1	#16001	[#_OFSZG[1]]	Z-axis compensation value (geometry) (*1)
2	#16002	[#_OFSZG[2]]	
:	:	:	
199	#16199	[#_OFSZG[199]]	
200	#16200	[#_OFSZG[200]]	
1	#17001	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#17002	[#_OFSRG[2]]	
:	:	:	
199	#17199	[#_OFSRG[199]]	
200	#17200	[#_OFSRG[200]]	
1	#19001	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#19002	[#_OFSYG[2]]	
:	:	:	
199	#19199	[#_OFSYG[199]]	
200	#19200	[#_OFSYG[200]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Workpiece coordinate system shift amount #2501, #2601 (Attribute: R/W)

T

System variables #2501 and #2601 can be used to read the workpiece coordinate system shift amount of the X-axis and Z-axis, respectively. The workpiece coordinate system shift amount of the X-axis or Z-axis can be changed by entering a value in the corresponding system variable.
(X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes)

Variable number	Variable name	Description
#2501	[#_WKSFTX]	X-axis workpiece shift amount
#2601	[#_WKSFTZ]	Z-axis workpiece shift amount

- Alarm #3000 (Attribute: W)

When an error is detected in a macro, an unit can enter the alarm state. In addition, an alarm message of up to 60 characters with alphabet and numerals can be specified between a control-out and a control-in after the expression. When an alarm message is not specified, a macro alarm is used instead.

Variable number	Variable name	Description
#3000	[#_ALM]	Macro alarm

When bit 1 (MCA) of parameter No.6008 = 0

#3000 = n (ALARM MESSAGE); (n: 0-200)

On the screen, the alarm number obtained by adding the value of #3000 to 3000 and alarm message appear.

(Example) #3000 = 1 (ALARM MESSAGE);

→ "3001 ALARM MESSAGE" appears on the alarm screen.

When bit 1 (MCA) of parameter No.6008 = 1

#3000 = n (ALARM MESSAGE); (n: 0-4095)

On the screen, the alarm number of #3000 and alarm message appear after MC.

(Example) #3000=1 (ALARM MESSAGE);

→ "MC0001 ALARM MESSAGE" appears on the alarm screen.

- Clock #3001, #3002 (Attribute: R/W)

The clock time can be obtained by reading system variables #3001 and #3002 for clocks. The time can be preset by entering a value in the system variables.

Type	Variable number	Variable name	Unit	At power-on	Count condition
Clock 1	#3001	[#_CLOCK1]	1 ms	Reset to 0	Anytime
Clock 2	#3002	[#_CLOCK2]	1 hour	Same as at power-down	When the STL signal is on

The clock accuracy is 16 ms. Clock 1 returns to 0 after a lapse of 2147483648 ms. Clock 2 returns to 0 after a lapse of 9544.37176 hours.

[Example]

Timer

Macro calling command

G65 P9101 T (wait time) ms ;

A macro is created as follows.

```
O9101;
#3001 = 0;   Initial setting
WHILE [#3001 LE #20] DO1:   Wait for a specified time
END1 ;
M99 ;
```

- Controlling of single block stop and waiting for the auxiliary function completion signal #3003 (Attribute: R/W)

Assigning the following values in system variable #3003 allows the specification of whether single block stop is disabled in the following blocks or whether a wait for the completion signal (FIN) of the auxiliary function (M, S, T, or B) before going to the next block is enabled. When a wait for completion signal is disabled, the distribution end signal (DEN) is not sent. Be careful not to specify the next auxiliary function without waiting for the completion signal.

Variable number and variable name	Value	Single block stop	Auxiliary function completion signal
#3003 [#_CNTL1]	0	Enabled	Waiting
	1	Disabled	Waiting
	2	Enabled	Not waiting
	3	Disabled	Not waiting

In addition, the following variable names can be used to enable or disable single block stop and a wait for the auxiliary function completion signal, individually.

Variable name	Value	Single block stop	Auxiliary function completion
[#_M_SBK]	0	Enabled	-
	1	Disabled	-
[#_M_FIN]	0	-	Waiting
	1	-	Not waiting

[Example]

Drill cycle (for incremental programming)
(G81 equivalent)
Macro calling command
G65 P9081 L Iterations R R point Z Z point;
A custom macro body is created as follows.

```
O9081 ;
#3003 = 1 ;                                 Disable single block stop.
G00 Z#18 ;                                  #18 corresponds to R and
G01 Z#26 ;                                  #26 to Z.
G00 Z-[ ROUND[#18] + ROUND[#26] ] ;
#3003 = 0 ;
M99 ;
```

> **NOTE**
> #3003 is cleared by a reset.

- Enabling of feed hold, feedrate override, and exact stop check #3004 (Attribute: R/W)

Assigning the following values in system variable #3004 allows the specification of whether feed hold and feedrate override are enabled in the following blocks or whether exact stop in G61 mode or by G09 command is disabled.

Variable number and variable name	Value	Field hold	Feedrate override	Exact stop
#3004 [#_CNTL2]	0	Enabled	Enabled	Enabled
	1	Disabled	Enabled	Enabled
	2	Enabled	Disabled	Enabled
	3	Disabled	Disabled	Enabled
	4	Enabled	Enabled	Disabled
	5	Disabled	Enabled	Disabled
	6	Enabled	Disabled	Disabled
	7	Disabled	Disabled	Disabled

In addition, the following variable names can be used to enable or disable feed hold, feedrate override, and exact stop in G61 mode or by the G09 command, individually.

Variable number and variable name	Value	Feed hold	Feedrate override	Exact stop
[#_M_FHD]	0	Enabled	-	-
	1	Disabled	-	-
[#_M_OV]	0	-	Enabled	-
	1	-	Disabled	-
[#_M_EST]	0	-	-	Enabled
	1	-	-	Disabled

NOTE

1 These system variables are provided to maintain compatibility with conventional NC programs. It is recommended that functions provided by G63, G09, G61, and other G codes be used to enable or disable feed hold, feedrate override, and exact stop.

2 When the feed hold button is pressed during execution of a block for which feed hold is disabled:
<1> If the feed hold button is kept pressed, operation stops after execution of the block. When single block stop is disabled, however, operation does not stop.
<2> If the pressed feed hold button is released, the feed hold lamp lights, but operation does not stop until the end of the first block that was enabled.

3 #3004 is cleared by a reset.

4 If exact stop is disabled by #3004, the original exact stop position between cutting feed and positioning block is not affected. #3004 can temporarily disable exact stop in G61 mode or by the G09 command between cutting feed and cutting feed.

- Settings #3005 (Attribute: R/W)

Settings can be read and written.
Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting							FCV	

	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#9 (FCV) : Whether to use the FANUC Series 15 program format conversion capability
#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO) : Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

- Stop with a message #3006 (Attribute: W)

When "#3006=1 (MESSAGE);" is commanded in the macro, the program executes blocks up to the immediately previous one and then stops. When a message of up to 60 characters with alphabet and numerals, which is enclosed by a control-in character and control-out character, is programmed in the same block, the message is displayed on the external operator message screen.

Variable number	Variable name	Description
#3006	[#_MSGSTP]	Stop with a message

- Status of a mirror image #3007 (Attribute: R)

The status of an mirror image (setting or DI) at that point in time can be obtained for each axis by reading #3007.

Variable number	Variable name	Description
#3007	[#_MRIMG]	Status of a mirror image

When the status is indicated in binary, each bit corresponds with an axis as follows.

Bit	4	3	2	1	0
nth axis	5	4	3	2	1

For the 5 bits, 0 indicates that a mirror image is disabled and 1 indicates that a mirror image is enabled.
[Example] When #3007 is 3, a mirror image is enabled for the 1st and 2nd axes.

NOTE
1. The status of a programmable mirror image is not reflected on this variable.
2. When the mirror image function is set for the same axis by the mirror image signal and setting, the signal value and setting value are ORed and then output.
3. When mirror image signals for axes other than the controlled axes are turned on, they are not read into system variable #3007.

- Status during restart of a program #3008 (Attribute: R)

Whether a program is restarting can be determined by reading #3008.

Variable number	Variable name	Description
#3008	[#_PRSTR]	0: Program is not restarting. 1: Program is restarting.

- Time #3011, #3012 (Attribute: R)

Year/month/date and hour/minute/second can be obtained by reading system variables #3011 and #3012. This variable is read-only. To change year/month/date and hour/minute/second, use the timer screen.

[Example] May 20, 2004, PM 04:17:05
#3011 = 20040520
#3012 = 161705

- Total number of parts and the number of required parts #3901 and #3902 (Attribute: R/W)

The number of required parts and the number of machined parts can be displayed on the screen by using the operation time and part number displaying function. When the (total) number of machined parts reaches the number of required parts, a signal indicating the fact is sent to the machine (PMC side).
The system variables can be used to read or write the total number of parts and the number of required parts.

Variable number	Variable name	Description
#3901	[#_PRTSA]	Total number of parts
#3902	[#_PRTSN]	Number of required parts

- Type of tool compensation memory #3980 (Attribute: R)

M

System variable #3980 can be used to read the type of compensation memory.

Variable number	Variable name	Description
#3980	[#_OFSMEM]	Types of tool compensation memory 0: Tool compensation memory A 2: Tool compensation memory C

- Main program number #4000 (Attribute: R)

System variable #4000 can be used to read the main program number regardless of the level of a subprogram.

Variable number	Variable name	Description
#4000	[#_MAINO]	Main program number

NOTE

1. The main program number indicates the number of the program that is first started.
2. When an O number is specified by MDI during execution of the main program or when the second O number is specified in DNC mode, the value of #4000 changes to the specified O number. In addition, when no programs are registered or when no O numbers are specified in DNC mode, the value of #4000 changes to 0.

- Modal information #4001-#4130, #4201-#4330, #4401-#4530 (Attribute: R)

The modal information specified before the previous block of the macro statement that reads system variables #4001 to #4130 can be obtained in the block currently being looked ahead, by reading system variables #4001 to #4130.
The modal information of the block currently being executed can be obtained by reading system variables #4201 to #4330.
The modal information specified before the block interrupted by an interruption type custom macro by reading system variables #4401 to #4530.
The unit used when it was specified is applied.

M

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4001 #4201 #4401	[#_BUFG[1]] [#_ACTG[1]] [#_INTG[1]]	Modal information (G code: group 1)
: :	: :	: :	: :
<1> <2> <3>	#4030 #4230 #4430	[#_BUFG[30]] [#_ACTG[30]] [#_INTG[30]]	Modal information (G code: group 30)
<1> <2> <3>	#4102 #4302 #4502	[#_BUFB] [#_ACTB] [#_INTB]	Modal information (B code)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4107 #4307 #4507	[#_BUFD] [#_ACTD] [#_INTD]	Modal information (D code)
<1> <2> <3>	#4108 #4308 #4508	[#_BUFE] [#_ACTE] [#_INTE]	Modal information (E code)
<1> <2> <3>	#4109 #4309 #4509	[#_BUFF] [#_ACTF] [#_INTF]	Modal information (F code)
<1> <2> <3>	#4111 #4311 #4511	[#_BUFH] [#_ACTH] [#_INTH]	Modal information (H code)
<1> <2> <3>	#4113 #4313 #4513	[#_BUFM] [#_ACTM] [#_INTM]	Modal information (M code)
<1> <2> <3>	#4114 #4314 #4514	[#_BUFN] [#_ACTN] [#_INTN]	Modal information (sequence number N)
<1> <2> <3>	#4115 #4315 #4515	[#_BUFO] [#_ACTO] [#_INTO]	Modal information (program number O)
<1> <2> <3>	#4119 #4319 #4519	[#_BUFS] [#_ACTS] [#_INTS]	Modal information (S code)
<1> <2> <3>	#4120 #4320 #4520	[#_BUFT] [#_ACTT] [#_INTT]	Modal information (T code)
<1> <2> <3>	#4130 #4330 #4530	[#_BUFWZP] [#_ACTWZP] [#_INTWZP]	Modal information (additional workpiece coordinate system number P)

T

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4001 #4201 #4401	[#_BUFG[1]] [#_ACTG[1]] [#_INTG[1]]	Modal information (G code: group 1)
: :	: :	: :	: :
<1> <2> <3>	#4030 #4230 #4430	[#_BUFG[30]] [#_ACTG[30]] [#_INTG[30]]	Modal information (G code: group 30)
<1> <2> <3>	#4108 #4308 #4508	[#_BUFE] [#_ACTE] [#_INTE]	Modal information (E code)
<1> <2> <3>	#4109 #4309 #4509	[#_BUFF] [#_ACTF] [#_INTF]	Modal information (F code)
<1> <2> <3>	#4113 #4313 #4513	[#_BUFM] [#_ACTM] [#_INTM]	Modal information (M code)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4114 #4314 #4514	[#_BUFN] [#_ACTN] [#_INTN]	Modal information (sequence number N)
<1> <2> <3>	#4115 #4315 #4515	[#_BUFO] [#_ACTO] [#_INTO]	Modal information (program number O)
<1> <2> <3>	#4119 #4319 #4519	[#_BUFS] [#_ACTS] [#_INTS]	Modal information (S code)
<1> <2> <3>	#4120 #4320 #4520	[#_BUFT] [#_ACTT] [#_INTT]	Modal information (T code)

NOTE

Previous block and running block
Since the CNC reads the block that is ahead of the block currently being executed by the machining program, the block being retrieved by the CNC is normally different from that currently being executed. The previous block indicates the block that is ahead of the block being retrieved by the CNC, that is, the block that is ahead of the program block in which #4001 to #4130 are specified.

[Example]
```
O1234 ;
N10 G00 X200. Y200. ;
N20 G01 X1000. Y1000. F10. ;
  :
  :
N50 G00 X500. Y500. ;
N60 #1 = #4001 ;
```

Assume that the CNC is currently executing N20. If the CNC retrieved and processed the blocks up to N60 as shown above, the running block is N20 and the previous block is N50. Therefore, group 1 modal information in the running block is G01 and group 1 modal information in the previous block is G00.
When N60 #1 = #4201, #1 = 1.
When N60 #1 = #4001, #1 = 0.

- Position information #5001-#5065 (Attribute: R)

The end position of the previous block, the specified current position (for the machine coordinate system and workpiece coordinate system), and the skip signal position can be obtained by reading the values of system variables #5001 to #5065.

Variable number	Variable name	Position information	Coordinate system	Tool position/tool length/cutter compensation	Reading operation during movement
#5001 : #5005	[#_ABSIO[1]] : [#_ABSIO[5]]	1st axis block end point position : 5th axis block end point position	Workpiece coordinate system	Not included	Enabled
#5021 : #5025	[#_ABSMT[1]] : [#_ABSMT[5]]	1st axis current position : 5th axis current position	Machine coordinate system	Included	Disabled

Variable number	Variable name	Position information	Coordinate system	Tool position/tool length/cutter compensation	Reading operation during movement
#5041 : #5045	[#_ABSOT[1]] : [#_ABSOT[5]]	1st axis current position : 5th axis current position	Workpiece coordinate system	Included	Disabled
#5061 : #5065	[#_ABSKP[1]] : [#_ABSKP[5]]	1st axis skip position : 5th axis skip position	Workpiece coordinate system	Included	Enabled

NOTE

1. When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.
2. The block end point position (ABSIO) of the skip (G31) is the position where the skip signal is turned on. If the skip signal is not turned on, the position is the end position of the block.
3. "Read operation during movement is disabled" means that the accurate reading of values during movement is not guaranteed.

- Tool length compensation value #5081#5085 (Attribute: R)

M

Tool length compensation in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5085.

Variable number	Variable name	Position information	Read operation during movement
#5081 : #5085	[#_TOFS[1]] : [#_TOFS[5]]	1st axis tool length compensation value : 5th axis tool length compensation value	Disabled

NOTE

When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.

- Tool offset #5081#5085, #5121-#5125 (Attribute: R)

T

Tool offset in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5085 or #5121 to #5125. (X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes)

<1> Without tool geometry/wear compensation memory (bit 6 (NGW) of parameter No.8136 = 1)

Variable number	Variable name	Position information	Read operation during movement
#5081 #5082 #5083 #5084 #5085	[#_TOFSWX] [#_TOFSWZ] [#_TOFSWY] [#_TOFS[4]] [#_TOFS[5]]	X-axis tool offset value Z-axis tool offset value Y-axis tool offset value 4th axis tool offset value 5th axis tool offset value	Disabled

<2> With tool geometry/wear compensation memory (bit 6 (NGW) of parameter No.8136 = 0)

Variable number	Variable name	Position information	Read operation during movement
#5081 #5082 #5083 #5084 #5085	[#_TOFSWX] [#_TOFSWZ] [#_TOFSWY] [#_TOFS[4]] [#_TOFS[5]]	X-axis tool offset value (wear) Z-axis tool offset value (wear) Y-axis tool offset value (wear) 4th axis tool offset value (wear) 5th axis tool offset value (wear)	Disabled
#5121 #5122 #5123 #5124 #5125	[#_TOFSGX] [#_TOFSGZ] [#_TOFSGY] [#_TOFSG[4]] [#_TOFSG[5]]	X-axis tool offset value (geometry) Z-axis tool offset value (geometry) Y-axis tool offset value (geometry) 4th axis tool offset value (geometry) 5th axis tool offset value (geometry)	

When the tool geometry/wear compensation memory is present (bit 6 (NGW) of parameter No. 8136 is 0), the system variables depend on the settings of bit 2 (LWT) of parameter No. 5002 and bit 4 (LGT) of parameter No. 5002, as shown below.

Variable number	LWT=0 LGT=0	LWT=1 LGT=0	LWT=0 LGT=1	LWT=1 LGT=1
#5081 #5082 #5083 #5084 #5085	Wear compensation	0	Wear compensation	Wear compensation
#5121 #5122 #5123 #5124 #5125	Geometry compensation	Wear compensation + Geometry compensation	Geometry compensation	Geometry compensation

NOTE

1 The set value is read as the tool offset regardless of bit 1 (ORC) of parameter No.5004 and bit 0 (OWD) of parameter No.5040.
2 To read the tool offset (geometry) using #5121 to #5125, set bit 2 (VHD) of parameter No.6004 to 0.

- Servo position deviation #5101-#5105 (Attribute: R)

The servo position deviation for each axis can be obtained by reading system variables #5101 to #5105.

Variable number	Variable name	Position information	Read operation during movement
#5101 : #5105	[#_SVERR[1]] : [#_SVERR[5]]	1st axis servo position deviation : 5th axis servo position deviation	Disabled

NOTE

When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

- Manual handle interruption #5121-#5125 (Attribute: R)

The manual handle interruption for each axis can be obtained by reading system variables #5121 to #5125.

Variable number	Variable name	Position information	Read operation during movement
#5121 : #5125	[#_MIRTP[1]] : [#_MIRTP[5]]	1st axis manual handle interruption : 5th axis manual handle interruption	Disabled

NOTE
When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.

T

NOTE
#5121 to #5125 are enabled only when bit 2 (VHD) of parameter No.6004 is set to 1.

- Distance to go #5181-#5185 (Attribute: R)

The distance to go value for each axis can be obtained by reading system variables #5181 to #5185.

Variable number	Variable name	Position information	Read operation during movement
#5181 : #5185	[#_DIST[1]] : [#_DIST[5]]	1st axis distance to go value : 5th axis distance to go value	Disabled

NOTE
When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.

- Workpiece origin offset value #5201-#5325 (Attribute: R/W)

The workpiece origin offset value can be obtained by reading system variables #5201 to #5325. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#5201 : #5205	[#_WZCMN[1]] : [#_WZCMN[5]]	1st axis external workpiece origin offset value : 5th axis external workpiece origin offset value	External workpiece origin offset value (applied to all coordinate systems)
#5221 : #5225	[#_WZG54[1]] : [#_WZG54[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	G54
#5241 : #5245	[#_WZG55[1]] : [#_WZG55[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	G55
#5261 : #5265	[#_WZG56[1]] : [#_WZG56[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	G56
#5281 : #5285	[#_WZG57[1]] : [#_WZG57[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	G57

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#5301 : #5305	[#_WZG58[1]] : [#_WZG58[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	G58
#5321 : #5325	[#_WZG59[1]] : [#_WZG59[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	G59

M

The following variables can also be used when bit 5 (D15) of parameter No. 6004 is set to 0:

Axis	Function	Variable number
1st axis	External workpiece origin offset value	#2500
	G54 workpiece origin offset value	#2501
	G55 workpiece origin offset value	#2502
	G56 workpiece origin offset value	#2503
	G57 workpiece origin offset value	#2504
	G58 workpiece origin offset value	#2505
	G59 workpiece origin offset value	#2506
2nd axis	External workpiece origin offset value	#2600
	G54 workpiece origin offset value	#2601
	G55 workpiece origin offset value	#2602
	G56 workpiece origin offset value	#2603
	G57 workpiece origin offset value	#2604
	G58 workpiece origin offset value	#2605
	G59 workpiece origin offset value	#2606
3rd axis	External workpiece origin offset value	#2700
	G54 workpiece origin offset value	#2701
	G55 workpiece origin offset value	#2702
	G56 workpiece origin offset value	#2703
	G57 workpiece origin offset value	#2704
	G58 workpiece origin offset value	#2705
	G59 workpiece origin offset value	#2706
4th axis	External workpiece origin offset value	#2800
	G54 workpiece origin offset value	#2801
	G55 workpiece origin offset value	#2802
	G56 workpiece origin offset value	#2803
	G57 workpiece origin offset value	#2804
	G58 workpiece origin offset value	#2805
	G59 workpiece origin offset value	#2806

T

The following variables can be used to maintain compatibility with conventional models.

Axis	Function	Variable number
1st axis	External workpiece origin offset value	#2550
	G54 workpiece origin offset value	#2551
	G55 workpiece origin offset value	#2552
	G56 workpiece origin offset value	#2553
	G57 workpiece origin offset value	#2554
	G58 workpiece origin offset value	#2555
	G59 workpiece origin offset value	#2556

Axis	Function	Variable number
2nd axis	External workpiece origin offset value	#2650
	G54 workpiece origin offset value	#2651
	G55 workpiece origin offset value	#2652
	G56 workpiece origin offset value	#2653
	G57 workpiece origin offset value	#2654
	G58 workpiece origin offset value	#2655
	G59 workpiece origin offset value	#2656
3rd axis	External workpiece origin offset value	#2750
	G54 workpiece origin offset value	#2751
	G55 workpiece origin offset value	#2752
	G56 workpiece origin offset value	#2753
	G57 workpiece origin offset value	#2754
	G58 workpiece origin offset value	#2755
	G59 workpiece origin offset value	#2756
4th axis	External workpiece origin offset value	#2850
	G54 workpiece origin offset value	#2851
	G55 workpiece origin offset value	#2852
	G56 workpiece origin offset value	#2853
	G57 workpiece origin offset value	#2854
	G58 workpiece origin offset value	#2855
	G59 workpiece origin offset value	#2856

NOTE

1. When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2. For the workpiece origin offset values for up to the number (5) of control axes, variables #5201 to #5325 can also be used.

M

NOTE

To use variables #2500 to #2806 and #5201 to #5325, enable the workpiece coordinate system (bit 0 (NWZ) of parameter No. 8136 is 0).

T

NOTE

To use variables #2550 to #2856 and #5201 to #5325, enable the workpiece coordinate system (bit 0 (NWZ) of parameter No. 8136 is 0).

- Workpiece origin offset value of the additional workpiece coordinate system #7001-#7945, #14001-#14945 (Attribute: R/W)

M

The workpiece origin offset value of the additional workpiece coordinate system can be obtained by reading system variables #7001 to #7945, #14001 to #14945. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Additional workpiece system number
#7001 : #7005	[#_WZP1[1]] : [#_WZP1[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	1 (G54.1 P1)

Variable number	Variable name	Controlled axis	Additional workpiece system number
#7021 : #7025	[#_WZP2[1]] : [#_WZP2[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	2 (G54.1 P2)
:	:	:	:
#7941 : #7945	[#_WZP48[1]] : [#_WZP48[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	48 (G54.1 P48)

System variable number = 7000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 48
Axis number: 1 to 5

Variable number	Variable name	Controlled axis	Additional workpiece system number
#14001 : #14005	[#_WZP1[1]] : [#_WZP1[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	1 (G54.1 P1)
#14021 : #14025	[#_WZP2[1]] : [#_WZP2[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	2 (G54.1 P2)
:	:	:	:
#14941 : #14945	[#_WZP48[1]] : [#_WZP48[5]]	1st axis workpiece origin offset value : 5th axis workpiece origin offset value	48 (G54.1 P48)

System variable number = 14000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 48
Axis number: 1 to 5

NOTE

1. When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.
2. To use variables #7001 to #7945 and #14001 to #14945 (G54.1 P1 to G54.1 P48), enable the addition of workpiece coordinate system pairs (48 pairs) (bit 2 (NWN) of parameter No. 8136 is 0).

- Switching between P-CODE variables and system variables (#10000 or later) #8570 (Attribute: R/W)

This system variable allows read/write operations of P-CODE variables (#10000 to #89999) for the macro executor function. For details on P-CODE variables, refer to the Macro Compiler / Macro Executor Programming Manual (B-64303EN-2).
System variable #8570 can be used to make variables #10000 or later correspond to either P-CODE variables or system variables.

#8570 setting	Specified variable	Corresponding variable
#8570 = 0	#10000 : #89999	System variables (#10000) : System variables (#89999)

#8570 setting	Specified variable	Corresponding variable
#8570 = 1	#10000 : #89999	P-CODE variables (#10000) : P-CODE variables (#89999)

Example

#8570 = 0 ;
#10001 = 123 ; → Writing to system variable #10001 (tool compensation)
#8570 = 1 ;
#10001 = 456 ; → Writing to P-CODE variable #10001 (tool compensation)

NOTE

1 Variable #8570 can be used only when the macro executor function is enabled.
2 System variables (#10000 or later) always correspond to system variables specified by their variable names even when #8570 is 1.
3 When an attempt is made to access a variable that cannot be used with P-CODE variables (#10000 or later), an alarm PS0115 occurs.

14.3 ARITHMETIC AND LOGIC OPERATION

Various operations can be performed on variables. Program an arithmetic and logic operation in the same way as for a general arithmetic expression.

#i=<expression>

<Expression>

The expression to the right of the arithmetic and logic operation contains constants and/or variables combined by a function or operator. Variables #j and #k below can be replaced with a constant. If a constant used in an expression has no decimal point, it is assumed to end with a decimal point.

Table 14.3 (a) Arithmetic and logic operation

Type of operation	Operation	Description
<1> Definition or replacement	#i=#j	Definition or replacement of a variable
<2> Addition-type operations	#i=#j+#k #i=#j-#k #i=#j OR #k #i=#j XOR #k	Addition Subtraction Logical OR (bit by bit of 32 bits) Exclusive OR (bit by bit of 32 bits)
<3> Multiplication-type operations	#i=#j*#k #i=#j/#k #i=#j AND #k #i=#j MOD #k	Multiplication Division Logical AND (bit by bit of 32 bits) Remainder (A remainder is obtained after #j and #k are rounded to their nearest whole numbers. When #j is a negative value, #i is assumed to be a negative value.)

Type of operation	Operation	Description
<4> Functions	#i=SIN[#j] #i=COS[#j] #i=TAN[#j] #i=ASIN[#j] #i=ACOS[#j] #i=ATAN[#j] #i=ATAN[#j]/[#k] #i=ATAN[#j,#k] #i=SQRT[#j] #i=ABS[#j] #i=BIN[#j] #i=BCD[#j] #i=ROUND[#j] #i=FIX[#j] #i=FUP[#j] #i=LN[#j] #i=EXP[#j] #i=POW[#j,#k] #i=ADP[#j]	Sine (in degrees) Cosine (in degrees) Tangent (in degrees) Arc sine Arc cosine Arc tangent (one argument), ATN can also be used. Arc tangent (two arguments), ATN can also be used. Arc tangent (two arguments), ATN can also be used. Square root, SQR can also be used. Absolute value Conversion from BCD to binary Conversion from binary to BCD Rounding off, RND can also be used. Rounding down to an integer Rounding up to an integer Natural logarithm Exponent using base e (2.718...) Power (#j to the #kth power) Addition of a decimal point

Explanation

- Angle units

The units of angles used with the SIN, COS, ASIN, ACOS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

- ARCSIN #i = ASIN[#j];

- The solution ranges are as indicated below:
  When the bit 0 (NAT) of parameter No.6004 is set to 0: 270° to 90°
  When the bit 0 (NAT) of parameter No.6004 is set to 1: -90° to 90°
- When #j is beyond the range of -1 to 1, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- ARCCOS #i = ACOS[#j];

- The solution ranges from 180° to 0°.
- When #j is beyond the range of -1 to 1, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- ARCTAN #i = ATAN[#j]/[#k]; (two arguments)

- ATAN[#j,#k] is equivalent to ATAN[#j]/[#k].
- When point (#k,#j) on plane X-Y is given, this function returns the value of the arc tangent for the angle made by the point.
- A constant can be used instead of the #j variable.
- The solution ranges are as follows:
  When the bit 0 (NAT) of parameter No.6004 is set to 0: 0° to 360°
  Example:
  When #1 = ATAN[-1]/[-1]; is specified, #1 is 225.0.
  When the bit 0 (NAT) of parameter No.6004 is set to 1: -180° to 180°
  Example:
  When #1 = ATAN[-1]/[-1]; is specified, #1 is -135.0.

- ARCTAN #i = ATAN[#j]; (one argument)

- When ATAN is specified with one argument, this function returns the main value of arc tangent (-90° ≤ ATAN[#j] ≤ 90°). In other word, this function returns the same value as ATAN in calculator specifications.
- To use this function as the dividend of a division, be sure to enclose it with brackets ([ ]). If this function is not enclosed, ATAN[#j]/[#k] is assumed.

Example:

#100 = [ATAN[1]]/10 ; : Divides ATAN with one argument by 10.
#100 = ATAN[1]/[10] ; : Executes ATAN with two arguments.
#100 = ATAN[1]/10 ; : Assumes ATAN with two arguments, but issues an alarm PS1131 because the X coordinate specification is not enclosed with brackets ([ ]).

- Natural logarithm #i = LN[#j];

- When the antilogarithm (#j) is zero or smaller, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- Exponential function #i = EXP[#j];

- When the result of the operation overflows, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- ROUND function

- When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.

Example:

When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.

- When the ROUND function is used in NC statement addresses, the ROUND function rounds off the specified value according to the least input increment of the address.

Example:

Creation of a drilling program that cuts according to the values of variables #1 and #2, then returns to the original position
Suppose that the increment system is 1/1000 mm, variable #1 holds 1.2345, and variable #2 holds 2.3456. Then,
G00 G91 X-#1 ; Moves 1.235 mm in negative direction.
G01 X-#2 F300 ; Moves 2.346 mm in negative direction.
G00 X[#1+#2] ; Since 1.2345 + 2.3456 = 3.5801 in positive direction, the travel distance is 3.580, which does not return the tool to the original position.

This difference comes from whether addition is performed before or after rounding off. G00X-[ROUND[#1]+ROUND[#2]] ; must be specified to return the tool to the original position.

- Add decimal point (ADP) function

- ADP[#n] (n = 1 to 33) can be executed to add a decimal point to an argument passed with no decimal point, in the subprogram.

Example:

In the subprogram called with G65 P_X10;, the value of ADP[#24] is a value to which a decimal point is added at its end (that is, 10.). Use this function when you do not want to consider the increment system in the subprogram. When bit 4 (CVA) of parameter No. 6007 is set to 1, however, the ADP function cannot be used because any argument is converted to 0.01 the moment it is passed.

NOTE

For compatibility among programs, it is recommended that the ADP function be not used, and decimal points be added in the argument specification for a macro call.

- Rounding up and down to an integer (FUP and FIX)

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.

Example:

Suppose that #1=1.2 and #2=-1.2.
When #3=FUP[#1] ; is executed, 2.0 is assigned to #3.
When #3=FIX[#1] ; is executed, 1.0 is assigned to #3.
When #3=FUP[#2] ; is executed, -2.0 is assigned to #3.
When #3=FIX[#2] ; is executed, -1.0 is assigned to #3.

- Abbreviations of arithmetic and logic operation commands

When a function is specified in a program, the first two characters of the function name can be used to specify the function.

Example:

ROUND → RO
FIX → FI

NOTE
1 POW cannot be abbreviated.
2 When an operation command is entered in an abbreviated form, the abbreviated form is displayed as is.
For example, when "RO" is entered, "RO" is displayed as is without being converted to "ROUND".

- Priority of operations

<1> Functions
<2> Operations such as multiplication and division (*, /, AND)
<3> Operations such as addition and subtraction (+, -, OR, XOR)

Example) #1=#2+#3*SIN[#4];
<1>
<2>
<3>
<1>, <2> and <3> indicate the order of operations.

- Bracket nesting

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, an alarm PS0118 occurs.

Example) #1=SIN [[[#2+#3] *#4 +#5] *#6];
<1>
<2>
<3>
<4>
<5>
<1> to <5> indicate the order of operations.

Limitation

- **Caution concerning decreased precision**
  When bit 0 (F0C) of parameter No. 6008 is set to 0

- Addition and subtraction

  Note that when an absolute value is subtracted from another absolute value in addition or subtraction, the relative error may become 10^{-15} or greater.
  For example, assume that #1 and #2 have the following true values in the process of operation.
  (The following values are examples in the process of operation and cannot actually be specified from any program.)

  ```
  #1=9876543210.987654321
  #2=9876543210.987657777
  ```

  You cannot obtain the following result with operation #2-#1:

  ```
  #2-#1=0.000003456
  ```

  This is because the precision of custom macro variables is 15 decimal digits. With this precision, the values of #1 and #2 become:

  ```
  #1=9876543210.987650000
  #2=9876543210.987660000
  ```

  (Precisely, the actual values are slightly different from the above values because they are internally processed in binary.) Therefore, the result is:

  ```
  #2-#1=0.000010000
  ```

  A large error occurs.

- Logical expressions

  Be aware of errors that can result from conditional expressions using EQ, NE, GT, LT, GE, and LE because they are processed basically in the same way as addition and subtraction. For example, if the following statement is used to decide whether #1 is equal to #2 in the above example, a correct decision may not be resulted because errors may occur:

  ```
  IF [#1 EQ #2]
  ```

  Evaluate the difference between #1 and #2 with:

  ```
  IF [ABS [#1-#2]LT 0.1]
  ```

  Then, assume that the values are equal when the difference does not exceed the allowable error range.

- Trigonometric functions

  The absolute error is guaranteed for trigonometric functions. However, the relative error is 10^{-15} or greater. Carefully perform multiplication or division after executing a trigonometric function.

- FIX function

  When using the FIX function for the result of an operation, be careful with the precision. For example, when the following operations are performed, the value of #3 may not always be 2.

  ```
  N10 #1=0.002;
  N20 #2=#1*1000;
  N30 #3=FIX[#2];
  ```

  This is because an error may occur in operation N20 and the result may not be

  ```
  #2=2.0000000000000000
  ```

  but a value a little smaller than 2 such as the following:

  ```
  #2=1.9999999999999997
  ```

  To prevent this, specify N30 as follows:

  ```
  N30 #3=FIX[#2+0.001];
  ```

  Generally, specify the FIX function as follows:

  FIX[expression] → FIX[expression ±ε]

  (Specify +ε when the value of the expression is positive or -ε when it is negative, and 0.1, 0.01, 0.001, ... for ε as required.)

> **NOTE**
> The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 790.

When bit 0 (F0C) of parameter No. 6008 is set to 1

Errors may occur when operations are performed.

Table 14.3 (b) Errors involved in operations

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.
4 Function TAN performs SIN/COS.
5 Note that, in the case of natural logarithm #i=LN[#j]; and exponential function #i=EXP[#j];, the relative error may become 10^{-8} or greater.
6 The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 110.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.

  Example:

  When an attempt is made to assign the following values to variables #1 and #2:

  #1=9876543210123.456
  #2=9876543277777.777

  the values of the variables become:

  #1=9876543200000.000
  #2=9876543300000.000

  In this case, when #3=#2-#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)

- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.

  Example:

  IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.
  Therefore, instead find the difference between the two variables with IF[ABS[#1-#2]LT0.001].
  Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).

- Also, be careful when rounding down a value.

  Example:

  When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997.
  Here, when #3=FIX[#2]; is specified, the resulting value of variable #3 is not 2.0 but 1.0.
  In this case, round down the value after correcting the error so that the result is greater than the expected number, or round it off as follows:

  #3=FIX[#2+0.001]
  #3=ROUND[#2]

- Brackets

Brackets ([ ]) are used to enclose an expression.
Note that parentheses ( ) are used for comments.

- Divisor

When a divisor of zero is specified in a division, an alarm PS0112 occurs.

14.4 READING PARAMETERS

Overview

By using the PRM function, it is possible to read parameters.

Format	Remarks
#i = PRM[#j, #k] ;	In the case of parameters other than axis type parameters
#i = PRM[#j, #k] / [#l] ;	In the case of axis type parameters

Explanation

- Reading a parameter
#i=PRM[#j,#k]
#i=PRM[#j,#k]/[#l]

- For #j, enter a parameter number. If the number of a parameter that cannot be read, alarm PS0119, "ARGUMENT VALUE OUT OF RANGE", is issued.
- To read a bit type parameter, specify, for #k, the bit number of the bit type parameter in the range of 0 to 7. If a bit number is specified, data with the specified bit is read. If none is specified, data with all bits is read. For parameters other than bit type parameters, the bit number is ignored.
- For #l, set the axis number of an axis type parameter in the range of 1 to 5. If an axis type parameter is to be read but #l is not specified, alarm PS0119 is issued. For parameters other than axis type parameters, #1 may be omitted together with '/'.

Example

1. Reading the value of the third axis of bit 0 (MIR) of bit axis type parameter No. 0012
   If parameter No. 0012 (third axis) = 10000001
   #2=12 ; Parameter number setting
   #3=0 ; Bit number setting
   #4=3 ; Axis number setting
   If reading data with all bits
   #1=PRM[#2]/[#4] ; #1=10000001
   If reading data with a specified bit
   #1=PRM[#2, #3]/[#4] ; #1=1

2. Reading the value of the fourth axis of axis type parameter No. 1322
   #2=1322 ; Parameter number setting
   #4=4 ; Axis number setting
   #1=PRM[#2]/[#4] ;

3. Reading bit 2 (SBP) of bit type parameter No. 3404
   If parameter No. 3404 = 10010000
   #2=3404 ; Parameter number setting
   #3=2 ; Bit number setting
   If reading data with all bits
   #1=PRM[#2] ; #1=10010000
   If reading data with a specified bit
   #1=PRM[#2,#3] ; #1=0

14.5 MACRO STATEMENTS AND NC STATEMENTS

The following blocks are referred to as macro statements:

- Blocks containing an arithmetic or logic operation (=)
- Blocks containing a control statement (such as GOTO, DO, END)
- Blocks containing a macro call command (such as macro calls by G65, G66, G67, or other G codes, or by M codes)

Any block other than a macro statement is referred to as an NC statement.

Explanation

- Differences from NC statements

- Even when single block mode is on, the machine does not stop. Note, however, that the machine stops in the single block mode when bit 5 of parameter SBM No. 6000 is 1.

M

- Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode.

- NC statements that have the same property as macro statements

- An NC statement has the same property as a macro statement when the NC statement is a subprogram call command (subprogram call by an M98, M code or subprogram call by a T code) and is also a block that does not include any command address other than O, N, P, and L.
- An NC statement has the same property as a macro statement when the NC statement is an M99 command and is also a block that does not include any command address other than O, N, P, and L.

14.6 BRANCH AND REPETITION

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

Branch and repetition	GOTO	(unconditional branch)
	IF	(conditional branch: if ..., then...)
	WHILE	(repetition while ...)

14.6.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999 is specified, an alarm PS1128 occurs. A sequence number can also be specified using an expression.

GOTOn ; n: Sequence number (1 to 99999)

Example:

```
GOTO 1;
GOTO #10;
```

> ⚠ **WARNING**
> Do not specify multiple blocks with the same sequence number in a single program. It is very dangerous to specify such blocks because the destination of a branch from the GOTO statement is undefined.

NOTE
1 A backward branch takes more time as compared with a forward branch.
2 In the block with sequence number n, which is the branch destination of the GOTO n command, sequence number n must be located at the beginning of the block. Otherwise, the branch cannot be executed.

14.6.2 GOTO Statement Using Stored Sequence Numbers

When the GOTO statement is executed in a custom macro control command, a sequence number search is made for sequence numbers stored at previous execution of the corresponding blocks at a high speed.
Sequence numbers stored at previous execution indicate the sequence numbers for a subprogram call and the sequence numbers that are unique in the same program of the sequence numbers at previous execution, and the CNC records these sequence numbers.
The storage type differs depending on the values of the following parameters.

(1) When bit 1 (MGO) of parameter No. 6000 is set to 1
- Fixed type: Up to 20 sequence numbers stored at execution of the corresponding blocks from the start of operation

(2) When bit 4 (HGO) of parameter No. 6000 is set to 1
- Variable type: Up to 30 sequence numbers stored at execution of the corresponding blocks before execution of the GOTO statement
- History type: Up to 10 sequence numbers stored by a sequence number search previously made using the GOTO statement

The stored sequence numbers are canceled in the following cases:
- Immediately after power-on
- After a reset
- Operation after program registration or editing (including background editing and MDI program editing)

⚠ WARNING
Do not specify multiple blocks with the same sequence number in a single program.
It is very dangerous to specify the sequence number of the branch destination before and after the GOTO statement and execute the GOTO statement because the branch destination changes according to the values of the parameters as shown below:

When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1	When both bits 1 (MGO) and 4 (HGO) of parameter No. 6000 are set to 0
: N10; : GOTO10; : N10; A branch to N10 before the GOTO statement occurs.	: N10; : GOTO10; : N10; A branch to N10 after the GOTO statement occurs.

When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1 and the GOTO statement is executed, the sequence number of the branch destination may not be contained in the sequence numbers stored at previous execution of the corresponding blocks. In this case, a branch to the sequence number in a block following the GOTO statement occurs (the destination is the same as when both bits are set to 0).

> **NOTE**
> When an external program is read and executed by DNC operation, the executed sequence numbers are not stored.
> When a program registered in memory is executed by a subprogram call, the sequence numbers are stored.

> ⚠ **CAUTION**
> According to the restrictions on the GOTO statement, no branch to a sequence number within a DO-END loop cannot be made. If a program in which a branch to a sequence number within a loop occurs is executed, operation may differ depending on whether the GOTO statement using stored sequence numbers is used.

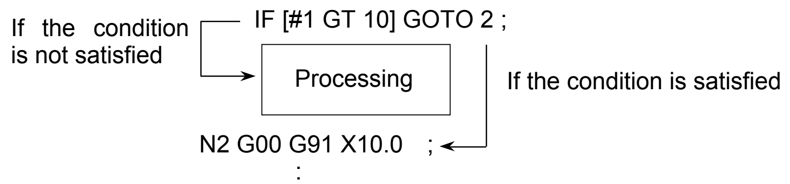
14.6.3 Conditional Branch (IF Statement)

Specify a <conditional expression> after IF.

IF[<conditional expression>]GOTOn

If the specified <conditional expression> is satisfied (true), a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.



IF[<conditional expression>]THEN

If the specified <conditional expression> is satisfied (true), a macro statement specified after THEN is executed.
Only a single macro statement is executed.

If the values of #1 and #2 are the same, 0 is assigned to #3.
IF[#1 EQ #2] THEN #3=0 ;

If the values of #1 and #2 are the same and those of #3 and #4 are also the same, 0 is assigned to #5.
IF[[#1 EQ #2] AND [#3 EQ #4]] THEN #5 = 0 ;

If the values of #1 and #2 are the same or those of #3 and #4 are the same, 0 is assigned to #5.
IF[[#1 EQ #2] OR [#3 EQ #4]] THEN #5 = 0 ;

Explanation

- <Conditional expression>

<Conditional expressions> are divided into <simple conditional expressions> and <complex conditional expressions>. In a <simple conditional expression>, a relational operator described in Table 14.5 (a) is specified between two variables or between a variable and constant to be compared. An <expression> can be used instead of a variable. With a <complex conditional expression>, an AND (logical AND), OR (logical OR), or XOR (exclusive OR) operation is performed for the results (true or false) of multiple <simple conditional expressions>.

- Relational operators

Relational operators each consist of two letters and are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value. Note that the equal sign (=) and inequality sign (>, <) cannot be used as a relational operator.

Table 14.6 (a) Relational operators

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≥)
LT	Less than(<)
LE	Less than or equal to(≤)

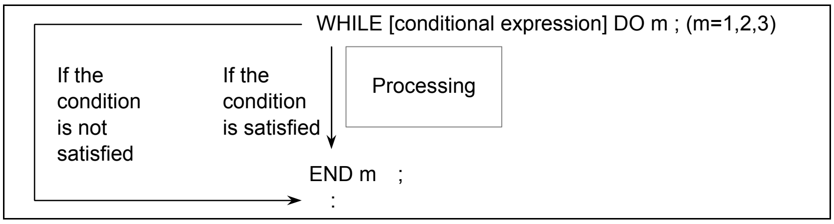
Sample program

The sample program below finds the total of numbers 1 to 10.

```
O9500 ;
  #1=0 ; ............................... Initial value of the variable to hold the sum
  #2=1 ; ............................... Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2 ;    Branch to N2 when the addend is greater than 10
  #1=#1+#2 ; ....................... Calculation to find the sum
  #2=#2+1 ; ......................... Next addend
  GOTO 1 ;.......................... Branch to N1
N2 M30 ; ............................. End of program
```

14.6.4 Repetition (WHILE Statement)

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If the specified condition is not satisfied, program execution proceeds to the block after END.



Explanation

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, an alarm PS0126 occurs.

- Nesting

The identification numbers (1 to 3) in a DO-END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), an alarm PS0124 occurs.

1. The identification numbers (1 to 3) can be used as many times as required.

```
WHILE [ … ] DO 1 ;
    Processing
END 1 ;
  :
WHILE [ … ] DO 1 ;
    Processing
END 1 ;
  :
```

2. DO ranges cannot overlap.

```
WHILE [ … ] DO 1 ;
    Processing
WHILE [ … ] DO 2 ;
  :
END 1 ;
    Processing
END 2 ;
```

3. DO loops can be nested to a maximum depth of three levels.

```
WHILE [ … ] DO 1 ;
  :
  WHILE [ … ] DO 2 ;
    :
    WHILE [ … ] DO 3 ;
        Processing
    END 3 ;
      :
  END 2 ;
    :
END 1 ;
```

4. Control can be transferred to the outside of a loop.

```
WHILE [ … ] DO 1 ;
IF [ … ] GOTO n ;
END 1 ;
Nn
```

5. Branches cannot be made to a location within a loop.

```
IF [ … ] GOTO n ;
  :
WHILE [ … ] DO 1 ;
  :
Nn … ;
END 1 ;
```

Limitation

- Infinite loops

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

- Processing time

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Therefore, in the case of processing in the reverse direction, use the WHILE statement for repetition to reduce processing time.

- Undefined variable

In a conditional expression that uses EQ or NE, a <null> and zero have different effects. In other types of conditional expressions, a <null> is regarded as zero.

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10] DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

14.7 MACRO CALL

A macro program can be called using the following methods. The calling methods can roughly be divided into two types: macro calls and subprogram calls.
A macro program can also be called during MDI operation in the same way.

- Macro call
  - Simple call (G65)
  - Modal call (G66, G67)
  - Macro call using a G code
  - Macro call using an M code
- Subprogram call
  - Subprogram call using an M code
  - Subprogram call using a T code
  - Subprogram call using a specific code

Limitation

- Call nesting

Macro calls can be nested to a depth of up to five levels and subprogram calls can be nested to a depth of up to ten levels; calls can be nested to a depth of up to 15 levels in total.

- Differences between macro calls and subprogram calls

Macro call (G65, G66, Ggg, or Mmm) differs from subprogram call (M98, Mmm, or Ttt) as described below.

- With a macro call, an argument (data passed to a macro) can be specified. A subprogram call does not have this capability.
- If a macro call block contains another NC command (such as G01 X100.0 G65 Pp), an alarm PS0127 occurs.
- If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the subprogram is called after the command is executed.
- In any macro call block, the machine does not stop in the single block mode.
  If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the machine stops in the single block mode.
- With a macro call, the level of local variables changes. With a subprogram call, the level of local variables does not change. (See "Local variable levels" in Limitation of Subsection 14.6.1.)

14.7.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.

G65 P p L ℓ <argument-specification> ;	P : Number of the program to call ℓ : Repetition count (1 by default) Argument : Data passed to the macro

```
O0001 ;
    :
G65 P9010 L2 A1.0 B2.0 ;
    :
M30 ;
```

```
O9010 ;
#3=#1+#2 ;
IF [#3 GT 360] GOTO 9 ;
G00 G91 X#3 ;
N9 M99 ;
```

Explanation

- Call

- After G65, specify at address P the program number of the custom macro to call.
- When a number of repetitions is required, specify a number from 1 to 999999999 after address L. When L is omitted, 1 is assumed.
- By using argument specification, values are assigned to corresponding local variables.

- Argument specification

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

- Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

- Addresses G, L, N, O, and P cannot be used in arguments.
- Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.
- Addresses do not need to be specified alphabetically. They conform to word address format.
  I, J, and K need to be specified alphabetically, however.
  Argument specification I is always used for I, J, and K by setting bit 7 (IJK) of parameter No. 6008 to 1

Example

- When bit 7 (IJK) of parameter No. 6008 is 0, I_ J_ K_ means that I = #4, J = #5, and K = #6 while K_ J_ I_ means K = #6, J = #8, and I= #10 because argument specification II is used.
- When bit 7 (IJK) of parameter No. 6008 is 1, K_ J_ I_ means that I = #4, J = #5, and K = #6, which is the same as with I_ J_ K_, because argument specification I is used.

- Argument specification II
  Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three-dimensional coordinates as arguments.

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	IK_3	#12	J_7	#23
B	#2	I_4	#13	K_7	#24
C	#3	J_4	#14	I_8	#25
I_1	#4	K_4	#15	J_8	#26
J_1	#5	I_5	#16	K_8	#27
K_1	#6	J_5	#17	I_9	#28
I_2	#7	K_5	#18	J_9	#29
J_2	#8	I_6	#19	K_9	#30
K_2	#9	J_6	#20	I_{10}	#31
I_3	#10	K_6	#21	J_{10}	#32
J_3	#11	I_7	#22	K_{10}	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

NOTE
When bit 7 (IJK) of parameter No. 6008 is 1, argument specification II cannot be used.

Limitation

- Format

G65 must be specified before any argument.

- Mixture of argument specifications I and II

The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.

[Example]

G65 A1.0 B2.0 I-3.0 I4.0 D5.0 P1000 ;

(Variables)
#1:1.0
#2:2.0
#3:
#4:-3.0
#5:
#6:
#7:~~4.0~~ 5.0

When both the I4.0 and D5.0 arguments are commanded for variable #7 in this example, the latter, D5.0, is valid.

- Position of the decimal point

The units used for argument data passed without a decimal point correspond to the least input increment of each address.

⚠ CAUTION

The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

M

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
D, E, H, M, S, or T	0	
Q or R	α (NOTE 2)	
A, C, I, J, K, X, Y, or Z	α (NOTE 2)	β (NOTE 3)
B, U, V (NOTE 1), or W	0	β (NOTE 3)
Second auxiliary function	γ (NOTE 4)	

Address	Metric input	Inch input
F (G93 mode)	3	
F (G94 mode)	0	2
F (G95 mode)	2 (NOTE 5)	4 (NOTE 5)

NOTE

1 When V is used in a call using a specific address, the number of decimal places is determined according to the setting for the reference axis.

2 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 4.

3 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective.
When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.

4 γ is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the following table. (When bit 7 (BDX) of parameter No. 3450 is set to 1, γ is also determined in the same way.)

<table>
<tr><th rowspan="3">Increment system for the reference axis</th><th rowspan="3">AUP (No.3450#0) = 0</th><th colspan="4">AUP(3450#0) = 1</th></tr>
<tr><th colspan="2">AUX (No.3405#0) = 0</th><th colspan="2">AUX (No.3405#0) = 1</th></tr>
<tr><th>Metric</th><th>Inch</th><th>Metric</th><th>Inch</th></tr>
<tr><td>IS-A</td><td rowspan="3">0</td><td colspan="2">2</td><td>2</td><td>3</td></tr>
<tr><td>IS-B</td><td colspan="2">3</td><td>3</td><td>4</td></tr>
<tr><td>IS-C</td><td colspan="2">4</td><td>4</td><td>5</td></tr>
</table>

NOTE

5 When bit 1 (FR3) of parameter No. 1405 is 1, the values in the table need to be incremented by 1.

6 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.

T

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
H, M, Q, S, or T	0	
R	α (NOTE 1)	
D	0/α (NOTE 6)	
A, B, C, I, J, K, U, V, W, X, Y, or Z	α (NOTE 1)	β (NOTE 2)
Second auxiliary function	γ (NOTE 3)	

Address	Metric input	Inch input
E, F (G98 mode)	0 (NOTE 4)	2 (NOTE 4)
E, F (G99 mode)	4	6

NOTE

1 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 2.

2 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective.

When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.

3 γ is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the following table. (When bit 7 (BDX) of parameter No. 3450 is set to 1, γ is also determined in the same way.)

<table>
<tr><th rowspan="3">Increment system for the reference axis</th><th rowspan="3">AUP (No.3450#0) = 0</th><th colspan="4">AUP(3450#0) = 1</th></tr>
<tr><th colspan="2">AUX (No.3405#0) = 0</th><th colspan="2">AUX (No.3405#0) = 1</th></tr>
<tr><th>Metric</th><th>Inch</th><th>Metric</th><th>Inch</th></tr>
<tr><td>IS-A</td><td rowspan="3">0</td><td colspan="2">2</td><td>2</td><td>3</td></tr>
<tr><td>IS-B</td><td colspan="2">3</td><td>3</td><td>4</td></tr>
<tr><td>IS-C</td><td colspan="2">4</td><td>4</td><td>5</td></tr>
</table>

NOTE

4 When bit 2 (FM3) of parameter No. 1404 is 1, the values in the table need to be incremented by 3.

5 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.

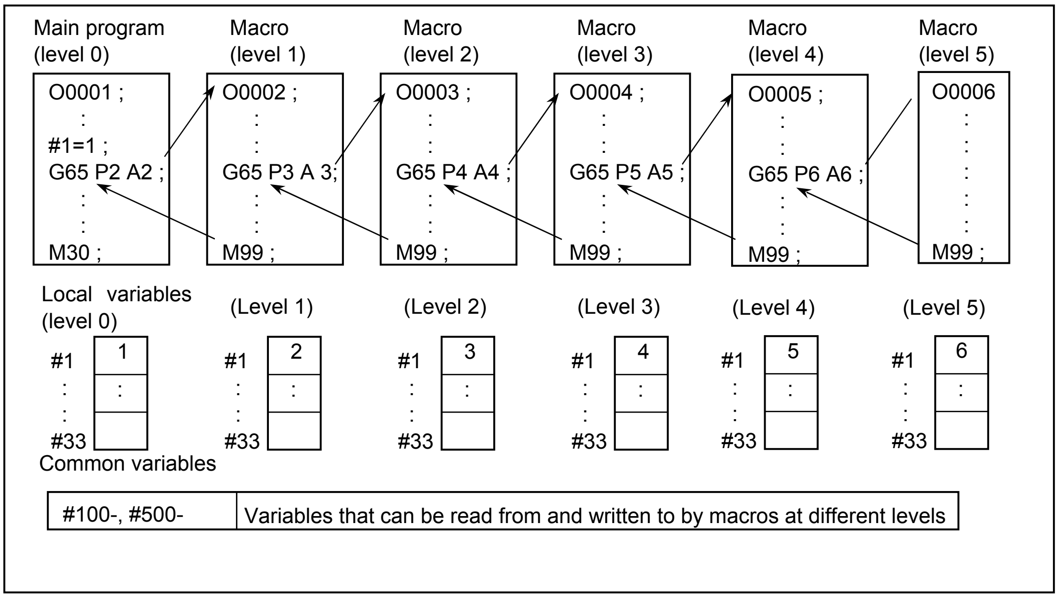
6 When bit 2 (DPD) of parameter No. 6019 is set to 0, the number of decimal places is 0.
When bit 2 (DPD) of parameter No. 6019 is set to 1, the number of decimal places is α.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.
A macro program can also be called during MDI operation in the same way.

- Local variable levels

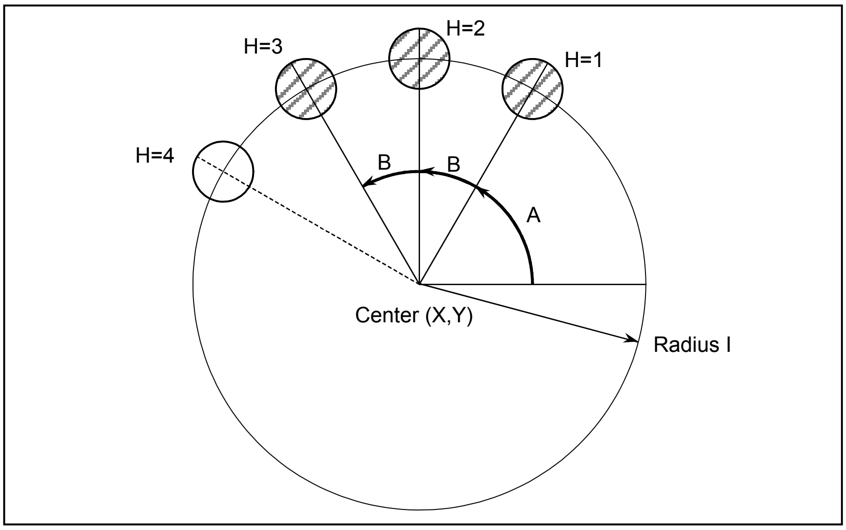
- Local variables from level 0 to 5 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G65, G66, Ggg, or Mmm), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC.
- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program (bolt hole circle)

M

A macro is created which drills H holes at intervals of B degrees after a start angle of A degrees along the periphery of a circle with radius I.
The center of the circle is (X,Y). Commands can be specified in either the absolute or incremental programming.
To drill in the clockwise direction, specify a negative value for B.



- Calling format

G65 P9100 Xx Yy Zz Rr Ff Ii Aa Bb Hh ;

X : X coordinate of the center of the circle
(absolute or incremental programming)........ (#24)
Y : Y coordinate of the center of the circle
(absolute or incremental programming)........ (#25)
Z : Hole depth........ (#26)
R : Coordinates of an approach point .... (#18)
F : Cutting feedrate........ (#9)
I : Radius of the circle .... (#4)
A : Drilling start angle .... (#1)
B : Incremental angle
(clockwise when a negative value is specified) .... (#2)
H : Number of holes........ (#11)

- Program calling a macro program

```
O0002 ;
G90 G92 X0 Y0 Z100.0 ;
G65 P9100 X100.0 Y50.0 R30.0 Z-50.0 F500 I100.0 A0 B45.0 H5 ;
M30 ;
```

- Macro program (called program)

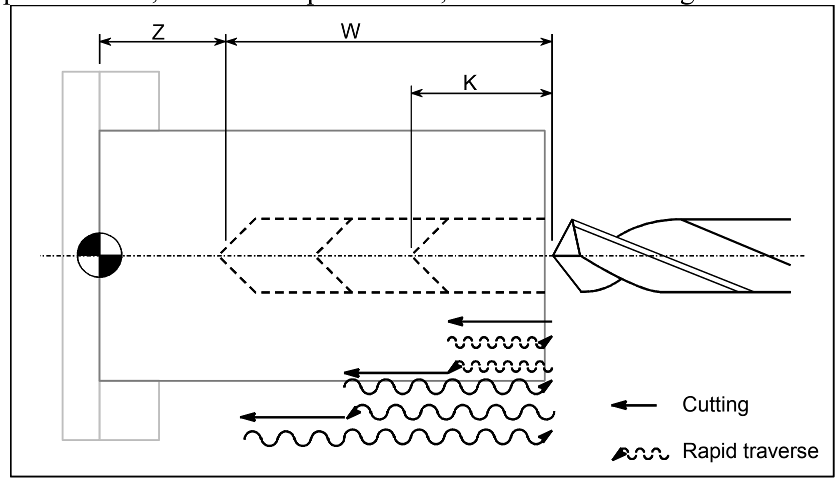
```
O9100 ;
#3=#4003 ; .............................. Stores G code of group 3.
G81 Z#26 R#18 F#9 K0 ; (Note).. Drilling cycle.
                                 Note:   L0 can also be used.
IF[#3 EQ 90]GOTO 1 ; ............ Branches to N1 in the G90 mode.
#24=#5001+#24 ; ..................... Calculates the X coordinate of the center.
#25=#5002+#25 ;   ................... Calculates the Y coordinate of the center.
N1 WHILE[#11 GT 0]DO 1 ;..... Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1] ;.............. Calculates a drilling position on the X-axis.
#6=#25+#4*SIN[#1] ; ............... Calculates a drilling position on the Y-axis.
G90 X#5 Y#6;........................... Performs drilling after moving to the target position.
#1=#1+#2 ; .............................. Updates the angle.
#11=#11-1 ; ............................. Decrements the number of holes.
END 1 ;
G#3 G80 ;................................ Returns the G code to the original state.
M99 ;
```

Meaning of variables:
Stores the G code of group 3. #5: X coordinate of the next hole to drill #6: Y coordinate of the next hole to drill

Sample program (Drill cycle)

T

Move the tool beforehand along the X- and Z-axes to the position where a drilling cycle starts. Specify Z or W for the depth of a hole, K for the depth of a cut, and F for the cutting feedrate to drill the hole.



- Calling format

G65 P9100 { Zz / Ww } Kk Ff ;

Z : Hole depth (absolute programming)
W : Hole depth (incremental programming)
K : Cutting amount per cycle
F : Cutting feedrate

- Program calling a macro program

```
O0002 ;
G50 X100.0 Z200.0 ;
G00 X0 Z102.0 S1000 M03 ;
G65 P9100 Z50.0 K20.0 F0.3 ;
G00 X100.0 Z200.0 M05 ;
M30 ;
```

- Macro program (called program)

```
     O9100;
     #1=0 ; .......................................Clear the data for the depth of the current hole.
     #2=0 ; .......................................Clear the data for the depth of the preceding hole.
     IF [#23 NE #0] GOTO 1 ; .........If incremental programming, specifies the jump to N1.
     IF [#26 EQ #0] GOTO 8 ; .........If neither Z nor W is specified, an error occurs.
     #23=#5002-#26 ; ......................Calculates the depth of a hole.
N1   #1=#1+#6 ; ...............................Calculates the depth of the current hole.
     IF [#1 LE #23] GOTO 2 ;..........Determines whether the hole to be cut is too deep?
     #1=#23 ; ...................................Clamps at the depth of the current hole.
N2   G00 W-#2 ; ...............................Moves the tool to the depth of the preceding hole at the cutting feedrate.
     G01 W- [#1-#2] F#9 ; ...............Drills the hole.
     G00 W#1 ; ................................Moves the tool to the drilling start point.
```

```
     IF [#1 GE #23] GOTO 9 ; .........Checks whether drilling is completed.
     #2=#1 ; .....................................Stores the depth of the current hole.
     GOTO 1 ;
N9   M99 ;
N8   #3000=1 (NOT Z OR W COMMAND) ; .....Issues an alarm.
```

14.7.2 Modal Call: Call After the Move Command (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

```
G66 P p L ℓ <argument-specification> ;   P : Number of the program to call
                                         ℓ : Repetition count (1 by default)
                                         Argument : Data passed to the macro
```

O0001 ; : G66 P9100 L2 A1.0 B2.0 ; G00 G90 X100.0 ; Y200.0 ; X150.0 Y300.0 ; G67 ; : M30 ;	O9100 ; : G00 Z-#1 ; G01 Z-#2 F300 ; : : : : M99 ;

Explanation

- Call

- After G66, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 999999999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.
- In the G66 mode, a macro can be called.

- Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

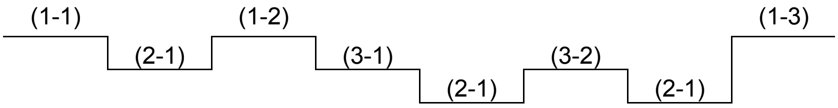
- Modal call nesting

For a single modal call (when G66 is specified only once), each time the move command is executed, the specified macro is called. When nested modal macro calls are specified, the macro at the next higher level is called each time the move command for a macro call is executed.
Macros are called in reverse order in which they are specified. Each time G67 is issued, the macros are canceled one by one in reverse order in which they are specified.

[Example]

```
G66 P9100 ;                 O9100 ;              O9200 ;
X10.0 ;      (1-1)          Z50.0 ;   (2-1)      X60.0 ;      (3-1)
G66 P9200 ;                 M99 ;                Y70.0 ;      (3-2)
X15.0 ;      (1-2)                               M99;
G67 ;        Cancels P9200.
G67 ;        Cancels P9100.
X-25.0 ;     (1-3)
```

Execution order of the above program (blocks not containing the move command omitted)



* No modal call is performed after (1-3) because the mode is not the macro call mode.

Limitation

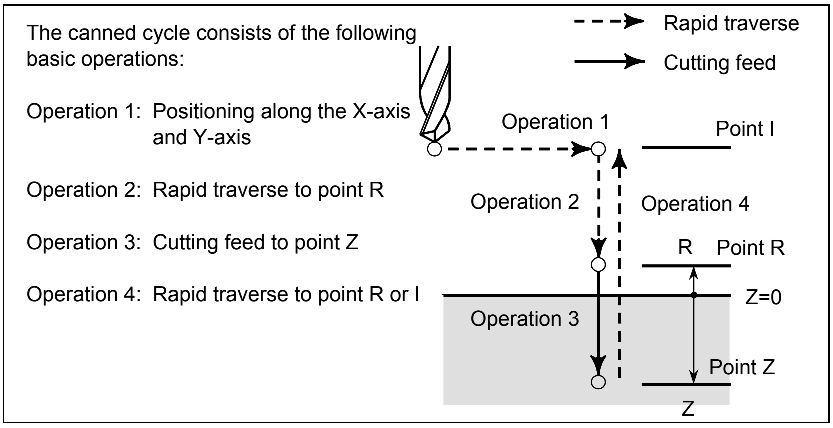
- G66 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66 mode, an alarm PS1100 occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- In a G66 block, no macros can be called. Local variables (arguments) are set, however.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a auxiliary function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

> **NOTE**
> If M99 is specified in a block in which a call is performed, it is executed after the call is performed.

Sample program

M

The same operation as the drilling canned cycle G81 is created using a custom macro and the machining program makes a modal macro call. For program simplicity, all drilling data is specified using absolute values.



- Calling format

G66 P9110 Zz Rr Ff Ll ;

Z : Coordinates of position Z (absolute programming only) .... (#26)
R : Coordinates of position R (absolute programming only)................. (#18)
F : Cutting feedrate.. (#9)
L : Repetition count

- Program that calls a macro program

```
O0001 ;
G28 G91 X0 Y0 Z0 ;
G92 X0 Y0 Z50.0 ;
G00 G90 X100.0 Y50.0 ;
```

```
G66 P9110 Z-20.0 R5.0 F500 ;
G90 X20.0 Y20.0 ;
X50.0 ;
Y50.0 ;
X70.0 Y80.0 ;
G67 ;
M30 ;
```

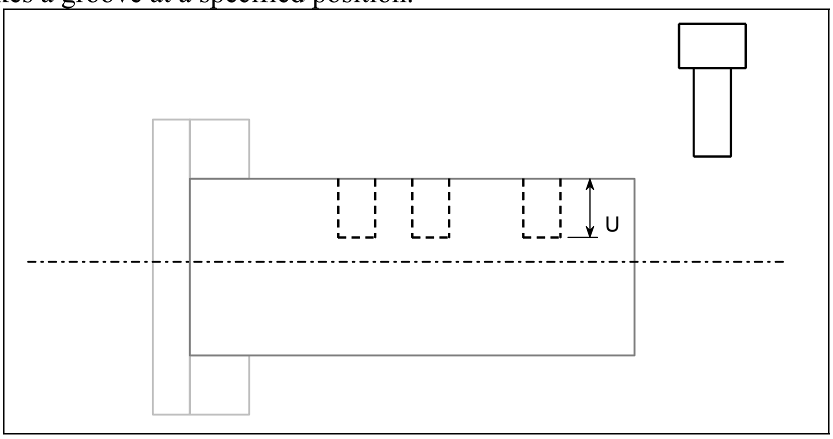
- Macro program (program called)

```
O9110 ;
    #1=#4001 ;  ............................Stores G00/G01.
    #3=#4003 ; ..............................Stores G90/G91.
    #4=#4109 ; ..............................Stores the cutting feedrate.
    #5=#5003 ; ..............................Stores the Z coordinate at the start of drilling.
    G00 G90 Z#18 ;.......................Positioning at position R
    G01 Z#26 F#9 ; .......................Cutting feed to position Z
    IF[#4010 EQ 98]GOTO 1 ; .......Return to position I
    G00 Z#18 ; ..............................Positioning at position R
    GOTO 2 ;
N1 G00 Z#5 ;...................................Positioning at position I
N2 G#1 G#3 F#4 ; ............................Restores modal information.
    M99 ;
```

Sample program

T

This program makes a groove at a specified position.



- Calling format

G66 P9110 Uu Ff ;

U : Groove depth (incremental programming)
F : Cutting feed of grooving

- Program that calls a macro program

```
O0003 ;
G50 X100.0 Z200.0 ;
S1000 M03 ;
G66 P9110 U5.0 F0.5 ;
G00 X60.0 Z80.0 ;
Z50.0 ;
Z30.0 ;
G67 ;
G00 X00.0 Z200.0 M05 ;
```

M30 ;

- Macro program (program called)

O9110 ;
G01 U-#21 F#9 ;...... Cuts the workpiece.
G00 U#21 ;.............. Retracts the tool.
M99 ;

14.7.3 Macro Call Using a G Code

By setting a G code number used to call a macro program in a parameter, the macro program can be called in the same way as for a simple call (G65).

```
O0001 ;                              O9010 ;
    :                                    :
G81 X10.0 Y20.0 Z-10.0 ;                 :
    :                                    :
M30 ;                                N9 M99 ;

Parameter   No.6050=81
```

Explanation

By setting a G code number from -9999 to 9999 used to call a custom macro program (O9010 to O9019) in the corresponding parameter (No.6050 to No.6059), the macro program can be called in the same way as with G65.

If a negative G code is set, a modal call (equivalent to G66) is made.
For example, when a parameter is set so that macro program O9010 can be called with G81, a user-specific cycle created using a custom macro can be called without modifying the machining program.

- Correspondence between parameter numbers and program numbers

Parameter number	Program number
6050	O9010
6051	O9011
6052	O9012
6053	O9013
6054	O9014
6055	O9015
6056	O9016
6057	O9017
6058	O9018
6059	O9019

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitation

- Nesting of calls using G codes

- To call another program in a program called using a G code, only G65, M98, or G66 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using an M code, T code, or specific code can be performed in a program called using a G code.

14.7.4 Macro Call Using a G Code (Specification of Multiple Definitions)

By setting the starting G code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple G codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6040 can be called using as many G codes as the number specified in parameter No. 6040. The numeric value set in parameter No. 6038 indicates the starting G code number and the program number set in parameter No. 6039 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6040.
If parameter No. 6038 is set to a negative G code, a modal call (equivalent to G66) is made.
The number of repetitions and argument specification are set in the same way as with a macro call using a G code.

[Example]

Set parameter No. 6038 to 900, parameter No. 6039 to 1000, and parameter No. 6040 to 100.

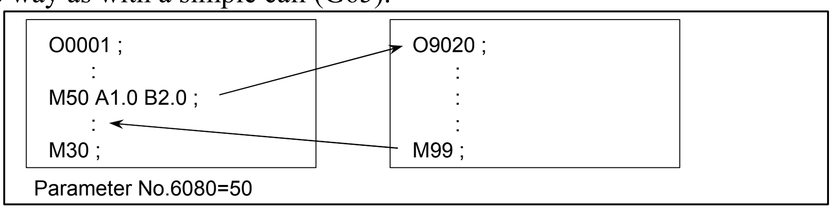
G900 → O1000
G901 → O1001
G902 → O1002
:
G999 → O1099

Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6038 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

> **NOTE**
> 1 The calls defined by this setting become all invalid in the following cases:
> <1> A value outside the valid data range is set in one of the above parameters.
> <2> (Setting of parameter No.6039 + Setting of parameter No.6040 - 1) > 9999
> 2 Simple and modal calls cannot be mixed in the specification.
> 3 If the G code set in parameter Nos. 6050 to 6059 to call the corresponding macro program is within the G code range for calling programs using multiple G codes, the macro program corresponding to the G code set in parameter Nos. 6050 to 6059 is called.

14.7.5 Macro Call Using an M Code

By setting an M code number used to call a macro program in a parameter, the macro program can be called in the same way as with a simple call (G65).



Explanation

By setting an M code number from 3 to 99999999 used to call custom macro program O9020 to O9029 in the corresponding parameter (Nos. 6080 to 6089), the macro program can be called in the same way as with G65.

- Correspondence between parameter numbers and program numbers

Parameter number	Corresponding program number
6080	O9020
6081	O9021
6082	O9022
6083	O9023
6084	O9024
6085	O9025
6086	O9026
6087	O9027
6088	O9028
6089	O9029

Example)

When parameter No. 6080 is set to 990, O9020 is called using M990.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitation

- An M code used to call a macro program must be specified at the start of a block.
- To call another program in a program called using an M code, only G65, M98, or G66 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

14.7.6 Macro Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple M codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6049 can be called using as many M codes as the number specified in parameter No. 6049. The numeric value set in parameter No. 6047 indicates the starting M code number and the program number set in parameter No. 6048 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6049.
The number of repetitions and argument specification are set in the same way as with a macro call using a M code.

[Example]

Set parameter No. 6047 to 90000000, parameter No. 6048 to 4000, and parameter No. 6049 to 100.

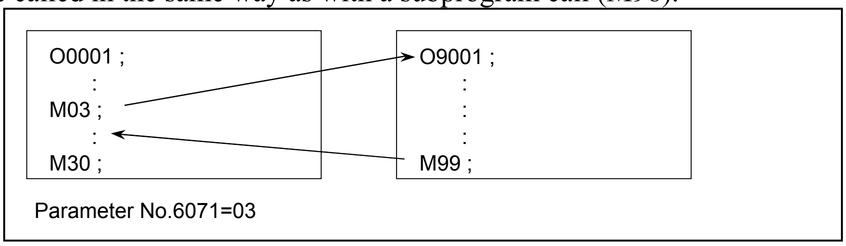
M90000000 → O4000
M90000001 → O4001
M90000002 → O4002
:
M90000099 → O4099

Custom macro calls (simple calls) for 100 combinations are defined as shown above.

> **NOTE**
> 1 The calls defined by this setting become all invalid in the following cases:
> <1> A value outside the valid data range is set in one of the above parameters.
> <2> (Setting of parameter No.6048 + Setting of parameter No.6049 - 1) > 9999
> 2 If the M code set in parameter Nos. 6080 to 6089 to call the corresponding macro program is within the M code range for calling programs using multiple M codes, the macro program corresponding to the M code set in parameter Nos. 6080 to 6089 is called.

14.7.7 Subprogram Call Using an M Code

By setting an M code number used to call a subprogram (macro program) in a parameter, the macro program can be called in the same way as with a subprogram call (M98).



Explanation

By setting an M code number from 3 to 99999999 used to call subprogram O9001 to O9009 in the corresponding parameters Nos. 6071 to 6079, the subprogram can be called in the same way as with M98.

- Correspondence between parameter numbers and program numbers

Parameter number	Program number
6071	O9001
6072	O9002
6073	O9003
6074	O9004
6075	O9005
6076	O9006
6077	O9007
6078	O9008
6079	O9009

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

- M code

An M code in a macro program that has been called is treated as an ordinary M code.

Limitation

- To call another program in a program called using an M code, only G65, M98, or G66 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

14.7.8 Subprogram Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a subprogram, the number of the starting subprogram to be called, and the number of definitions, subprogram calls using multiple M codes can be defined.

Explanation

As many sub programs as the number specified in parameter No. 6046 can be called using as many M codes as the number specified in parameter No. 6046. The numeric value set in parameter No. 6044 indicates the starting M code number and the program number set in parameter No. 6045 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6046.

[Example]

Set parameter No. 6044 to 80000000, parameter No. 6045 to 3000, and parameter No. 6046 to 100.

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

Subprogram calls for 100 combinations are defined as shown above.

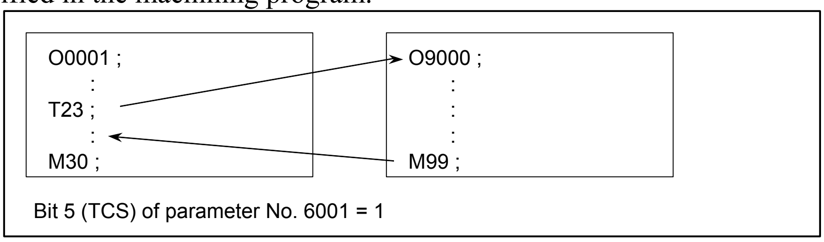
NOTE

1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (Setting of parameter No.6045 + Setting of parameter No.6046 - 1) > 9999

2 If the M code set in parameter Nos. 6071 to 6079 to call the corresponding subprogram is within the M code range for calling subprograms using multiple M codes, the subprogram corresponding to the M code set in parameter Nos. 6071 to 6079 is called.

14.7.9 Subprogram Calls Using a T Code

By enabling subprograms to be called with a T code in a parameter, a subprogram can be called each time the T code is specified in the machining program.



Explanation

- Call

By setting bit 5 (TCS) of parameter No. 6001 to 1, subprogram O9000 can be called each time a T code is specified in a machining program. A T code specified in a machining program is assigned to common variable #149.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

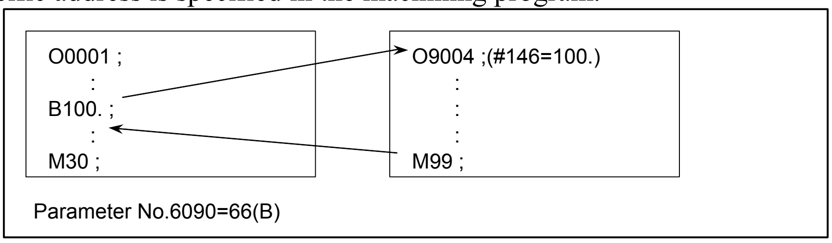
Argument specification is not allowed.

Limitation

- To call another program in a program called using a T code, only G65, M98, or G66 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a T code.

14.7.10 Subprogram Call Using a Specific Address

By enabling subprograms to be called with a specific address in a parameter, a subprogram can be called each time the specific address is specified in the machining program.



Explanation

- Call

By setting the code (ASCII code converted to decimal) corresponding to a specific address in parameter No. 6090 or No. 6091, the custom macro program, O9004 or O9005, corresponding to each parameter can be called when the specific address is specified in a machining program. The code value corresponding to a specific address specified in a machining program is assigned to the common variables (#146, #147). The table below indicates the addresses that can be set.

M

Address	Parameter setting
A	65
B	66
D	68
F	70
H	72
I	73
J	74
K	75
L	76
M	77
P	80
Q	81

Address	Parameter setting
R	82
S	83
T	84
V	86
X	88
Y	89
Z	90

> **NOTE**
> When address L is set, the number of repetitions cannot be set.

T

Address	Parameter setting
A	65
B	66
F	70
H	72
I	73
J	74
K	75
L	76
M	77
P	80
Q	81
R	82
S	83
T	84

> **NOTE**
> When address L is set, the number of repetitions cannot be set.

- Correspondence between parameter numbers and program numbers and between the parameter numbers and common variables

Parameter number	Program number	Common variable
6090 6091	O9004 O9005	#146 #147

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

Limitation

- To call another program in a program called using a specific address, only G65, M98, or G66 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a specific address.

Sample program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tools T01 to T05 is measured.
  No measurement is made for tools with numbers greater than T05.
- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

- Parameter setting

Set 3 in parameter No.6071, and set 5 in parameter No.6072.

- Variable value setting

Set 0 in variables #501 to #505.

- Program that calls a macro program

```
O0001 ;
T01 M06 ;
M03 ;
    :
M05 ; ..................Changes #501.
T02 M06 ;
M03 ;
    :
M05 ; ..................Changes #502.
T03 M06 ;
M03 ;
    :
M05 ; ..................Changes #503.
T04 M06 ;
M03 ;
    :
M05 ; ..................Changes #504.
T05 M06 ;
M03 ;
    :
M05 ; ..................Changes #505.
M30 ;
```

- Macro program (program called)

```
O9001(M03) ; ........................................................ Macro to start counting
    M01 ;
    IF[#4120 EQ 0]GOTO 9 ; ............................... No tool specified
    IF[#4120 GT 5]GOTO 9 ; ............................... Out-of-range tool number
```

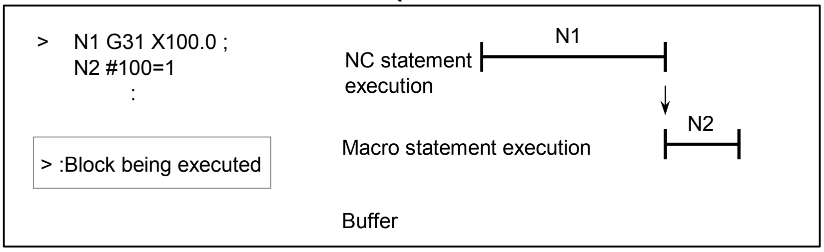
```
        #3002=0 ; ...................................... Clears the timer.
N9 M03 ;................................................ Rotates the spindle in the
        M99 ; .......................................... forward direction.

O9002(M05) ; ........................................... Macro to end counting
        M01 ;
        IF[#4120 EQ 0]GOTO 9 ; ......................... No tool specified
        IF[#4120 GT 5]GOTO 9 ; ......................... Out-of-range tool number
        #[500+#4120]=#3002+#[500+#4120] ;............... Calculates cumulative time.

N9 M05 ;................................................ Stops the spindle.
        M99 ;
```

14.8 PROCESSING MACRO STATEMENTS

For smooth machining, the CNC prereads the NC statement to be performed next. This operation is referred to as buffering. For example, many NC statements are buffered during look-ahead by AI advanced preview control (M series) / AI contour control (M series).
In the cutter compensation mode (G41 or G42) for M series, the CNC prereads the NC statements at least three blocks ahead to find intersections even if look-ahead by AI contour control and so forth is not applied.
Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.
At the blocks containing M00, M01, M02 or M30, blocks containing M-codes for which buffering is suppressed by setting parameters Nos. 3411 to 3420 and No.3421 to 3432, and blocks containing prevention buffering G codes such as G31 or G53, the CNC stops to preread the NC statement after that. Then, the stop of the macro statement execution is guaranteed until such M codes or G codes complete its execution.

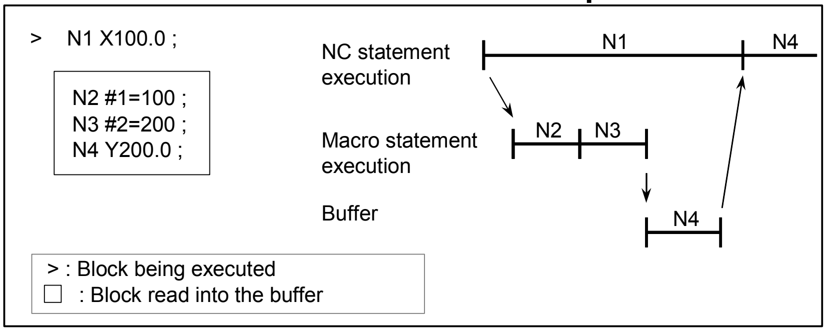
Explanation

- When the next block is not buffered (M codes that are not buffered, G31, etc.)



> ⚠**CAUTION**
> In case that you need to execute the macro statement after completing the block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, offset value, etc., it may different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

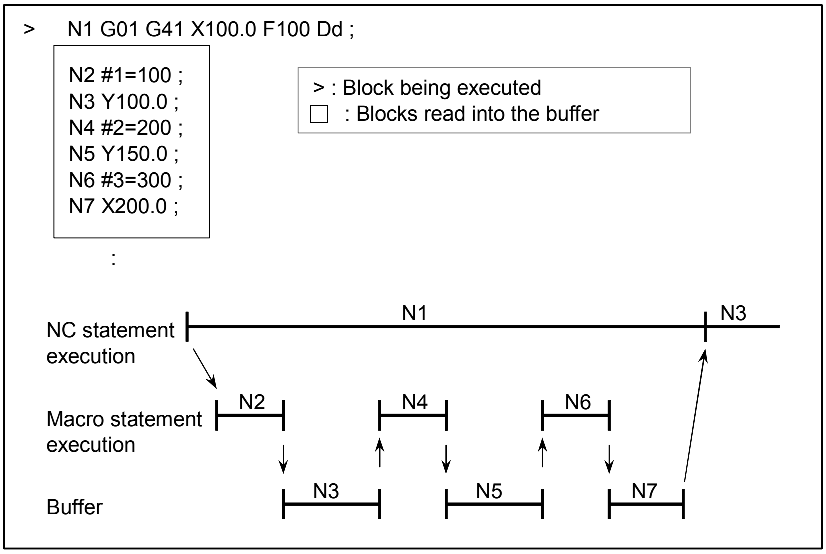
- Buffering the next block in other than cutter compensation mode (G41, G42)



When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- In cutter compensation mode (G41, G42)

M



When N1 is being executed, the NC statements in the next three blocks (up to N7) are read into the buffer. The macro statements (N1, N4, N6) between N1 and N7 are processed during execution of N1.

14.9 REGISTERING CUSTOM MACRO PROGRAMS

Custom macro programs are similar to subprograms. They can be registered and edited in the same way as subprograms. The storage capacity is determined by the total length of tape used to store both custom macros and subprograms.

14.10 CODES AND RESERVED WORDS USED IN CUSTOM MACROS

In addition to the codes used in ordinary programs, the following codes are used in custom macro programs.

Explanation

- Codes

(1) When the ISO code is used or when bit 4 (ISO) of parameter No. 6008 is set to 0
(The codes are represented in hexadecimal.)

Meaning	Code
*	0AAh
=	0BDh
#	0A3h
[	0DBh
]	0DDh
?	03Fh
@	0C0h
&	0A6h
_	05Fh
O	0CFh

(2) When the EIA code is used or when the ISO code is used with bit 4 (ISO) of parameter No. 6008 set to 1

Meaning	Code
*	Code set in parameter No. 6010
=	Code set in parameter No. 6011
#	Code set in parameter No. 6012
[	Code set in parameter No. 6013
]	Code set in parameter No. 6014
?	Code set in parameter No. 6015
@	Code set in parameter No. 6016
&	Code set in parameter No. 6017
_	Code set in parameter No. 6018

For O, the same code as for O indicating a program number is used. Set a hole pattern for each of *, =, #, [,], ?, @, &, and _ in the ISO or EIA code in the corresponding parameter (Nos. 6010 to 6018).
The code 00h cannot be used. The code indicating an alphabetic character can be used for the code indicating a character listed above, but the code can be no longer used to indicate the original character.

- Reserved words

The following reserved words are used in custom macros:
AND, OR, XOR, MOD, EQ, NE, GT, LT, GE, LE, SIN, COS, TAN, ASIN, ACOS, ATAN, ATN, SQRT, SQR, ABS, BIN, BCD, ROUND, RND, FIX, FUP, LN, EXP, POW, ADP, IF, GOTO, WHILE, DO, END, BPRNT, DPRNT, POPEN, PCLOS, SETVN
System variable (constant) names and registered common variable names are also used as reserved words.

14.11 EXTERNAL OUTPUT COMMANDS

In addition to the standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- BPRNT
- DPRNT
- POPEN
- PCLOS

These commands are provided to output variable values and characters through the reader/puncher interface.

Explanation

Specify these commands in the following order:

Open command: POPEN
: Before specifying a sequence of data output commands, specify this command to establish a connection to an external input/output device.

Data output command: BPRNT or DPRNT
: Specify necessary data output.

Close command: PCLOS
: When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

- Open command POPEN

The POPEN command establishes a connection to an external input/output device. It must be specified before a sequence of data output commands. The CNC outputs a DC2 control code.

- Data output command BPRNT

The BPRNT command outputs characters and variable values in binary.

BPRNT [a #b [c] …]

- c: Number of significant decimal places
- #b: Variable
- a: Character

(i) Specified characters are converted to the codes according to the setting data (ISO) that is output at that time.
Specifiable characters are as follows:
- Letters (A to Z)
- Numbers
- Special characters (*, /, +, -, ?, @, &, _)

> **NOTE**
> 1 An asterisk (*) is output by a space code.
> 2 When using ?, @, &, and/or _, use the ISO code as the punch code (setting data (ISO) = 1).

(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2-word (32-bit) data, including the decimal digits. It is output as binary data starting from the highest byte.

(iii) When specified data has been output, an EOB code is output according to the setting code (ISO).

(iv) <Null> variables are regarded as 0.

Example

BPRNT [C** X#100 [3] Y#101 [3] M#10 [0]]

Variable value

#100=0.40956

#101=-1638.4

#10=12.34

are output as follows:

```
C3 A0 A0 D8 00 00 01 9A 59 FF E7 00 00 4D 00 00 00 0C 0A
↓                                                      ↓
C   sp sp   X0000019A    YFFE70000     M0000000C       LF
    ( * * )  (410)       (-1638400)    (12)            ( ; )
```

- Data output command DPRNT

DPRNT [a #b [c d] …]

- a: Character
- #b: Variable
- c: Number of significant digits in the integer part
- d: Number of significant decimal places

The DPRNT command outputs characters and each digit in the value of a variable according to the code set in the settings (ISO).

(i) For an explanation of the DPRNT command, see Items (i), (iii), and (iv) for the BPRNT command.

(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.
For the value of a variable, as many codes as the specified number of digits are output according to the settings one by one, starting with the highest digit. The decimal point is also output using the set code.
Each variable must be a numeric value consisting of up to nine digits. When high-order digits are zeros, these zeros are not output if bit 1 (PRT) of parameter No. 6001 is 1. If parameter PRT is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.
When bit 1 (PRT) of parameter No. 6001 is 0, a space code is output to indicate a positive number instead of +; if parameter PRT is 1, no code is output.

Example
DPRNT [X#2 [53] Y#5 [53] T#30 [20]]
Variable value
#2=128.47398
#5=-91.2
#30=123.456
are output as follows:

(1) Parameter PRT (No.6001#1) = 0

```
D8  A0 A0 A0  B1 B2 B8 2E B4 B7 B4  59 2D  A0 A0 A0  39 B1 2E B2 30 30  D4  A0  B2 33  0A
X   sp sp sp  128.474               Y-     sp sp sp  91.200             T   sp  023    LF
```

(2) Parameter PRT (No.6001#1) = 1

```
D8 B1 B2 B8 2E B4 B7 B4  59 2D 39 B1 2E B2 30 30  D4 A0 B2 33  0A
X128.474                 Y-91.200                 T023         LF
```

- Close command PCLOS

The PCLOS command releases a connection to an external input/output device. Specify this command when all data output commands have terminated. DC4 control code is output from the CNC.

- Required setting

Specify the specification number of an input/output device to be used for the I/O device specification number.
According to the above settings, set data items (such as the baud rate) for the reader/puncher interface.
Do not specify the FANUC Cassette or FLOPPY unit as an external output device.
When specifying a DPRNT command to output data, specify whether leading zeros are output as spaces (by setting bit 1 (PRT) of parameter No. 6001 to 1 or 0).
To indicate the end of a line of data in ISO code, specify whether to use only an LF (bit 4 (CRO) of parameter 6001 is 0) or an LF and CR bit 4 (CRO) of parameter 6001 is 1).

NOTE

1 It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.
2 Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.
3 When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. If a reset is caused by M30 or other commands at the end of a program that is outputting data, wait until all data is output by, for example, specifying the close command at the end of the program and then execute M30 or other commands.

14.12 RESTRICTIONS

- Single block

Even while a macro program is being executed, blocks can be stopped in the single block mode.
A block containing a macro call command (G65, G66, Ggg, Mmm, or G67) does not stop even when the single block mode is on.
Whether blocks containing arithmetic and logic operation commands and control commands are stopped depends on the settings of bits 5 (SBM) and 7 (SBV) of parameter No. 6000 as shown in the following table.

<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Bit 5 (SBM) of parameter No. 6000</th></tr>
<tr><th>0</th><th>1</th></tr>
<tr><td rowspan="2">Bit 7 (SBV) of parameter No. 6000</td><td>0</td><td>Not stopped when the single block mode is on.</td><td rowspan="2">Can be stopped in the single block mode. (Variable #3003 cannot be used to disable single block stop. Single block stop is always enabled.)</td></tr>
<tr><td>1</td><td>Can be stopped in the single bock mode. (Variable #3003 can be used to enable or disable single block stop.)</td></tr>
</table>

M

Note that when a single block stop occurs at a macro statement in cutter compensation mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)

- Optional block skip

A / appearing in the middle of an <expression> (enclosed in brackets [ ] on the right-hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.

- Operation in EDIT mode

By setting bit 0 (NE8) of parameter No.3202 and bit 4 (NE9) of parameter No.3202 to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. This prevents registered custom macro programs and subprograms from being destroyed by accident. When the entire memory is cleared, the contents of memory such as custom macro programs are deleted.

- Reset

With a reset operation, local variables and common variables #100 to #199 are cleared to null values. However, bit 6 (CCV) of parameter No. 6001 can be set to prevent variables #100 to #199 from being cleared.
A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- Display of the PROGRAM RESTART

As with M98, the M and T codes used for subprogram calls are not displayed.

- Feed hold

When a feed hold is enabled during execution of a macro statement, the machine stops after execution of the macro statement. The machine also stops when a reset or alarm occurs.

- DNC operation

The control commands (such as GOTO and WHILE-DO) cannot be executed during DNC operation.

However, this restriction is removed when a program registered in program memory is called during DNC operation.

- Constant values that can be used in <expression>

+0.00000000001 to +999999999999
-999999999999 to -0.00000000001
The number of significant digits is 12 (decimal).
If this range is exceeded, an alarm PS0012 occurs.

14.13 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. Program an interrupt command in the following format:

Format

M96Pxxxx ;	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.

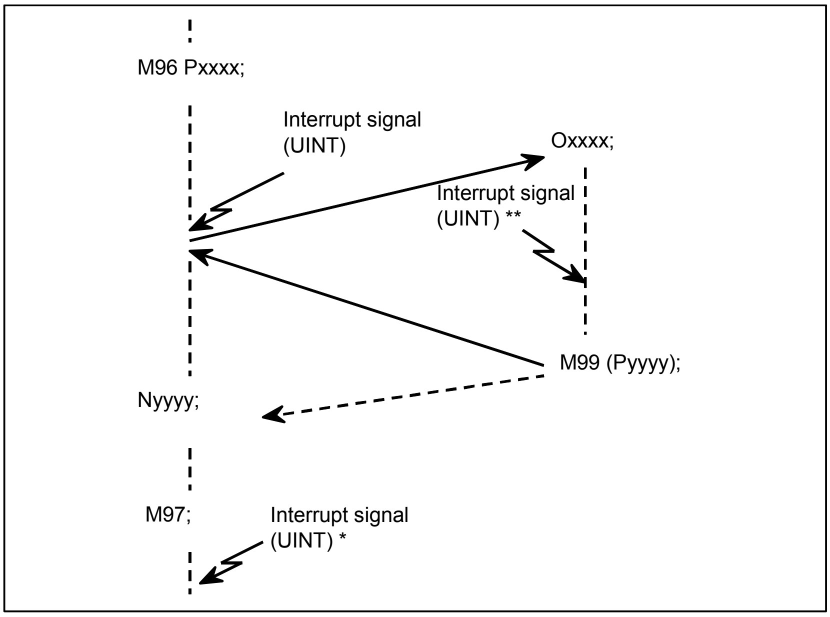


Fig 14.13 (a) Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx. When the interrupt signal

((UINT)** and (UINT)* in Fig 14.13 (a)) is input during execution of the interrupt program or after M97, it is ignored.

14.13.1 Specification Method

Explanation

- Interrupt conditions

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

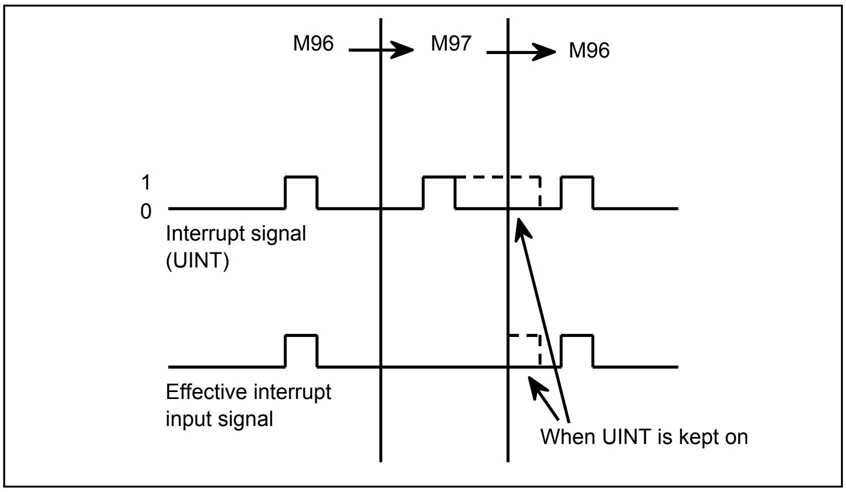
- When memory operation, DNC operation, or MDI operation is selected
- When STL (start lamp) is on
- When a custom macro interrupt is not currently being processed

A custom macro interrupt cannot be performed during manual operation.

- Specification

Generally, the custom macro interrupt function is used by specifying M96 to enable the interrupt signal (UINT) and M97 to disable the signal.

Once M96 is specified, a custom macro interrupt can be initiated by the input of the interrupt signal (UINT) until M97 is specified or the CNC is reset. After M97 is specified or the CNC is reset, no custom macro interrupts are initiated even when the interrupt signal (UINT) is input. The interrupt signal (UINT) is ignored until another M96 command is specified.



The interrupt signal (UINT) becomes valid after M96 is specified. Even when the signal is input in M97 mode, it is ignored. When the signal input in M97 mode is kept on until M96 is specified, a custom macro interrupt is initiated as soon as M96 is specified (only when the status-triggered scheme is employed); when the edge-triggered scheme is employed, the custom macro interrupt is not initiated even when M96 is specified.

NOTE

For the status-triggered and edge-triggered schemes, see Item "Custom macro interrupt signal (UINT)" of II-14.12.2.

14.13.2 Details of Functions

Explanation

- Subprogram-type interrupt and macro-type interrupt

There are two types of custom macro interrupts: Subprogram-type interrupts and macro-type interrupts. The interrupt type used is selected by bit 5 (MSB) of parameter No.6003.

(a) Subprogram-type interrupt: When bit 5 (MSB) of parameter No.6003 is set to 1
An interrupt program is called as a subprogram.
This means that the levels of local variables remain unchanged before and after the interrupt.
This interrupt is not included in the nesting level of subprogram calls.

(b) Macro-type interrupt: When bit 5 (MSB) of parameter No.6003 is set to 0
An interrupt program is called as a custom macro.
This means that the levels of local variables change before and after the interrupt.
The interrupt is not included in the nesting level of custom macro calls.

When a subprogram call or a custom macro call is performed within the interrupt program, this call is included in the nesting level of subprogram calls or custom macro calls.
Arguments cannot be passed from the current program even when the custom macro interrupt is a macro-type interrupt. The local variables immediately after interruption are all cleared to null.

- M codes for custom macro interrupt control

In general, custom macro interrupts are controlled by M96 and M97. However, these M codes, may already being used for other purposes (such as an M function or M code for macro call) by some machine tool builders.
For this reason, bit 4 (MPR) of parameter No.6003 is provided to set M codes for custom macro interrupt control.
When specifying this parameter to use the custom macro interrupt control M codes set by parameters, set parameters Nos.6033 and 6034 as follows:
Set the M code to enable custom macro interrupts in parameters Nos.6033, and set the M code to disable custom macro interrupts in parameter 6034.
When specifying that parameter-set M codes are not used, M96 and M97 are used as the custom macro control M codes regardless of the settings of parameters Nos.6033 and 6034.
The M codes used for custom macro interrupt control are processed internally (they are not output to external units). However, in terms of program compatibility, it is undesirable to use M codes other than M96 and M97 to control custom macro interrupts.

- Custom macro interrupts and NC statements

When performing a custom macro interrupt, the user may want to interrupt the NC statement being executed, or the user may not want to perform the interrupt until the execution of the current block is completed. Bit 2 (MIN) of parameter No.6003)is used to select whether to perform interrupts even in the middle of a block or to wait until the end of the block. The type of interrupt performed even in the middle of a block is called type I and the type of interrupt performed at the end of the block is called type II.

⚠ **CAUTION**

For interrupt type I, operation after control is returned differs depending on whether the interrupt program contains an NC statement.
When the program number block contains EOB (;), it is assumed to contain an NC statement.

(Program containing an NC statement)	(Program containing no NC statement)
O0013 ;	O0013#101=#5041 ;
#101=#5041 ;	#102=#5042 ;
#102=#5042 ;	#103=#5043 ;
#103=#5043 ;	M99 ;
M99 ;	

Type I (when an interrupt is performed even in the middle of a block)

(i) When the interrupt signal (UINT) is input, any movement or dwell being performed is stopped immediately and the interrupt program is executed.

(ii) If there are NC statements in the interrupt program, the command in the interrupted block is lost and the NC statement in the interrupt program is executed. When control is returned to the interrupted program, the program is restarted from the next block after the interrupted block.

(iii) If there are no NC statements in the interrupt program, control is returned to the interrupted program by M99, then the program is restarted from the command in the interrupted block.

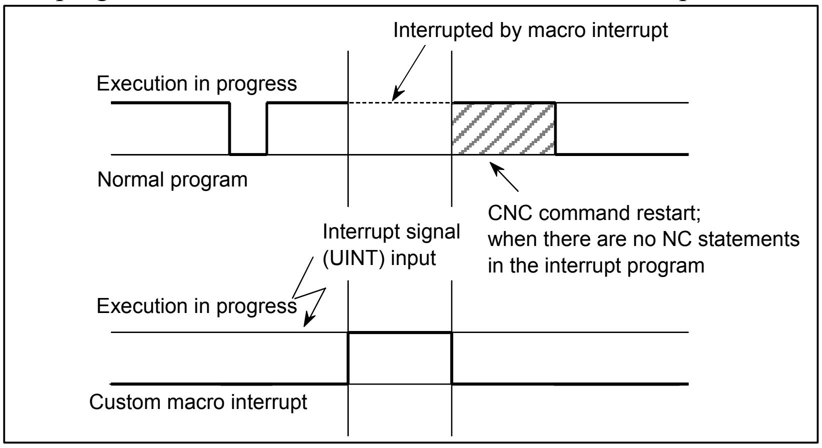


Fig. 14.13 (b) Custom macro interrupt and NC command (type I)

Type II (when an interrupt is performed at the end of the block)

(i) If the block being executed is not a block that consists of several cycle operations such as a drilling canned cycle and automatic reference position return (G28), an interrupt is performed as follows:
When an interrupt signal (UINT) is input, macro statements in the interrupt program are executed immediately unless an NC statement is encountered in the interrupt program. NC statements are not executed until the current block is completed.

(ii) If the block being executed consists of several cycle operations, an interrupt is performed as follows:
When the last movement in the cycle operations is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after all cycle operations are completed.

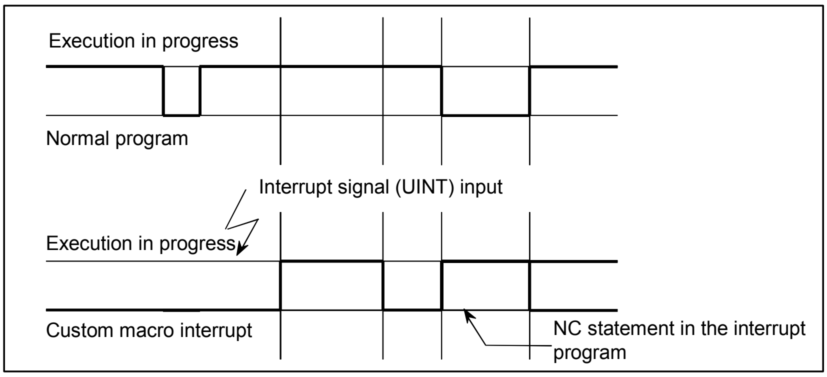


Fig. 14.13 (c) Custom macro interrupt and NC command (type II)

M

NOTE

During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:

<1> Automatic reference position return
<2> Cutter compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
<3> Canned cycle
<4> Automatic tool length measurement
<5> Normal direction control

T

NOTE

During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:

<1> Automatic reference position return
<2> Tool nose radius compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
<3> Canned cycle (No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle, however.)
<4> Automatic tool compensation
<5> Chamfering/corner R

- Conditions for enabling and disabling the custom macro interrupt signal

The interrupt signal becomes valid after execution starts of a block that contains M96 for enabling custom macro interrupts. The signal becomes invalid when execution starts of a block that contains M97.

While an interrupt program is being executed, the interrupt signal becomes invalid. The signal become valid when the execution of the block that immediately follows the interrupted block in the main program is started after control returns from the interrupt program. In type I, if the interrupt program consists of only macro statements, the interrupt signal becomes valid when execution of the interrupted block is started after control returns from the interrupt program.

- Custom macro interrupt signal (UINT)

There are two schemes for custom macro interrupt signal (UINT) input: The status-triggered scheme and edge-triggered scheme. When the status-triggered scheme is used, the signal is valid when it is on. When the edge triggered scheme is used, the signal becomes valid on the rising edge when it switches from off to on status.
One of the two schemes is selected with bit 3 (TSE) of parameter No.6003.

When the status-triggered scheme is selected by this parameter, a custom macro interrupt is generated if the interrupt signal (UINT) is on at the time the signal becomes valid. By keeping the interrupt signal (UINT) on, the interrupt program can be executed repeatedly.

When the edge-triggered scheme is selected, the interrupt signal (UINT) becomes valid only on its rising edge. Therefore, the interrupt program is executed only momentarily (in cases when the program consists of only macro statements). When the status-triggered scheme is inappropriate, or when a custom macro interrupt is to be performed just once for the entire program (in this case, the interrupt signal may be kept on), the edge-triggered scheme is useful.

Except for the specific applications mentioned above, use of either scheme results in the same effects. The time from signal input until a custom macro interrupt is executed does not vary between the two schemes.

In the example shown in Fig 14.12 (d), an interrupt is executed four times when the status-triggered scheme is used; when the edge-triggered scheme is used, the interrupt is executed just once.

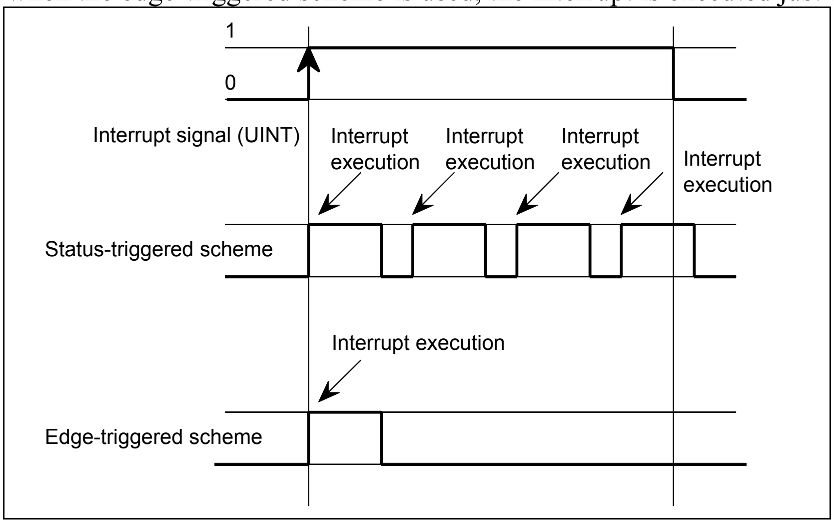


Fig. 14.13 (d) Custom macro interrupt signal

- Return from a custom macro interrupt

To return control from a custom macro interrupt to the interrupted program, specify M99. A sequence number in the interrupted program can also be specified using address P. If this is specified, the program is searched from the beginning for the specified sequence number. Control is returned to the first sequence number found.

NOTE

If a block containing M99 is alone or has address O, N, P, L, or M only, this block is programmatically assumed to be the same as the previous block. Therefore, a single-block stop does not occur for this block. In terms of programming, the following <1> and <2> are basically the same. (The difference is whether Gxx is executed before M99 is recognized.)

```
<1>        Gxx Xxxx ;
           M99 ;
<2>        Gxx Xxxx M99 ;
```

- Custom macro interrupt and modal information

A custom macro interrupt is different from a normal program call. It is initiated by an interrupt signal (UINT) during program execution. In general, any modifications of modal information made by the interrupt program should not affect the interrupted program.
For this reason, even when modal information is modified by the interrupt program, the modal information before the interrupt is restored when control is returned to the interrupted program by M99.

When control is returned from the interrupt program to the interrupted program by M99 Pyyyy, however, modal information can again be controlled by the program. In this case, the new continuous information modified by the interrupt program is passed to the interrupted program.

In this case, take the following action as required:
<1> The interrupt program provides modal information to be used after control is returned to the interrupted program.
<2> After control is returned to the interrupted program, modal information is specified again as necessary.

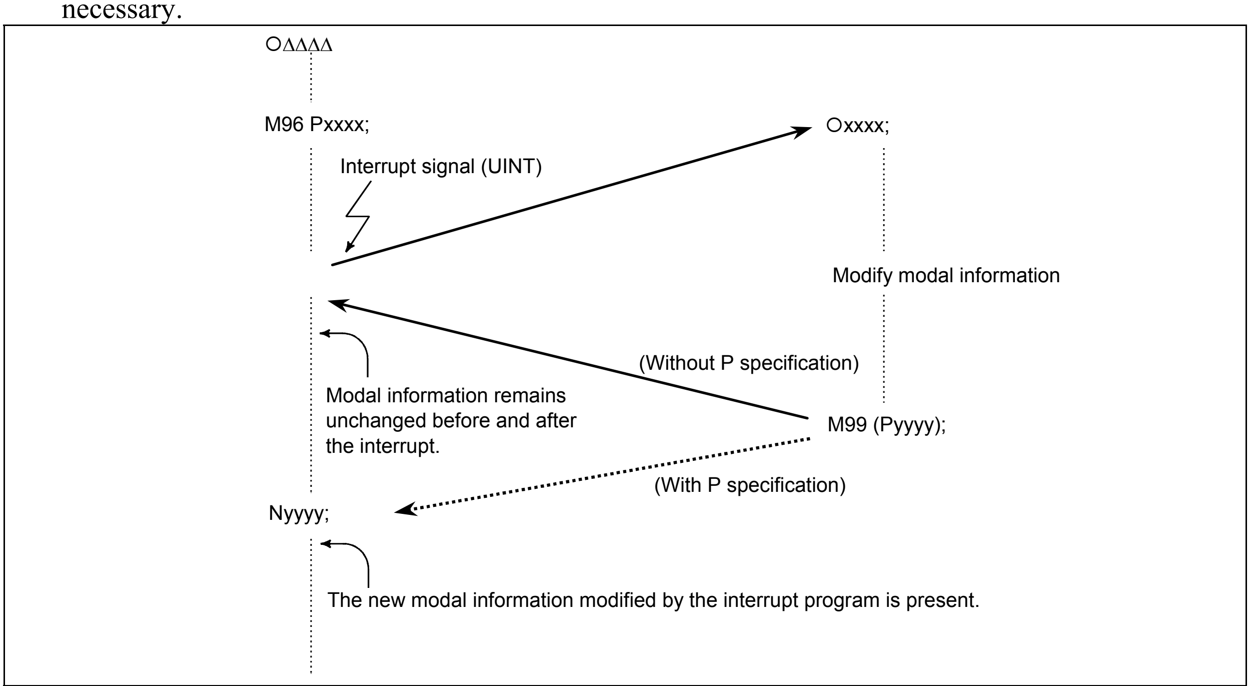


Fig. 14.13 (e) Custom macro interrupt and modal information

Modal information when control is returned by M99

The modal information present before the interrupt becomes valid. The new modal information modified by the interrupt program is made invalid.

Modal information when control is returned by M99 Pyyyy
The new modal information modified by the interrupt program remains valid even after control is returned.

Modal information which was valid in the interrupted block
The old modal information which was valid in the interrupted block can be read using custom macro system variables #4401 to #4530.

M

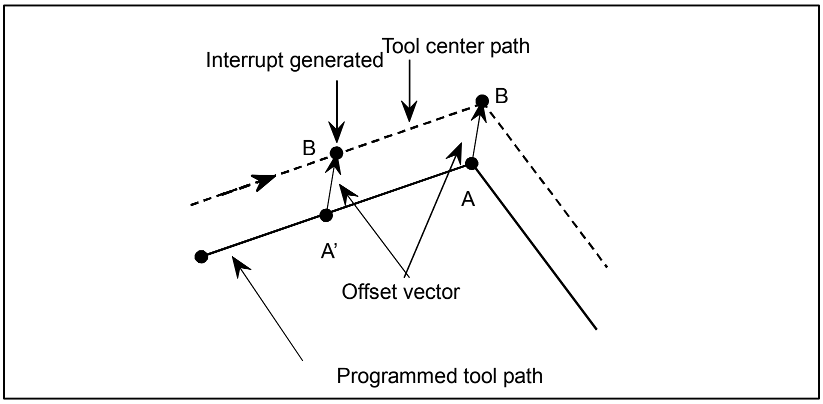
System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4430	G code (group 30)
#4502	B code
#4507	D code
#4508	E code
#4509	F code
#4511	H code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code
#4530	Additional workpiece coordinate system number

T

System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4430	G code (group 30)
#4508	E code
#4509	F code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code

- System variables (position information values) for the interrupt program
Position information can be read as follows.

Macro variable	Condition	Position information value
#5001 or above	Until the first NC statement appears	Coordinates of point A
	After an NC statement with no move command appears	Coordinates of point A'
	After an NC statement with a move command appears	Coordinates of the end point of the move command
#5021 or above		Machine coordinates of point B'
#5041 or above		Workpiece coordinates of point B'



- Custom macro interrupt and custom macro modal call

When the interrupt signal (UINT) is input and an interrupt program is called, the custom macro modal call is canceled (G67). However, when G66 is specified in the interrupt program, the custom macro modal call becomes valid. When control is returned from the interrupt program by M99, the modal call is restored to the state it was in before the interrupt was generated. When control is returned by M99 Pyyyy ;, the modal call in the interrupt program remains valid.

- Custom macro interrupt and program restart

In program restart, when the interrupt signal (UINT) is input during dry run recovery after a search, the interrupt program is called after restart of all axes is completed.
That is, interrupt type II is assumed regardless of the parameter setting.

M

NOTE

1 Alarm PS1101 occurs in the following cases:
<1>An interrupt is generated in the programmable mirror image (G51.1) mode and another G51.1 is specified in the interrupt program.
<2>An interrupt is generated in the coordinate system rotation (G68) mode and another G68 is specified in the interrupt program.
<3>An interrupt is generated in the scaling (G51) mode and another G51 is specified in the interrupt program.
2 In program restart, do not input the interrupt signal (UINT) during dry run recovery after a search.

T

NOTE

1 No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle.
2 In program restart, do not input the interrupt signal (UINT) during dry run recovery after a search.

15 PROGRAMMABLE PARAMETER INPUT (G10)

Overview

The values of parameters and pitch error compensation data can be entered in a program. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.

Format

- Parameter entry mode

```
G10 L52 ;            Parameter entry mode setting
N_ (Q_) R_ ;         For parameters other than the axis type or spindle type
N_ P_ (Q_) R_ ;      For axis type or spindle type parameters
  :
G11 ;                Parameter entry mode cancel
```

N_ : Parameter number
R_ : Parameter setting value (Leading zeros can be omitted.)
(Q_) : Bit number 0 to 7 (to be set when a bit type parameter is input)
(Enabled when bit 4 (G1B) of parameter No.3454 = 1.)
P_ : Axis number 1 to maximum controlled axis number (to be specified when an axis type parameter or spindle type parameter is specified)

NOTE
G10L52 cannot be used to enter pitch error compensation data.

- Pitch error compensation data entry mode

```
G10 L50 ;   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

NOTE
G10L50 cannot be used to enter parameter.

Explanation

- Setting value (R_)

Do not use a decimal point in the setting (R_) of a parameter or pitch error compensation data.
To change a bit type parameter if bit 4 (G1B) of parameter No. 3454 is 1, specify 0 or 1. If a value other than 0 and 1 is specified, alarm PS1144, "G10 FORMAT ERROR", is issued.
As the value of R, a custom macro variable can be used.
When a parameter of real type is used, set an integer value in (R_) according to the increment system of the parameter.

- Bit number (Q__)

Bit number (Q_) is effective if bit 4 (G1B) of parameter No. 3454 is 1. To set a bit type parameter, set a number in the range of 0 to 7.
A custom macro variable can be used as the value of Q.

- Axis number (P_)

As the axis number (P_), specify the order of a controlled axis to be displayed on the CNC display screen, by using an axis type parameter.
For example, specify P2 for the control axis which is displayed second.
For a spindle type also, specify the order of an axis to be display on the CNC display screen.
A custom macro variable can be used as the value of P.

> ⚠ **WARNING**
> 1 Do not fail to perform reference position return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
> 2 The canned cycle mode must be cancelled before entering of parameters. When not cancelled, the drilling motion may be activated.

> ⚠ **CAUTION**
> Compatibility with the Series 0*i*-C:
> This model has parameters that are not compatible with the Series 0*i*-C.
> So, before using this function, make a check according to the Parameter Manual (B-64310EN) of this model.

> **NOTE**
> Other NC statements cannot be specified while in parameter input mode.

Example

1. Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 0)

G10 L52 ;	Parameter entry mode
N3404 R00000100 ;	SBP setting
G11 ;	Cancel parameter entry mode

2. Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 1)

G10 L52 ;	Parameter entry mode
N3404 Q2 R1 ;	SBP setting
G11 ;	Cancel parameter entry mode

3. Change the values for the Z-axis (3rd axis) and A-axis (4th axis) in axis type parameter No. 1322 (the coordinates of stored stroke limit 2 in the positive direction for each axis).
(When the increment systems for the 3rd and 4th axes are IS-B and millimeter machine, respectively)

G10 L52 ;	Parameter entry mode
N1322 P3 R4500 ;	Change the value for the Z-axis to 4.500
N1322 P4 R12000 ;	Change the value for the A-axis to 12.000
G11 ;	Cancel parameter entry mode

4. Change compensation point numbers 10 and 20 of pitch error compensation.

G10 L50 ;	Pitch error compensation data entry mode
N10010 R1 ;	Change the compensation point number from 10 to 1
N10020 R5 ;	Change the compensation point number from 20 to 5
G11 ;	Pitch error compensation data entry mode

16 HIGH-SPEED CUTTING FUNCTIONS

Chapter 16, "HIGH-SPEED CUTTING FUNCTIONS", consists of the following sections:

16.1 ADVANCED PREVIEW CONTROL (T SERIES) / AI ADVANCED PREVIEW CONTROL (M SERIES) / AI CONTOUR CONTROL (II) (M SERIES)

Overview

Advanced preview control (T series), AI advanced preview control (M series), and AI contour control (II) (M series) are intended for high-speed, high-precision machining. The use of these functions suppresses the acceleration/deceleration delay that tends to increase as the feedrate becomes faster, as well as the delay in the servo system, reducing the machining profile error.

The following table shows the functions included in these functions.

	APC	AI APC		AICC	AICC II
Model	0*i*-TD	0*i* Mate-MD	0*i*-MD	0*i*-MD	0*i*-MD
Basic/Option	Option	Basic		Option	Option
Look-ahead block count	1	12	20	40	200
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 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	–	○		○	○
Smooth speed control	–	–		–	○
Speed control with cutting load	–	–		–	○
Disregard of feedrate command	–	–		–	○
Jerk control - Speed control with change of acceleration on each Axis - Look-ahead smooth bell-shaped acceleration/deceleration before Interpolation	–	–		–	☆
Nano smoothing	–	–		–	☆

APC : Advanced preview control

AI APC : AI advanced preview control
AICC : AI contour control
AICC II : AI contour control II

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

M

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

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 (II)

G05.1 Q_ ;
Q1 : AI advanced preview control mode/AI contour control (II) mode on
Q0 : AI advanced preview control mode/AI contour control (II) mode off

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

Explanation

- Look-ahead acceleration/deceleration before interpolation

T

Acceleration/deceleration type of look-ahead acceleration/ deceleration before interpolation function is look-ahead linear acceleration/deceleration before interpolation function.

M

There are two look-ahead acceleration/deceleration before interpolation functions - look-ahead linear acceleration/deceleration before interpolation and look-ahead bell-shaped acceleration/ deceleration before interpolation. Look-ahead bell-shaped acceleration/deceleration before interpolation offers more smooth acceleration and deceleration.

* Look-ahead bell-shaped acceleration/deceleration before interpolation is an optional function.

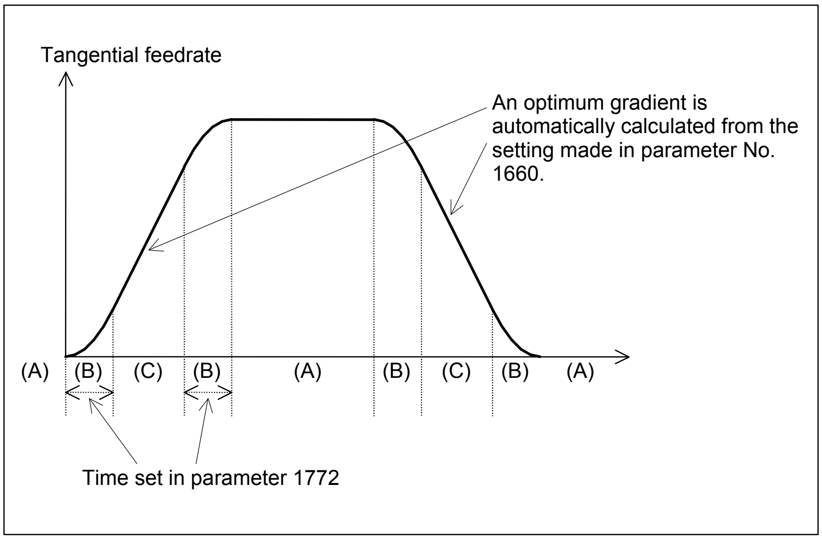
- Setting an acceleration

T

A permissible acceleration for the linear acceleration/deceleration of each axis is set in parameter No. 1660. The acceleration/deceleration is performed with the maximum tangential acceleration not exceeding the permissible acceleration of each axis specified in parameter No. 1660.

M

A permissible acceleration for the linear acceleration/deceleration of each axis is set in parameter No. 1660. For bell-shaped acceleration/deceleration, acceleration change time (B) (period of transition from constant speed state (A) to constant acceleration/ deceleration state (C)) is set in parameter No. 1772. In the constant acceleration/deceleration state (C), acceleration/deceleration is performed with the maximum tangential acceleration not exceeding the permissible acceleration of each axis specified in parameter No. 1660.
The acceleration change time specified in parameter No. 1772 is held constant, regardless of the tangential acceleration.



- Method of determining the tangent acceleration

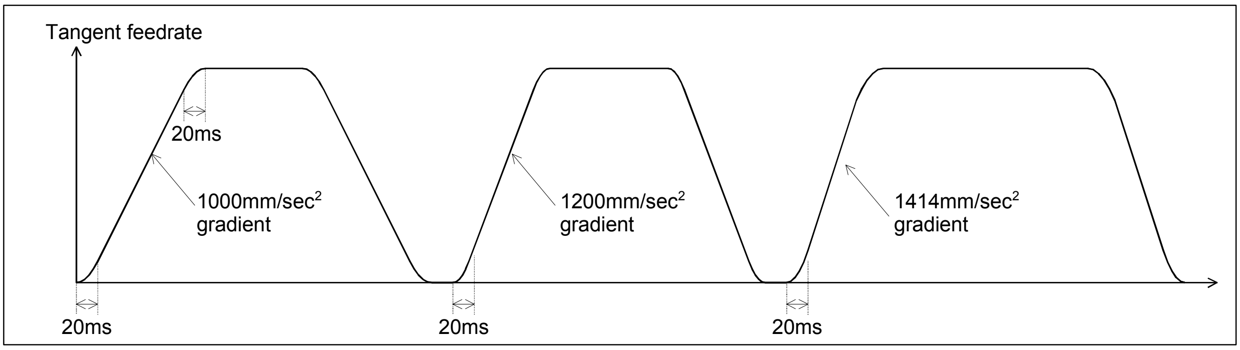
Acceleration/deceleration is performed with the largest tangent acceleration/deceleration that does not exceed the permissible acceleration set for each axis.

(Example)
X-axis permissible acceleration: 1000 mm/sec^2
Y-axis permissible acceleration: 1200 mm/sec^2
Acceleration change time: 20 msec

Program:

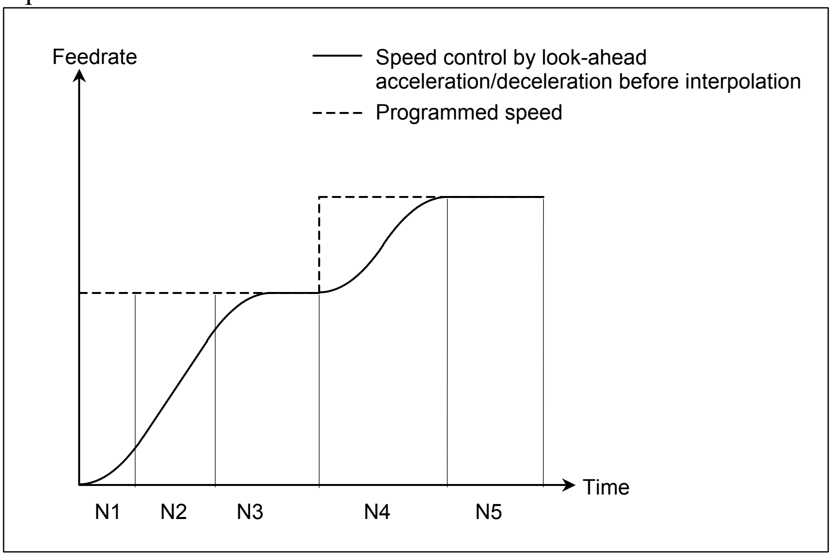
```
N1 G01 G91 X20. F6000 ; (Move on the X-axis.)
G04 X0.01 ;
N2 Y20. ; (Move on the Y-axis.)
G04 X0.01 ;
N3 X20. Y20. ; (Move in the XY direction (at 45 degrees).)
```

Since N3 performs interpolation for the X and Y axes in the 45-degree direction, the acceleration of the Y axis is controlled according to the X axis to become 1000 mm/s^2. Therefore, the combined acceleration is 1414 mm/s^2.



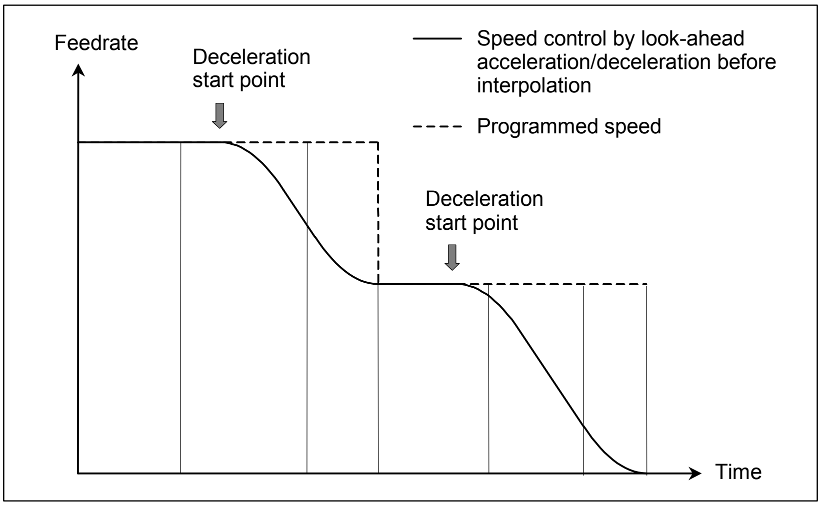
- Acceleration

Acceleration is performed so that the feedrate programmed for a block is attained at the beginning of the block. When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, acceleration can be performed across more than one block.



- Deceleration

Deceleration starts in advance so that the feedrate programmed for a block is attained at the beginning of the block.
When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, deceleration can be performed across more than one block.



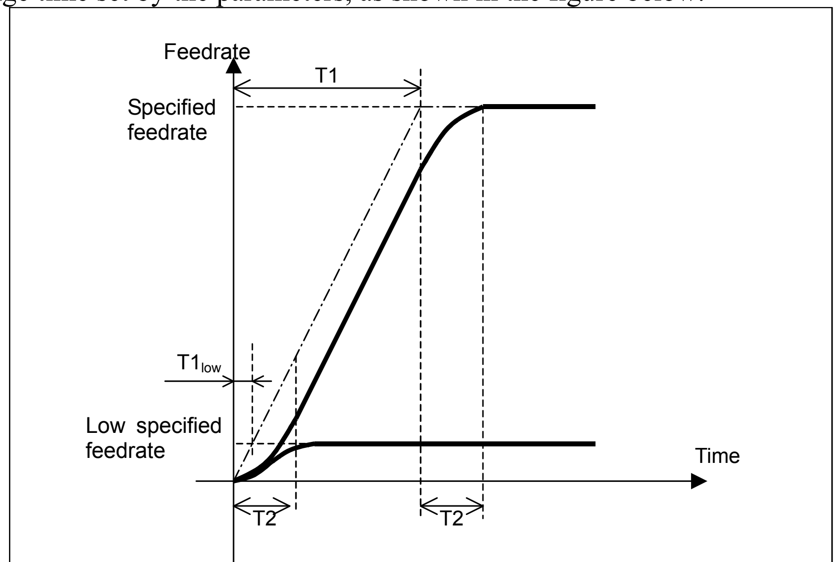
- Deceleration based on a distance

If the total distance of the blocks read ahead becomes shorter than or equal to the deceleration distance obtained from the current feedrate, deceleration starts.
If the total distance of the blocks read ahead during deceleration increases, acceleration is performed.
If the blocks of a small amount of travel are successively specified, deceleration and acceleration may be performed alternately, making the feedrate inconsistent.
To avoid this, decrease the programmed feedrate.

M

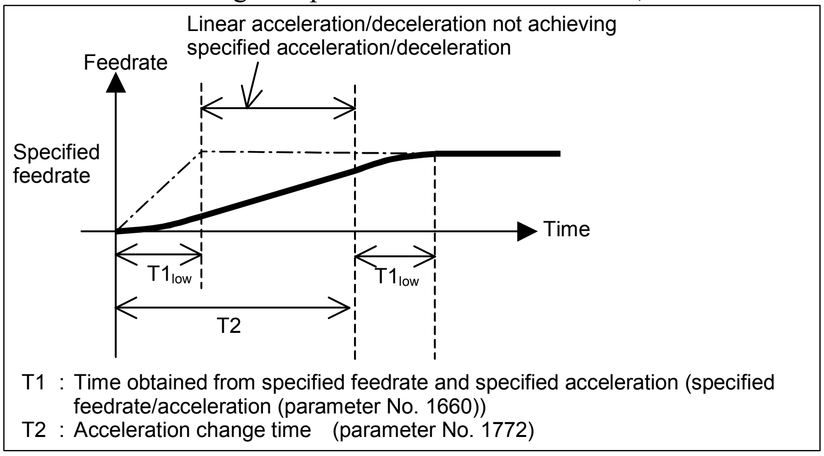
- Function for changing time constant of bell-shaped acceleration/deceleration

Bell-shaped acceleration/deceleration before interpolation is performed according to the acceleration and acceleration change time set by the parameters, as shown in the figure below.

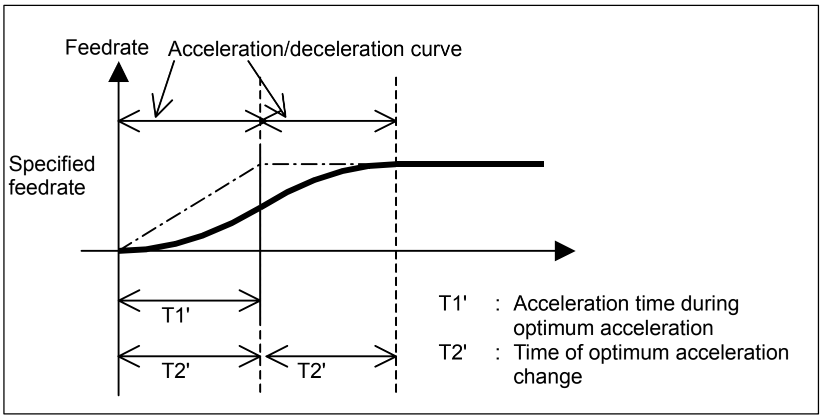


T1 : Time obtained from specified feedrate and specified acceleration (specified feedrate/acceleration (parameter No. 1660))
T2 : Acceleration change time (parameter No. 1772)

Here, the acceleration change time (T2) remains constant regardless of the specified feedrate, while the acceleration time for the linear section (T1), which is determined by acceleration, varies with the specified feedrate. If T1 becomes shorter than T2 when the specified feedrate is low, linear acceleration/deceleration not achieving the specified acceleration results, as shown in the figure below.



In such a case, set bit 3 (BCG) of parameter No. 7055 to 1. Then, the internal acceleration and vector time constant of acceleration/ deceleration before interpolation are changed to make the acceleration/deceleration pattern as close as possible to the optimum bell-shaped acceleration/deceleration before interpolation based on a specified acceleration/deceleration reference speed, and so acceleration/deceleration time is reduced.



There are three methods for specifying the acceleration/deceleration reference speed.

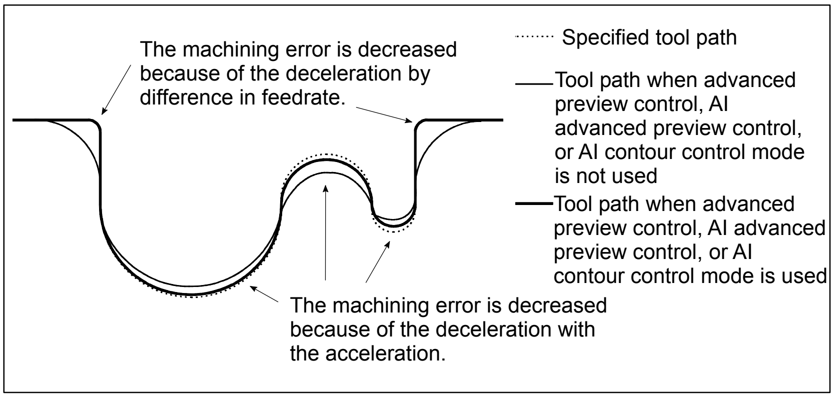
(1) Specifying the speed using an F in a G05.1 Q1 block
(2) Setting the speed on parameter No. 7066
(3) Setting the speed specified with the F command issued at the start of cutting as the reference speed

When F is specified in a G05.1Q1 block, the specified feedrate is assumed to be the acceleration/deceleration reference speed. This command can be used only in the feed per minute mode. If no F command is specified in a G05.1Q1 block, the feedrate specified in parameter No. 7066 is assumed to be the acceleration/deceleration reference speed. If 0 is set in parameter No. 7066, the F command specified in the cutting start block is assumed to be the acceleration/deceleration reference speed.

- Automatic feedrate control function

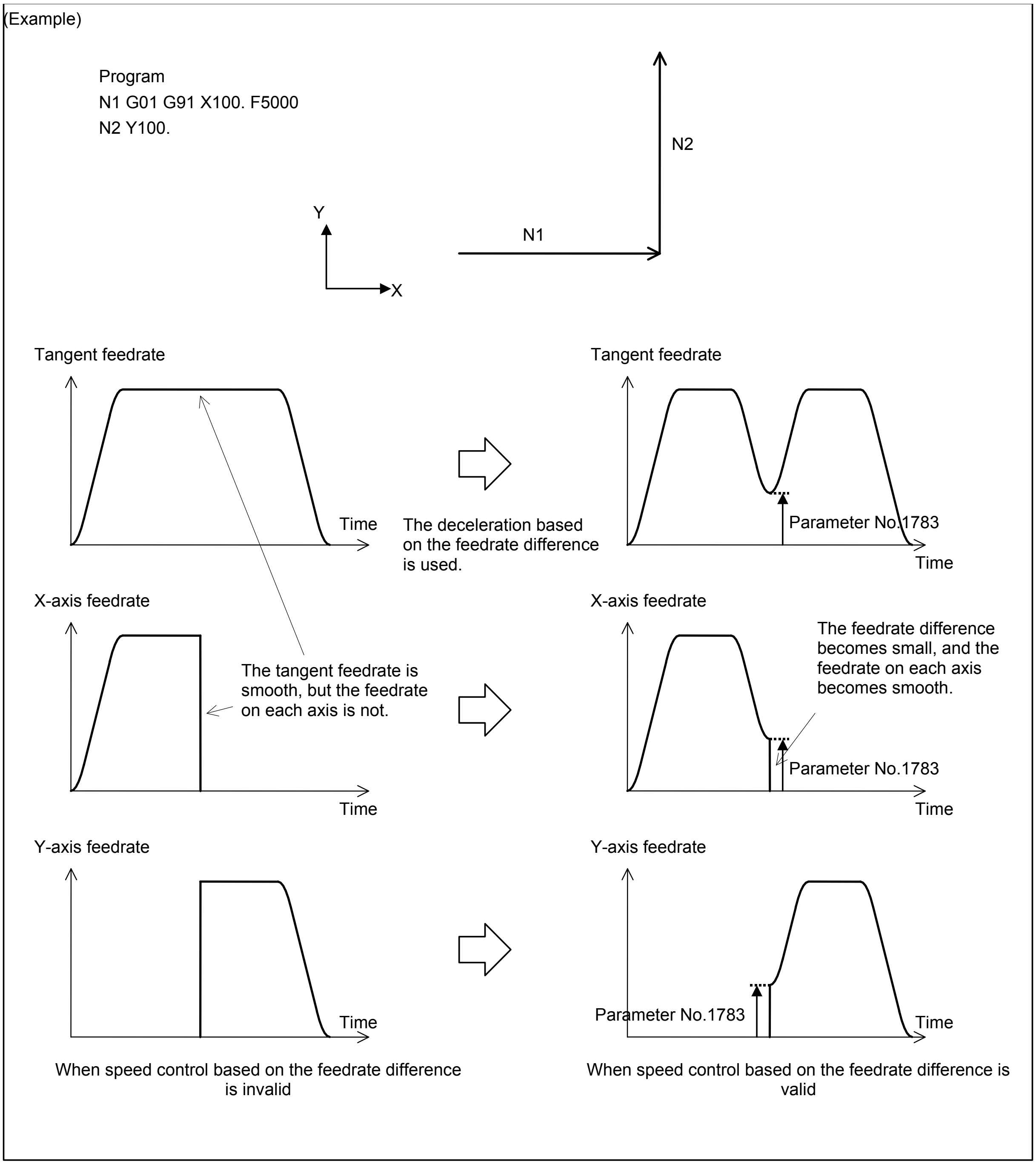
During the advanced preview control, AI advanced preview control, or AI contour control (II) mode, the feedrate is automatically controlled by reading blocks in advance.
The feedrate is determined using the following conditions. If the specified feedrate exceeds the determined feedrate, acceleration/ deceleration before interpolation is performed to achieve the determined feedrate.

<1> Feedrate changes on each axis at a corner and the permissible feedrate change that has been set
<2> Expected acceleration on each axis and the permissible acceleration that has been set
<3> Cutting load that is expected from the travel direction on the Z-axis

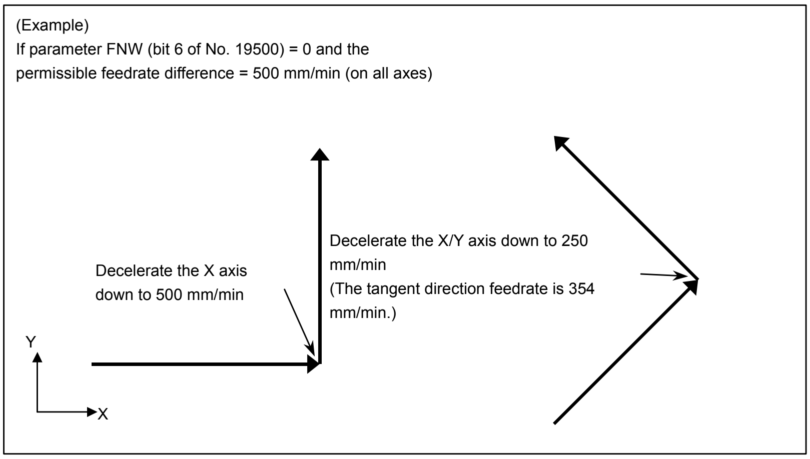


- Speed control based on the feedrate difference on each axis at a corner

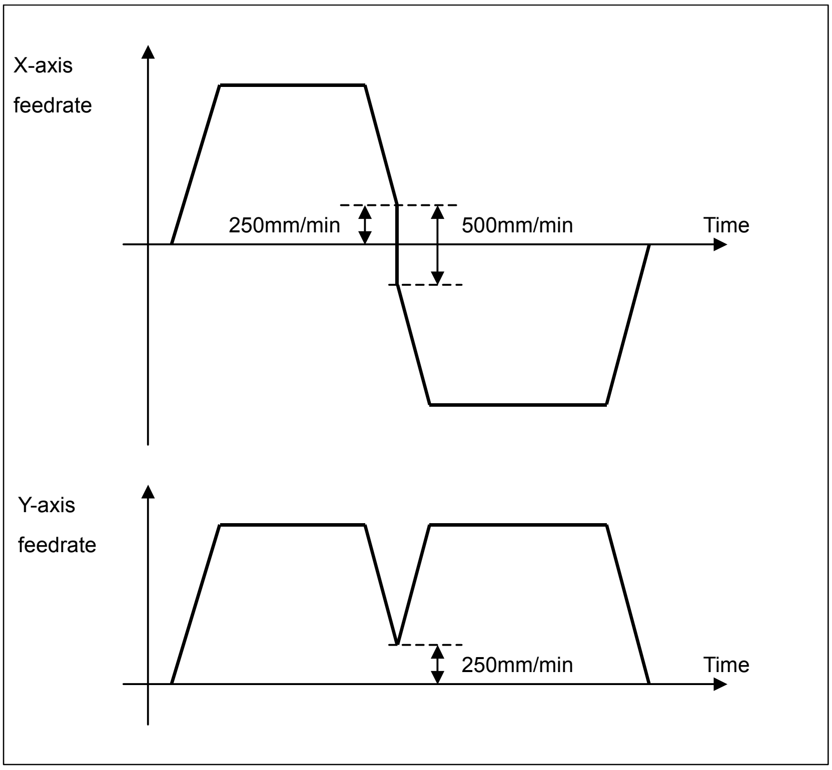
By using the speed control based on the feedrate difference on each axis at a corner, if a feedrate change occurs on an axis on each axis at a corner, the feedrate is determined so that any feedrate difference exceeding the permissible feedrate difference on that axis that has been set for parameter No. 1783 does not occur, and deceleration is automatically performed.



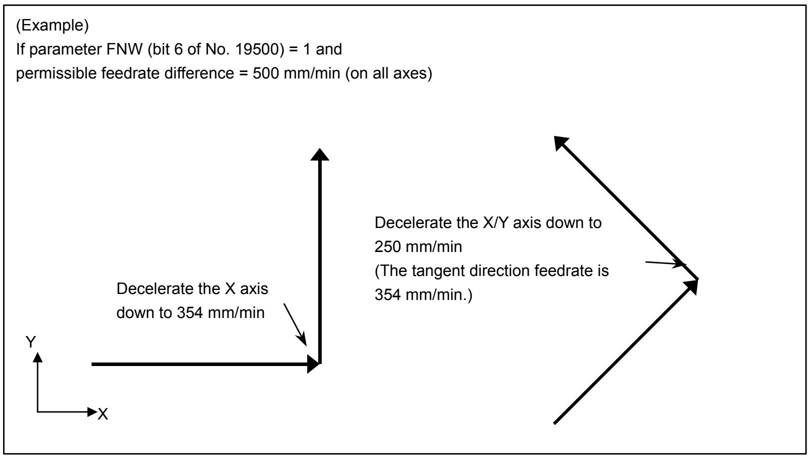
The method of deceleration based on the feedrate difference differs depending on the setting made for parameter FNW (bit 6 of No. 19500).
If "0" is set, the largest feedrate that does not exceed the permissible feedrate difference set for parameter No. 1783 is assumed to be the deceleration feedrate.
In this case, the deceleration feedrate differs if the travel direction differs, even if the shape is the same.



In the left-side example in the figure above, the X axis is inverted at the corner from the position direction to the negative direction, and deceleration is performed so that the feedrate difference becomes 500 mm/min. In other words, the feedrate is 250 mm/min both when the axis moves in the position direction and when it moves in the negative direction. As a result, the tangent direction feedrate becomes 354 mm/min.

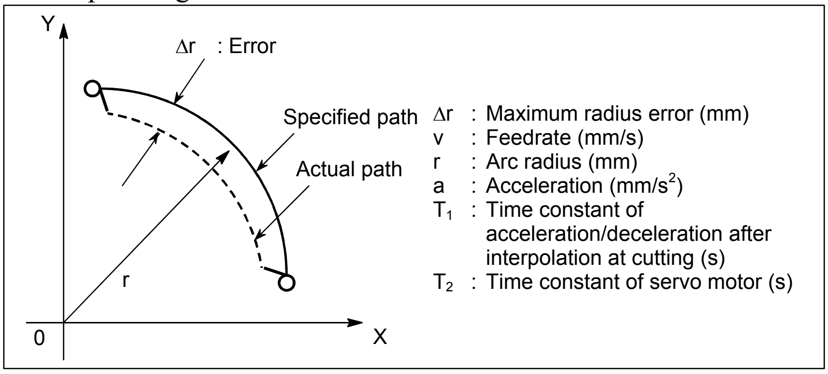


If "1" is set, the feedrate is determined not only with the condition that
the permissible feedrate difference and permissible acceleration on each axis are not exceeded, but also that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If 1 is set for this parameter, the deceleration feedrate determined with the feedrate difference may be up to 30% lower than that determined if 0 is set.



- Speed control with acceleration in circular interpolation

When high-speed cutting is performed in circular interpolation or helical interpolation, the actual tool path has an error with respect to the programmed path. In circular interpolation, this error can be approximated from the equation given below.



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

In actual machining, the permissible error Δr is given as the machining accuracy. Therefore, the permissible acceleration a (mm/sec^2) is determined by equation 1.
When a specified feedrate causes the radial error from an arc having a programmed radius to exceed the permissible error, speed control with acceleration in circular interpolation automatically clamps the arc-cutting feedrate by using parameter settings.

Let the permissible acceleration calculated from the permissible acceleration set for each axis be A. Then, maximum permissible feedrate v with programmed radius r is expressed as follows:

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

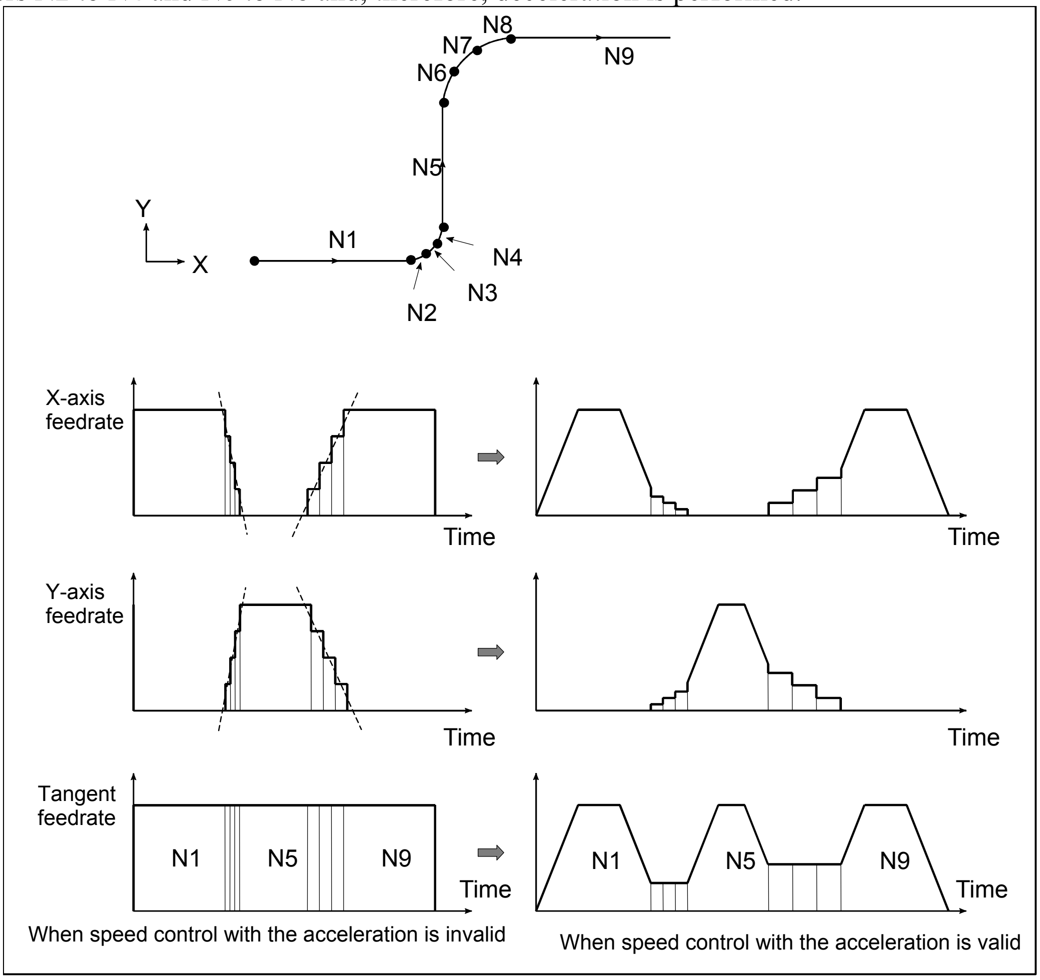
If a specified feedrate exceeds feedrate v obtained from equation 2, the feedrate is clamped at feedrate v automatically.
The permissible acceleration is specified in parameter No. 1735. If there is a difference in permissible acceleration between two axes for circular interpolation, the lower acceleration is regarded as the permissible acceleration.
If the radius of an arc is small, too small value can be calculated as deceleration v. In such a case, the lower feedrate limit can be set in parameter No. 1732 to prevent the feedrate from being decreased too much.

M

- Speed control with the acceleration on each axis

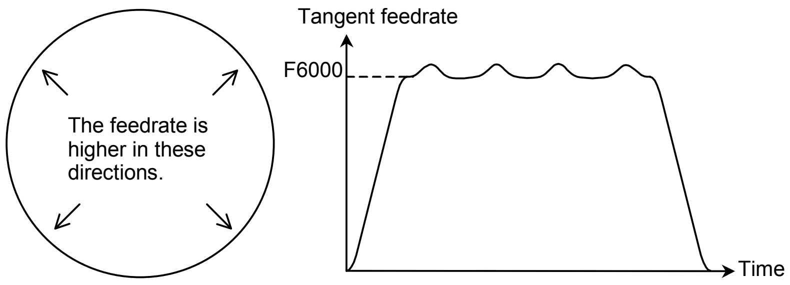
When consecutive small lines are used to form a curve, as in the example shown in the figure below, the feedrate differences on each axis at the individual corners are not very large. Thus, deceleration with the feedrate differences is not effective. Consecutive small feedrate differences, however, cause a large acceleration on each axis, as a whole.
In such a case, deceleration can be performed to reduce the impact on the machine and the machining error caused by too large an acceleration. The deceleration feedrate is determined to be the feedrate that does not cause the acceleration on each axis to exceed the permissible acceleration set for parameter No. 1737.
The deceleration feedrate is determined for each corner. The actual feedrate is the smaller of the deceleration feedrate determined at the start point of the block and that determined at the end point.
Depending on the specified figure, a very low deceleration feedrate may be calculated. In such a case, the lower feedrate limit can be set in parameter No. 1738 to prevent the feedrate from being decreased too much.

In the following example, the acceleration (gradient of the broken line in the feedrate graph) at too large at corners N2 to N4 and N6 to N8 and, therefore, deceleration is performed.



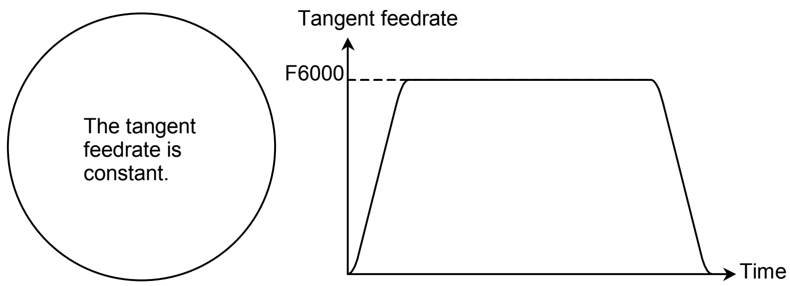
The method of determining the feedrate with the acceleration differs depending on the setting of parameter FNW (bit 6 of No. 19500).
If "0" is set, the highest feedrate that does not cause the permissible acceleration set for parameter No. 1737 to be exceeded is assumed to be the deceleration feedrate. In this case, the deceleration feedrate differs depending on the travel direction even if the shape is the same, as shown in the figure below.

(Example) If a circular shape with a radius of 10 mm is specified with small line blocks
Parameter FNW (bit 6 of No. 19500) = 0
Permissible acceleration = 1000 mm/s^2 (on all axes)



If "1" is set, the feedrate is determined with not only the condition that the permissible acceleration on each axis is not exceeded but also the condition that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If 1 is set for this parameter, the deceleration feedrate determined with the feedrate difference or acceleration may be up to 30% lower than that determined if 0 is set.

(Example) If a circular shape with a radius of 10 mm is specified with small line blocks
Parameter FNW (bit 6 of No. 19500) = 1,
radius = 10 mm, permissible acceleration = 1000 mm/s^2 (on all axes)

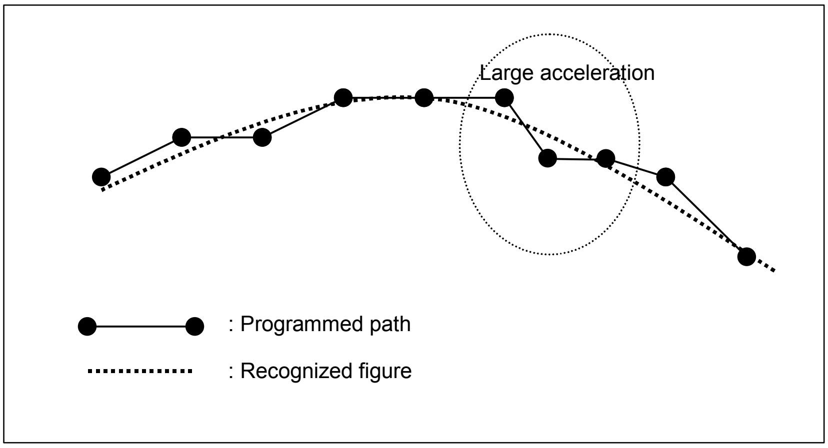


NOTE
In circular interpolation, the tangent feedrate is constant regardless of the setting of the parameter.

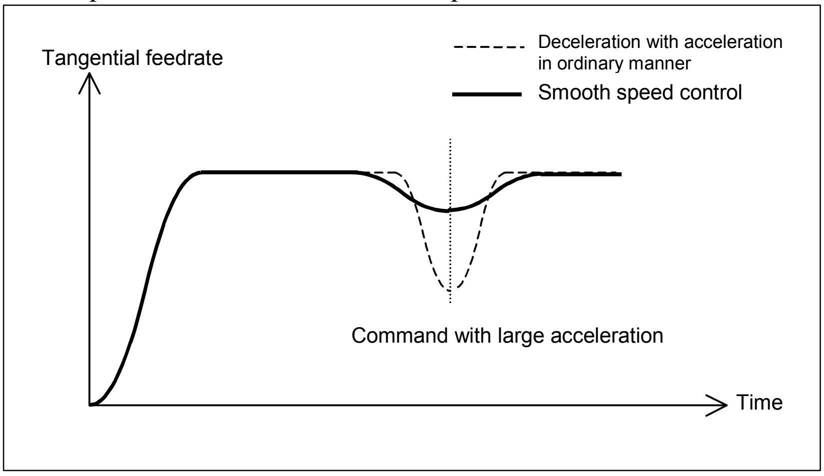
M

- Smooth speed control

In speed control with acceleration, the smooth speed control function recognizes the entire figure from preceding and following blocks including blocks read ahead to make a smooth feedrate determination.
When a curve is specified with successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line.
When the feedrate is determined with acceleration in an ordinary manner, an optimum feedrate is automatically calculated exactly for a programmed figure, so a large acceleration may result depending on the command, which can lead to deceleration.
In such a case, the use of smooth speed control enables speed control by recognizing the entire figure, which provides smooth speed control while suppressing local deceleration, therefore increasing the feedrate.



Also for a part of a programmed figure in which a large acceleration would be required, the acceleration is obtained based on the figure recognized from multiple blocks, and the feedrate is determined so that the acceleration is within the permissible acceleration set in parameter No. 1737.



Smooth speed control obtains the acceleration by using the figure recognized from the preceding and following blocks including blocks read ahead, so smooth speed control is enabled even in parts in which the acceleration increases.
Smooth speed control is enabled under the following conditions:
<1> Speed control with acceleration is enabled in the AI contour control II mode.
<2> Successive linear interpolation commands are specified.
<3> Bit 0 (HPF) of parameter No. 19503 is set to 1.

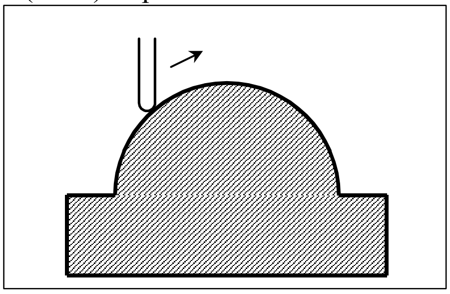
> ⚠ **CAUTION**
> When smooth speed control is used, the feedrate in a certain figure such as a corner may become larger than the feedrate obtained by ordinary speed control with acceleration. For corners, set parameter No. 1783, which is the permissible feedrate difference parameter for speed control with the feedrate difference at corners, to perform appropriate deceleration by speed control with the corner feedrate difference.

M

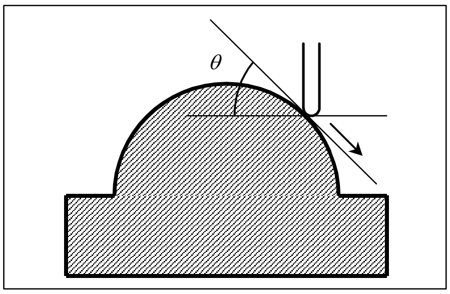
- Speed control with the cutting load

Usually, the cutting resistance produced when machining is performed with the bottom of the cutter as the tool lowers along the Z-axis is greater than the cutting resistance produced when machining is performed with the side of the cutter as the tool rises along the Z-axis. Therefore, deceleration is required.

In AI contour control II, the tool travel direction on the Z-axis is used as a condition for calculating the machining feedrate.
This function is enabled when bit 4 (ZAG) of parameter No. 8451 is set to 1.



During ascent on the Z-axis



During descent on the Z-axis

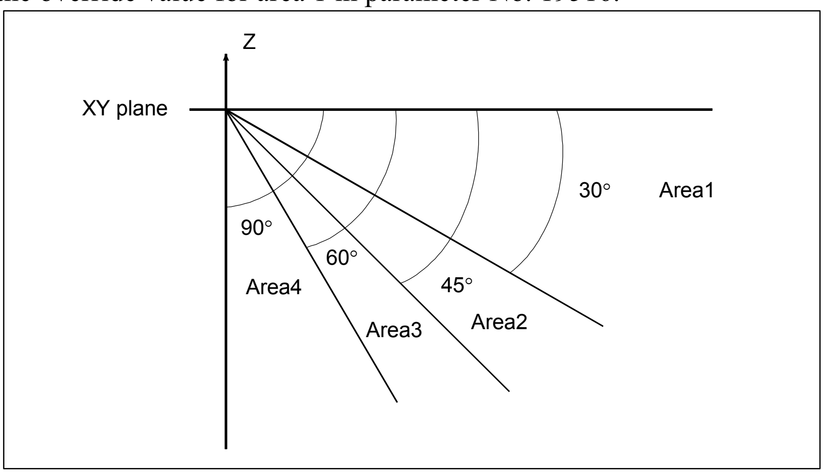
The descent angle θ during descent on the Z-axis (angle formed by the XY plane and the tool center path) is as shown in the figure. The descent angle is divided into four areas, and the override values for the individual areas are set for the following parameters:

Parameter No. 8456 for area 2
Parameter No. 8457 for area 3
Parameter No. 8458 for area 4

For area 1, however, no parameter is available, and an override of 100% is used at all times. The feedrate obtained according to other feedrate control is multiplied by the override value of the area to which descent angle θ belongs.

Area1 $0^\circ \le \theta < 30^\circ$
Area2 $30^\circ \le \theta < 45^\circ$
Area3 $45^\circ \le \theta < 60^\circ$
Area4 $60^\circ \le \theta < 90^\circ$

The feedrate can be overridden with an inclination by setting bit 1 (ZG2) of parameter No. 19515 to 1. In this case, specify the override value for area 1 in parameter No. 19516.



> ⚠ **CAUTION**
> 1 The speed control with the cutting feed is effective only when the tool is parallel with the Z-axis. Thus, it may not be possible to apply this function, depending on the structure of the machine used.
> 2 In the speed control with the cutting feed, the travel direction on the Z-axis is determined with the appropriate NC command. If, therefore, manual intervention is performed on the Z-axis with manual absolute on, or if a mirror image is applied on the Z-axis, the direction on the Z-axis cannot be determined. When using the speed control with the cutting load, do not use these functions.
> 3 When performing 3-dimensional coordinate conversion, determine the descent angle on the Z-axis using the converted coordinate system.
> 4 Speed control with the cutting load is enabled for all interpolations in the AI contour control II mode. This function, however, can be made valid only for linear interpolations by setting bit 4 (ZOL) of parameter No. 19503 to 1.

M

- Ignoring feedrate commands

In a block in which AI contour control II is enabled, all feedrate commands (F commands) can be ignored by setting bit 7 (NOF) of parameter No. 8451.
The term feedrate commands, as used here, refer to the following commands:

<1> Modal F commands before the block in which AI contour control II is enabled
<2> F commands and modal F commands in the block in which AI contour control II is enabled

When the feedrate commands are ignored, it is assumed that the upper feedrate limit specified for parameter No. 8465 is specified.
Note, however, that any issued F commands and modal F commands are stored within the CNC.
Thus, in a block in which AI contour control II changes from the enabled state to the disabled state, the modal values of the F commands described in <1> and <2> described above are used as modal F commands, instead of the modal values of the F commands calculated by AI contour control II.

- Another example of determining the feedrate

If a specified feedrate exceeds the upper feedrate limit of advanced preview control/AI advanced preview control/AI contour control (II) (in parameter No. 8465), the feedrate is clamped at the upper feedrate. The upper feedrate limit is clamped at the maximum cutting feedrate (parameter No. 1432).

Limitations

- Conditions for temporarily canceling the advanced preview control, AI advanced preview control, or AI contour control (II) mode

If any of the commands listed below is executed during advanced preview control, AI advanced preview control, or AI contour control (II) mode is temporarily canceled. Note that the advanced preview control, AI advanced preview control, or AI contour control (II) mode resumes as soon as it becomes available.

T

Function name	G code
Positioning (rapid traverse) (NOTE 1)	G00
Spindle positioning	G00
Rigid tapping	G84,G88
Threading (NOTE 2)	G32

Function name	G code
Variable-lead threading (NOTE 2)	G34
Single threading cycle (NOTE 2)	G92
Multiple repetitive threading cycle (NOTE 2)	G76
When no move command is specified	—
One-shot G code other than those shown at right (NOTE 1)	G09 G38,G39

M

Function name	G code
Positioning (rapid traverse) (NOTE 1)	G00
Single direction positioning	G60
Rigid tapping	G74,G84
Threading (NOTE 2)	G33
Electronic gear box (EGB)	G81
When no move command is specified	—
One-shot G code other than those shown at right (NOTE 1)	G09 G38,G39 G45,G46,G47,G48

NOTE

1 If the first three conditions (1) to (3) below are all met, the mode is not canceled even when the rapid traverse command is specified. If all the conditions (1) to (5) are met, the mode is not canceled even when the G28, G30, or G53 command is specified.
   (1) Bit 1 (LRP) of parameter No. 1401 is set to 1. (Interpolation type positioning is valid.)
   (2) Parameter No. 1671 (maximum acceleration during rapid traverse) is set.
   (3) Bit 5 (FRP) of parameter No. 19501 is set to 1 (acceleration/deceleration before interpolation is valid for rapid traverse).
   (4) Bit 4 (ZRL) of parameter No. 1015 is set to 1 (the G28, G30, and G53 commands are of the interpolation type).
   (5) Bit 1 (AMP) of parameter No. 11240 is set to 1 (acceleration/deceleration before interpolation is valid for the G28, G30, and G53 commands in the high-speed, high-precision mode).

2 Acceleration/deceleration before interpolation is invalid for a threading command. Therefore, if acceleration/deceleration before interpolation is enabled by a command that precedes or follows a threading command, the tool is decelerated and stopped temporarily when the block changes.
   Since the state of acceleration/deceleration before interpolation does not change during continuous threading, deceleration does not occur when the block changes.

- Parameter list

Positioning

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Positioning type (non-linear (0)/interpolation (1))	1401#1 LRP		

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Acceleration/deceleration type (acceleration constant (0)/time constant (1))	1603#4 PRT		
Acceleration/deceleration type (after interpolation (0)/before interpolation (1))	19501#5 FRP		
Time constant of acceleration/deceleration after interpolation in rapid traverse	1620		
Time constant of bell-shaped acceleration/deceleration after interpolation in rapid traverse	1621		
Maximum permissible acceleration of acceleration/deceleration after interpolation in rapid traverse	1671		
Acceleration change time of bell-shaped acceleration/deceleration before interpolation in rapid traverse	1672		

Acceleration/deceleration before interpolation

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Maximum permissible acceleration of acceleration/deceleration before interpolation	1660		
Acceleration change time of bell-shaped acceleration/deceleration before interpolation	None		1772
Valid/invalid state of the function for changing time constant of bell-shaped acceleration/deceleration before interpolation	None		7055#3 BCG
Reference acceleration/deceleration speed for the function for changing time constant of bell-shaped acceleration/deceleration before interpolation	None		7066

Acceleration/deceleration after interpolation

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Acceleration/deceleration type of acceleration/deceleration after interpolation in cutting feed	1602#3 BS2, 1602#6 LS2		
FL rate of acceleration/deceleration after interpolation in cutting feed	1763		
Time constant of acceleration/deceleration after interpolation in cutting feed	1769		

Speed control based on the feedrate difference on each axis

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Permissible feedrate difference when determining the feedrate based on the feedrate difference at a corner	1783		
Method of determining the feedrate based on the feedrate difference or based on the acceleration	19500#6 FNW		

Speed control with acceleration in circular interpolation

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Lower-limit feedrate for the deceleration function with the acceleration in circular interpolation	1732		
Permissible acceleration for the deceleration function with the acceleration in circular interpolation	1735		

M

Speed control with the acceleration on each axis

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Permissible acceleration for the deceleration function with the acceleration	None	1737	
Lower-limit feedrate for the deceleration function with the acceleration	None	1738	
Method of determining the feedrate based on the feedrate difference or based on the acceleration	None	19500#6 FNW	

Others

Parameter	Parameter No.		
	Advanced preview control	AI advanced preview control	AI contour control (II)
Maximum cutting feedrate during the mode of acceleration/deceleration before interpolation	1432		
Upper-limit feedrate for advanced preview control, AI advanced preview control, or AI contour control (II)	8465		
Upper-limit feedrate for advanced preview control, AI advanced preview control, or AI contour control (II) (when only the rotation axis is specified)	8466		

16.2 MACHINING CONDITION SELECTING FUNCTION

Overview

By setting a speed- or precision-focused parameter set in an advanced preview control (T series) / AI advanced preview control (M series) / AI contour control (II) (M series) function and specifying a precision level in accordance with the machining conditions during machining, parameters suitable to the conditions can be automatically calculated so that machining can be performed.
This function is an optional one.

Format

- Changing the precision level using a program

In addition to being switched on the precision level selection screen, the precision level can be changed using a program in the format below.

T

For advanced preview control

G08 P1 Rx ;
x.......Level (1 to 10)

> ⚠ **CAUTION**
> Once specified, a level remains effective even if the advanced preview control mode is canceled.

M

For AI advanced preview control/AI contour control (II)

G05.1 Q1 Rx ;
x.......Level (1 to 10)

> ⚠ **CAUTION**
> Once specified, a level remains effective even if the AI advanced preview control / AI contour control (II) mode is canceled.

16.3 MACHINING QUALITY LEVEL ADJUSTMENT (M Series)

M

Overview

In nano smoothing, if the “level 1” and “level 10” parameters of a precision level and smoothing level are set in order to specify a precision level and smoothing level according to the machining condition during machining, the parameter values corresponding to the condition can be automatically calculated for machining.
On the machining quality level adjustment screen, the machining quality/precision/speed level in nano smoothing can easily be adjusted.
This function is an optional function.

Format

- Changing the smoothing level by a program

The smoothing level can be switched on the machining level selection screen or machining quality level adjustment screen; it can also be changed by a program with the following format.

G05.1 Q3 Rx ;
x ....Level (1 to 10)

> ⚠ **CAUTION**
> Once a level is specified, it remains valid even after the nano smoothing mode is canceled.

- Changing the precision level by a program

For information on the changing the precision level by a program, see Section 16.2, "MACHINING CONDITION SELECTION FUNCTION".

16.4 JERK CONTROL (M Series)

M

16.4.1 Speed Control with Change of Acceleration on Each Axis

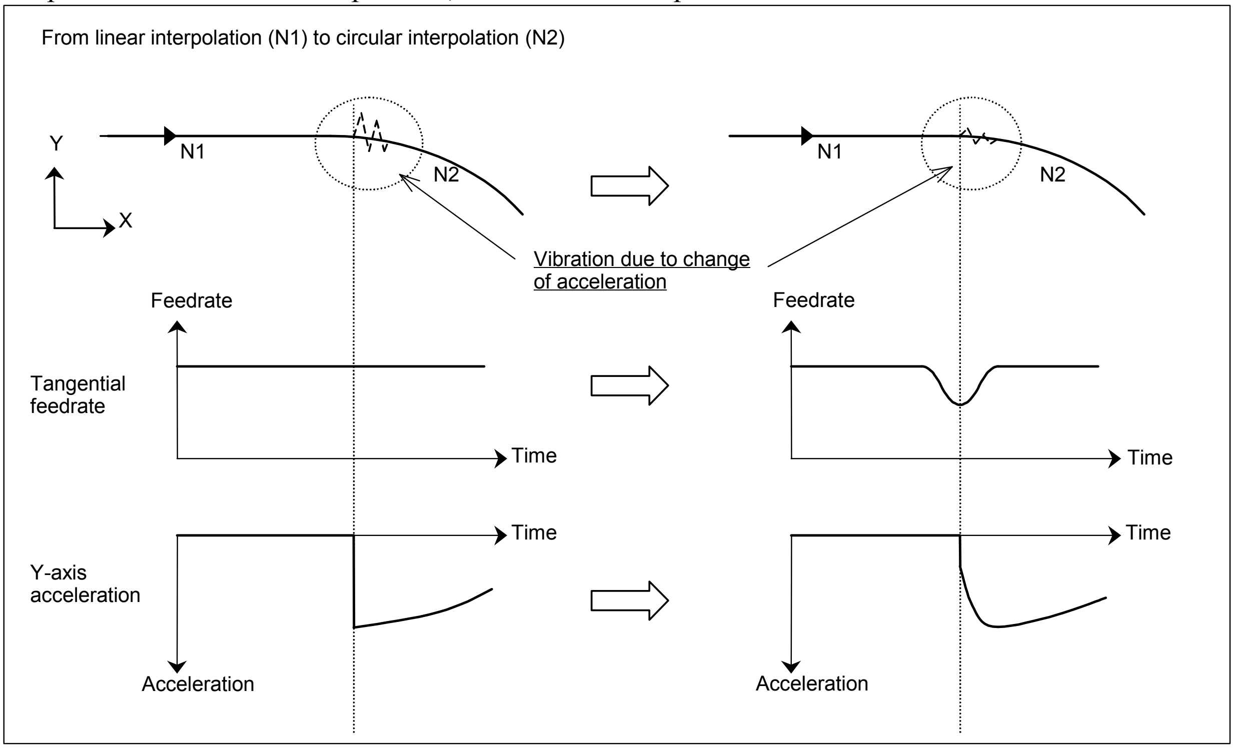
Overview

In portions in which acceleration changes largely, such as a portion where a programmed figure changes from a straight line to curve, vibration or shock on the machine may occur. Speed control with change of acceleration on each axis is a function to suppress machining errors due to vibration and machine shock generated by change of acceleration. This function obtains a feedrate so that change of acceleration is within the parameter-set permissible acceleration change amount for each axis, and performs deceleration by using acceleration/deceleration before interpolation.

> ⚠ **CAUTION**
> Before speed control with change of acceleration on each axis can be used, the options for jerk control and AI contour control II are required.

Explanation

In the following example, the Y-axis acceleration changes largely at the contact point between a linear interpolation and circular interpolation, so deceleration is performed.



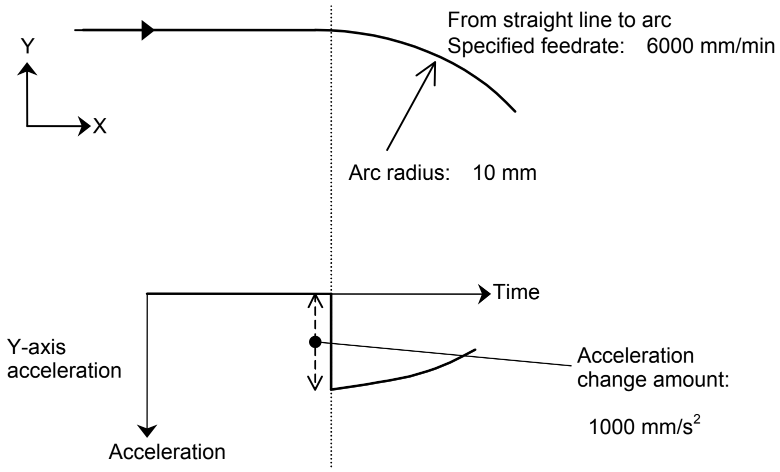
- Setting the permissible acceleration change amount

The permissible acceleration change amount for each axis is set in parameter No. 1788. When 0 is set in this parameter for a certain axis, speed control with change of acceleration is not performed for that axis.

- Parameter setting example

Suppose a figure shown below in which a straight line is followed by an arc. Let the specified feedrate and the arc radius be 6000 mm/min and 10 mm, respectively. Then, the Y-axis acceleration change amount at the contact point of the linear and arc portions is obtained as follows:

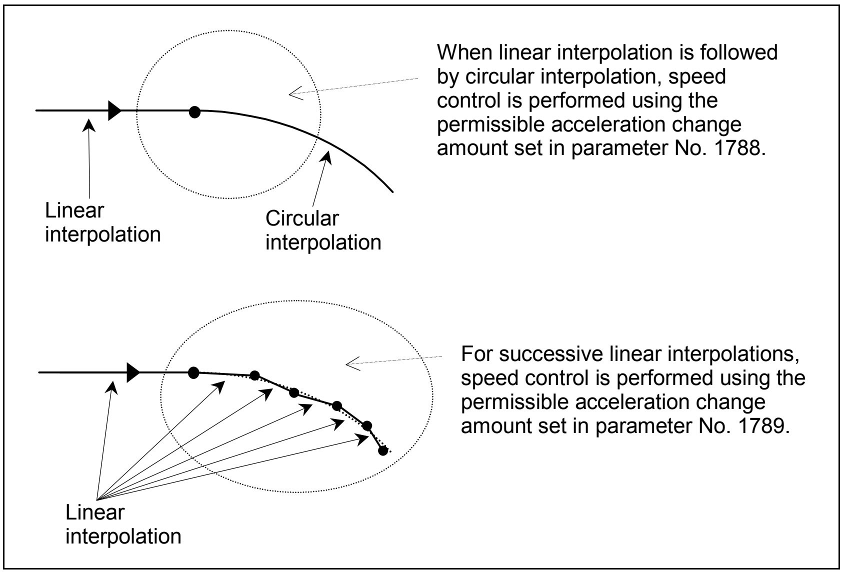
$$\frac{v^2}{r} = 1000 mm / s^2$$



To suppress the change of acceleration to 300 mm/s^2, set 300 mm/s^2 for the Y-axis in parameter No. 1788. Note that the change of acceleration is determined from the interpolation data of the CNC, so it may differ from the theoretical value.
The actual machine is affected by acceleration/deceleration and other factors, so the value to be set in the parameter should be determined after adjustments are made.

- For successive linear interpolations

When there are successive linear interpolations, speed control with change of acceleration obtains the deceleration feedrate from the change in acceleration between the start point and end point of a specified block.
When a curve is specified using successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line. The error due to rounding may increase change of acceleration, and especially when the line segments specified by blocks are short, deceleration is performed frequently. As a result, the machining speed cannot increase enough. In such a case, a relatively large value should be set in parameter No. 1789 as the permissible acceleration change amount for each axis in successive linear interpolations to improve the machining speed.
When a value other than 0 is set in parameter No. 1789 for an axis for which deceleration with change of acceleration is enabled, this setting is regarded as the permissible acceleration change amount at corners in which linear interpolations meet. (For portions where a linear interpolation and circular interpolation meet and where circular interpolations meet, the setting in parameter No. 1788 is used.)
When 0 is set in parameter No. 1789 for an axis, the setting in parameter No. 1788 specifying the ordinary permissible acceleration change amount is used even at a corner in which linear interpolations meet.
When smooth speed control is used in speed control with permissible acceleration in AI contour control II, the deceleration feedrate is obtained from the change of acceleration calculated by smooth speed control. Therefore, the deceleration feedrate may be higher than the ordinary deceleration feedrate.

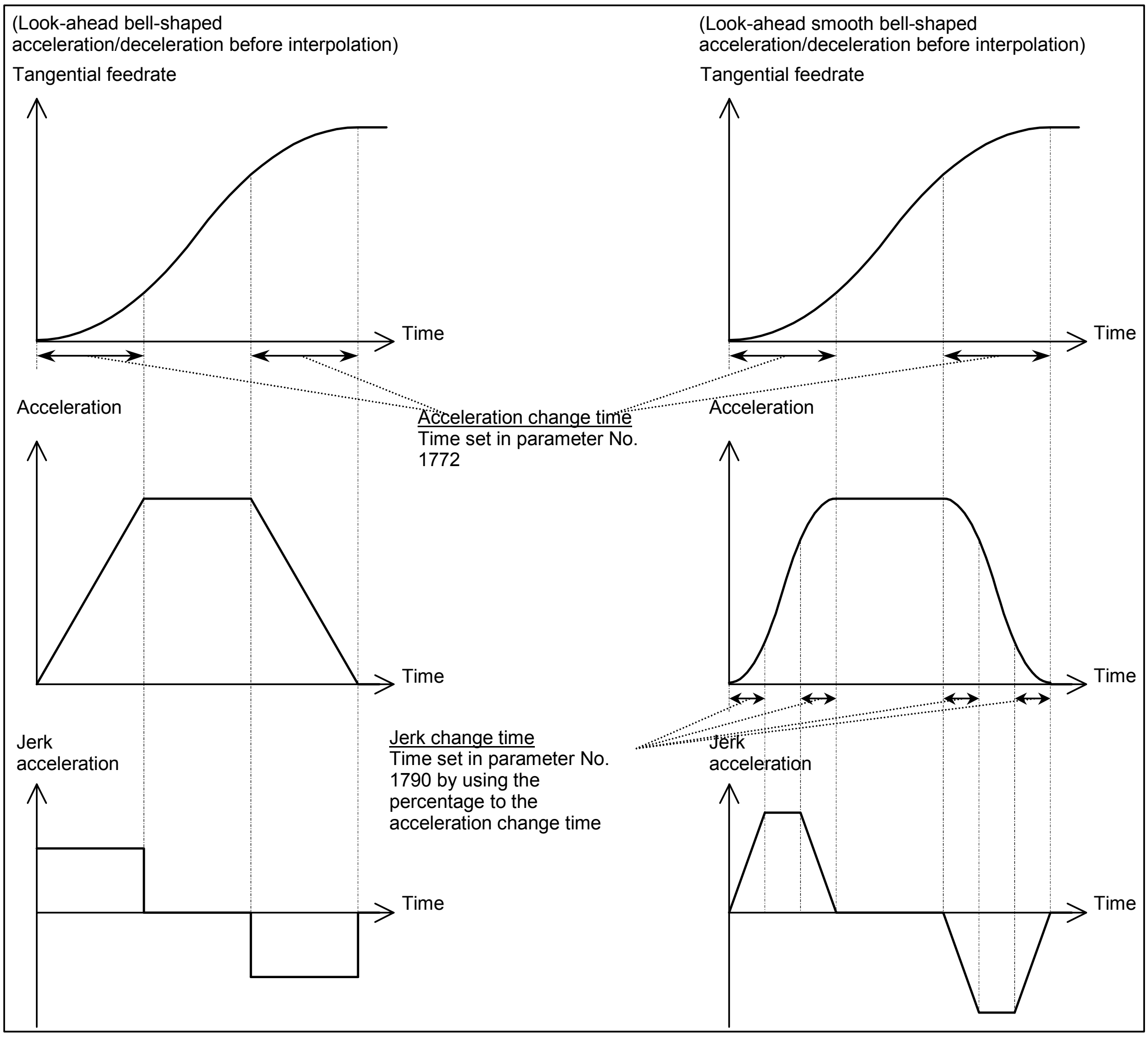

When linear interpolation is followed by circular interpolation, speed control is performed using the permissible acceleration change amount set in parameter No. 1788.
Linear interpolation
Circular interpolation
For successive linear interpolations, speed control is performed using the permissible acceleration change amount set in parameter No. 1789.
Linear interpolation

16.4.2 Look-Ahead Smooth Bell-Shaped Acceleration/Deceleration before Interpolation

Overview

In look-ahead bell-shaped acceleration/deceleration before interpolation performs smooth acceleration/deceleration by changing the acceleration at a constant rate in specified acceleration change time.

In look-ahead smooth bell-shaped acceleration/deceleration before interpolation, the jerk change time is specified in parameter No. 1790 by using the percentage to the acceleration change time for look-ahead bell-shaped acceleration/deceleration before interpolation, and change of acceleration is also controlled so that the change is bell-shaped. This enables smoother acceleration/deceleration, therefore reducing machine vibration and shock due to acceleration/ deceleration.



> ⚠ **CAUTION**
> Before look-ahead smooth bell-shaped acceleration/deceleration before interpolation can be used, the option for jerk control and AI contour control II is required.

Explanation

- Setting the jerk change time

The jerk change time is set in parameter No. 1790 by using the percentage to the acceleration change time.
The actual jerk change time is represented by the percentage to the acceleration change time set in parameter No. 1772.
The jerk change time must be within a half of the acceleration change time, so the value to be set in the parameter ranges 0 to 50 (percent).
If 0 or a value beyond the specifiable range is specified in parameter No. 1790, look-ahead smooth bell-shaped acceleration/deceleration before interpolation is not enabled.

- Acceleration/deceleration before interpolation for linear type rapid traverse

When bell-shaped acceleration/deceleration is used in acceleration/
deceleration before interpolation for linear type rapid traverse, enabling look-ahead smooth bell-shaped acceleration/deceleration before interpolation applies smooth bell-shaped acceleration/deceleration to acceleration/deceleration before interpolation for linear type rapid traverse.
In this case, the jerk change time is represented by the percentage set in parameter No. 1790 to the acceleration change time set in parameter No. 1672.

17 AXIS CONTROL FUNCTIONS

Chapter 21, "AXIS CONTROL FUNCTIONS", consists of the following sections:

17.1 AXIS SYNCHRONOUS CONTROL

Overview

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. 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).

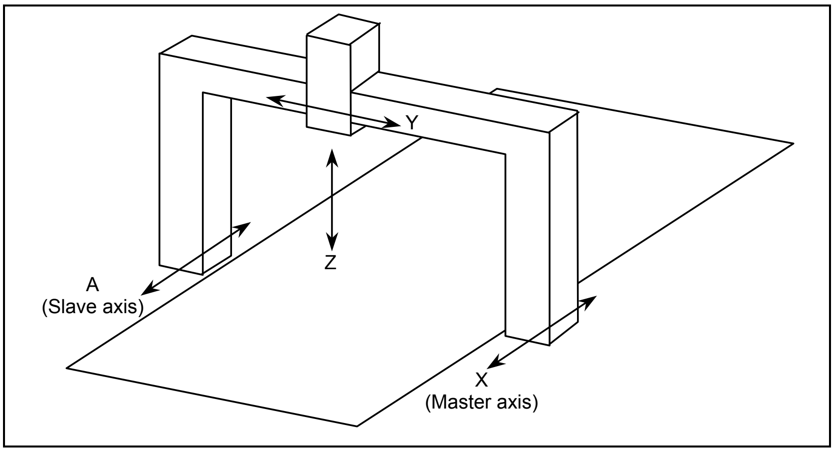


Fig. 17.1 (a) Example of machine with X and A being synchronous axes

The synchronous establishment function can be used for automatic compensation to eliminate a machine coordinate error in cases such as emergency stop cancellation.

An external signal can be used to turn synchronization on and off.

17.1.1 Axis Configuration for Axis Synchronous Control

Explanation

- Master axis and slave axis for axis synchronous control

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).
By setting the axis number of a master axis in the parameter No. 8311 of the slave axis, the axis configuration for axis synchronous control is determined.

- Synchronous operation and normal operation

Operation where axis synchronous control is turned on (enabled) to make a movement along the slave axis in synchronism with the master axis is referred to as synchronous operation. Operation where axis synchronous control is turned off (disabled) to make movements along the master axis and slave axis independently of each other is referred to as normal operation.

(Example)
Automatic operation when the master axis is the X-axis and the slave axis is the A-axis

In synchronous operation, movements are made along the X-axis and A-axis according to the programmed command Xxxxx for the master axis.

In normal operation, movements are made along the master axis and slave axis independently of each other as in the case of normal CNC control. The programmed command Xxxxx makes a movement along the X-axis. The programmed command Aaaaa makes a movement along the A-axis. The programmed command Xxxxx Aaaaa makes movements along the X-axis and A-axis at the same time.

The mode of operation can be switched between synchronous operation and normal operation by an input signal, or synchronous operation can be performed at all times. Which mode to use can be set using bit 5 (SCA) of parameter No. 8304.

- Switching between synchronous operation and normal operation by using an input signal

When bit 5 (SCA) of parameter No. 8304 is set to 0 for the slave axis, the signal SYNCx/SYNCJx (with x representing a slave axis number) is used to switch between synchronous operation and normal operation. When SYNCx/SYNCJx = 1, synchronous operation is selected. When SYNCx/SYNCJx = 0, normal operation is selected.

During feed axis synchronization control, the output signal SYNOx is set to "1".

- Setting for using synchronous operation at all times

When bit 5 (SCA) of parameter No. 8304 for the slave axis is set to 1, synchronous operation is performed at all times, regardless of the setting of the signal SYNCx/SYNCJx.

- Synchronous control axis name

The name of a master axis and the name of a slave axis may be the same or may be different from each other.

- Restrictions on using the same name for the master axis and slave axis

If the same axis name is assigned to the master axis and slave axis, manual operation only is allowed in normal operation. Automatic operation cannot be performed.

- Setting of an axis name subscript

A subscript can be attached to an axis name like X1, X2, XM, and XS. If the same axis name is used for multiple axes, and a unique subscript is assigned to each of those axes, the axes can be distinguished from each other on the screen display, or which of those axes issued an alarm can be identified.
Set a subscript in parameter No. 3131.

- Setting of multiple slave axes

One master axis can have multiple slave axes.

(Example)

In the example below, movements along the X1-axis and X2-axis are made in synchronism with the XM-axis.

Axis name indication	Controlled axis number	Axis name Parameter (No. 1020)	Subscript Parameter (No.3131)	Master axis number Parameter (No.8311)	Operation
XM	1	88	77	0	
Y	2	89	0	0	
X1	3	88	49	1	A movement is made in synchronism with the XM-axis.
X2	4	88	50	1	A movement is made in synchronism with the XM-axis.

When one master axis has multiple slave axes, synchronous establishment, and synchronous error check are performed for each slave axis independently.

- Combination with tandem control

Tandem control can be used with each of the master and slave axes. The same restriction on axis arrangement as imposed in the case of normal tandem control is imposed. No particular restriction is imposed on axis synchronous control.

- Axis selection on the screen display

On a screen such as the current position display screen, a slave axis is also displayed. The display of a slave axis can be disabled by setting bit 0 (NDP) of parameter No. 3115 to 1 and setting bit 1 (NDA) of parameter No. 3115 to 1.

- Axis selection in actual cutting feedrate display

By setting bit 2 (SAF) of parameter No. 8303 to 1 for a slave axis, the slave axis can be included in an actual cutting feedrate display calculation during synchronous operation.

- Axis synchronous control with an absolute-position detector

When bit 7 (SMA) of parameter No. 8302 is set to 1 to attach an absolute-position detector, and bit 4 (APZ) of parameter No. 1815 for an axis placed in synchronous operation is turned off, APZ for the axis (axes) placed together in synchronous operation is also turned off.

- Slave axis mirror image

By setting parameter No. 8312, a mirror image can be applied to a slave axis placed in synchronous operation. When the mirror image function is enabled, the direction in which the absolute and relative coordinates change is the same as for the machine coordinates.
At this time, synchronization establishment, synchronization error check, and correction mode cannot be used.
The mirror image set by bit 0 (MIR) of parameter No. 0012 cannot be applied to the slave axis. Because this mirror image differs from the mirror image set by parameter MIR, it does not affect input signal MIx <G106> or output signal MMIx <F108>.

- External machine coordinate system shift

Bit 7 (SYE) of parameter No. 8304 can be set to 1 for the slave axis to shift the slave axis by the same amount as specified for the master axis when external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control.

- Manual operation to slave axis

The move command cannot be performed to the slave axis in axis synchronous control with manual operation (JOG feed, HANDLE feed, etc.).

17.1.2 Synchronous Establishment

Explanation

Upon power-up or after emergency stop cancellation, the machine positions on the master axis and slave axis under axis synchronous control are not always the same. In such a case, the synchronous establishment function matches the machine position on the master with that on the slave axis.

- Synchronous establishment based on machine coordinates

Enable synchronous establishment based on machine coordinates by setting bit 7 (SOF) of parameter No. 8303 to 1. This method of synchronous establishment outputs the machine coordinate difference between the master axis and slave axis as command pulses for the slave axis to establish synchronization. A machine coordinate difference is output at a time as command pulses. So, if the compensation value is large, the machine abruptly makes a large movement. Taking this into consideration, set a maximum allowable compensation value to be used for synchronous establishment in parameter No. 8325. As a maximum allowable compensation value, set a maximum allowable value by which the machine may move abruptly. If a compensation value is larger than the value set in this parameter, an alarm SV0001 is issued, and synchronous establishment is not performed. Moreover, when parameter No. 8325 is set to 0, synchronous establishment is not performed.
The result of comparing the positional difference between the master axis and slave axis with a maximum allowable compensation value for synchronous establishment can be checked using the synchronous establishment enable state output signal SYNOF <F0211>.

- First synchronous establishment after power-up

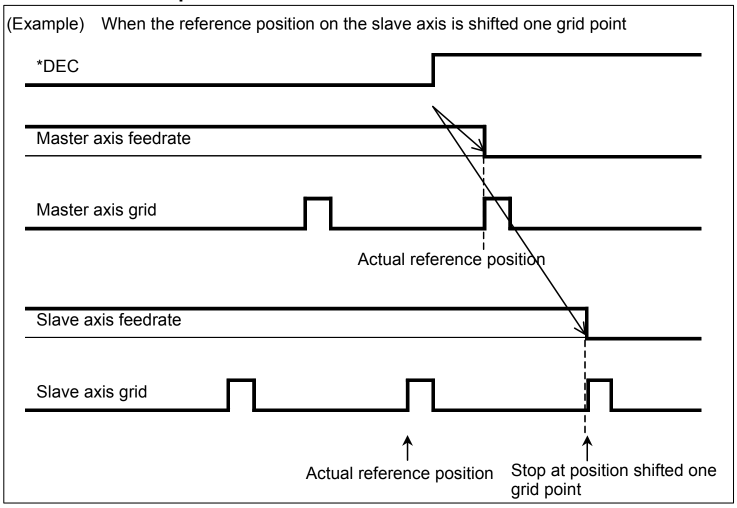
Two methods of performing the first synchronous establishment after power-up are available. One method is based on manual reference position return operation, and the other is based on absolute position detection.
A synchronization error value is checked until this synchronous establishment is completed.

- Synchronous establishment based on manual reference position return operation

When manual reference position return operation is performed along axes under axis synchronous control, the machine is placed at the reference position on the master axis and slave axis according to the same sequence as for normal reference position return operation.
The sequence is the same as the grid method for one axis only. However, only the deceleration signal for the master axis is used. When the deceleration signal is set to 0, the machine gradually stops along the master axis and slave axis, then an FL feedrate is set. When the deceleration signal is set to 1, the machine moves to a grid point along each of the master axis and slave axis, then stops.

NOTE

When the grid position difference between the master axis and slave axis is large, a reference position shift can occur, depending on the timing of the *DEC signal set to 1. In the example below, the shift along the slave axis is so large that the position shifted one grid point from the actual reference position is regarded as the reference position.



In such a case, match the grid position according to Subsection 17.1.3, "Automatic Setting for Grid Position Matching."

- Synchronous establishment based on absolute position detection

When an absolute-position detector is used as the position detector, the machine positions on the master axis and slave axis are found at power-up time for automatic establish synchronization.

- Synchronous establishment after emergency stop cancellation, etc.

Synchronous establishment is also performed when servo position control is turned on, for example, at emergency stop cancellation, servo alarm cancellation, or servo-off cancellation time.
However, synchronous establishment is not performed at the time of axis removal cancellation. So, synchronous establishment based on manual reference position return operation is required as in the case of power-up time.

- One-direction synchronous establishment

Synchronous establishment can be performed by setting bit 0 (SSO) of parameter No. 8305 to 1 to move the machine in one direction along the master axis and slave axis. The move direction depends on the reference position setting based on bit 0 (SSA) of parameter No. 8304. When SSA = 0, for example, the machine coordinate on the master axis or slave axis, whichever larger, is used as the reference point. So, the machine moves in the + direction along the axes.
When bit 1 (SSE) of parameter No. 8305 is set to 1, normal synchronous establishment is performed instead of one-direction synchronous establishment after an emergency stop.

17.1.3 Automatic Setting for Grid Position Matching

Explanation

Before axis synchronous control can be performed, the reference position on the master axis must be matched with the reference position on the slave axis. With this function, the CNC automatically matches the reference positions (grid positions) on the master axis and slave axis under axis synchronous control.

[Operation procedure]

The procedure below is usable when bit 0 (ATE) of parameter No. 8303 is set to 1.

1. Set bit 1 (ATS) of parameter No. 8303 to 1.
2. Turn off the power then turn on the power.
3. Set the REF mode (or JOG mode in the case of reference position setting without dogs) when synchronous operation is ready, and make movements in the reference position return direction along the master axis and slave axis.
4. The movements along the master axis and slave axis automatically stop, and a grid difference value is set in parameter No. 8326. At this time, bit 1 (ATS) of parameter No. 8303 is set to 0, and the power-off request alarm PW0000 is issued.
5. Turn off the power then turn on the power again.
6. Perform normal reference position return operation.

NOTE

1 Parameter setting
When bit 1 (ATS) of parameter No. 8303 is set, bit 4 (APZ) of parameter No. 1815 and parameter No. 8326 for the master axis and slave axis are set to 0.
When the operator sets parameter No. 8326 (MDI, G10L50), bit 0 (ATE) of parameter No. 8303 is set to 0.

2 This function cannot be used together with the reference position shift function.

17.1.4 Synchronous Error Check

Explanation

A synchronous error value is monitored at all times. If an error exceeding a certain limit is detected, an alarm is issued and the movement along the axis is stopped.

A synchronous error check based on machine coordinates and a synchronous error check based on a positional deviation value are performed.

- Synchronous error check based on machine coordinates

A synchronous error check based on machine coordinates is made.

The machine coordinate on the master axis is compared with that on the slave axis. When the error between the machine coordinates exceeds the value set in parameter No. 8314, the SV0005 alarm is issued, and the motor is stopped immediately.

A check can be made even in the emergency stop, servo off, and servo alarm states.

A synchronous error check is performed during normal operation as well as during synchronous operation. So, even if the axis synchronous control selection signal (SYNCx) or the axis synchronous control manual feed selection signal (SYNCJx) is set to 0 by mistake during synchronous operation, damage to the machine can be prevented.

The machine coordinates on the master axis and slave axis can be checked using the machine coordinate match state output signal SYNMT <F0210>.

- Synchronous error check based on a positional deviation value

The servo positional deviation value of the master axis and slave axis is monitored during axis synchronous control. When the positional deviation value exceeds the limit value set in parameter No. 8323, the DS0001 alarm is issued, and the axis synchronous control positional deviation error alarm signal <F403.0> is output.

The DS0001 alarm is issued to the master axis and slave axis.

When bit 4 (SYA) of parameter No. 8301 is set to 1, the positional deviation limit value of the master axis and slave axis is checked even if a servo-off occurs during axis synchronous control.

17.1.5 Methods of Alarm Recovery by Synchronous Error Check

Explanation

To recover from an alarm issued as a result of synchronous error check, two methods are available. One method uses the correction mode, and the other uses normal operation.
If the mode of operation is switched between synchronous operation and normal operation by using an input signal, only the method using normal operation can be used.
If synchronous operation is used at all times, only the method using the correction mode can be used.

- Procedure for correcting a synchronous error by using the correction mode

Use this method if synchronous operation is used at all times without using an input signal (when bit 5 (SCA) of parameter No. 8304 is set to 1).
When the correction mode is used, synchronous error check can be temporarily disabled, and a movement can be made along the master axis or slave axis to correct a synchronous error.
In the correction mode, error check are not performed, so that an alarm DS0003 is issued as a warning.

1. Select the correction mode, and select an axis along which a movement is to be made by manual master axis feed. Set bit 2 (ADJ) of parameter No. 8304 of the master axis or slave axis to 1 to set the correction mode. Thus, by manual master axis feed, a movement can be made along the axis with this parameter set to 1.
   When this parameter is set to 1, the DS0003 (axis synchronous control correction mode) alarm is issued.
2. Reset the synchronous error excessive alarm.
   In this state, error check are not performed. Be careful.
3. Select the manual mode (jog, incremental feed, or handle).
4. While checking the synchronous error value, make a movement along the master axis or slave axis in the direction that reduces the error.
   If one master axis has multiple slave axes, an attempt to reduce the synchronous error of one slave axis by master axis movement may increase the synchronous error of another slave axis, thus disabling a movement in any direction. In such a case, by setting bit 4 (MVB) of parameter No. 8304 to 1, a movement can be made in a direction that increases the synchronous error.
5. When the synchronous error is reduced to within the allowable value for suppressing the alarm, reset the value of bit 2 (ADJ) of parameter No. 8304 to the original value to switch from the correction mode to the normal synchronization mode.
   Synchronous error check are restarted.
6. Reset the correction mode alarm.

- Method of recovery using normal operation

Use this method when switching between synchronous operation and normal operation by using an input signal.
Use the procedure below for recovery from alarm SV0005.

1. Set SYNCx/SYNCJx (with x representing a slave axis number) to 0 to select normal operation.
2. Set a value greater than the current value in the parameter No. 8314 for specifying a maximum allowable synchronous error, then reset the alarm.
3. Make a movement along the master axis or slave axis by using the manual handle so that the machine coordinates of the master axis and slave axis match to a maximum possible extent.
4. Return the value of parameter No. 8314 for specifying a maximum allowable synchronous error to the original value.

17.1.6 Axis Synchronous Control Torque Difference Alarm

Explanation

If a movement made along the master axis differs from a movement made along the slave axis during axis synchronous control, the machine can be damaged. To prevent such damage, the torque command difference between the two axes is observed. If the difference is abnormal, a servo alarm SV0420 can be issued.

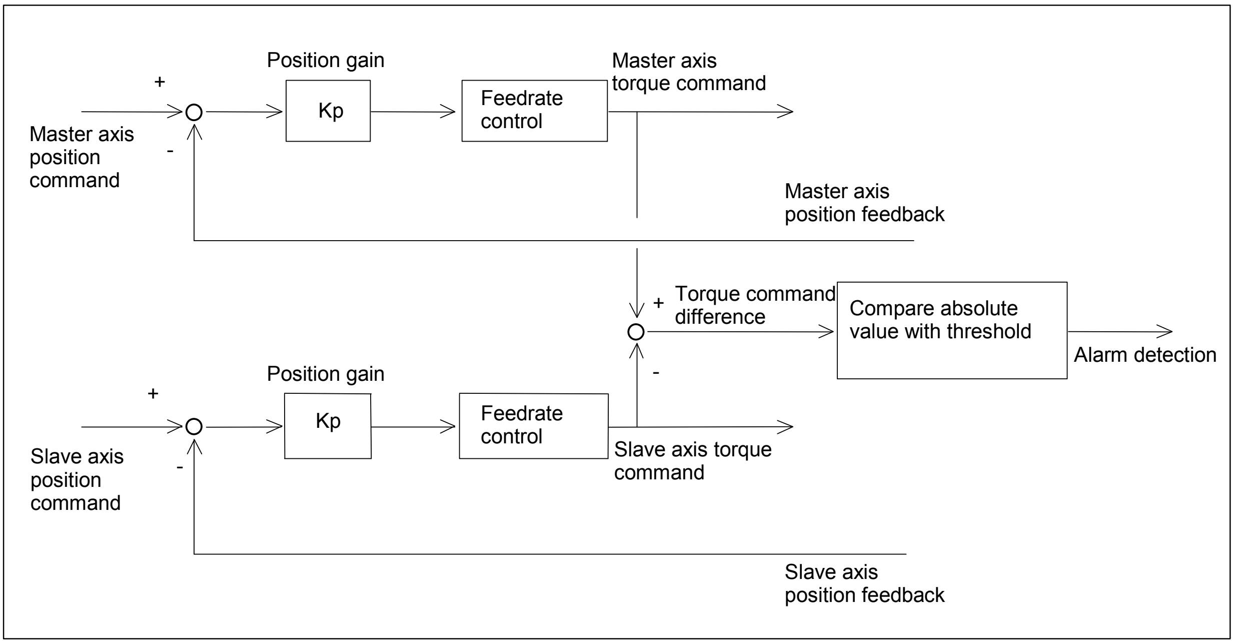
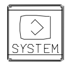


Fig. 17.1.6 (a) System configuration

[Method of use]

Specify the threshold parameter No. 2031 according to the procedure below.

1. Set 0 in parameter No. 2031, and disable the torque difference alarm detection function.
2. To check the absolute value of the torque difference between the synchronous axes, set the parameters below. Set the same value for the two axes placed under axis synchronous control.
   Parameter No. 2115 = 0
   Parameter No. 2151 is as described below.
   - For the T series (2-path control system), set it to 434 if the setting of parameter No. 1023 is 1, 2, 5, 6, 9, 10… and to 6578 if it is 3, 4, 7, 8, 11, 12…
   - For 1-path control system, set it to 434.
3. Display the diagnostic screen by pressing the function key  then the [DGNOS] soft key.
   Diagnose No. 0353 indicates the absolute value of the torque difference between the two axes.
4. Read the absolute torque difference value presented when normal operation is being performed. In the threshold parameter No. 2031, set a value obtained by adding some margin to the read absolute value.
   The absolute torque difference value can be observed with the Servo Guide.

- Enabling/disabling of alarm detection

Alarm detection is enabled when the time set in parameter No. 8327 has elapsed after the servo ready signal SA <F000.6> is set to 1. When the input signal NSYNCA <G059.7> is set to 1, alarm detection is disabled.

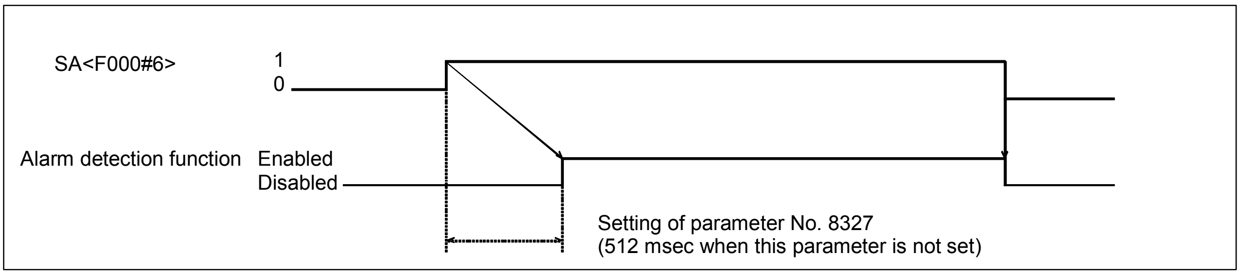


Fig. 17.1.6 (b) Timing chart

When the servo ready signal SA <F000.6> is set to 0, torque difference alarm detection is disabled.

NOTE

The servo axis number combination of the master axis and slave axis synchronized with each other must be such that an odd servo axis number is assigned to the master axis and the next servo axis number is assigned to the slave axis like (1,2) and (3,4).

⚠ CAUTION

1 When making a synchronous error check, ensure that the reference position on the master axis and the reference position on the slave axis must be at the same position.
2 In manual reference position return operation, the same operation is performed along the master axis and slave axis until a deceleration operation starts. After a deceleration operation starts, grid detection is performed for the master axis and slave axis independently of each other.
3 Pitch error compensation and backlash compensation are performed for the master axis and slave axis independently of each other.

NOTE

1 During axis synchronous control, a movement based on the reference position return check (G27), automatic reference position return (G28), 2nd/3rd/4th reference position return (G30), or machine coordinate system selection (G53) command is made as described below according to the setting of bit 7 (SRF) of parameter No. 8304.
<1> When SRF = 0, the same movement as made along the master axis is made along the slave axis.
<2> When SRF = 1, a movement is made along the slave axis to the specified position independently of a movement made along the master axis to the specified position.
2 A command not involving a movement along an axis such as the workpiece coordinate system setting command and local coordinate system setting command is set with the master axis according to the master axis programming.
3 During synchronous operation, the signals provided for each axis, such as the external deceleration, interlock, and machine lock signals, are enabled only on the master axis side and ignored on the slave axis side.
4 When switching the synchronization state in a program, be sure to specify M codes (parameter No. 8337 and No. 8338) for turning synchronization on and off. By switching between the input signals SYNCx <G138> and SYNCJx <G140> from the PMC with the M codes, the synchronization state can be switched in the program.

NOTE

5 When controlled axis removal is performed, the synchronization state is cancelled. When performing controlled axis removal, perform removal for the master axis and slave axis at the same time.

6 If a programmed command is specified for the slave axis during synchronous operation, an alarm PS0213 is issued.
A programmed command can be specified for the slave axis when switching between synchronous operation and normal operation is set to 0 (with bit 5 (SCA) of parameter No. 8304 set to 0) to select normal operation.

7 Axis synchronous control and PMC axis control cannot be used at the same time.

17.2 ROTARY AXIS ROLL-OVER

Overview

The roll-over function prevents coordinates for the rotary axis from overflowing. The roll-over function is enabled by setting bit 0 (ROAx) of parameter No. 1008 to 1.

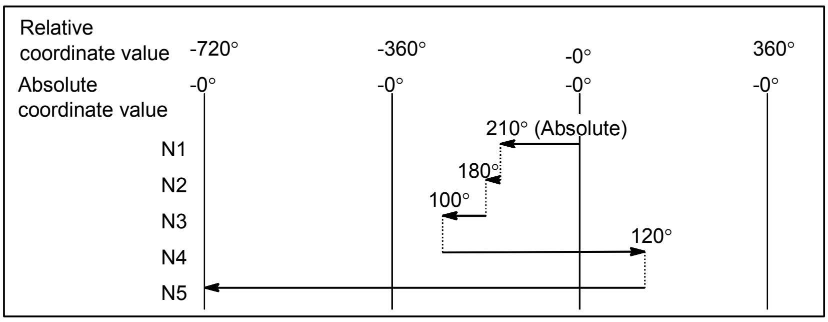
Explanation

For an incremental programming, the tool moves the angle specified in the command. For an absolute programming, the coordinates after the tool has moved are values set in parameter No. 1260, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when bit 1 (RABx) of parameter No. 1008 is set to 0. Relative coordinates can be rounded to the angle corresponding to one rotation by setting bit 2 (RRLx) of parameter No. 1008 is set to 1.

Example

Assume that axis A is the rotary axis and that the amount of movement per rotation is 360.000 (parameter No. 1260). When the following program is executed using the roll-over function of the rotary axis, the axis moves as shown below.

G90 A0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 G90 A-150.0 ;	N1	-150	210
N2 G90 A540.0 ;	N2	-30	180
N3 G90 A-620.0 ;	N3	-80	100
N4 G91 A380.0 ;	N4	+380	120
N5 G91 A-840.0 ;	N5	-840	0



M

> **NOTE**
> This function cannot be used together with the index table indexing function.

17.3 ARBITRARY ANGULAR AXIS CONTROL

Overview

When the angular axis installed makes an angle other than 90° with the perpendicular axis, the Arbitrary 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. The coordinate system used at this time is referred to as the program coordinate system. (The program coordinate system may be referred to as the Cartesian coordinate system, and the actual move coordinate system may be referred to as the angular coordinate system or machine coordinate system.)

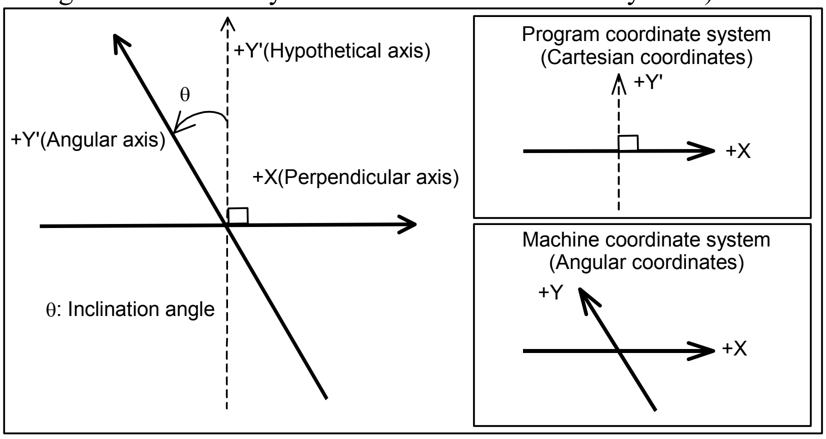


Fig. 17.3 (a)

Explanation

When the amounts of travel along the angular axis and the perpendicular axis are Ya and Xa, respectively, the amounts are controlled according to the formulas shown below.

$$Ya = \frac{Yp}{\cos\theta}$$

Xa,*Ya*: Actual distance
Xp,*Yp*: Programmed distance

The amount of travel along the perpendicular axis is corrected by the influence of travel along the angular axis, and is determined by the following formula:

$$Xa = Xp - C \times Yp \times \tan\theta$$

> **NOTE**
> The coefficient C is 1/2 in the case of diameter specification for the perpendicular axis (X) or 1 in the case of radius specification.

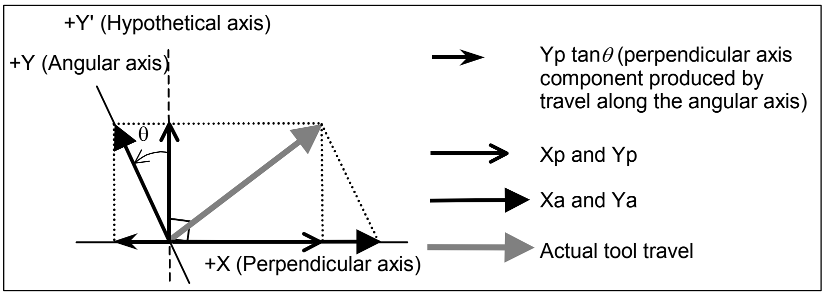


Fig. 17.3 (b)

- Feedrate

When the Y-axis is an angular axis, and the X-axis is a perpendicular axis, the feedrate along each axis is controlled as described below so that the feedrate in the tangent direction becomes Fp.
The feedrate component along the Y-axis is determined by the following expressions:

$$Fay = \frac{Fp}{\cos\theta}$$

Fa represents the actual feedrate.
Fp represents a programmed feedrate.

$$Fax = Fp - Fp \times \tan\theta$$

- Absolute and relative position display

An absolute and a relative position are indicated in the programmed Cartesian coordinate system.

- Machine position display

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle.

Method of use

The angular and perpendicular axes for which arbitrary angular axis control is to be applied must be specified beforehand, using parameters Nos. 8211 and 8212. When 0 is set in one of the parameters, the same number is specified in the parameters, or a number other than the controlled axis numbers is specified in a parameter, however, an angular axis and perpendicular axis are selected according to the table below.

	Angular axis	Perpendicular axis
M series	Y-axis of the basic three axes (axis with 2 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)
T series	X-axis of the basic three axes (axis with 1 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)

- Bit 0 (AAC) of parameter No. 8200 enables or disables the arbitrary angular axis control. If the function is enabled, the distance traveled along each axis is controlled according to an angular angle parameter No. 8210.
- By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis by a movement made along the angular axis when a manual reference position return operation is performed along the angular axis can be chosen.
- By setting the normal axis/angular axis control invalid signal NOZAGC to 1, angular axis control only for the angular axis can be available. In this time the angular axis are converted to those along the angular coordinate system without affecting commands to normal axis.
  Use this signal when operating each axis independently.

- Manual reference position return operation

A movement is made to the reference position (machine position) set in parameter No. 1240. By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis when a reference position return operation is performed along the angular axis can be chosen.

- Automatic reference position return operation (G28, G30)

A movement to the middle point along the angular axis affects a movement along the perpendicular axis. As a movement from the middle point to the reference position along the angular axis, a selection can be made with bit 0 (ARF) of parameter No. 8209 between a Cartesian coordinate system operation (FS0*i*-C compatibility) and an angular coordinate system operation. If manual reference position return operation is not performed even once after the power is turned on, operation is performed in the same sequence as for manual reference position return operation. So, specify commands first for the angular axis then for the perpendicular axis.

Example 1)

When the Y-axis is an angular axis and the X-axis is a perpendicular axis

(1) If the angular axis is first specified then the perpendicular axis is specified, reference position return operation is performed normally.
G28 Y_ ;
G28 X_ ;

(2) If the perpendicular axis is first specified then the angular axis is specified, or if the perpendicular axis and the angular axis are specified at the same time, alarm PS0372 is issued when a movement is made along the perpendicular axis.

$\begin{cases} \text{G28X\_;} \\ \text{G28Y\_;} \end{cases}$ or $\begin{cases} \text{G28X\_Y\_;} \\ \end{cases}$

Example 2)

Automatic reference position return examples
(If the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is –30. This example assumes that the reference position is already established once.)

<1> Command for automatic reference position return along the Y-axis from point P2
>G91 G28 X200. ;

<2> Command for automatic reference position return along the X-axis from point P1
>G91 G28 Y100. ;

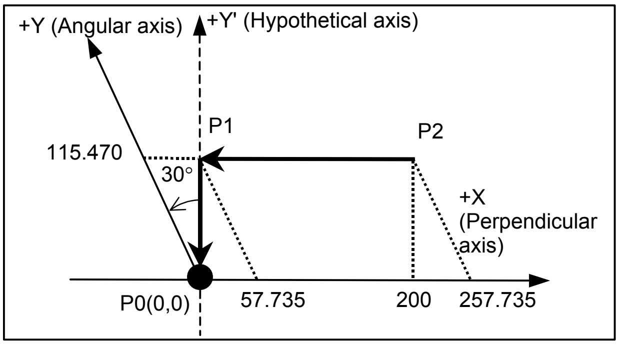
(1) If bit 0 (ARF) of parameter No. 8209 is 1 (FS0*i*-C compatibility)

<1> Coordinates at P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	57.735
Y	100.000	Y	115.470

<2> Coordinates at P0

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	0.000	Y	0.000



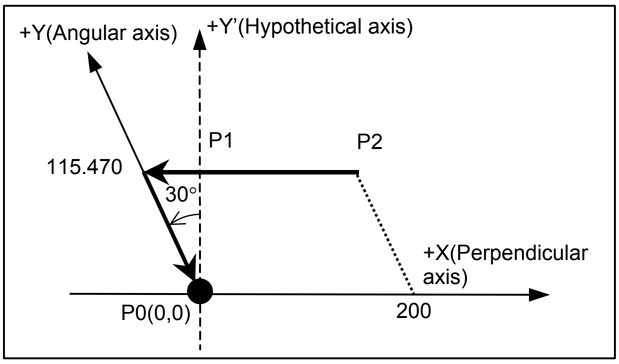
(2) If bit 0 (ARF) of parameter No. 8209 is 0

<1> Coordinates at P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	100.000	Y	115.470

<2> Coordinates at P0

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	0.000	Y	0.000



- Reference position return operation of high-speed type

When a reference position is already established and a reference position return operation of high-speed type is to be performed, the reference position return operation need not be performed in the order from the angular axis to the perpendicular axis.

- Machine coordinate selection (G53)

By specifying (G90)G53X_Y_:, a movement is made by rapid traverse.
However, a movement along the angular axis (G53 command) does not affect a movement along the perpendicular axis, regardless of whether the perpendicular axis/angular axis control disable signal (NOZAGC) is turned on or off.
Example)

(when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)

1 Move command for movement from point P0 to point P1
>G90G53Y100.

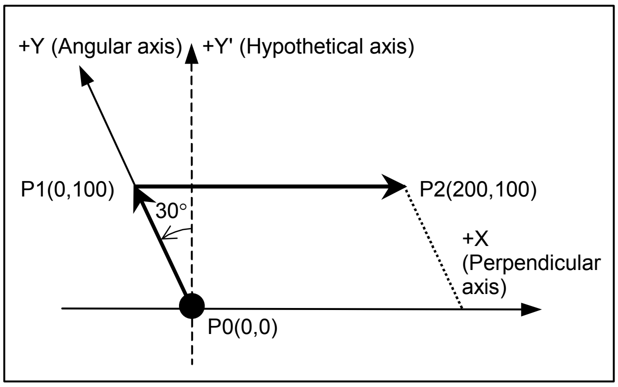
2 Move command for movement from point P1 to point P2
>G90G53X200.

<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	-50.000	X	0.000
Y	86.603	Y	100.000

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	150.000	X	200.000
Y	86.603	Y	100.000



- Commands for linear interpolation and linear interpolation type positioning (G01, G00)

The tool moves to a specified position in the Cartesian coordinate system when the following is specified:
(G90)G00X_Y_;
or
(G90)G01X_Y_F_;

Example)
Examples of positioning
(when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)

1 Move command for movement from point P0 to point P1
> G90 G00 Y100. ;
2 Move command for movement from P1 to P2
> G90 G00 X200. ;

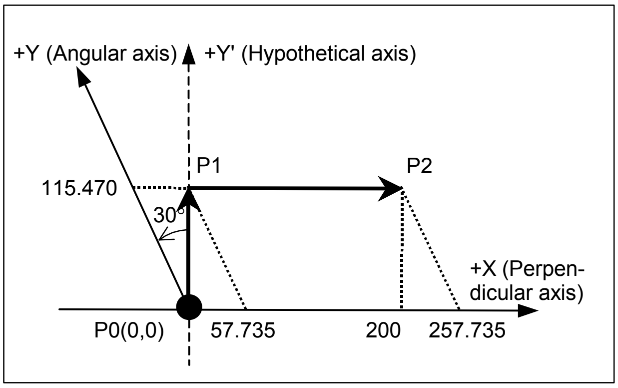
(1) When the perpendicular axis/angular axis control disable signal (NOZAGC) is set to 0

<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	57.735
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	200.000	X	257.735
Y	100.000	Y	115.470



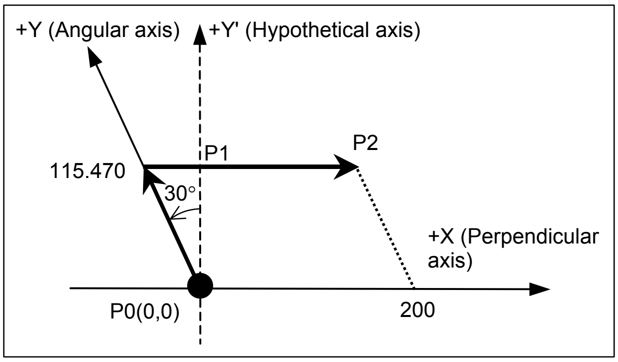
(2) When the perpendicular axis/angular axis control disable signal (NOZAGC) is set to 1

<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	200.000	X	200.000
Y	100.000	Y	115.470



- Stored stroke limit

Stored stroke limits under arbitrary angular axis control can be set not in a angular coordinate system but in the Cartesian coordinate system by setting bits 2, 1, and 0 (AO3, AO2, and AOT) of parameter No. 8201.

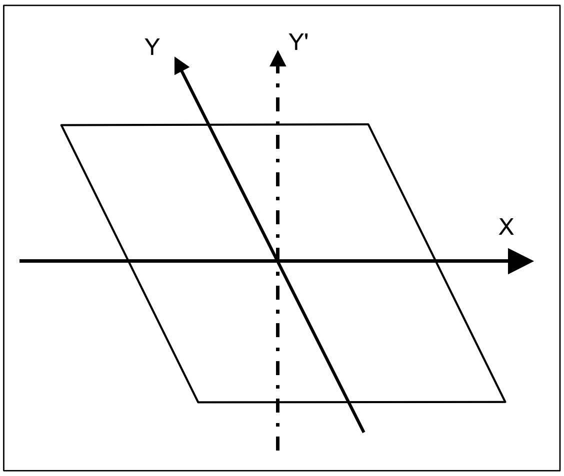


Fig. 17.3 (c) OT area in a angular coordinate system

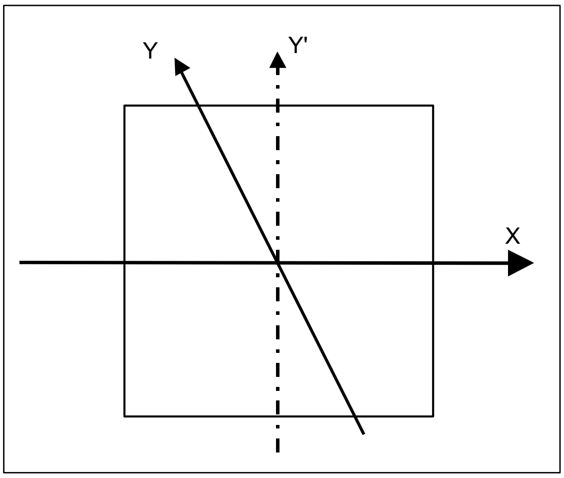


Fig. 17.3 (d) OT area in a Cartesian coordinate system

Machine coordinates include a value converted for the angular axis and a compensation value for the perpendicular axis, so that a angular machine coordinate system as shown in Fig. 17.3 (c) results.
A stored stroke limit is checked in the machine coordinate system, so that the limit area is slanted to form a rhombus as shown in Fig. 17.3 (c). In this case, the area cannot be identified intuitively. So, stroke limits are checked not in an actual angular machine coordinate system but in a virtual Cartesian machine coordinate system as shown in Fig. 17.3 (d).

The functions that operate in the Cartesian coordinate system are:

- Stored stroke check 1 (Both of I and II)
- Stored stroke check 2 (G22/G23)
- Stored stroke check 3
- Stored stroke check before move
  The stored stroke check function before move does not work in a angular coordinate system. Unless this function is enabled, and the coordinate system is converted to the Cartesian coordinate system, no stroke check is made.
- Bit 7 (BFA) of parameter No. 1300 for specifying whether to issue an alarm before or after a stroke limit is exceeded (valid for OT1 and OT3)

M

- Stroke limit external setting (valid only for OT1)

The stored stroke limit functions other than the above work in a angular coordinate system.

- Relationships between this function and axis-by-axis input/output signals

The table below indicates the relationships between this function and the meaning of each controlled axis signal.
The input/output signals are classified as signals valid for the program coordinate system (Cartesian coordinate system) and signals valid for the machine coordinate system (angular coordinate system). In the "Classification" column, "Cartesian" is indicated for a signal that is valid for the Cartesian coordinate system, and "Angular" is indicated for a signal that is valid for the angular coordinate system.
A signal valid for the Cartesian coordinate system means a signal valid for a specified axis, and a signal valid for the angular coordinate system is a signal valid for actual machine movement.
That is, when the perpendicular axis is moved by a command only for the angular axis:
A signal valid for the Cartesian coordinate system is affected by a movement along the angular axis.
A signal valid for the angular coordinate system is not affected by a movement along the angular axis.

Input signal				
Signal name		Address	Classification	Remarks
Interlock for each axis	*ITx	G130	Cartesian	When a movement is made along the angular axis only, interlocking the perpendicular axis does not interlock a movement along the perpendicular axis made by a movement along the angular axis. Caution) When using the interlock signal for each axis, make both of the angular axis and perpendicular axis high.
Overtravel	*+Lx *-Lx	G114 G116	Angular	This signal is applied to each axis independently. (If the perpendicular axis is made high, no alarm is issued for the perpendicular axis even when an OT alarm is issued for the angular axis.)
Deceleration signal for reference position return	*DECx	X009	Angular	This signal is applied to each axis independently.
Servo-off signal	SVFx	G126	Angular	This signal is applied to each axis independently.
Control axis detach signal	DTCHx	G124	Angular	This signal is applied to each axis independently.
Feed axis direction selection signal	+Jx -Jx	G100 G102	Cartesian	A movement is made in the Cartesian coordinate system. (When the +J/-J signal for the angular axis is made high, a movement is made also along the perpendicular axis.)
Mirror image	MIx	G106	Angular	Mirror image is applied to the angular coordinate system for each axis independently. Caution) Be sure to turn off the mirror image signal for the angular axis and perpendicular axis engaged in manual operation.
Manual feed interlock signal for each axis direction, tool compensation value write signal	+MIT1, +MIT2	X004.2, 4	Cartesian	Set the tool compensation parameter in the Cartesian coordinate system.
Machine lock for each axis	MLKx	G108	Angular	This signal is applied to each axis independently.

Output signal				
Signal name		Address	Classification	Remarks
In-position signal	INPx	F104	Angular	Applied to each axis independently.
Mirror image check signal	MMIx	F108	Angular	Applied to each axis independently.
Controlled axis removal in-progress signal	MDTCHx	F110	Angular	Applied to each axis independently.
Travel in-progress signal	MVx	F102	Angular	Applied to each axis independently.
Reference position return completion signal	ZPx	F094	Cartesian	Applied to each axis independently. (A manual reference position return operation and the first automatic reference position return operation after power-up need to be performed first for the angular axis.)
2nd reference position return completion signal	ZP2x	F096	Cartesian	Applied to each axis independently.
3rd reference position return completion signal	ZP3x	F098	Cartesian	Applied to each axis independently.
4th reference position return completion signal	ZP4x	F100	Cartesian	Applied to each axis independently.

Limitation

- Linear scale with absolute address reference mark

- For both of the angular axis and perpendicular axis, a linear scale with an absolute address reference mark must be used.
- Reference position return operation must be first completed along the angular axis.
- Return operation cannot be performed along the perpendicular axis while return operation is being performed along the angular axis.

T

- Synchronous control

For synchronous control on axes related to arbitrary angular axis control, the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side must be placed under synchronous control at the same time. Moreover, synchronous control can be exercised between angular axes only or between Cartesian axes only.
If an attempt is made to perform operation under a condition other than the above, the alarm PS0375 is issued.
Example)

Path 1		Path 2
X1 (Cartesian axis)	←Synchronous→	X2 (Cartesian axis)
Y1 (angular axis)	←Synchronous→	Y2 (angular axis)

- Composite control

For composite control on axes related to arbitrary angular axis control, the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side must be placed under composite control at the same time. Moreover, composite control can be exercised between angular axes only or between Cartesian axes only.
If an attempt is made to perform operation under a condition other than the above, the alarm PS0375 is issued.
Example)

Path 1		Path 2
X1 (Cartesian axis)	←composite→	X2 (Cartesian axis)
Y1 (angular axis)	←composite→	Y2 (angular axis)

- Rigid tapping

As a rigid tapping axis, no angular axis can be used.

- Functions that cannot be used simultaneously

- Axis synchronous control, rigid tapping, PMC axis control

T

- Polygon turning, superimposed control

M

- Electronic gear box function

> ⚠ **CAUTION**
>
> 1 After arbitrary angular axis control parameter setting, be sure to perform manual reference position return operation.
> 2 Before manual reference position return operation is performed along the perpendicular axis, reference position return operation along the angular axis must be completed (with the reference position return completion signal for the angular axis (ZPx) set to 1). If reference position return operation is performed along the perpendicular axis first, an alarm PS0372 is issued.
> 3 When the setting is made so that the tool moves along the perpendicular axis during manual reference position return along the angular axis (bit 2 (AZK) of parameter No. 8200 is set to 0), if once manual reference position return has been performed along the angular axis, also perform manual reference position return along the perpendicular axis immediately after the operation.
> 4 To move the perpendicular axis and the angular axis independently for each other during manual operation, set perpendicular/angular axis control disable signal NOZAGC to 1.
> 5 Once the tool has been moved along the angular axis when perpendicular/angular axis control disable signal NOZAGC has been set to 1, manual reference position return must be performed.
> 6 The same increment system must be used with the angular axis and perpendicular axis.
> 7 Before a perpendicular axis reference position return check can be made, angular axis reference position return operation must be completed.
> 8 No rotary axis must be set for the angular axis and perpendicular axis. A rotary axis may be specified only for a linear axis.
> 9 Set a position switch operation range (parameter Nos. 6930 to 6965) in a angular coordinate system.

17.4 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis. Positioning is performed by the main motor only. The submotor is used only to produce torque. With this tandem control function, the torque produced can be doubled.

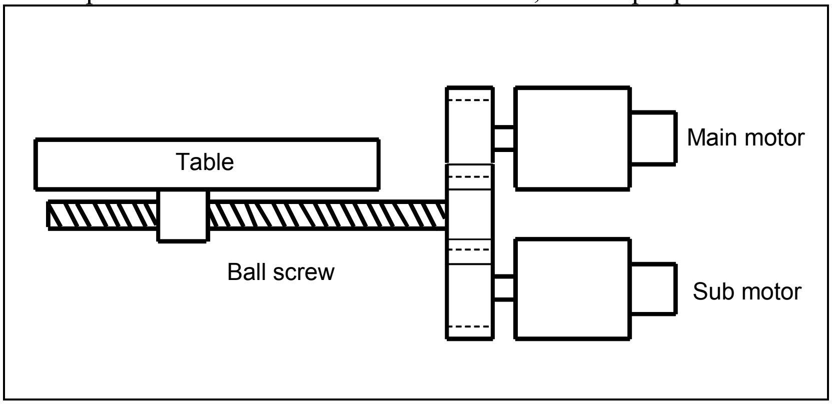


Fig. 17.4 (a) Example of operation

In general, the NC regards tandem control as being performed for one axis. However, for servo parameter management and servo alarm monitoring, tandem control is regarded as being performed for two axes.
For details, refer to the relevant manual published by the machine tool builder.

18 PATTERN DATA INPUT

Chapter 18, "PATTERN DATA INPUT", consists of the following sections:

18.1 OVERVIEW

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.

18.2 EXPLANATION

This function is consist of Pattern menu screen and Custom macro screen.
The process pattern is selected on the pattern menu screen.
Then the process pattern is selected, the custom macro screen is displayed.
On this custom macro screen, the variable with the name and comment is displayed according to the selected process pattern.
The process data can be input by referring to the variable name with the numerical value on the drawing.

Bit 7 (NPD) of parameter No. 8135 can be set to enable or disable this function (0: enabled, 1: disabled).
When this function is disabled, the above screens are not displayed.

The following is the example for the pattern menu and the custom macro.

(1) Pattern menu screen

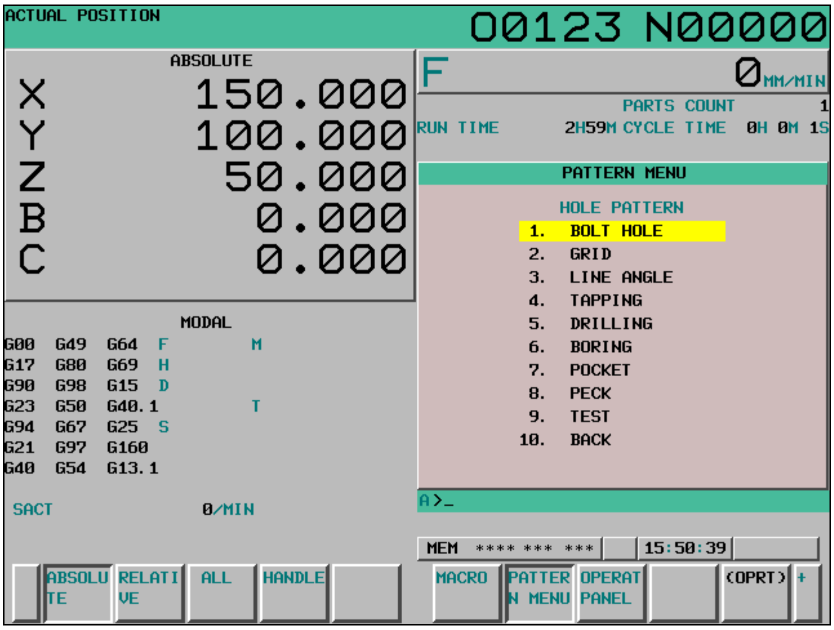


Fig. 18.2 (a) Pattern data menu screen (10.4-inch)

(2) Custom macro screen

The name of variable and comment can be displayed on the usual custom macro screen.

The menu title and pattern name on the pattern menu screen and the variable name on the custom macro screen can be defined

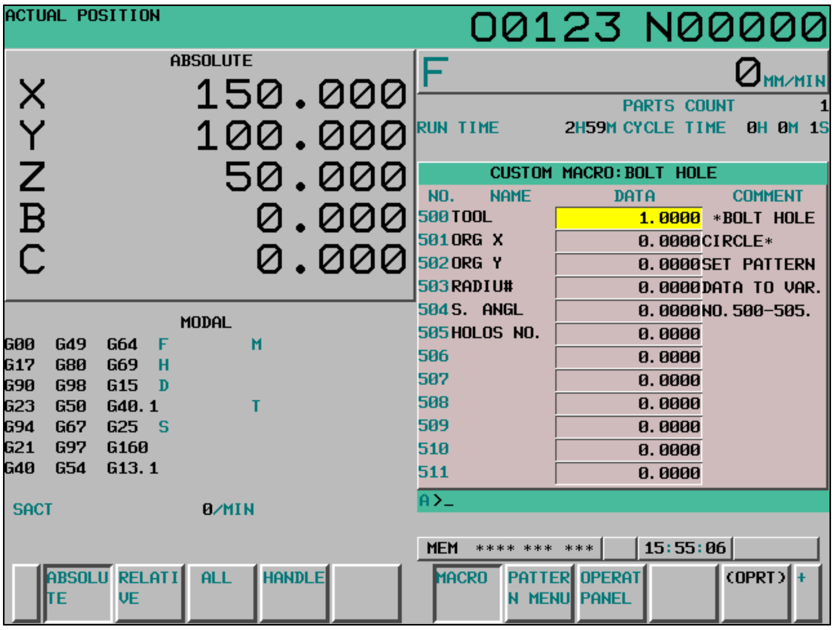


Fig. 18.2 (b) Custom macro screen (10.4-inch)

18.3 EXPLANATION OF OPERATION

The following explains how to display the pattern menu screen.

1 Press function key .
2 Press continuous menu key .
3 Press soft key [PATTERN MENU] ([MENU] for the 8.4-inch display unit).

Pattern menu screen

The following pattern menu is displayed.

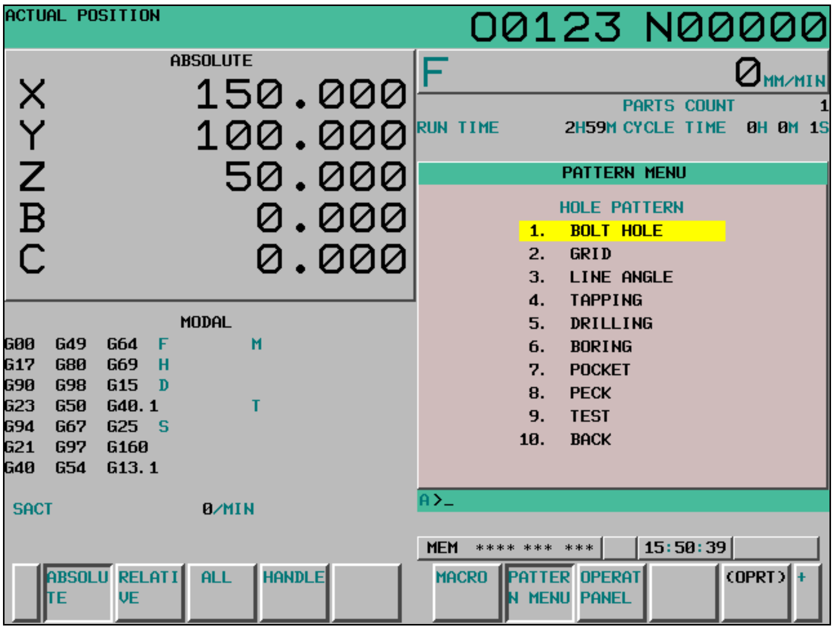


Fig. 18.3 (a) Pattern menu screen (10.4-inch)

Select the pattern on this screen
The following two methods are effective.

- Selection by cursor

  Move the cursor to the pattern name with the cursor move keys , and press the soft key [SELECT] or key.

- Selection by setting of pattern number

  The number that is displayed in the left side of the pattern name is input and press the soft key [select] or key.

The selected pattern number is registered to system variable #5900. The custom macro of the selected pattern can be started by starting a fixed program (external program No. search) with an external signal. This program refers to the system variable #5900 in the program. This system variable #5900 is kept after power-off.

Custom macro variable screen

The following custom macro screen is displayed.

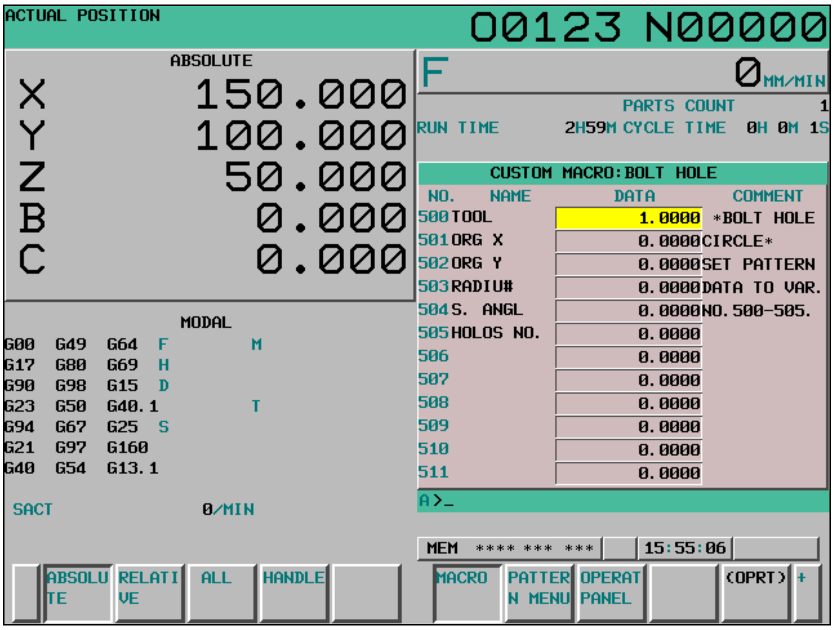


Fig. 18.3 (b) Custom macro screen when the pattern data is input (10.4-inch)

When the screen is changed to the custom macro screen, the macro variable number that is selected first is specified with the parameters Nos.6101 to 6110. The macro variables that variable name is not defined can be input, too.

> **NOTE**
> 1 The variable name that is displayed cannot be used as the common variable name of the NC program.
> 2 When the common variable name is defined by SETVN command, the variable name defined by pattern data input function is given priority.

18.4 DEFINITION OF THE SCREEN

The definition of the screen is performed by NC program.

Program configuration

This function is consist of one program for the definition of pattern menu screen and maximum ten programs for the definition of custom macro screen.
The program number is as follows

Table 18.4 (a) Numbers of subprograms employed in the pattern data input function

Sub program No.	Screen
O9500	Specifies character strings displayed on the pattern data menu.
O9501	Specifies a character string of the pattern data corresponding to pattern No.1
O9502	Specifies a character string of the pattern data corresponding to pattern No.2
O9503	Specifies a character string of the pattern data corresponding to pattern No.3
O9504	Specifies a character string of the pattern data corresponding to pattern No.4
O9505	Specifies a character string of the pattern data corresponding to pattern No.5
O9506	Specifies a character string of the pattern data corresponding to pattern No.6
O9507	Specifies a character string of the pattern data corresponding to pattern No.7
O9508	Specifies a character string of the pattern data corresponding to pattern No.8

Sub program No.	Screen
O9509	Specifies a character string of the pattern data corresponding to pattern No.9
O9510	Specifies a character string of the pattern data corresponding to pattern No.10

Table 18.4 (b) Macro commands used in the pattern data input function

G code	H code	Function
G65	H90	Specifies the menu title.
G65	H91	Specifies the pattern name.
G65	H92	Specifies the pattern data title.
G65	H93	Specifies the variable name.
G65	H94	Specifies the comment.

Table 18.4 (c) System variables employed in the pattern data input function

System variable	Function
#5900	Pattern No. selected by user.

18.4.1 Definition of the Pattern Menu Screen

Menu title and pattern name are defined as follows.

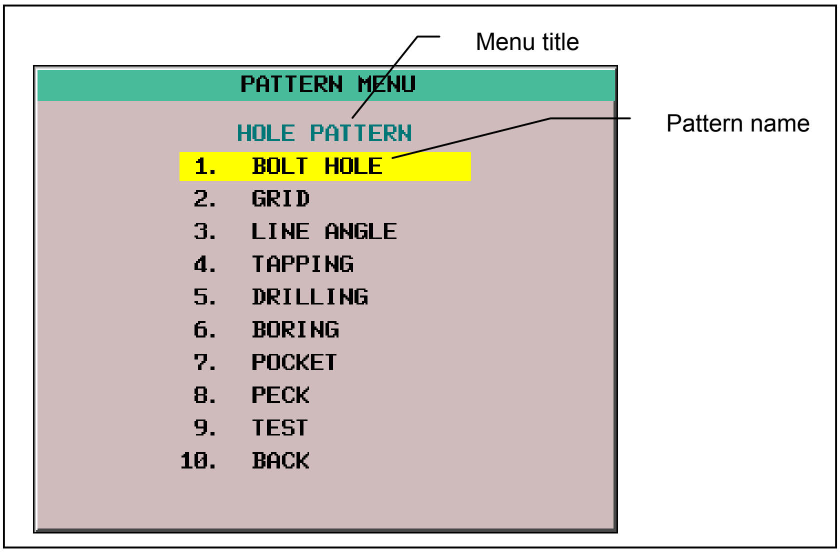


Fig. 18.4.1 (a) Pattern menu screen

Definition of menu title

The character string displayed in the menu title of the pattern menu screen is defined.
The menu title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter such as kanji character.

- Format

G65 H90 P_ Q_ R_ I_ J_ K_ ;

H90 : Specifies the menu title
P_ : The code of 1st and 2nd characters of title
Q_ : The code of 3rd and 4th characters of title
R_ : The code of 5th and 6th characters of title
I_ : The code of 7th and 8th characters of title
J_ : The code of 9th and 10th characters of title
K_ : The code of 11th and 12th characters of title
As for the way of setting the character-code, refer to the Subsection 18.4.3, "Setting the Character-codes" in the Part II, "Programming."

Definition of pattern name

The character string displayed in the pattern name which becomes a menu item is defined.
The pattern name is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

- Format

G65 H91 P_ Q_ R_ I_ J_ K_ ;

H91 : Specifies the pattern name
P_ : Specifies the menu number of the pattern name
The menu number = 1 to 10
Q_ : The code of 1st and 2nd characters of pattern name
R_ : The code of 3rd and 4th characters of pattern name
I_ : The code of 5th and 6th characters of pattern name
J_ : The code of 7th and 8th characters of pattern name
K_ : The code of 9thd and 10th characters of pattern name
As for the way of setting the character-code, refer to the Subsection 18.4.3, "Setting the Character-codes" in the Part II, "Programming."

Example

The following is example for pattern menu screen.

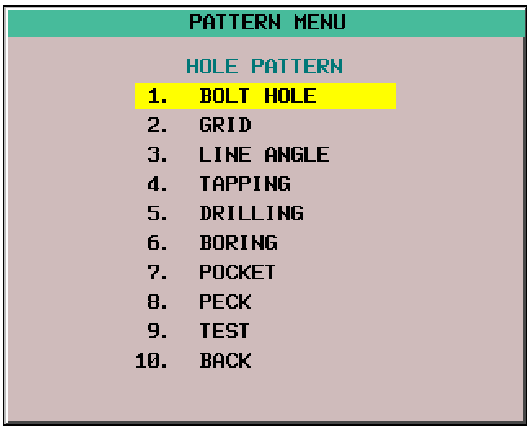


Fig. 18.4.1 (b) Pattern menu screen

```
O9500 ;
N1 G65 H90 P072079 Q076069 R032080 I065084 J084069 K082078 ;.. "HOLE PATTERN"
N2 G65 H91 P1 Q066079 R076084 I032072 J079076 K069032 ; ........... "BOLT HOLE"
N3 G65 H91 P2 Q071082 R073068 ; ....................................................... "GRID"
N4 G65 H91 P3 Q076073 R078069 I032065 J078071 K076069 ; ........... "LINE ANGLE"
N5 G65 H91 P4 Q084065 R080080 I073078 J071032 ;  ......................... "TAPPING"
N6 G65 H91 P5 Q068082 R073076 I076073 J078071 ;........................... "DORILLING"
N7 G65 H91 P6 Q066079 R082073 I078071 ; ........................................ "BORING"
N8 G65 H91 P7 Q080079 R067075 I069084 ; ........................................ "POCKET"
N9 G65 H91 P8 Q080069 R067075 ; ....................................................... "PECK"
N10 G65 H91 P9 Q084069 R083084 ; ..................................................... "TEST"
N11 G65 H91 P10 Q066065 R067075 ; ................................................... "BACK"
N12 M99 ;
```

18.4.2 Definition of the Custom Macro Screen

The title, variable name and comment are defined as follows.

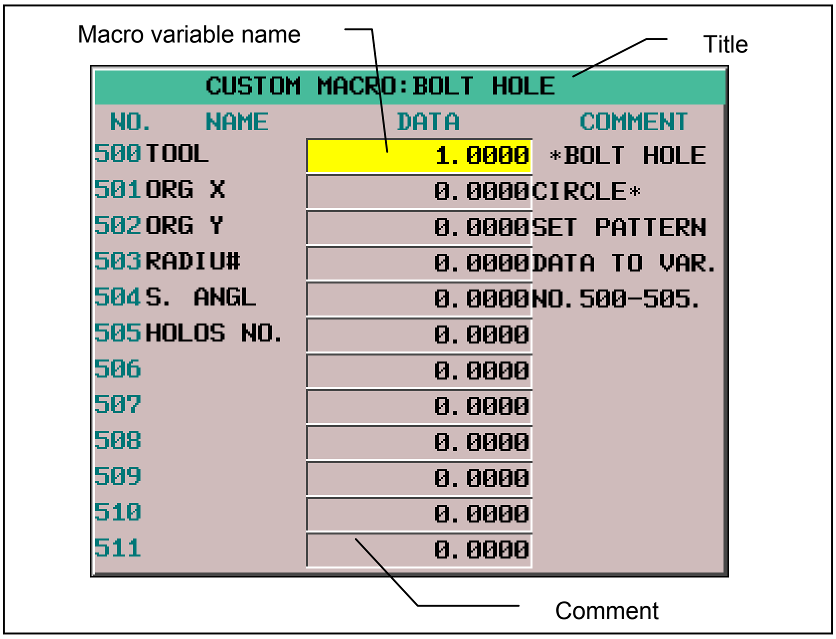


Fig. 18.4.2 (a) Custom macro screen

Definition of title

The character string displayed in the title of the custom macro screen is defined.
The title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter.

- Format

G65 H92 P_ Q_ R_ I_ J_ K_ ;

H92 : Specifies the menu title
P_ : The code of 1st and 2nd characters of the menu title
Q_ : The code of 3rd and 4th characters of the menu title
R_ : The code of 5th and 6th characters of the menu title
I_ : The code of 7th and 8th characters of the menu title
J_ : The code of 9th and 10th characters of the menu title
K_ : The code of 11th and 12th characters of the menu title
As for the way of setting the character-code, refer to the Subsection 18.4.3, "Setting the Character-codes" in the Part II, “Programming.”

Definition of macro variable

The character string displayed in the macro variable name is defined.
The macro variable is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

The variable which can be used is as follows
#100 to 199 (100 variables)
#500 to 999 (500 variables), 600 variables in total

- Format

G65 H93 P_ Q_ R_ I_ J_ K_ ;

H93 : Specifies the variable name
P_ : Specifies the variable number
Specifies 100 to 199 or 500 to 999
Q_ : The code of 1st and 2nd characters of the variable name
R_ : The code of 3rd and 4th characters of the variable name
I_ : The code of 5th and 6th characters of the variable name
J_ : The code of 7th and 8th characters of the variable name
K_ : The code of 9th and 10th characters of the variable name
As for the way of setting the character-code, refer to the Subsection 18.4.3, "Setting the Character-codes" in the Part II, “Programming.”

Definition of a comment

The character string of the comment displayed on the custom macro screen is defined.
The comment is specified by up to 12 characters in a half size letter and up to 6 characters in a full size letter per one block.

1 line is composed by 1 blocks, the maximum number of lines is 8 on the 8.4-inch display unit or 12 on the 10.4-inch display unit.
Blocks are displayed from the first comment line in the order specified in the program.

- **Format**

G65 H94 P_ Q_ R_ I_ J_ K_ ;

H94 : Specifies the comment
P_ : The code of 1st and 2nd characters of comment
Q_ : The code of 3rd and 4th characters of comment
R_ : The code of 5th and 6th characters of comment
I_ : The code of 7th and 8th characters of comment
J_ : The code of 9th and 10th characters of comment
K_ : The code of 11th and 12th characters of comment
As for the way of setting the character-code, refer to the Subsection 18.4.3, "Setting the Character-codes" in the Part II, "Programming."

Example

The following is example of the custom macro screen.

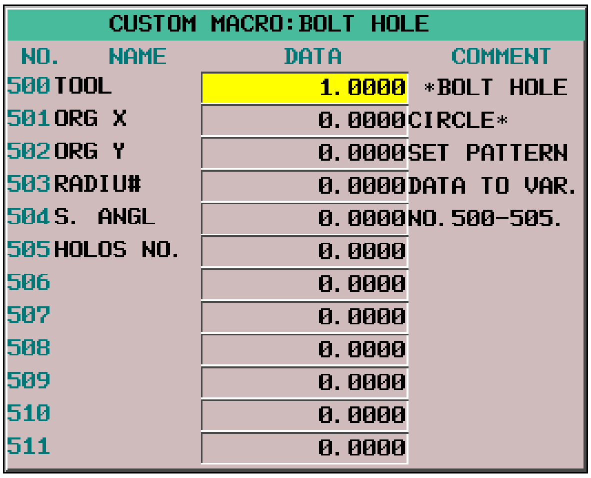


Fig. 18.4.2 (c) Custom macro screen

```
O9501 ;
N1 G65 H92 P066079 Q076084 R032072 I079076 J069032 ;..................."BOLT HOLE"
N2 G65 H93 P500 Q084079 R079076 ; ......................................................."TOOL"
N3 G65 H93 P501 Q079082 R071032 I08832 ; ........................................."ORG X"
N4 G65 H93 P502 Q079082 R071032 I08932 ; ........................................."ORG Y"
N5 G65 H93 P503 Q082065 R068073 I085803 ; ......................................."RADIUS"
N6 G65 H93 P504 Q083046 R032065 I078071 J076032 ;........................."S. ANGL"
N7 G65 H93 P505 Q072079 R076079 I083032 J078079 K046032 ;.........."HOLES NO."
N8 G65 H94 P032042 Q066079 R076084 I032072 J079076 K069032 ;...." *BOLT HOLE"
N9 G65 H94 P067073 Q082067 R076069 I042032 ; ................................"CIRCLE*"
N10 G65 H94 P083069 Q084032 R080065 I084084 J069082 K078032 ;.."SET PATTERN"
N11 G65 H94 P068065 Q084065 R032084 I079032 J086065 K082046 ;.."DATA NO VAR."
N12 G65 H94 P078079 Q046053 R048048 I045053 J048053 K046032 ;.."NO500-505"
N13 M99 ;
```

18.4.3 Setting the Character-codes

The character cannot be used to specify the NC program.
Therefore, the code corresponding to the character is specified.
One character is consist of three figures in a half size letter and six figures in a full size letter.
The character code is specified for each address of the G65 instruction by six digits.

Refer to the table for the character code.

Example)

When "ABCDEFGH" is specified, the description of the code is as follows.

Encoded character string : 065 066 067 068 069 070 071 072

```
P065066 Q067068 R069070 I071072 ;
  AB      CD      EF      GH
```

> **NOTE**
> 1 Space (032) is added ahead of the character-code, when the character-code of three digits or less is specified.
> Example)
> P065066 Q067 ; → " AB   C "
> 032(space) is put at the end, when "ABC" is displayed.
> P065066 Q067032 ; → " ABC "
> 2 It is assumed in that the space of two characters was defined in the address when there is an address not defined.
> Example)
> P065066 I067068 ; → "AB   CD"

Characters and codes to be used for the pattern data input function

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		”	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		*	042	Asterisk
N	078		+	043	Plus sign
O	079		,	044	Comma
P	080		-	045	Minus sign
Q	081		.	046	Period
R	082		/	047	Slash
S	083		:	058	Colon
T	084		;	059	Semicolon
U	085		<	060	Left angle bracket
V	086		=	061	Equal sign
W	087		>	062	Right angle bracket
X	088		?	063	Question mark
Y	089		@	064	At mark
Z	090		[	091	Left square bracket
0	048		¥	092	Yen sign
1	049		]	093	Right square bracket
2	050		^	094	
3	051		_	095	Underscore
4	052				
5	053				

The characters and the codes of the katakana is as follows.

Character	Code	Comment	Character	Code	Comment
ア	177		ム	209	
イ	178		メ	210	
ウ	179		モ	211	
エ	180		ヤ	212	
オ	181		ユ	213	
カ	182		ヨ	214	
キ	183		ラ	215	
ク	184		リ	216	
ケ	185		ル	217	
コ	186		レ	218	
サ	187		ロ	219	
シ	188		ワ	220	
ス	189		ヲ	166	
セ	190		ン	221	
ソ	191		ァ	167	
タ	192		ィ	168	
チ	193		ゥ	169	
ツ	194		ェ	170	
テ	195		ォ	171	
ト	196		ャ	172	
ナ	197		ュ	173	
ニ	198		ョ	174	
ヌ	199		ッ	175	
ネ	200		″	222	Diacritical mark
ノ	201		°	223	Diacritical mark
ハ	202		。	161	Punctuation
ヒ	203		「	162	Left quotation mark
フ	204		」	163	Right quotation mark
ヘ	205		、	164	Comma
ホ	206		・	165	Point
マ	207			000	Space
ミ	208				

NOTE
Diacritical mark is one character.

The characters and the codes of the hiragana and the kanji are as follows. The following hiraganas and kanjis use two characters of the alphanumeric character.

ぁ	あ	ぃ	い	ぅ	う	ぇ	え	ぉ	お
002 000	002 002	002 004	002 006	002 008	002 010	002 012	002 014	002 016	002 018
か	が	き	ぎ	く	ぐ	け	げ	こ	ご
002 020	002 022	002 024	002 026	002 028	002 030	002 032	002 034	002 036	002 038
さ	ざ	し	じ	す	ず	せ	ぜ	そ	ぞ
002 040	002 042	002 044	002 046	002 048	002 050	002 052	002 054	002 056	002 058
た	だ	ち	ぢ	っ	つ	づ	て	で	と
002 060	002 062	002 064	002 066	002 068	002 070	002 072	002 074	002 076	002 078
ど	な	に	ぬ	ね	の	は	ば	ぱ	ひ
002 080	002 082	002 084	002 086	002 088	002 090	002 092	002 094	002 096	002 098
び	ぴ	ふ	ぶ	ぷ	へ	べ	ぺ	ほ	ぼ
002 100	002 102	002 104	002 106	002 108	002 110	002 112	002 114	002 116	002 118

ぽ	ま	み	む	め	も	ゃ	や	ゅ	ゆ
002 120	002 122	002 124	002 126	002 128	002 130	002 132	002 134	002 136	002 138
ょ	よ	ら	り	る	れ	ろ	わ	わ	素
002 140	002 142	002 144	002 146	002 148	002 150	002 152	002 154	002 156	002 158
材	を	ん	種	類	棒	穴	成	形	質
002 160	002 162	002 164	002 166	002 168	002 170	002 172	002 174	002 176	002 178
寸	法	外	径	長	端	面	最	小	内
002 180	002 182	002 184	002 186	002 188	002 190	002 192	002 194	002 196	002 198
大	加	工	切	削	倣	正	途	中	荒
002 200	002 202	002 204	002 206	002 208	002 210	002 212	002 214	002 216	002 218
具	番	号	仕	上	込	点	方	向	速
002 220	002 222	002 224	002 226	002 228	002 230	002 232	002 234	002 236	002 238
度	送	量	開	始	深	主	軸		
002 240	002 242	002 244	002 246	002 248	002 250	002 252	002 254		
回	転	数	位	置	決	直	線	時	円
003 000	003 002	003 004	003 006	003 008	003 010	003 012	003 014	003 016	003 018
反	現	在	指	令	値	領	域	診	断
003 020	003 022	003 024	003 026	003 028	003 030	003 032	003 034	003 036	003 038
操	作	手	引	機	械	残	移	動	次
003 040	003 042	003 044	003 046	003 048	003 050	003 052	003 054	003 056	003 058
早	電	源	投	入	間	分	秒	自	運
003 060	003 062	003 064	003 066	003 068	003 070	003 072	003 074	003 076	003 078
負	荷	実	使	用	寿	命	新	規	除
003 080	003 082	003 084	003 086	003 088	003 090	003 092	003 094	003 096	003 098
隅	取	単	補	能	独	終	了	記	角
003 100	003 102	003 104	003 106	003 108	003 110	003 112	003 114	003 116	003 118
溝	刃	幅	広	設	定	一	覧	表	部
003 120	003 122	003 124	003 126	003 128	003 130	003 132	003 134	003 136	003 138
炭	合	金	鋼	超	硬	先	付	摩	耗
003 140	003 142	003 144	003 146	003 148	003 150	003 152	003 154	003 156	003 158
仮	想	副	行	挿	消	去	山	高	準
003 160	003 162	003 164	003 166	003 168	003 170	003 172	003 174	003 176	003 178
備	完	後	弧	助	択	無	視	器	原
003 180	003 182	003 184	003 186	003 188	003 190	003 192	003 194	003 196	003 198
登	録	再	処	理	描	画	過	容	編
003 200	003 202	003 204	003 206	003 208	003 210	003 212	003 214	003 216	003 218
集	未	対	相	座	標	示	名	歯	変
003 220	003 222	003 224	003 226	003 228	003 230	003 232	003 234	003 236	003 238
呼	推	馬	力	系	選	達	閉		
003 240	003 242	003 244	003 246	003 248	003 250	003 252	003 254		
禁	復	帰	書	個	桁	稼	由	両	半
004 000	004 002	004 004	004 006	004 008	004 010	004 012	004 014	004 016	004 018
逃	底	逆	下	空	四	触	平	代	辺
004 020	004 022	004 024	004 026	004 028	004 030	004 032	004 034	004 036	004 038
格	子	周	心	本	群	停	止	巾	微
004 040	004 042	004 044	004 046	004 048	004 050	004 052	004 054	004 056	004 058
状	路	範	囲	倍	率	注	側	特	殊
004 060	004 062	004 064	004 066	004 068	004 070	004 072	004 074	004 076	004 078
距	離	連	続	増	隔	件	初	期	条
004 080	004 082	004 084	004 086	004 088	004 090	004 092	004 094	004 096	004 098
経	握	圧	扱	陰	隠	右	押	横	黄
004 100	004 102	004 104	004 106	004 108	004 110	004 112	004 114	004 116	004 118

億	屋	化	何	絵	階	概	該	巻	換
004 120	004 122	004 124	004 126	004 128	004 130	004 132	004 134	004 136	004 138
気	起	軌	技	疑	供	共	境	強	教
004 140	004 142	004 144	004 146	004 148	004 150	004 152	004 154	004 156	004 158
掘	繰	係	傾	型	検	権	研	肩	見
004 160	004 162	004 164	004 166	004 168	004 170	004 172	004 174	004 176	004 178
験	元	弦	減	孔	巧	控	更	校	構
004 180	004 182	004 184	004 186	004 188	004 190	004 192	004 194	004 196	004 198
根	左	差	雑	参	散	産	算	治	耳
004 200	004 202	004 204	004 206	004 208	004 210	004 212	004 214	004 216	004 218
式	失	修	十	従	勝	商	少	尚	昇
004 220	004 222	004 224	004 226	004 228	004 230	004 232	004 234	004 236	004 238
植	色	食	伸	信	侵	振	浸		
004 240	004 242	004 244	004 246	004 248	004 250	004 252	004 254		
真	暗	以	意	異	影	鋭	越	価	可
005 000	005 002	005 004	005 006	005 008	005 010	005 012	005 014	005 016	005 018
科	果	箇	課	各	拡	核	学	掛	漢
005 020	005 022	005 024	005 026	005 028	005 030	005 032	005 034	005 036	005 038
簡	観	関	含	却	客	休	急	業	曲
005 040	005 042	005 044	005 046	005 048	005 050	005 052	005 054	005 056	005 058
均	筋	継	計	軽	言	限	互	降	採
005 060	005 062	005 064	005 066	005 068	005 070	005 072	005 074	005 076	005 078
済	細	姿	思	写	射	斜	者	車	借
005 080	005 082	005 084	005 086	005 088	005 090	005 092	005 094	005 096	005 098
縦	重	出	述	術	渉	照	省	章	証
005 100	005 102	005 104	005 106	005 108	005 110	005 112	005 114	005 116	005 118
象	身	進	人	図	違	印	沿	遠	央
005 120	005 122	005 124	005 126	005 128	005 130	005 132	005 134	005 136	005 138
奥	往	応	会	解	改	割	活	願	基
005 140	005 142	005 144	005 146	005 148	005 150	005 152	005 154	005 156	005 158
奇	寄	岐	既	近	区	矩	駆	偶	旧
005 160	005 162	005 164	005 166	005 168	005 170	005 172	005 174	005 176	005 178
求	球	究	級	欠	結	口	語	誤	交
005 180	005 182	005 184	005 186	005 188	005 190	005 192	005 194	005 196	005 198
厚	項	刻	告	黒	財	策	糸	試	資
005 200	005 202	005 204	005 206	005 208	005 210	005 212	005 214	005 216	005 218
事	持	似	釈	弱	受	収	純	順	所
005 220	005 222	005 224	005 226	005 228	005 230	005 232	005 234	005 236	005 238
序	剰	場	常	飾	水	錐	据		
005 240	005 242	005 244	005 246	005 248	005 250	005 252	005 254		
制	整	製	前	全	然	則	属	即	他
006 000	006 002	006 004	006 006	006 008	006 010	006 012	006 014	006 016	006 018
多	存	谷	探	短	徴	鎮	調	頂	鉄
006 020	006 022	006 024	006 026	006 028	006 030	006 032	006 034	006 036	006 038
添	頭	同	導	道	熱	年	濃	箱	発
006 040	006 042	006 044	006 046	006 048	006 050	006 052	006 054	006 056	006 058
抜	伴	必	百	複	物	文	聞	併	忘
006 060	006 062	006 064	006 066	006 068	006 070	006 072	006 074	006 076	006 078
末	密	有	余	与	裏	立	略	青	席
006 080	006 082	006 084	006 086	006 088	006 090	006 092	006 094	006 096	006 098
石	積	赤	接	折	粗	創	双	捜	太
006 100	006 102	006 104	006 106	006 108	006 110	006 112	006 114	006 116	006 118

打	体	待	態	替	段	知	地	致	遅
006 120	006 122	006 124	006 126	006 128	006 130	006 132	006 134	006 136	006 138
追	通	伝	得	読	凸	凹	突	鈍	敗
006 140	006 142	006 144	006 146	006 148	006 150	006 152	006 154	006 156	006 158
杯	背	配	品	不	布	並	頁	別	片
006 160	006 162	006 164	006 166	006 168	006 170	006 172	006 174	006 176	006 178
返	勉	弁	保	明	滅	木	目	歪	揺
006 180	006 182	006 184	006 186	006 188	006 190	006 192	006 194	006 196	006 198
様	溶	要	抑	良	輪	和	話	枠	節
006 200	006 202	006 204	006 206	006 208	006 210	006 212	006 214	006 216	006 218
説	絶	千	専	浅	旋	総	走	退	台
006 220	006 222	006 224	006 226	006 228	006 230	006 232	006 234	006 236	006 238
第	題	卓	室	着	柱	鋳	丁		
006 240	006 242	006 244	006 246	006 248	006 250	006 252	006 254		
低	訂	肉	日	白	薄	比	皮	被	非
007 000	007 002	007 004	007 006	007 008	007 010	007 012	007 014	007 016	007 018
美	普	伏	歩	包	門	問	絡	列	万
007 020	007 022	007 024	007 026	007 028	007 030	007 032	007 034	007 036	007 038
利	訳	礼	乱	放	枚	約	練	油	劣
007 040	007 042	007 044	007 046	007 048	007 050	007 052	007 054	007 056	007 058
例	郭	戻	冷	垂	緑	紫	許	測	精
007 060	007 062	007 064	007 066	007 068	007 070	007 072	007 074	007 076	007 078
効	→	↗	↑	↖	←	↙	↓	↘	
007 080	007 082	007 084	007 086	007 088	007 090	007 092	007 094	007 096	007 098
				板	予	〃	家	装	管
007 100	007 102	007 104	007 106	007 108	007 110	007 112	007 114	007 116	007 118
粉	等					貫	安	α	β
007 120	007 122	007 124	007 126	007 128	007 130	007 132	007 134	007 136	007 138
程	抗	張	任	破	損	御	足	守	般
007 140	007 142	007 144	007 146	007 148	007 150	007 152	007 154	007 156	007 158
納	義	丸	汎	固	毎	当	的	詳	鳥
007 160	007 162	007 164	007 166	007 168	007 170	007 172	007 174	007 176	007 178
適	論	額	縁	温	給	界	混	監	締
007 180	007 182	007 184	007 186	007 188	007 190	007 192	007 194	007 196	007 198
護	己	称	樹	脂	料	落	確	認	報
007 200	007 202	007 204	007 206	007 208	007 210	007 212	007 214	007 216	007 218
排	性	生	績	判	搬	砥	θ	島	壁
007 220	007 222	007 224	007 226	007 228	007 230	007 232	007 234	007 236	007 238
				]	[	ˌ	■		
007 240	007 242	007 244	007 246	007 248	007 250	007 252	007 254		

III. OPERATION

1 GENERAL

Chapter 1, "GENERAL", consists of the following sections:

1.1 MANUAL OPERATION

Explanation

- Manual reference position return

The CNC machine tool has a position used to determine the machine position.
This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.
Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel. (See Section III-3.1)

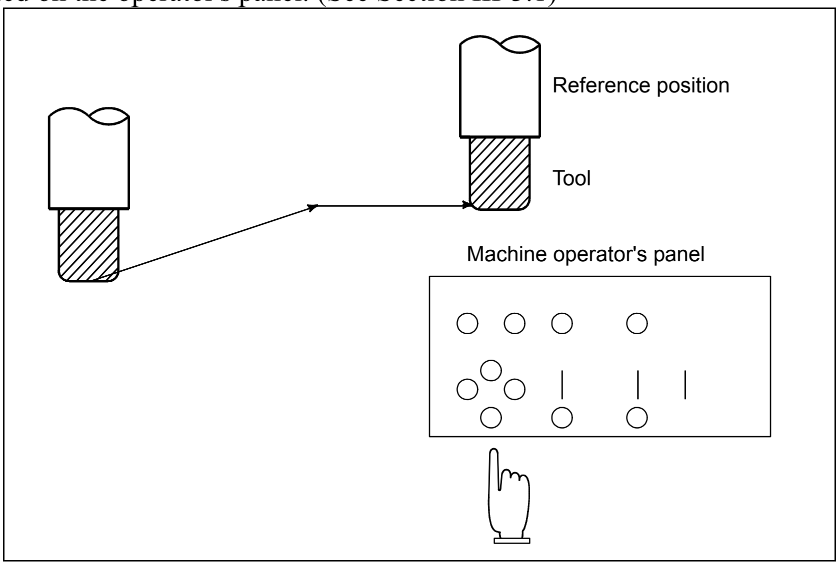


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return (See Section II-6.1).

- The tool movement by manual operation

Using machine operator's panel switches, pushbuttons, or the manual handle, the tool can be moved along each axis.

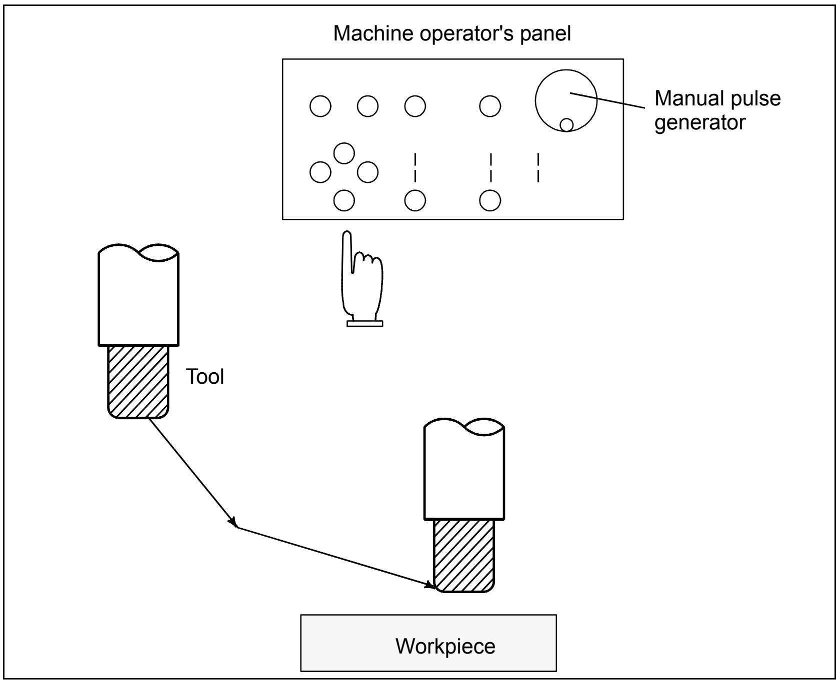


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i) Jog feed (See Section III-3.2)
The tool moves continuously while a pushbutton remains pressed.

(ii) Incremental feed (See Section III-3.3)
The tool moves by the predetermined distance each time a button is pressed.

(iii) Manual handle feed (See Section III-3.4)
By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, MDI and DNC operations. (See Section III-4).

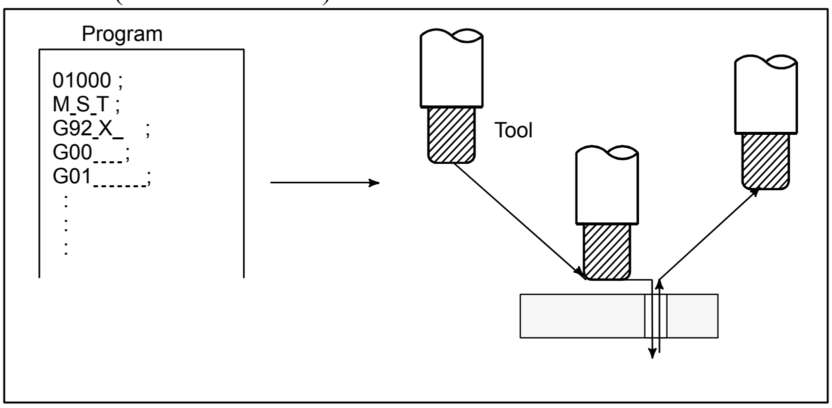


Fig. 1.2 (a) Tool Movement by programming

Explanation

- Memory operation

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

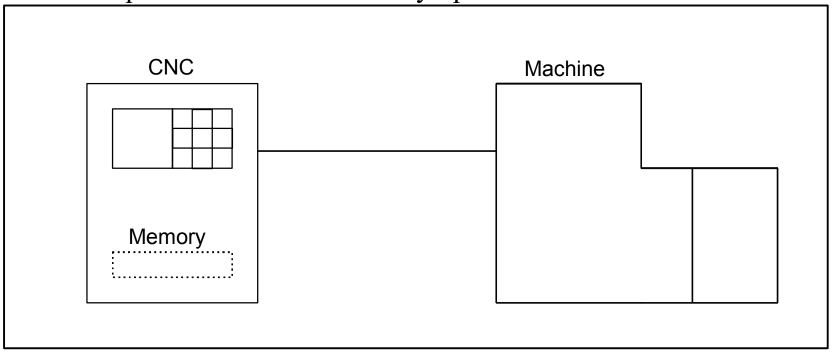


Fig. 1.2 (b) Memory operation

- MDI operation

After the program is entered, as an command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.

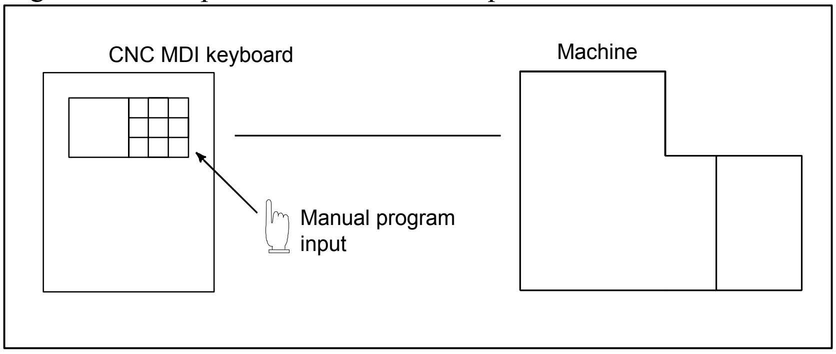


Fig. 1.2 (c) MDI operation

- DNC operation

In this mode of operation, the program is not registered in the CNC memory. It is read from the external input/output devices instead. This is called DNC operation.

1.3 AUTOMATIC OPERATION

Explanation

- Program selection

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number (Section III-9.3).

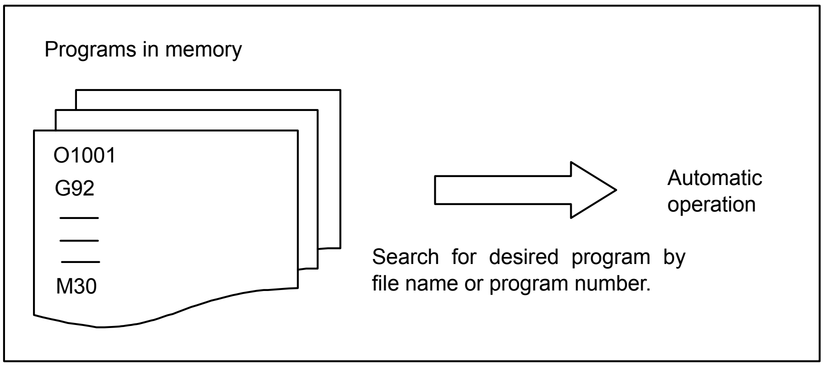


Fig. 1.3 (a) Program selection for automatic operation

- Start and stop

Pressing the cycle start pushbutton causes automatic operation to start. By pressing the feed hold or reset pushbutton, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops. (See Section III-4)

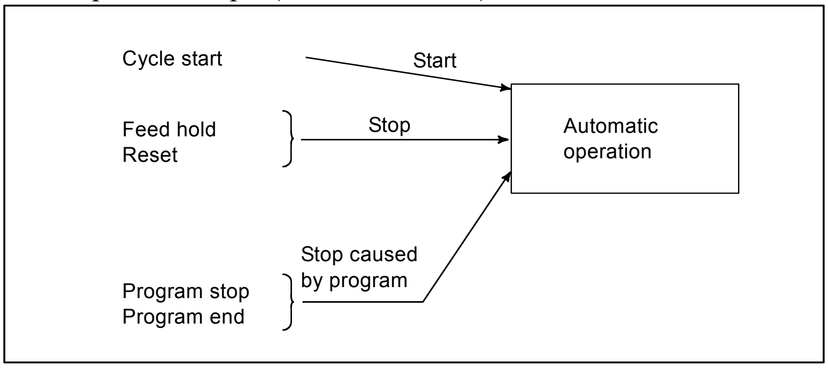


Fig. 1.3 (b) Start and stop for automatic operation

- Handle interruption

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle. (See Section III-4.6)

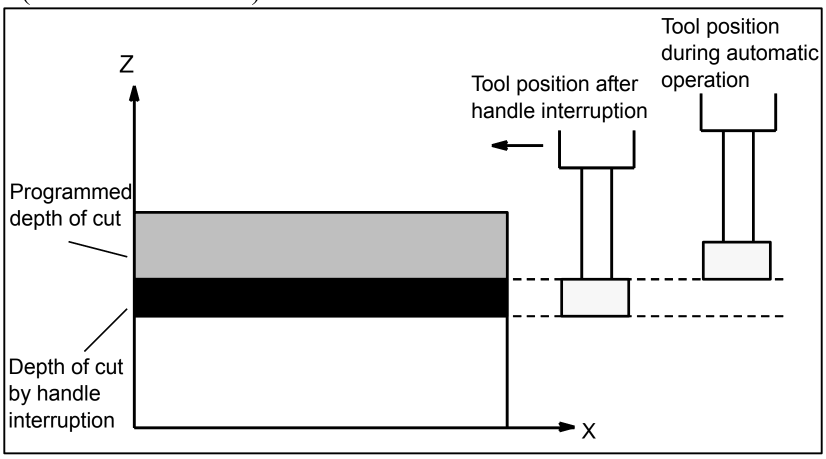


Fig. 1.3 (c) Handle interruption for automatic operation

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed.
It checks whether the created program can operate the machine as desired.
This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Section III-5).

1.4.1 Check by Running the Machine

Explanation

- Dry run

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel. (See Section III-5.4)

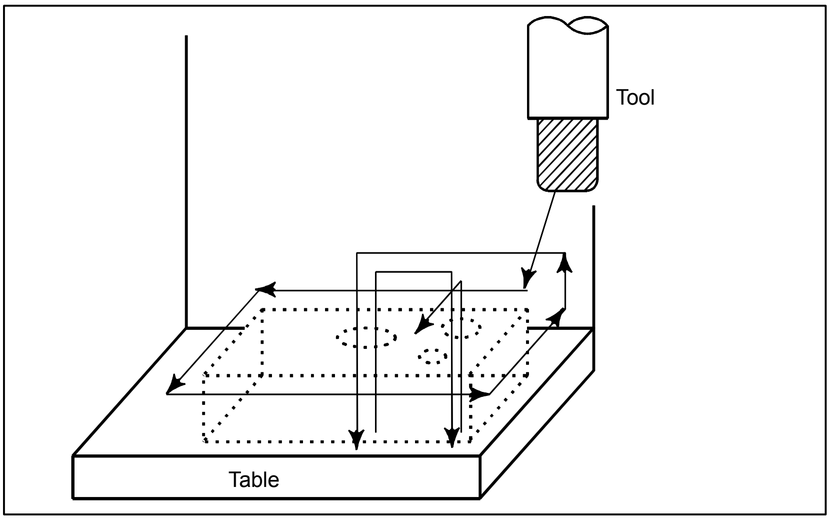


Fig. 1.4.1 (a) Dry run

- Feedrate override

Check the program by changing the feedrate specified in the program. (See Section III-5.2)

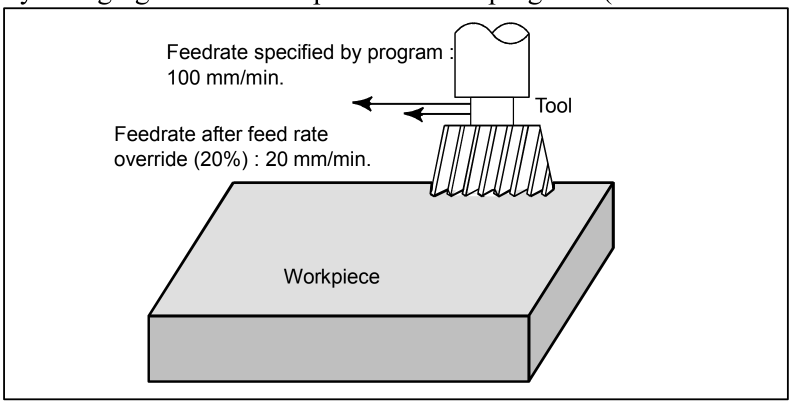


Fig. 1.4.1 (b) Feedrate override

- Single block

When the cycle start pushbutton is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner. (See Section III-5.5)

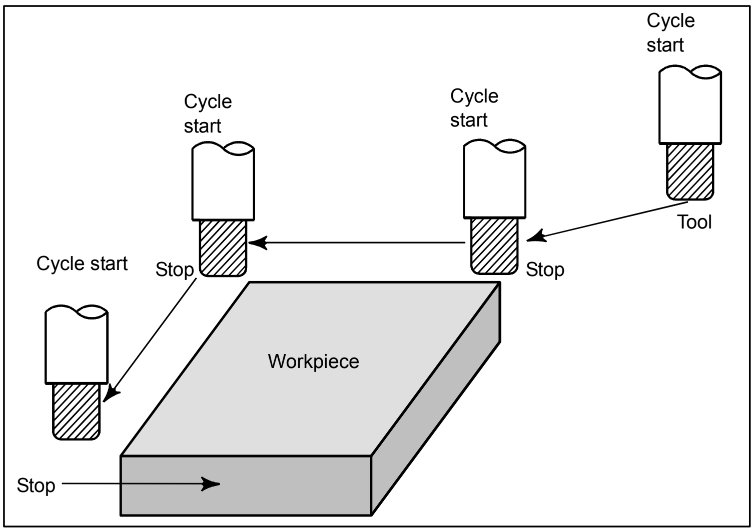


Fig. 1.4.1 (c) Single Block

1.4.2 How to View the Current Position Display Change without Running the Machine

Explanation

- Machine Lock

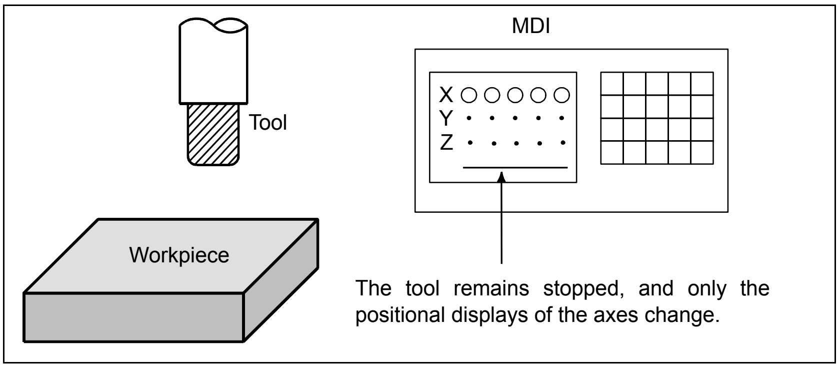


Fig. 1.4.2 (a) Machine Lock

- Auxiliary function lock

When automatic running is placed into the auxiliary function lock mode during the machine lock mode (See Sections III-5.1), all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) (See Section III-5.1) are disabled.

1.5 EDITING A PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the MDI panel (See Section III-10).
This operation can be executed using the program edit function.

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the MDI screen (See III-12).

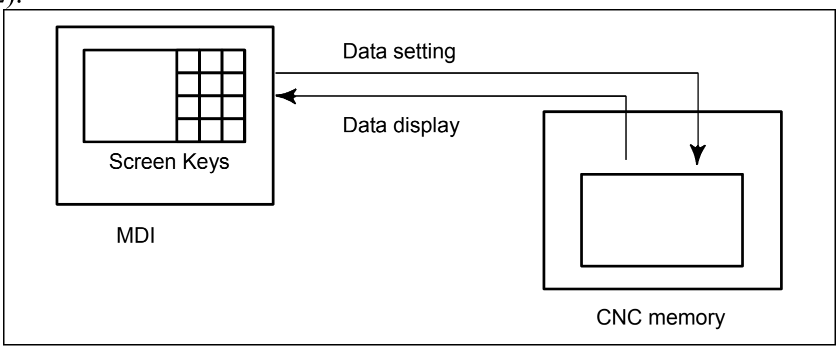


Fig. 1.6 (a) Displaying and setting data

Explanation

- Offset value

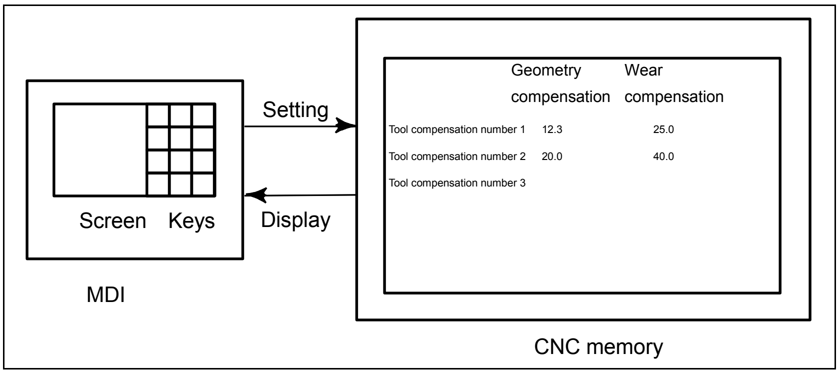


Fig. 1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions.
By setting tool dimension data in CNC memory beforehand, CNC automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value.

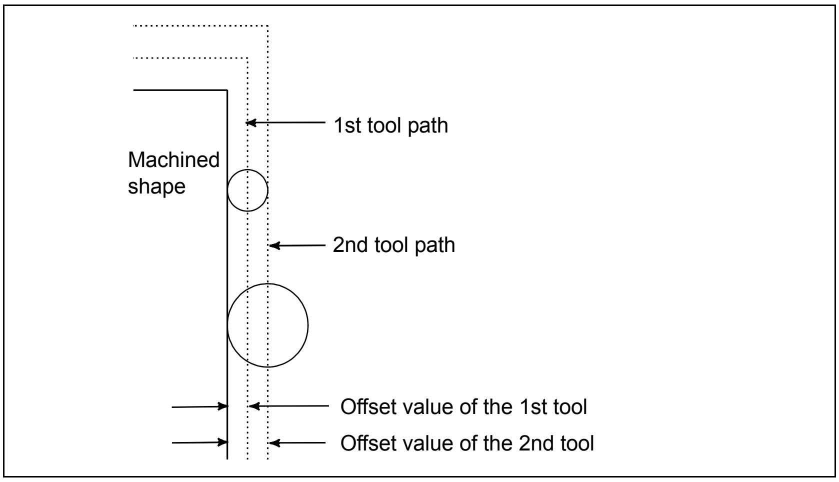


Fig. 1.6 (c) Offset value

- Displaying and setting operator's setting data

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.
For example, the following data can be set:

- Inch/Metric switching
- Selection of I/O devices
- Mirror image cutting on/off

The above data is called setting data (See Section III-12.3.1).

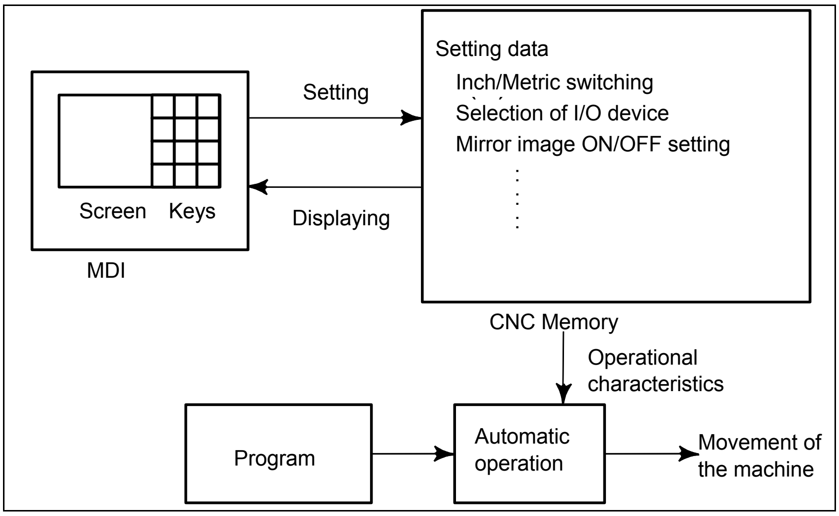


Fig. 1.6 (d) Displaying and setting operator's setting data

- Displaying and setting parameters

The CNC functions have versatility in order to take action in characteristics of various machines. For example, CNC can specify the following:

- Rapid traverse rate of each axis
- Whether increment system is based on metric system or inch system.
- How to set command multiply/detect multiply (CMR/DMR)

Data to make the above specification is called parameters (See Section III-12.4.1). Parameters differ depending on machine tool.

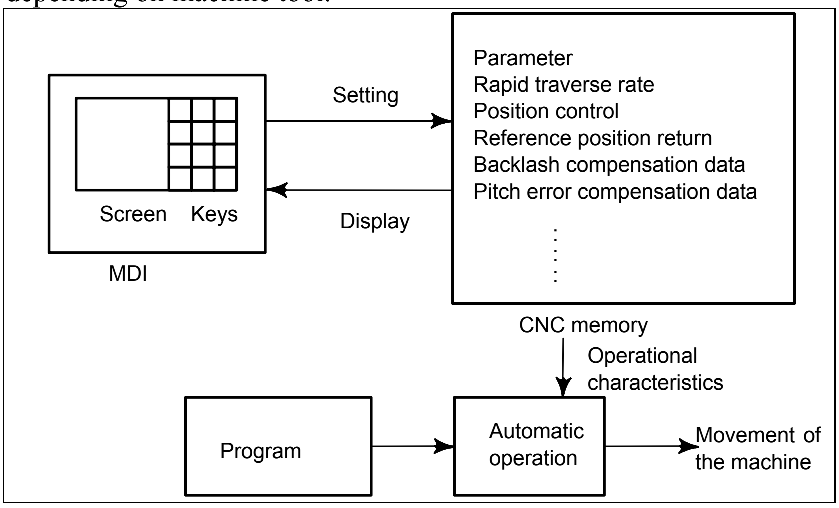


Fig. 1.6 (e) Displaying and setting parameters

- Data protection key

A key called the data protection key can be defined. It is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously (See Section III-12).

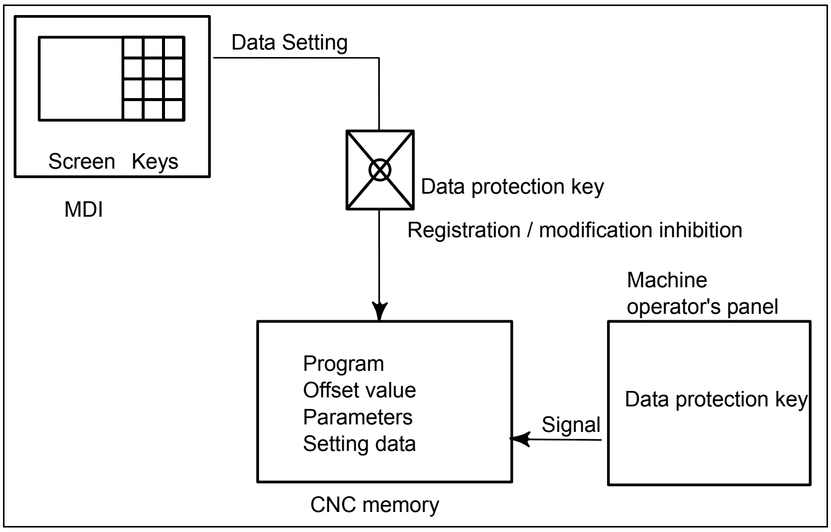


Fig. 1.6 (f) Data protection key

1.7 DISPLAY

1.7.1 Program Display

The contents of the currently active program are displayed.
(See Section III-12.2.1)

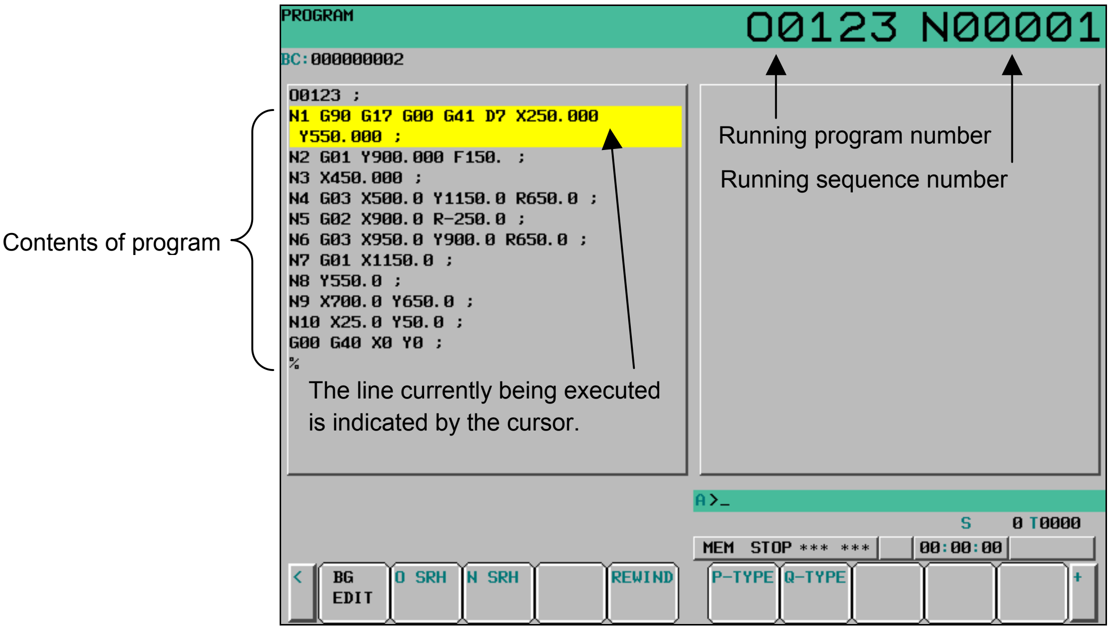


Fig. 1.7.1 (a)

The programs in the program memory are listed.

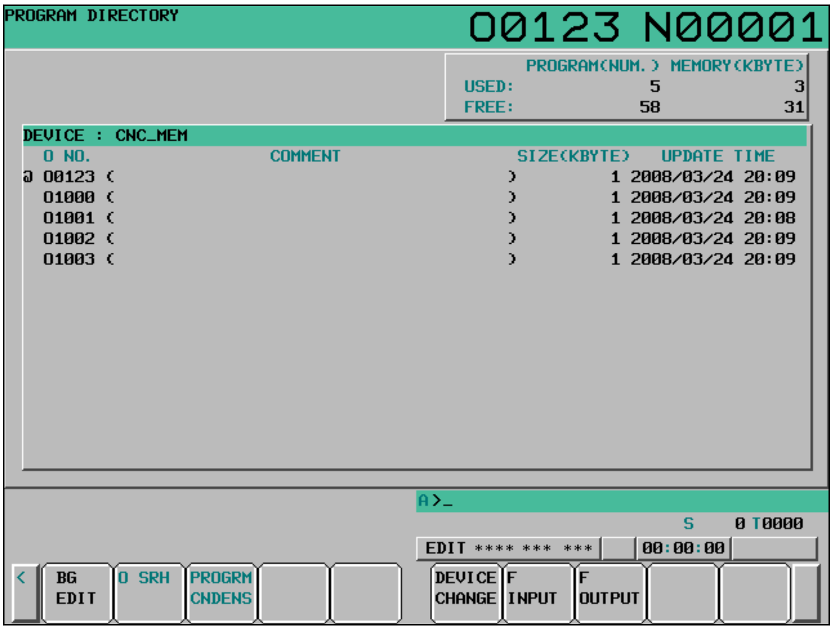


Fig. 1.7.1 (b)

1.7.2 Current Position Display

The current position of the tool is displayed with the coordinate values.
Moreover, the distance from the current position to a target point can be displayed as a remaining travel distance.
(See Subsections III-12.1.1 to 12.1.3.)

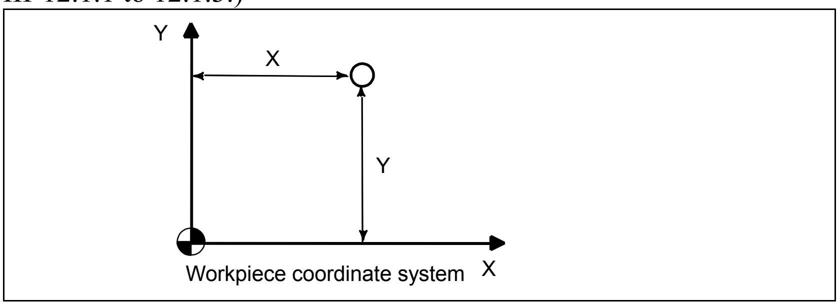


Fig. 1.7.2 (a)

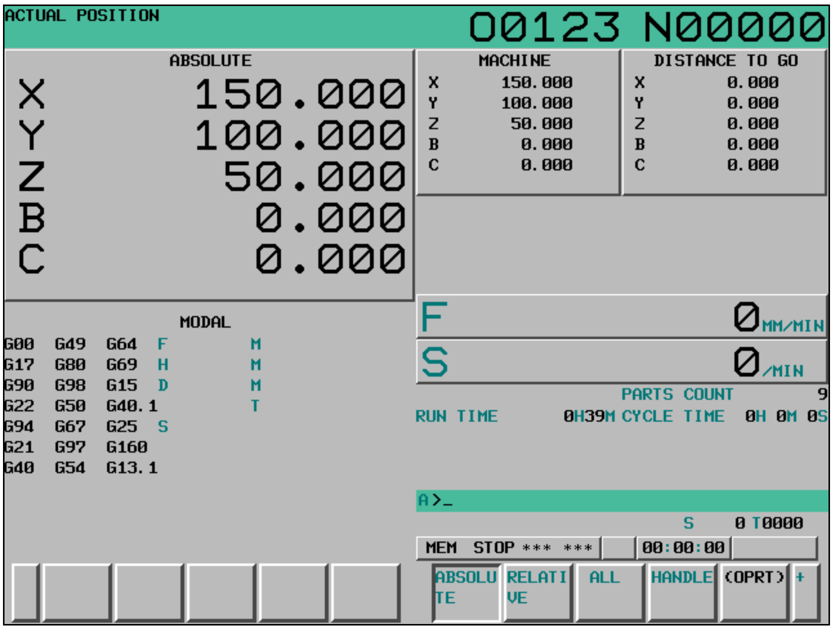


Fig. 1.7.2 (b)

1.7.3 Alarm Display

When a trouble occurs during operation, error code and alarm message are displayed on the screen. (See Section III-7.1.)
See APPENDIX G for the list of error codes and their meanings.

Fig. 1.7.3 (a)

1.7.4 Parts Count Display, Run Time Display

The position display screen displays a machined parts count, run time, and cycle time. (See Section lll-12.3.3.)

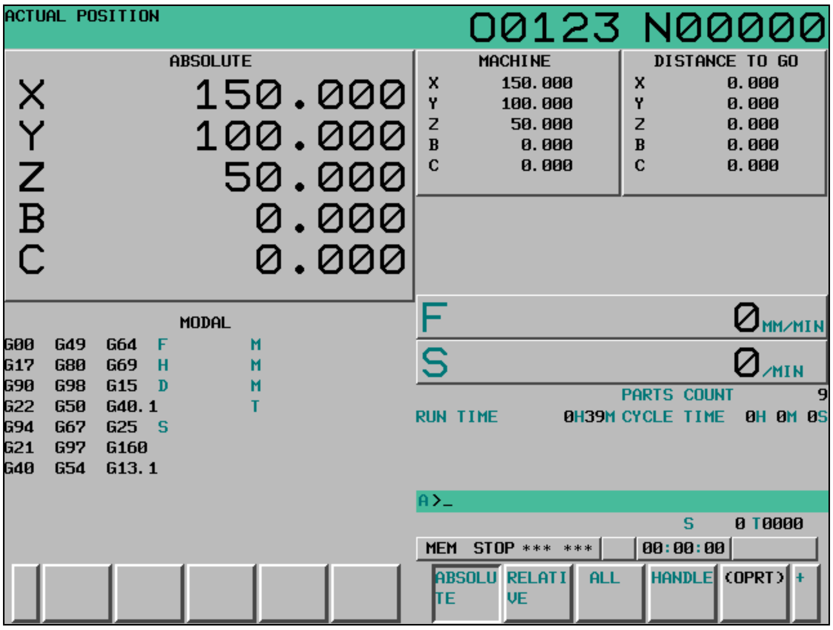


Fig. 1.7.4 (a)

2 OPERATIONAL DEVICES

As operational devices, setting and display devices attached to the CNC, and machine operator's panels are available.
For machine operator's panels, refer to the relevant manual of the machine tool builder.

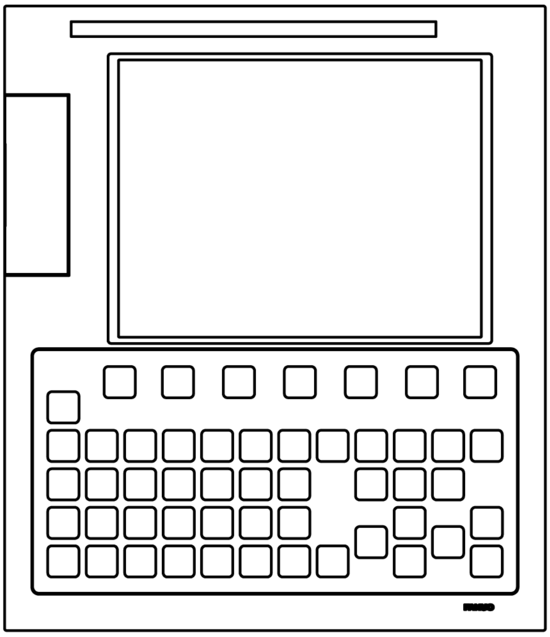
Chapter 2, "OPERATIONAL DEVICES", consists of the following sections:

2.1 SETTING AND DISPLAY UNITS

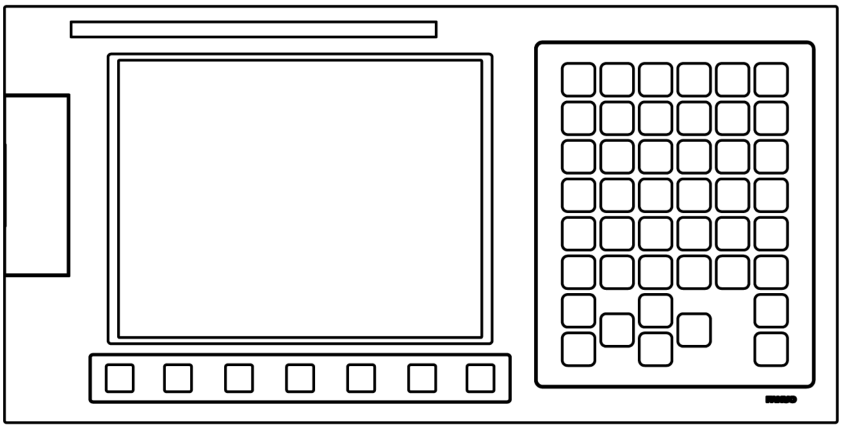
The setting and display units are shown in Subsections 2.1.1 to 2.1.4 of Part III.

8.4” LCD/MDIIII-2.1.1
10.4” LCD III-2.1.2
Standard MDI Unit (ONG Key) III-2.1.3
Small MDI Unit (ONG Key).................... III-2.1.4

2.1.1 8.4” LCD/MDI

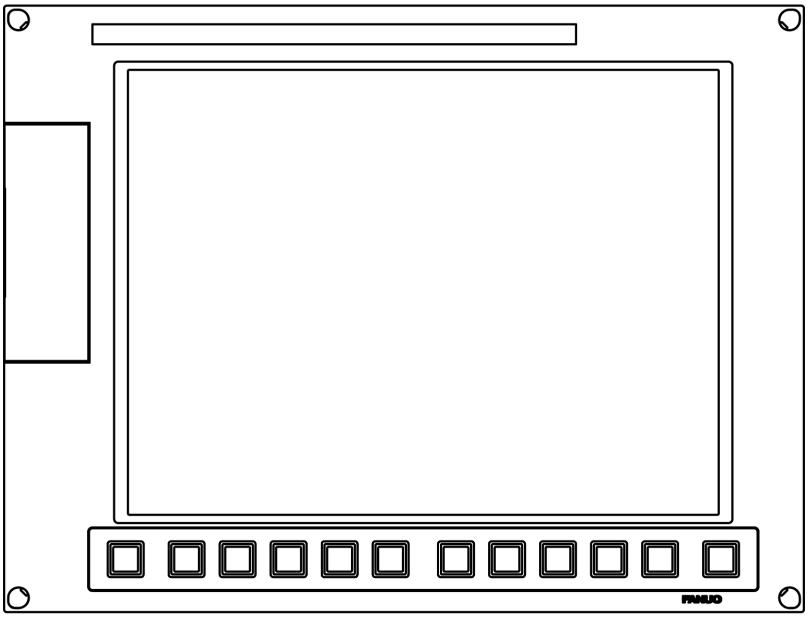


8.4” LCD/MDI (vertical type)



8.4” LCD/MDI (horizontal type)

2.1.2 10.4" LCD

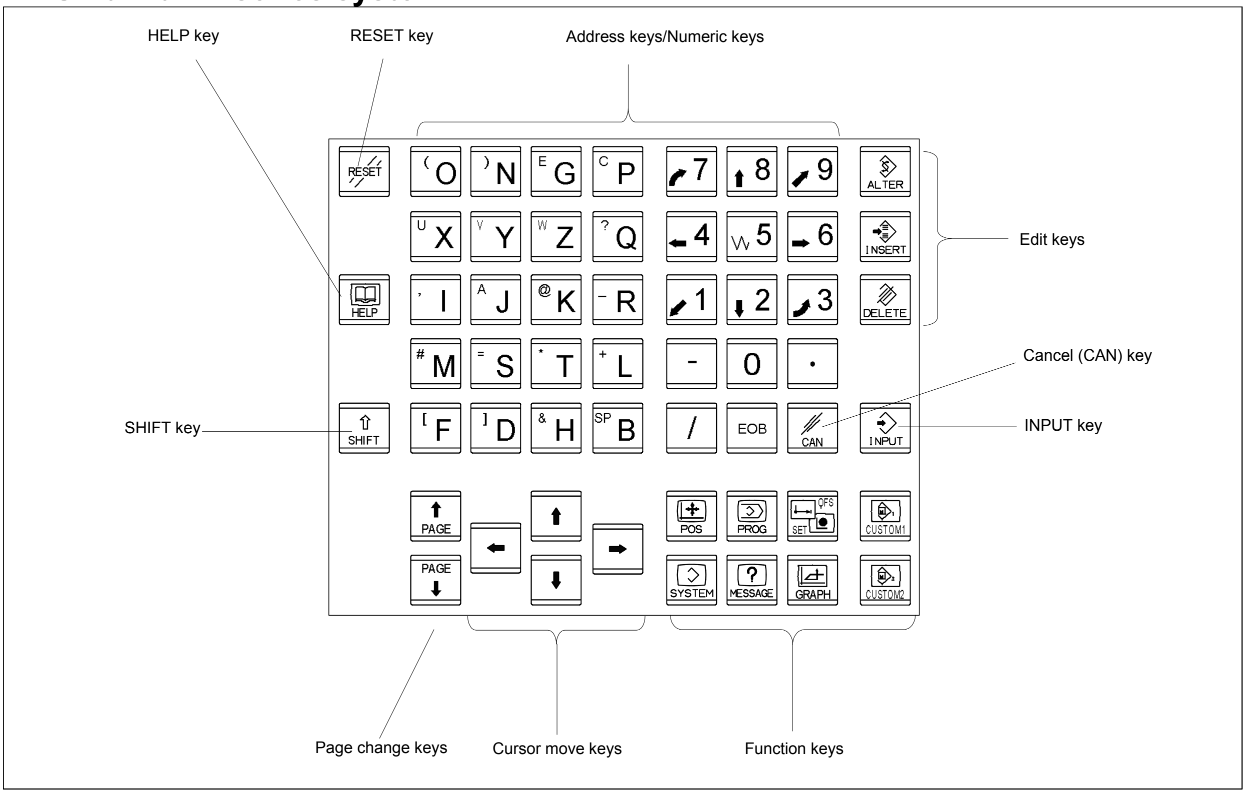


10.4" LCD

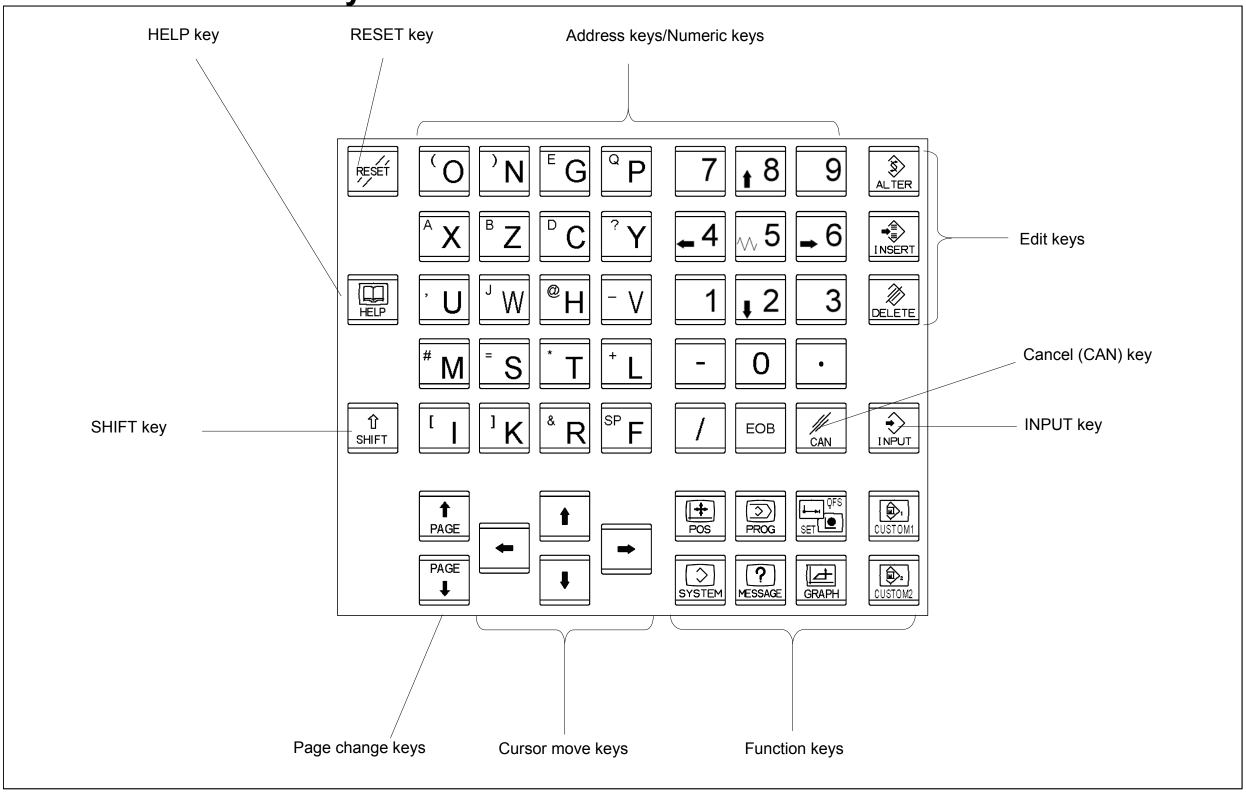
(Note) The touch panel display unit has no soft keys.

2.1.3 Standard MDI Unit (ONG Key)

- Unit with M series system

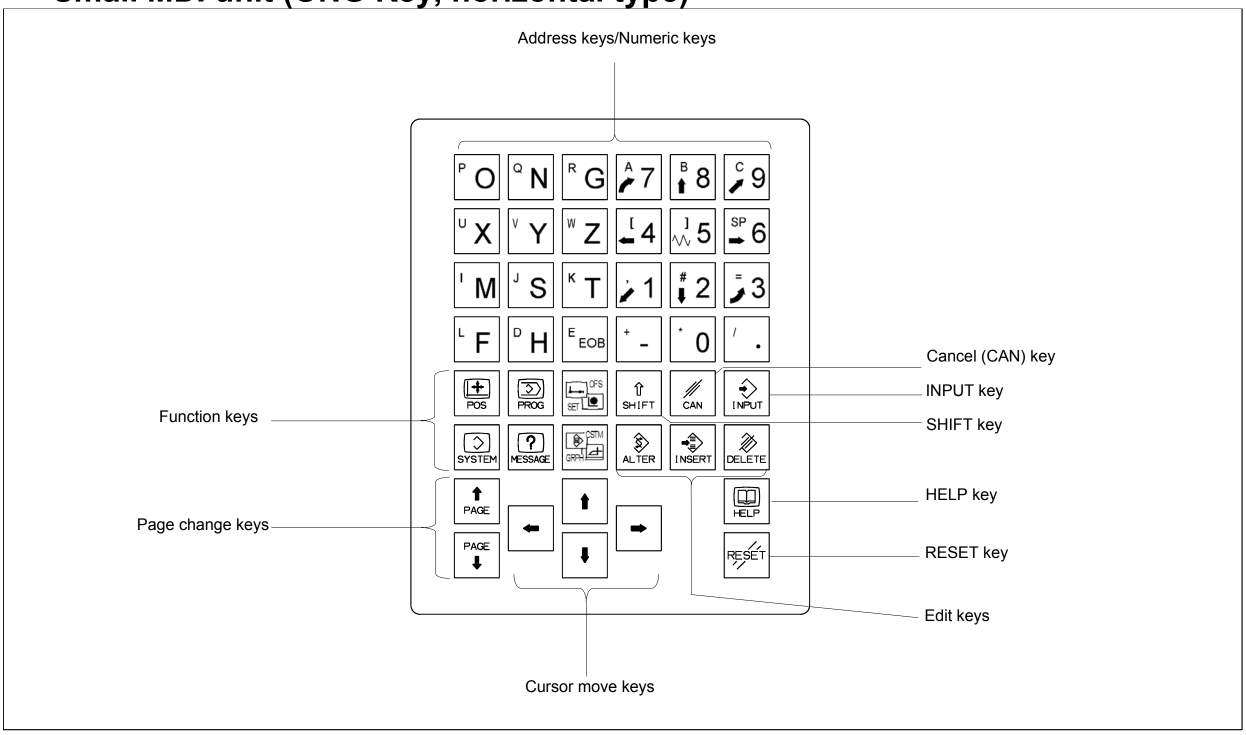


- **Unit with T series system**

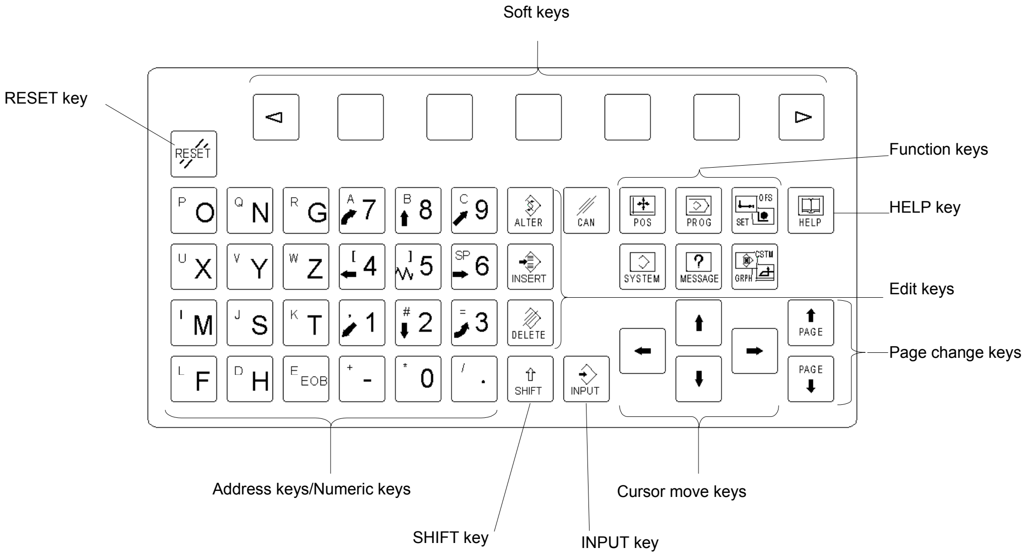


2.1.4 Small MDI Unit (ONG Key)

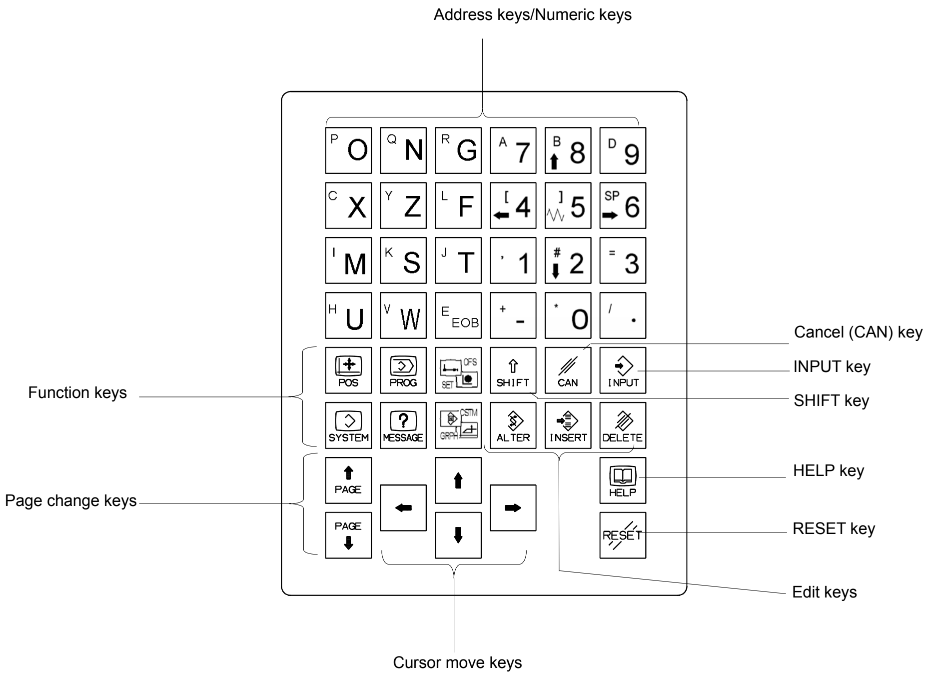
- **Unit with M series system**
  Small MDI unit (ONG Key, horizontal type)



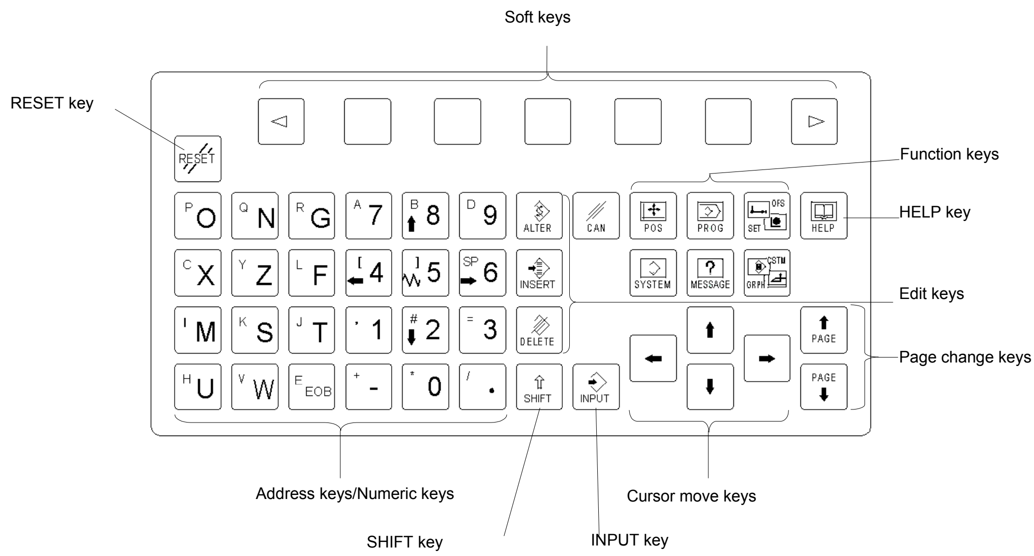
Small MDI unit (ONG Key, vertical type)



- Unit with T series system
Small MDI unit (ONG Key, horizontal type)



Small MDI unit (ONG Key, vertical type)



2.2 OPERATIONAL DEVICES

Table 2.2 (a) Explanation of the MDI keyboard

Number	Name	Explanation
1	RESET key	Press this key to reset the CNC, to cancel an alarm, etc.
2	HELP key	Press this key to use the help function when uncertain about the operation of an MDI key (help function).
3	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed on the display unit.
4	Address and numeric keys . . .	Press these keys to input alphabetic, numeric, and other characters.
5	SHIFT key	Some address keys or numeric keys have two characters on their top faces. Pressing the <SHIFT> key switches the characters. Special character ^ is displayed on the screen when a character indicated at the upper left corner on the keytop can be entered.
6	INPUT key	When an address key or a numerical key is pressed, the data is input to the key input buffer, and it is displayed on the screen. To copy the data in the key input buffer to the offset register, etc., press the INPUT key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result.

Number	Name	Explanation
7	CANCEL (CAN) key CAN	Press this key to delete the last character or symbol input to the key input buffer. Example) When the key input buffer displays >N001X100Z_ and the cancel key CAN is pressed, Z is canceled and >N001X100_ is displayed.
8	Edit keys ALTER INSERT DELETE	Press these keys when editing the program. ALTER : ALTER INSERT : INSERT DELETE : IDELETE
9	Function keys POS PROG . . .	Press theses keys to switch display screens for each function. See III-2.3 for details of the function keys.
10	Cursor keys	There are four different cursor move keys. → : This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction. ← : This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction. ↓ : This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction. ↑ : This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.
11	Page change keys (Page keys) PAGE ↑ PAGE ↓	Two kinds of page change keys are described below. PAGE ↓ : This key is used to changeover the page on the screen in the forward direction. PAGE ↑ : This key is used to changeover the page on the screen in the reverse direction.

Explanation

- Key operation with 2-path control

In the 2-path control, be sure to select the tool post for which data is specified, using the path selection switch on the machine operator's panel. Then, perform keyboard operation, such as displaying or specifying various data items, and editing a program.

2.3 FUNCTION KEYS AND SOFT KEYS

The function keys are used to select the type of screen (function) to be displayed. When a soft key (section select soft key) is pressed immediately after a function key, the screen (section) corresponding to the selected function can be selected.

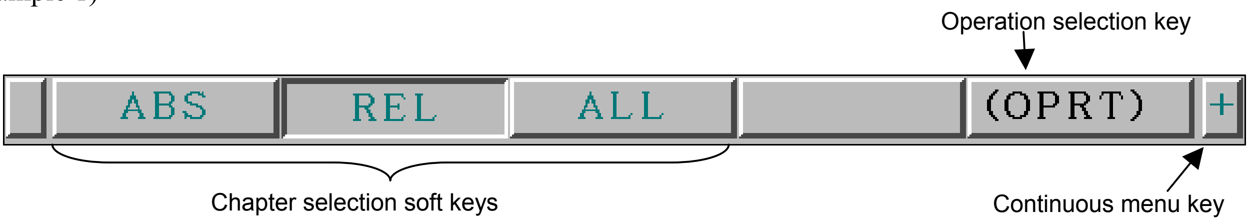
This section assumes the 8.4-inch display unit with seven soft keys.

2.3.1 General Screen Operations

- Procedure

1 By pressing a function key on the MDI panel, the chapter selection soft keys that belong to the function are displayed.

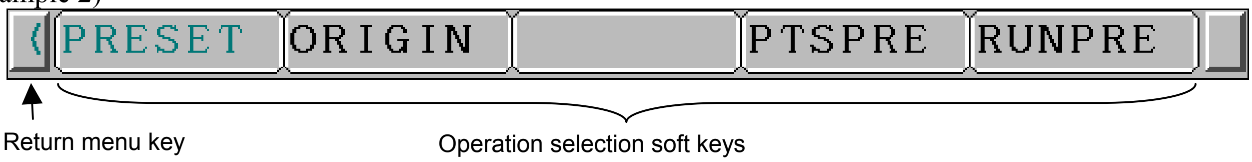
Example 1)



2 When one of the chapter selection soft keys is pressed, the screen of the chapter is displayed. If the soft key of a desired chapter is not displayed, press the continuous menu key.
In a chapter, a further choice may be made from multiple chapters.

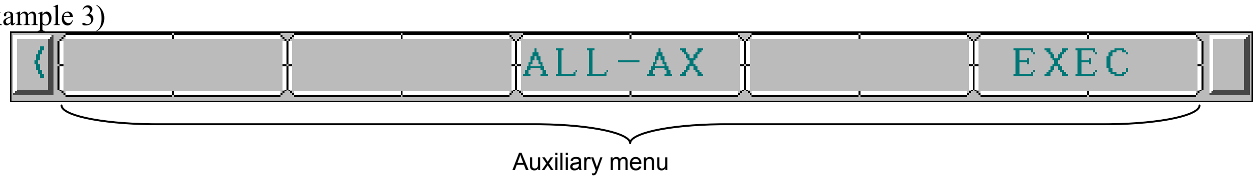
3 When the screen of a desired chapter is displayed, press the operation selection key to display operations (operation selection soft keys). If address/numeric keys are used, operation selection soft keys may be displayed automatically.

Example 2)



4 Select a desired operation with the operation selection soft key.
Depending on the operation to be executed, an auxiliary menu of soft keys is displayed. Perform an operation according to the indications on the auxiliary menu.

Example 3)



5 To return to the display of chapter selection soft keys, press the return menu key.

A general screen display procedure is provided above.
The actual display procedure varies from one screen to another.
For details, see each description of operation.

- Button design change depending on soft key state

The soft keys to be displayed depend on the object to be selected.

- Chapter selection soft keys
- Operation selection soft keys
- Auxiliary menu of operation selection soft keys

Depending on the state, the button images of the soft keys change.
From the button images, which state the soft keys are assuming can be known.

Example 1)

For the 8.4-inch LCD display unit

Chapter selection soft keys

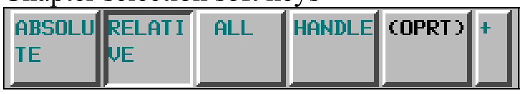
Operation selection soft keys

Operation selection keys, auxiliary menu

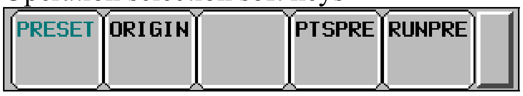
Example 2)

For the 10.4-inch LCD display unit

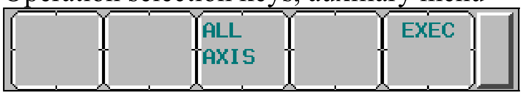
Chapter selection soft keys



Operation selection soft keys

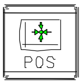


Operation selection keys, auxiliary menu

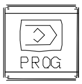


2.3.2 Function Keys

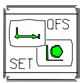
Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI panel:



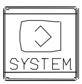
Press this key to display the position screen.



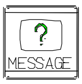
Press this key to display the program screen.



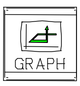
Press this key to display the offset/setting screen.



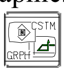
Press this key to display the system screen.



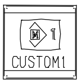
Press this key to display the message screen.



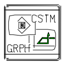
Press this key to display the graphics screen.

For the small MDI unit, press 

.



Press this key to display the custom screen 1 (conversational macro screen or C language executor screen).

For the small MDI unit, press 

.

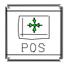
Press this key to display the custom screen 2 (conversational macro screen or C language executor screen).
For the small MDI unit, there is no key that corresponds to this key.

2.3.3 Soft Keys

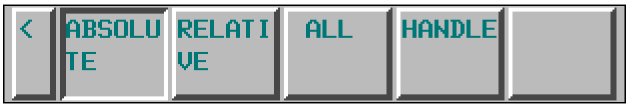
By pressing a soft key after a function key, the corresponding screen of the function can be displayed.

The chapter selection soft keys of each function are described below.
The four keys on the right-hand side are assigned to chapter selection soft keys. When multiple pages are used for chapter selection soft keys, [+] is displayed on the continuous menu key (rightmost soft key). Press the continuous menu key to switch between chapter selection soft keys.

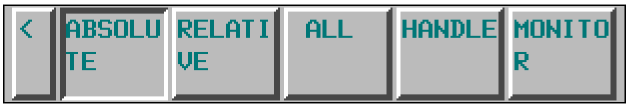
> **NOTE**
> 1 Press function keys to switch between screens that are used frequently.
> 2 Some soft keys are not displayed depending on the option configuration or parameter setting.

For the 10.4-inch LCD display unit, when pressing other than function key 

indicates positional display on the left side of the screen, the left half of the soft keys are shown below.

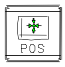


or



As for the soft key [MONITOR], refer to Section III-12.8.
Refer to the next page for other soft keys.

Position display screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

		(1)	(2)	(3)	(4)	(5)	
Page 1		ABS	REL	ALL	HNDL	(OPRT)	+

		(6)	(7)	(8)	(9)	(10)	
Page 2		MONI				(OPRT)	+

Table 2.3.3 (a) Position display screen

No.	Chapter menu	Description
(1)	ABS (ABSOLUTE)	Selects the absolute coordinate display screen.
(2)	REL (RELATIVE)	Selects the relative coordinate display screen.
(3)	ALL (ALL)	Selects the overall coordinate display screen.
(4)	HNDL (HANDLE)	Selects the operation screen for manual handle operation.
(6)	MONI (MONITOR)	Selects the screen for displaying the servo axis load meter, serial spindle load meter, and speedometer.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

Program screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

In the MEM/RMT mode

		(1)	(2)	(3)	(4)	(5)	
Page 1		PROGRM	CHECK	CURRENT	NEXT	(OPRT)	+

		(6)	(7)	(8)	(9)	(10)	
Page 2		RESTART	DIR			(OPRT)	+

In the MDI mode

		(1)	(2)	(3)	(4)	(5)	
Page 1		PROGRM	MDI	CURRENT	NEXT	(OPRT)	+

		(6)	(7)	(8)	(9)	(10)	
Page 2		RESTART	DIR			(OPRT)	+

In the EDIT/TJOG/THND mode

		(1)	(2)	(3)	(4)	(5)	
Page 1		PROGRM	DIR		C.A.P	(OPRT)	+

In the JOG/HND/REF mode

		(1)	(2)	(3)	(4)	(5)	
Page 1		PROGRM		CURRENT	NEXT	(OPRT)	+

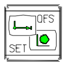
		(6)	(7)	(8)	(9)	(10)	
Page 2		RESTART	DIR			(OPRT)	+

Table 2.3.3 (b) Program

No.	Chapter menu	Description
(1)	PROGRM (PROGRAM)	Selects the screen for displaying a list of part programs currently registered.
(2)	CHECK	Selects the program check screen for displaying program and axis positions and modal command values. (Only for the 8.4- or 10.4-inch display unit used for simultaneous 2-path displays)
(2)	MDI (MDI)	Selects the screen for editing and displaying a program in the MDI mode. (Only in the MDI mode)
(3)	CURRENT	Selects the screen for displaying the modal command value and the command value of the block currently being executed from command values. (Only for the 8.4-inch display unit)
(4)	NEXT (NEXT BLOCK)	Selects the screen for displaying the command value of the block currently being executed and the command value of the block to be executed next from command values.
(4)	C.A.P (C.A.P)	Selects the screen for figure conversational input or the screen of MANUAL GUIDE 0i.
(6)	RESTART (RESTART)	Selects the operation screen for restarting an interrupted program operation.
(2) (7)	DIR (DIR)	Selects the screen for displaying a list of part programs currently registered.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

Offset/setting screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

		(1)	(2)	(3)	(4)	(5)	
Page 1		OFFSET	SETTING	WORK		(OPRT)	+

		(6)	(7)	(8)	(9)	(10)	
Page 2		MACRO	MENU	OPR	TOOLLF	(OPRT)	+

		(11)	(12)	(13)	(14)	(15)	
Page 3		OFST.2	W.SHFT		BARRIER	(OPRT)	+

		(16)	(17)	(18)	(19)	(20)	
Page 4			PR-LEV			(OPRT)	+

		(21)	(22)	(23)	(24)	(25)	
Page 5			LANG.	PROT.	GUARD	(OPRT)	+

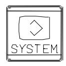
Table 2.3.3 (c) Offset

No.	Chapter menu	Description
(1)	OFFSET (OFFSET)	Selects the screen for setting tool offset values.
(2)	SETTING (SETTING)	Selects the screen for setting the setting parameters.
(3)	WORK (WORK)	Selects the screen for setting a workpiece coordinate system offset.
(6)	MACRO (MACRO)	Selects the screen for setting macro variables.
(7)	MENU (PATTERN MENU)	Selects the screen for setting pattern data. (Pattern data input)
(8)	OPR (OPERAT PANEL)	Selects the screen for operating part of the operation switches on the machine operator's panel as soft switches on the CNC screen. (Software operator's panel)
(9)	TOOLLF (TOOL LIFE)	Selects the screen for setting tool life data.
(11)	OFST.2 (Y OFFSET)	Selects the screen for setting Y-axis offsets. (Only for the T series)
(12)	W.SHFT (WORK SHIFT)	Selects the screen for setting workpiece coordinate system shift values. (Only for the T series)

No.	Chapter menu	Description
(14)	BARRIER (BARRIER)	Selects the chuck tail stock barrier screen. (Only for the T series)
(17)	PR-LEV (PRECI LEVEL)	Selects the screen for setting precision levels. (Machining condition selection function)
(22)	LANG. (LANGUAGE)	Selects the screen for setting a display language.
(23)	PROT. (PROTECT)	Selects the screen for setting 8-level data protection.
(24)	GUARD (GUARD)	Selects the screen for setting wrong operation prevention.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

System screen

The chapter selection soft keys that belong to the function key 

and the function of each screen are described below.

Page 1

	(1)	(2)	(3)	(4)	(5)	
	PARAM	DGNOS		SYSTEM	(OPRT)	+

Page 2

	(6)	(7)	(8)	(9)	(10)	
		PITCH	SV.SET	SP.SET	(OPRT)	+

Page 3

	(11)	(12)	(13)	(14)	(15)	
	W.DGNS	ALL IO		OPEHIS		+

Page 4

	(16)	(17)	(18)	(19)	(20)	
	PMCMNT	PMCLAD	PMCCNF	PM.MGR	(OPRT)	+

Page 5

	(21)	(22)	(23)	(24)	(25)	
	COLOR	MAINTE	M-INFO		(OPRT)	+

Page 6

	(26)	(27)	(28)	(29)	(30)	
		FSSB	PRMSET		(OPRT)	+

Page 7

	(31)	(32)	(33)	(34)	(35)	
	EMBED	PCMCIA	ETHBRD		(OPRT)	+

		(36)	(37)	(38)	(39)	(40)	
Page 8		RMTDIAG	M-TUN			(OPRT)	+

		(41)	(42)	(43)	(44)	(45)	
Page 9		ID-INF		MEMORY		(OPRT)	+

		(46)	(47)	(48)	(49)	(50)	
Page 10		PROF.M	PROF.S			(OPRT)	+

Table 2.3.3 (d) System

No.	Chapter menu	Description
(1)	PARAM (PARAMETER)	Selects the screen for setting parameters.
(2)	DGNOS (DIAGNOSIS)	Selects the screen for displaying CNC state.
(4)	SYSTEM (SYSTEM)	Selects the screen for displaying the current system status.
(7)	PITCH (PITCH ERROR)	Selects the screen for setting pith error compensation.
(8)	SV.SET (SERVO SETTING)	Selects the screen for setting the servo-related parameters.
(9)	SP.SET (SPINDLE SETTING)	Selects the screen for spindle-related setting.
(11)	W.DGNS (WAVE DIAG)	Selects the screen for displaying data such as servo positional deviation values, torque values, machine signals, and so forth as graphs.
(12)	ALL IO (ALL IO)	Selects the screen for inputting or outputting data.
(14)	OPEHIS (OPERAT HISTRY)	Selects the screen for displaying the history of operations performed by the operator and issued alarms.
(16)	PMCMNT (PMC MAINTE)	Selects the screen related to PMC maintenance such as PMC signal state monitoring and tracing, and PMC parameter display/editing.
(17)	PMCLAD (PMC LADDER)	Selects the screen related to ladder display/editing.
(18)	PMCCNF (PMC CONFIG)	Displays the screen for displaying/editing data other than ladders that makes up a sequence program and for setting the PMC function.
(19)	PM.MGR (P.MATEMGR.)	Select the screen of Power Mate CNC Manager.
(21)	COLOR (COLOR)	Selects the screen for setting colors to be used on the screen.
(22)	MAINTE (PERIOD MAINTE)	Selects the screen for setting maintenance items to be managed periodically.
(23)	M-INFO (MAINTE INFO)	Selects the screen for displaying information about maintenance performed.
(27)	FSSB (FSSB)	Selects the screen for making settings related to the high-speed serial servo bus (FSSB: Fanuc Serial Servo Bus).
(28)	PRMSET (PARAMETER)	Selects the screen for setting parameters necessary for start-up and tuning.

No.	Chapter menu	Description
(31)	EMBED (EMBED PORT)	Selects the screen for making settings related to the embedded Ethernet (embedded port).
(32)	PCMCIA (PCMCIA LAN)	Selects the screen for making settings related to the embedded Ethernet (PCMCIA Ethernet card).
(33)	ETHBRD (ETHER BOARD)	Selects the screen for making settings related to the fast Ethernet/fast data server.
(36)	RMTDIAG (REMOTE DIAG)	Selects the screen for making settings related to remote diagnosis.
(37)	M-TUN (MCHN TUNING)	Displays the screen for setting the parameter set for emphasis on speed (LV1) or emphasis on precision (LV10).
(43)	MEMORY (MEMORY)	Selects the screen for displaying the contents of memory.
(46)	PROF.M (PROFI MASTER)	Selects the screen for making settings related to the Profibus master function.
(47)	PROF.S (PROFI SLAVE)	Selects the screen for making settings related to the Profibus slave function.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

Message screen

The chapter selection soft keys that belong to the function key ? MESSAGE and the function of each screen are described below.

		(1)	(2)	(3)	(4)	(5)	
Page 1		ALARM	MSG	HISTRY			+
		(6)	(7)	(8)	(9)	(10)	
Page 2			MSGHIS				+
		(11)	(12)	(13)	(14)	(15)	
Page 3		EMB LOG	PCM LOG	BRD LOG			+

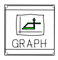
Table 2.3.3 (e) Message

No.	Chapter menu	Description
(1)	ALARM (ALARM)	Selects the alarm message screen.
(2)	MSG (MESSAGE)	Selects the operator message screen.
(3)	HISTRY (HISTRY)	Selects the screen for displaying the details of alarms issued so far.
(7)	MSGHIS (MESAGE HISTRY)	Selects the external operator message screen.
(11)	EMB LOG (EMBED LOG)	Selects the screen for displaying error messages related to the embedded Ethernet (embedded port).
(12)	PCM LOG (PCMCIA LOG)	Selects the screen for displaying error messages related to the embedded Ethernet (PCMCIA Ethernet card).

No.	Chapter menu	Description
(13)	BRD LOG (BOARD LOG)	Selects the screen for displaying error messages related to the fast Ethernet/fast data server.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

Graphic screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

When the graphic display function is enabled:

	(1)	(2)	(3)	(4)	(5)	
	PARAM	GRAPH	GRAPH	LARGE	(OPRT)	

Table 2.3.3 (f) Graphic

No.	Chapter menu	Description
(1)	PARAM (PARAMETER)	Selects the screen for setting graphic parameters.
(2)	GRAPH (GRAPH)	Selects the screen for performing the graphic display of tool paths. (For the M series system)
(3)	GRAPH (GRAPH)	Selects the screen for performing the graphic display of tool paths. (For the T series system)
(4)	LARGE (LARGE)	Displays the soft key for setting the scaling factor of graphic display.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

When the dynamic graphic display function is enabled

M series :

	(1)	(2)	(3)	(4)	(5)	
	PARAM	EXEC	EXEC	POS	(OPRT)	+

	(6)	(7)	(8)	(9)	(10)	
	PARAM	3-PLN			(OPRT)	+

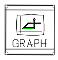
T series :

	(6)	(7)	(8)	(9)	(10)	
	PARAM		GRAPH	LARGE	(OPRT)	

Table 2.3.3 (g) Graphic (for dynamic graphic)

No.	Chapter menu	Description
(1) (6) (11)	PARAM (PARAMETER)	Selects the screen for setting drawing parameters.
(2)	EXEC (EXEC)	Selects the screen for drawing tool paths.
(3)	EXEC (EXEC)	Selects the screen for drawing animation.
(4)	POS (POSITION)	Selects the screen for displaying tool positions on tool paths during drawing tool paths.
(7)	3-PLN (3-PLN)	Selects the screen for displaying 3-plane drawings in animated simulation.
(13)	GRAPH (GRAPH)	Selects the screen for performing the graphic display of tool paths.
(14)	LARGE (LARGE)	Displays the soft key for setting the scaling factor of graphic display.

*1 The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

*2 Item (2) and item (3) are displayed alternately each time function key  is pressed.

2.3.4 Key Input and Input Buffer

When an address and a numeric key are pressed, the character corresponding to that key is input once into the key input buffer. The contents of the key input buffer is displayed at the bottom of the screen.
In order to indicate that it is key input data, a ">" symbol is displayed immediately in front of it. A "_" is displayed at the end of the key input data indicating the input position of the next character.

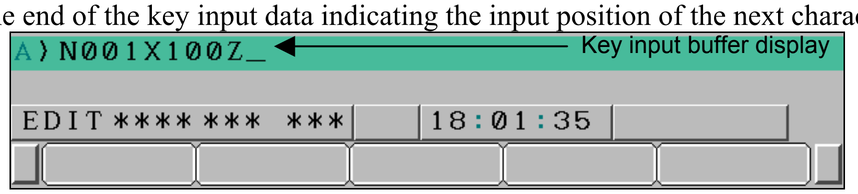


Fig. 2.3.4 (a) Key input buffer display

- Switching between upper and lower key characters

To input the upper character or symbol of the keys that have two characters inscribed on them, first press the  key and then the key in question.

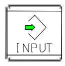
When the  key is pressed, "_" indicating the input position of the next character changes to "∧" and the upper character can be input. This status is called the shift status.

When a character is input in shift status the shift status is canceled. Furthermore, if the  key is pressed in shift status, the shift status is canceled.
It is possible to input up to 128 characters at a time in the key input buffer.

Press the  key to cancel a character or symbol input in the key input buffer.

(Example)
When the key input buffer displays
>N001X100Z_
and the cancel  key is pressed, Z is canceled and
>N001X100_
is displayed.

2.3.5 Warning Messages

After a character or number has been input from the MDI panel, a data check is executed when  key or a soft key is pressed. In the case of incorrect input data or the wrong operation a flashing warning message will be displayed on the status display line.

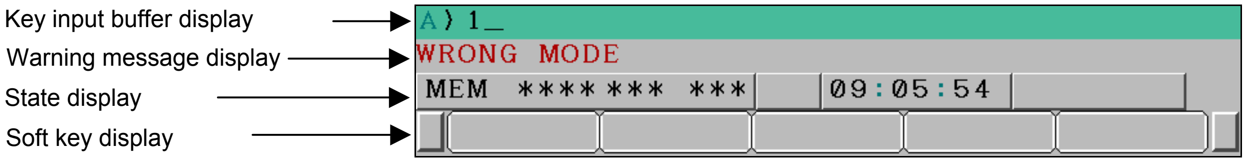


Fig. 2.3.5 (a) Warning message display

Table 2.3.5 (a) Warning Messages

Warning message	Content
FORMAT ERROR	The format is incorrect.
WRITE PROTECT	Key input is invalid because of data protect key or the parameter is not write enabled.
DATA IS OUT OF RANGE	The input value exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.
WRONG MODE	Parameter input is not possible in any mode other than MDI mode.
EDIT REJECTED	It is not possible to edit in the current CNC status.
CANNOT USE I/O DEVICE	Because other functions occupy I/O device, I/O device cannot be used.

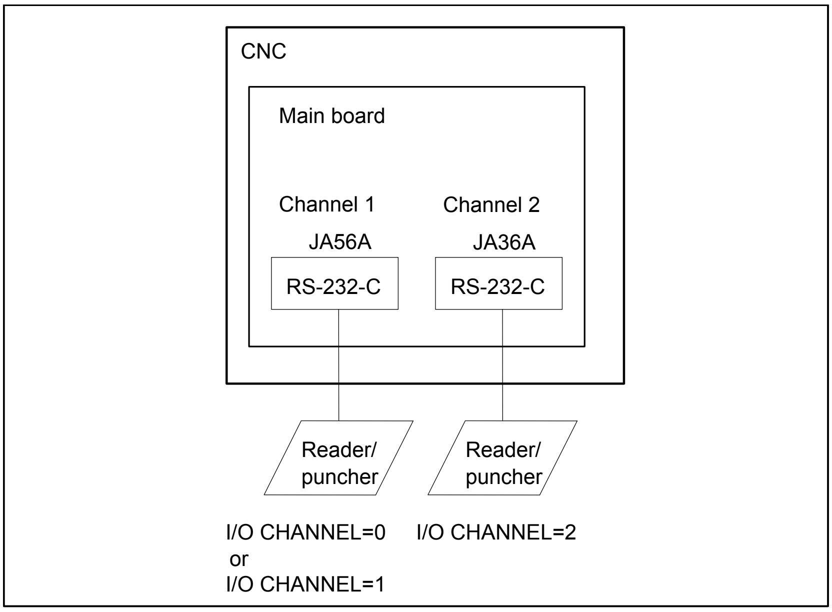
2.4 EXTERNAL I/O DEVICES

External I/O devices such as a memory card are available.
By using an external I/O device such as a memory card, the following data can be input or output:
1. Programs
2. Offset data
3. Parameters
4. Custom macro common variables

For how to input or output data and input data from or output it to a memory card, see III-8.

- Parameter setting

Before an external input/output device can be used, parameters must be set as follows.



This CNC has a total of two channels of reader/puncher interfaces. It also has a memory card interface. The input/output device to be used is specified by setting the channel (interface) connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance. (These settings are not required for the memory card interface.)
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the reader/puncher interface parameters for the channels.

0020 I/O CHANNEL or foreground input

Set channels to be used for data input/output.

I/O CHANNEL (0 to 9)
=0 : Channel 1
=1 : Channel 1
=2 : Channel 2
=3 : Channel 3
:
:
:

Input/output to and from the memory card interface, etc. is also possible.

When bit 0 (IO4) of parameter No.0110 is set

0021 Foreground output

0022 Background input

0023 Background input

The channel setting is the same as No.0020.

Input/output channel number (parameter No.0020)

↓

I/O CHANNEL=0 (Channel 1)
- 0101 Stop bit and other data
- 0102 Number specified for the input/output device
- 0103 Baud rate

I/O CHANNEL=1 (Channel 1)
- 0111 Stop bit and other data
- 0112 Number specified for the input/output device
- 0113 Baud rate

I/O CHANNEL=2 (Channel 2)
- 0121 Stop bit and other data
- 0122 Number specified for the input/output device
- 0123 Baud rate

:
:
:

I/O CHANNEL=9

2.5 POWER ON/OFF

2.5.1 Turning on the Power

Procedure of turning on the power

Procedure

1. Check that the CNC or machine is visually normal. (For example, check that front door and rear door are closed.)
2. Turn on the power according to the manual issued by the machine tool builder.
3. After the power is turned on, check that the position screen is displayed. An alarm screen is displayed if an alarm occurs upon power-on.

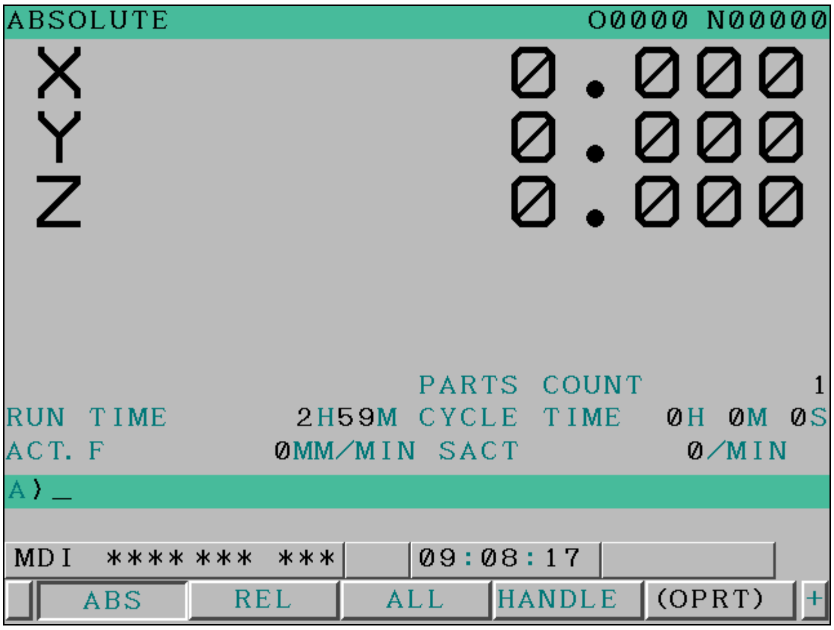


Fig. 2.5.1 (a) Position screen (example for the 8.4-inch display unit)

4. Check that the fan motor is rotating.

> ⚠ **WARNING**
> Until the positional or alarm screen is displayed at the power on, do not touch them. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

2.5.2 Power Disconnection

Procedure of power disconnection

Procedure

1. Check that the LED indicating the cycle start is off on the operator's panel.
2. Check that all movable parts of the CNC machine tool is stopping.
3. If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.
4. Continue to press the <POWER OFF> button for about 5 seconds.
5. Refer to the machine tool builder's manual for turning off the power to the machine.

3 MANUAL OPERATION

MANUAL OPERATION are eight kinds as follows :

3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :
The tool is moved in the direction specified in bit 5 (ZMI) of parameter No.1006 for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters Nos. 1424,1421, and 1425.
Four step rapid traverse override is effective during rapid traverse.
When the tool has returned to the reference position, the reference position return completion LED goes on. The tool generally moves along only a single axis, but can move along three axes simultaneously when specified so in bit 0 (JAX) of parameter No.1002.

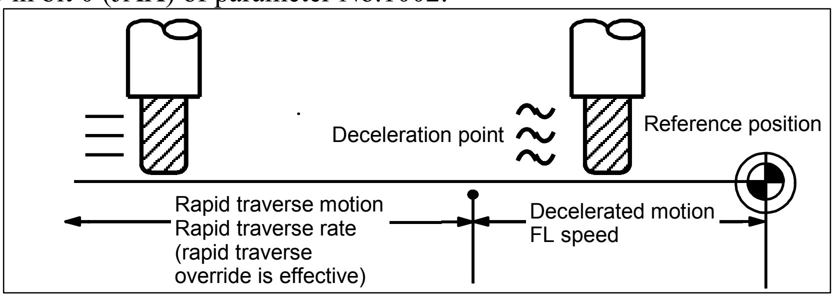


Fig. 3.1 (a) Manual reference position return

Procedure for manual reference position return

Procedure

1 Press the reference position return switch, one of the mode selection switches.
2 To decrease the feedrate, press a rapid traverse override switch.
3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter. When the tool has returned to the reference position, the reference position return completion LED goes on.
4 Perform the same operations for other axes, if necessary.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

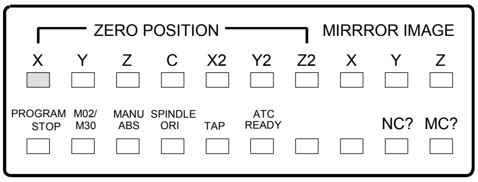


Fig. 3.1 (b)

Explanation

- Automatically setting the coordinate system

Bit 0 (ZPR) of parameter No.1201 is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.

When α, β and γ are set in parameter 1250, the workpiece coordinate system is determined so that reference point on the tool holder or the position of the tip of the reference tool is X= α, Y = β, Z = γ when reference position return is performed. This has the same effect as specifying the following command for reference position return:

G92X$\underline{\alpha}$Y$\underline{\beta}$Z$\underline{\gamma}$;

When the workpiece coordinate system is used (bit 0 (NWZ) of parameter No. 8136 is 0), this function cannot be used.

Limitation

- Moving the tool again

Once the reference position return completion LED lights at the completion of reference position return, the tool does not move unless the reference position return switch is turned off.

- Reference position return completion LED

The reference position return completion LED is extinguished by either of the following operations:

- Moving from the reference position.
- Entering an emergency stop state.

- The distance to return to reference position

For the distance (Not in the deceleration condition) to return the tool to the reference position, refer to the manual issued by the machine tool builder.

3.2 JOG FEED (JOG)

In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.

The jog feedrate is specified in a parameter No.1423.

The jog feedrate can be adjusted with the jog feedrate override dial.

Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate No. 1424 regardless of the position of the jog feedrate override dial. This function is called the manual rapid traverse.

Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by bit 0 (JAX) of parameter No.1002.

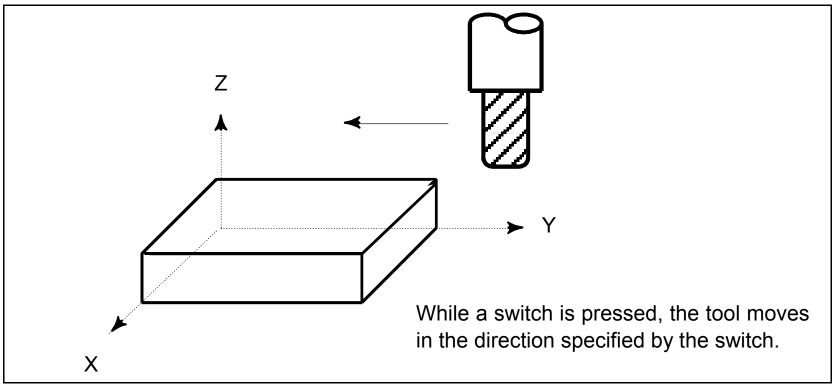


Fig. 3.2 (a) Jog Feed (JOG)

Procedure for JOG feed

Procedure

1. Press the jog switch, one of the mode selection switches.
2. Press the feed axis and direction selection switch corresponding to the axis and the direction where the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in the parameter No. 1423. The tool stops when the switch is released.
3. The jog feedrate can be adjusted with the jog feedrate override dial.
4. Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Manual per revolution feed

The manual per revolution feed is enabled for jog feed by setting bit 4 (JRV) of parameter No.1402.
In manual per revolution feed, jog feed is performed at the feedrate of the feed amount per spindle revolution set by parameter No. 1423 multiplied by the jog feedrate override multiplied by the spindle speed.
During manual per revolution feed, the tool is jogged at the following feedrate:
Feed distance per rotation of the spindle (mm/rev) (specified with parameter No. 1423) × JOG feedrate override × actual spindle speed (rev/min).

Limitation

- Acceleration/deceleration for rapid traverse

Feedrate, time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

- Change of modes

Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode at first, then press a feed axis and direction selection switch.

- Rapid traverse prior to reference position return

If reference position return is not performed after power-on, pushing rapid traverse button does not actuate the rapid traverse but the remains at the JOG feedrate. This function can be disabled by setting bit 0 (RPD) of parameter No.1401.

3.3 INCREMENTAL FEED

In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.
With using bit 2 (HNT) of parameter No. 7103, each step can be additionally 10 times the lest input increment.
The feedrate set in parameter No. 1423 is applied.
By using the manual feedrate override signal, the feedrate can be increased or decreased.
The tool can also be moved at the rapid traverse rate by using the manual rapid traverse selection signal, independent of the manual feedrate override signal.

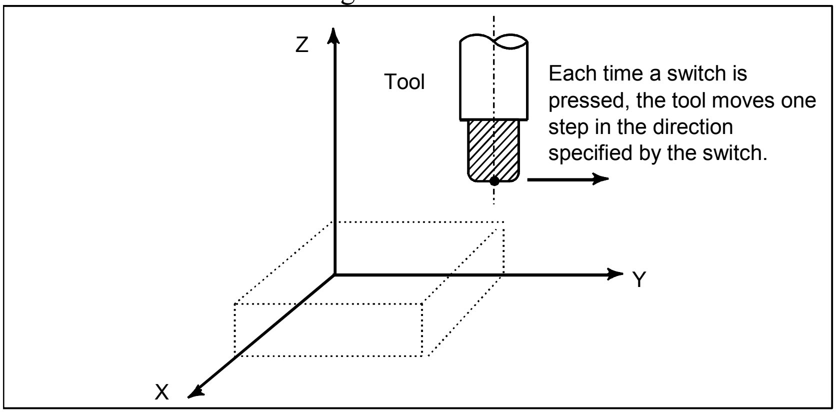


Fig. 3.3 (a) Incremental feed

Procedure for incremental feed

Procedure

1. Press the INC switch, one of the mode selection switches.
2. Select the distance to be moved for each step with the magnification dial.
3. Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.
4. Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate. Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Travel distance specified with a diameter

For specification with a diameter, the travel distance is a diameter value.

3.4 MANUAL HANDLE FEED

In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. It can be multiplied by one of four scaling factors: 1, 10, and two arbitrary values set in parameters Nos. 7113 and 7114. It is possible to set arbitrary scaling factors for each axis (set in parameters Nos. 12350 and 12351) as well as arbitrary scaling factors common to all axes (set in parameters Nos. 7113 and 7114). If parameter No. 12350 is not set, the setting of parameter No. 7113 is used. If parameter No. 12351 is not set, the setting of parameter No. 7114 is used. With bit 2 (HNT) of parameter No. 7103, the minimum distance can be further 10 times greater.
The above parameters are valid for manual handle interruption.

The number of manual pulse generators is given below.

T

- Up to two (Two axes can be moved at the same time.)
  Up to three with an optional function for 0*i*-TD.

M

- Up to three (Three axes can be moved at the same time.)

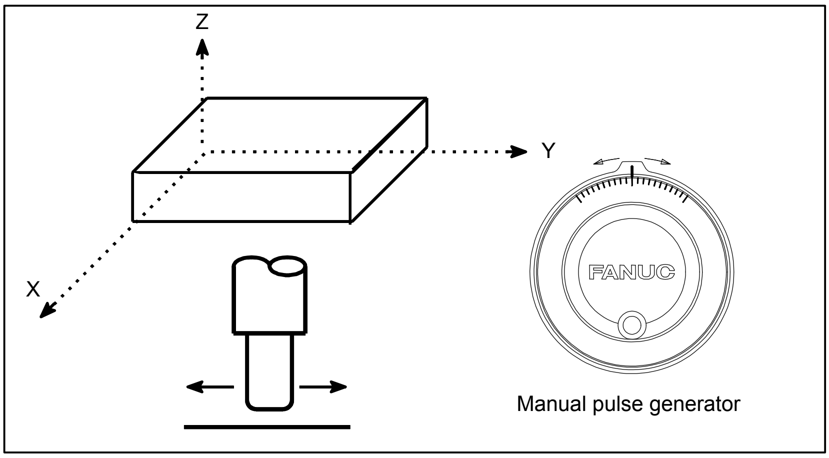


Fig. 3.4 (a) Manual handle feed

Procedure for manual handle feed

Procedure

1. Press the handle switch, one of the mode selection switches.
2. Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.
3. Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.
4. Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Availability of manual pulse generator in Jog mode (JHD)

When bit 0 (JHD) of parameter No. 7100 is set to 1, both jog feed and manual handle feed can be used in JOG mode.
When bit 0 (JHD) of parameter No. 7100 is set to 1, both manual handle feed and incremental feed can be used in HANDLE mode.

- Manual handle feed in TEACH IN JOG mode (THD)

By setting bit 1 (THD) of parameter No. 7100, manual handle feed in TEACH IN JOG mode can be enabled or disabled.

- When manual handle feed exceeding the rapid traverse rate is specified

The amount of pulses exceeding the rapid traverse rate can be saved by CNC as B. And amount of pulses B will be output as pulses C.

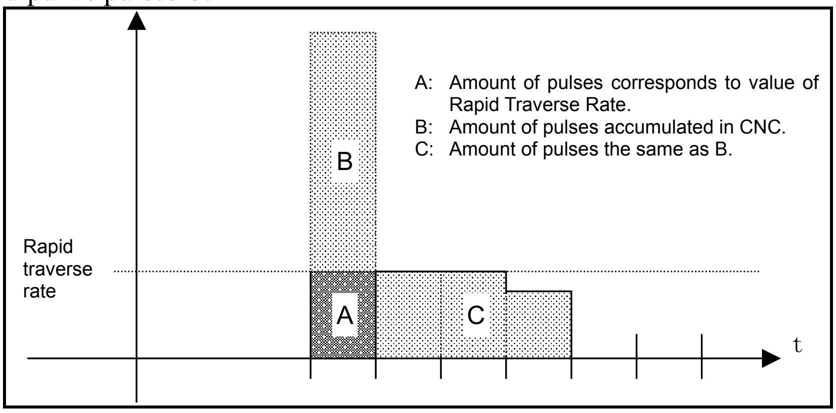


Fig. 3.4 (b) Amount of pulses output by CNC in Manual Handle Feed

Amount of pulses B is calculated in 2 cases as following:

In case of
1) Parameter No.7117 = 0
   The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0).

In case of
2) Parameter No.7117 > 0
   The feedrate is clamped as the Rapid Traverse Rate, but the pulses exceeding the Rapid Traverse Rate is not ignored. Amount of pulses accumulated in CNC is calculated as following. (Although the rotation of manual pulse generator is stopped, if there is pulses accumulated in CNC, it will be output and the tool will move as long as amount of it.)

   Magnification set by MP1, MP2<Gn019.4,5> is m, value of parameter No.7117 is n.

   $n < m$: Clamping is set performed at value of parameter No.7117.
   $n \geq m$: Amount A+B, shown in figure, which's value is multiple of m and small than n. As a result, clamping is performed as an integral multiple of the selected magnification.

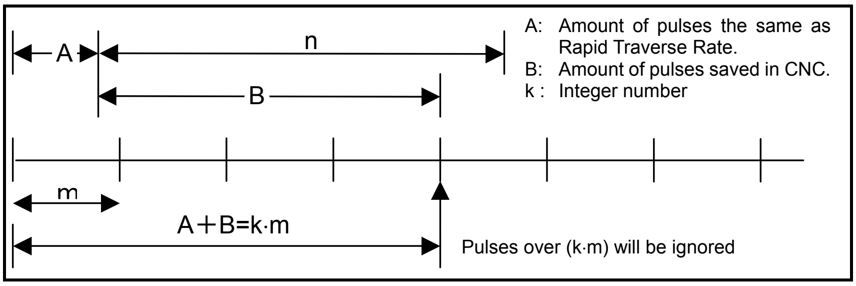


Fig. 3.4 (c) Amount of pulses exceeding the Rapid Traverse Rate (n ≥ m)

NOTE

Due to change of mode, clamping can be performed not as an integral multiple of the selected magnification.
The distance the tool moves may not match the graduations on the manual pulse generator.

- Upper feedrate limit in manual handle feed

The upper feedrate limit depends on the input signal (maximum manual handle feedrate switch signal HNDLF) from the PMC as follows:

- When HNDLF is set to 0, the feedrate is clamped to the manual rapid traverse rate (parameter No. 1424).
- When HNDLF is set to 1, the feedrate is clamped to the feedrate set in parameter No. 1434.

- Movement direction of an axis to the rotation of MPG (HNGx)

Bit 0 (HNGx) of parameter No 7102 switches the direction of MPG in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

This parameter is valid only for the following functions:

- Manual handle feed
- Manual handle interruption

Limitation

⚠ WARNING

Rotating the handle quickly with a large magnification such as ×100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

NOTE

Rotate the manual pulse generator at a rate of five rotations per second or lower.
If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

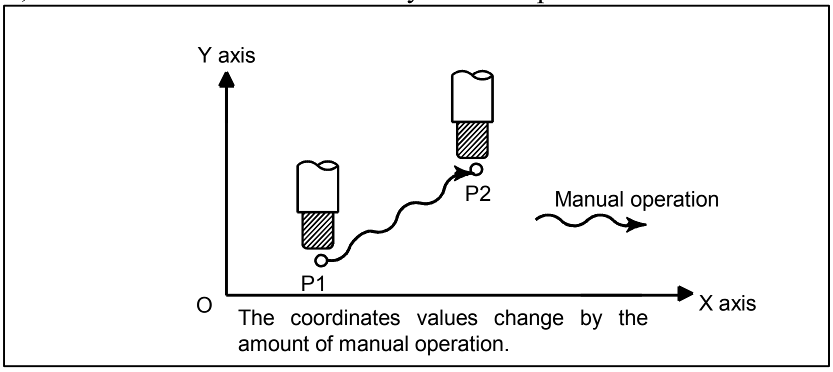


Fig. 3.5 (a) Coordinates with the switch ON

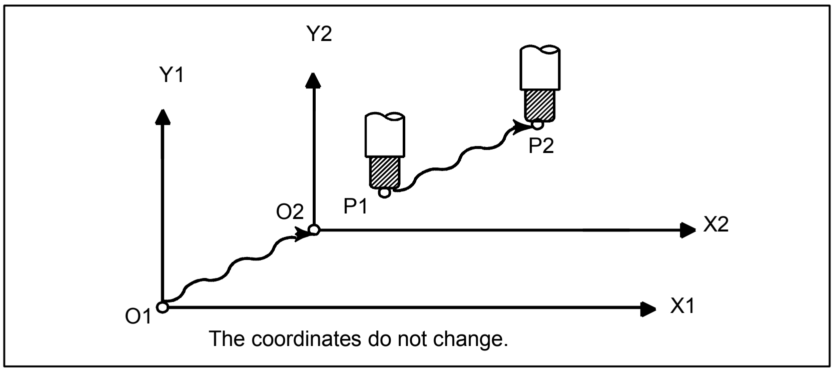


Fig. 3.5 (b) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01G90  X100.0Y100.0F10 ;          <1>
        X200.0Y150.0     ;          <2>
        X300.0Y200.0     ;          <3>
```

Fig. 3.5 (c) Program example

The subsequent figures use the following notation:

——→ Movement of the tool when the switch is on

- - -→ Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

- Manual operation after the end of block

Coordinates when block <1> has been executed after manual operation (X-axis +20.0, Y-axis +100.0) at the end of movement of block <2>.

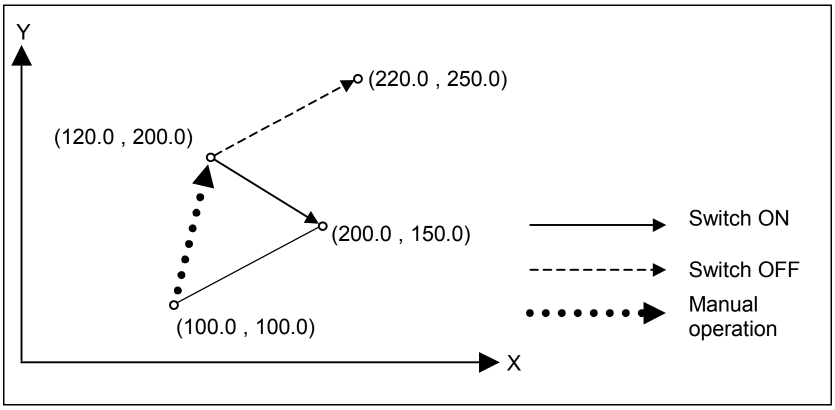


Fig. 3.5 (d) Manual operation after the end of block

- Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis + 75.0) is performed, and the cycle start button is pressed and released.

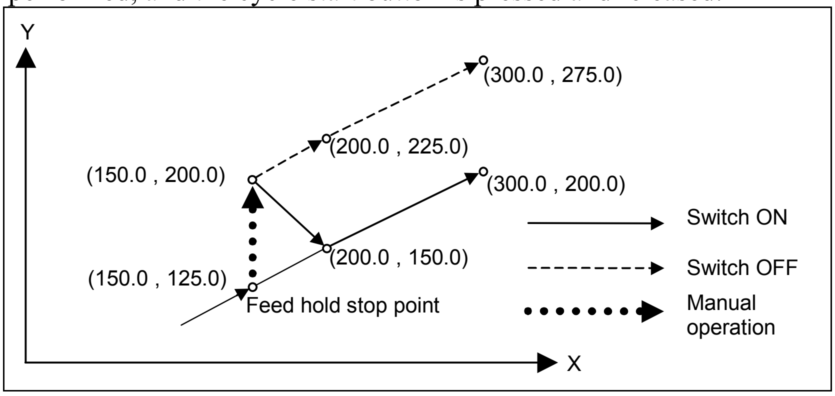


Fig. 3.5 (e) Manual operation after a feed hold

- When reset after a manual operation following a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis +75.0) is performed, the control unit is reset with the RESET button, and block <2> is read again.

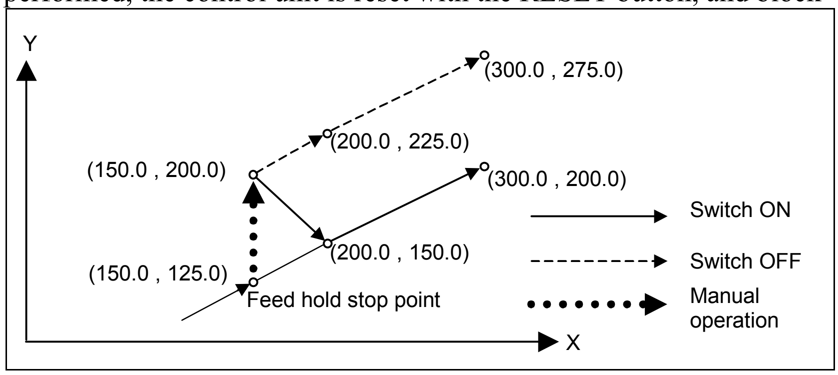


Fig. 3.5 (f) When reset after a manual operation following a feed hold

- When a movement command in the next block is only one axis

When there is only one axis in the following command, only the commanded axis returns.

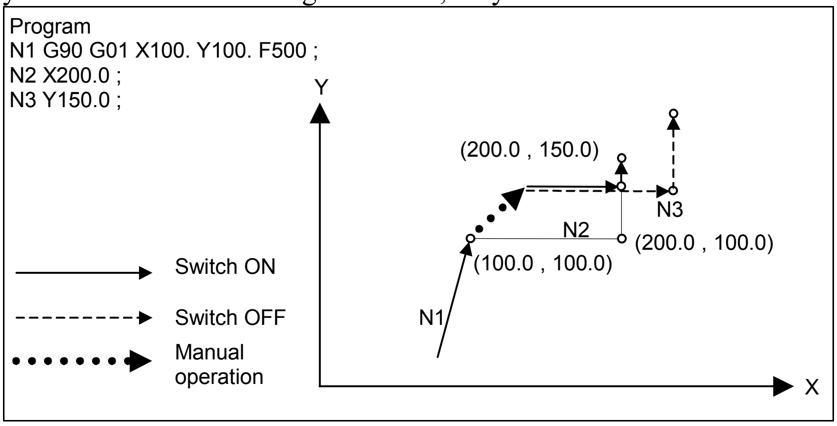


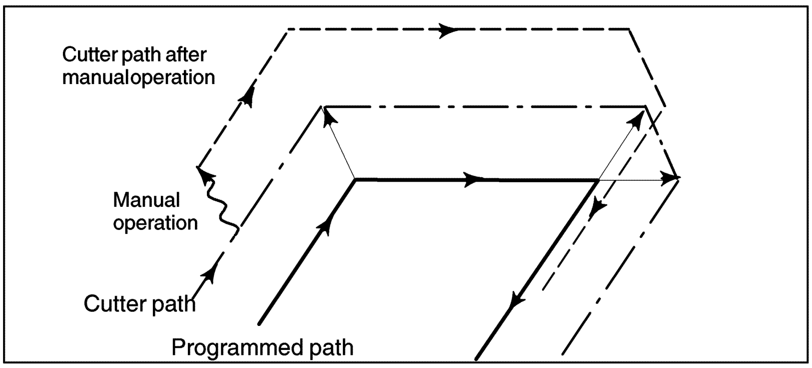
Fig. 3.5 (g) When a movement command in the next block is only one axis

- When the next move block is an incremental

When the following commands are incremental commands, operation is the same as when the switch is OFF.

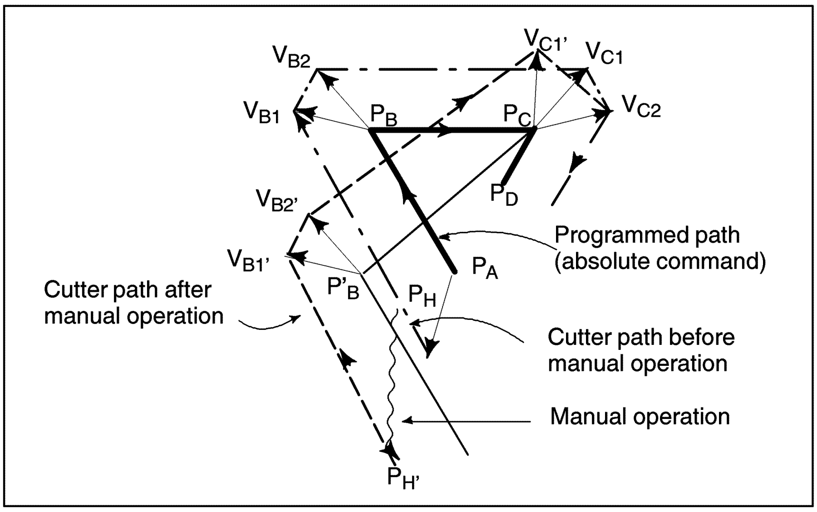
- Manual operation during cutter or tool nose radius compensation

- When the switch is OFF
  After manual operation is performed with the switch OFF during cutter or tool nose radius compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed.
  The amount of separation equals to the amount that was performed manually.

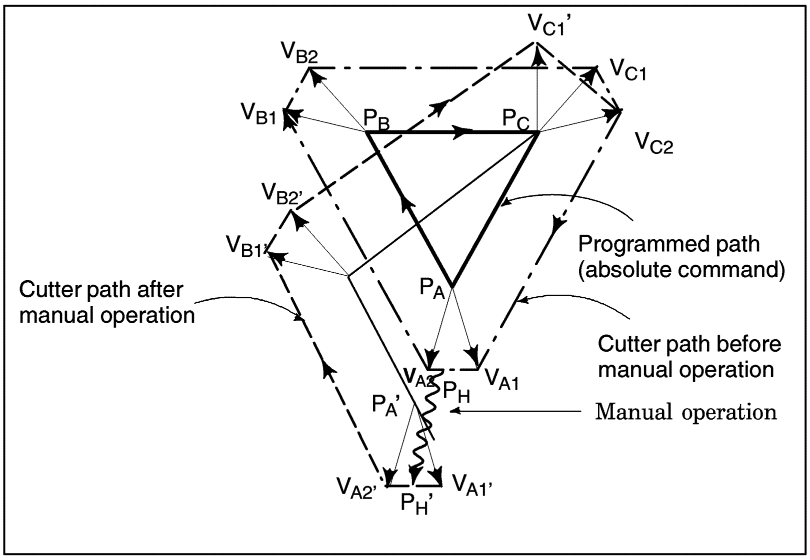


- When the switch is ON during cutter or tool nose radius compensation
  Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the cutter or tool nose radius compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

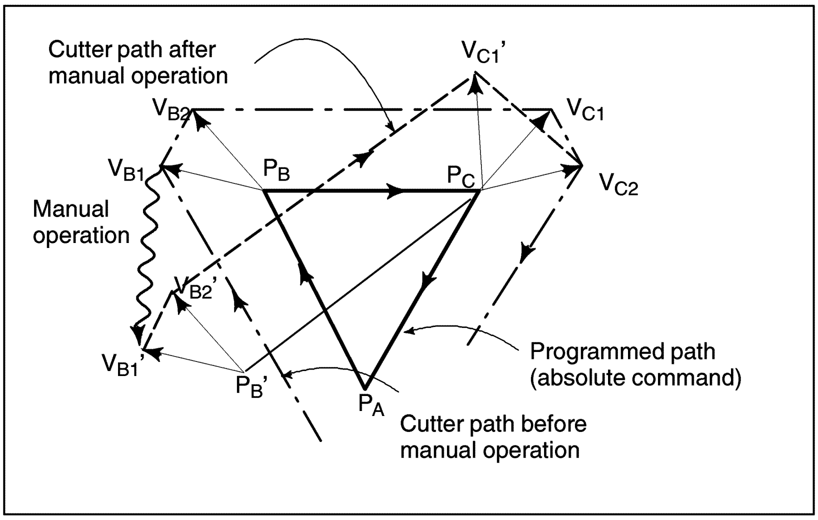
- Manual operation performed in other than cornering
  Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to PH'. The block end point P_B moves to the point P_B' by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to V_{B1}' and V_{B2}'. Vectors V_{C1} and V_{C2} between the next two blocks P_B - P_C and P_C - P_D are discarded and new vectors V_{C1}' and V_{C2}' ($V_{C2}' = V_{C2}$ in this example) are produced from the relation between P_B' - P_C and P_C - P_D. However, since V_{B2}' is not a newly calculated vector, correct offset is not performed at block P_B' - P_C. Offset is correctly performed after P_C.



- Manual operation during cornering
  This is an example when manual operation is performed during cornering. V_{A2}', V_{B1}', and V_{B2}' are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct cutter or tool nose radius compensation is performed for the blocks following P_C.



- Manual operation after single block stop
  Manual operation was performed when execution of a block was terminated by single block stop. Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub-sequent processing is the same as case a described above. An MDI operation can also be intervened as well as manual operation. The movement is the same as that by manual operation.



3.6 DISTANCE CODED LINEAR SCALE INTERFACE

Overview

The interval of each reference marks of distance coded linear scale are variable. Accordingly, if the interval is determined, the absolute position can be determined. The CNC measures the interval of reference marks by axis moving of short distance and determines the absolute position. Consequently the reference position can be established without moving to reference position.

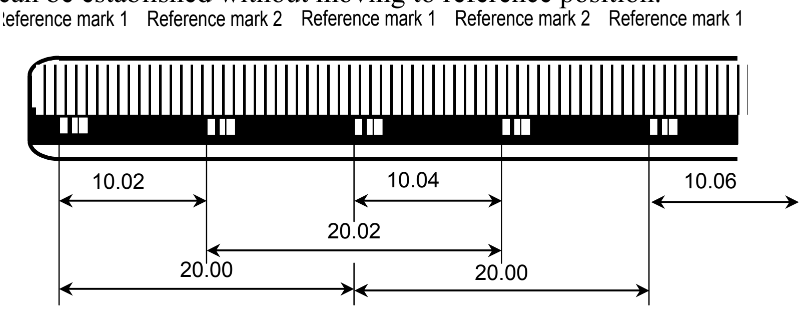


Fig. 3.6 (a) Example of distance coded linear scale

This is an optional function.

3.6.1 Procedure for Reference Position Establishment

Procedure

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter (No.1425) setting).
(4) When a reference mark is detected, the axis stops, then the axis is fed at a constant low speed again.
(5) Above (4) is executed repeatedly until two, three or four reference marks are detected. And absolute position is determined and reference position establishment signal (ZRF1,ZRF2,ZRF3, …) turns to "1".
(A number of reference marks is determined by bit 2 (DC2x) and 1 (DC4x) of parameter No.1802.)

The timing chart for this procedures is given below.

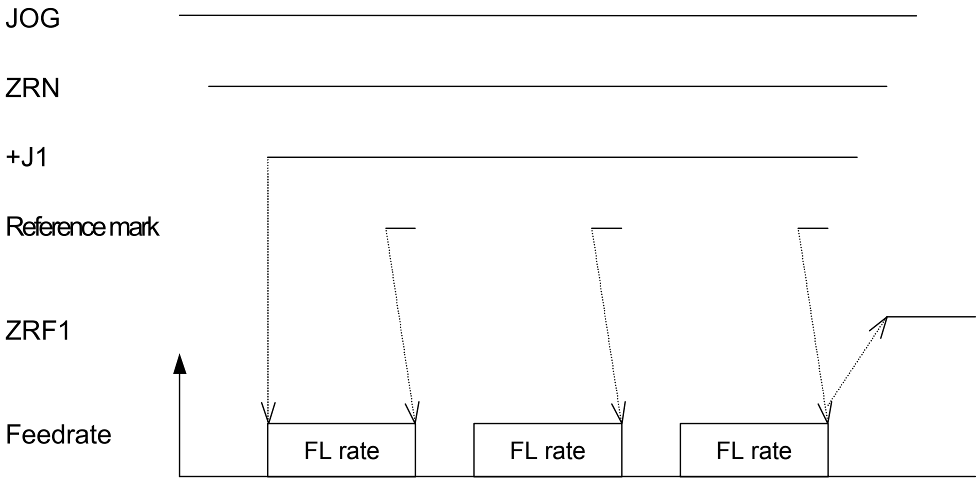


Fig. 3.6.1 (a) Timing chart for reference position establishment

- Procedure for establishing a reference position through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (5) above are performed automatically.
After the reference position is established, the automatic reference position return is performed.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of the following operations is performed in steps (3) to (5), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position

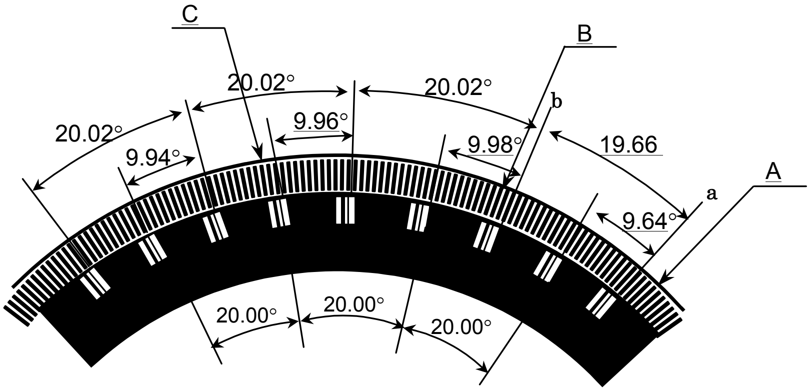
If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

3.6.2 Reference Position Return

(1) When the reference position is not established and the axis moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the reference position establishment procedure is executed.
(2) When the reference position is already established and the axis is moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the axis is moved to the reference point without executing the reference position establishment procedure.
(3) When the reference position is not established and the reference position return command (G28) is executed, the reference position establishment procedure is executed. The next movement the axis depends on the setting of parameter RFS (No.1818#0).
(4) When the reference position is already established and the reference position command (G28) is executed, the movement of the axis depends on the setting of bit 1 (RF2) of parameter No.1818.

3.6.3 Distance Coded Rotary Encoder

In case of setting a rotary axis, if bit 3 (DCRx) of parameter No.1815 is set, the setting axis is regarded as being equipped with a distance coded rotary encoder.
In case of distance coded rotary encoder, the marker interval may be different from parameter setting value. (a-b section of the following figure) When the reference point return is executed through this section, it is not able to establish the reference point. Therefore, in case of distance coded rotary encoder, if the reference point return is started for B point from A point of below figure, the reference point is not established yet at B point. The reference point return is re-started for C point. The reference point return procedure is finished at C point.



- When using the distance coded rotary encoder in the case of parameter rotary axis B type (bits 0 and 1 of parameter No. 1006 are 1 and 1, respectively (the machine coordinate system of the rotary axis is of linear axis type)), even if the machine turns more than one turn, the reference position established by this function is rounded to a movement amount per revolution of the rotary axis.
- When using the distance coded rotary encoder, only 3-point measurement or 4-point measurement is enabled; 2-point measurement (bit 2 (DC2) of parameter No. 1802) is disabled.

3.6.4 Axis Synchronization Control

Requirements when this function is used with axis synchronization control axes

When this function is used with axis synchronization control axes, the distance coded linear scale used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
(Set identical values in parameters Nos. 1821 and 1882 for both the master and slave axes.)
This function does not work unless the use of this function is specified
for both the master and slave axes (bit 2 (DCL) of parameter No. 1815 is 1).
Also, in all parameters related to this function, except parameter No. 1883, 1884 (distance from the scale zero point to reference position 1, 2), set identical values for both the master and slave axes.
If a parameter value for the master axis differs from the corresponding parameter value for the slave axis, alarm SV1051 is issued.

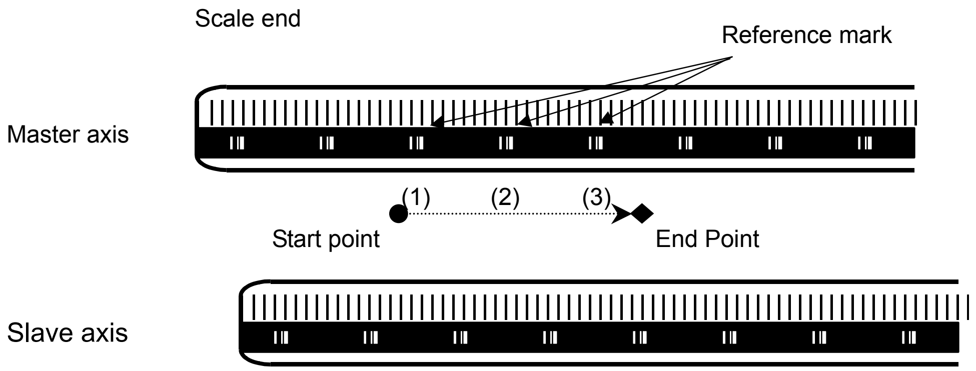
> **NOTE**
> When this function is used with axis synchronization control axes for which the operation mode is switched between synchronization operation and normal operation, this function is enabled only if the synchronization select signal (SYNC1 to SYNC5 <Gn138>) is 1. (During establishment of a reference position, the synchronization select signal status must be maintained.)

Reference position establishment with axis synchronization control axes

With axis synchronization control axes, a reference position is established as follows. When a reference mark for the master or slave axis is detected, a stop takes place temporarily. Then, a feed operation is performed again at the reference position return FL feedrate. This sequence is repeated until a reference mark is detected three or four times for both the master and slave axes. Then the absolute position is calculated for both the master and slave axes, and the reference position establish signals ZRF1, ZRF2, ... <F120> are set to 1.

After the reference position has been established by the above operation, a synchronization error is corrected. (Checking for excessive synchronization error alarm 2 is made even during reference position establishment.)

(Example of 3 points measurement system)



In this example, reference mark (1) of the master axis is first detected, a pause takes place, movement operation is performed at the FL feedrate, and a pause takes place again in a position where the reference mark of the slave axis is detected.

Then, movement operation is performed again, reference mark (2) of the master axis, a pause takes place when the reference mark of the slave axis, and reference mark (3) of the master axis are detected during movement at the FL feedrate, and reference position establishment operation of both axes is completed on the slave axis where the third reference mark is detected.

NOTE

In case of this function is used with axis synchronization control axes, if the value of parameters Nos. 1883 and 1884 for both the master and slave axes is 0, the reference position is not established. Also, the reference position establish signals ZRF1, ZRF2, ... <F120> are set to 0.

3.6.5 Axis Control by PMC

In PMC axis control, if the reference position return command (axis control command code 05H) is issued for an axis having a distance coded linear scale, reference position return is performed according to the reference position return sequence for the distance coded linear scale.

Specifically, the following operations take place:

Before reference position establishment	The reference position is established by detecting two, three or four reference marks. Movement to the reference position is not performed.
After reference position establishment	Positioning at the reference position is performed.

3.6.6 Angular Axis Control

There are the following limitations when the angular axis control is used.

(a) It is necessary to use the linear scale with the distance coded reference mark for both the perpendicular axis and the angular axis.

(b) When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020 is generated.

(c) During the reference point establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

3.6.7 Note

(1) In the case of the actual interval of reference marks is different from parameter setting value, the alarm DS1449 occurs.

(2) This function is disabled if any of the following conditions is satisfied:

- Either parameter No.1821 (mark-1 interval) or parameter No.1882 (mark-2 interval) is set to 0.
- The setting of parameter No.1821 is greater than or equal to the setting of parameter No.1882.
- The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting.
- The absolute-position detection function is enabled.(Bit 5 (APCx) of parameter No.1815 is set to 1.)

(3) A difference of parameters Nos.1821 and 1882 must be more than 4.

Example)

When the scale, which is that mark1 interval is 20.000mm and mark2 interval is 20.004mm, is used on IS-B machine :

When the detection unit of 0.001mm is selected, parameter No.1821 and No.1882 must be set "20000" and "20004", and the difference of them is "4".

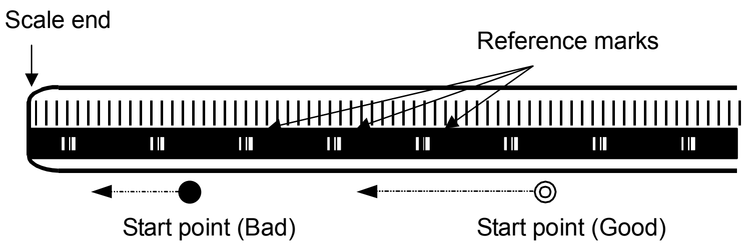
To use such a scale, please adjust the detection unit by modification of parameters No.1820 (CMR) and Nos.2084/2085(flexible feed gear) to make the difference of parameters Nos.1821 and 1882 more than 4 as following examples.

(a) Set the detection unit=0.0001mm, and set No.1821=200000, No.1882=200040

(b) Set the detection unit=0.0005mm, and set No.1821=40000, No.1882=40008

> **NOTE**
> When the detection unit is changed, parameters relating to the detection unit (such as the effective area and positional deviation limit) must also be changed accordingly.

(4) In this procedure, the axis does not stop until two, three or four reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three or four reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.



(5) When the axis used this function, the following function can not be used.
- Absolute position detection (bit 5 (APCx) of parameter No.1815 = 1)

(6) If axial movement is made in the direction opposite to that of reference position return, the movement is reversed to the direction of reference position return after three or four reference marks have been detected. Steps (3) to (5) of the basic procedure for establishing a reference position are carried out to establish the reference position.

M

(7) Simple straightness compensation
When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by simple straightness compensation amount when the reference point of moving axis is established.

T

(8) The reference point establishment is not performed during synchronous control is activated.
(9) The reference point establishment is not performed during composite control is activated.
(10) The reference point establishment is not performed during superimposed control is activated.

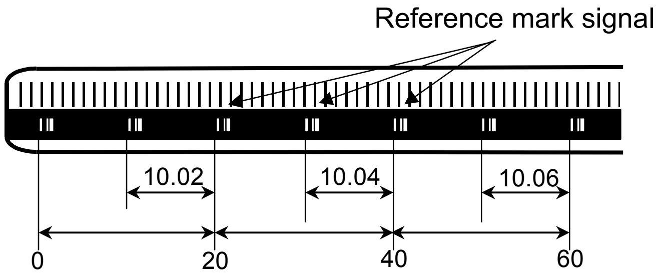
3.7 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

Overview

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.

Explanation

The linear scale with distance-coded reference marks (serial) is combined the irregular reference marked linear scale with the High-resolution serial output circuit, it can detect the accurate position.

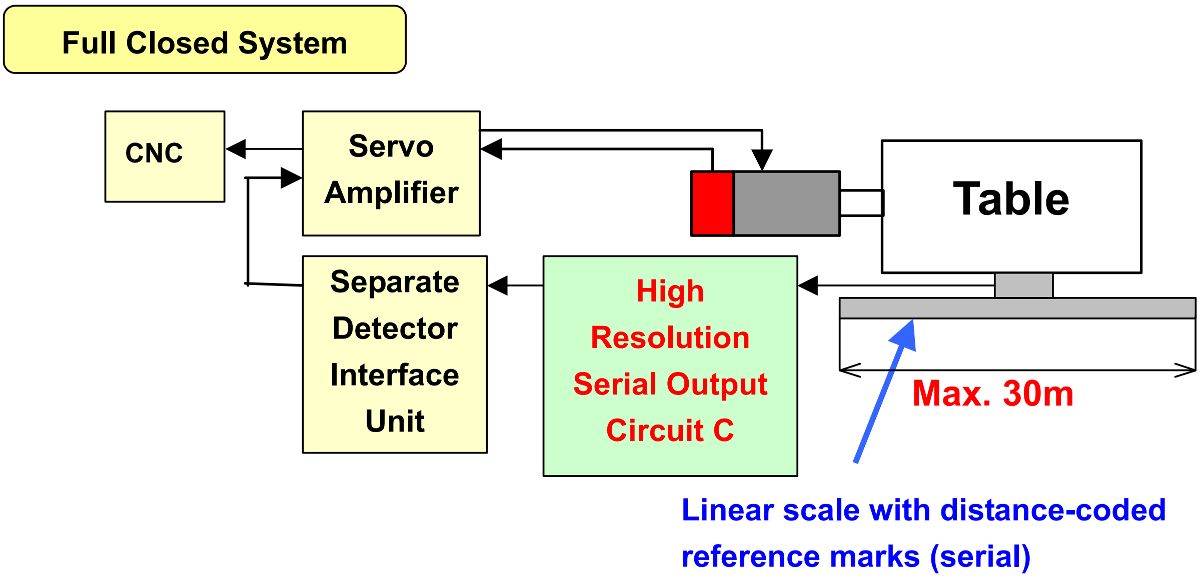


The CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position, because of the interval of each reference mark is different with regular interval.
It is not necessary that the axis is moved to the reference position for establishment of reference 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.

- Connection

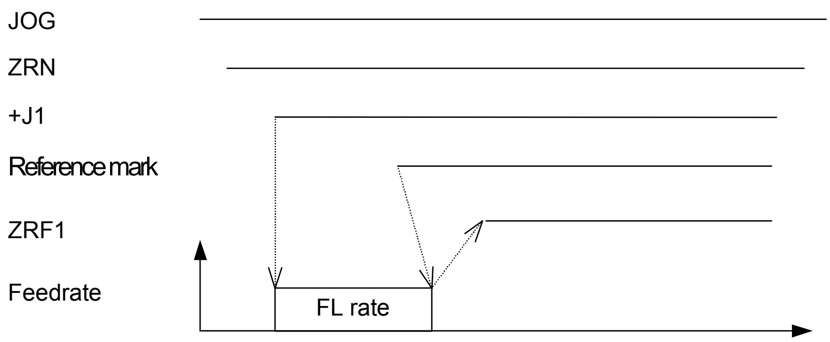
It is available under full closed system.



- Procedure for reference position establishment through manual operation

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter No.1425 setting).
(4) When the absolute position of linear scale with distance-coded reference marks (serial) is detected, the axis stops. Then the absolute position of CNC is calculated and reference position establishment signal (ZRF1,ZRF2,ZRF3,…) turns to "1".

The timing chart for this procedures is given below.



- Procedure for reference position establishment through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (4) above are performed automatically.
After the reference position is established, the automatic reference position return is performed by setting of parameter RFS No.1818#0.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of the following operations is performed in steps (3) to (4), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0
- Setting the Servo off signals (SVF1, SVF2, etc.) to 1

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position
- Setting the Servo off signals (SVF1, SVF2, etc.) to 1

If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

- Establishing a reference position and moving to the reference position

By following operation, establishing a reference position and moving to the reference position is performed.

	Moving through manual operation in REF mode	Moving through automatic operation by automatic reference position return (G28)
The reference position is not established.	Establishing the reference position	Firstly, moving to the intermediate position, and establishing the reference position. Secondly, whether moving to the reference position or not is performed by setting bit 0 (RFS) of parameter No.1818.
The reference position is established.	Moving to the reference position	Whether moving to the intermediate position and the reference position or not is performed by setting bit 1 (RF2) of parameter No.1818.

- Feed axis synchronization control

In case of using the axis synchronization control, please confirm the following items.

- When this function is used with axis synchronization control axes, the linear scale with distance-coded reference marks (serial) used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
- The master axis scale and the slave axis scale should be installed in parallel direction. (The zero positions should be faced the same direction.)
- To the parameters, which relate to this function (except No.1883, No.1884), the same value must be set for the master axis and for the slave axis.
- The linear scale with distance-coded reference marks (serial) should be applied for the master axis and the slave axis.
  If either of the master axis or the slave axis is not the linear scale with distance-coded reference marks (serial), alarm DS0018 occurs when reference position establishment is tried.
- During operating the establishment of reference position, the state of signal for selecting synchronized axis(SYNCn<Gn138> or SYNCJn<Gn140>) should be kept.

Procedure for reference position establishment by axis synchronization control is as follows.

- Both of axes (master axis and slave axis) are fed on the reference position return FL feedrate until distance coded scales of both axes detect the absolute position.
- Then absolute position of both axes are calculated and Reference Position Establishment Signals (ZRF1,ZRF2,...) turn to "1".

- Angular axis control

In case of using the angular axis control, please confirm the following items.

- It is necessary to use the linear scale with distance-coded reference marks (serial) for both the perpendicular axis and the angular axis.
  If not, the alarm DS0019 occurs when reference position establishment is commanded.
- When the reference point establishment of angular and perpendicular axes are tried, please set bit 2 (AZR) of parameter No. 8200 to '0' and input signal NOZAGC <G063.5> to '0'.
  If not, the alarm DS0019 occurs when reference position establishment is commanded.

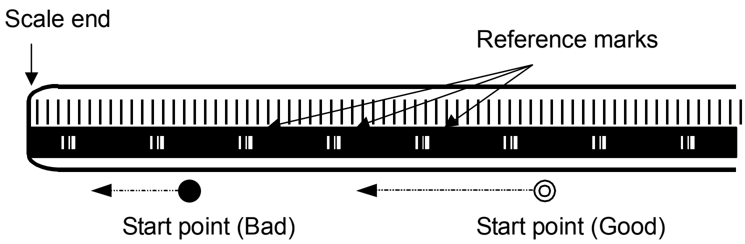
- When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020 occurs.
- On angular axis control, if you use automatic setting of parameter No.1883,1884 on reference point establishment (bit 2 (DATx) of parameter No.1819 = 1), please establish reference point of perpendicular axis after reference point establishment and return of angular axis.

In manual reference position return, the perpendicular axis cannot be specified while the angular axis reference point is being established. The perpendicular axis, if specified, is ignored.

⚠ **CAUTION**

1 When the Linear scale with distance-coded reference marks (serial) is used, please set bit 3 (SDCx) of parameter No.1818 to 1.

2 On the Linear scale with distance-coded reference marks (serial), the axis does not stop until three reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.
And if establishment of reference position is failed, the establishment is retried. Then axis does not stop until still more three reference marks are detected. So please set the maximum move amount (detection unit : parameter No.14010) not to reach the scale end.



3 Simple straightness compensation (M series)
When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by simple straightness compensation amount when the reference point of moving axis is established.

4 It is not available to use this function and the temporary absolute coordinate setting together.

5 Angular axis control cannot be performed together with synchronous control (T series), composite control (T series), or superposition control (T series).

3.8 MANUAL HANDLE RETRACE

Overview

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.

- Checking mode

In this mode, the program can be executed forward and backward and the program can be checked.
To change to the checking mode, it is necessary to change the mode to the memory mode (MEM mode), and the checking mode signal MMOD<Gn067.2> is set to "1". This function makes the data to execute the program backward when the program is executed forward in the checking mode.
To work a machine synchronizing with a pulse generated by a manual handle in the checking mode, the manual handle check signal MCHK <Gn067.3> is set to "1" in addition to the above-mentioned. As a result, it becomes possible to check the program with a manual handle.

> **NOTE**
> During the checking mode, it is not possible to change the parameter and offset.

- Forward movement with a manual handle

The "forward movement" is that the program is executed forward by turning a manual handle in the positive direction (when the manual handle check signal is set to "1".) or in no relation to rotation of a manual handle (when the manual handle check signal is set to "0".).
When the manual handle check signal is set to "1", the execution speed of the program is proportional to the number of rotations of a manual handle. The program is executed forward rapidly when a manual handle is turned to the positive direction rapidly. And, the program is executed forward slowly when a manual handle is turned to the positive direction slowly. The distance magnification traveled per pulse from manual handle can be switched as same as a usual manual handle feed function.
When the manual handle check signal is set to "0", the execution of the program is controlled as same as an automatic operation.

- Backward movement

The "backward movement " is that the program executed forward once is executed backward by turning a manual handle in the negative direction.
The program can be executed backward only for the block executed forward. And, the number of blocks for it is about 190 blocks. This block number changes by the content of the commanded program.
The program is executed backward rapidly when a manual handle is turned to the negative direction rapidly. And the program is executed backward slowly when a manual handle is turned to the negative direction slowly. The distance traveled per pulse from manual handle can switch magnification as well as a usual manual handle feed.

Explanation

- Control by the manual handle

Program execution start

The checking mode signal MMOD<Gn067.2> is set to "1" in the memory mode (MEM mode) in order to change the checking mode. Then, the program execution is begun by turning ST signal from "1" to "0".
If the manual handle check signal MCHK <Gn067.3> is set to "1" at this time, the execution of the program is controlled by a manual handle. The program is executed synchronizing with rotation of a manual handle.
When a manual handle check signal MCHK<Gn067.3> is set to "0", it is controlled as usual execution.
When check mode signal MMOD<Gn067.2> is set to "1" during the operation of the program, it is enabled a check mode from the block that next buffering is done.
That is, even if check mode signal set to "1", check mode is not always enabled at once.
When check mode is enabled, check mode confirmation signal MMMOD<Fn091.3> is set to "1".

> **NOTE**
> After the signal MMOD is turned to "0" during the execution of the program, the program cannot be executed forward and backward.

Control with the manual handle

The value of the parameter No.6410 and the scale factors decide the moving speed of the machine by one pulse generated by a manual handle.
When a manual handle is turned, the actual movement speed of the machine is as follows.
[Feedrate command value] × [Number of the handle pulse per a second]
× [Handle magnification] × ([Parameter setting value]/100) × (8/1000) (mm/min or inch/min)
Example)

When feedrate command value is 30mm/min, handle magnification is 100, parameter No.6410 is set to 1 and manual pulse generator is rotated at 100 pulse/rev, the feedrate of axis is decided as follow.
[Feedrate]=30[mm/min] × 100[pulse/s] × 100 × (1/100) × (8/1000)[s] =24[mm/min]

When the feedrate exceeds the override 100% feedrate by turning a manual handle rapidly, the feedrate is clamped at the speed of override 100%. That is, if the pulse of the following formula exceeds "1", the feedrate is clamped.
[Number of the handle pulse per a second]
× [Handle magnification] × ([Parameter setting value]/100) × (8/1000)
The rapid traverse feedrate is clamped at 10%. However, the feedrate of the rapid traverse is clamped at 100% when bit 0 (HDRPD) of parameter No.6400 is set to "1".
And if parameter No.6405 is set to an optional value, it can be clamped to override by nearly optional value.
When the parameter No.6405 is set to larger value than "100", it is clamped to nearly 100%.
When parameter No.6405 is set to "0", the setting of bit 0 (RPO) of parameter No.6400 becomes valid.
The single block signal and the feed hold signal are effective in the checking mode. When the execution of a program is stopped by the single block stop or the feed hold stop, it is necessary to turn ST signal from "1" to "0" in order to restart the program.
In the block with the movement and the block of dwell, the execution speed of the program can be controlled by turning a manual handle. As for the block of neither movement nor dwell such as the block of only address M, S, T, and F, the program advances to the following block even if a manual handle does not turn.
The rotation of the spindle does not synchronize with a pulse of a manual handle. During the checking mode, the spindle rotates at the specified rotation speed. As for the feed per revolution, a program is executed at the feedrate which was converted from the rotation speed of the spindle to the corresponding feed per minute inside CNC.

NOTE
The manual handle used by this function is always the first. The second and third manual handles cannot be used.

Forward movement and backward movement with a manual handle

The program is executed forward when a manual handle is turned to the positive direction. And, the program is executed backward when a manual handle is turned to the negative direction.
The program is executed backward as soon as a manual handle is turned to the negative direction in executing the program forward.
When a manual handle keeps being turned in a negative direction, the program is executed backward and the execution stops in the block of O number. Then, if a manual handle is turned to the positive direction, the program is executed forward again.
Even if a manual handle controls the program execution, the program is executed forward in no relation to a pulse generated by a manual handle on setting the manual handle check signal to "0".

Program end

When the block of M2 or M30 is executed, the manual handle retrace ends. It is not possible to execute the program backward from the block of M2 or M30.
When the execution of the program ends, RESET signal must be set to "1", and the checking mode signal and the manual handle check signal must be set to "0".

In 2 path control system, FIN signal must not be set to "1" when the block of M2 or M30 is executed in only one of paths. After the block of M2 or M30 has been executed in both paths, FIN signal is set to "1". (Except for the block of waiting M code is commanded before M2 or M30 in both paths.)

- Notice of the operation

- Dry-run can not operate during the checking mode. Dry-run signal must be set to "0".
- Automatic operation starts immediately with the feedrate commanded by the program, when the checking mode signal or the synchronous operation with handle signal is turned off during executing the program in the checking mode.
- The edit of the program and the change of the parameter and the offset must not be done.

- Backward movement of each code

All modal information of G, T, S-code is memorized in executing the program forward. And, the memorized data of the modal G, T, S-code are used in executing the program backward.
As for M-codes, they are grouped and the modal information is managed by parameter No.6411 to 6490. Therefore, M-code can be executed backward according to the information. As for the modal information of the M-code, a change in each group is memorized in the execution data.
As for the codes except for G, M, S, and T, the same code is output between forward movement and backward movement.

- G-code

If G-code that changes modal information is commanded in backward movement, the modal information of previous block is executed.
Example)

```
N1 G99 ;
N2 G01 X_ F_ ;
N3 X_ Z_ ;
N4 G98 ;............ backward movement starts from this block
N5 X_ Y_ Z_ ;
```

If backward movement starts from N4 block, the modal information is changed from G98 to G99 and G99 is executed from N3.
G-code with a movement is traced along the route opposite to forward movement.
G-code that can be command in executing the program backward is as follows.
The other G-codes cannot be command in executing the program backward.
The G-codes in the G-code system B and C (for T series) also can be used.

T series (for G-code system A)

G00	G01	G02	G03	G04	G22	G23
G25	G26	G28	G30	G40	G41	G42
G50	G53	G65	G70	G71	G72	G73
G75	G80	G83	G85	G87	G89	G90
G94	G96	G97	G98	G99		

M series

G00	G01	G02	G03	G04	G22	G23
G25	G26	G28	G30	G40	G41	G42
G43	G44	G49	G53	G65	G73	G76
G80	G81	G82	G83	G85	G86	G87
G88	G89	G82	G94	G95	G96	G97

> **NOTE**
> 1 In Small-Hole Pecking Drilling Cycle(G83) (M series), backward movement is prohibited.
> 2 In forward movement of Boring Cycle(G88) (M series), the sequence of actions at bottom of hole is shown as follows (dwell -> stop of spindle motor -> hold state). But in backward movement, that is (rotation of spindle -> hold state -> dwell after restart).

- M-code

If there is M-code in the same group is commanded in previous blocks, modal information of the M-code, commanded at the last in previous blocks, is output.
If no M-code is commanded in previous blocks, the M-code set to the first parameter in the same M-code group is output.
If M-code is not set to group M-code in parameter, the same M-code is output in backward movement.
If the parameter RVN(6400#5) is set to "1", the backward movement is prohibited when the M-code, which is not set to group M-code, is commanded in backward movement.

> **NOTE**
> When setting the parameter RVN, backward movement prohibition is enabled except the M-code which was set in the grouping but backward movement can be enabled for the following M-code exceptionally.
> 1. Subprogram Call by M98/M99.
> 2. Subprogram Call using an M code
> 3. Macro Call using an M code
> 4. Waiting M code
> 5. M0

Example) Output of M-codes that are set to groups by parameters in backward movement
Setting of parameters:

No.6400#2=1, #3=0 (5 M-codes/group and 16 groups)

No.6411=100
No.6412=101
No.6413=102 Group A
No.6414=103
No.6415=104

No.6416=200
No.6417=201
No.6418=202 Group B
No.6419=203
No.6420=204

Program O10 is executed in forward movement from N1 to N15 and backward movement is executed from N15. In backward movement, the output of M-codes is shown as next table.

	Forward movement	Backward movement
O0010 ;		
N1 G4 X1. ;		
N2 M101 ;	M101	M100 (*1)
N3 G4 X1. ;		
N4 M204 ;	M204	M200 (*1)
N5 G4 X1. ;		
N6 M104 ;	M104	M101 (*2)
N7 G4 X1. ;		
N8 M300 ;	M300	M300 (*3)
N9 G4 X1. ;		
N10 M200 ;	M200	M204 (*2)
N11 G4 X1. ;		
N12 M0 ;	M0	M0 (*3)
N13 G4 X1. ;		
N14 M102 ;	M102	M104 (*2)
N15 G4 X1. ;		Backward movement starts from this block
M2 ;		

*1 No M-code in the same group is commanded before this block, so the M-code, which is set in the 1st. parameter of the same group, is output.
*2 M-code in the same group is commanded before this block, so the M-code, which is commanded at the last before this block, is output.
*3 M-code is not set to group M-code, so the same M-code is output.

- S and T-code

A modal value of the previous block is output.
When movement command and S-code or T-code is commanded in the same block, the timing of the output of the S-code and T-code is different. Because, the timing where S-code and T-code are output at the forward movement is different from that at the backward movement. By setting bit 7 (STO) of parameter No.6401 to "1", the timing of the output of S and T code at the forward movement is the same as the one at the backward movement.

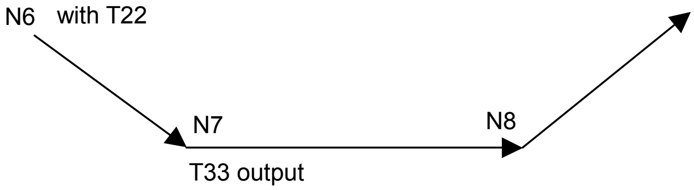
Example)
T-code output timing at the backward movement
T-code is output as follows when the program proceeds backward after the forward movement to N8 block.

	Forward movement	Backward movement	
		Parameter STO=0	Parameter STO=1
O1000 ;			
N1 G98 G00 X0 Z0 ;			Default T output
N2 G00 X-10. T11 ;	T11 output	Default T output	
N3 G00 X100. ;			T11 output
N4 G00 X10. Z20. T22 ;	T22 output	T11 output	No T code output
N5 G00 X30. Z30. ;			
N6 G00 X-10. Z-20. ;			T22 output
N7 G00 X50. Z40. T33 ;	T33 output	T22 output	T33 output
N8 G04 X5. ;		(Backward start)	(Backward start)
M30 ;			

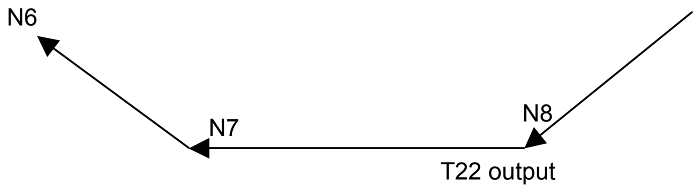
The “Default T” means a T-code status at N1 block in forward movement. If the status is T0, “T0” signal is output as “Default T” in the backward movement.

The timing of T-code output of N7 and N8 in O1000 shown in the example above is as follows.

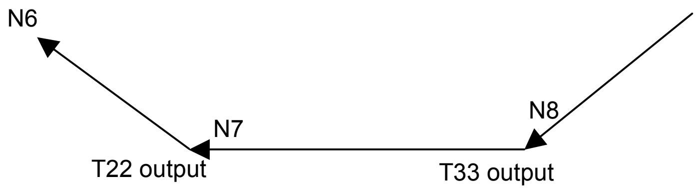
Forward movement :



Backward movement (When parameter STO is set to “0”) :



Backward movement (When parameter STO is set to “1”) :



- Direction change prohibition

The direction change prohibition is a state not changing the direction where the program is executed.
In the state, even if the rotating direction of a manual handle is reversed, the reversed rotation is ignored.
A manual handle must be rotate in the same direction as present direction for removing this state.
The direction change prohibition can be confirmed by output signal MNCHG<Fn091.1>.

It becomes the change prohibition state under the following condition.

- During axis movement
- While the block with the code waiting for FIN is executing
- After a block has done and until the next block begins to operate
- During thread cutting
- Modal G code of G68 (M series) and G51.2 (T series)
- The block with the axis that ends movement earlier in the block with G02 or nonlinear type position (G00) etc.
- During waiting at a block boundary (Only for a 2-path system. See "Waiting in 2-path system".)

- Backward movement prohibition

The backward movement prohibition is a state that the program cannot be executed from a certain block backward. In this state, the negative rotation of a manual handle is ignored, and the only positive rotation is effective. The program must be executed forward by rotating a manual handle in the positive direction for removing this state.
The backward movement prohibition can be confirmed by output signal MRVSP<Fn091.2>.
If the following blocks are executed in backward movement, backward movement is prohibited.

- Program number block of main program (except subprogram and macro program)
- Over the maximum number of the blocks for reverse movement

- The block including backward movement prohibition G-code (which is not described in the paragraph "G-code")
- The block which is executed while in modal including backward movement prohibition G-code (which is not described in the paragraph "G-code")

- Status display

In manual handle retrace, the status of manual handle retrace is displayed on clock display of CNC state display line. This status display is displayed during the execution of manual handle retrace. The clock is displayed usually.

When the all condition is filled, "M.H.RTR." is displayed on clock display of CNC state display line. This status is displayed by the color of color setting 3 (INPUT KEY, O/N NO. and STATUS are the same color). The screen display is as shown in Fig.3.8(a). When the following conditions are not full, the clock is displayed.

- When bit 2 (CHS) of parameter No.6401 is set to "0":
  1) Software option of handle manual retrace is enabled.
  2) Status display disable/enable bit 6 (HST) of parameter No.6401 is set to "1".
  3) Check mode confirmation signal MMMOD<Fn091.3> is set to "1".

- When bit 2 (CHS) of parameter No.6401 is set to "1":
  1) Software option of handle manual retrace is enabled.
  2) Status display disable/enable bit 6 (HST) of parameter No.6401 is set to "1".
  3) Cycle start signal STL<Fn000.5> is set to "1".
  4) Check mode signal MMOD<Gn067.2> is set to "1".
  5) Manual handle check signal MCHK<Gn067.3> is set to "1".

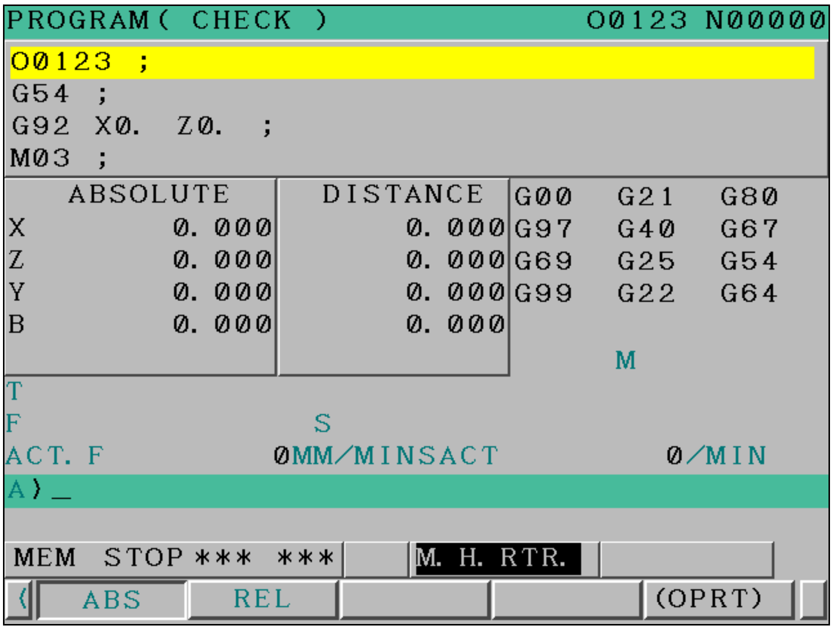


Fig. 3.8 (a) "M.H.RTR." status display

Besides, when reverse movement prohibition signal MRVSP<Fn091.2> is set to "1", the "NO RVRS." is displayed. This status is displayed by blinking/reversing in the color of color number 1 (ALARM is the same color). The screen display is as shown in Fig.3.8(b). When reverse movement prohibition signal MRVSP<Fn091.2> is set to "0", the " M.H.RTR." is displayed again.

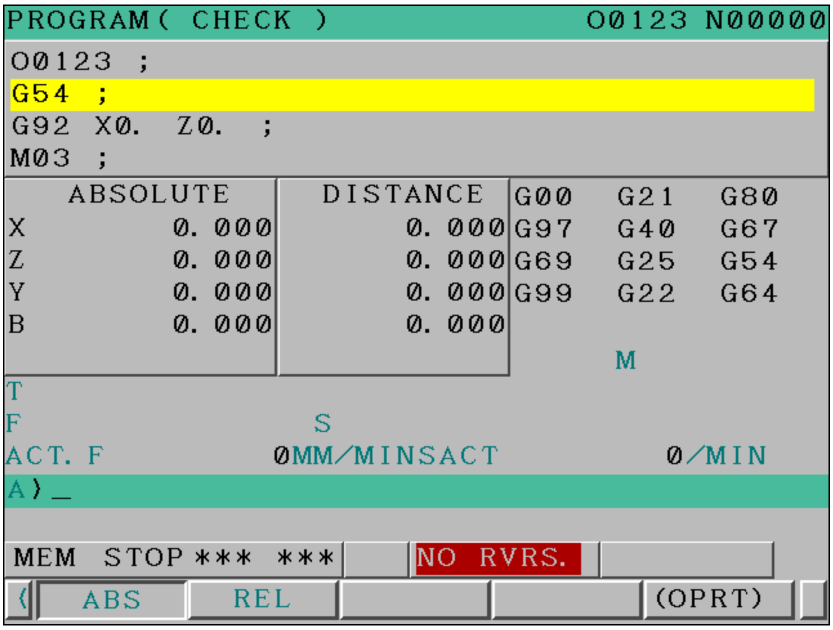


Fig. 3.12 (b) " NO RVRS." status display

Besides, when direction change prohibition signal MNCHG<F0091.1> is set to "1" and the direction of program’s execution is changed by manual handle, this status display changes from "M.H.RTR." to "NO.CHAG.".

This status is displayed by blinking/reversing in the color of color number 3 (INPUT KEY, O/N NO. and STATUS are the same color) . The screen display is shown as Fig.3.12(c). When the program is executed in the direction as the same as before by manual handle or direction change prohibition signal MNCHG<Fn091.1> is set to "0", the " M.H.RTR." is displayed again.

Moreover, when parameter FWD(No.6400#1) is set to "1" and the program is executed to change direction by manual handle, this status display changes from "M.H.RTR." to "NO.CHAG.".

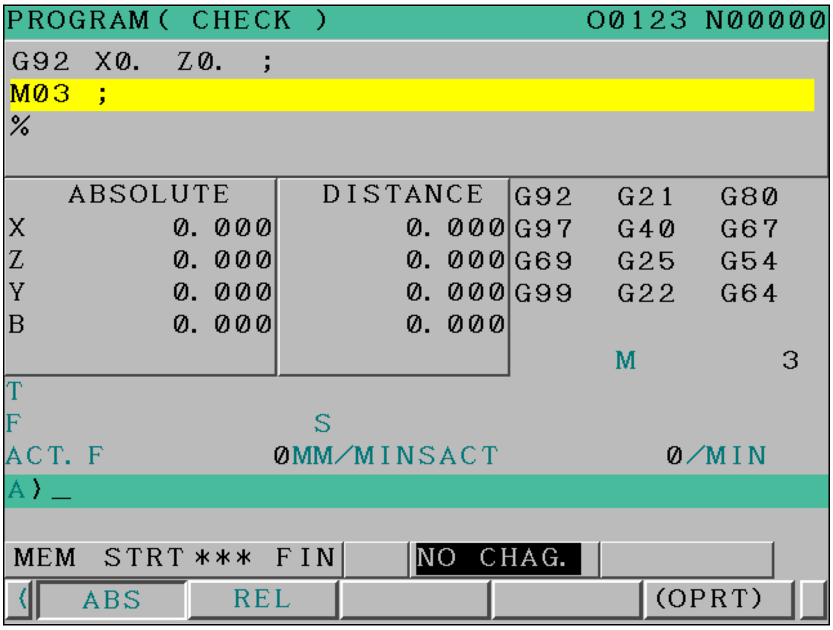


Fig. 3.12 (c) "NO.CHAG." status display

Limitation

- Movement in automatic operation by DNC operation mode(RMT)

In the automatic operation by DNC operation mode(RMT), the backward movement is prohibited though the forward movement is enable.

- Movement in subprogram operation by external subprogram call

In M198 or M-code for subprogram operation by external subprogram call (parameter No.6030), the backward movement is prohibited though the forward movement is enable.

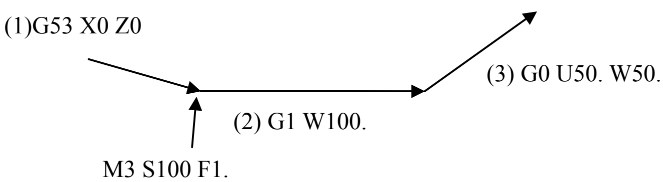
- Movement command and M,S,T-code

When M,S,T-codes and movement commands are in the same block, the timing outputting codes changes between in forward movement and backward movement. Therefore, M, S, T-codes should be commanded in backward movement after confirming that "DEN" signal is set to "1".

Example of executing the following programs with T series

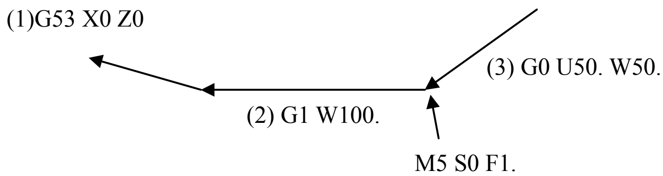
```
O0001 ;
M5 S0 F0 ;
G53 X0 Z0 ; ..........................(1)
G1 W100 M3 S100 F1. ; ........(2)
G0 U50. W50. ; ......................(3)
M2 ;
```

[Forward movement]



The block of (2) moves with M3 S100 F1.

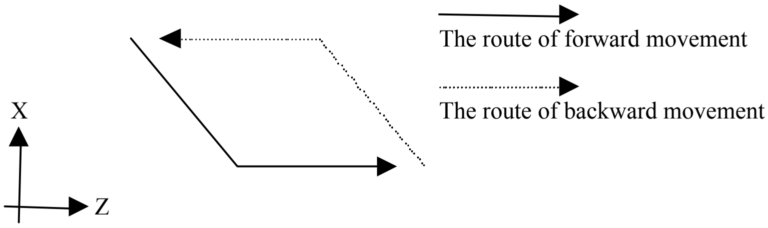
[Backward movement]



The block of (2) moves with M5 S0 F1.

- Non linear interpolation type positioning

In the non-linear interpolation type positioning, the route is different between forward movement and backward movement.



Please use the interpolation type positioning to ward off danger. (Set the bit 1 (LRP) of parameter No.1401 to "1") When non linear interpolation type positioning is used, a direction change is prohibited when any axis stops moving.

- Threading in forward movement

Threading is always executed at 100% override speed. That is to say, a pulse generated by a manual handle is ignored in executing a threading block. In thread cutting cycle, the pulse is invalid at the time actually cutting thread, but the one are valid in the other movements.

- Macro

In macro statement, the setting, operation, and so on of the macro variable is executed in only first forward movement. That is to say, the setting, operation, and so on of the macro variable is never executed in the block executing them once.

- Axis control by PMC

The movement of axis control by PMC cannot be controlled by this function.

T

- Two path simultaneous check in the 2-path system

When using the manual handle retrace function at the same time in two paths, the timing of block operation may slightly differ between these paths due to the repetition of forward and backward motion or differences in the rotation speed of the manual handle. To synchronize block operation between the paths, use the wait M code.

- Waiting in 2-path system

In the 2-path system, the total of handle pulses input between the beginning and the end of each block is recorded during forward movement. During backward movement, control is made to prevent the processing from proceeding to the next block until as many handle pulses as were input during forward movement are input. Since the handle pulses input during an in-position check are also recorded, when the handle rotation speed (axis feedrate) changes between forward movement and backward movement, the time required for an in-position check varies, causing a difference between the total of handle pulses recorded during forward movement and that recorded during backward movement. In this case, even if the block is completed (the remaining movement amount is indicated to be 0) during backward movement, processing may not proceed to the next block until the handle is turned by the amount equivalent to the number of the handle pulses recorded during forward movement. A direction change is prohibited in this case, so switching to forward movement is not allowed until processing proceeds to the next block in forward movement.

- Check of path unit by 2-path system

In the 2-path system, the program check of an arbitrary path is possible. In the path not to check, please select the mode excluding the MEM mode. Even if bit 4 (HMP) of parameter No.6400 is set to "1", it is possible to execute the check of forward movement, direction change and backward movement.
If waiting M code exist in the program, please set No-wait signal of two path NOWT <Gn063.1> to "1".

- Multi Spindle

During the backward movement, both TYPE-A and TYPE-B multi spindle control may not be operated exactly.

- Modal display

In the backward movement with manual handle, the modal display is updated according to the operation condition of the program.

- Modal information

In the backward movement with manual handle, the state of modal information is updated according to the operation condition of the program.

- Change in operation mode

When you change to EDIT mode during the checking mode, the backward movement and the re-forward movement cannot be executed in the blocks which have been already executed.

- ON/OFF of Manual Handle Retrace mode

When check mode signal MMOD<Gn067.2> is set to "0" and handle available signal in checking mode MCHK<Gn067.3> is set to "0", the check mode might not be turned off at once.
Basically, in the middle of block, the check mode doesn't switch from ON to OFF or from OFF to ON. After the block is ended, the check mode switches from ON to OFF or from OFF to ON.

- Advanced preview control (T series) / AI advanced preview control (M series) / AI contour control (M series)

When check mode signal MMOD<Gn067.2> is 1, advance preview control (G08 P1), AI advanced preview control (G05.1 Q1), and AI contour control (G05.1 Q1) are disabled. Forward movement or backward movement is performed with advance preview control (G08 P1), AI advanced preview control (G05.1 Q1), and AI contour control (G05.1 Q1) disabled.
If check mode signal MMOD<Gn067.2> is set to 1 in the advance preview control (G08 P1) mode, AI advanced preview control (G05.1 Q1) mode, or AI contour control (G05.1 Q1) mode, then the check mode is disabled until advance preview control is disabled (G08 P0) or AI advanced preview control/AI contour control is disabled (G05.1 Q0).

- Execution of measurement G-code with the speed of override 100%

When bit 6 (MGO) of parameter No.6400 is set to "1", a handle pulse is invalid and it is always executed at 100% override. When bit 6 (MGO) of parameter No.6400 is set to "0", this function is invalid and a handle pulse is valid.
In the 2-path system, this function is not effective in the execution of another path and handle pulse is valid in another path. The measurement G code to which this function is effective is as follows.

1) G31 for skip
2) G31, G31 P1, G31 P2, G31 P3, G31 P4, G04, G04 Q1, G04 Q2, G04 Q3 and G04 Q4 for multistage skip
3) G31 P99 and G31 P98 for torque limit skip

T

During measurement of G36 and G378 for automatic tool compensation, handle pulses are disabled and execution at a feedrate of override 100% is assumed, regardless of the setting of bit 6 (MGO) of parameter No. 6400. During rapid traverse before measurement, handle pulses are enabled.

When bit 7 (SKF) of parameter No.6200 is set to "0" and bit 2 (SFN) of parameter No.6207 is set to "0", handle pulse at G31 is invalid and it is always executed at 100% override regardless of setting bit 6 (MGO) of parameter No.6400.

When bit 7 (MG4) of parameter No.6400 is set to "1" and the software option of multistage skip is enabled and the setting of parameter from No.6202 to No.6206 is enabled, the backward movement prohibition is enabled in G04 block for multistage skip.
The G code to which this function is effective is as follows.

1) G04, G04 Q1, G04 Q2, G04 Q3, and G04 Q4 for multistage skip

M

- Relation to another function

This function cannot coexist with the following functions

- Retrace

4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.
This chapter explains the following types of automatic operation:

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.
When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.
When the 

key on the MDI panel is pressed, automatic operation terminates and the reset state is entered.

T

For the 2-path control, the programs for the two paths can be executed simultaneously so the two paths can operate independently at the same time.

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

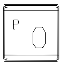
Memory operation

Procedure

T

1 For the 2-path control, select the path to be operated with the path selection switch on the machine operator's panel.

2 Press the MEMORY mode selection switch.
3 Select a program from the registered programs. To do this, follow the steps below.
2-1 Press 

key to display the program screen.
2-2 Press address 

key.
2-3 Enter a program number using the numeric keys.
2-4 Press the [O SRH] soft key.
4 Press the cycle start switch on the machine operator's panel.
Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.
5 To stop or cancel memory operation midway through, follow the steps below.
a. Stopping memory operation
Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
(i) When the machine was moving, feed operation decelerates and stops.
(ii) When dwell was being performed, dwell is stopped.
(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.
b. Terminating memory operation
Press the 

key on the MDI panel.
Automatic operation is terminated and the reset state is entered.
When a reset is applied during movement, movement decelerates then stops.

Explanation

- Memory operation

After memory operation is started, the following are executed:
(1) A one-block command is read from the specified program.
(2) The block command is decoded.
(3) The command execution is started.
(4) The command in the next block is read.
(5) Buffering is executed. That is, the command is decoded to allow immediate execution.
(6) Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.
(7) Hereafter, memory operation can be executed by repeating the steps (4) to (6).

- Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

- The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
- There are two keys to stop memory operation: The feed hold key and reset key.

- Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Optional stop (M01)

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

- Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

- Reset

Automatic operation can be stopped and the system can be made to the reset state by using RESET key on the MDI panel or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

- Optional block skip

When the optional block skip switch on the machine operator's panel is turned on, blocks containing a slash (/) are ignored.

T

- Cycle start for the 2-path control

For the 2-path control, the cycle start switch is provided for each path. Accordingly, it is possible to operate a single path by starting the path or to operate both paths at the same time by starting the paths during memory operation or MDI operation. Generally, select the path to be operated with the path selection switch on the machine operator's panel and press the cycle start button to start the path. (The operation method may vary depending on the machine tool builder, so refer to the manual provided by the machine tool builder.)

4.2 MDI OPERATION

In the MDI mode, a program consisting of up to 511 characters can be created in the same format as normal programs and executed from the MDI panel.
MDI operation is used for simple test operations.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

MDI Operation

Procedure

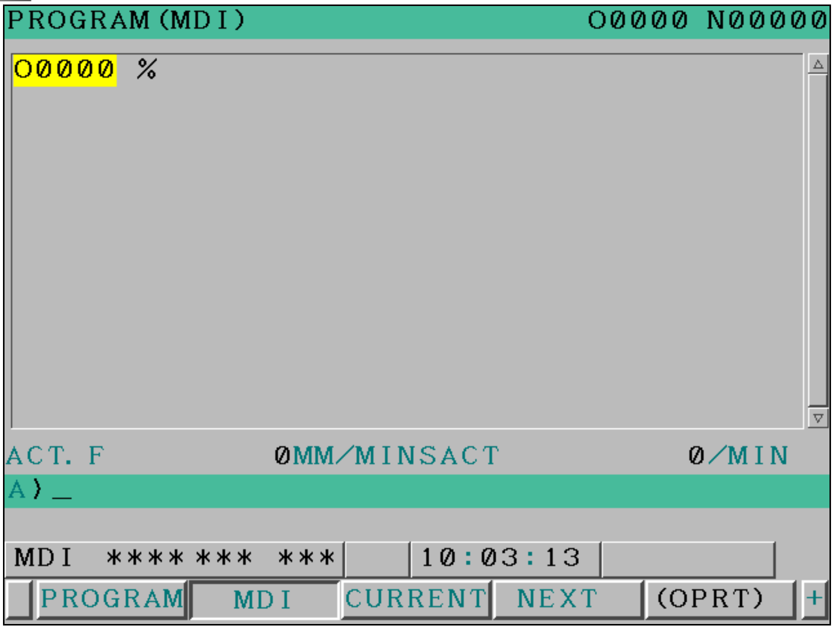
1 Select the MDI mode.

T

For the 2-path control, select the path for which a program is created and select MDI mode. A program is created for each path.

2 Press the 

key to select the program screen. The following screen appears:

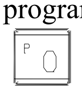


MDI program screen

At this time, program number “O0000” is inserted automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode.

4 To entirely erase a program created in MDI mode, use one of the following methods:

a. Enter address 

key, then press the 

key.

b. Alternatively, press the 

key. In this case, set parameter MCL (No. 3203#7) to 1 in advance.

5 To execute a program, set the cursor on the head of the program.
Push Cycle Start button on the operator's panel. By this action, the prepared program will start.
When the program end (M02, M30) or EOR(%) is executed, the prepared program will be automatically erased and the operation will end.
By command of M99, control returns to the head of the prepared program.

6 To stop or terminate MDI operation in midway through, follow the steps below.

a. Stopping MDI operation
Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
(i) When the machine was moving, feed operation decelerates and stops.
(ii) When dwell was being performed, dwell is stopped.
(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b. Terminating MDI operation

Press the RESET key.

Automatic operation is terminated and the reset state is entered.
When a reset is applied during movement, movement decelerates then stops.

Explanation

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program (M99 performs this function).

- Erasing the program

Programs prepared in the MDI mode will be erased in the following cases:

- In MDI operation, if M02, M30 or EOR(%) is executed.
- When bit 6 (MER) of parameter No. 3203 is set to 1, and the last block of the program is executed in single block operation

> **NOTE**
> In the two cases above, program erasure can be prevented by setting bit 6 (MKP) of parameter No. 3204 to 1.

- In MEM mode, if memory operation is performed.
- In EDIT mode, if any editing is performed.
- When the P 0 and DELETE keys are pressed.
- Upon reset when bit 7 (MCL) of parameter No.3203 is set to 1

> **NOTE**
> Upon reset when the parameter MCL = 0, the cursor moves to the end of the program.

- Restart

If a program is not executed even once after the program is input, the program is executed from the beginning, regardless of where the cursor is placed. However, a program is executed starting at the beginning of the block where the cursor is place, if the program is stopped for a reason such as single block operation after restart of an MDI operation then is restarted after an editing operation.

> ⚠ **CAUTION**
> When an MDI program is restarted, the program is executed starting at the beginning of the block where the cursor is placed, regardless of the cursor position in the block.
> (Example)
> When the cursor is placed on G90
> :
> G91 X100.0 G90 Y200.0 Z300.0 ;
> :
> The program is executed starting at the beginning (namely, G91) of this block. So, the tool moves by 100.0 along the X-axis in the incremental programming, and moves to 200.0 and 300.0 along the Y-axis and Z-axis, respectively, in the absolute programming.

- Editing a program during MDI operation

A program can be edited during MDI operation. By setting bit 5 (MIE) of parameter No. 3203 to 1, editing can be disabled. However, even when bit 5 (MIE) of parameter No. 3203 is set to 1, editing can be enabled by resetting the operation.

- Absolute/incremental command

When bit 4 (MAB) of parameter No. 3401 is set to 1, the absolute/incremental programming of MDI operation does not depend on G90/G91. In this case, the incremental programming is set when bit 5 (ABS) of parameter No. 3401 is set to 0, and the absolute programming is set when bit 5 (ABS) of parameter No. 3401 is set to 1.

Parameter MAB (No.3401#4)=0	Parameter MAB (No.3401#4)=1	
Absolute mode operation with G90 command, and incremental programming operation with G91 command	Parameter ABS (No.3401#5)=0	Parameter ABS (No.3401#5)=1
	Incremental mode operation at all times, independent of G90/G91 command	Absolute mode operation at all times, independent of G90/G91 command

NOTE

When G code system A is used on a T series, the parameters MAB and ABS are invalid.

Limitation

- Program registration

Programs created in MDI mode cannot be registered.

- Number of characters in a program

A created program can consist of up to 511 characters including "O0000" automatically inserted.

- Subprogram nesting

The subprogram call command (M98) can be described in a program created in MDI mode. That is, programs that are registered in memory through MDI operation can be called and executed. The level of subprogram call nesting is the same as in MEM operation.

- Macro call

When the custom macro function is enabled (bit 5 (NMC) of parameter No. 8135 is 0), a macro program can be created and executed even in the MDI mode. Moreover, a macro program can be called for execution.

NOTE

The GOTO statement, WHILE statement, and DO statement cannot be executed in a program created in the MDI mode. An alarm PS0377 is issued.
When a program including those statements is to be executed, register the program in the program memory then call the program for execution.

4.3 DNC OPERATION

By activating automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read in via reader/puncher interface.
To use the DNC operation function, it is necessary to set the parameters related to the reader/punch interface in advance.
The procedure described below is just an example. For actual operation, refer to the relevant manual of the machine tool builder.

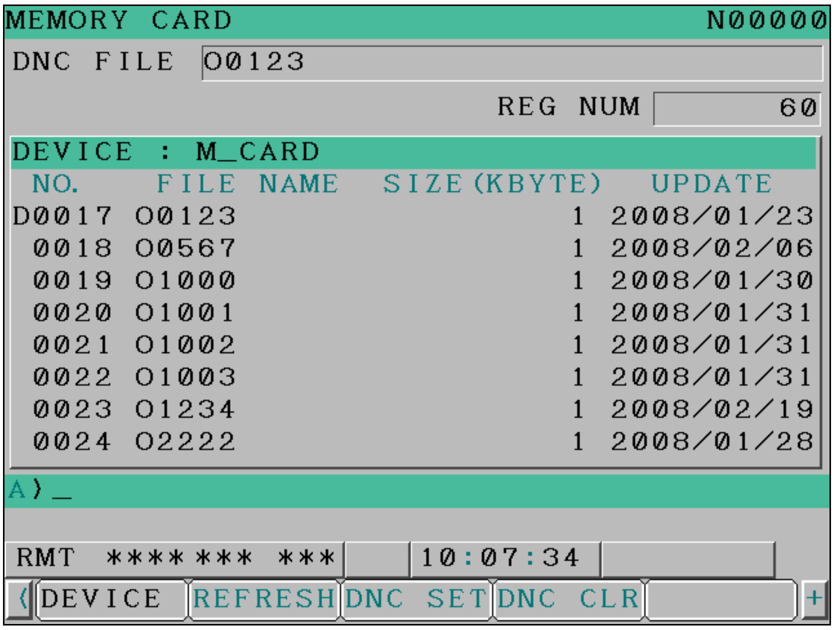
DNC operation

Procedure

1 Press the REMOTE switch on the machine operator's panel to enter the RMT mode.

2 Select the program to be executed.

- Selecting a DNC operation file
  Enter the number of the file to be subjected to DNC operation is performed on the memory card (or floppy cassette) list screen with the keyboard and press soft key [DNC SET] (or [DNC SET] for the 10.4-inch display unit) to select the file to be subjected to DNC operation. (The selected file is marked with "D".)
- Releasing a DNC operation file
  Press soft key [DNC CLR] (or [DNC CLR] for the 10.4-inch display unit) on the memory card (or floppy cassette) list screen to release the DNC operation file. (The "D" mark for the file is removed.)



3 Press the cycle start switch to execute the selected file. For details on the REMOTE switch, refer to the manual provided by the machine tool builder.

4 During DNC operation, executed programs are listed on the program check screen and program screen.

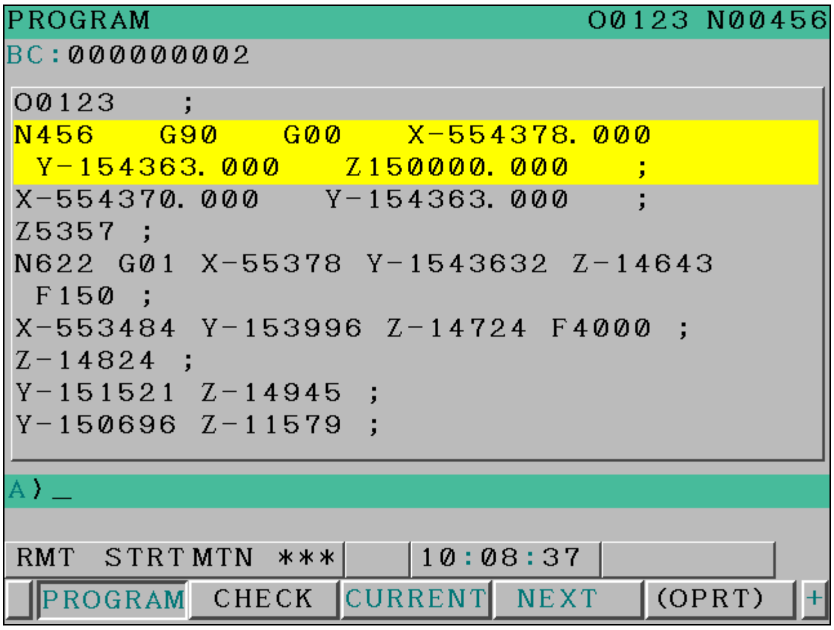


Fig. 4.3 (a) PROGRAM screen

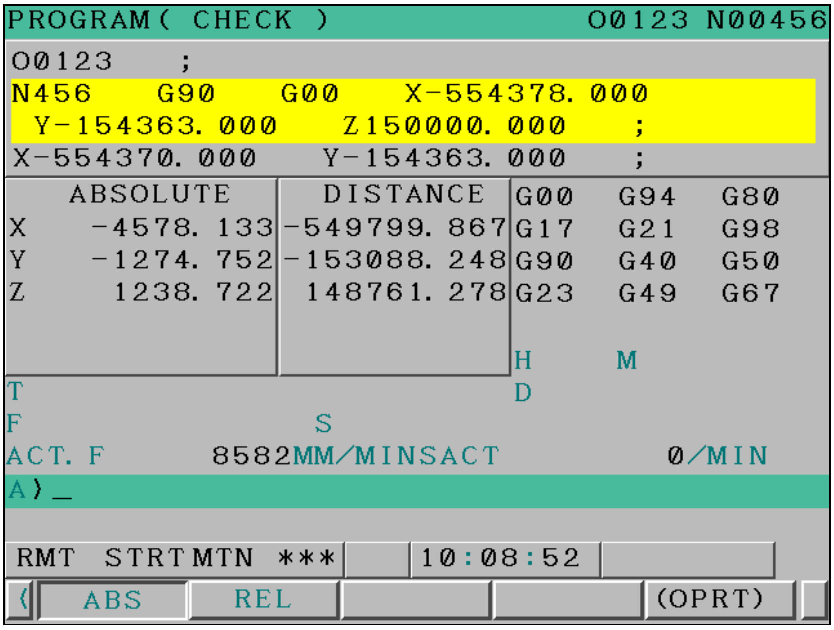


Fig. 4.3 (b) PROGRAM CHECK screen

> **NOTE**
> 1 Before selecting a DNC operation file, be sure to release all schedule data. DNC operation and schedule operation cannot be specified at the same time.
> 2 A DNC operation file cannot be released during DNC operation.
> 3 To switch between devices when DNC settings are made, release the settings and then make the settings again.

Explanation

During DNC operation, subprograms and macro programs stored in memory can be called.

Limitation

- M198 (command for calling a program from within an external input/output unit)

In DNC operation, M198 cannot be executed. If M198 is executed, alarm PS0210 is issued.

- Custom macro

In DNC operation, custom macros can be specified, but no repeat instruction and branch instruction can be programmed. If a repeat instruction or branch instruction is executed, alarm PS0123 is issued.

- M99

For returning from a subprogram or macro program to the calling program during DNC operation, the specification of a return command (M99P...) with a sequence number specified is not allowed.

T

- 2-path concurrent operation

DNC operation cannot be performed concurrently on two paths.
DNC operation can be performed on one path at a time.

4.4 SCHEDULE OPERATION

To perform schedule operation, select files (programs) registered in a memory card and specify the sequence of execution and the repetition count of each program.

Schedule operation

Procedure

1 Press the REMOTE switch on the machine operator's panel to enter the RMT mode.

2 Select the program to be subjected to schedule operation.

- Selecting a schedule
  Select the file to be subjected to schedule operation. After selecting the file, press soft key [SCHDL] to display the schedule list screen.

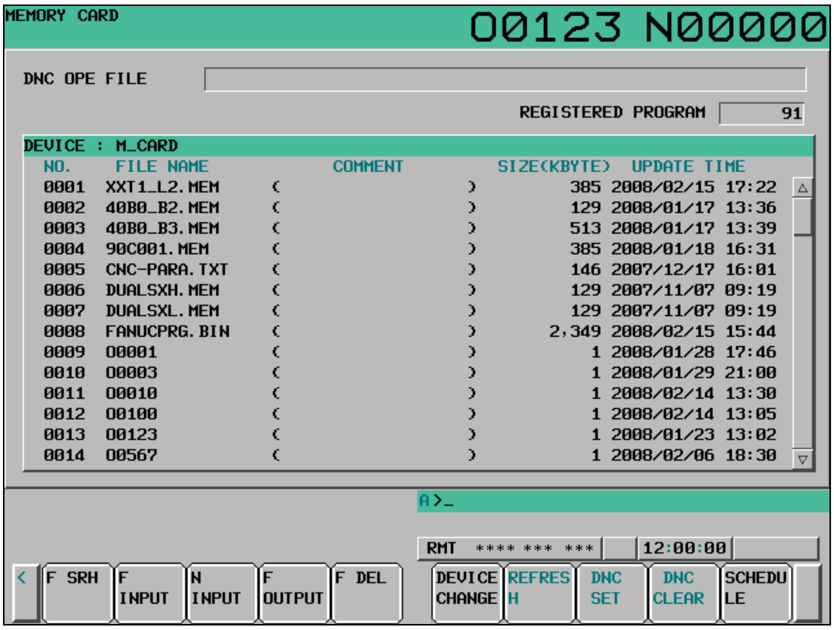
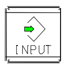


Fig. 4.4 (a) Program list screen (10.4-inch)

[SCHDL]
Lists the settings of schedule data to edit the repetition count or the like (see the next page for details).

- Setting and editing a schedule (for the 10.4-inch display unit)
  Move the cursor to the FILE NUMBER field or FILE NAME field of the desired number, enter the file number or file name with the keyboard, and press edit key  to schedule the file. Schedule operation is performed in ascending order of the number. When a schedule is set in this procedure, the file execution repeat count is set to 1. The repeat count and the order of schedule operation can be edited on this screen.

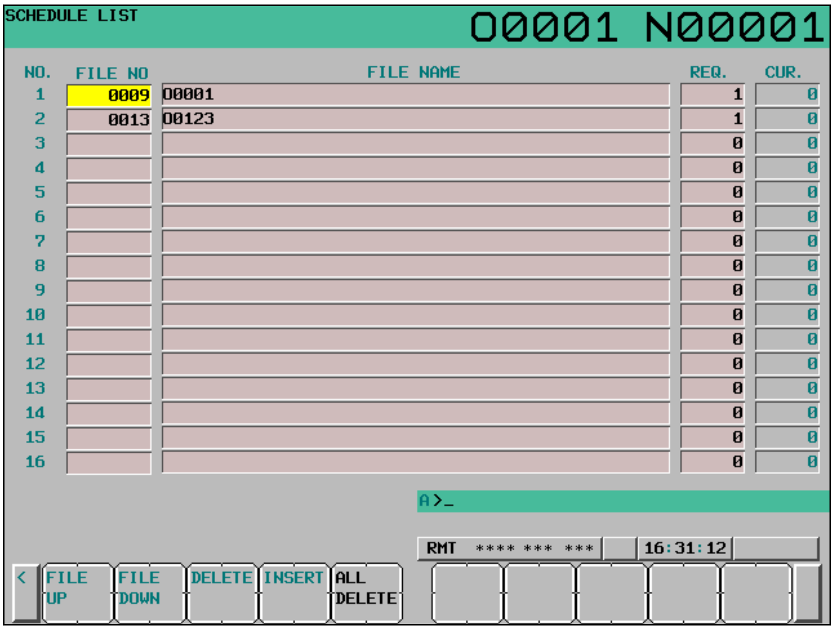
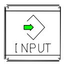


Fig. 4.4 (b) Schedule list screen (10.4-inch)

[FILE UP]
Moves the file at the cursor position up one line and moves the replaced file down one line.
[FILE DOWN]
Moves the file at the cursor position down one line and moves the replaced file up one line.
[DELETE]
Deletes the file at the cursor position and moves the files below the cursor up one line.
[INSERT]
Moves the files below the cursor down one line.
[ALL DELETE]
Deletes all records.

- Setting and editing a schedule (for the 8.4-inch display unit)
  There are two schedule list screens for the 8.4-inch display unit: the file number screen for setting file numbers and the file name screen for setting file names. On the schedule list screen, press continuous menu key  and press soft key [F-NAME] or [F-NO] to select one of the two screen. (When the file number screen is displayed, soft key [F-NAME] appears. When the file name screen is displayed, soft key [F-NO] appears.)
  Move the cursor to the FILE NUMBER field or FILE NAME field of the desired number, enter the file number or file name with the keyboard, and press edit key 

  to schedule the file. Schedule operation is performed in ascending order of the number. When a schedule is set in this procedure, the file execution repeat count is set to 1. The repeat count and the order of schedule operation can be edited on this screen.

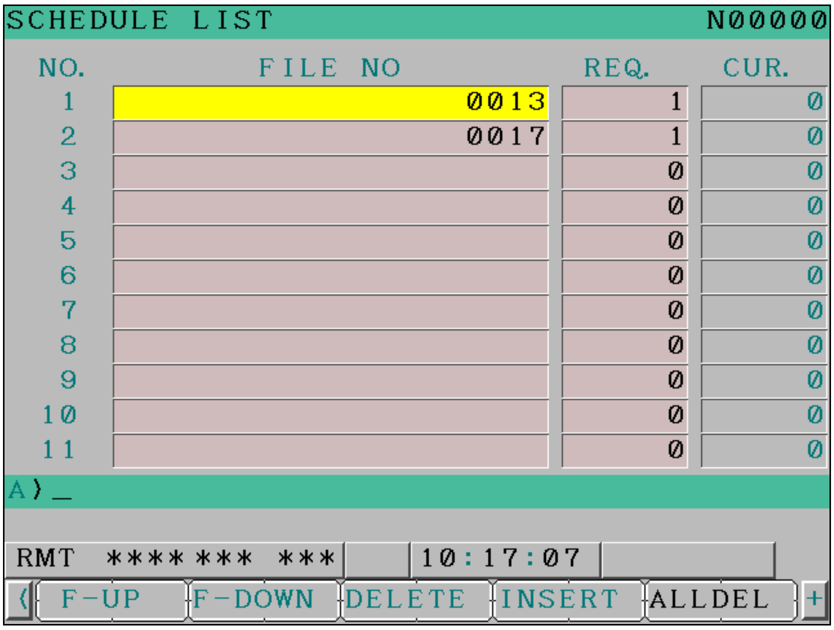


Fig. 4.4 (c) File number screen (schedule list screen)(8.4-inch)

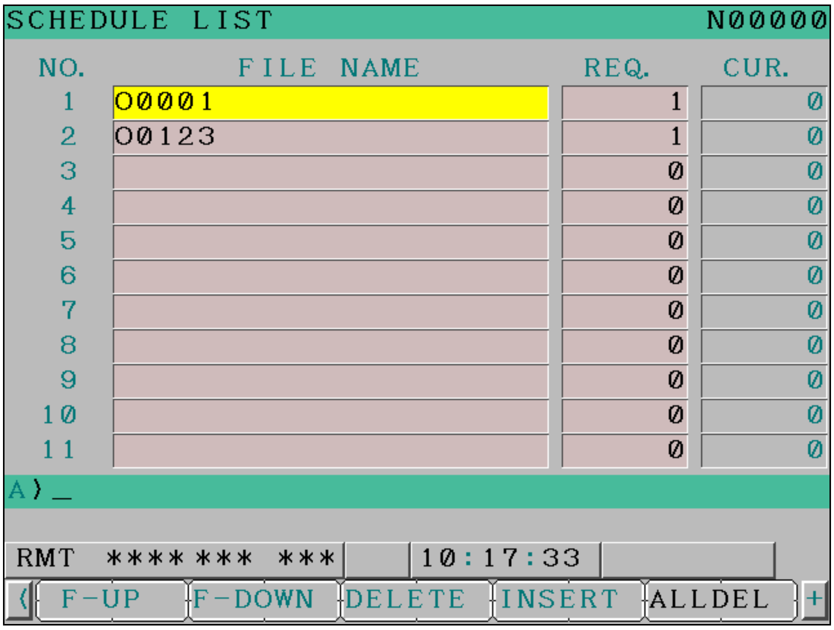


Fig. 4.4 (d) File name screen (schedule list screen) (8.4-inch display unit)

Fig. 4.4 (e) Soft key [F-NO] (8.4-inch display unit)

Fig. 4.4 (f) Soft key [F-NAME] (8.4-inch display unit)

[F-UP]	Replaces the file at the cursor position with the file located one line above.
[F-DOWN]	Replaces the file at the cursor position with the file located one line below.
[DELETE]	Deletes the file at the cursor position and moves the files at the lower positions up one line.
[INST]	Moves the files at the cursor and lower positions down one line.
[ALLDEL]	Deletes all records.
[F-NO]	Displays the file number screen.

[F-NAME] Displays file name screen.

The files registered as schedule data are marked with "S" to the left of their file names on the program list screen.

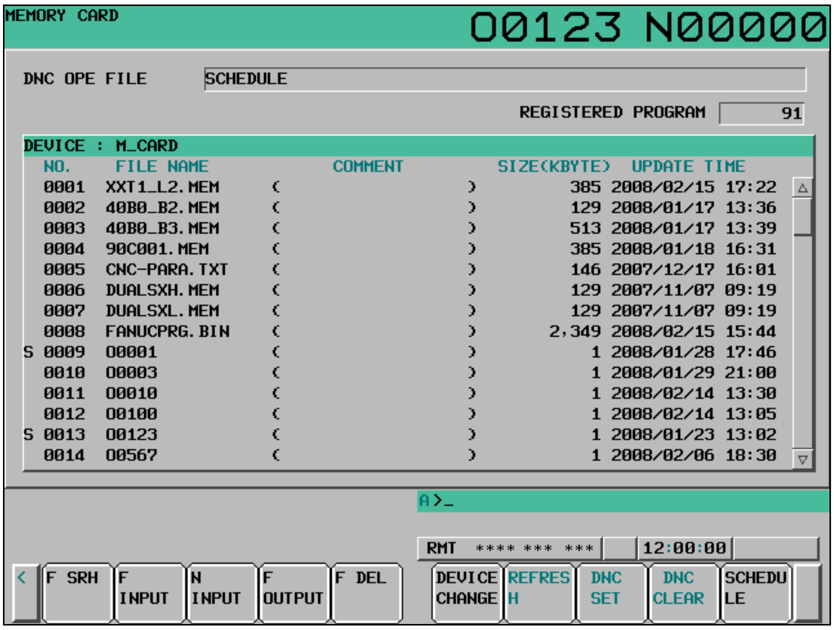


Fig. 4.4 (g) Program list screen (after setting schedule data) (10.4-inch display unit)

3 Press the cycle start switch to execute the selected files. For details on the REMOTE switch, refer to the manual provided by the machine tool builder.

> **NOTE**
> 1 Before setting schedule operation, release DNC operation files in the MDI mode. DNC operation and schedule operation cannot be specified at the same time.
> 2 Before starting schedule operation, confirm that schedule data is set correctly on the schedule list screen.
> 3 Schedule data cannot be changed or edited during schedule operation. Before changing schedule data, make a reset to stop operation.

Restrictions

- Repetition count

The maximum repetition count during schedule operation is 9999. When a negative value is specified, an endless loop (LOOP display) is assumed. The file for which 0 is set is skipped and processing proceeds to the next file.

- Number of registered files

The maximum number of programs that can be registered as schedule setting data is 20.

- Selectable files

The files to be selected as schedule setting data must be registered in the same directory. (Files in the different directories cannot be selected.)

- M code

Even if a code other than M02 and M30 in the execution program is executed, the current count on the schedule execution status screen is not increased.

- Floppy disk directory display during execution of a file

During schedule operation, directories in a floppy disk cannot be displayed in a background edit.

- Intervention during automatic operation

Intervention in schedule operation cannot be performed during automatic operation.

T

- During 2-path control

The scheduling function cannot be used by two paths at the same time.

4.5 EXTERNAL SUBPROGRAM CALL (M198)

During memory operation, you can call and execute a subprogram registered in an external device (such as a Memory Card, Handy File, or Data Server) connected to the CNC.

Format

M198 Pxxxxxxxx Lyyyyyyyy ;

↑ ↑

Pxxxxxxxx : Program number (or file number)
Lyyyyyyyy : Number of repetitive calls
When address L is omitted, the number of repetitive calls is assumed to be 1.

FS0*i*-C compatible command format

M198 P<u>xxxx</u> <u>yyyy</u> ;

↑ ↑

<u>xxxx</u> : Number of repetitive calls
<u>yyyy</u> : Program number (or file number)
When the number of repetitive calls is omitted, it is assumed to be 1.

Explanation

M code M198 specifies an external subprogram call. You can also call an external subprogram using an M code set in parameter No. 6030. (When an M code other than M198 is set as an M code for calling an external subprogram, M198 is executed as a normal M code.)

Specify a program number (file number) registered in an external device at address P. If the specified program number (file number) is not registered in the connected external device, an alarm (PS1079) is issued. (When the memory card is used as external device, alarm (SR1966) is issued.)

Example)

M198 P0123 L3 ;

This command specifies that the subprogram having external subprogram number O0123 is to be called three times repeatedly.
The subprogram is called from the main program and executed as follows:

Main program	1 2 3	Sub program
N0010 ... ;		0123 ... ;
N0020 ... ;		N1020 ... ;
N0030 M198 P0123 L3 ;		N1030 ... ;
N0040 ... ;		N1040 ... ;
N0050 ;		N0050 ... ;
N0060 ... ;		N1060 ... M99 ;

- Program number call

You can also specify a subprogram call with its program number instead of the file number by the setting of bit 2 (SBP) of parameter No. 3404.

NOTE

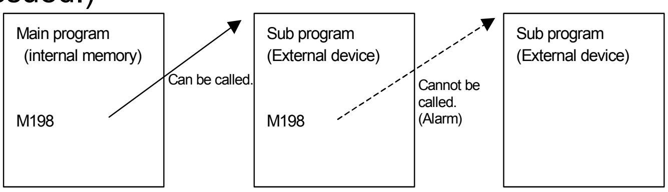
1 An external subprogram call can be specified during program operation in the MEM mode or MDI mode. To make an external subprogram call in the MDI mode, set bit 1 (MDE) of parameter No. 11630 to 1.

2 An external subprogram call is available for the following external devices:

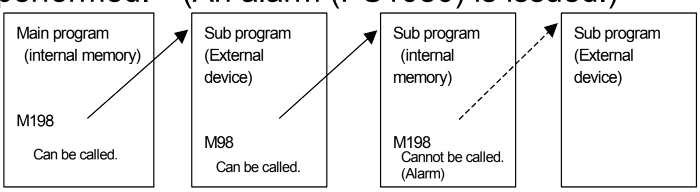
External device name	Program number call	File number call
Handy File	Available	Available
FLOPPY CASSTTE	Available	Available
Memory Card	Available	Unavailable
Data Server	Available	Unavailable

3 To perform a subprogram call using a Memory Card as the external device, set bit 7 (MNC) of parameter No. 138 to 1 and I/O channel (parameter No. 0020) to 4.
A program number call is always enabled regardless of the setting of bit 2 (SBP) of parameter No. 3404.

4 An external device subprogram call cannot be performed from a subprogram called using another external device subprogram call. (An alarm (PS1080) is issued.)



5 A subprogram registered in internal memory can be called from a subprogram called using an external device subprogram call. From the called subprogram in internal memory, another external device subprogram call cannot be performed. (An alarm (PS1080) is issued.)



NOTE

6 A call using the external device subprogram call function is counted as one level of subprogram nesting.

7 In a 2-path system (T series), an external device subprogram call cannot be performed simultaneously from both paths.

4.6 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. One of four types of magnifiers selected with MP1 and MP2 <G019.4 and 5> can be applied. With bit 3 (HIT) of parameter No. 7103, the minimum unit of travel distance can be further 10 times greater. A handle feed magnifier is selected using the manual handle feed amount selection signal. (See Section III-4.6, "MANUAL HANDLE FEED".)

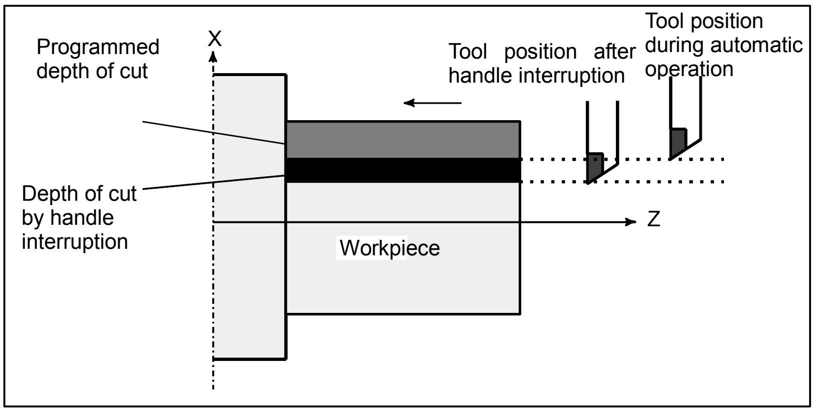


Fig. 4.6(a) Manual handle interruption

⚠ **WARNING**

The travel distance per scale division by manual handle interruption is the least input increment as with manual handle feed. For example, in the case of IS-B, 254 ticks correspond to 0.01 inch for a millimeter input/inch output machine and 100 ticks correspond to 0.254 millimeter for an inch input/millimeter output machine.

Explanation

- Interruption operation

1 When the handle interruption axis selection signal for a handle interruption axis is set to 1 in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, manual handle interruption can be performed by rotating the handle of the manual pulse generator.

NOTE

Even when the feedrate override signal sets 0%, manual handle interruption can be accepted.

2 For the method of selecting a manual handle interruption axis, refer to the relevant manual of the machine tool builder.

3 The feedrate during manual handle interruption is the sum of feedrate used for automatic operation and the feedrate used for movement by manual handle interruption. However, the feedrate during manual handle interruption is controlled so that it does not exceed the maximum allowable cutting feedrate for the axis.

Example

Suppose that the maximum allowable cutting feedrate for an axis is 5 m/min, and that a movement is made in the + direction at 2 m/min along the axis. In this case, manual handle interruption can be accepted even when the manual pulse generator is rotated up to a speed equivalent to 3 m/min. Manual handle interruption by rotation in one direction can be accepted even when the manual pulse generator is rotated to a speed equivalent to 7 m/min.
If the manual pulse generator is rotated to a speed beyond the upper limits, those pulses from the manual pulse generator that correspond to the excess are lost, resulting in a mismatch between the scale mark of the manual pulse generator and the actually interrupted travel distance.

4 For a magnifier for manual handle interruption, refer to the relevant manual of the machine tool builder.

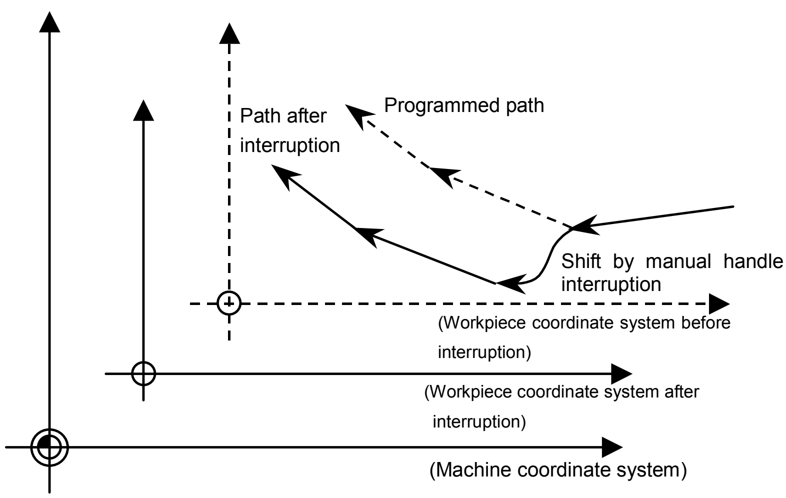
5 If the travel direction is reversed as a result of manual handle interruption, backlash compensation is performed. Pitch error compensation is performed for the position after interruption.

6 In manual handle interruption, only acceleration/deceleration for cutting feed is enabled. By setting bit 0 (MNJ) of parameter No. 1606 to 1, acceleration/deceleration for both of cutting feed and jog feed can be applied to manual handle interruption.

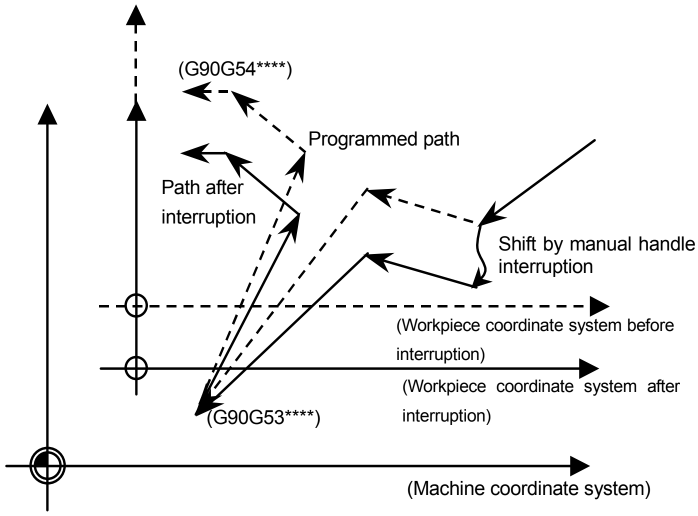
- Manual handle interruption and coordinate system

1 The amount of manual handle interruption shifts the workpiece coordinate systems and the local coordinate system. So, the machine moves, but the coordinates in the workpiece coordinate systems and the local coordinate system remain unchanged.
Regardless of which coordinate system is selected, all workpiece coordinate systems and the local coordinate system shift by the same amount.

- Absolute coordinates
  → Remain unchanged by handle interruption.
- Relative coordinates
  → Change by the amount of handle interruption.
- Machine coordinates
  → Change by the amount of handle interruption.



2 Even when manual handle interruption is performed, the machine coordinate system remains unchanged. The absolute command (G53) in the machine coordinate system is not affected by manual handle interruption.

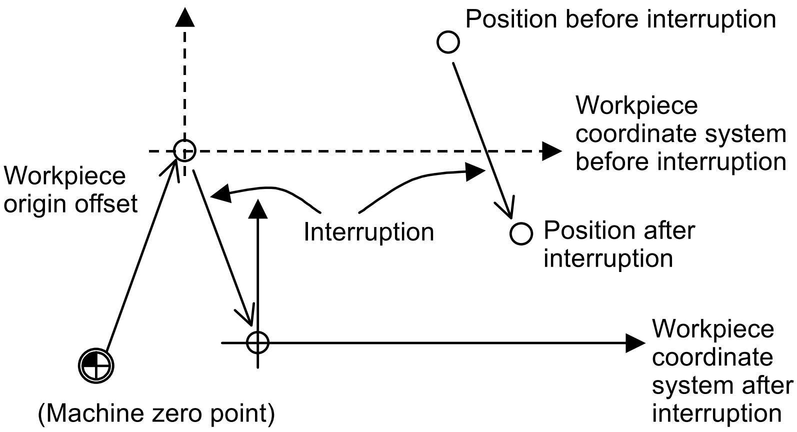


3 In automatic reference position return (G28), the end point (reference position) is not affected by manual handle interruption. However, the midpoint is in the workpiece coordinate system, so that the position shifted by the amount of interruption becomes the midpoint.

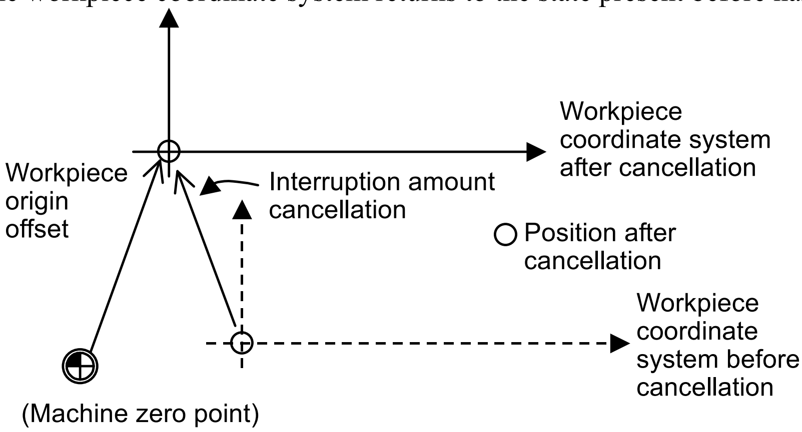
- Cancellation of the amount of interruption

Operation by which the workpiece coordinate system shifted by manual handle interruption from the machine coordinate system is returned to the original workpiece coordinate system is referred to cancellation of the amount of interruption.
When the amount of interruption is canceled, the workpiece coordinate system is shifted by the amount of manual handle interruption, and the amount of interruption is reflected in the absolute coordinates.

Interruption shifts the workpiece coordinate system from the machine coordinate system.



By cancellation, the workpiece coordinate system returns to the state present before handle interruption.



In the following cases, the amount of interruption is canceled:

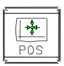
- When a reset is made (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When emergency stop state is canceled (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When a manual reference position return operation is performed
  (when G28 is specified before a reference position is established)
- When a reference position is set without dogs
- When the workpiece coordinate system is preset

> **NOTE**
> When the amount of interruption is cleared using soft keys, only the indication of the amount of interruption becomes 0, and the workpiece coordinate system remains unchanged.

- Clearing the amount of interruption by soft keys

Clearing of the amount of interruption means that the indication of the amount of interruption by manual handle interruption is set to 0. The workpiece coordinate system does not change.
"Clearing all axes" or "Clearing any axis" is performed on the path for which the amount of manual handle interruption is indicated.
When bit 3 (HLC) of parameter No. 7100 is enabled, soft key [INTRPT CANCEL], which is used for this operation, appears. When HLC is disabled, soft key [INTRPT CANCEL] does not appear.

To select "Clearing all axes" or "Clearing any axis", follow the procedure below.

1 Press function key  on the MDI panel.

2 Press soft key [HANDLE].

3 Press soft key [(OPRT)].

4 To prepare for "Clearing all axes" or "Clearing any axis", press soft key [CANCEL].

To prepare for "Clearing all axes" or "Clearing any axis", perform one of the following.

- Clearing all axes
  Press soft key [CANCEL] and then press soft key [ALL-AX].

- Clearing any axis (there are the following two methods.)
  - Enter the axis name and then press [INTRPTCANCEL].
  - Press soft key [INTRPTCANCEL], enter the axis name, and press soft key [EXEC].

  If an incorrect axis name is entered, a warning message stating "FORMAT ERROR" appears.

- Relation with other functions

The following table indicates the relation between other functions and the movement by handle interruption.

Table 4.6(a) Relation between other functions and the movement by handle interruption

Signals	Relation
Machine lock	Machine lock is effective. When machine lock is on, no movement is made due to handle interruption.
Interlock	Interlock is effective. When interlock is on, no movement is made due to handle interruption.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

- Position display

The following table shows the relation between various position display data and the movement by handle interruption.

Table4.6(b) relation between various position display data and the movement by handle interruption

Signals	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Relative coordinates are changed by the travel distance specified by handle interruption.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

- Travel distance display

Press the function key POS, then press the chapter selection soft key [HNDL]. The move amount by the handle interruption is displayed. The following 4 kinds of data are displayed concurrently.

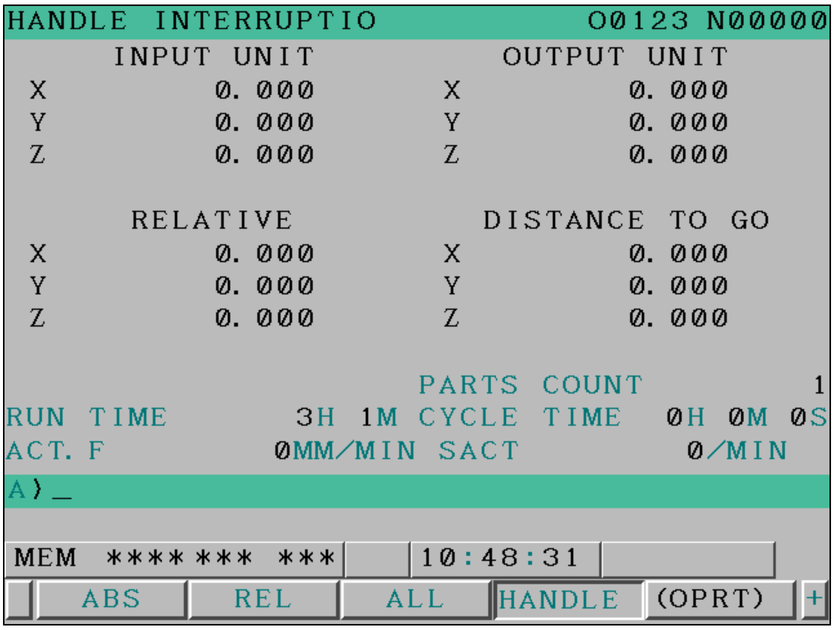


Fig. 4.6 (b)

(a) INPUT UNIT:
Handle interruption move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT :
Handle interruption move amount in output unit system
Indicates the travel distance specified by handle interruption according to the least command increment.

(c) RELATIVE:
Position in relative coordinate system
Relative coordinates are changed by the travel distance specified by handle interruption.

(d) DISTANCE TO GO:
The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

The handle interruption move amount is cleared when the manual reference position return ends every axis.

- Fifth axis display

The fifth axis display of each path is the same as overall position display. See III-12.1.3.

Note

NOTE

1 In a manual operation mode such as the jog feed mode, manual handle feed mode, or TEACH IN HANDLE mode, handle interruption cannot be performed.
2 During a machine lock or interlock, handle interruption does not cause movement.
3 Manual handle interruption is disabled for the axis in any of the following states.
   - Follow-up state
   - PMC axis control state
4 Manual handle interruption cannot be performed for the axis specified in the G00 mode.

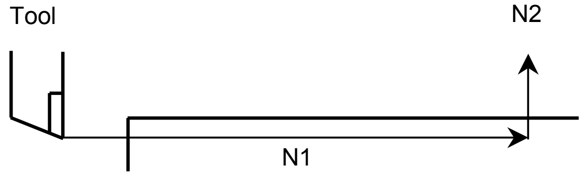
4.7 MANUAL INTERVENTION AND RETURN

When movement along an axis is stopped by feed hold during automatic operation, manual intervention is performed to check the cut surface, and a restart is made, then the tool returns to the position where it was before the intervention and automatic operation is restarted.

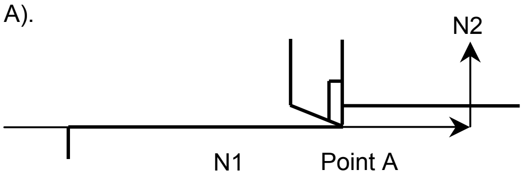
Explanation

To enable automatic return and intervention, set bit 0 (MIT) of parameter No. 7001 to be enabled.
The automatic return and intervention sequence is described below.

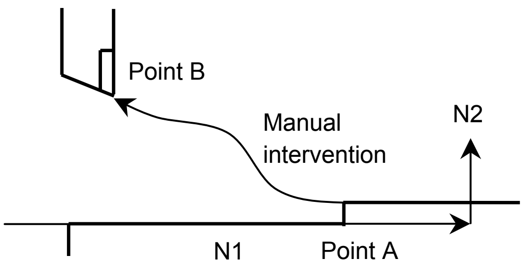
1. The N1 block cuts a workpiece



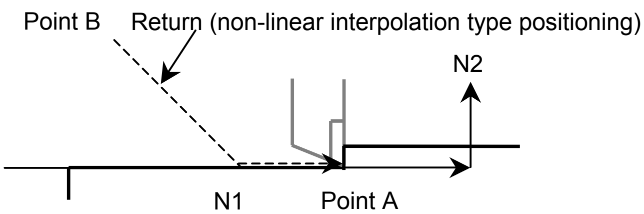
2. The tool is stopped by pressing the feed hold switch in the middle of the N1 block (point A).



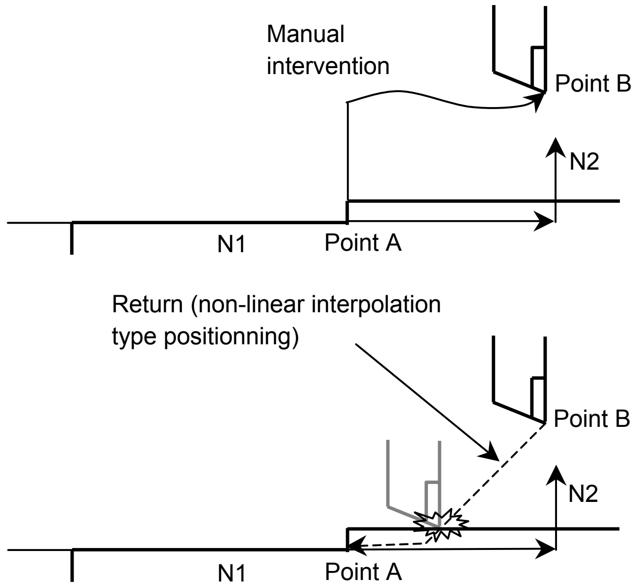
3. After retracting the tool manually to point B, tool movement is restarted.



4. After automatic return to point A at the dry run feedrate, the remaining move command of the N1 block is executed.



> ⚠ **WARNING**
>
> Be sure to perform correct intervention according to the machining direction and workpiece figure. Otherwise, the workpiece, machine, or tool may be broken.



- Manual absolute on/off

In cases such as when tool movement along an axis is stopped by feed hold during automatic operation so that manual intervention can be used to replace the tool: When automatic operation is restarted, this function returns the tool to the position where manual intervention was started.

- Return feedrate

The return feedrate is a dry run feedrate and jog feedrate override is enabled. When manual rapid traverse signal RT(G0019.7) is 1, the return feedrate is not a dry run feedrate, but a rapid traverse rate.

- Return operation

Return operation is performed according to non-linear interpolation type positioning.

- Single block

If the single block stop switch is on during return operation, the tool stops at the stop position and restarts movement when the cycle start switch is pressed.

- Cancellation

When a reset, alarm, or emergency stop occurs during manual intervention or return, the manual intervention and return function is canceled.

- MDI mode

The manual intervention and return function is enabled in the MDI mode.

Limitation

- Enabling and disabling manual intervention and return

This function is enabled only when the automatic operation hold LED is on. When the remaining travel distance is 0, if a feed hold stop is made and manual intervention is performed, manual intervention and return is disabled and operation is performed according to the specification of the manual absolute on/off function.

- Offset
When the tool is broken, if the tool is replaced by manual intervention and then processing is restarted from the midpoint in the interrupted block, a change in the offset is not reflected.

- Machine lock and mirror image
When performing manual intervention and return, do not apply a machine lock or mirror image.

M

- Scaling
When performing manual intervention and return, do not apply scaling.

4.8 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the MDI panel.

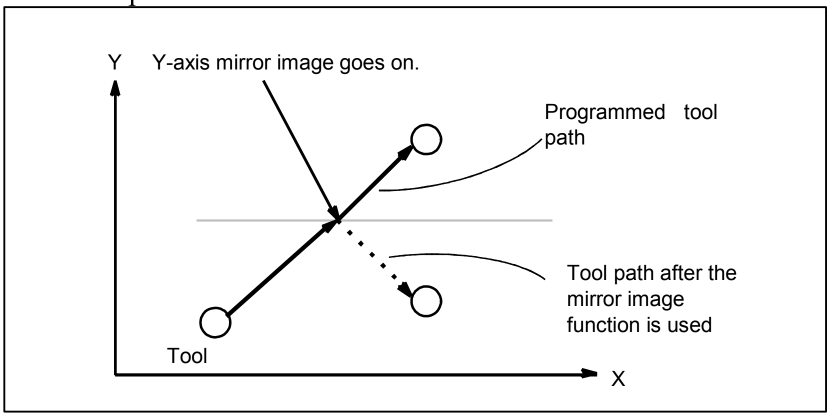
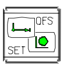


Fig. 4.8 (a) Mirror image

Procedure for mirror image

Procedure
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

1 Press the single block switch to stop automatic operation.
  When the mirror image function is used from the beginning of operation, this step is omitted.
2 Press the mirror image switch for the target axis on the machine operator's panel.
  Alternatively, turn on the mirror image setting by following the steps below:
  2-1 Set the MDI mode.
  2-2 Press the function key .

2-3 Press the [SETING] soft key for chapter selection to display the setting screen.

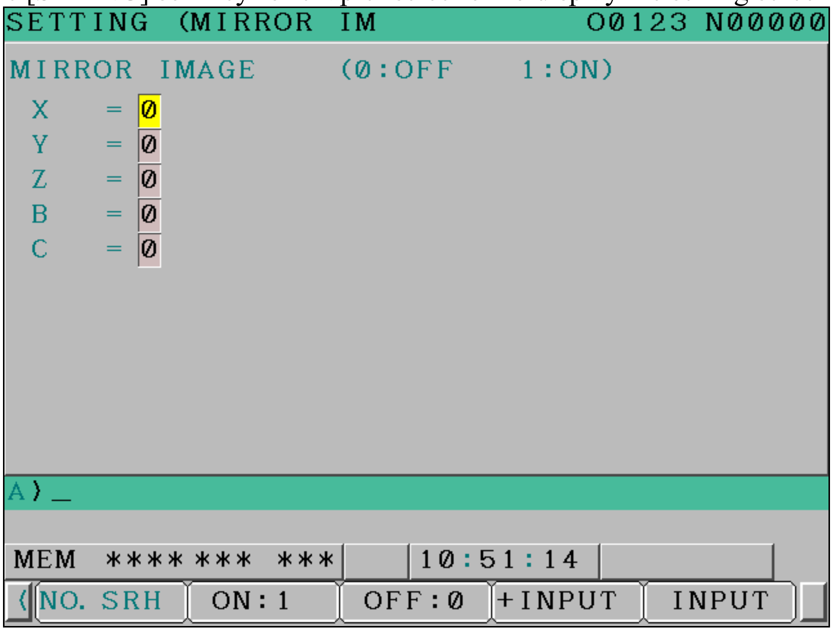


Fig. 4.8 (b) Setting screen

2-4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (memory mode or MDI mode), then press the cycle start button to start automatic operation.

Explanation

- The mirror image function can also be turned on and off by setting bit 0 (MIRx) of parameter No.0012 to 1 or 0.
- For the mirror image switches, refer to the manual supplied by the machine tool builder.

Limitation

T

The direction of movement during machine coordinate system setting (G53), the direction of movement during manual operation, and the direction of movement from an intermediate point to the reference position during automatic reference position return (G28) cannot be reversed.

M

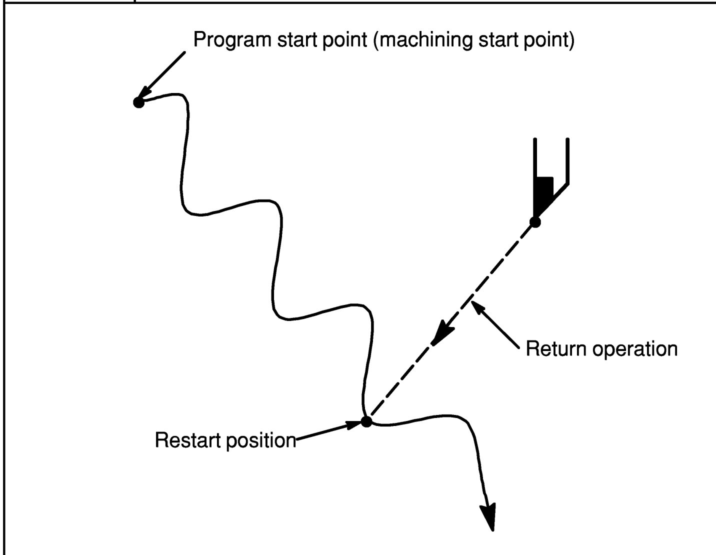
The direction of movement during machine coordinate system setting (G53), the direction of movement during manual operation, the direction of movement from an intermediate point to the reference position during automatic reference position return (G28), the direction of approach during single direction positioning (G60), and the shift direction in a boring cycle (G76, G87) cannot be reversed.

4.9 PROGRAM RESTART

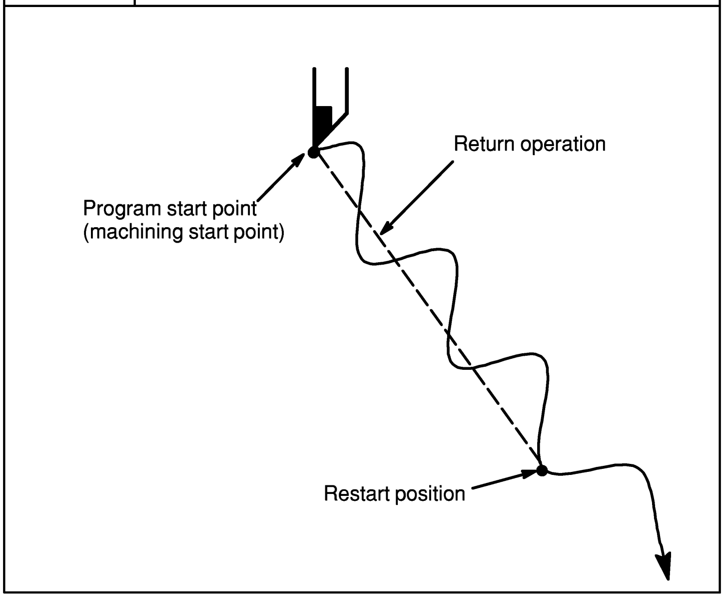
This function specifies Sequence No. of a block to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high-speed program check function.

There are two restart methods: the P-type method and Q-type method.

P TYPE	Operation can be restarted anywhere. This restart method is used when operation is stopped because of a broken tool.



Q TYPE	Before operation can be restarted, the machine must be moved to the programmed start point (machining start point)



Procedure for program restart by specifying a sequence number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to step 2.)

Procedure 2

[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.

2 Press PROG key to display the desired program.

3 Find the program head. Press RESET key.

4 Enter the sequence number of the block to be restarted, then press the [P TYPE] or [Q TYPE] soft key.

N xxxxx [Q TYPE] or [P TYPE]

Sequence number

If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number.

N xxxyyyyy [Q TYPE] or [P TYPE]

Sequence number (The rightmost five digits.)

Frequency

5 The sequence number is searched for, and the program restart screen appears on the LCD display.

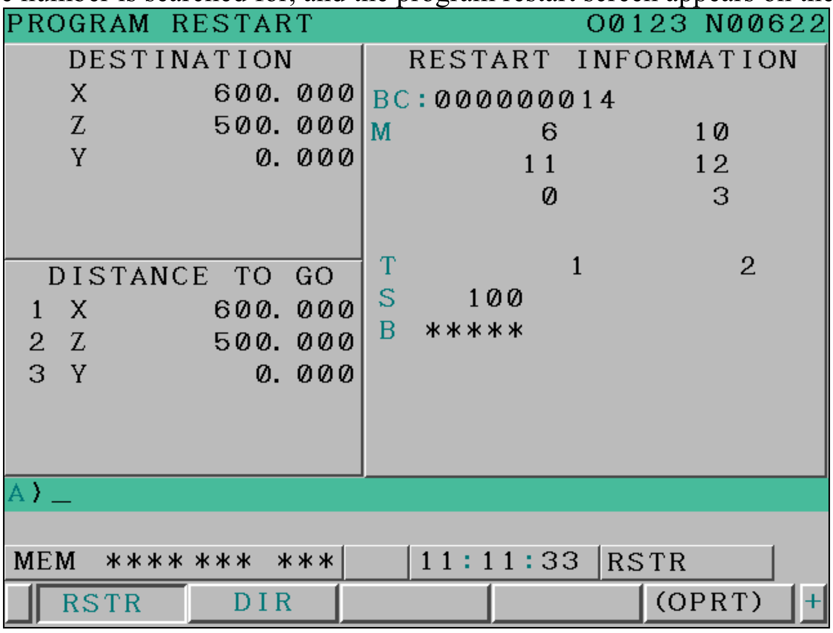


Fig. 4.9 (a) Program restart screen

DESTINATION shows the position at which machining is to restart.

DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.

The coordinates and amount of travel for restarting the program can be displayed for up to four axes. If your system supports five or more axes, pressing the [RSTR] soft key again displays the data for the fifth and subsequent axes.

M : Up to 35 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.

With 10.4-inch LCD/MDI panel : Up to 30 M codes

With 8.4-inch LCD/MDI panel : Up to 6 M codes

T : Two most recently specified T codes
S : Most recently specified S code
B : Most recently specified B code
Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.
7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.
8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.
9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter No. 7310 settings. Machining is then restarted.

Procedure for program restart by specifying a block Number

Procedure 1

[P TYPE]

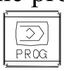
1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to step 2.)

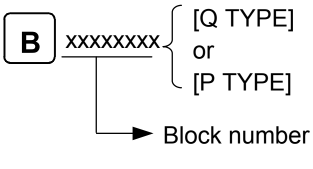
Procedure 2

[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.
2 Press 

key to display the desired program.
3 Find the program head. Press 

key.
4 Enter the number of the block to be restarted then press the [P TYPE] or [Q TYPE] soft key. The block number cannot exceed eight digits.



5 The block number is searched for, and the program restart screen appears on the LCD display.

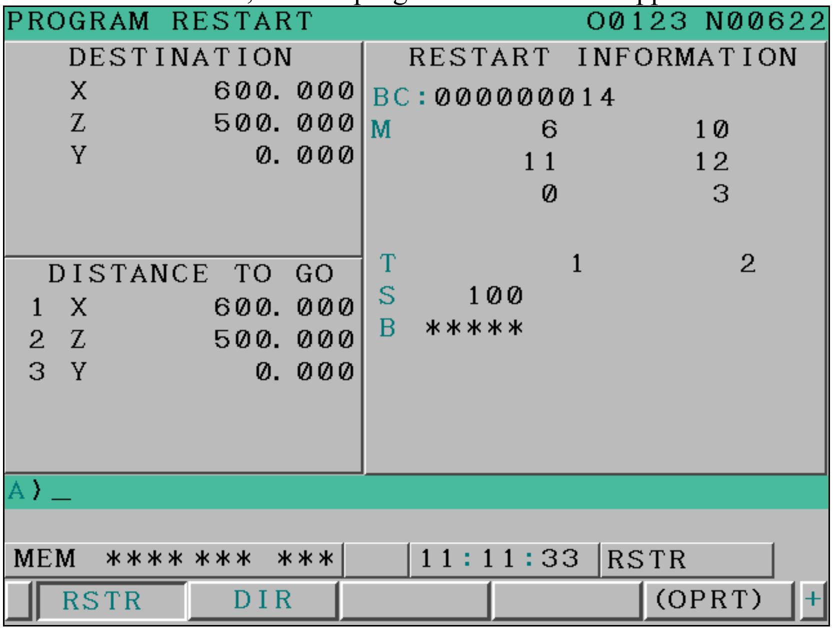


Fig. 4.9 (b) Program restart screen

DESTINATION shows the position at which machining is to restart.
DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.
The coordinates and amount of travel for restarting the program can be displayed for up to four axes. For a path with five axes, when soft key [RSTR] is pressed again to display the fifth axis.

M : Up to 35 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.
With 10.4-inch LCD/MDI panel : Up to 30 M codes
With 8.4-inch LCD/MDI panel : Up to 6 M codes

T : Two most recently specified T codes

S : Most recently specified S code

B : Most recently specified B code

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter No. 7310 settings. Machining is then restarted.

Outputting the M, S, T, and B codes for program restart

After the block to be restarted is searched for, you can perform the following operations:

1 Before the tool is moved to the machining restart position
<1> The most recently specified M, S, T, and B codes can automatically be output to the PMC. The most recently specified S code is output as the maximum spindle speed when the S code is specified in the block containing G92 or as the specified spindle speed in other cases. As the most recently specified S code, only one S code is displayed on the program restart screen regardless of whether the S code is specified in the block containing G92.
<2> While the block to be restarted is being searched for, all sampled M codes and most recently specified S, T, and B codes can automatically be output to the PMC. Up to 35 M codes can be sampled. If the number of sampled M codes exceeds 35, the 35 most recently specified M codes are output to the PMC.
Switch between operations <1> and <2> using bit 6 (MOA) of parameter No. 7300.
2 Before the tool reaches the machining restart position
On the program restart screen, you can specify M, S, T, and B codes from the MDI panel in the MEM or RMT mode without changing the mode.

Outputting the most recently specified M, S, T, and B codes

When bit 7 (MOU) of parameter No. 7300 is 1 and bit 6 (MOA) of parameter No. 7300 is 0, if the cycle start button is pressed after searching for the block to be restarted, the last M, S, T, and B codes are automatically output to PMC before moving to the machining restart point.
In the single block stop status, after the most recently specified M, S, T, and B codes are output, pressing the cycle start switch again moves the tool to the machining restart position.

Outputting all M codes and most recently specified S, T, and B codes

When bit 7 (MOU) of parameter No. 7300 is 1 and bit 6 (MOA) of parameter No. 7300 is 1, if the cycle start button is pressed after searching for the block to be restarted, all M codes and the last S, T, and B codes are automatically output to PMC before moving to the machining restart point.
(Example)
When M10, M11, M12, M13, M14, T0101, S1000, and B10 are sampled, a program is executed in the format shown below before the tool is moved to the machining restart position:
M10 T0101 S1000 B10 ;
M11 ;
M12 ;
M13 ;
M14 ;

Outputting M, S, T, and B codes on the program restart screen

When bit 7 (MOU) of parameter No. 7300 is set to 1, you can specify M, S, T, and B codes from the MDI panel in the MEM or RMT mode without changing the mode after searching for the block to be restarted until the tool reaches the machining restart position.

Procedure

1 When the block to be restarted is searched for using the program restart function, the program restart screen appears. When bit 7 (MOP) of parameter No. 7300 is set to 1, operation soft keys [OVERSTORE], [ERASE], and [INPUT] are displayed.

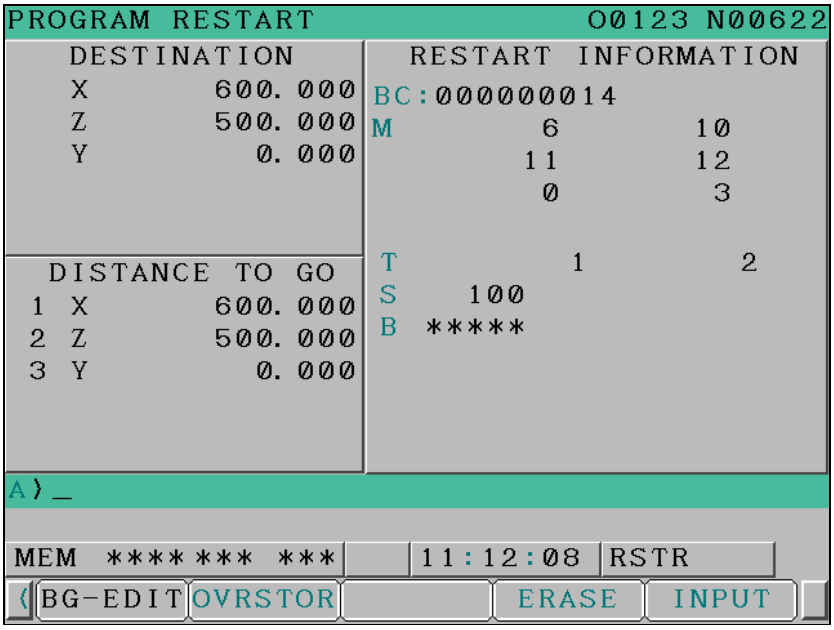


Fig. 4.9 (c) Program restart screen (outputting M, S, T, and B codes)

2 Before the tool reaches the machining restart position, pressing soft key [OVERSTORE] selects the over store mode. In the over store mode, data can be entered in the M, S, T, and B fields displayed in the (OVERSTORE) section.
To select the over store mode while the tool is moving to the machining restart position, hold restart operation by feed hold and press soft key [OVERSTORE].
Enter M, S, T, and B codes to be output in the (OVERSTORE) section from the MDI panel.
(Example)
To enter M10, S1000, T101, and B20 in the (OVERSTORE) section:
<1> Enter M 1 0 from the MDI panel.
<2> Press the [INPUT] key.
You can also enter the S, T, and B codes by performing steps <1> and <2>.

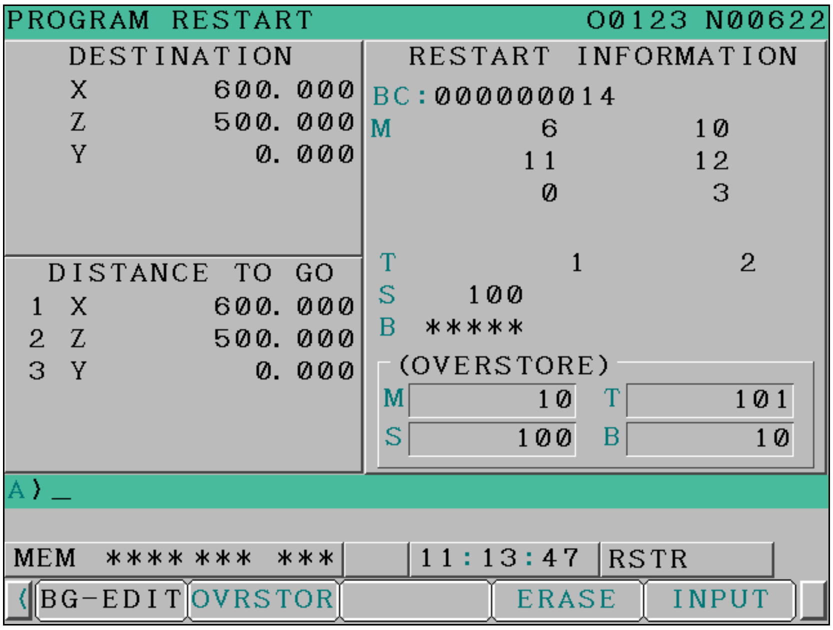


Fig. 4.9 (d) Program restart screen when M, S, T, and B codes are output

3 When values have been entered in the (OVERSTORE) section, pressing the cycle start switch outputs each code in the (OVERSTORE) section. The values in the (OVERSTORE) section are cleared.
4 To clear the values entered in the (OVERSTORE) section as M, S, T, and B codes, press soft key [ERASE]. All entered values are cleared.
5 Pressing soft key [OVERSTORE] again in the over store mode cancels the mode. Pressing the reset key also cancels the over store mode.
6 To continue with restart operation, cancel the over store mode and press the cycle start switch.

⚠ CAUTION

1 The M, S, T, and B codes specified in the over store mode are not displayed on the program restart screen.
2 In the over store mode, changing the operation mode to other than the MEM or RMT mode does not cancel the over store mode. In this case, no values can be entered in the (OVERSTORE) section.
3 In T series, do not specify a T code in the over store mode. If a T code is specified, it is not executed.

Explanation

- Block number

When the CNC is stopped, the number of executed blocks is displayed on the program screen or program restart screen. The operator can specify the number of the block from which the program is to be restarted, by referencing the number displayed on the screen. The displayed number indicates the number of the block that was executed most recently. For example, to restart the program from the block at which execution stopped, specify the displayed number, plus one.
The number of blocks is counted from the start of machining, assuming one NC line of a CNC program to be one block.

(Example 1)

CNC Program	Number of blocks
O0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G01 X100. F100 ;	3
G01 Z-50. F50 ;	4
M30 ;	5

(Example 2)

CNC Program	Number of blocks
O0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G90 G00 Z100. ;	3
G81 X100. Y0. Z120. R-80. F50. ;	4
#1=#1+1 ;	4
#2=#2+1 ;	4
#3=#3+1 ;	4
G00 X0 Z0 ;	5
M30 ;	6

Macro statements are not counted as blocks.

- Storing / clearing the block number

The block number is held in memory while no power is supplied. The number can be cleared by cycle start in the reset state.

- Block number when a program is halted or stopped

The program screen usually displays the number of the block currently being executed. When the execution of a block is completed, the CNC is reset, or the program is executed in single-block stop mode, the program screen displays the number of the program that was executed most recently. When a CNC program is halted or stopped by feed hold, reset, or single-block stop, the following block numbers are displayed:

Feed hold	: Block being executed
Reset	: Block executed most recently
Single-block stop	: Block executed most recently

For example, when the CNC is reset during the execution of block 10, the displayed block number changes from 10 to 9.

- MDI intervention

When MDI intervention is performed while the program is stopped by single-block stop, the CNC commands used for intervention are not counted as a block.

- Block number exceeding eight digits

When the block number displayed on the program screen exceeds eight digits, the block number is reset to 0 and counting continues.

Limitation

- P type Restart

In the following conditions, P type restart cannot be performed:

- Automatic operation has not been performed since power-on.
- Automatic operation has not been performed since the coordinate system was changed or shifted (change of the external workpiece origin offset value).

- Restart block

The block where the program is to restart is not necessarily be the block at which the program was interrupted. You can restart the program from any block. For P-type restart, however, the block where the program is to restart must use the same coordinate system as when program execution was interrupted.

- Single block

When the single-block operation is enabled at the time of a movement to the restart point, a single-block stop occurs each time an axis operation takes place. In this case, no MDI operation is allowed.

- Manual intervention

During movement to the restart point, manual intervention is allowed for an axis for which a return operation has not yet been performed. However, manual operations do not cause any movement along axes for which a return operation has already been completed.

- MDI

When the search operation has ended, no move command can be specified by MDI before axis movement.

- Reset

Do not perform a reset operation during the time from the start of the search operation of the restart sequence until machining is restarted.
If a reset operation is performed, the restart steps must be performed again from the beginning.

- Feed hold
If a feed hold operation is performed during the search, the restart steps must be performed again from the beginning.

- Manual absolute
Every manual operation must be performed with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return
Unless an absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-up, then perform restart operation.

- Program restart switch
When the program restart switch is on, pressing the cycle start switch does not start operation.

- Blocks specifying a macro statement, macro call, and subprogram call
Blocks specifying a macro statement, macro call, and subprogram call are not searched for even when they have a sequence number. In such a case, search for a block previously preceding such a block.

- Interruption type custom macro
During movement to the machining restart point at a dry run feedrate, no interruption type custom macro can be started. If an interruption type custom macro is started, alarm DS024 is issued.

M

- Index table indexing
For a machine that uses index table indexing, position the machine at the restart point in advance before restarting the program.

- Commands that prevent program restart
Program restart cannot be performed for blocks placed in the following modes:
- Cs contouring control
- Threading (G32,G33)
- Rigid tapping

T

- Polygon turning (G50.2)
- Threading cycle (G92)
- Multiple repetitive threading cycle (G76)
- Polar coordinate interpolation (G12.1)
- Balance cutting (G68)

If any of the following commands is included between the beginning of a program and the block where the program is to restart, program restart cannot be performed:
- Workpiece coordinate system preset (G92.1,G50.3)
- Commands for enabling and disabling axis synchronous control

T

- Commands for enabling and disabling synchronous/composite control and superimposed control

- M, S, and T commands not usable in over store mode

The M, S, and T functions listed below, unlike the other M, S, and T functions, have special meanings within the CNC. These M, S, and T commands cannot be specified from the over store screen. To specify these commands, cancel the over store mode, and execute them in MDI operation.

Example:

- Rigid tapping

T

- Spindle positioning

> ⚠ **WARNING**
>
> As a rule, the tool cannot be returned to a correct position under the following conditions.
>
> Special care must be taken in the following cases since none of them cause an alarm:
>
> - Manual operation is performed when the manual absolute mode is OFF.
> - Manual operation is performed when the machine is locked.
> - When the mirror image is used. However, P type return is possible for a block that switched between ON and OFF most recently or a subsequent block. In this case, the mirror image signal status present when the program was interrupted must be maintained.
> - When no coordinate system is set up at the beginning of a program in which main commands are executed in the incremental mode.
> - When manual operation is performed in the course of axis movement for returning operation.
> - When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
> - When program restart is specified in the machine lock state, then the machine lock is canceled.
> - When program restart specified for an intermediate block for a multiple repetitive canned cycle (T series)
> - In general, when a coordinate system is set up, changed, or shifted after the search operation ends, the tool cannot be returned to a correct position.

> ⚠ **CAUTION**
>
> Keep the following in mind when restarting a program including macro variables.
>
> - Common variable
>
>   When the program is restarted, the previous values are inherited as common variables without being preset automatically. Before restarting the program, initialize the appropriate variables to the original values used at start of the previous automatic operation.
> - DI/DO
>
>   At restart of the program, DI can be read by a system variable, but DO cannot be output.
> - Clock
>
>   When the program is being restarted, the clock time can be obtained by a system variable, but the time cannot be preset.
> - Tool offset and workpiece origin offset
>
>   When the program is being restarted, the offset can be read by a system variable, but change of the offset is allowed only for the Q type.

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.
There are two types of machine lock: all-axis machine lock, which stops the movement along all axes, and specified-axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, T, and B (2nd auxiliary function) commands, is available for checking a program together with machine lock.

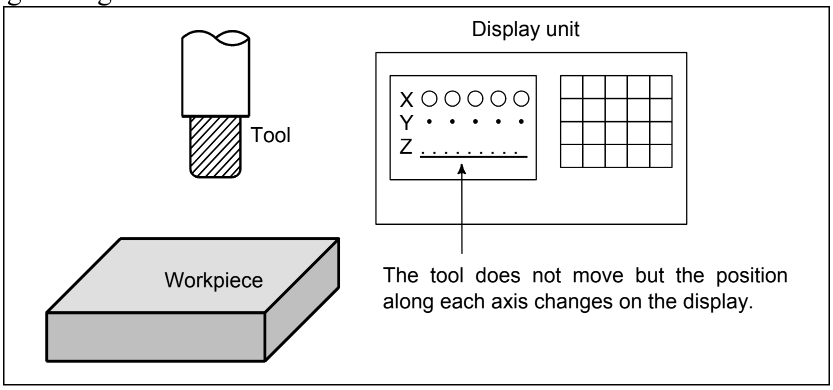


Fig. 5.1 (a) Machine lock

Machine lock and auxiliary function lock

Procedure

- Machine Lock

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.
Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.

> ⚠ **WARNING**
> The positional relationship between the workpiece coordinates and machine coordinates may differ before and after automatic operation using machine lock. In such a case, specify the workpiece coordinate system by using a coordinate setting command or by performing manual reference position return.

- Auxiliary function lock

Press the auxiliary function lock switch on the operator's panel. M, S, T, and B codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Limitation

- M, S, T, B command by only machine lock

M, S, T and B commands are executed in the machine lock state.

- Reference position return under machine lock

When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

- M codes not locked by auxiliary function lock

M00, M01, M02, M30, M98, M99, and M198 (external subprogram call function) commands are executed even in the auxiliary function lock state. M codes for calling a subprogram (parameters No. 6071 to 6079) and those for calling a custom macro (parameters No. 6080 to 6089) are also executed.

5.2 FEEDRATE OVERRIDE

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial. This feature is used to check a program.
For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

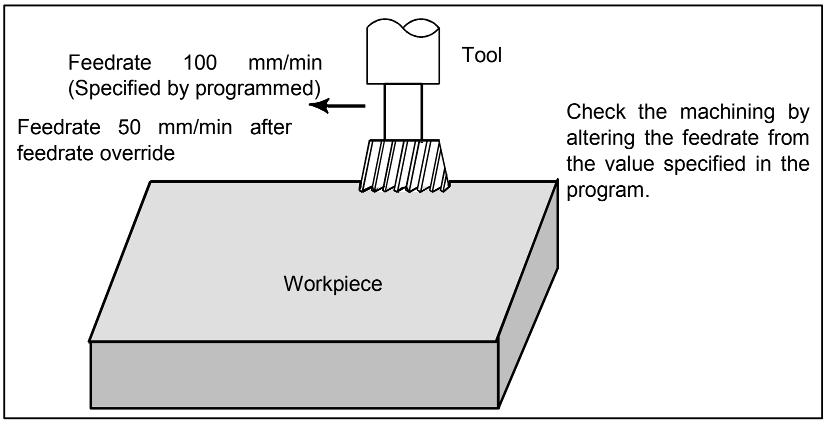


Fig. 5.2 (a) Feedrate override

Feedrate override

Procedure

Set the feedrate override dial to the desired percentage (%) on the machine operator's panel, before or during automatic operation.
On some machines, the same dial is used for the feedrate override dial and jog feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Limitation

- Override range

The override that can be specified ranges from 0 to 254%. For individual machines, the range depends on the specifications of the machine tool builder.

- Override during thread

During the threading process, the override setting is ignored; it is always regarded as 100% during the process.

5.3 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 No. 1421.
A rapid traverse override can be selected in steps of 1% or 0.1% in the range of 0 to 100%.

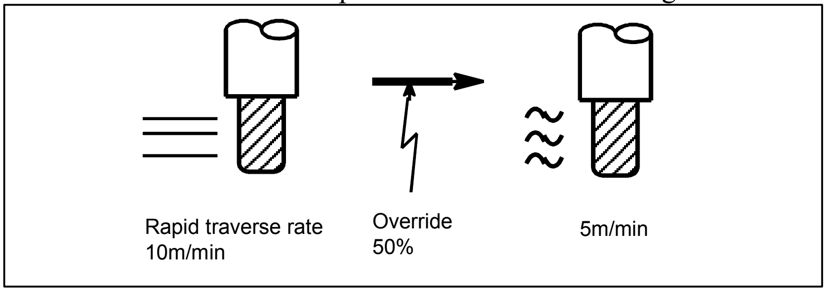


Fig. 5.3 (a) Rapid traverse override

Rapid traverse override

Procedure

Select one of the four feedrates with the rapid traverse override switch during rapid traverse.
Select a rapid traverse override in steps of 1% or 0.1%.
Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

(1) Rapid traverse by G00
(2) Rapid traverse during a canned cycle
(3) Rapid traverse in G27, G28, G29 (M series), G30, G53
(4) Manual rapid traverse
(5) Rapid traverse of manual reference position return

5.4 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

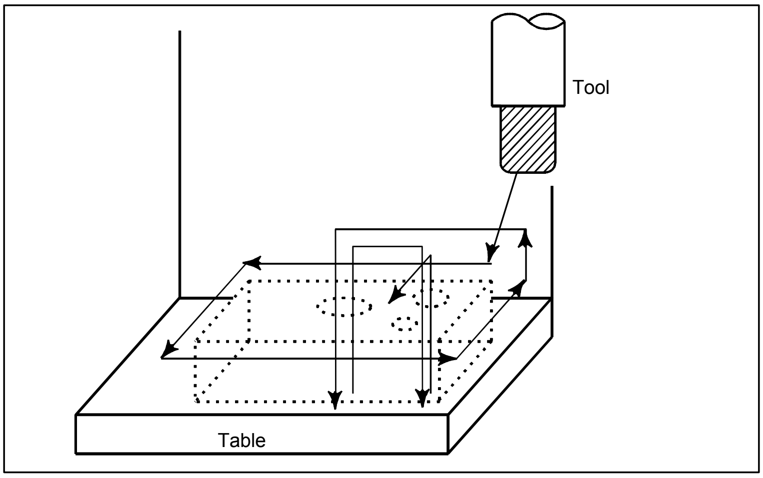


Fig. 5.4 (a) Dry run

Dry run

Procedure

Press the dry run switch on the machine operator's panel during automatic operation.
The tool moves at the feedrate specified in a parameter. The rapid traverse switch (manual rapid traverse selection signal) can also be used for changing the feedrate. Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

- Dry run feedrate

The dry run feedrate changes as shown in the table below according to the rapid traverse switch (manual rapid traverse selection signal) and parameters.

Table 5.4 (a) Feedrate during dry run

Rapid traverse switch	Program command	
	Rapid traverse	Cutting feed
ON	Rapid traverse rate	Dry run feedrate × Jvmax(*2)
OFF	Dry run feedrate × JV, or rapid traverse rate (*1)	Dry run feedrate × JV(*2)

Max. cutting feedrate...Setting by parameter No.1430
Rapid traverse rate...Setting by parameter No.1420
Dry run feedrate ....Setting by parameter No.1410

(*1) Dry run feedrate × JV when parameter RDR (No. 1401#6) is 1. Rapid traverse rate when parameter RDR is 0.
JV Jog feedrate override

(*2) Clamped to the maximum cutting feedrate
Jvmax Maximum value of jog feedrate override

5.5 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.

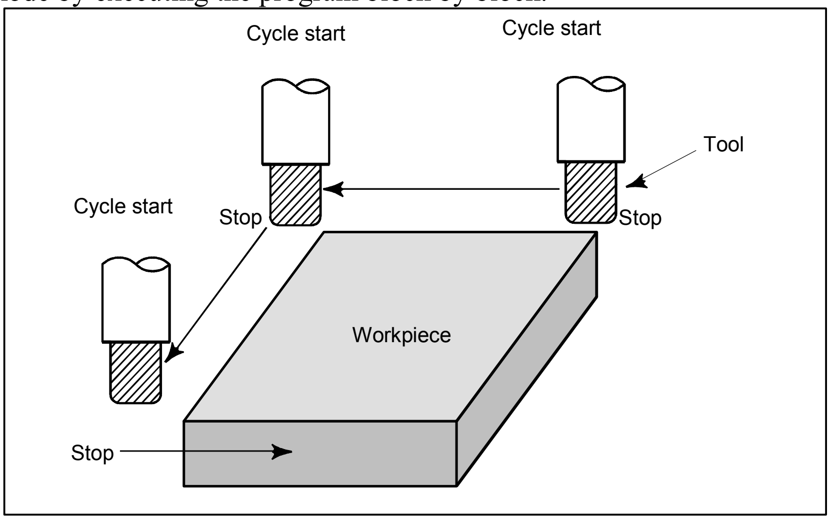


Fig. 5.5 (a) Single block

Single block

Procedure

1 Press the single block switch on the machine operator's panel.
The execution of the program is stopped after the current block is executed.

2 Press the cycle start button to execute the next block. The tool stops after the block is executed.
Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

- Reference position return and single block

If G28, G29 (M series), and G30 are issued, the single block function is effective at the intermediate point.

- Single block during a canned cycle

In a canned cycle, the single block stop points are the end of <1>, <2>, and <6> shown below. When the single block stop is made after the point <1> or <2>, the feed hold LED lights.

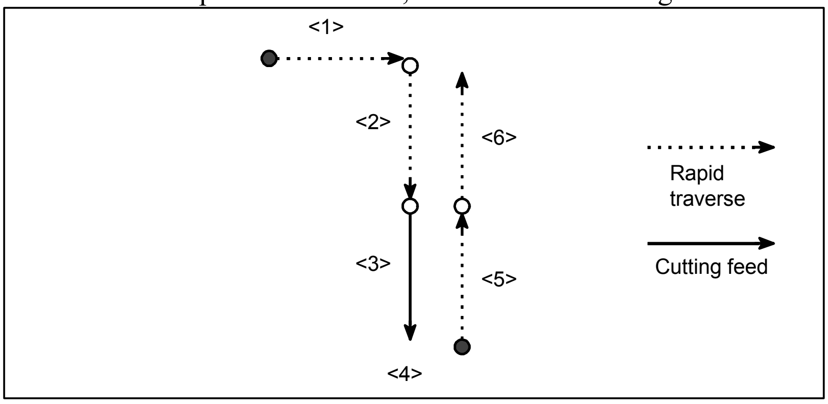


Fig. 5.5 (b) Single block during canned cycle

- Subprogram call and single block

Single block stop is not performed in a block containing M98P_ ;. M99 ; or G65.
However, single block stop is even performed in a block with M98P_ or M99 command, if the block contains an address other than O, N, P, L.

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stored stroke check are available. This chapter describes emergency stop, overtravel check, and stored stroke check.

Chapter 6, "SAFETY FUNCTIONS", consists of the following sections:

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

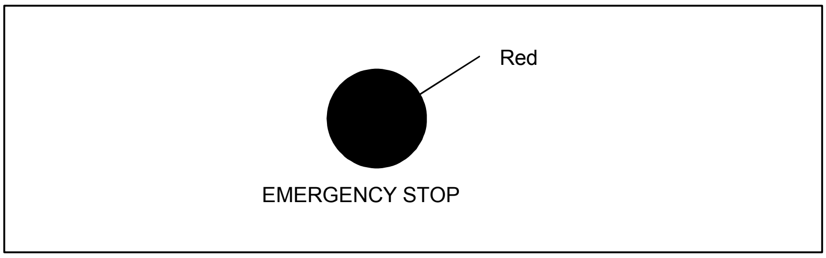


Fig. 6.1 (a) Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 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 is displayed.

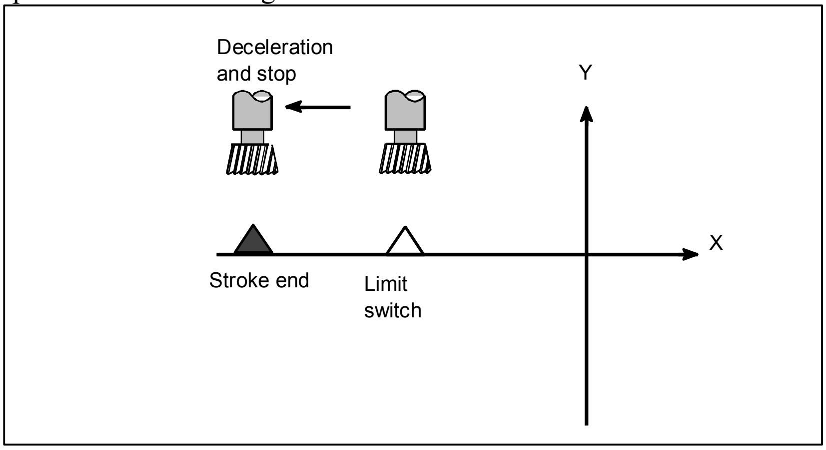


Fig. 6.2 (a) Overtravel

Explanation

- Overtravel during automatic operation

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

- Overtravel during manual operation

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

- Releasing overtravel

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Alarm

Table6.2 (a)

Alarm No.	Message	Description
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.

6.3 STORED STROKE CHECK

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

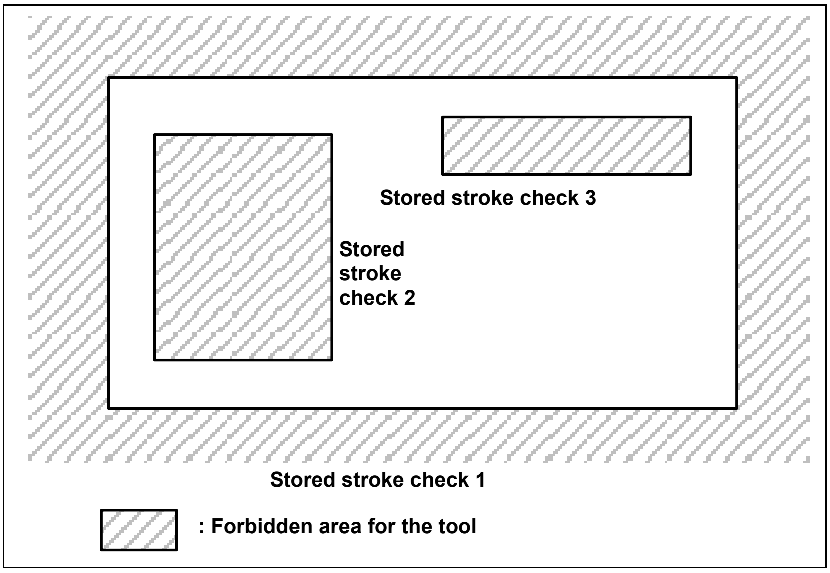


Fig. 6.3 (a) Stroke check

The following shows the areas which the tool cannot enter for each stored stroke check.

- Stored stroke check 1: Outside
- Stored stroke check 2: Outside or inside (switchable)
- Stored stroke check 3: Inside

When the tool moves into the forbidden area, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

- Stored stroke check 1

Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.
At this time, a signal (overtravel alarm signal) can be output to the PMC if bit 6 (OTS) of parameter No. 1301 is set to 1. In addition, when the tool enters the forbidden area during manual operation, the signal (overtravel alarm signal) can be output to the PMC without generating the alarm by setting bit 1 (NAL) of parameter No. 1300 to 1. With this parameter setting, the alarm is generated when the tool enters the forbidden area during automatic operation.

> ⚠ **CAUTION**
> 1 If the two points for specifying a forbidden area are identical, all areas are handled as forbidden areas for stored stroke check 1.
> 2 The size of a forbidden area must be set carefully. If the size is set incorrectly, the stroke becomes infinite.

- Stored stroke check 2

Parameters (Nos. 1322, 1323) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter OUT (No. 1300#0) selects either inside or outside as the forbidden area.
In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area.
Each of G22; and G23; should be commanded independently of another commands in a block.
The command below creates or changes the forbidden area:

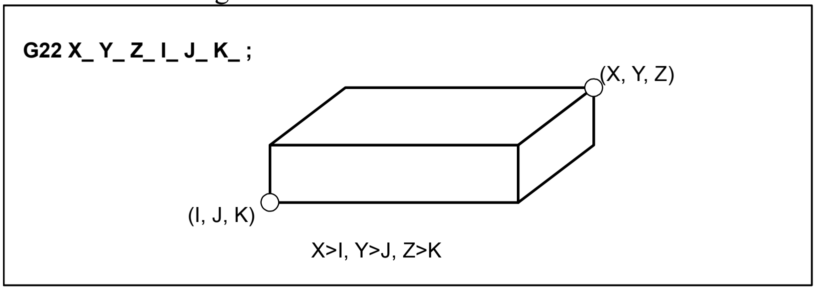


Fig. 6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

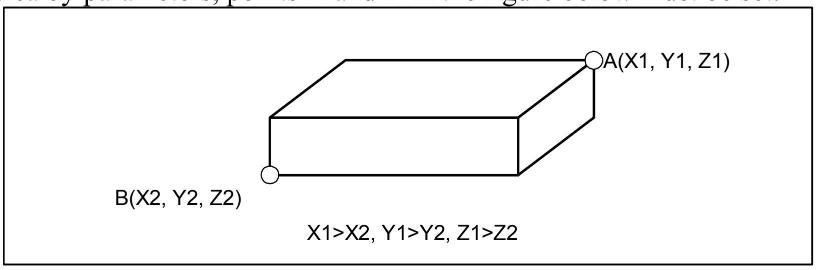


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

The values X1, Y1, Z1, X2, Y2, and Z2, which are set by parameters No. 1322 and No. 1323, must be specified by the distance from the machine coordinate system (machine unit). The values X, Y, Z, I, J, and K, which are set by a G22 command, must be specified by the distance in the least input increment (input unit).
Values set by a program are then converted in the machine increment and the values are set as the parameters.

- Stored stroke check 3

Set the boundary with parameters No. 1324 and 1325. The area inside the boundary becomes the forbidden area. The values X1, Y1, Z1, X2, Y2, and Z2 must be set as coordinates (machine unit) in the machine coordinate system.

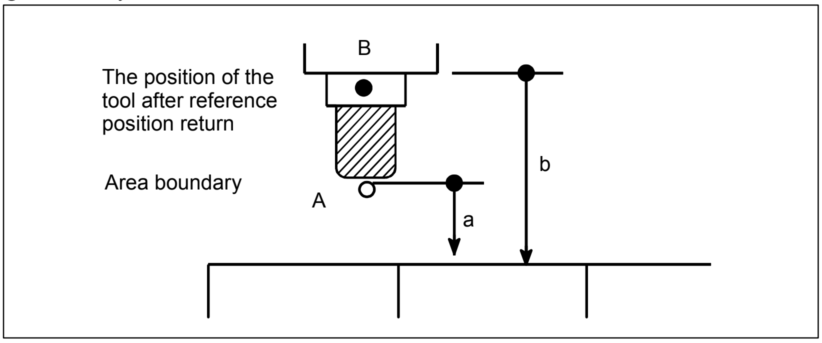
> ⚠ **CAUTION**
>
> 1 If the two points for specifying a forbidden area are identical, all areas are handled as movable areas for stored stroke check 2/3.
>
> 2 Even if the magnitude relation of the two points for specifying a forbidden area is incorrectly set, a rectangular parallelepiped having the two points as apexes on its diagonal line is assumed as the boundaries in stored stroke check 2/3.
>
> 3 Since an axis without the reference position return function has no forbidden areas, there are no alarms about forbidden areas for the axis.

- Checkpoints in the forbidden area

The parameter setting or programmed value (XYZIJK) depends on which part of the tool or tool holder is checked for entering the forbidden area.

If point A (the top of the tool) is checked in Fig. 6.3(d), the distance "a" should be set as the data for the stored stroke limit function. If point B (the tool chuck) is checked, the distance "b" must be set. A 点
When a tool tip such as point A is checked, if the length and diameter of the tool change variously, make settings using the maximum length and diameter. This eliminates the setting for each tool and makes machining safe.

- For machining center system



- For lathe system

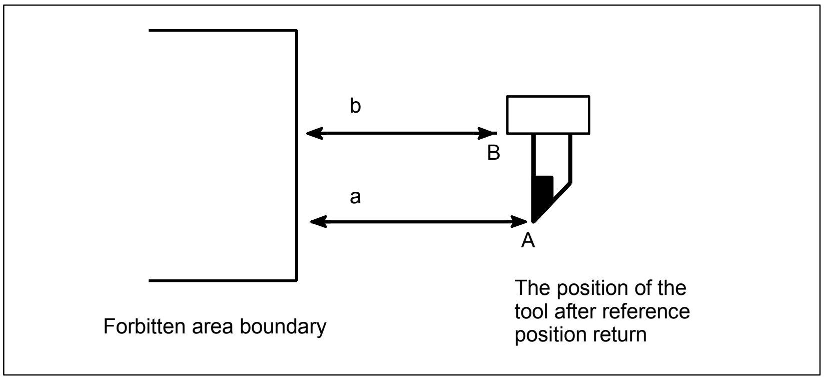


Fig. 6.3 (d) Setting the forbidden area

- Forbidden area overlapping

Area can be set in piles.

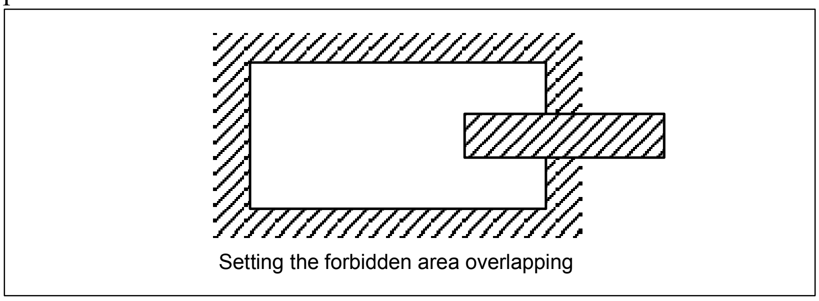


Fig. 6.3 (e) Setting the forbidden area overlapping

Unnecessary limits should be set beyond the machine stroke.

- Condition under which each check is enabled

Each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed.
After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke check 2).

- Releasing the alarms

If the enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.
When bit 4 (OF1) of parameter No.1301 is 1, if the axis moves within the movable area after an alarm occurs in stored stroke check 1, the OT alarm is cleared without a reset (automatic clearance function).

> **NOTE**
> In the following cases, the automatic clearance function is disabled. To clear an alarm, make a reset.
> 1 An alarm is set to occur before the stored stroke limit is exceeded (bit 7 (BFA) of parameter No. 1300 is 1).
> 2 Another over travel alarm (such as stored stroke check 2/3 and interference check) occurs.

- Change from G23 to G22 in a forbidden area

When G23 is switched to G22 in the forbidden area, the following results.
<1> When the forbidden area is inside, an alarm is informed in the next move.
<2> When the forbidden area is outside, an alarm is informed immediately.

- Timing for displaying an alarm

In stored stroke check 1/2/3, parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

Alarm

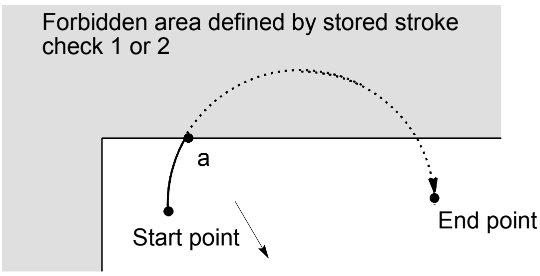
Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	A movement in the positive direction exceeded stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	A movement in the negative direction exceeded stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	A movement in the positive direction exceeded stored stroke check 2.
OT0503	- OVERTRAVEL (SOFT 2)	A movement in the negative direction exceeded stored stroke check 2.
OT0504	+ OVERTRAVEL (SOFT 3)	A movement in the positive direction exceeded stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	A movement in the negative direction exceeded stored stroke check 3.

6.4 STROKE LIMIT CHECK BEFORE MOVE

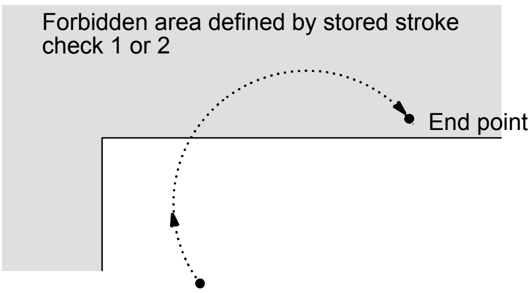
During automatic operation, before the movement specified by a given block is started, whether the tool enters the forbidden area defined by stored stroke check 1, 2, or 3 is checked by determining the position of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the forbidden 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.

> ⚠ **WARNING**
> Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a forbidden area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the forbidden area defined by stored stroke check 1, 2, or 3, an alarm is issued. (See the examples below.)

Example 1)

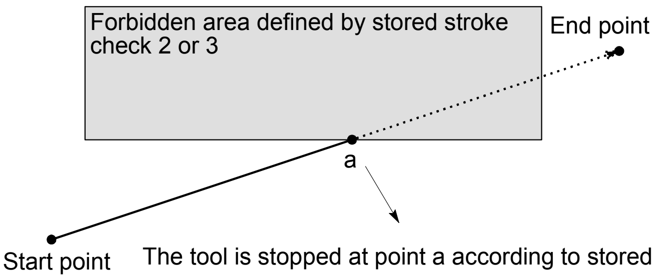


The tool is stopped at point a according to stored stroke check 1 or 2.

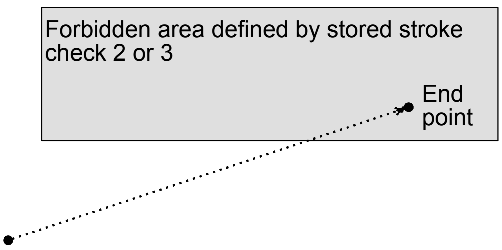


Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

Example 2)



The tool is stopped at point a according to stored stroke check 2 or 3.



Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

Explanation

When a stroke limit check before moving is performed, whether to check the movement performed by a G31 (skip) block and G37 (automatic tool length measurement (M series) or automatic tool compensation (T series)) block can be determined using (parameter NPC (No. 1301#2)).

Limitation

- Machine lock

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- G23

When stored stroke check 2 is disabled (G23 mode), no check is made to determine whether the tool enters the forbidden area defined by stored stroke check 2.

- Program restart

When a program is restarted, an alarm is issued if the restart position is within a forbidden area.

- Manual intervention following a feed hold stop

When the execution of a block is restarted after manual intervention following a feed hold stop, no alarm is issued even if the end point following a manual intervention is within a forbidden area.

- A block consisting of multiple operations

If a block consisting of multiple operations (such as a canned cycle and automatic reference position return) is executed, an alarm is issued at the start point of any operation whose end point falls within a inhibited area.

- Cylindrical interpolation mode

In cylindrical interpolation mode, no check is made.

T

- Polar coordinate interpolation mode

In polar coordinate interpolation mode, no check is made.

- PMC axis control

No check is made for a movement based on PMC axis control.

Alarm

Table 6.4 (a) Alarm

Number	Message	Description
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point was found in the + side stroke limit prohibition area during a stroke check before movement. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point was found in the - side stroke limit prohibition area during a stroke check before movement. Modify the program.

6.5 WRONG OPERATION PREVENTION FUNCTIONS

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 operation confirmation functions described below are meant to prevent the operator from performing any unintended operation (hereinafter referred to as an improper 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

6.5.1 Functions that are Used When Data is Set

The following functions are provided to prevent improper operations when data is set.
- Input data range check
- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process

Set these functions on the operation confirmation function setting screen. For the input data range check, set a valid input data range, e.g. the upper and lower limits, for each input screen. For the other functions, specify whether to enable or disable them.
For information about how to display the individual setting screens, how to manipulate them, and other details, see the item “Operation confirmation setting screen” that describes the operation procedures.

6.5.1.1 Input data range check

This function allows an effective data range to be set and checks whether the input data is within the set range.

Input data range check

Explanation

- Outline of the input data range check

This function allows an effective data range to be set for the data of each input screen listed later and checks whether the input data is within the set range. If the input data is out of the effective data range, the warning message "DATA IS OUT OF RANGE" is displayed and the data is rejected.

For example, assume that the effective data range for a certain tool offset number is set to -200. to 200, and that you are going to input 100.[INPUT]. Even if you inadvertently press the 0 key one more time, resulting in 1000.[INPUT], the input of 1000. is not accepted.
The function detects a setting mistake and prevents the program from running with invalid data.

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- Settings

To enable this function, set an effective data range for each input screen on the operation confirmation function setting screen. For information about how to display the individual setting screens, how to set data ranges, and other details, see the items that describe the setting of the data ranges.
If the set data range is invalid, no data input is accepted. Correct the data range setting, and then input data.

- Disabling the function

The input data range check is disabled if you make any of the following settings on the operation confirmation function setting screen.

- Both the upper and lower limit values for the tool offset number or workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

- Messages displayed during the input data range check

When the cursor moves into an input field of an input screen, one of the following messages and warning messages is displayed. No message is displayed when the input data range check is disabled.

When the set effective data range is valid

Message list 1		
Input data status	**Message**	**Color**
The data in the input field is within the range.	Input range xxx - xxx	Black
The data in the input field is out of the range.	Input range xxx - xxx	Red

xxx: Upper and lower limit values

When the set effective data range is invalid

Message list 2		
Range check status	**Message**	**Color**
Tool offset number overlap	NG SETTING (OFFSET NUM OVERLAP)	Red
Workpiece coordinate system overlap	NG SETTING (WORK COORD VAL OVERLAP)	Red
Invalid upper and lower limit values	NG SETTING (U-LMT AND L-LMT ILLEGAL)	Red

The message "NG SETTING (U-LMT AND L-LMT ILLEGAL)" is displayed in the following cases:

- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

- Range check for data changed by G10 or system variable

If the data changed by G10 or system variable is out of the effective data range, the alarm PS0334 "OFFSET DATA OUT OF RANGE" is displayed.

6.5.1.2 Confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key.

Confirmation of incremental input

Explanation

- Outline of the confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key in any of the input screens listed below. It lets you confirm whether you really want to change data or not before making that change.

For example, when you set 5.[+INPUT] for 10., the message "15. INPUT OK?" is displayed.
The function prevents improper absolute or incremental input operations.

> **NOTE**
> This function cannot be used to input two or more values consecutively by delimiting them by semicolons (;).

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset
- Settings
- Parameter
- Pitch error compensation

T

- Workpiece shift
- Y-axis tool offset
- Chuck tail stock barrier

- Settings

In the operation confirmation function setting screen, check or uncheck the "INCREMENTAL INPUT" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

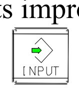
6.5.1.3 Prohibition of the absolute input by the soft key

This function prohibits the absolute input using the [INPUT] soft key.

Prohibition of the absolute input by the soft key

Explanation

- Outline of the prohibition of the absolute input by the soft key

This function prohibits the absolute input by the [INPUT] soft key in the input screens listed later.
It prevents improper absolute or incremental input operations by requiring that the absolute input be made using the 

MDI key and that the incremental input be made using the [+INPUT] soft key.

- **Input screens for which this function is effective**
- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- **Settings**

In the operation confirmation function setting screen, check or uncheck the "DISABLED SOFTKEY[INPUT] IN" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.1.4 Confirmation of the deletion of the program

This function displays the confirmation message "DELETE PROGRAM ?" when you attempt to delete the program.

Confirmation of the deletion of the program

Explanation

- **Outline of the confirmation of the deletion of the program**

When you attempt to delete the program, this function displays the confirmation message "DELETE PROGRAM (program name)?" It lets you confirm whether you really want to delete the program or not before executing the deletion.
The function prevents the program from being deleted due to an improper operation.

- **Settings**

In the operation confirmation function setting screen, check or uncheck the "PROGRAM DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.1.5 Confirmation of the deletion of all data

This function displays the confirmation message "DELETE ALL DATA?" when you attempt to delete all data.

Confirmation of the deletion of all data

Explanation

- **Outline of the confirmation of the deletion of all data**

When you attempt to delete all data on the input screen described later, this function displays the confirmation message "DELETE ALL DATA?". It lets you confirm whether you really want to delete all data or not before executing the deletion.
The function prevents all data from being deleted due to an improper operation.

- **Input screens for which this function is effective**
- Tool compensation

T

- Y-axis tool offset

- Settings

In the operation confirmation function setting screen, check or uncheck the "ALL DATA DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.1.6 Confirmation of a data update during the data setting process

This function displays the [CAN] and [EXEC] soft keys for confirmation when you attempt to update the data of an input screen during the data setting process.

Confirmation of a data update during the data setting process

Explanation

- Outline of the confirmation of a data update during the data setting process

When you input data in input screen during the data setting process, this function displays the [CAN] and [EXEC] soft keys for confirmation. It lets you confirm whether you really want to update the data or not before executing the update.
The function prevents set values from being lost due to an improper operation.

If you input data using the [+INPUT] soft key when the confirmation of incremental input is enabled, a message is displayed to confirm the incremental input.

- Settings

In the operation confirmation function setting screen, check or uncheck the "INPUT IN SETTING" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.2 Functions that are Used when the Program is Executed

Overview

The following functions are provided to prevent improper operations when the program is executed.

- Display of updated modal information
- Start check signal
- Axis status display
- Confirmation of the start from a middle block
- Data range check
- Maximum incremental value check
- Warning indication during a reset in program operation

Enable or disable each of the functions on the wrong operation prevention function setting screen.
To enable or disable "Warning indication during a reset in program operation", set bit 0 (MDW) of parameter No. 10334 instead of the wrong operation prevention function setting screen.
For information about how to display the setting screen, how to manipulate it, and other details, see the item “Operation confirmation setting screen” that describes the operation procedures.

6.5.2.1 Display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.

Display of updated modal information

Explanation

- Outline of the display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.
For example, when a absolute command has been changed to an incremental command or when the workpiece coordinate system has been initialized by RESET, the function displays the changed part of the data in an easy-to-recognize manner, in order to prevent improper operations during the execution of the program.

- Settings

In the operation confirmation function setting screen, check or uncheck the "UPDATE MODAL HIGHLIGHT DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.2.2 Start check signal

This function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt before the program is executed.

Start check signal

Explanation

- Outline of the start check signal

When a cycle start is made with the start check signal STCHK <G0408.0> set to 1, the function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt. Making the cycle start again resumes the execution of the program.
The function lets you check the status of the block before executing it, thus helping to prevent improper operations at the time of execution.
Using this function in combination with the updated modal information display function described in the preceding subsection makes it easier to check the status of the block to be executed.

- Settings

This function does not require any setting on the operation confirmation function setting screen.

6.5.2.3 Axis status display

This function displays the axis status to the left of the axis name in the coordinate display screen.

Axis status display

Explanation

- Outline of the axis status display

This function displays the axis status to the left of the axis name in the display of the machine coordinates, absolute coordinates, relative coordinates, and remaining travel amounts.
For example, when the mirror image function is enabled for the X1 axis, the absolute coordinates are displayed as follows.

```
ABSOLUTE
M  X1   10.000
   Y1   10.000
   Z1    0.000
```

By displaying the axis status as shown above, the function prevents improper operations at the time of execution.

- Axis status indication

The axis status is indicated as follows. These indications are listed in order of priority.
AXIS DETACH : D
INTERLOCK : I
MACHINE LOCK : L
SERVO OFF : S
Move command in progress or not in-position : *
MIRROR IMAGE : M

- Settings

In the operation confirmation function setting screen, check or uncheck the "AXIS STATUS DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

NOTE
For the 8.4-inch display unit, there is no indication on the program check screen.

6.5.2.4 Confirmation of the start from a middle block

This function displays a confirmation message when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program.

Confirmation of the start from a middle block

Explanation

- Outline of the confirmation of the start from a middle block

This function displays the confirmation message "START FROM MIDDLE OF PROG (START/RESET)" when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program. It lets you confirm whether you really want to start execution from that block or not before executing the program.
The function prevents you from inadvertently making a cycle start from a block in the middle of the program.

- Settings

In the operation confirmation function setting screen, check or uncheck the "START FROM MIDDLE OF PROGRAM" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.2.5 Data range check

This function lets you set an effective data range and check whether the data to be used for execution is within the set range.

Data range check

Explanation

- Outline of the data range check

This function lets you set an effective data range for each data item listed later and check whether the data to be used for execution is within the set range. If the data is out of the effective range, the alarm PS0334 "OFFSET DATA OUT OF EFFECTIVE RANGE" is displayed.
The function detects data setting mistakes and prevents the program from running with invalid data.

- Data for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

> **NOTE**
> To use this function, you need to set each effective data range correctly. For information about how to set the data ranges, see the item "Effective value range for each data".

6.5.2.6 Maximum incremental value check

This function checks the maximum incremental value specified for each axis by the NC command.

Maximum incremental value check

Explanation

- Outline of the maximum incremental value check

When the maximum incremental value is specified by the NC command described later, make sure that the absolute value of the travel distance by the incremental command does not exceed the specified value. If the specified value is exceeded, the alarm PS0337 "EXCESS MAXIMUM INCREMENTAL VALUE" is displayed.
A maximum incremental value can be specified on a per-axis basis and remains effective until 0 is set or the value is reset.
For example, when advanced preview control (T series) / AI advanced preview control (M series)/AI contour control (M series) is used, the function checks whether the amount of movement between blocks is kept to the specified value or less. Through this process, it detects erroneous program settings and prevents the program from running with invalid data.

- Format

The format of the NC command used to specify the maximum incremental value is as follows.
G91.1 IP_ ;
 IP_ ; Maximum incremental value

 To cancel the maximum incremental value check, set 0.

6.5.2.7 Warning display during a reset in program operation

When bit 6 (CLR) of parameter No. 3402 is 0, if a reset occurs during block execution in program operation, modal information returns to the state before block execution.
This function display a warning to notifies the operator that modal information is not updated by information of the interrupted block.

Warning display during a reset in program operation

Explanation

- Overview of warning display during a reset in program operation

If a reset occurs during program operation, a warning saying "MODAL DATA IS CHANGED BY BLOCK STOP" is issued. To enable or disable the warning, set bit 0 (MDW) of parameter No. 10334.

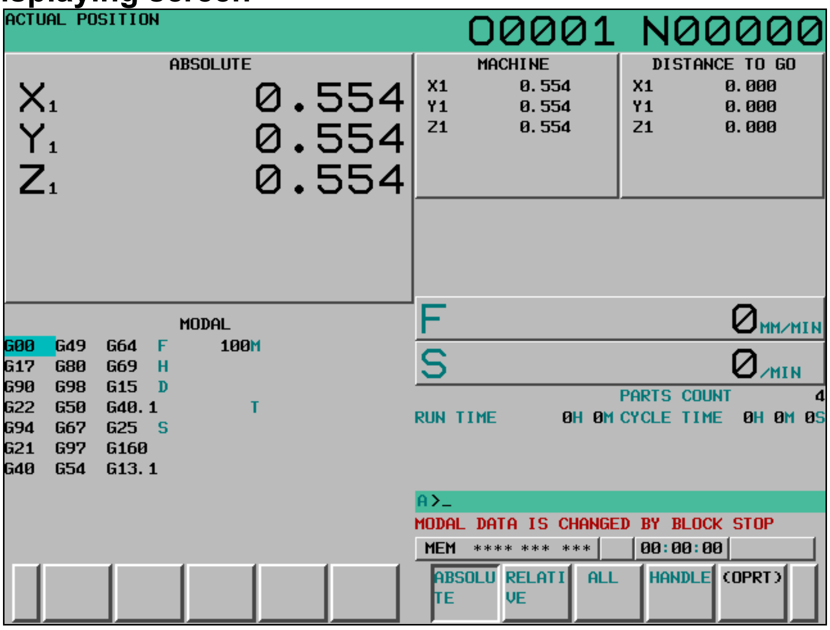
- Warning occurrence condition

- When a reset occurs during program operation, if address G, F, H, D, T, S, M, or B (second auxiliary function) was changed

- Warning clear condition

- When a reset is issued
- When the <CAN> key is pressed

- Warning displaying screen



> **NOTE**
> There is no function for acquiring or displaying a warning message in C Language Executor. Accordingly, this warning cannot be displayed on a screen created by the machine tool builder.

6.5.3 Setting Screen

This section describes how to display the operation confirmation function setting screen and how to set the individual data items on this screen.
The operation confirmation function setting screen allows you to set the following items:

- Enabling or disabling each operation confirmation function
- Effective value range for the tool offset
- Effective value range for the workpiece origin offset

T

- Effective value range for the Y-axis tool offset
- Effective value range for the work shift

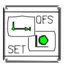
6.5.3.1 Operation confirmation function setting screen

This screen displays the enable/disable setting status of the following operation confirmation functions and lets you change their settings. (Hereinafter, the screen is referred to as the operation confirmation function setting screen.)

- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process
- Display of updated modal information
- Axis status display
- Confirmation of the start from a middle block

Displaying and setting the operation confirmation function setting screen

Procedure

1 Press the  function key.
2 Press the continuous menu key  at the right edge of the screen several times until the [GUARD] soft key is displayed.
3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).
4 If any screen other than the operation confirmation function setting screen is displayed, click the [GUARD] soft key. The operation confirmation function setting screen is displayed.

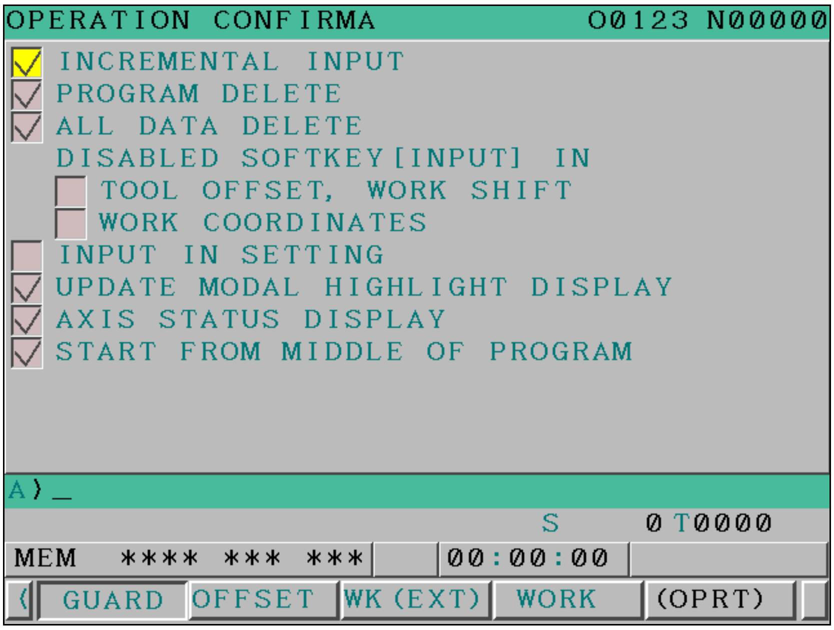


Fig. 6.5.3.1 (a) Operation confirmation function setting screen

5 In the operation confirmation function setting screen, the check box of each enabled function is checked (✓). Move the cursor to the check box of the item you want to set, by pressing the ↑, ↓, ←, and → keys.

6 Click the operation soft key [ON:1] or [OFF:0]. When you click the [ON:1] soft key, a check mark (✓) appears in the corresponding check box, indicating that the function is enabled. When you click the [OFF:0] soft key, the check mark disappears from the check box, indicating that the function is disabled.

Explanation

- Items to be set

The following table shows what is displayed for each item to be set and the corresponding functions.

Displayed item	Corresponding function
INCREMENTAL INPUT	Confirmation of incremental input
DISABLED SOFTKEY[INPUT] IN TOOL OFFSET, WORK SHIFT	Prohibition of the absolute input by the soft key (tool offset, Y-axis tool offset (T series), and work shift (T series))
DISABLED SOFTKEY[INPUT] IN WORK COORDINATES	Prohibition of the absolute input by the soft key (workpiece origin offset)
PROGRAM DELETE	Confirmation of the deletion of the program
ALL DATA DELETE	Confirmation of the deletion of all data
INPUT IN SETTING	Confirmation of a data update during the data setting process
UPDATE MODAL HIGHLIGHT DISPLAY	Display of updated modal information
AXIS STATUS DISPLAY	Axis status display
START FROM MIDDLE OF PROGRAM	Confirmation of the start from a middle block

6.5.3.2 Tool offset range setting screen

This screen displays the setting status of tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the tool offset range setting screen.)
Up to 20 pairs of numbers can be specified to identify tool offset number ranges, and an effective offset value range can be defined for each of these 20 pairs.

Displaying and setting the tool offset range setting screen

Procedure

1 Press the function key.

2 Press the continuous menu key at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the tool offset range setting screen is displayed, click the [OFFSET] soft key. The tool offset range setting screen is displayed. What is displayed in this screen differs depending on the system configuration described later.

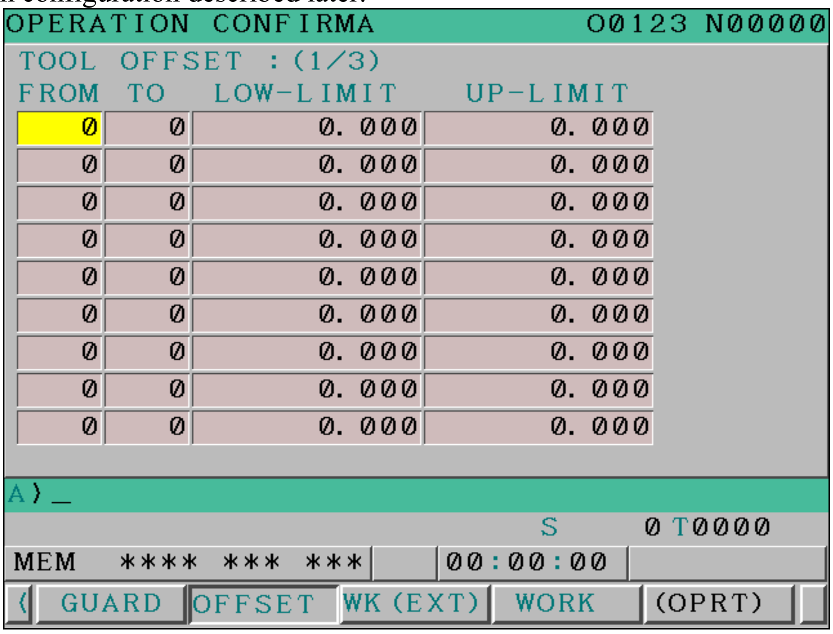


Fig. 6.5.3.2 (a) Tool offset range setting screen

5 Move the cursor to the item you want to set, by using the and keys, , , , and keys, or the [SWITCH] soft key.

6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- Control type

The setting depend on the control type shown below.

M

- Tool offset memory A (bit 6 (NGW) of parameter No. 8136 is 1)
- Tool offset memory C (bit 6 (NGW) of parameter No. 8136 is 0)

T

- Without geometry and wear offset (bit 6 (NGW) of parameter No. 8136 is 1)
- With geometry and wear offset bit 6 (NGW) of parameter No. 8136 is 0)

M

- Settings with tool offset memory A (bit 6 (NGW) of parameter No. 8136 is 1)

With tool offset memory A, an effective data range is specified using the following four items.

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified tool offset number range.
	UP-LIMIT	

- Settings with tool offset memory C (bit 6 (NGW) of parameter No. 8136 is 0)

With tool offset memory C, an effective data range is specified using the following ten items.

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for geometry length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry radius in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for wear length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this configuration, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

T

- Settings without geometry and wear offset (bit 6 (NGW) of parameter No. 8136 is 1)

Without geometry/wear offset, an effective data range is specified using the following eight items.

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
X	LOW-LIMIT	Specify a valid tool offset value range for the X-axis in connection with a specified tool offset number range.
	UP-LIMIT	
Z	LOW-LIMIT	Specify a valid tool offset value range for the Z-axis in connection with a specified tool offset number range.
	UP-LIMIT	

Displayed item		What to set
RADIUS	LOW-LIMIT	Specify a valid tool offset value range for tool-nose radius in connection with a specified tool offset number range.
	UP-LIMIT	

> **NOTE**
> The radius items are not displayed when tool-nose radius compensation is not provided (bit 7 (NCR) of parameter No. 8136 is 1).

- Settings with geometry and wear offset (bit 6 (NGW) of parameter No. 8136 is 0)

With geometry/wear offset, an effective data range is specified using the following 14 items.

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	X	LOW-LIMIT	Specify a valid tool offset value range for the geometry X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the geometry Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	X	LOW-LIMIT	Specify a valid tool offset value range for the wear X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the wear Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this system, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

> **NOTE**
> The radius items are not displayed when tool-nose radius compensation is not provided (bit 7 (NCR) of parameter No. 8136 is 1).

- Example of setting an input data range

For example, suppose that the following values are set with offset memory A (M series).

FROM : TO LOW-LIMIT : UP-LIMIT
1 : 20 0.000 : 100.000

In this case, the tool offset input screen accepts only offset values from 0.000 to 100.000 for offset numbers 1 to 20.
If you attempt to input any other value, the warning message "DATA IS OUT OF RANGE" is displayed.

6.5.3.3 Workpiece origin offset range setting screen

This screen displays the setting status of workpiece origin offset and external workpiece origin offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece origin offset range setting screen.)

Up to six pairs of values can be specified to identify workpiece coordinate ranges for the workpiece origin offset, and an effective offset value range can be defined for each of the axes of these six pairs.
As for the external workpiece origin offset, an effective offset value range can be specified for each axis.

Displaying and setting the workpiece origin offset range setting screen

Procedure

1 Press the function key.

2 Press the continuous menu key at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the workpiece origin offset range setting screen is displayed, click the [WORK] soft key. The workpiece origin offset range setting screen is displayed.

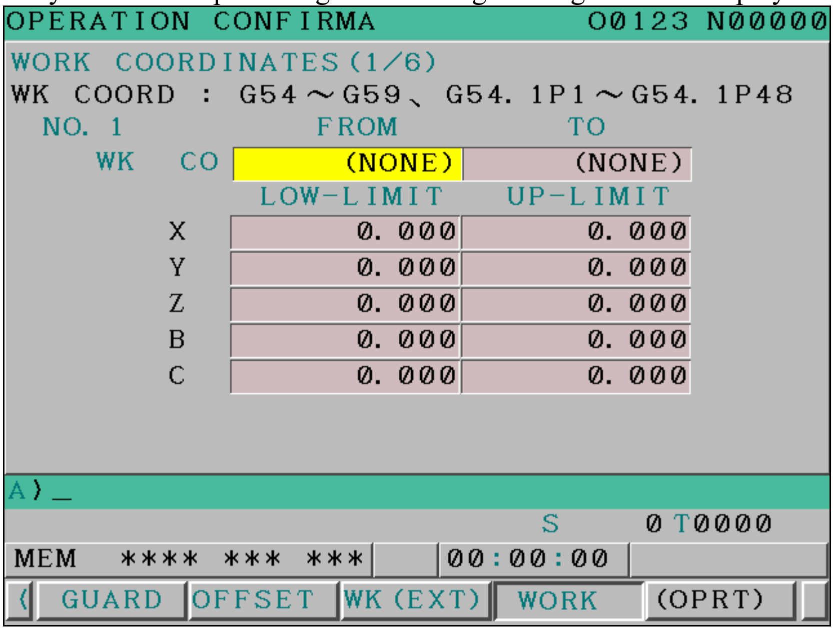


Fig. 6.5.3.3 (a) Workpiece origin offset range setting screen

5 Move the cursor to the item you want to set, by using the and keys, , , , and keys, or the [SWITCH] soft key.

6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a workpiece coordinate overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., an invalid workpiece coordinate system is set).
- The upper limit value is set for the workpiece coordinate system when 0 is set for the lower limit value.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

Explanation

- What to set for the workpiece origin offset

For the workpiece origin offset, an effective data range is specified using the following four items.

Displayed item		What to set
RANGE	FROM	Specify a workpiece coordinate system range.
	TO	
AXIS NAME	LOW-LIMIT	Specify a valid offset value range in connection with a specified workpiece coordinate system range.
	UP-LIMIT	

- What to set for the external workpiece origin offset

For the external workpiece origin offset, an effective data range is specified using the following two items.

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid external workpiece origin offset value range on each axis.
	UP-LIMIT	

6.5.3.4 Y-axis tool offset range setting screen

T

In the case of a T series system, this screen displays the setting status of Y-axis tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the Y-axis tool offset range setting screen.)
Up to four pairs of values can be specified to identify Y-axis tool offset number ranges, and an effective offset value range can be defined for each of these four pairs.

Displaying and setting the Y-axis tool offset range setting screen

Procedure

1 Press the OFS/SET function key.
2 Press the continuous menu key ▷ at the right edge of the screen several times until the [GUARD] soft key is displayed.
3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).
4 If any screen other than the Y-axis tool offset range setting screen is displayed, click the [OFST.2] soft key. The Y-axis tool offset range setting screen is displayed. What is displayed in this screen differs depending on such factors as whether tool geometry/wear offsets are present.

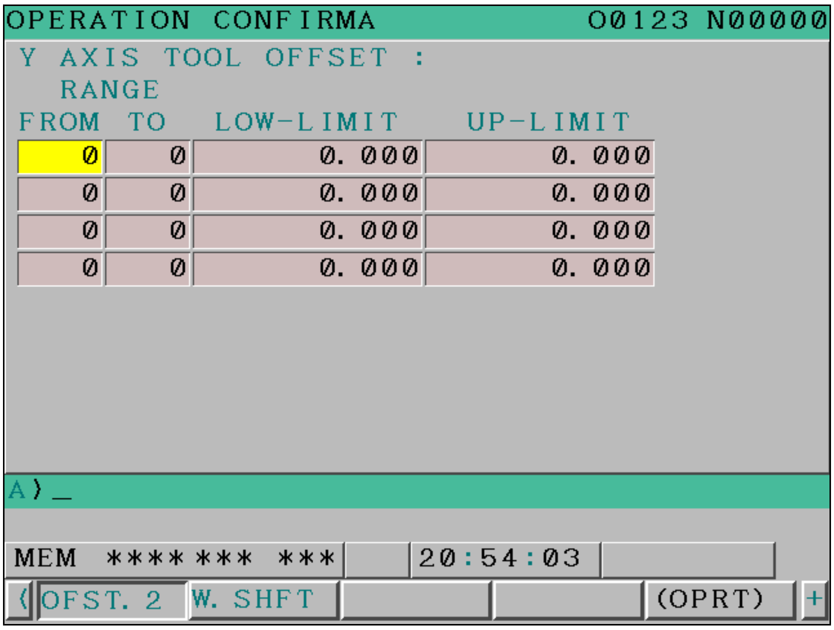


Fig. 6.5.3.4 (a) Y-axis tool offset range setting screen

5 Move the cursor to the item you want to set, by using the PAGE ↑ and PAGE ↓ keys, ↑, ↓, ←, and → keys, or the [SWITCH] soft key.

6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- Settings without geometry and wear offset (bit 6 (NGW) of parameter No. 8136 is 1)

Without geometry/wear offsets, an effective data range is specified using the following four items.

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

- Settings with geometry and wear offset (bit 6 (NGW) of parameter No. 8136 is 0)

With geometry/wear offsets, an effective data range is specified using the following six items.

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	

Displayed item		What to set
GEOM	LOW-LIMIT	Specify a valid tool offset value range for geometry in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	
WEAR	LOW-LIMIT	Specify a valid tool offset value range for wear in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

6.5.3.5 Workpiece shift range setting screen

T

In the case of a T series system, this screen displays the setting status of shift effective data ranges of workpiece shift and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece shift range setting screen.)
An workpiece shift value range can be specified for each axis.

Displaying and setting workpiece shift input ranges

Procedure

1 Press the function key.
2 Press the continuous menu key at the right edge of the screen several times until the [GUARD] soft key is displayed.
3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).
4 If any screen other than the workpiece shift range setting screen is displayed, click the [WORK SHIFT] soft key. The workpiece shift range setting screen is displayed.

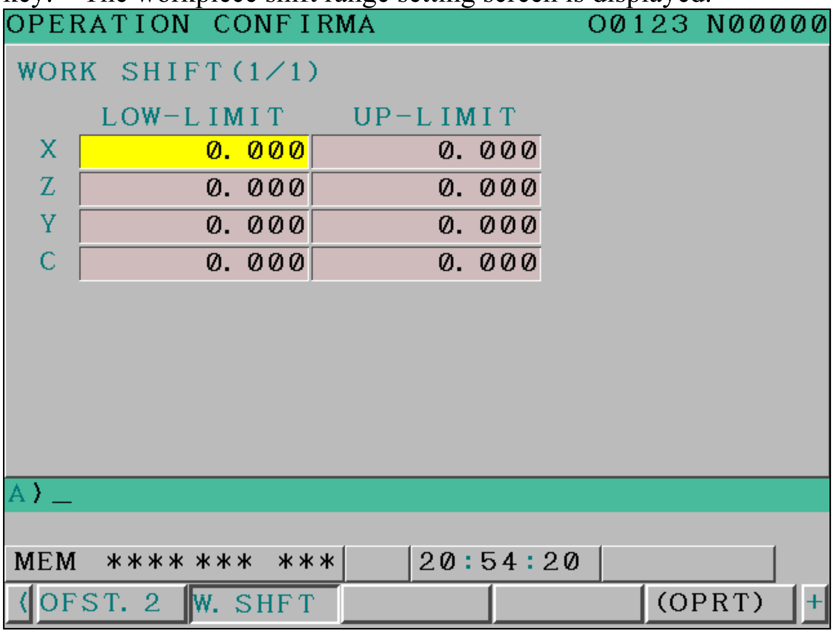


Fig. 6.5.3.5 (a) Workpiece shift range setting screen

5 Move the cursor to the item you want to set, by using the and keys, , , , and keys, or the [SWITCH] soft key.
6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- The upper and lower limit values are reversed.

Also, the input data range check is invalidated in the following cases.

- The upper and lower workpiece shift limit values are identical.

Explanation

- What to set for the workpiece shift

For the workpiece shift, an effective data range is specified using the following two items.

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid workpiece shift value range on each axis.
	UP-LIMIT	

7 ALARM AND SELF-DIAGNOSIS FUNCTIONS

When an alarm occurs, the corresponding alarm screen appears to indicate the cause of the alarm. The causes of alarms are classified by error codes and number. Up to 50 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self-diagnosis function.

7.1 ALARM DISPLAY

Explanation

- Alarm screen

If an alarm occurs, the alarm screen (error code and number) appears to indicate the cause. Alarms are classified by an error code and number.

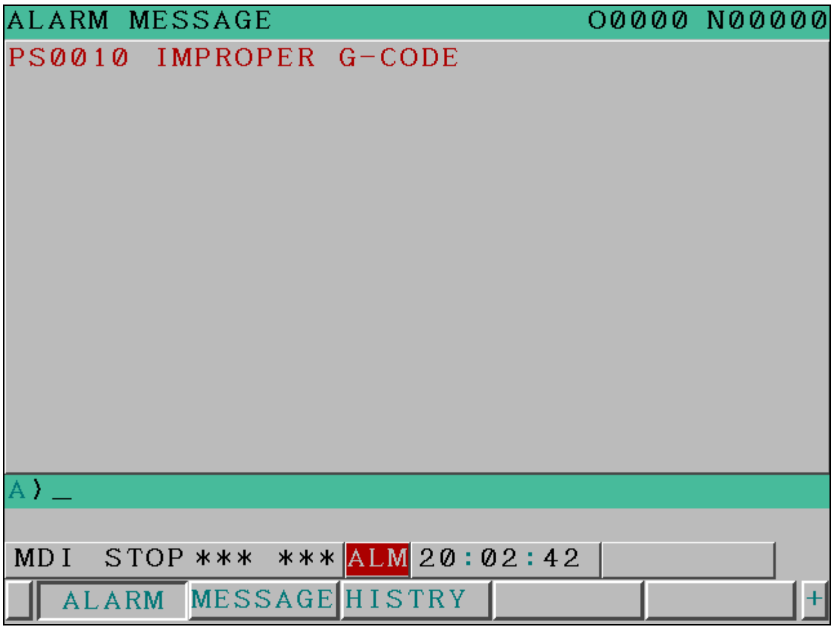


Fig. 7.1 (a) Alarm screen (example for the 8.4-inch display unit)

Display wrapping

If an alarm message does not fit one line, the display is wrapped and the rest of the message begin at the start position on the next line.

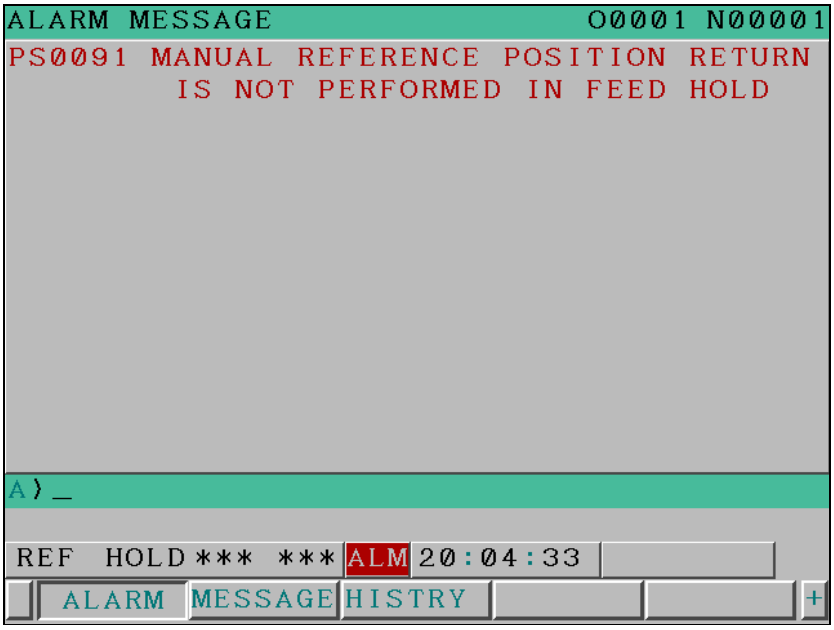


Fig. 7.1(b) Display wrapping (example for the 8.4-inch display unit)

7.1.1 Operation

- How to display the alarm screen

In some cases, no switching occurs to the alarm screen, and “ALM” is displayed on the bottom of the current screen (for example, if bit 7 (NPA) of parameter No. 3111 = 1).

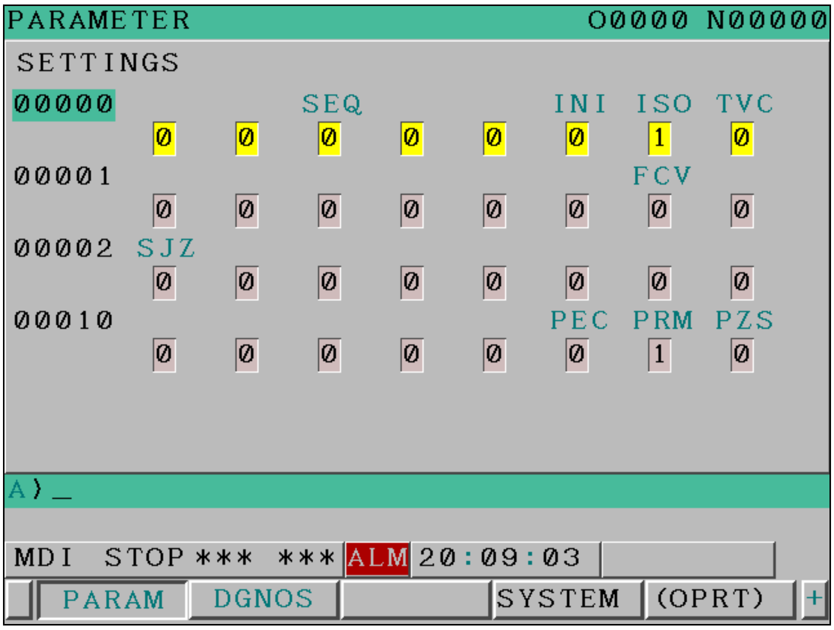


Fig. 7.1 (c) Parameter screen (example for the 8.4-inch display unit)

In this case, display the alarm screen by following the steps below.

1 Press the function key 

.

2 Press the soft key [ALARM].

3 The page change key can be used to switch between pages.

- Releasing alarm

The cause of an alarm can be determined from the error code, number, and associated message. To release the alarm, generally correct the cause, then press the reset key.

- Error code and number

The type of an alarm is indicated by an error code and number.
Example: PS0010, SV0004, etc.
For details, see Appendix G, "ALARMS".

Screen scrolling

If alarm information does not fit one screen, the page keys (PageDown and PageUp) can be used to scroll the alarm information screen by screen.

Line scrolling

If alarm information does not fit one screen, the cursor keys ↑ ↓ can be used to scroll the alarm information alarm by alarm.

> **NOTE**
> In simultaneous 2-path display, the alarm displays of both the paths are subjected to screen/line scrolling simultaneously.

7.1.2 Alarm Display in a 2-Path System

T

2-path concurrent display

Alarms for two paths are displayed concurrently in a 2-path system.
The path name is displayed on the first line of each screen.

Display order change

Parameter No. 13130 can be set to change the display order of two paths.

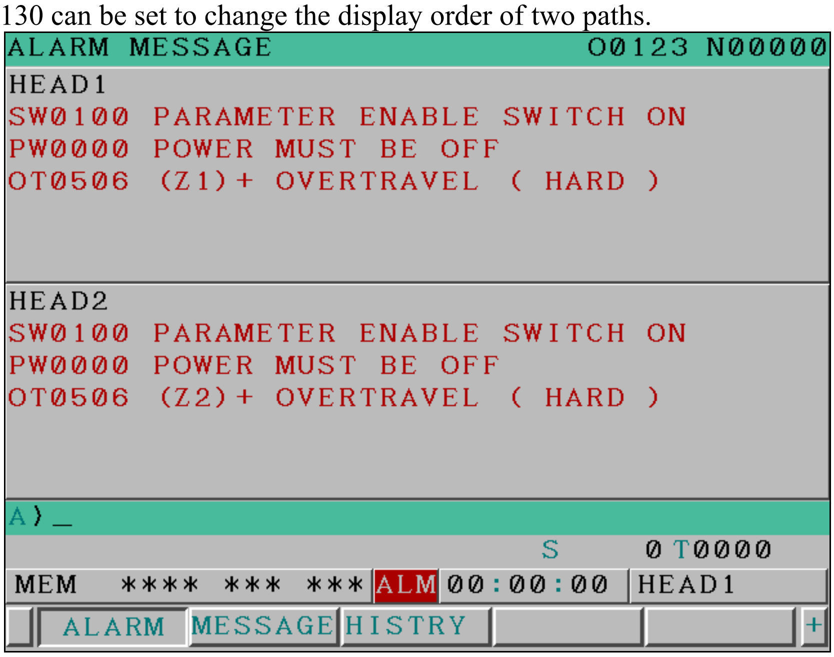


Fig. 7.1.2(a) 2-path display on the alarm display screen (8.4-inch display unit)

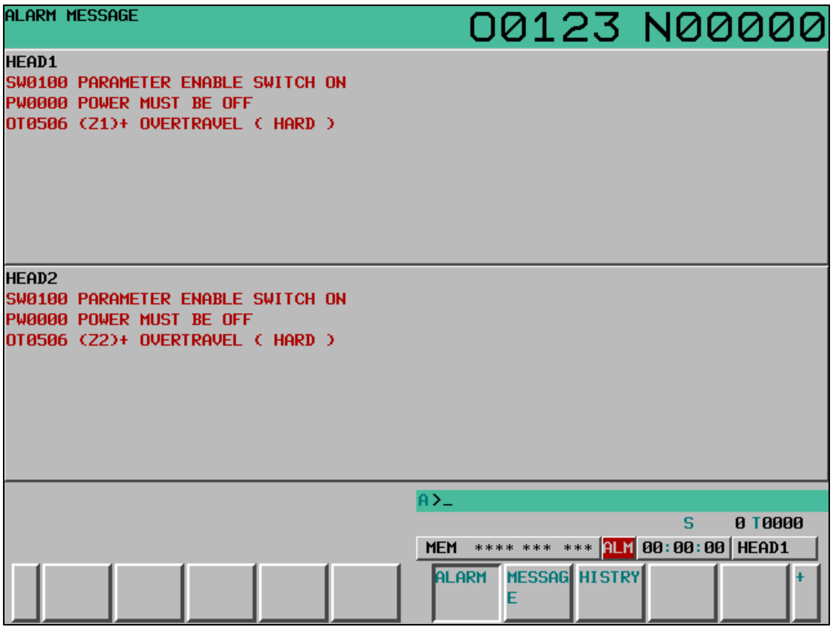


Fig. 7.1.2(b) 2-path display on the alarm display screen (10.4-inch display unit)

> **NOTE**
> If an arbitrary name is set (by parameters Nos. 3141 to 3147) for each path, the arbitrary name is displayed at the upper left of each split screen instead.

Single path display

For a 2-path system, if bit 2 of parameter No. 3193 is set to 1, it is possible to switch from 2-path concurrent display to single path display.

Alarms for the selected path are displayed in the full screen mode.

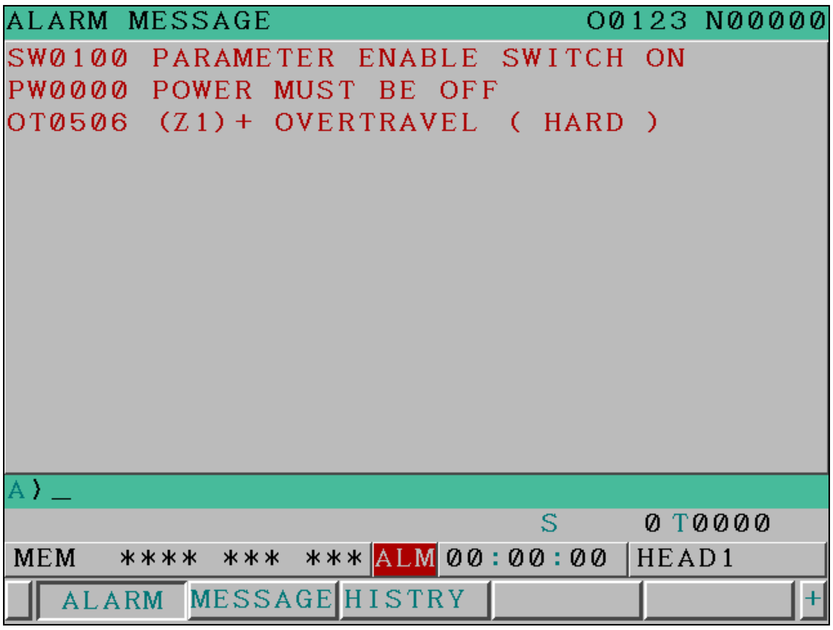


Fig. 7.1.2(c) Alarm screen (single path display for the 8.4-inch display unit)

7.2 ALARM HISTORY DISPLAY

Up to 50 alarms issued by the CNC including the latest alarm are stored and displayed on the screen. The display procedure is explained below.

Alarm history display

Procedure

1 Press the function key 

.

2 Press the soft key [HISTRY].
An alarm history is displayed.
The following information is displayed:
<1> Date and time of alarm issuance
<2> Alarm type
<3> Alarm number
<4> Alarm message (sometimes not displayed depending on the alarm)
<5> Number of recorded alarms

3 You can change pages by using the page key.

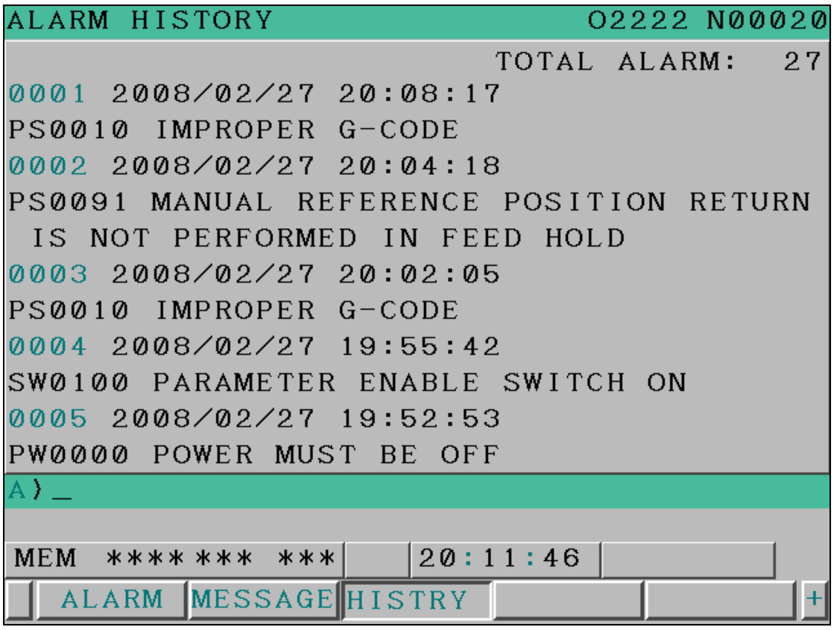


Fig. 7.2 (a) Alarm history screen (example for the 8.4-inch display unit)

T

For a 2-path system, alarms that were issued on both paths are displayed on one screen, regardless of the selected path.
Each history data is preceded by the path on which the alarm was issued.
A total of 50 alarms issued on both paths are recorded.

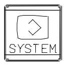
Fig. 7.2 (b) Alarm history screen (or a 2-path system, example of the 10.4-inch display unit)

7.3 CHECKING BY DIAGNOSTIC DISPLAY

The system may sometimes seem to be at a halt, although no alarm has occurred. In this case, the system may be performing some processing. Diagnostic display can be used to check the system status.

Procedure for Diagnostic display

Procedure

1 Press the function key 

.
2 Press the soft key [DGNOS].
3 The diagnosis screen has more than 1 pages. Select the screen by the following operation.
 (1) Change the page by the page change key.
 (2) Method by soft key
  - Key input the number of the diagnosis data to be displayed.
  - Press the soft key [NO.SRH].

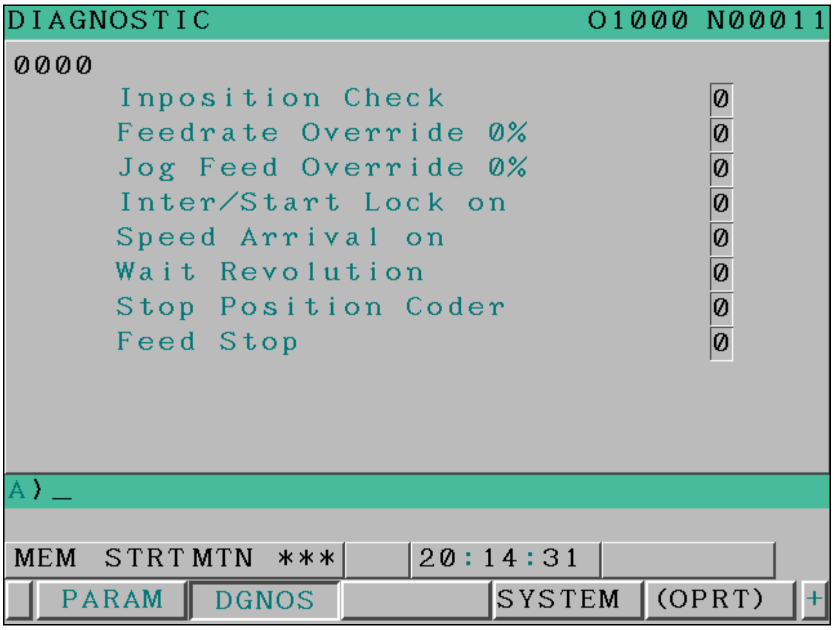


Fig. 7.3 (a) Diagnostic display (example for the 8.4-inch display unit)

7.4 RETURN FROM THE ALARM SCREEN

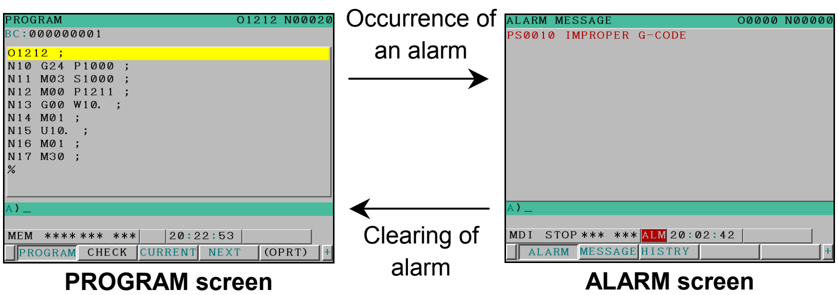
7.4.1 Return from the Alarm Screen

When alarms are cleared or function key  is pressed on the alarm screen, the screen displayed before the alarm screen appears.
To enable this function, set bit 4 (ADC) of parameter No. 11302 is set to 1.

Switching between screens when alarms are cleared

When all alarms are cleared on the alarm screen, the screen displayed before the alarm screen appears again.
When the alarm screen was displayed automatically due to occurrence of an alarm, the screen displayed immediately before the alarm appears again.
When the alarm screen was displayed by pressing function key  during occurrence of an alarm, the screen displayed immediately before the alarm appears again.

(Example)

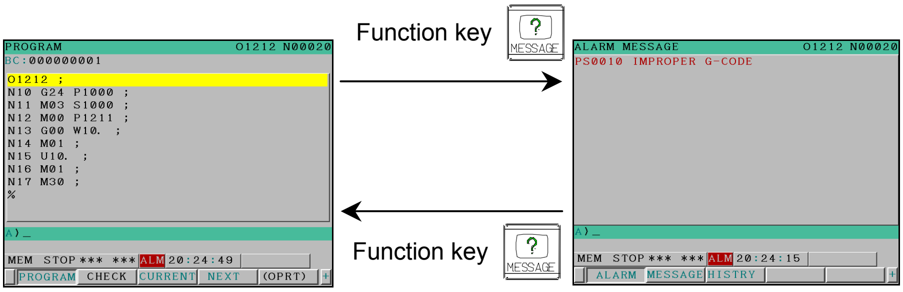


> **NOTE**
> Even if alarms are cleared when the alarm screen is not displayed, the current screen is not changed.

Switching between screens by the function key

When function key  is pressed on the alarm screen, the screen displayed before the alarm screen appears. Press function key  to switch to the alarm screen for checking for alarms and then press function key  to return to the previous screen.

(Example)



PROGRAM screen **ALARM screen**

If function key  is pressed when the alarm screen was displayed automatically due to occurrence of an alarm, the screen displayed before the alarm screen appears again.

Restrictions

- Switching to the interactive macro screen is not performed.
- The screens to which the alarm screen can be switched are only the screen selected by the chapter selection soft key.

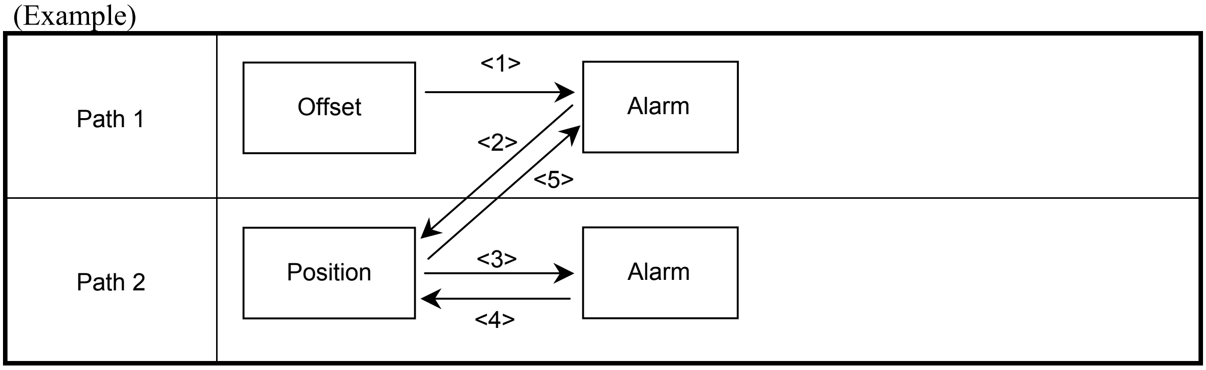
7.4.2 Relationship with Other Functions (For 2-Path Control)

T

Relationship between the screen switching function and a return from the alarm screen during switching between paths

(1) When bit 5 (PSC) of parameter No. 3208 is set to 0, if paths are switched by the path switching signal, the screen last selected in the path appears again.
At this time, even if a return from the alarm screen to the previous screen is performed in one path, a return is not performed in the another path and the alarm screen remains displayed.

(Example)

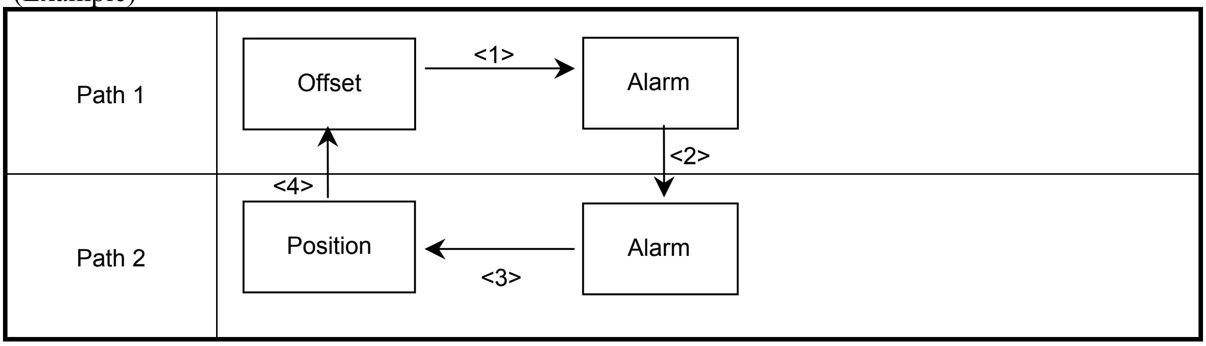


<1> When the message key is pressed on the offset screen of path 1, the alarm screen (path 1) appears.

<2> When switching to path 2 is performed on the alarm screen of path 1, the position screen of path 2 appears (when the screen last displayed in path 2 is the position screen).
<3> When the message key is pressed on the position screen of path 2, the alarm screen (path 2) appears.
<4> When the alarm is cleared or the message key is pressed on the alarm screen of path 2, a return to the position screen (path 2) is performed.
<5> When switching to path 1 is performed, the alarm screen (path 1) appears.

(2) When PSC of parameter No. 3208 is set to 1, if paths are switched by the path switching signal, the screen displayed immediately before the path switching appears again.
At this time, if a return from the alarm screen to the previous screen is performed in one path, the screen of the path in which a return was performed appears in the other path.

(Example)



<1> When the message key is pressed on the offset screen of path 1, the alarm screen (path 1) appears.
<2> When switching to path 2 is performed on the alarm screen of path 1, the alarm screen (path 2) appears.
<3> When the alarm is cleared on the alarm screen of path 2, the offset screen (path 2) appears.
<4> When switching to path 1 is performed, the offset screen of path 1 appears.

8 DATA INPUT/OUTPUT

Information stored in external I/O devices can be read into the CNC, and information can be written into external I/O devices.
External I/O devices include memory cards that can be mounted to the memory card interface located on the left side of the display unit and personal computers and data servers that can be connected via embedded Ethernet.
The following types of data can be input and output.

Data type	Default file name
Program	ALL-PROG.TXT
Offset data	TOOLOFST.TXT
Parameter	CNC-PARA.TXT
Pitch error compensation data	PITCH.TXT
Custom macro common variable	MACRO.TXT
Workpiece coordinate system data	EXT_WKZ.TXT
Operation history data	OPRT_HIS.TXT
Maintenance information	MAINTINF.TXT
Periodic maintenance data (periodic maintenance: status screen)	MAINTENA.TXT
System configuration data	SYS-CONF.TXT
PMC signal protect	DIDOENBL.TXT
Servo/spindle information	SV_SP_ID.TXT
Machine system name data (periodic maintenance: machine system screen)	MAINTEMC.TXT
Servo waveform diagnosis	WAVE-DGN.TXT
Tool geometry data (Interference check for each path) (T series)	TOOL-FRM.TXT
P code variable (macro executor)	PCODE.TXT

The above types of data can be input and output on the screens for displaying and setting those types of data.
If NC data such as programs and parameters is to be written to a memory card, and if a file with the same name already exists, it is possible to select whether to overwrite the existing file or cancel writing with appropriate operation.

The external I/O device set in parameter No. 0020 is selected. See the table below for details.

Correspondence between settings and input/output units	
Setting	**Description**
0,1	RS-232-C serial port 1
2	RS-232-C serial port 2
4	Memory card interface
5	Data server interface
9	Embedded Ethernet interface

⚠ **WARNING**

1. Always use ISO codes for input/output except when ASCII code data is to be input.
   ISO code input/output is enabled for memory cards and data servers by setting, respectively, bit 0 (ISO) of parameter No. 0139 and bit 0 (ISO) of parameter No. 0908 to “1”.
2. ASCII data input/output is risky because ASCII data does not contain parity information and therefore any data error cannot be detected.

> ⚠ **CAUTION**
> 1 This control unit supports the use of a Memory card as an input/output device. The Flash ATA card is available:
> See the order list for details of the supported Memory card types.
> 2 On a Memory card, only those files that are in the root directory can be accessed for display, reading, and writing. Those in subdirectories cannot be used.
> 3 The time required to read or write each data item varies depending on the Memory card type, the status of use, and other factors.
> 4 For flash ATA cards, only those recommended by FANUC are available.
> 5 When formatting a flash ATA card, use the quick formatting method, which clears the file allocation table and the directory information on the root directory. An unformatted flash ATA card needs to be formatted in FAT16 with a PC or the like. (A FAT32-formatted ATA card cannot be recognized.)

> **NOTE**
> External I/O devices can handle file names of up to 12 characters each.

8.1 OVERWRITING FILES ON A MEMORY CARD

Screen display

When an attempt is made to output NC data to a memory card, and if the specified file name or the default file name is the same as an existing file name on the memory card, a confirmation message "OVERWRITE?" appears.

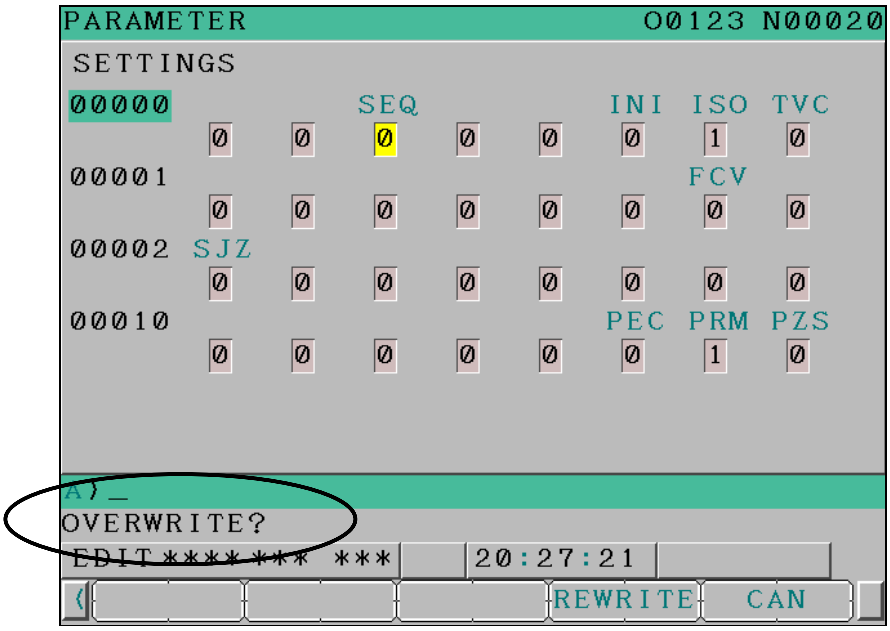


Fig. 8.1 (a) Screen display example

Procedure

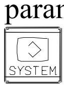
On the output screen for the desired function, perform the following operation.

1 Press the soft key [F OUTPUT].

2 Press the soft key [EXEC]. If a file with the same name does not exist on the memory card, the file is output in this step.

3 If a file with the same name exists on the memory card, soft keys [REWRITE] and [CAN] appear.
Pressing the soft key [REWRITE] causes the file to be overwritten.
Pressing the soft key [CAN] causes output to be canceled.

Example) Output from the parameter screen

1 Press the function key 

.
2 Press the soft key [PARAMETER].
3 Enter the EDIT mode or state emergency stop.
4 Press the soft key [(OPRT)].
5 For 8.4-inch display unit, press the continuous menu key .
6 Press the soft key [F OUTPUT]. The soft key display switches from the one in Fig. 8.1 (b) to the one in Fig. 8.1 (c).
7 If all parameters are to be output, press the soft key [ALL]. If only the parameters with nonzero values are to be output, press the soft key [NON-0]. The soft key displays change from those displayed in Fig. 8.1 (c) to those displayed in Fig. 8.1 (d).
8 Press the soft key [EXEC]. Because no file name is specified, the file is output with a file name of CNC-PARA.TXT, but if a file with the same name exists on the memory card, the soft key display switches from the one in Fig. 8.1 (d) to the one in Fig. 8.1 (e), with a confirmation message appearing.
If a file with the same name does not exist on the memory card, the file is output directly.
9 Pressing the soft key [REWRITE] causes the file to be overwritten.
Pressing the soft key [CAN] causes output to be canceled. If wishing to output the file after changing the file name, specify a file name after step 6, and perform step 7 again.

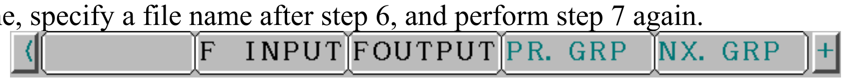


Fig. 8.1 (b) Soft key display before [F OUTPUT] is pressed

Fig. 8.1 (c) Soft key display after [F OUTPUT] is pressed

Fig. 8.1 (d) Soft key display after [ALL] or [NON-0] is pressed

Fig. 8.1 (e) Soft key display after [EXEC] is pressed

> ⚠ **CAUTION**
> If a file to be overwritten has the read-only attribute, the warning message "OVER WRITE FAILED" appears to cancel output even when the soft key [REWRITE] is pressed for that file.
> If a memory card is pulled out or inserted while a message for confirming overwriting is displayed, it is likely that the write operation may fail and, at worst, files on the memory card may be broken.

8.2 INPUT/OUTPUT ON EACH SCREEN

This section explains how to input and output data of the following types to and from each operation screen: program, parameter, offset, pitch error compensation, macro variable, workpiece coordinate system data, and operation history.

Section 8.2, "INPUT/OUTPUT ON EACH SCREEN", consists of the following subsections:

8.2.1 Inputting and Outputting a Program

8.2.1.1 Inputting a program

The following explains how to input a program from an external device to the memory of the CNC by using the program editing screen or program folder screen.

Inputting a program

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key 

to display the program editing screen or program folder screen.
3 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
4 Press the soft key [(OPRT)].
5 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
6 Type the name of the file that you want to input.
Press the soft key [F-NAME]. To specify the program number to input, type the program number and press the soft key [O SET].
For an explanation of the operations to be performed if an input file name [F-NAME] and an input program number [O SET] are omitted, see the table below.

7 Press the soft key [EXEC].
This starts reading the program, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

The read program is registered in the program memory of the path currently selected.

[F-NAME]	[O SET]	Input file name	Input program	Input program number
BLANK	INPUT	File for the program number specified with [O SET]	All programs in the program specified with [O SET]	Continuous program numbers starting at one specified with [O SET]
INPUT	BLANK	File name set with [F-NAME]	All programs in the file specified with [F-NAME]	File name at the time the file is saved
INPUT	INPUT	File name set with [F-NAME]	All programs in the file specified with [F-NAME]	Continuous program numbers starting at one specified with [O SET]

8.2.1.2 Outputting a program

A program stored in the memory of the CNC unit is output to an external device.

Outputting a program

Procedure

1 Make sure the output device is ready for writing.
2 Press the function key 

to display the program editing screen or program folder screen.
3 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
4 Press the soft key [(OPRT)].
5 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
6 Type the program number to output and press the soft key [O SET]. To specify an output file name, type the output file name and press the soft key [F-NAME].
If no output file name or output program number is specified here, the main program or the program being subjected to background editing is output.
For an explanation of the operations to be performed if an output file name [F-NAME] and an output program name [O SET] are omitted, see the table below.
7 Press the soft key [EXEC].
This starts outputting the program, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

[F-NAME]	[O SET]	Output file name	Output program
BLANK	BLANK	Main program or program number being subjected to background editing	Main program or program being subjected to background editing
BLANK	-9999	ALL-PROG.TXT	All programs in the program memory that are displayed in the program list
BLANK	INPUT	Program number set with [O SET]	Program in the NC that is set with [O SET]
INPUT	BLANK	File name set with [F-NAME]	Main program or program being subjected to background editing
INPUT	-9999	File name set with [F-NAME]	All programs in the program memory that are displayed in the program list
INPUT	INPUT	File name set with [F-NAME]	Program in the NC that is set with [O SET]

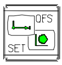
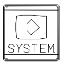
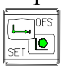
8.2.2 Inputting and Outputting Parameters

8.2.2.1 Inputting parameters

Parameters are loaded into the memory of the CNC unit from an external device. The input format is the same as the output format. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Inputting parameters

Procedure

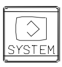
1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
4 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
5 Enter 1 in response to the prompt for “PARAMETER WRITE” in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the soft key [PARAMETER], then the parameter screen appears.
8 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “CNC-PARA.TXT” is assumed.
12 Press the soft key [EXEC].
This starts reading the parameter, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].
13 Press the function key .
14 Press the soft key [SETTING].
15 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
16 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
17 Turn the power of the CNC on again.

8.2.2.2 Outputting parameters

All parameters are output in a defined output format from the memory of the CNC to an external device.

Outputting parameters

Procedure

1 Make sure the output device is ready for writing.
2 Press the function key .
3 Press the soft key [PARAMETER], then the parameter screen appears.
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].

7 If all parameters are to be output, press the soft key [ALL]. If only the parameters with nonzero values are to be output, press the soft key [NON-0].
8 Type the file name that you want to output.
If the file name is omitted, default file name “CNC-PARA.TXT” is assumed.
9 Press the soft key [EXEC].
This starts outputting the parameter, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Explanation

- Suppressing output of parameters set to 0

When bit 1 (PRM) of parameter No. 0010 is set to 1, and soft key [EXEC] is pressed, the following parameters are not output:

	Other than axis type	Axis type
Bit type	Parameter for which all bits are set to 0.	Parameter for an axis for which all bits are set to 0.
Value type	Parameter whose value is 0.	Parameter for an axis for which the value is 0.

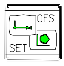
8.2.3 Inputting and Outputting Offset Data

8.2.3.1 Inputting offset data

Offset data is loaded into the memory of the CNC from an external device. The input format is the same as for offset value output. When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Inputting offset data

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key 

.
3 Press the continuous menu key  until soft key [OFFSET] appears.
Press the soft key [OFFSET].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “TOOLOFST.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the offset data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

8.2.3.2 Outputting offset data

All offset data is output in a defined output format from the memory of the CNC to an external device.

Outputting offset data

Procedure

1. Make sure the output device is ready for writing.
2. Press the function key .
3. Press the continuous menu key until soft key [OFFSET] appears.
   Press the soft key [OFFSET].
4. Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5. Press the soft key [(OPRT)].
6. Press the continuous menu key until soft key [F OUTPUT] appears.
   Press the soft key [F OUTPUT].
7. Type the file name that you want to output.
   If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
8. Press the soft key [EXEC].
   This starts outputting the offset data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
   To cancel the output, press the soft key [CAN].

Explanation

- Output format

Output format is as follows:

M

- Tool compensation memory A (bit 6 (NGW) of parameter No.8136 = 1)

```
%
G10 G90 P01 R_
G10 G90 P02 R_
 ...
G10 G90 P_ R_
%
```

P_ : Tool offset number (1 to the number of tool compensation pairs)
R_ : Tool compensation data. Output with a decimal point in the input unit used at output.

- Tool compensation memory C (bit 6 (NGW) of parameter No.8136 = 0)

```
%
G10 G90 L10 P01 R_
G10 G90 L11 P01 R_
G10 G90 L12 P01 R_
G10 G90 L13 P01 R_
G10 G90 L10 P02 R_
 ...
G10 G90 L12 P_ R_
G10 G90 L13 P_ R_
%
```

L10 : Geometry compensation amount corresponding to the H code
L11 : Wear compensation amount corresponding to the H code
L12 : Geometry compensation amount corresponding to the D code
L13 : Wear compensation amount corresponding to the D code
P_, and R_ have the same meanings as for tool compensation memory A.

T

- Without geometry/wear offset (bit 6 (NGW) of parameter No.8136 = 1)
  The tool compensation amount and tool nose radius compensation amount are output in the following format.

```
%
G10 P01 X_ Z_ R_ Q_ Y_
G10 P02 X_ Z_ R_ Q_ Y_
...
G10 P__ X_ Z_ R_ Q_ Y_
%
```

P_: Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Tool compensation amount
X_: Tool compensation data (X). Output with a decimal point in the input unit used at output.
Z_: Tool compensation data (Z). Same as X_.
R_: Tool nose radius offset amount (R). The data format is the same as for X_.
When tool nose radius compensation is not provided, this item is not output.
Q_: Virtual tool nose number (TIP). When tool nose radius compensation is not provided, this item is not output.
Y_ : Tool compensation data (Y). The data format is the same as for X_.
When no Y-axis offset is provided, this item is not output.

- With geometry/wear offset (bit 6 (NGW) of parameter No.8136 = 0)
  The tool compensation amount and tool nose radius compensation amount are output in the following format.

%
G10 P01 X_ Z_ R_ Q_ Y_
G10 P02 X_ Z_ R_ Q_ Y_
...
G10 P__ X_ Z_ R_ Q_ Y_
G10 P10001 X_ Z_ R_ Y_
G10 P10002 X_ Z_ R_ Y_
...
G10 P100__ X_ Z_ R_ Y_
%

P_: Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Tool wear compensation amount
10000 + tool offset number: Tool geometry compensation amount
X_: Tool compensation data (X). Output with a decimal point in the input unit used at output.
Z_: Tool compensation data (Z). Same as X_.
R_: Tool nose radius offset amount (R). The data format is the same as for X_. When tool nose radius compensation is not provided, this item is not output.
Q_: Virtual tool nose number (TIP). When tool nose radius compensation is not provided, this item is not output.
Y_ : Tool compensation data (Y). The data format is the same as for X_. When no Y-axis offset is provided, this item is not output.

NOTE
The input format and output format do not depend on the G-code system A/B/C.

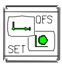
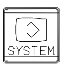
8.2.4 Inputting and Outputting Pitch Error Compensation Data

8.2.4.1 Inputting pitch error compensation data

Pitch error compensation data are loaded into the memory of the CNC from an external device. The input format is the same as the output format. When a pitch error compensation data is loaded which has the corresponding data number as a pitch error compensation data already registered in the memory, the loaded data replaces the existing data.

Inputting pitch error compensation data

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the continuous menu key  until soft key [PITCH ERROR] appears.
Press the soft key [PITCH ERROR].

8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key [▷] until soft key [F INPUT] appears.
Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
12 Press the soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].
13 Press the function key [OFS/SET].
14 Press the soft key [SETTING].
15 Press the MDI switch on the machine operator's panel or enter state emergency stop.
16 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
17 Turn the power of the CNC on again.

8.2.4.2 Outputting pitch error compensation data

All pitch error compensation data are output in a defined output format from the memory of the CNC to an external device.

Outputting pitch error compensation data

Procedure

1 Make sure the output device is ready for writing.
2 Press the function key [SYSTEM].
3 Press the continuous menu key [▷] until soft key [PITCH ERROR] appears.
Press the soft key [PITCH ERROR].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key [▷] until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "PITCH.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

8.2.4.3 Input/output format of pitch error compensation data

Pitch error compensation data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Keyword	Meaning of the following numeric value
N	Pitch error compensation data number + 10000
Q	Data identification (1 : Parameter data, 0 : Pitch error compensation data)
P	Pitch error compensation data value

- Format

Pitch error compensation data is output in the following format:

N	*****	Q0	P	****	;

The 5-digit numeric value following N indicates a pitch error compensation data number to which a value of 10000 is added.
Q0 indicates pitch error compensation data
The numeric value following P indicates the value (integer value) of pitch error compensation data between -7 to 7.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example

N10001 Q0 P100 ;
Pitch error compensation data number 1
Pitch error compensation data value 100

- Beginning and end of a record

A pitch error compensation data record begins with % and ends with %.

Example

% .... Beginning of record
N10000 Q0 P10
N10001 Q0 P100
:
N11023 Q0 P0
% .... End of record

When parameters and pitch error compensation data are integrated into one file, % is added to the beginning and end of the file.

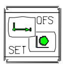
8.2.5 Inputting and Outputting Custom Macro Common Variables

8.2.5.1 Inputting custom macro common variables

The value of a custom macro common variable is loaded into the memory of the CNC from an external device. The same format used to output custom macro common variables is used for input.

Inputting custom macro common variables

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [MACRO] appears.
Press the soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].

7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “MACRO.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the custom macro common variables, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Explanation

- Common variables

The common variables (#500 to 999) can be input and output.
#100 to #199 can be input when bit 3 (PV5) of parameter No. 6001 is set to 1.

8.2.5.2 Outputting custom macro common variables

Custom macro common variables stored in the memory of the CNC can be output in a defined output format to an external device.

Outputting custom macro common variables

Procedure

1 Make sure the output device is ready for writing.
2 Press the function key .
3 Press the continuous menu key until soft key [MACRO] appears.
Press the soft key [MACRO].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name “MACRO.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the custom macro common variables, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Explanation

- Output format

The output format is as follows:
The values of custom macro variables are output in a bit-image hexadecimal representation of double-precision floating-point type data.

```
%
G10 L85 P500(4024000000000000)
G10 L85 P501(4021000000000000)
G10 L85 P502(0000000000000000)
 ･
SETVN500[ABC,DEF]
SETVN501[GHI,JKL]
SETVN502[MNO,PQR]
 ･
M02
%
```

> **NOTE**
> The conventional custom macro statement program format cannot be used for output.

By setting bit 0 (MCO) of parameter No. 6019, it is possible to output macro variable numbers and variable data values as comments after normally output data.
The output comments do not affect data input.

- Common variable

The common variables (#500 to #999) can be input and output.
#100 to #199 can be output when bit 3 (PV5) of parameter No. 6001 is set to 1.

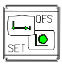
8.2.6 Inputting and Outputting Workpiece Coordinates System Data

8.2.6.1 Inputting workpiece coordinate system data

Coordinate system variable data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When coordinate system variable data with a data number corresponding to existing coordinate system variable data registered in the memory is loaded, the loaded coordinate system variable data replaces the existing coordinate system variable data.

Inputting workpiece coordinate system data

Procedure

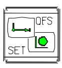
1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [WORK] appears.
Press the soft key [WORK].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

8.2.6.2 Outputting workpiece coordinate system data

All coordinate system variable data is output in the output format from the memory of the CNC to an external device.

Outputting workpiece coordinate system data

Procedure

1 Make sure the output device is ready for writing.
2 Press the function key .

3 Press the continuous menu key [▷] until soft key [WORK] appears.
  Press the soft key [WORK].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key [▷] until soft key [F OUTPUT] appears.
  Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
  If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
8 Press the soft key [EXEC].
  This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press the soft key [CAN].

8.2.7 Inputting and Outputting Operation History Data

Only output operation is permitted on operation history data.
The output data is in text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

8.2.7.1 Outputting operation history data

All operation history data is output in the output format form the memory of the CNC to an external device.

Outputting operation history data

Procedure

1 Make sure the output device is ready for writing.
2 Press the function key [SYSTEM].
3 Press the MDI switch on the machine operator's panel, make sure the display of the operation history screen is enabled by setting "1" in bit 4 (OPH) of parameter No. 3106.
4 Press the continuous menu key [▷] until soft key [OPERAT HISTRY] appears.
  Press the soft key [OPERAT HISTRY].
5 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the soft key [(OPRT)].
7 Press the soft key [F OUTPUT].
8 Type the file name that you want to output.
  If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
9 Press the soft key [EXEC].
  This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press the soft key [CAN].

8.3 INPUT/OUTPUT ON THE ALL IO SCREEN

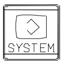
Just by using the ALL IO screen, you can input and output programs, parameters, offset data, pitch error compensation data, macro variables, and workpiece coordinate system data.

> **NOTE**
> The ALL IO screen can be operated only if a memory card interface is selected as an external I/O device.

The following explains how to display the ALL IO screen:

Displaying the ALL IO screen

Procedure

1 Press the function key .
2 Press the continuous menu key  several times.
3 Press the soft key [ALL IO] to display the ALL IO screen.

The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

Configuration of this section

Section 8.3, "INPUT/OUTPUT ON THE ALL IO SCREEN", consists of the following subsections:

8.3.1 Inputting/Outputting a Program

A program can be input and output using the ALL IO screen.

Inputting a program

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F-NAME].
If the input file name is omitted, default input file name “ALL-PROG.TXT” is assumed.
See the table below for details.
6 Set the program number to be used after the input.
Type a program number, and press the soft key [O SET].
If no program number is specified here, the program number in the file is adopted as is.
7 Press the soft key [EXEC].
This starts reading the program, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

[F-NAME]	[O SET]	Input file name	Input program	Input program number
BLANK	INPUT	File for the program number specified with [O SET]	All programs in the program specified with [O SET]	Continuous program numbers starting at one specified with [O SET]
INPUT	BLANK	File name set with [F-NAME]	All programs in the file specified with [F-NAME]	Program number at the time the file is saved
INPUT	INPUT	File name set with [F-NAME]	All programs in the file specified with [F-NAME]	Continuous program numbers starting at one specified with [O SET]

Outputting a program

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the program that you want to output.
Type a program number, and press the soft key [O SET].
If -9999 is typed, all programs in the memory are output.
6 Set the file name to be output.
Type a file name, and press the soft key [F-NAME].
When no file name is set, the output file name is assumed to be "O""number" if a single program number is specified; if -9999 is specified, the output file name is assumed to be "ALL-PROG.TXT".
See the table below for details.
7 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

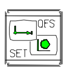
[F-NAME]	[O SET]	Output file name	Output program
BLANK	BLANK	Main program or program number being subjected to background editing	Main program or program being subjected to background editing
BLANK	-9999	ALL-PROG.TXT	All programs in the program memory that are displayed in the program list
BLANK	INPUT	Program number set with [O SET]	Program in the NC that is set with [O SET]
INPUT	BLANK	File name set with [F-NAME]	Main program or program being subjected to background editing
INPUT	-9999	File name set with [F-NAME]	All programs in the program memory that are displayed in the program list
INPUT	INPUT	File name set with [F-NAME]	Program in the NC that is set with [O SET]

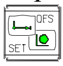
8.3.2 Inputting and Outputting Parameters

Parameters can be input and output using the ALL IO screen.

Inputting parameters

Procedure

1 Press the function key 
.
2 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].

3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100 appears.
5 Press the soft key [PARAMETER] on the ALL IO screen.
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the soft key [(OPRT)].
8 Press the soft key [N INPUT].
9 Set the name of the file that you want to input.
Type a file name, and press the soft key [F-NAME].
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
10 Press the soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].
11 Press the function key .

12 Press the soft key [SETTING].
13 Press the MDI switch on the machine operator's panel or enter state emergency stop.
14 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
15 Turn the power of the CNC on again.

Outputting parameters

Procedure

1 Press the soft key [PARAMETER] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F-NAME].
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the parameter, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

8.3.3 Inputting and Outputting Offset Data

Offset data can be input and output using the ALL IO screen.

Inputting offset data

Procedure

1 Press the soft key [OFFSET] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F-NAME].
If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.
6 Press the soft key [EXEC].
This starts reading the offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Outputting offset data

Procedure

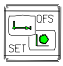
1 Press the soft key [OFFSET] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F-NAME].
If the file name is omitted, default file name “TOOLOFST.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the offset data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

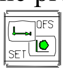
8.3.4 Inputting/Outputting Pitch Error Compensation Data

Pitch error compensation data can be input and output using the ALL IO screen.

Inputting pitch error compensation data

Procedure

1 Press the function key 

.
2 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
3 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
4 Enter 1 in response to the prompt for “PARAMETER WRITE” in setting data. Alarm SW0100 appears.
5 Press the soft key [PITCH] on the ALL IO screen.
6 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
7 Press the soft key [(OPRT)].
8 Press the soft key [N INPUT].
9 Set the name of the file that you want to input.
Type a file name, and press the soft key [F-NAME].
If the input file name is omitted, default input file name “PITCH.TXT” is assumed.
10 Press the soft key [EXEC].
This starts reading the pitch error compensation data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].
11 Press the function key 

.
12 Press the soft key [SETTING].
13 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
14 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
15 Turn the power of the CNC on again.

Outputting pitch error compensation data

Procedure

1 Press the soft key [PITCH] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].

5 Set the file name to be output.
Type a file name, and press the soft key [F-NAME].
If the file name is omitted, default file name “PITCH.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the pitch error compensation data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

8.3.5 Inputting/Outputting Custom Macro Common Variables

Custom macro common variables can be input and output using the ALL IO screen.

Inputting custom macro common variables

Procedure

1 Press the soft key [MACRO] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F-NAME].
If the input file name is omitted, default input file name “MACRO.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the custom macro common variables, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Outputting custom macro common variables

Procedure

1 Press the soft key [MACRO] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F-NAME].
If the file name is omitted, default file name “MACRO.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the custom macro common variables, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

8.3.6 Inputting and Outputting Workpiece Coordinates System Data

Workpiece coordinates system data can be input and output using the ALL IO screen.

Inputting workpiece coordinate system data

Procedure

1 Press the soft key [WORK] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].

5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F-NAME].
If the input file name is omitted, default input file name “EXT_WKZ.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the workpiece coordinate system data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Outputting workpiece coordinate system data

Procedure

1. Press the soft key [WORK] on the ALL IO screen.
2. Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3. Press the soft key [(OPRT)].
4. Press the soft key [F OUTPUT].
5. Set the file name to be output.
   Type a file name, and press the soft key [F-NAME].
   If the file name is omitted, default file name “EXT_WKZ.TXT” is assumed.
6. Press the soft key [EXEC].
   This starts outputting the workpiece coordinate system data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
   To cancel the output, press the soft key [CAN].

8.3.7 File Format

Explanation

- File format

All files output to or input from external I/O devices are text files. The format is described below.
A file starts with % or LF, followed by the actual data. A file always ends with %. In an input operation, data between the first % and the next LF is skipped. Each block ends with an LF, not a semicolon (;).

- LF: 0A (hexadecimal) of ASCII code
- When a file containing lowercase letters, kana characters, and several special characters (such as $, \, and !) is written, those letters and characters are ignored.

Example)

```
%
O0001(MEMORY CARD SAMPLE FILE)
G17 G49 G97
G92 X-11.3 Y2.33
 :
 :
M30
%
```

- ASCII codes are always used when data is input to or output from a memory card regardless of the setting parameter (ISO/EIA).
- Bit 3 (NCR) of parameter No. 0100 can be used to specify whether the end of block code (EOB) is output as "LF" only, or as "LF, CR, CR."

8.4 MEMORY CARD SCREEN

8.4.1 Displaying the Memory Card Screen

Procedure

1 Press the function key PROG.
2 Press the soft key [DIR]. The program list screen appears.
(If the soft key does not appear, press the continuous menu key ▷.)
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICE CHANGE].
5 Press the soft key [MEMORY CARD], then the memory card screen appears.

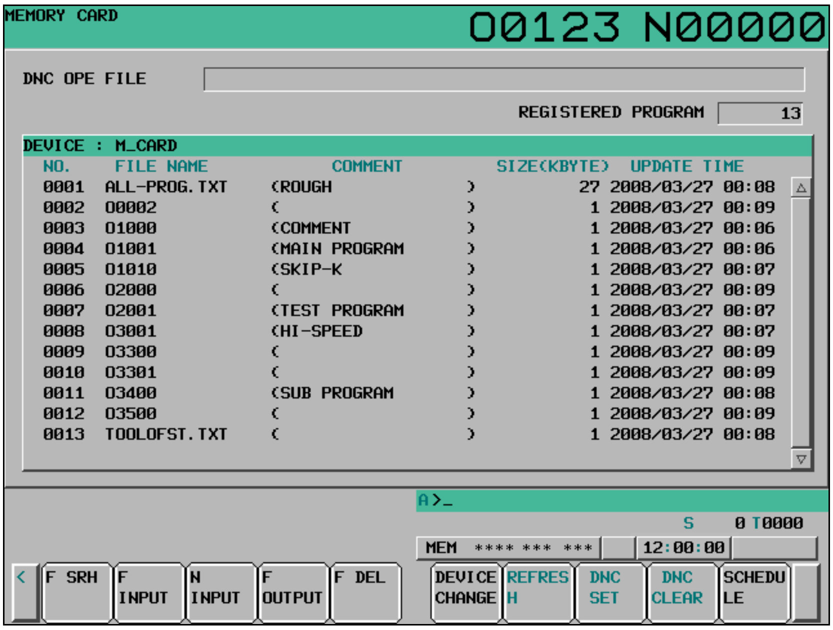


Fig. 8.4.1 (a) Memory card screen

Display item

DNC OPE FILE
The file name to be subjected to DNC operation is displayed.

REGISTERED PROGRAM
The number of registered files is displayed.

NO.
The file number is displayed.

FILE NAME
The file name is displayed.

COMMENT
The program comment is displayed.

SIZE(KBYTE)
The memory capacity the file takes is displayed.

UPDATE TIME
The update date of the file is displayed.

8.4.2 Displaying and Operating the File List

DIR +
For the 8.4-inch display unit, the displays can be changed between the comment and the size/date.

REFRESH
Display data can be updated.

F SRH
A file can be searched for. The file found is displayed at the beginning of the list.

1. Press the soft key [F SRH].
2. Enter the file number of a file to be searched for.
3. Press the soft key [F SET].
4.
   - To execute the search request, press the soft key [EXEC].
   - To cancel the search request, press the soft key [CAN].

F DEL
A file can be deleted.

<Using a file number to specify a file to be deleted>
1. Press the soft key [F DEL].
2. Enter the file number of a file to be deleted.
3. Press the soft key [F SET].
4.
   - To execute the delete request, press the soft key [EXEC].
   - To cancel the delete request, press the soft key [CAN].

<Using a file name to specify a file to be deleted>
1. Press the soft key [F DEL].
2. Enter the name of a file to be deleted.
3. Press the soft key [F-NAME].
4.
   - To execute the delete request, press the soft key [EXEC].
   - To cancel the delete request, press the soft key [CAN].

DEVICE CHANGE
A device can be selected on the program list screen.

1. Press the soft key [CHANGE DEVICE].
2. Press the soft key for a device to be changed.

DNC SET
A file to be subjected to DNC operation can be selected.
See Section 4.3, "DNC OPERATION" for details.

DNC CLEAR
The selection of a file to be subjected to DNC operation can be canceled.
See Section 4.3, "DNC OPERATION" for details.

SCHEDULE
The schedule list screen can be displayed.
See Section 4.4, "SCHEDULE OPERATION" for details.

8.4.3 Inputting/Outputting a File

A program can be input and output using the memory card screen.

Inputting a program (F INPUT)

1 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
2 Press the soft key [(OPRT)].
3 Press the soft key [F INPUT].
4 Set the file number that you want to input.
Type a file number, and press the soft key [F SET].
See the table below for details.
5 Set the program number to be used after the input.
Type a program number, and press the soft key [O SET].
If no program number is specified here, the program number in the file is adopted as is.
6 Press the soft key [EXEC].
This starts reading the program, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

[F SET]	[O SET]	Input file name	Input program	Input program number
BLANK	INPUT	File for the program number specified with [O SET]	All programs in the program specified with [O SET]	Continuous program numbers starting at one specified with [O SET]
INPUT	BLANK	File name for the file number specified with [F SET]	All programs in the file specified with [F SET]	Program number at the time the file is saved
INPUT	INPUT	File name for the file number specified with [F SET]	All programs in the file specified with [F SET]	Continuous program numbers starting at one specified with [O SET]

Inputting a file (N INPUT)

1 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
2 Press the soft key [(OPRT)].
3 Press the soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the soft key [F-NAME].
See the table below for details.
5 Set the program number to be used after the input.
Type a program number, and press the soft key [O SET].
If no program number is specified here, the program number in the file is adopted as is.
6 Press the soft key [EXEC].
This starts reading the program, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

[F-NAME]	[O SET]	Input file name	Input program	Input program number
BLANK	INPUT	File for the program number specified with [O SET]	All programs in the program specified with [O SET]	Continuous program numbers starting at one specified with [O SET]
INPUT	BLANK	File name set with [F-NAME]	All programs in the file specified with [F-NAME]	Program number at the time the file is saved

[F-NAME]	[O SET]	Input file name	Input program	Input program number
INPUT	INPUT	File name set with [F-NAME]	All programs in the file specified with [F-NAME]	Continuous program numbers starting at one specified with [O SET]

Outputting a file

1 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
2 Press the soft key [(OPRT)].
3 Press the soft key [F OUTPUT].
4 Set the program that you want to output.
Type a program number, and press the soft key [O SET].
If -9999 is typed, all programs in the memory are output.
5 Set the file name to be output.
Type a file name, and press the soft key [F-NAME].
When no file name is set, the output file name is assumed to be "O""number" if a single program number is specified; if -9999 is specified, the output file name is assumed to be "ALL-PROG.TXT". See the table below for details.
6 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

[F-NAME]	[O SET]	Output file name	Output program
BLANK	BLANK	Main program or program number being subjected to background editing	Main program or program being subjected to background editing
BLANK	-9999	ALL-PROG.TXT	All programs in the program memory that are displayed in the program list
BLANK	INPUT	Program number set with [O SET]	Program in the NC that is set with [O SET]
INPUT	BLANK	File name set with [F-NAME]	Main program or program being subjected to background editing
INPUT	-9999	File name set with [F-NAME]	All programs in the program memory that are displayed in the program list
INPUT	INPUT	File name set with [F-NAME]	Program in the NC that is set with [O SET]

8.5 EMBEDDED ETHERNET OPERATIONS

8.5.1 FTP File Transfer Function

The operation of the FTP file transfer function is described below.

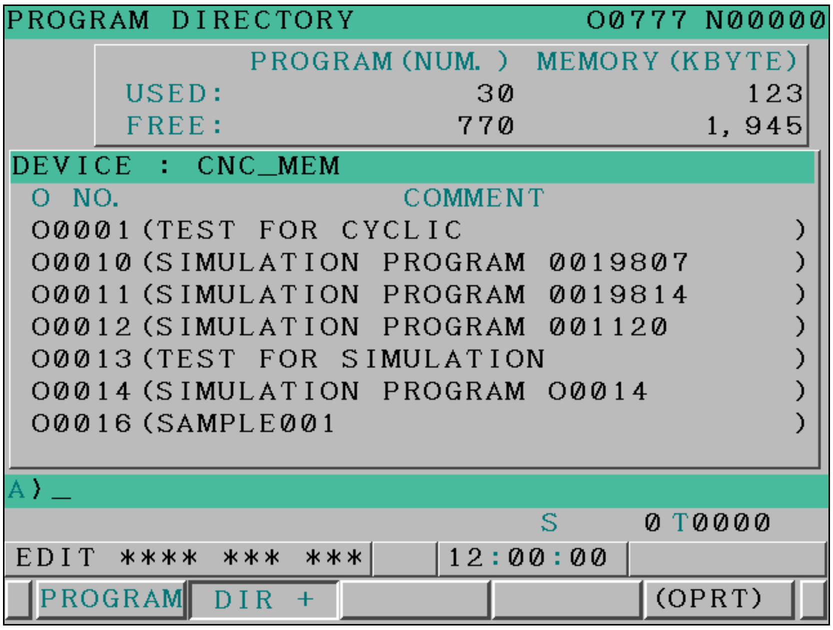
Host file list display

A list of the files held on the host computer is displayed.

Procedure

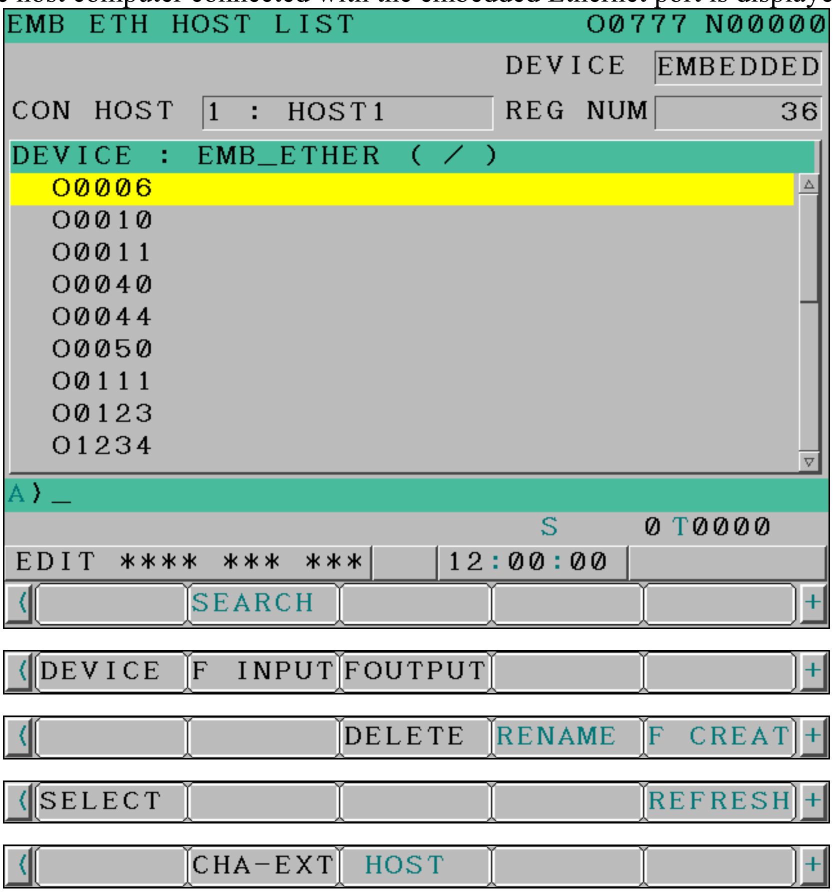
1 Press the function key 

.
2 Press soft key [DIR+]. The program folder screen appears. (If the soft key does not appear, press the continuous menu key.)

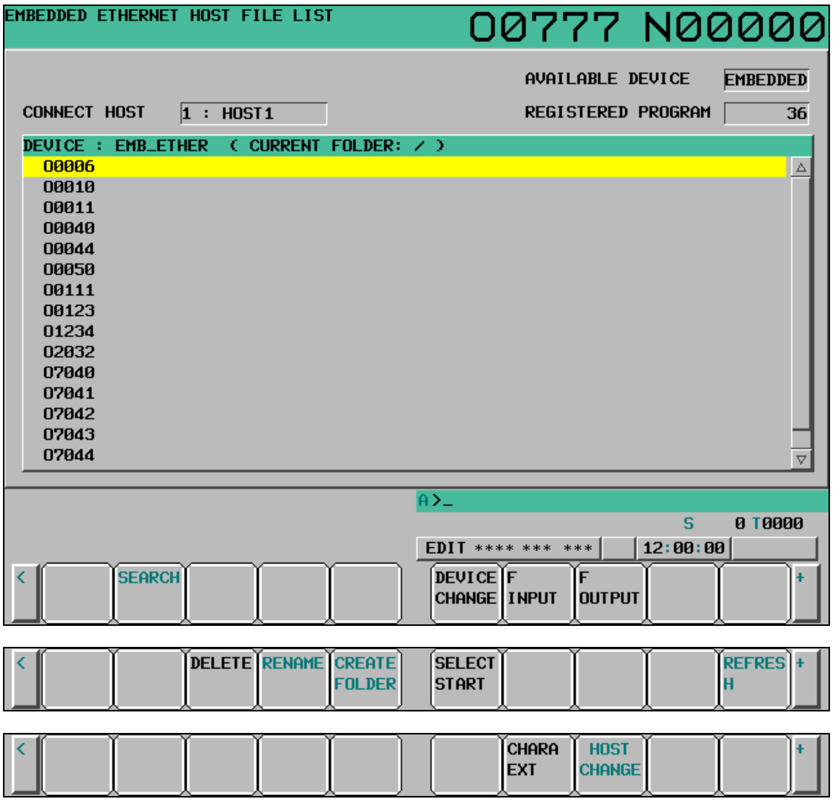


3 Press soft keys [(OPRT)] and [DEVICE] in that order. The soft keys for selectable devices appear.

4 Pressing soft key [EMB ETH] displays the Embedded Ethernet host file list screen, on which a list of files in the host computer connected with the embedded Ethernet port is displayed.

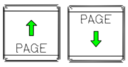


Embedded Ethernet host file list screen (8.4-inch LCD)



Embedded Ethernet host file list screen (10.4-inch LCD)

> **NOTE**
> When using the FTP file transfer function, check that the valid device is the embedded Ethernet port.
> The two conditions below determine a connection destination on the host file list screen:
> (1) Check that the valid device is the embedded Ethernet port. It is selected using soft key [EMB/PCM] on the Ethernet setting screen.
> (2) A host computer can be selected from connection destinations 1, 2, and 3. A computer to be connected is selected using the operation procedure described in Subsection 5.2.2.1, "Operation on the FTP file transfer setting screen" or “HOST CHANGE” in the Subsection 5.4.1.1, "Displaying and operating the file list" of MAINTENANCE MANUAL (B-64305EN).

5 When a list of files is larger than one page, the screen display can be switched using the page keys .

Display item

DEVICE (AVAILABLE DEVICE)

The currently selected device is displayed.

CON HOST (CONNECT HOST)

Number of the currently connected host of the host computer

REG NUM (REGISTERED PROGRAM)

The number of files in the current folder.

DEVICE

Current device. When the embedded Ethernet host file list is selected, “EMB_ETHER” is displayed.

CURRENT FOLDER
Current work folder in the host computer

FILE LIST
Information of the files and folders in the host computer

Operation list

DEVICE (DEVICE CHANGE)
Enables a device to be selected from the program folder screen. To select the embedded Ethernet host file list, press soft key [EMB ETH].

DIR +
Switches between the outline and detailed file lists.

F CREAT (CREATE FOLDER)
Create a subfolder in the current work folder in the host computer.

DELETE
Deletes a file or folder in the host computer.

RENAME
Renames a file or folder in the host computer.

HOST (HOST CHANGE)
Changes the connected host computer.

SEARCH
Searches for a file through the current folder in the host computer.

REFRESH
Updates the information displayed on the embedded Ethernet host file list screen.

F INPUT
Transfers a program from the host computer to the CNC memory.

F OUTPUT
Transfers a program from the CNC memory to the host computer.

> **NOTE**
> The character strings enclosed in parentheses are those displayed when the 10.4-inch LCD unit is used.

Inputting programs

The following procedure can be used to transfer programs from the host computer to the CNC memory.

1. Press soft key [F INPUT].
2. Select a program in the host computer.
   In the host computer, place the cursor on the file you want to input and press soft key [F GET], or key in the file name.
3. Press soft key [F NAME].
4. If you want to rename a program when inputting it, key in the program number and press soft key [O SET].
5. Press soft key [EXEC].

The following table summarizes what operation occurs when the input file name [F NAME] and input program number [O SET] are omitted.

[F NAME]	[O SET]	Key input buffer	Input file name	Input program	Input program No.
–	–	–	Warning message "NO PROGRAM SELECTED" is displayed, and nothing is input.		
		Other than Oxxxx	Warning message "THE WRONG DATA IS USED" is displayed, and nothing is input.		
		Oxxxx	File name set in the key input buffer (NOTE)	All programs in the input file	Continuous program numbers starting at one (xxxx) set in the key input buffer
	-9999	No relation	Warning message "NO PROGRAM SELECTED" is displayed, and nothing is input.		
	O		Same file name as program No. set by [O SET] (NOTE)	All programs in the input file	Continuous program numbers starting at one set by [O SET]
O	–		File name set by [F NAME]	All programs in a file specified by [F NAME]	Program No. used when the program was saved
	-9999		Warning message "THE WRONG DATA IS USED" is displayed, and nothing is input.		
	O		File name set by [F NAME]	All programs in the file specified by [F NAME]	Continuous program numbers starting at one set by [O SET]

O: Specified
– : Not specified

> **NOTE**
> The input file name consists of "O" followed by a 4-digit number.
> If program input is executed by specifying program No. 1, for example, a file whose file name is "O0001" is input.
> If this operation is performed for path 2, the file name is suffixed with the file extension "P-2" (for this example, "O0001.P-2").

Outputting programs

The following procedure can be used to transfer programs from the CNC memory to the host computer.

1 Press soft key [FOUTPUT].
2 Select a program in the CNC.
Key in the program No. of the program to be output.
3 Press soft key [O SET].
4 If you want to rename a program when outputting it, key in the file name and press soft key [F NAME].
5 Press soft key [EXEC].

The following table summarizes what operation occurs when the output file name [F NAME] and output program number [O SET] are omitted.

[F NAME]	[O SET]	Key input buffer	Output file name	Output program
–	–	–	Currently selected main program name (NOTE 1 and NOTE 2)	Currently selected main program (NOTE 1)
		Other than Oxxxx	Warning message "THE WRONG DATA IS USED" is displayed, and nothing is output.	
		Oxxxx	Program name set in the key input buffer (NOTE 2)	Program in the CNC memory set in the key input butter
		O-9999	ALL-PROG.TXT (NOTE 3)	All programs in the CNC memory
	-9999	No relation		
	O		Same file name as program No. set by [O SET] (NOTE 2)	Program in the CNC memory set by [O SET]
O	–		File name set by [F NAME]	Currently selected main program (NOTE 1)
	-9999			All programs in the CNC memory
	O			Program in the CNC memory set by [O SET]

O : Specified
– : Not specified

NOTE
1. If a file is undergoing background editing, it is output.
2. The output file name consists of "O" followed by a 4-digit number.
   If a program whose program No. is 1 is output, for example, it is output under the file name "O0001" to the host computer.
   If this operation is performed for path 2, the file name is suffixed with the file extension "P-2" (for this example, "O0001.P-2").
3. If this operation is performed for path 2, the file name "ALL-PROG.P-2" is used.

8.6 FLOPPY CASSETTE SCREEN

8.6.1 Displaying the Floppy Cassette Screen

Procedure

1. Press the function key .
2. Press the soft key [DIR]. The program list screen appears.
   (If the soft key does not appear, press the continuous menu key .)
3. Press the soft key [(OPRT)].
4. Press the soft key [DEVICE CHANGE].
5. Press the soft key [FLOPPY], then the floppy cassette screen appears.

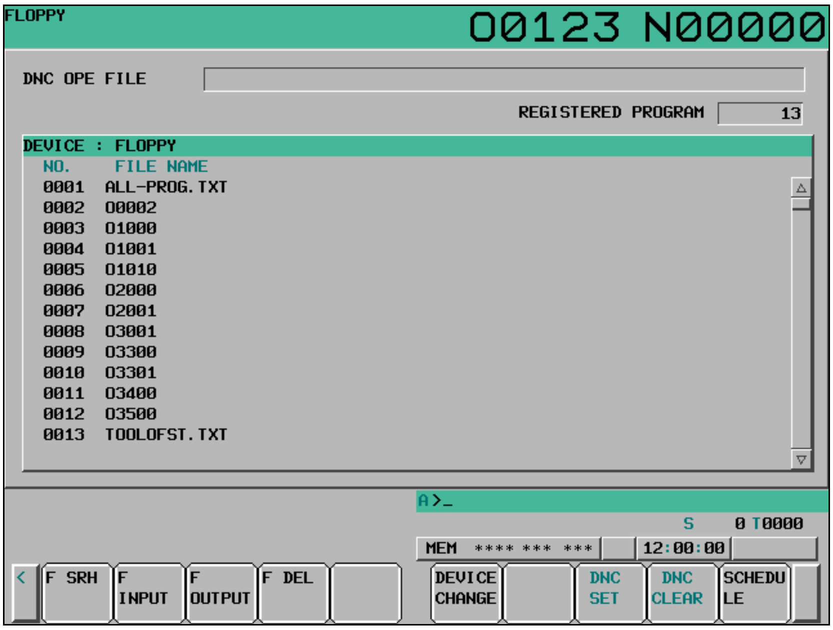


Fig. 8.6.1 (a) Floppy cassette screen

Display item

DNC OPE FILE
The file name to be subjected to DNC operation is displayed.

REGISTERED PROGRAM
The number of registered files is displayed.

NO.
The file number is displayed.

FILE NAME

The file name is displayed.

8.6.2 Displaying and Operating the File List

F SRH

A file can be searched for. The file found is displayed at the beginning of the list.

1. Press the soft key [F SRH].
2. Enter the file number of a file to be searched for.
3. Press the soft key [F SET].
4. - To execute the search request, press the soft key [EXEC].
   - To cancel the search request, press the soft key [CAN].

F DEL

A file can be deleted.

<Using a file number to specify a file to be deleted>

1. Press the soft key [F DEL].
2. Enter the file number of a file to be deleted.
3. Press the soft key [F SET].
4. - To execute the delete request, press the soft key [EXEC].
   - To cancel the delete request, press the soft key [CAN].

<Using a file name to specify a file to be deleted>

1. Press the soft key [F DEL].
2. Enter the name of a file to be deleted.
3. Press the soft key [F-NAME].
4. - To execute the delete request, press the soft key [EXEC].
   - To cancel the delete request, press the soft key [CAN].

DEVICE CHANGE

A device can be selected on the program list screen.

1. Press the soft key [CHANGE DEVICE].
2. Press the soft key for a device to be changed.

DNC SET

A file to be subjected to DNC operation can be selected.
See Section 4.3, “DNC OPERATION” for details.

DNC CLEAR

The selection of a file to be subjected to DNC operation can be canceled.
See Section 4.3, “DNC OPERATION” for details.

SCHEDULE

The schedule list screen can be displayed.
See Section 4.4, “SCHEDULE OPERATION” for details.

8.6.3 Inputting/Outputting a File

A program can be input and output using the floppy cassette screen.

Inputting a file

1. Press the EDIT switch on the machine operator's panel or enter state emergency stop.
2. Press the soft key [(OPRT)].
3. Press the soft key [F INPUT].
4. Set the file number that you want to input.
   Type a file number, and press the soft key [F SET].
   See the table below for details.
5. Set the program number to be used after the input.
   Type a program number, and press the soft key [O SET].
   If no program number is specified here, the program number in the file is adopted as is.
6. Press the soft key [EXEC].
   This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press the soft key [CAN].

[F SET]	[O SET]	Input file name	Input program	Input program number
BLANK	INPUT	File for the program number specified with [O SET]	All programs in the program specified with [O SET]	Continuous program numbers starting at one specified with [O SET]
INPUT	BLANK	File name for the file number specified with [F SET]	All programs in the file specified with [F SET]	Program number at the time the file is saved
INPUT	INPUT	File name for the file number specified with [F SET]	All programs in the file specified with [F SET]	Continuous program numbers starting at one specified with [O SET]

Outputting a file

1. Press the EDIT switch on the machine operator's panel or enter state emergency stop.
2. Press the soft key [(OPRT)].
3. Press the soft key [F OUTPUT].
4. Set the program that you want to output.
   Type a program number, and press the soft key [O SET].
   If -9999 is typed, all programs in the memory are output.
5. Set the file name to be output.
   Type a file name, and press the soft key [F-NAME].
   When no file name is set, the output file name is assumed to be "O""number" if a single program number is specified; if -9999 is specified, the output file name is assumed to be "ALL-PROG.TXT".
   See the table below for details.
6. Press the soft key [EXEC].
   This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
   To cancel the output, press the soft key [CAN].

[F-NAME]	[O SET]	Output file name	Output program
BLANK	BLANK	Main program or program number being subjected to background editing	Main program or program being subjected to background editing
BLANK	-9999	ALL-PROG.TXT	All programs in the program memory that are displayed in the program list

[F-NAME]	[O SET]	Output file name	Output program
BLANK	INPUT	Program number set with [O SET]	Program in the NC that is set with [O SET]
INPUT	BLANK	File name set with [F-NAME]	All programs in the program memory that are displayed in the program list
INPUT	-9999	File name set with [F-NAME]	Main program or program being subjected to background editing
INPUT	INPUT	File name set with [F-NAME]	Program in the NC that is set with [O SET]

8.7 SCREEN HARD COPY FUNCTION

Overview

This function converts screen information displayed on the CNC into bit map format data and output it to a memory card. Once output, bit map format data can be displayed and edited on a personal computer.

Explanation

- Start/cancellation methods

The screen hard copy function is started by pressing and holding down key SHIFT for five seconds. The function can be canceled by pressing key CAN or by changing the hard copy cancellation request signal HCABT<G067.6> to "1".

While the screen hard copy function is in progress, the hard copy execution status signal HCEXE<F061.3> is "1", and upon completion, it is set to "0". When a screen hard copy cancellation request is received, the hard copy cancellation request reception signal HCAB2<F061.2> is set to "1" and remains in the "1" state until the hard copy function is started again or a reset is made.

- Acquisition and output of screen data

When started, the screen hard copy function starts acquiring screen data. As soon as it has acquired it, the function outputs bit map format data to the memory card inserted into the LCD unit. While screen data is being acquired, the screen remains stationary for a few seconds. Acquired screen data can be output from the memory card screen. Also, while screen data is being output, "OUTPUT" blinks in the status display.

- Screen data file names

Bit map format screen data files created by this function are assigned the names below, starting with the one created after the power is turned on.

"HDCPY000.BMP" (name of the first file output to the memory card after the power is turned on)
"HDCPY001.BMP" (name of the second file output to the memory card after the power is turned on)
:
:
"HDCPY999.BMP" (name of the 1000th file output to the memory card after the power is turned on)

If, after a file with "HDCPY999.BMP" is output, the screen hard copy function is executed, the file name returns to "HDCPY000.BMP". If, however, a file with the same file name as the one to be output when the screen hard copy function is executed exists on the memory card, alarm SR1973 is issued. If the capacity of the memory card is exceeded, alarm SR1962 is issued.

Since no screen data in output in both cases, rename or delete the file or replace the memory card with a new one.

Limitation

- Screens whose hard copies cannot be made

Hard copies of the BOOT screen, the IPL screen, and the system alarm screen cannot be made.

- Foreground I/O devices

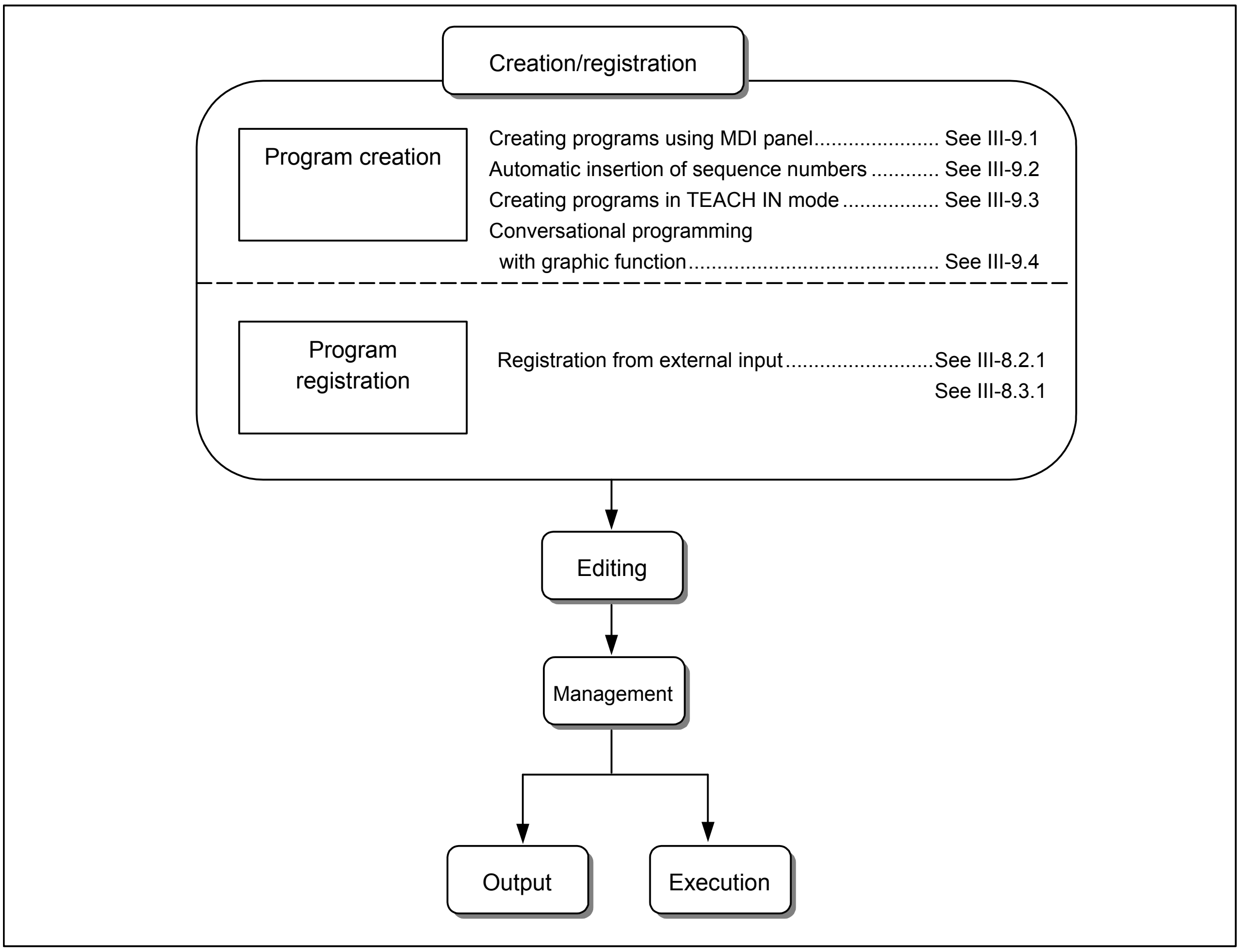
During DNC operation, for example, screen data cannot be output while a foreground I/O device is in use.

- Canceling the hard copy function

If the hard copy function is canceled before a hard copy is completed, an incomplete bit map file of data that has been output is created.

9 CREATING PROGRAMS

This chapter explains how to create programs by MDI of the CNC.
This chapter also explains automatic insertion of sequence numbers and how to create programs in TEACH IN mode.



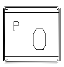
9.1 CREATING PROGRAMS USING THE MDI PANEL

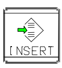
Programs can be created in the EDIT mode using the program editing functions described in III-10.

Procedure for Creating Programs Using the MDI Panel

1 Enter the EDIT mode.

2 Press the  key.

3 Press address key  and enter the program number.

4 Press the  key.

5 Create a program using the program editing functions described in III-10.

Explanation

- Comments in a program

Comments can be written in a program using the control in/out codes.
Example) O0001 (TEST PROGRAM) ;
M08 (COOLANT ON) ;

- When the INSERT key is pressed after the control-out code "(", comments, and control-in code ")" have been typed, the typed comments are registered.
- When the INSERT key is pressed midway through comments, to enter the rest of comments later, the data typed before the INSERT key is pressed may not be correctly registered (not entered, modified, or lost) because the data is subject to an entry check which is performed in normal editing.
- Control-out code "(" or Control-in code ")" cannot be registered by itself.

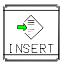
9.2 AUTOMATIC INSERTION OF SEQUENCE NUMBERS

Sequence numbers can be automatically inserted in each block when a program is created using the MDI keys in the EDIT mode.
Set the increment for sequence numbers in parameter No. 3216.

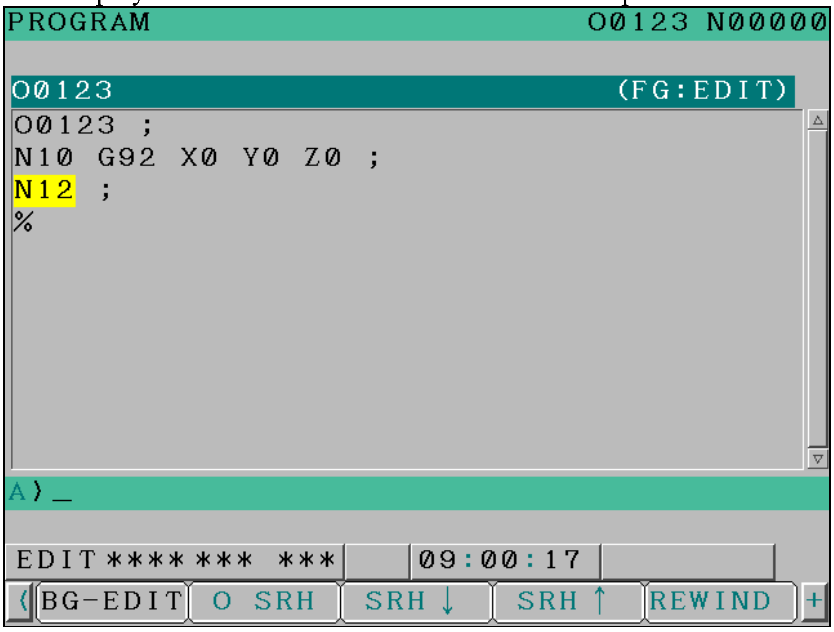
Procedure for automatic insertion of sequence numbers

Procedure

1 Enter 1 in response to the prompt for "SEQUENCE NO." in setting data. (see III-12.3.1).
2 Enter the EDIT mode.
3 Press PROG key to display the program screen.
4 Search for or register the number of a program to be edited and move the cursor to the EOB (;) of the block after which automatic insertion of sequence numbers is started.
When a program number is registered and an EOB (;) is entered with the INSERT key, sequence numbers are automatically inserted starting with 0. Change the initial value, if required, according to step 10, then skip to step 7.
5 Press address key N and enter the initial value of N.
6 Press INSERT key.
7 Enter each word of a block.
8 Press EOB key.

9 Press 

key. The EOB is registered in memory and sequence numbers are automatically inserted. For example, if the initial value of N is 10 and the parameter for the increment is set to 2, N12 inserted and displayed below the line where a new block is specified.



10 In the example above, if N12 is not necessary in the next block, pressing the 

key after N12 is displayed deletes N12.
If wishing to insert N100, not N12, into the next block, type N100 immediately after N12 is displayed, and press 

. This causes N100 to be registered and the initial value to be changed to 100.

9.3 CREATING PROGRAMS IN TEACH IN MODE (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.

Program screen in the TEACH IN JOG or TEACH IN HANDLE mode

Displayed items

In the TEACH IN JOG or TEACH IN HANDLE MODE, the following program screen is displayed.
On the left of the screen, the coordinates of the current position in the absolute and relative coordinate systems are displayed; on the right of the screen, the contents of a program are displayed. You can create a program while checking the current position by manual operation.

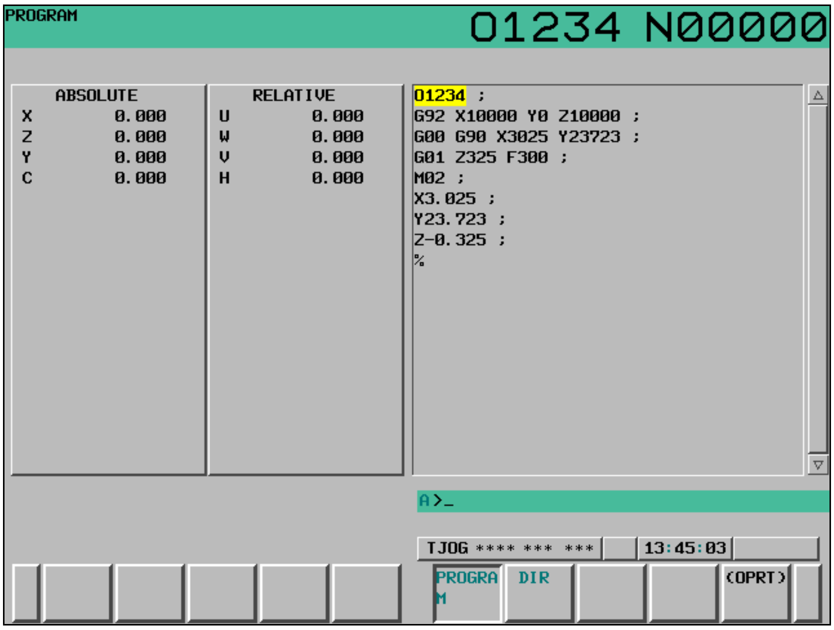


Fig. 9.3 (a) Program screen in the TEACH IN JOG mode

Inputting the coordinates of the current position

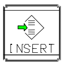
You can use the following procedure to insert the coordinate of the current position along each axis in the absolute coordinate system:

1 Select the TEACH IN JOG mode or TEACH IN HANDLE mode.

2 Press  key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the current position along each axis is to be inserted.

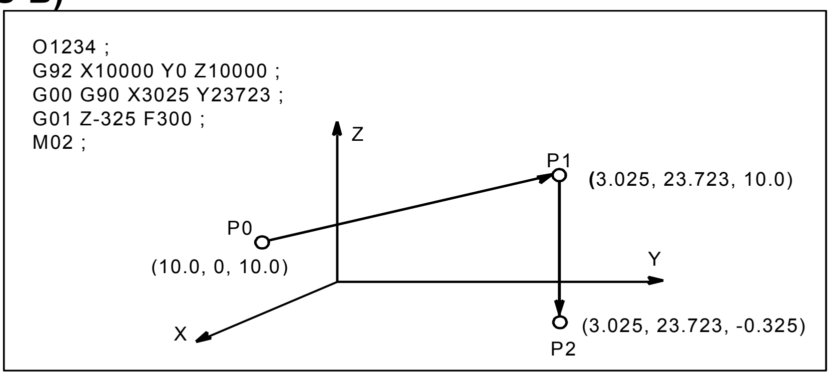
3 Move the tool to the desired position with jog or handle.

4 Key in the axis name of an axis along which you want to insert the coordinate of the current position.

5 Press the  key. Then, the coordinate of the current position along the specified axis is inserted in the program.

(Example) X10.521 Coordinate of the current position
X10521 Data inserted in the program (For IS-B)

Example (For IS-B)



1 Select the TEACH IN HANDLE mode.

2 Make positioning at position P0 by the manual pulse generator.

3 Select the program screen.

4 Enter program number O1234 as follows:

[O] [1] [2] [3] [4] [INSERT]

This operation input program number O1234 in memory.
Next, press the following keys:

[EOB E] [INSERT]

An EOB (;) is entered after program number O1234.

5 Enter the P0 machine position for data of the first block as follows:

[G] [9] [2] [INSERT] [X] [INSERT] [Y] [INSERT]
[Z] [INSERT] [EOB E] [INSERT]

This operation registers G92X10000Y0Z10000 ; in program.

6 Position the tool at P1 with the manual pulse generator.

7 Enter the P1 machine position for data of the second block as follows:

[G] [0] [0] [G] [9] [0] [INSERT]
[X] [INSERT] [Y] [INSERT] [EOB E] [INSERT]

This operation input G00G90X3025Z23723 ; in program.

8 Position the tool at P2 with the manual pulse generator.

9 Enter the P2 machine position for data of the third block as follows:

[G] [0] [1] [INSERT] [Z] [INSERT] [F] [3] [0] [0] [INSERT]
[EOB E] [INSERT]

This operation input G01Z-325F300 ; in program.

10 Input M02; in program as follows:

[M] [0] [2] [INSERT] [EOB E] [INSERT]

This completes the registration of the sample program.

Explanation

- Registering a position with compensation

When an axis name and a numeric value are keyed in and the [INSERT] key is pressed, the value keyed in is added to the absolute coordinate of the current position and the coordinate is inserted. This operation allows insertion of a corrected value for the absolute coordinate position.

- Registering commands other than position commands

Commands to be entered before and after a position command must be entered before and after the machine position is registered, by using the same operation as program editing in EDIT mode.

- Pocket calculator type input

When the pocket calculator type input format is disabled (bit 0 (DPI) of parameter No. 3401 is set to 0), the coordinate of the current position is inserted into the program in least input increments. When the pocket calculator type input format is enabled (the bit is set to 1), the coordinate is inserted with a decimal point.

(Example)

Coordinate of the current position	X10.521
At this time, the X-axis coordinate is inserted into the program as follows (For IS-B):	
When the pocket calculator type input format is disabled	X10521
When the pocket calculator type input format is enabled	X10.521

9.4 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

NC programs can be created on a block-by-block basis, viewing the displayed G code menu screen and G code details screen.

Procedure for Conversational Programming with Graphic Function

Procedure 1: Creating a program

1 Enter the EDIT mode.

2 Press function key 

. If no program is registered, the following screen is displayed.

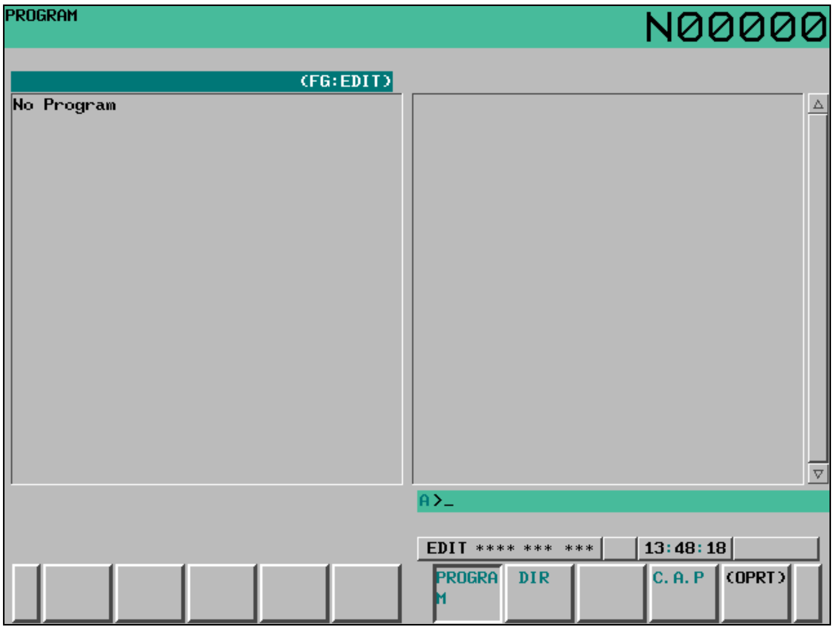


Fig. 9.4 (a) Program screen (with no program registered)

If a program is registered, the program currently selected is displayed.

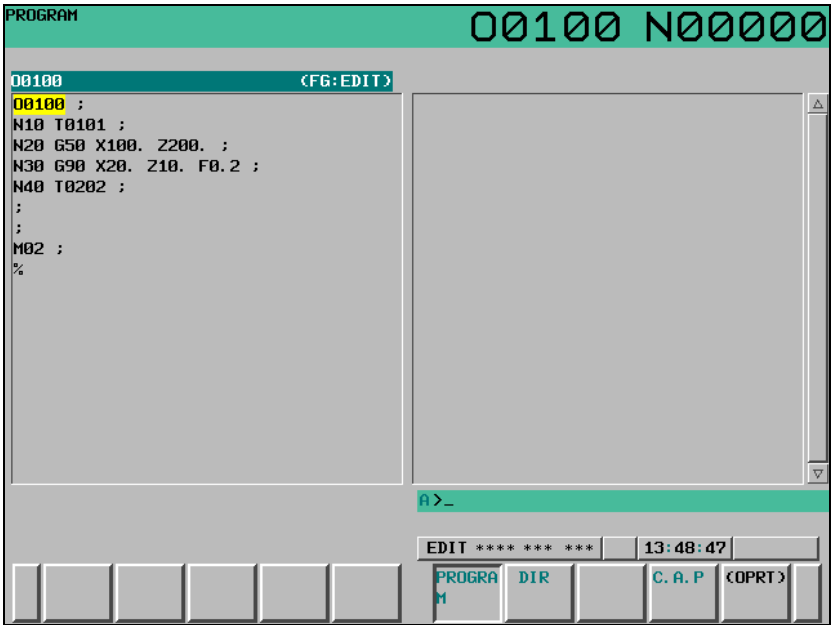


Fig. 9.4 (b) Program screen (with a program registered)

3 Key in the program number of a program to be registered after keying in address O, then press INSERT key. For example, when a program with program number 10 is to be registered, key in O10 , then press INSERT key. This registers a new program O0010.

4 Press the [C.A.P] soft key. The following G code menu is displayed on the screen. (If the [C.A.P] soft key is not displayed, hold down the rightmost continuous menu key ▷ or the leftmost return menu key ◁ until the [C.A.P] soft key is displayed.)

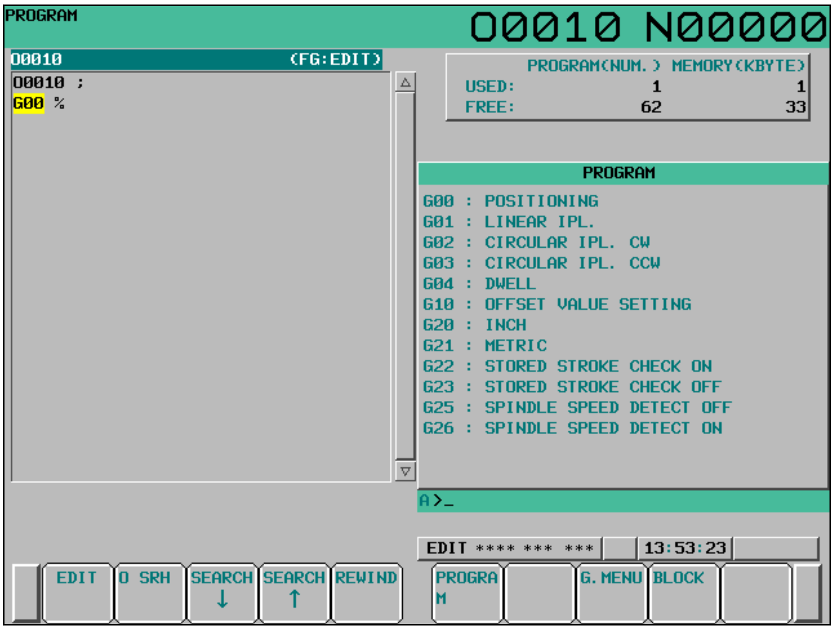


Fig. 9.4 (c) G code menu screen

5 Key in the G code corresponding to a function to be programmed. When the positioning function is desired, for example, the G code menu lists the function with the G code G00. So key in G00. If the screen does not indicate a function to be programmed, press the page key to display the next G code menu screen. Repeat this operation until a desired function appears. If a desired function is not a G code, key in no data.

6 Press the soft key [BLOCK] to display a detailed screen for a keyed in G code. The figure below shows an example of detailed screen for G00.

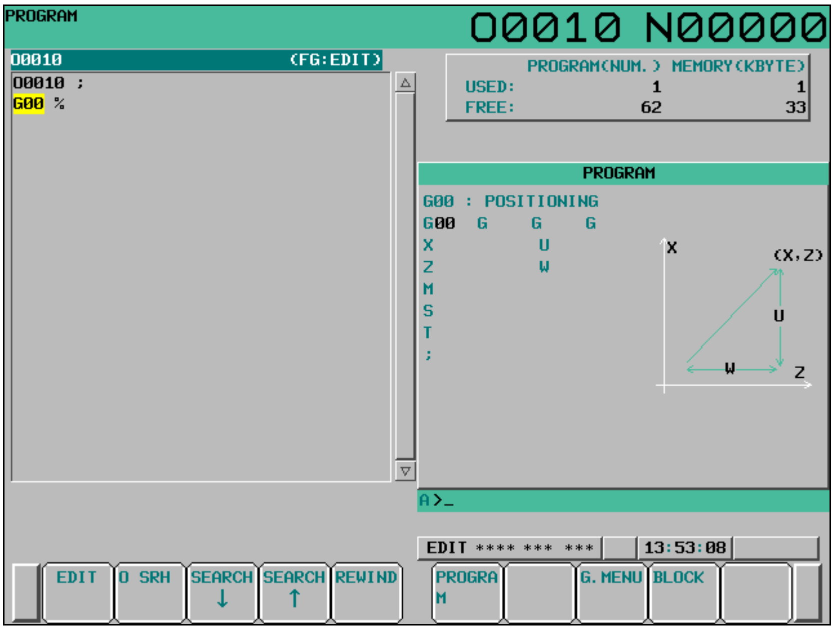


Fig. 9.4 (d) G code details screen (G00)

When no keys are pressed, the standard details screen is displayed.

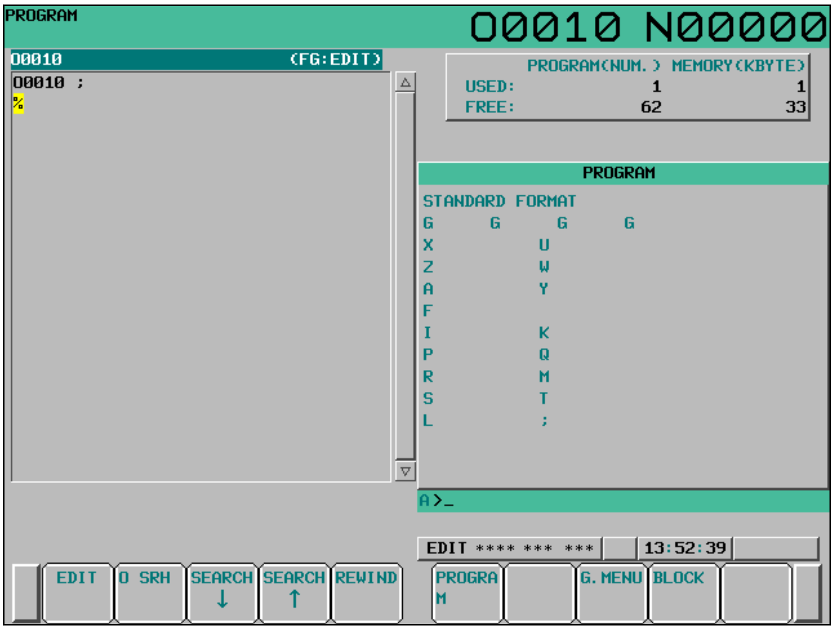
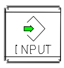


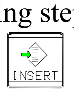
Fig. 9.4 (e) G code details screen (standard details screen)

7 Move the cursor to the block to be modified on the program screen. If the address where the cursor is placed on the program side is present on the figure side, the address on the figure side blinks.

8 Enter numeric data by pressing the numeric keys and press the [INPUT] soft key or 

key. This completes the input of one data item.

9 By repeating steps 7 and 8 above, enter all data necessary for the G code keyed in step 5 above.

10 Press the 

key. This completes the registration of data of one block in program memory. On the screen, the G code menu screen is displayed, allowing the user to enter data for another block. Repeat the procedure starting with 5 as required.

11 After registering all programs, press the [PRGRM] soft key. The registered programs are converted to the conversational format and displayed.

12 Press the RESET key to return to the program head.

Procedure 2: Modifying a block

1 Move the cursor to the block to be modified on the program screen and press the [C.A.P] soft key. Or, press the [C.A.P] soft key first to display the conversational screen, then press the ↑ PAGE or PAGE ↓ page key until the block to be modified is displayed.

2 When data other than a G code is to be altered, just move the cursor to the data and key in a desired value, then press the [INPUT] soft key or INPUT key.

3 When a G code is to be altered, press the menu return key ◁ and the soft key [G.MENU]. Then the G code menu appears. Select a desired G code, then key in the value. For example, to specify a cutting feed, since the G code menu indicates G01, key in G01. Then press the soft key [BLOCK]. The detailed screen of the G code is displayed, so enter the data.

4 After data is changed completely, press the CALC ALTER key. This operation replaces an entire block of a program.

Procedure 3: Inserting a block

1 On the conversational screen, display the block immediately before a new block is to be inserted, by using the page keys. On the program screen, move the cursor with the page keys and cursor keys to immediately before the point where a new block is to be inserted.

2 Press the soft key [G.MENU] to display the G code menu. Then enter new block data.

3 When input of one block of data is completed in step 2, press the INSERT key. This operation inserts a block of data.

Procedure 4: Deleting a block

1 On the conversational screen, display the contents of a block to be deleted, then press the DELETE key.

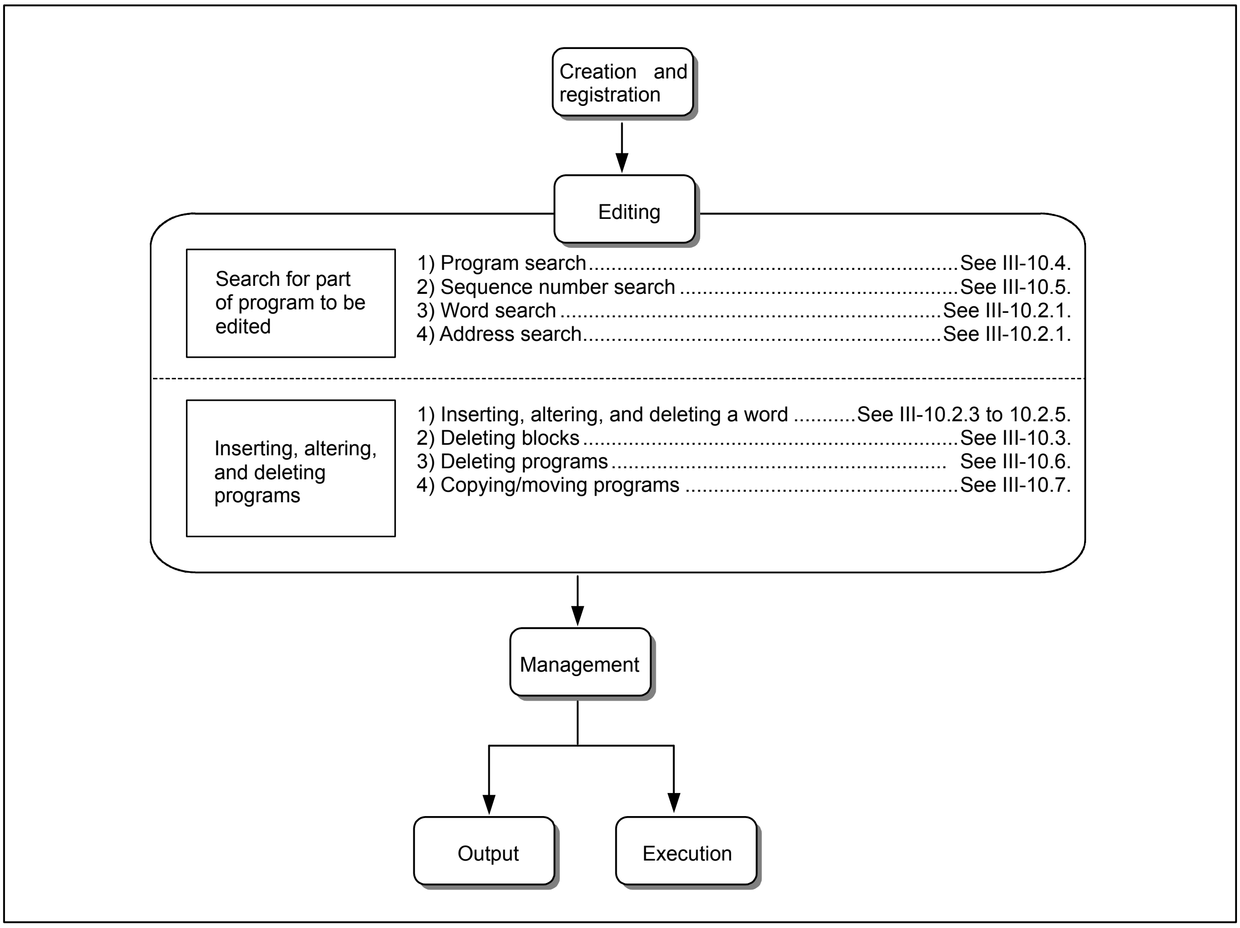
2 The contents of the block displayed are deleted from program memory. Then the contents of the next block are displayed on the conversational screen.

Notes

With a machining center system, the names of the basic three axes are X, Y, and Z at all times.
With a lath system, the names of the basic two axes are X and Z at all times.

10 EDITING PROGRAMS

This chapter describes how to edit programs registered in the CNC. Editing includes the insertion, modification, and deletion of words. Editing also includes deletion of the entire program and automatic insertion of sequence numbers. This chapter also describes program search, sequence number search, word search, and address search, which are performed before editing the program.



10.1 EDIT DISABLE ATTRIBUTE

When the 8-level data protection function is used, the edit disable attribute can be set for each program.
Programs with the edit disable attribute cannot be edited.
So, before a program can be edited, the edit disable attribute, if set for the program, needs to be removed.

NOTE
The 8-level data protection function is optional.

Procedure for removing the edit disable attribute

1 Select EDIT mode.
2 Press the function key 

.
3 Press the soft key [DIR].

4 Press the soft key [DIR+] to display a detailed program list.
(Each time the soft key [DIR+] is pressed, the program list display switches between detailed display and normal display.)
5 Select a program whose edit disable attribute is to be removed.
Move the cursor with the cursor key ⬇ or ⬆ to a program whose edit disable attribute is to be removed.
6 Press the soft key [(OPRT)].
7 Press the soft key [CHANGEATTRIB] (or [ATTRIB] when an 8.4-inch display unit is used).
8 Press the soft key [EDIT ENABLE] (or the soft key [EDT ON] when an 8.4-inch display unit is used).
9 Press the soft key [END].

> ⚠ **CAUTION**
> 1 After completing editing, set the edit disable attribute as necessary.
> 2 To set the edit disable attribute, follow the same procedure as for removing the attribute.
> In the step 7, press the soft key [EDIT DISABL].

10.2 INSERTING, ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, altering, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

1 Select EDIT mode.
2 Press the function key PROG.
3 Select a program to be edited.
If a program to be edited is selected, perform the operation 4.
If a program to be edited is not selected, search for the program number.
4 Move the cursor to the position to be edited, with one of the methods below.
- Scan method
- Word search method
- Address search method

For each method, refer to III-10.1.1.
5 Perform an operation such as altering, inserting, or deleting a word.

Explanation

- Concept of word and editing unit

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous.
So the editing unit is considered here.
The editing unit is a unit subject to alteration or deletion in one operation. In one scan operation, the cursor indicates the start of an editing unit.
An insertion is made after an editing unit.
Definition of editing unit

- Program portion from an address to immediately before the next address
- An address is an alphabet, IF, WHILE, GOTO, END, DO=, or ; (EOB).

According to this definition, a word is an editing unit.
For convenience in the explanation of editing below, the "editing unit" is referred to as the "word".

> ⚠ **WARNING**
> When a change, insertion, or deletion was performed on data of a program by pausing machining with the single block stop, feed hold, or other operations during execution of a program, be sure to return the cursor to the original position before restarting the program. To execute the program with the cursor positioned at another position, be sure to make a reset.
> Otherwise, the program may not be executed as expected from the program shown on the screen after machining restarts.

10.2.1 Word Search

For convenience, selecting a word with the cursor is referred to as scanning a word.
A word can be scanned by moving the cursor to the word, by performing a word search operation, or by performing an address search operation.
For convenience in the explanation below, selecting a word by performing a word search operation or address search operation may be referred to as searching for a word.

Procedure for scanning a program (cursor move operation)

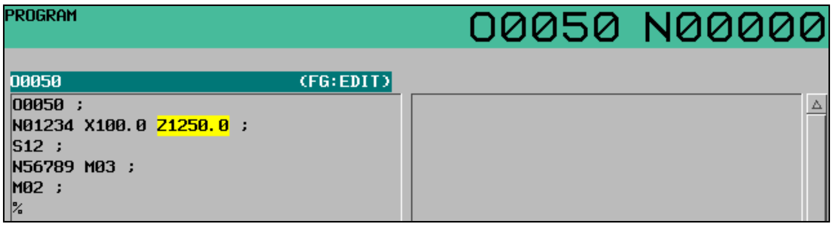
Move the cursor word by word.
Move the cursor according to the procedure below to scan a desired word.

1 Press the cursor key [→].

The cursor moves forward word by word; the cursor is displayed at a selected word.

2 Press the cursor key [←].

The cursor moves backward word by word; the cursor is displayed at a selected word.
Example: When the cursor moves to Z1250.0



3 Holding down the cursor key [→] or [←] moves words continuously.

4 Pressing the cursor key [↓] moves the cursor to the first word of the next block.

5 Pressing the cursor key [↑] moves the cursor to the first word of the previous block.

6 Holding down the cursor key [↓] or [↑] moves the cursor to the head of a block continuously.

7 Pressing the page key [PAGE ↓] changes the screen display to the next page and moves the cursor to the first word.

8 Pressing the page key [↑ PAGE] changes the screen display to the previous page and moves the cursor to the first word.

9 Holding down the page key [PAGE ↓] or [↑ PAGE] displays one page after another.

Procedure for searching a word

- Search using a soft key

1. Select EDIT or MEMORY mode.
2. Press function key PROG.
3. Key in a word to be found.
4. Pressing soft key [SRH↓] makes a word search in the forward direction from the cursor position.
5. If the program includes the word to be found, the cursor moves to the word. If the word cannot be found when a search has been made up to the end of the program, the cursor moves to the end of the program and the warning message "THE STRING IS NOT FOUND" is displayed.
6. Pressing soft key [SRH↓] again makes another search for the same word.
7. To search for a different word, enter the word in the key-in buffer then press soft key [SRH↓].
8. Pressing soft key [SRH↑] makes a search in the reverse direction.

- Search using the cursor keys

1. Select EDIT or MEMORY mode.
2. Press function key PROG.
3. Key in a word to be found.
4. Pressing cursor key ↓ makes a word search in the forward direction.
5. If the program includes the word to be found, the cursor moves to the word. If the word cannot be found, the warning message "THE STRING IS NOT FOUND" is displayed.

Pressing cursor key ↑ after keying in a word makes a search in the reverse direction.

> **NOTE**
> 1 To search for a word, the character string that completely matches the word to be found must be specified.
> Example)
> X100.0 cannot be found by specifying X1. Specify X100.0.
> G01 cannot be found by specifying G1. Specify G01.
> 2 Unlike word search using a soft key, word search using the cursor keys demands that a word to be found be set each time.

Procedure for searching an address

- Search using a soft key

1. Select EDIT or MEMORY mode.
2. Press function key PROG.
3. Key in an address to be found.
4. Pressing soft key [SRH↓] makes an address search in the forward direction from the cursor position.
5. If the program includes the word containing the set address, the cursor moves to the word. If the word containing the set address cannot be found when a search has been made up to the end of the program, the cursor moves to the end of the program and the warning message "THE STRING IS NOT FOUND" is displayed.
6. Pressing soft key [SRH↓] again makes another search for the same address.
7. To search for a different address, enter the address in the key-in buffer then press soft key [SRH↓].
8. Pressing soft key [SRH↑] makes a search in the reverse direction.

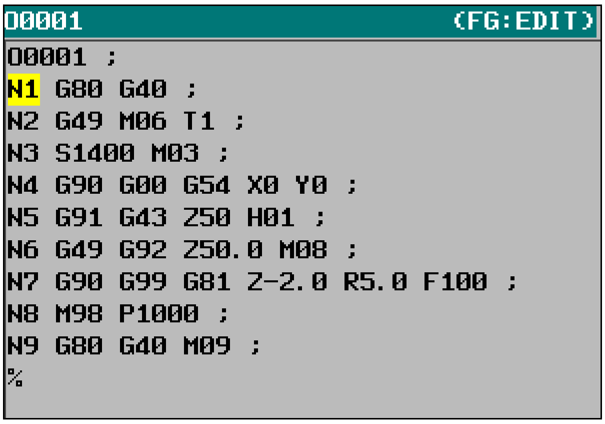
- Search using the cursor keys

1. Select EDIT or MEMORY mode.

2. Press function key .
3. Key in an address to be found.
4. Pressing cursor key makes an address search in the forward direction.
5. If the program includes the word containing the set address, the cursor moves to the word. If the word containing the set address cannot be found, the cursor moves to the end of the program and the warning message "THE STRING IS NOT FOUND" is displayed.
6. Pressing cursor key after keying in an address makes a search in the reverse direction.

Example) To make an address search for "M"

1. Key in "M".
2. Pressing soft key [SRH↓] moves the cursor to "M06" placed on the same line as N2.
3. Pressing soft key [SRH↓] again moves the cursor to "M03" placed on the same line as N3.
4. Pressing soft key [SRH↑] moves the cursor to "M06" placed on the same line as N2.
5. Pressing cursor key after keying in "M" moves the cursor to "M03" placed on the same line as N3.
6. If continuation of search operation in the forward direction finds no more address, the cursor moves to the last line and the warning message "THE STRING IS NOT FOUND" is displayed.



NOTE

Unlike address search using a soft key, address search using the cursor keys demands that an address to be found be set each time.

10.2.2 Heading a Program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the four methods for heading the program pointer.

Procedure for heading a program

Method 1

1 Press key when the program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2
1 Search for the program number.
When the program screen is selected in MEMORY or EDIT mode, enter a program number.
(Press address key O then type the program number.)
2 Press soft key [O SEARCH].

Method 3
1 Select the program screen or program check screen in MEMORY mode.
2 Press soft key [(OPRT)].
3 Press soft key [REWIND].

Method 4
1 Select the program screen in EDIT mode.
2 Press soft key [(OPRT)].
3 Press soft key [REWIND].

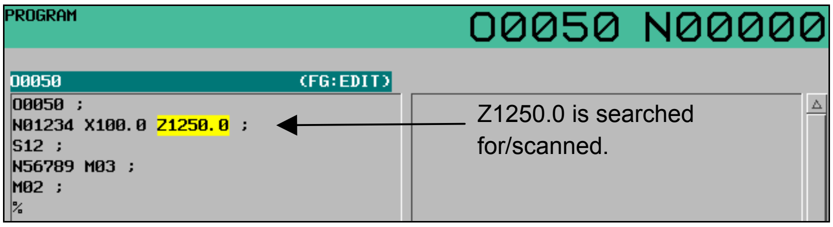
10.2.3 Inserting a Word

Procedure for inserting a word

1 Search for or scan the word immediately before a word to be inserted.
2 Key in an address to be inserted.
3 Key in data.
4 Press the INSERT key.

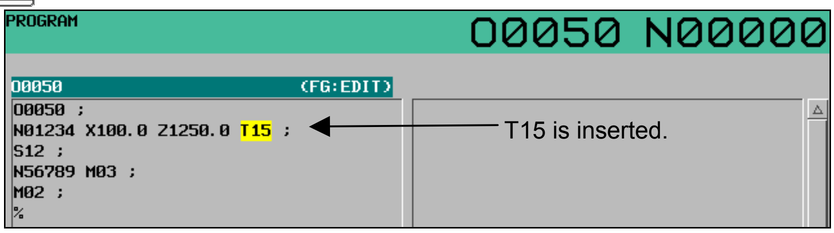
Example of Inserting T15

1 Search for or scan Z1250.



2 Key in T 1 5 .

3 Press the INSERT key.



10.2.4 Altering a Word

Procedure for altering a word

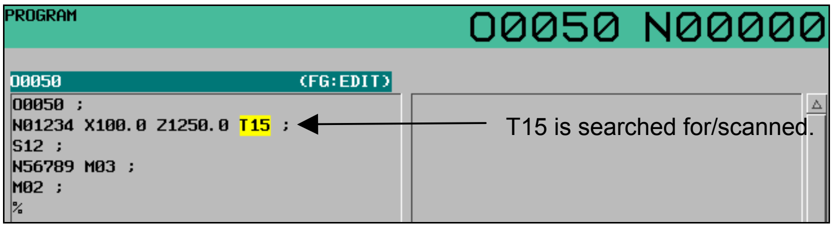
1 Search for or scan a word to be altered.
2 Key in an address to be inserted.

3 Key in data.

4 Press the ALTER key.

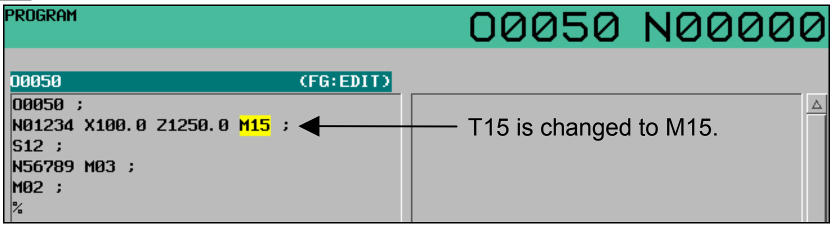
Example of changing T15 to M15

1 Search for or scan T15.



2 Key in M 1 5 .

3 Press the ALTER key.



10.2.5 Deleting a Word

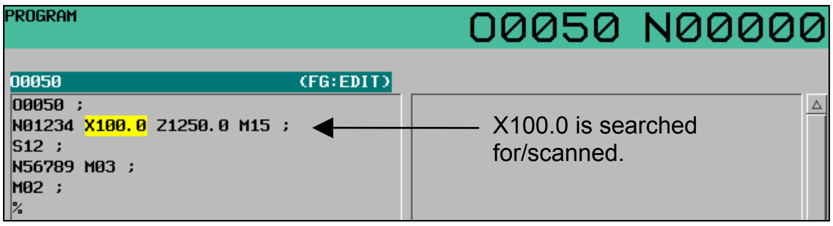
Procedure for deleting a word

1 Search for or scan a word to be deleted.

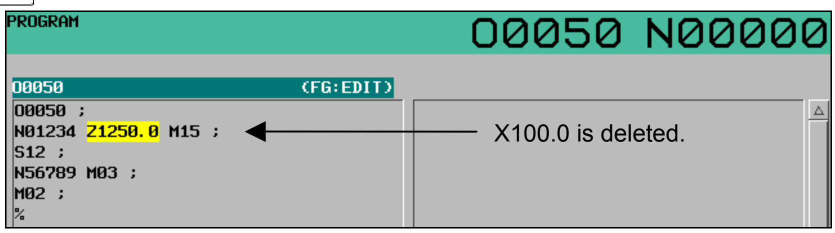
2 Press the DELETE key.

Example of deleting X100.0

1 Search for or scan X100.0.



2 Press the DELETE key.



10.3 DELETING BLOCKS

A block or blocks can be deleted in a program.

10.3.1 Deleting a Block

The portion from the current word position to the next EOB is deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting a block

1 Search for or scan address N for a block to be deleted.

2 Press the EOB key.

3 Press the editing key DELETE.

Example of deleting a block of N01234

1 Search for or scan N01234.

```
O0050 ;
N01234 Z1250.0 M15 ;
S12 ;
N56789 M03 ;
M02 ;
%
```

N01234 is searched for/scanned.

2 Press the EOB key.

3 Press the editing key DELETE.

```
O0050 ;
S12 ;
N56789 M03 ;
M02 ;
%
```

Block containing N01234 has been deleted.

10.3.2 Deleting Multiple Blocks

The several blocks in the forward direction from the current word position up to the EOB of the farthest of those blocks are deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting blocks

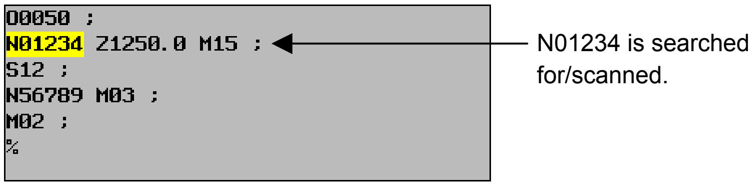
1 Search for or scan a word in the first block of a portion to be deleted.

2 Press the EOB key as many times as the number of blocks that you want to delete.

3 Press the editing key DELETE.

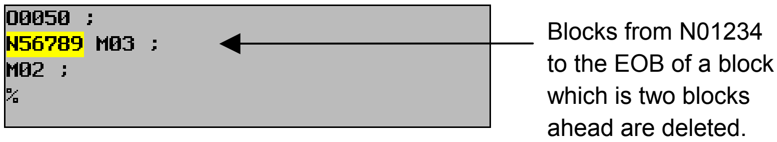
Example of deleting blocks from N01234 to the EOB of a block which is two blocks ahead

1 Search for or scan N01234.



2 Press EOB EOB.

3 Press the editing key DELETE.



10.4 PROGRAM SEARCH

When memory holds multiple programs, a program can be searched for. There are four methods as follows.

Procedure for program search

Method 1 To search for a program with this method, key in the desired program number then press operation selection soft key [O SEARCH].

1 Select EDIT or MEMORY mode.

2 Press function key PROG to display the program screen.

3 Press address key P O.

- For a program registered in the CNC memory, address key entry may be omitted.

4 Key in a program number to be found.

5 Press operation selection soft key [O SEARCH].
Upon completion of search operation, the found program number is displayed in the upper right area of the screen.
If the desired program number is not found, the warning message "SPECIFIED PROGRAM NOT FOUND" is displayed when a 5-digit or longer program number is specified for search operation.
When a character not following step 3 or 4 above is entered for search operation, the warning message "FORMAT ERROR" is displayed.

Method 2 To search for a program with this method, key in the desired program number then press cursor key ↑ or ↓.

1 Select EDIT or MEMORY mode.

2 Press function key PROG to display the program screen.

3 Press address key P O.

4 Key in a program number to be found.

5 Press cursor key ↑ or ↓.

- Pressing cursor key ↑ searches the previous program.
- Pressing cursor key ↓ searches the next program.

Upon completion of search operation, the found program number is displayed in the upper right area of the screen.
If the desired program number is not found, the warning message "SPECIFIED PROGRAM NOT FOUND" is displayed when a 5-digit or longer program number is specified for search operation.
When a character not following step 3 or 4 above is entered for search operation, the warning message "FORMAT ERROR" is displayed.

Method 3 This method searches the next program from the current program.

1 Select EDIT or MEMORY mode.
2 Press function key to display the program screen.
3 Press operation selection key [O SEARCH].
4 Upon completion of search operation, the found program number is displayed in the upper right area of the screen.

Method 4 To search for a program, press address key then press cursor key or .

1 Select EDIT or MEMORY mode.
2 Press function key to display the program screen.
3 Press address key .
4 Press cursor key or .
   - Pressing cursor key searches the previous program.
   - Pressing cursor key searches the next program.

   Upon completion of search operation, the found program number is displayed in the upper right area of the screen.

10.5 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example)
Sequence number 02346 in a program (O0002) is searched for.

```
                            PROGRAM
                            O0001 ;
                            N01234 X100.0 Z100.0 ;
                            S12 ;
                              :
Selected program      -->   O0002 ;                    } This section is searched
                            N02345 X20.0 Z20.0 ;       } starting at the beginning.
Target sequence       -->   N02346 X10.0 Z10.0 ;       } (Search operation is
number is found.              :                          performed only within a
                            O0003 ;                      program.)
                              :
```

Procedure for sequence number search

Procedure

1 Select MEMORY mode.

2 Press function key 

.

3 If the program contains a sequence number to be searched for, perform the operations 4 to 7 below.
If the program does not contain a sequence number to be searched for, select the program number of the program that contains the sequence number to be searched for.

4 Press address key 

.

5 Key in a sequence number to be searched for.

6 Press soft key [N SRH].

7 Upon completion of search operation, the sequence number searched for is displayed in the upper-right corner of the screen.
If the specified sequence number is not found in the program currently selected, alarm PS0060 occurs.

Explanation

- Operation during Search

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC states at that point.

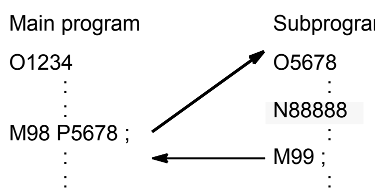
- When the optional block skip function is enabled

When the optional block skip function is enabled for a block, the sequence numbers included in the block are excluded as sequence number search targets.

Limitation

- Searching in sub-program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm PS0060 is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.



If an attempt is made to search for N8888 in the example above, an alarm occurs.

Fig. 10.5 (a)

10.6 DELETING PROGRAMS

Programs registered in memory can be deleted, either one program by one program or all at once.

10.6.1 Deleting One Program

A single program can be deleted.

Procedure for deleting one program

1 Select the EDIT mode.
2 Press function key [PROG] to display the program screen.
3 Press address key [O P].
4 Key in a desired program number.
5 Press the editing key [DELETE]. The program with the entered program number is deleted.

10.6.2 Deleting All Programs

All programs can be deleted.

Procedure for deleting all programs

1 Select the EDIT mode.
2 Press function key [PROG] to display the program screen.
3 Press address key [O P].
4 Key in -9999.
5 Press editing key [DELETE] to delete all programs.

10.7 COPYING/MOVING PROGRAMS

An entire program or a part of a program can be copied or moved.

10.7.1 Copying a Part of a Program

A part of a program being displayed can be copied and pasted to another area.

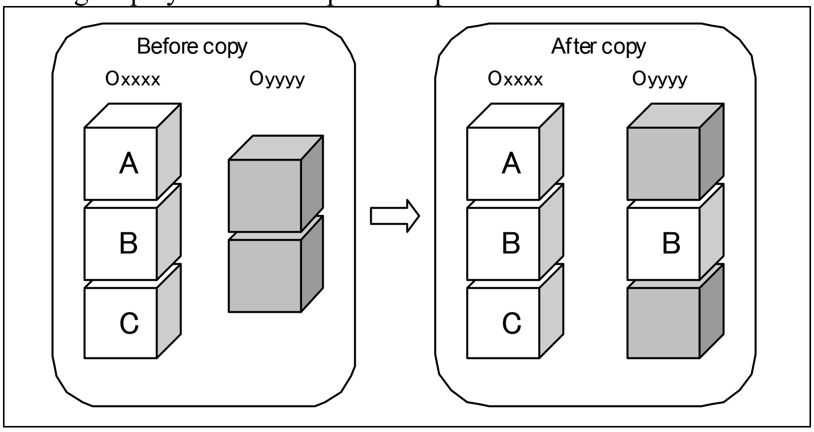
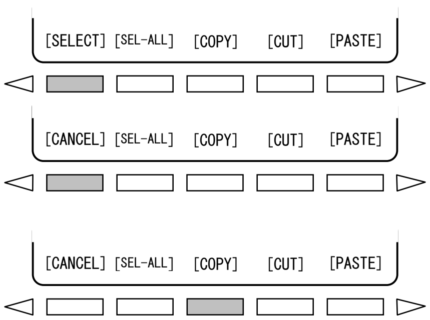


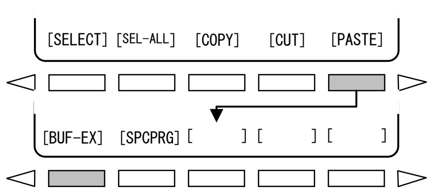
Fig. 10.7.1 (a) Copying a part of a program

In Fig. 10.7.1 (a), program B of program number xxxx is copied and pasted into the program of program number yyyy. The program of program number xxxx remains unchanged before and after the copy operation.

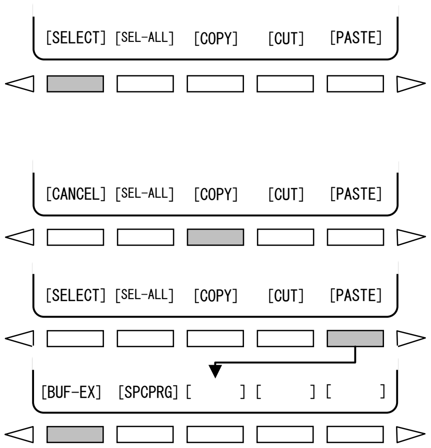
Procedure for copying a part of a program



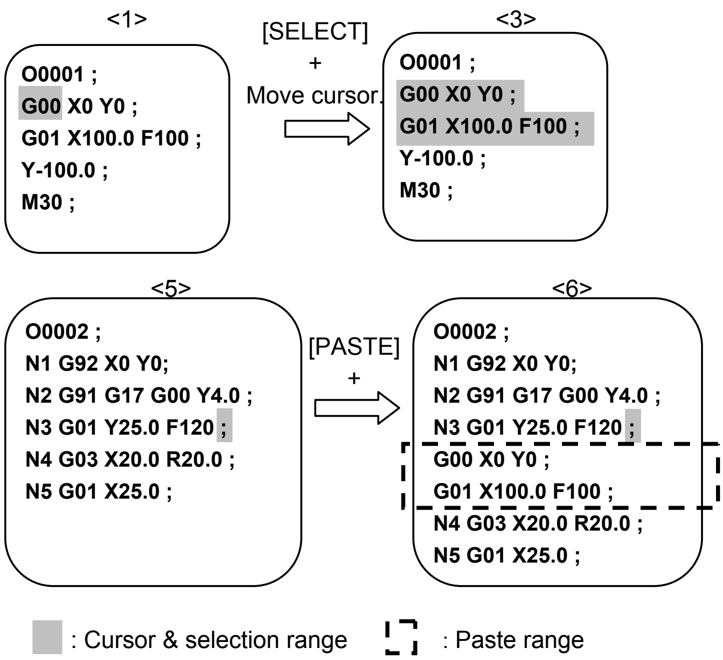
1. Move the cursor to a desired copy start position.
2. Press soft key [SELECT].
3. When the cursor is moved, the range from the copy start position to the current cursor position is selected and displayed in the same color as the cursor color.
   To cancel the selection, press soft key [CANCEL].
4. Press soft key [COPY]. The selected program range is stored in the copy buffer.
   If the size of the selected program range exceeds the capacity of the copy buffer, the selection is terminated with the warning message "EXCEED THE CAPACITY OF COPY BUFFER".
   If soft key [COPY] is pressed with no program range selected, the warning message "WORDS ARE NOT SELECTED" is displayed and the copy buffer is cleared. If the program size of the selected range exceeds the upper limit of the copy buffer, the warning message "EXCEED THE CAPACITY OF COPY BUFFER" is displayed. The selected range is not released.
5. Search for a paste target program.
6. Move the cursor to the desired paste position.
7. Press soft key [PASTE] then press soft key [BUF-EX]. The contents copied in step 4 are pasted after the cursor. If [BUF-EX] is pressed when the copy buffer is empty, the warning message "COPY BUFFER IS EMPTY" is displayed.



Example 1) A part of O0001 is copied to O0002.

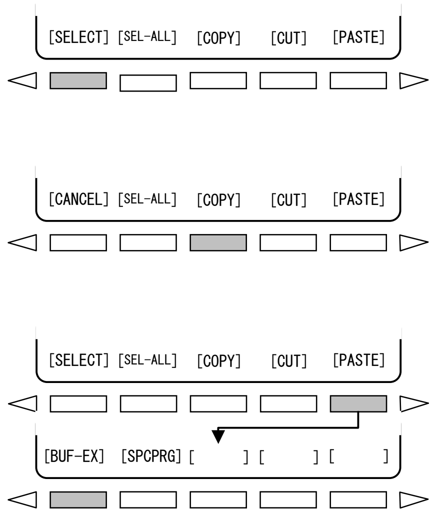


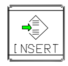
1. Display O0001 then move the cursor to a desired copy start position. (<1>)
2. Press soft key [SELECT].
3. When the cursor is moved, the range from the copy start position to the current cursor position is selected and displayed in the same color as the cursor color. (<3>)
4. Press soft key [COPY]. The selected program range is stored in the copy buffer.
5. Search for O0002 then move the cursor to the position where the contents copied from O0001 are to be inserted. (<5>)
6. Press soft key [PASTE] then press soft key [BUF-EX]. The contents copied from O0001 are pasted after the cursor. (<6>)

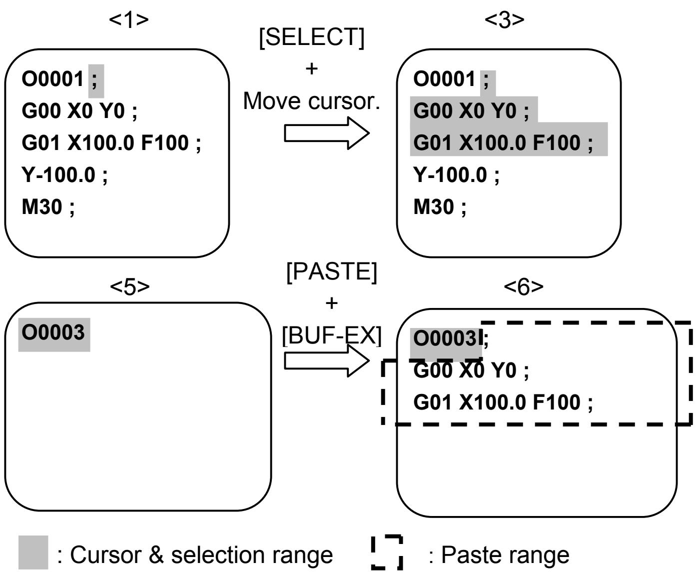


For convenience of explanation, the cursor of <6> is placed at the same position as for <5>.
Actually, the cursor moves to the last word of pasted contents.

Example 2) A part of O0001 is copied to create O0003 newly.



1. Display O0001 then move the cursor to a desired copy start position. (<1>)
2. Press soft key [SELECT].
3. When the cursor is moved, the range from the copy start position to the current cursor position is selected and displayed in the same color as the cursor color. (<3>)
4. Press soft key [COPY]. The selected program range is stored in the copy buffer.
5. Key in "O0003" then press editing key  to create the new program O0003. (<5>)
6. Press soft key [PASTE] then press soft key [BUF-EX]. The contents copied from O0001 are pasted. (<6>)



For convenience of explanation, the cursor of <6> is placed at the same position as for <5>.
Actually, the cursor moves to the last word of pasted contents.

10.7.2 Moving a Part of a Program

A part of a program being displayed can be cut and pasted to another area.

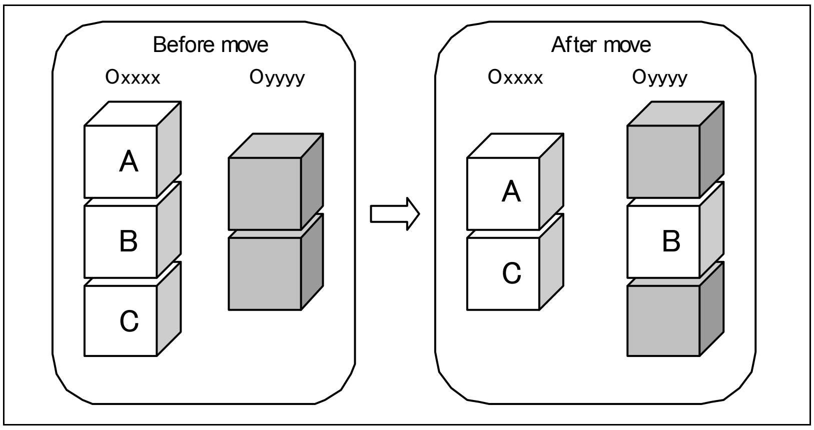
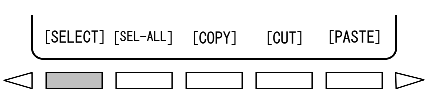


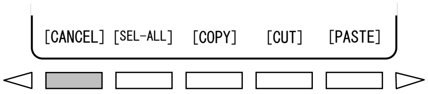
Fig. 10.7.2 (a) Moving a part of a program

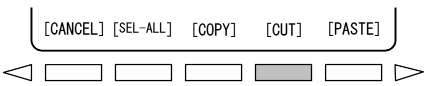
In Fig. 10.7.2 (a), program B of program number xxxx is cut and pasted into the program of program number yyyy.

Procedure for moving a part of a program

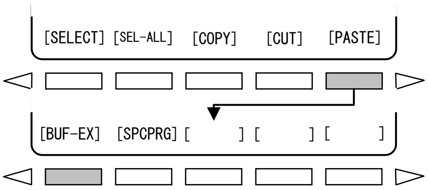


1. Move the cursor to a desired cut start position.
2. Press soft key [SELECT].



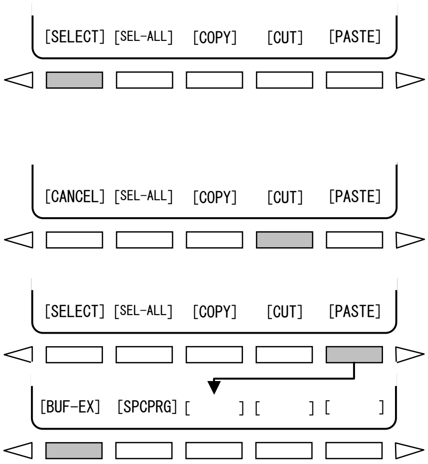


3. When the cursor is moved, the range from the cut start position to the current cursor position is selected and displayed in the same color as the cursor color.
   To cancel the selection, press soft key [CANCEL].
4. Press soft key [CUT]. The selected program range is stored in the copy buffer. At the same time, the selected program range is cut. After the cutting, the cursor moves to before the cut range.
   If soft key [CUT] is pressed with no program range selected, the warning message "WORDS ARE NOT SELECTED" is displayed and the copy buffer is cleared.
   If the program size of the selected range exceeds the upper limit of the copy buffer, the warning message "EXCEED THE CAPACITY OF COPY BUFFER" is displayed. The selected range is not released. In this case, no cut operation is performed.
5. Search for a paste target program.
6. Move the cursor to the desired paste position.

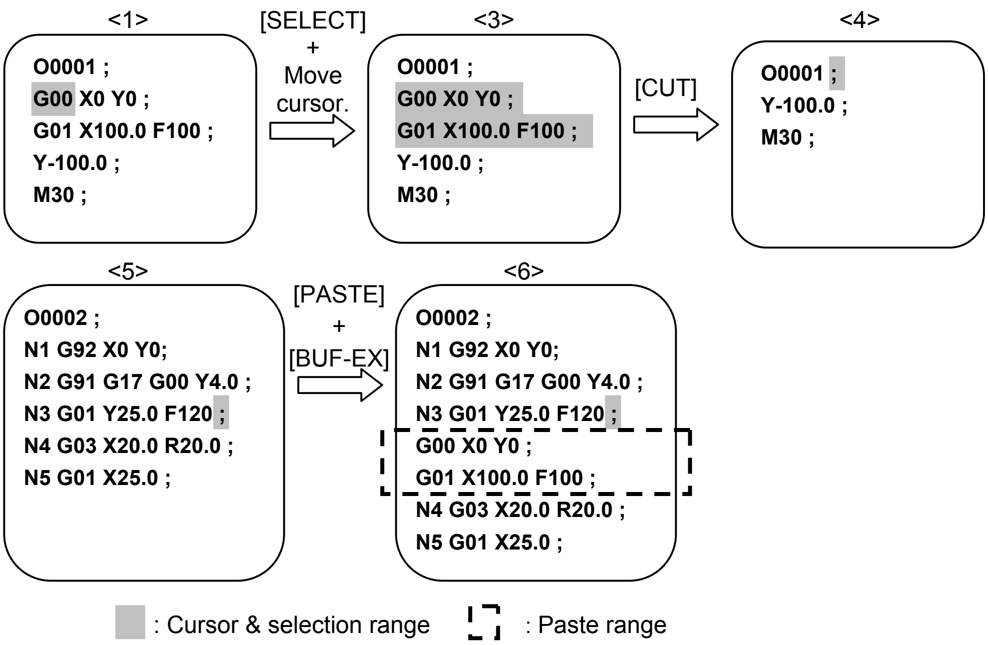


7. Press soft key [PASTE] then press soft key [BUF-EX]. The program range cut in step 4 is pasted after the cursor. If [BUF-EX] is pressed when the copy buffer is empty, the warning message "COPY BUFFER IS EMPTY" is displayed.

Example 1) A part of O0001 is moved to O0002.

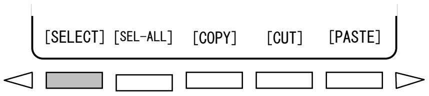


1. Display O0001 then move the cursor to a desired cut start position. (<1>)
2. Press soft key [SELECT].
3. When the cursor is moved, the range from the cut start position to the current cursor position is selected and displayed in the same color as the cursor color. (<3>)
4. Press soft key [CUT]. The selected program range is cut. (<4>)
5. Search for O0002 then move the cursor to the position where the contents cut from O0001 are to be inserted. (<5>)
6. Press soft key [PASTE] then press soft key [BUF-EX]. The contents cut from O0001 are pasted after the cursor. (<6>)

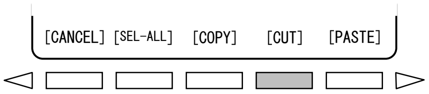


For convenience of explanation, the cursor of <6> is placed at the same position as for <5>.
Actually, the cursor moves to the last word of pasted contents.

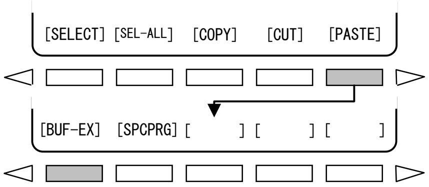
Example 2) A part of O0001 is cut to create O0003 newly.

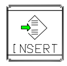


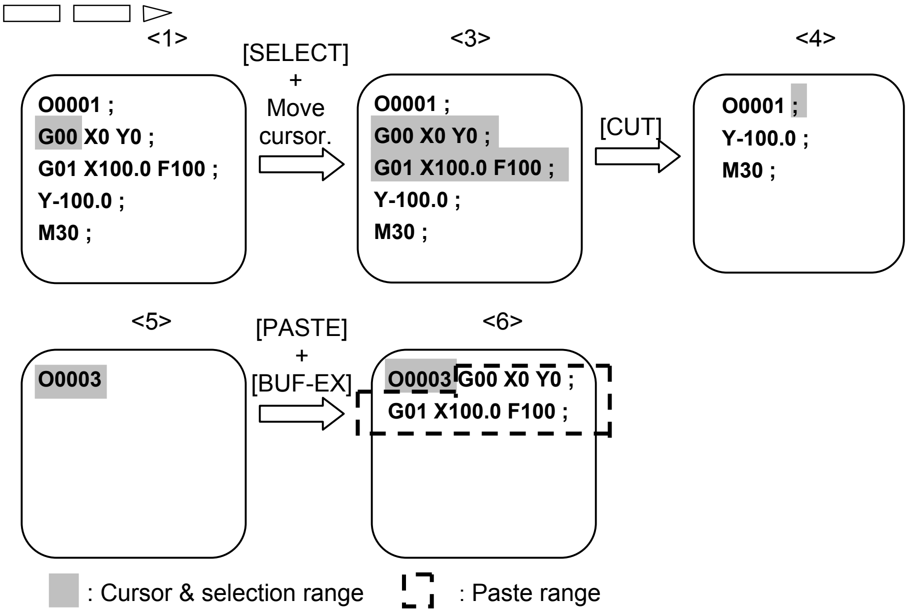
1. Display O0001 then move the cursor to a desired cut start position. (<1>)
2. Press soft key [SELECT].



3. When the cursor is moved, the range from the cut start position to the current cursor position is selected and displayed in the same color as the cursor color. (<3>)
4. Press soft key [CUT]. The selected program range is cut. (<4>)



5. Key in "O0003" then press editing key  to create the new program O0003. (<5>)
6. Press soft key [PASTE] then press soft key [BUF-EX]. The contents cut from O0001 are pasted. (<6>)



For convenience of explanation, the cursor of <6> is placed at the same position as for <5>. Actually, the cursor moves to the last word of pasted contents.

10.7.3 Copying an Entire Program

An entire program can be copied and pasted to another area.

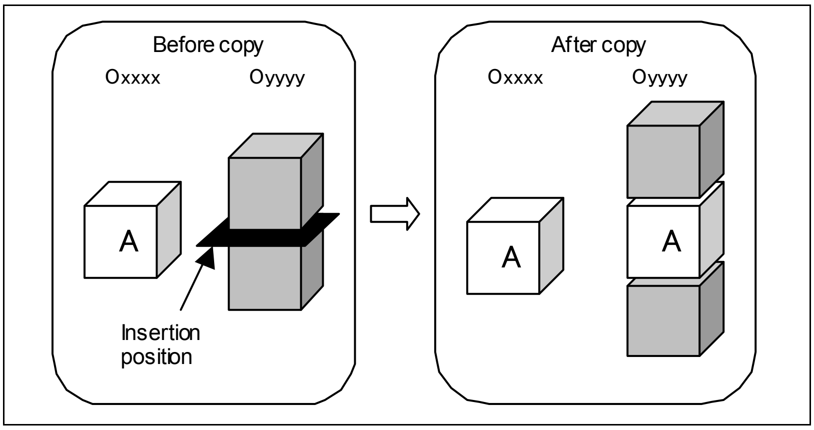
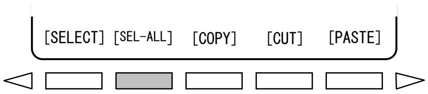


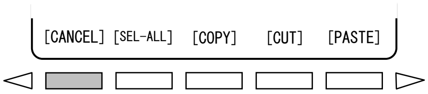
Fig. 10.7.3 (a)

In Fig. 10.7.3 (a), the program of program number xxxx is inserted into the program of program number yyyy. The program of program number xxxx remains unchanged before and after the insertion.

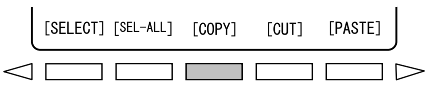
Procedure for copying an entire program



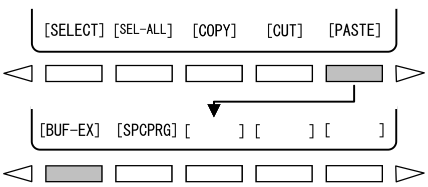
1. Display a program to be copied.
2. Press soft key [SEL-ALL]. The entire program except its O number is selected and displayed in the same color as the cursor color.



To cancel the selection, press soft key [CANCEL].

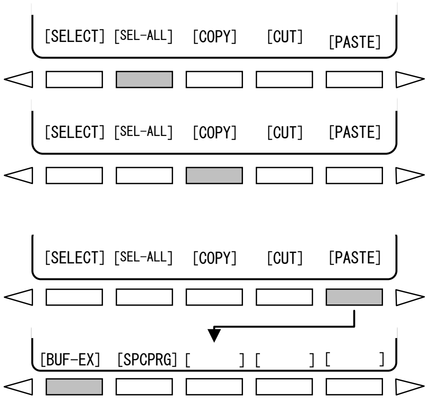


3. Press soft key [COPY]. The entire program is stored in the copy buffer.
   If soft key [COPY] is pressed with no program range selected, the warning message "WORDS ARE NOT SELECTED" is displayed and the copy buffer is cleared. If the program size of the selected range exceeds the upper limit of the copy buffer, the warning message "EXCEED THE CAPACITY OF COPY BUFFER" is displayed. The selected range is not released.
4. Search for a paste target program.

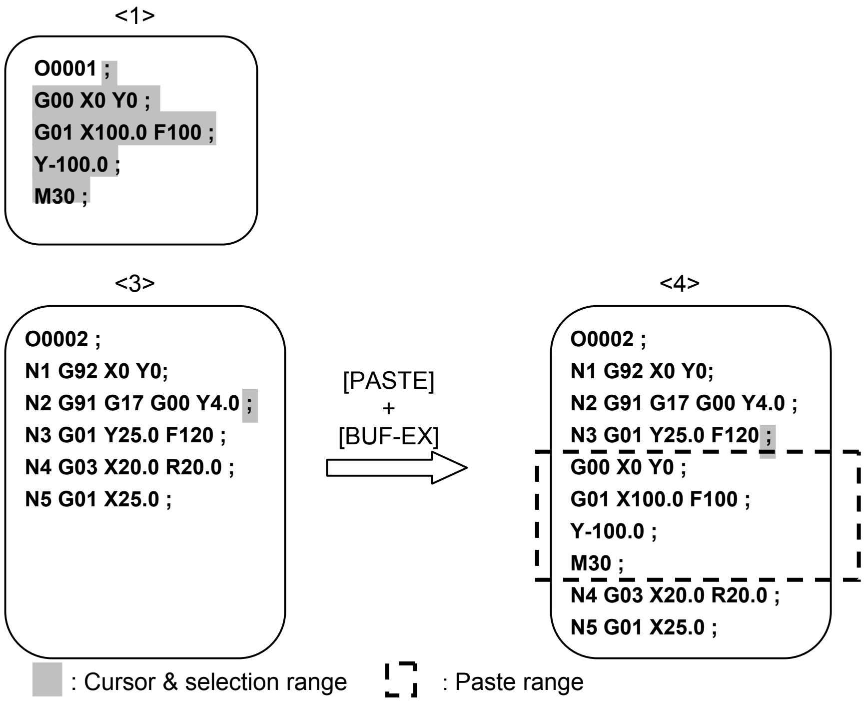


5. Move the cursor to the desired paste position.
6. Press soft key [PASTE] then press soft key [BUF-EX]. The contents copied in step 3 are pasted after the cursor. If [BUF-EX] is pressed when the copy buffer is empty, the warning message "COPY BUFFER IS EMPTY" is displayed.

Example) O0001 is copied and pasted to O0002.



1. Display O0001 then press soft key [SEL-ALL]. The entire program is selected and highlighted in the same color as the cursor color. (<1>)
2. Press soft key [COPY].
3. Display O0002 then move the cursor to the desired O0001 insertion position. (<3>)
4. Press soft key [PASTE] then press soft key [BUF-EX]. O0001 is pasted after the cursor. (<4>)



For convenience of explanation, the cursor of <4> is placed at the same position as for <3>. Actually, the cursor moves to the last word of pasted contents.

10.7.4 Moving an Entire Program

An entire program can be cut and pasted to another area.

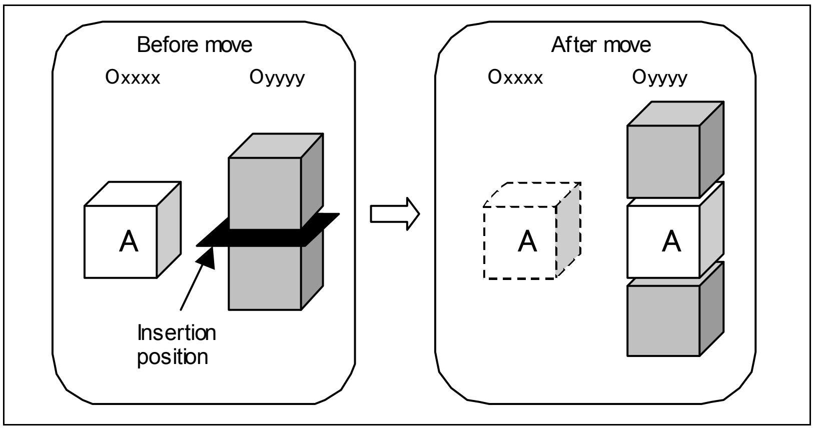
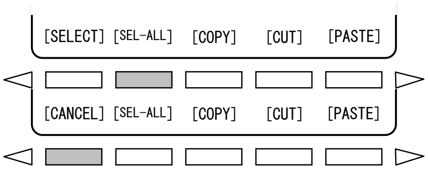


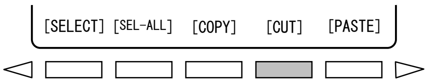
Fig. 10.7.4 (a)

In Fig. 10.7.4 (a), the program of program number xxxx is inserted into the program of program number yyyy. The contents of the program of program number xxxx are deleted. However, the deleted program remains as an empty program.

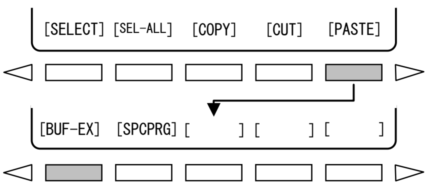
Procedure for moving an entire program



1. Display a program to be cut.
2. Press soft key [SEL-ALL]. The entire program except its O number is selected and displayed in the same color as the cursor color.
   To cancel the selection, press soft key [CANCEL].

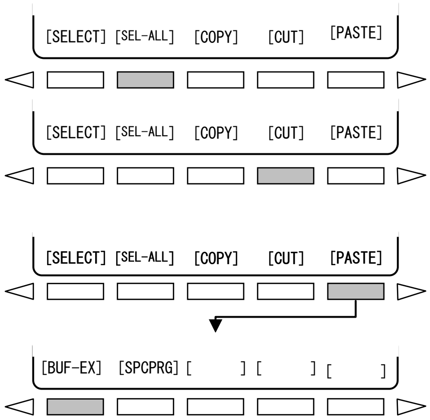


3. Press soft key [CUT]. The entire program is stored in the copy buffer. At the same time, the contents of the program are cut.
   If soft key [CUT] is pressed with no program range selected, the warning message "WORDS ARE NOT SELECTED" is displayed and the copy buffer is cleared. If the program size of the selected range exceeds the upper limit of the copy buffer, the warning message "EXCEED THE CAPACITY OF COPY BUFFER" is displayed. The selected range is not released. In this case, no cut operation is performed.
4. Search for a paste target program.
5. Move the cursor to the desired paste position.

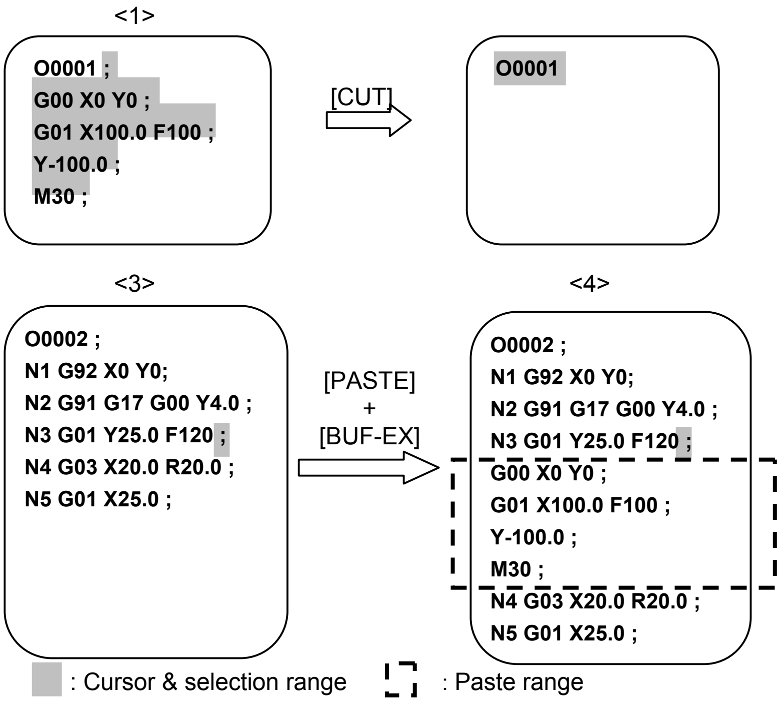


6. Press soft key [PASTE] then press soft key [BUF-EX]. The program cut in step 3 is pasted after the cursor. If [BUF-EX] is pressed when the copy buffer is empty, the warning message "COPY BUFFER IS EMPTY" is displayed.

Example) O0001 is cut and pasted to O0002.



1. Display O0001 then press soft key [SEL-ALL]. The entire program is selected and highlighted in the same color as the cursor color. (<1>)
2. Press soft key [CUT].
3. Display O0002 then move the cursor to the desired O0001 insertion position. (<3>)
4. Press soft key [PASTE] then press soft key [BUF-EX]. O0001 is pasted after the cursor. (<4>)

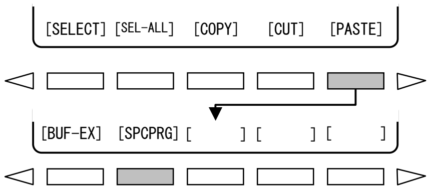


For convenience of explanation, the cursor of <4> is placed at the same position as for <3>.
Actually, the cursor moves to the last word of pasted contents.

10.7.5 Copy Specifying a Program Number

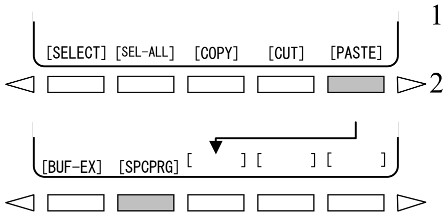
An entire program can be copied to the current cursor position by specifying its program number.
With this function, an entire program can be copied easily.
Even when the size of a program exceeds the capacity of the copy buffer, the entire program can be copied.

Procedure for copying a program by specifying its program number

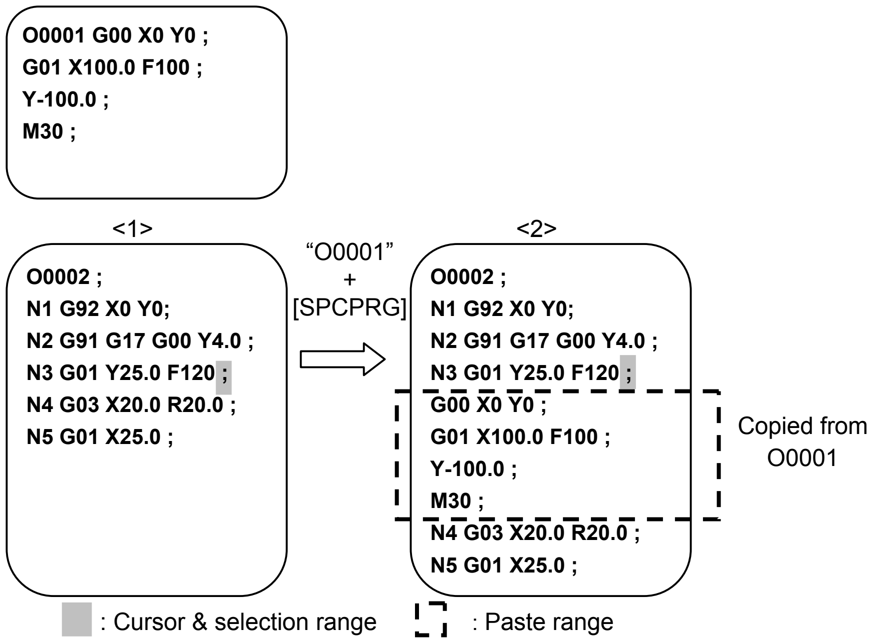


1. Display a paste target program then move the cursor immediately before the desired paste position.
2. Press soft key [PASTE] then key in the program number of a program to be pasted. Next, press [SPCPRG]. The program of the program number keyed in is pasted after the cursor. When [SPCPRG] is pressed with no program number selected, the warning message "PROGRAM IS NOT SPECIFIED" is displayed.

<u>Example) O0001 is copied and pasted to O0002.</u>



1. Search for O0002 then move the cursor to the desired O0001 insertion position. (<1>)
2. Press soft key [PASTE] then key in "O0001". Next, press [SPCPRG]. O0001 is pasted after the cursor. (<2>)

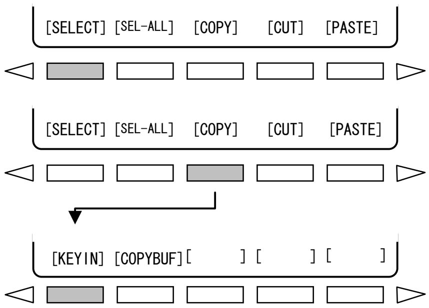


For convenience of explanation, the cursor of <2> is placed at the same position as for <1>.
Actually, the cursor moves to the last word of pasted contents.

10.7.6 Copying/Moving to the Key-in Buffer

The copy/move destination of a selected word is changed from the copy buffer to the key-in buffer. With this function, the user can perform editing while checking contents to be copied/moved.

Procedure: Copying to the key-in buffer

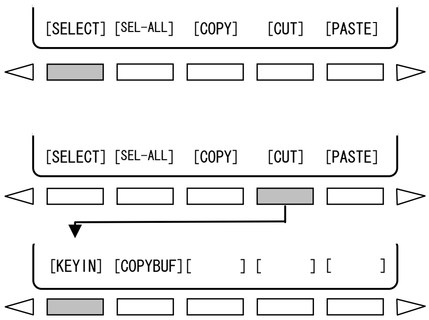


1. Set bit 2 of parameter No. 3205 to 1.
2. Display a desired program.
3. Press soft key [SELECT].
4. When the cursor is moved, the range from the copy start position to the current cursor position is selected and displayed in the same color as the cursor color.
5. If soft key [COPY] is pressed with no range selected, the warning message "WORDS ARE NOT SELECTED" is displayed and the copy buffer is cleared. Press soft key [COPY] then press soft key [KEYIN]. The selected word range is input to the key-in buffer.

NOTE

1. When performing a normal copy operation, press soft key [COPY] then press [COPYBUF].
2. Up to 127 characters can be copied. If [KEYIN] is pressed when more than 127 characters are selected, the warning message "EXCEED THE CAPACITY OF KEYIN BUFFER" is displayed.

Procedure: Moving to the key-in buffer



1. Set bit 2 of parameter No. 3205 to 1.
2. Display a desired program.
3. Press soft key [SELECT].
4. When the cursor is moved, the range from the cut start position to the current cursor position is selected and displayed in the same color as the cursor color.
5. Press soft key [CUT] then press soft key [KEYIN]. The selected word range is input to the key-in buffer.

NOTE

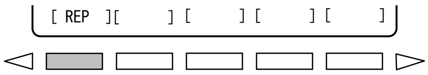
1. When performing a normal move operation, press soft key [CUT] then press [COPYBUF].
2. Up to 127 characters can be moved. If [KEYIN] is pressed when more than 127 characters are selected, the warning message "EXCEED THE CAPACITY OF KEYIN BUFFER" is displayed.

10.8 REPLACING

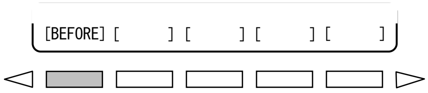
A string in a program is replaced with a specified string.

Procedure for replacing

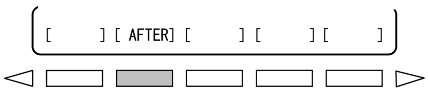
1. Enter the EDIT mode or MDI mode (MDI screen).
2. Press the function key PROG.
3. Press the soft key [(OPRT)].
4. Press the continuous menu key ▷ until soft key [REP] appears.



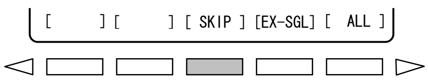
5. Press the soft key [REP].
6. Key in the word to be replaced.



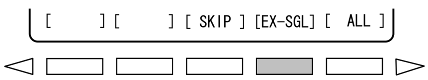
7. Press the soft key [BEFORE].
8. Key in the replacing word.



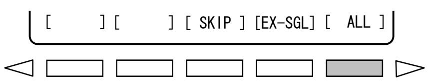
9. Press the soft key [AFTER]. The word set by soft key [BEFORE] is searched for and the cursor moves to the word.
   - If the word set by soft key [BEFORE] is not found, the warning message "THE STRING IS NOT FOUND" is displayed.
10. Press soft key [EX-SGL] or [ALL] to replace the string. To move the cursor to the next string without replacing the string, press [SKIP].



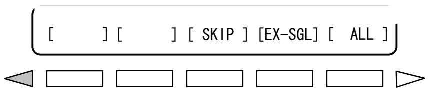
   - If soft key [SKIP] is pressed, the word at the cursor position is not replaced and the next word is searched for. If no word is found at the current cursor position and later, the cursor moves to the end of the program and the warning message "THE STRING IS NOT FOUND" is displayed.



   - If soft key [EX-SGL] is pressed, the word at the cursor position is replaced. Then, a search in the forward direction is made and the cursor moves. If no word is found at the current cursor position and later, the cursor moves to the end of the program and the warning message "THE STRING IS NOT FOUND" is displayed.



   - If soft key [ALL] is pressed, all words that are present at the current cursor position and later are replaced.



11. To finish the replacement, press soft key ◁.

10.9 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the III-9.1 for the comments of a program.

Explanation

- Editing unit

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:
(a) Address
(b) # located at the start of the left side of a substitution statement
(c) /, (,=, and ;
(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS
On the screen, a blank is placed before each of the above characters and symbols.
Example) Example of display where one blank is inserted

```
N001 X-#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X-SQRT[#3/3*[#4+1]] ;
N004 X-#2 Z#1 ;
N005 #5 =1+2-#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;
```

- Abbreviations of custom macro word

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH	GOTO → GO	XOR → XO	AND → AN	SIN → SI
ASIN → AS	COS → CO	ACOS → AC	TAN → TA	ATAN → AT
SQRT → SQ	ABS → AB	BCD → BC	BIN → BI	FIX → FI
FUP → FU	ROUND → RO	END → EN	POPEN → PO	BPRNT → BP
DPRNT → DP	PCLOS → PC	EXP → EX	THEN → TH	

Example) Keying in
WH [AB [#2] LE RO [#3]] has the same effect as
WHILE [ABS [#2] LE ROUND [#3]]
The program is also displayed in this way.

> **NOTE**
> The arithmetic function POW cannot be replaced with PO.

10.10 PASSWORD FUNCTION

With the password function, bit 4 (NE9) of parameter No. 3202 used to protect the programs of program numbers O9000 to O9999 can be locked using two parameters, namely, PASSWD (parameter No. 3210) and KEYWD (parameter No. 3211). In the locked state, bit 4 (NE9) of parameter No. 3202 cannot be set to 0. Thus, protection of the programs of program numbers O9000 to O9999 can be canceled only when the correct keyword is set.

A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.
2 Enable parameter writing (III-12.3.1). At this time, alarm SW0100 is issued on the CNC.
3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.
4 Disable parameter writing.
5 Press the 

key to release the alarm state.

Unlocking

1 Set the MDI mode.
2 Enable parameter writing (III-12.3.1). At this time, alarm SW0100 is issued on the CNC.
3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.
4 Set bit 4 (NE9) of parameter No.3202 to 0.
5 Disable parameter writing.
6 Press the 

key to release the alarm state.
7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanation

- Setting parameter PASSWD

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set the parameter PASSWD in a state other than this state, a write protect warning is issued. When the locked state is set (when parameter PASSWD ≠ 0 and parameter PASSWD ≠ parameter KEYWD), the parameter NE9 is automatically set to 1. In this case, a write protect warning is issued when an attempt is made to set NE9 to 0.

- Changing parameter PASSWD

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- Setting 0 in parameter PASSWD

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- Re-locking

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the NC off then on again to reset parameter KEYWD.

> ⚠ **CAUTION**
> Once the locked state is set, parameter NE9 cannot be set to 0 and parameter PASSWD cannot be changed until the locked state is released or the memory all-clear operation is performed. Special care must be taken in setting parameter PASSWD.

10.11 SIMULTANEOUS EDITING OF 2-PATH PROGRAMS

T

Simultaneous editing of 2-path programs enables simultaneous editing of programs for two paths on a single screen.
This function is enabled when the following conditions are satisfied:

- 2-path control system
- Bit 2 (DOP) of parameter No.3193 = 0
- Bit 0 (DHD) of parameter No.3106 = 1

Explanation

- Procedure

1 Place path 1 and path 2 in EDIT mode.

2 Press function key .

3 Press soft key [PROGRAM] to display the program editing screen.

- Screen display

Figure 10.11 (a) to (c) show examples of performing simultaneous editing of 2-path programs.
Above each program is a status line on which three items of information are displayed: the program number, "FG:EDIT", which indicates that the program is being edited in the foreground, and the path name.
For the program currently being edited, its status line is displayed in reverse video.

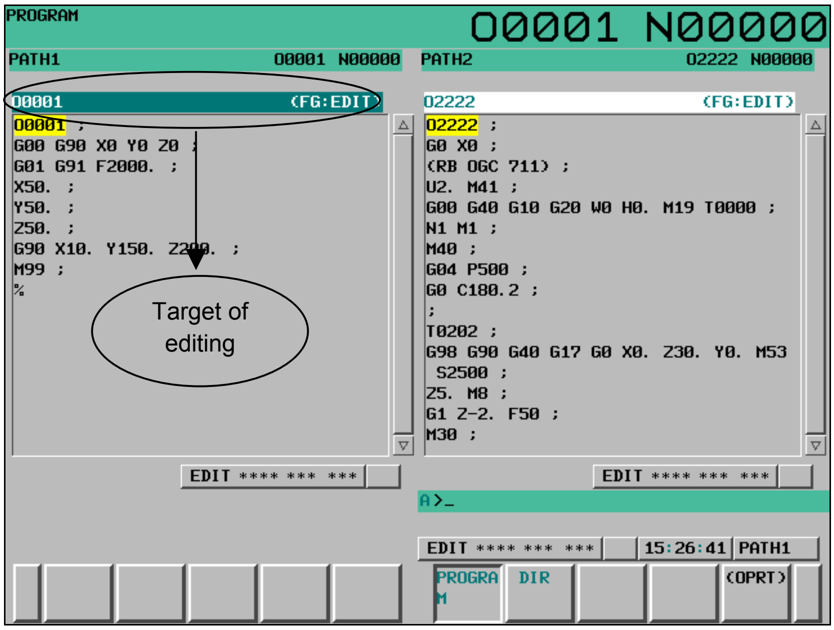


Fig. 10.11 (a) Simultaneous editing of 2-path programs screen (10.4-inch LCD)

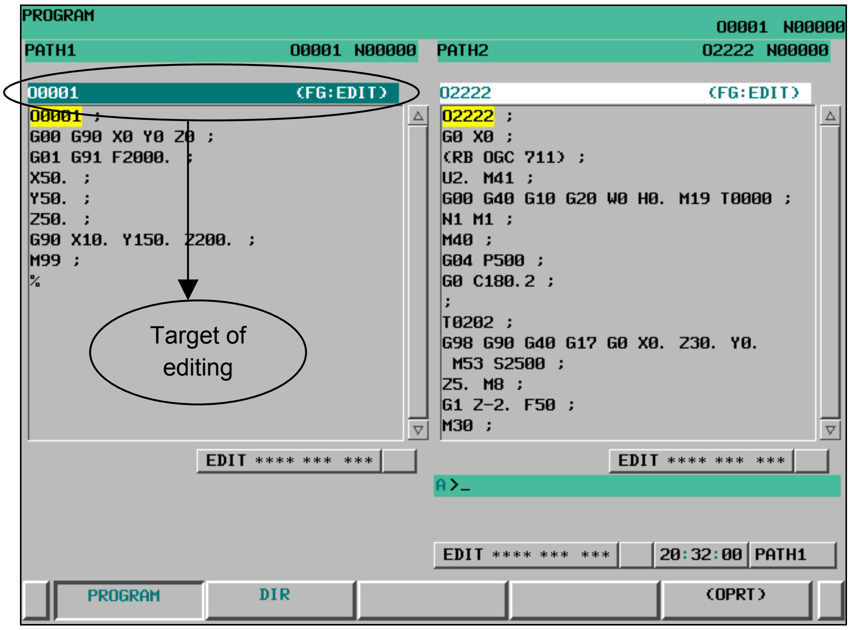


Fig. 10.11 (b) Simultaneous editing of 2-path programs screen (8.4-inch LCD)

- Modes

When both of path 1 and path 2 are placed in EDIT mode or MEM mode, the programs of both paths are simultaneously displayed on the program screen. When a path placed in EDIT mode is selected, program editing can be performed.

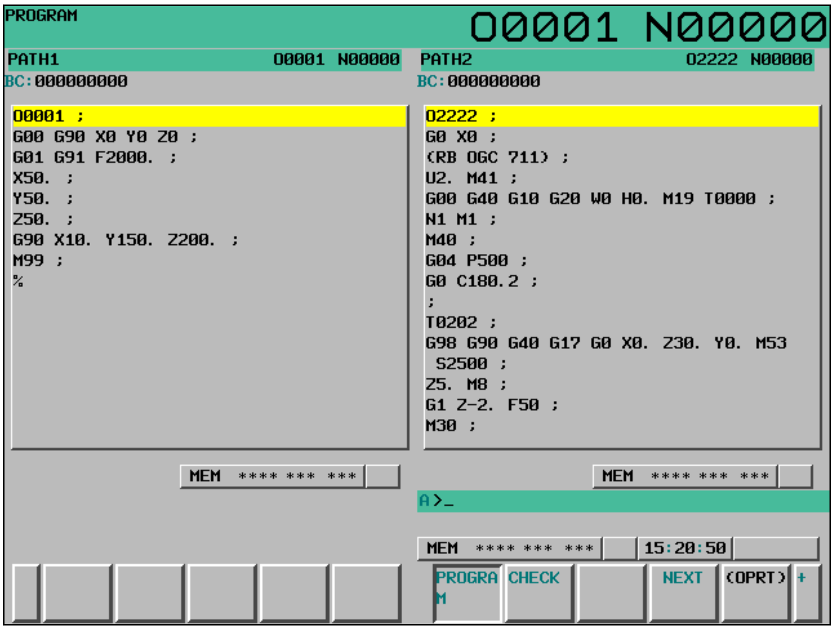


Fig. 10.11 (c) Screen for simultaneous 2-path program editing (MEM mode)

- Switching the path subject to editing

The path selected with the path selection signal is subject to editing.

- Conditions for enabling simultaneous display and editing

Simultaneous editing of 2-path programs is disabled in the following cases:

- When the program screen is selected as the full screen
- When both paths are placed in EDIT or MEM mode in foreground editing
- The virtual MDI key function is disabled.

If background editing is started when simultaneous 2-path program editing is enabled, background editing is started not only with the path for which operation is performed but also with the path not selected, and simultaneous 2-path program background editing is displayed. (Note, however, that background editing with the path not selected is started with no program.) The edit mode specified at the start of background editing is applied to all paths under simultaneous editing. On the simultaneous editing screen, no mixture of the edit mode and reference mode is allowed. When a background editing end operation is performed, background editing is ended not only with the path for which operation is performed but also with the path not selected.

- Simultaneous editing on 8.4-inch LCD

If the simultaneous editing screen is displayed on an 8.4-inch display unit, characters are displayed in smaller size.

10.12 COMPACT-TYPE MDI KEY INPUT

Use of this function allows the user to enter characters such as "@", "(", and ")" by using soft keys when the compact-type MDI unit is used.

Procedure of compact-type MDI key input

Procedure

1. Select the EDIT mode.
2. Press function key .

3. Press soft key [(OPRT)] then press continuous menu key  several times to display soft key [CHA-EXT].
4. Press soft key [CHA-EXT].
5. As shown in Fig. 10.12 (a), characters such as "@" and "(" appear as soft keys.

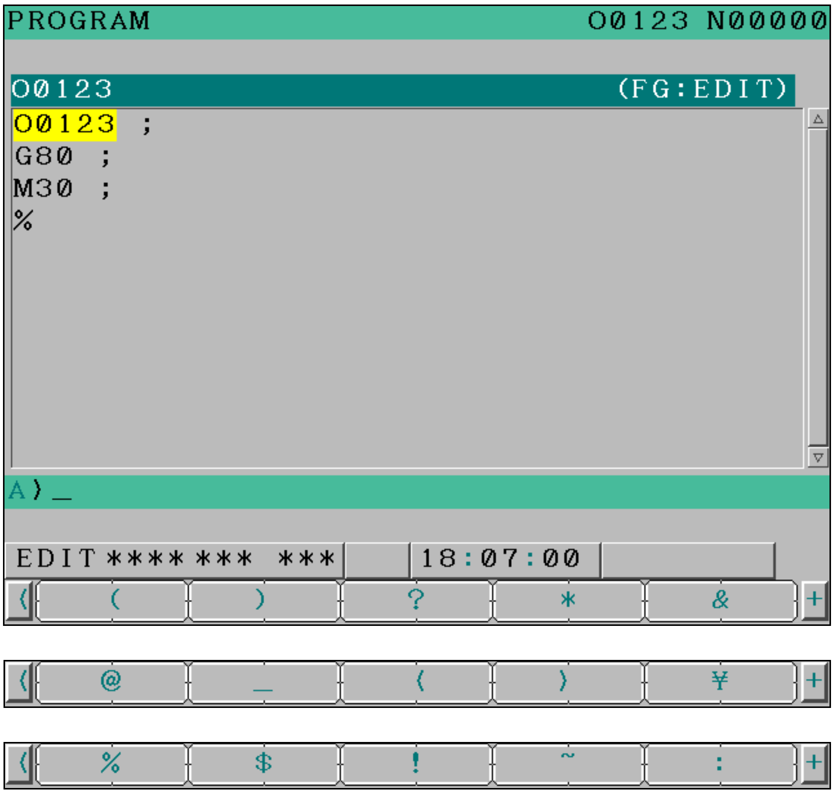


Fig. 10.12 (a) Compact-type MDI key input

6. When a soft key indicating the character to be input is pressed, the character is input to the key-in buffer.

Explanation

- Usable characters

The following characters can be entered using soft keys:

Table 10.12 (a) Characters that can be entered using soft keys

(	)	?	*	&
@	_	<	>	¥
%	$	!	~	:
"	‘			AB/ab(*)

* This soft key causes the input mode of alphabetical characters to alternate between the uppercase input mode and the lowercase input mode.
The character before the key-in buffer changes to "A" or "a" depending on the input mode.

- Setting that inputting < > ¥ % $! ~ : " ' AB/ab become effective

Soft key input of the characters shown below and switching between the uppercase and lowercase input modes by a soft key are enabled by setting 1 in bit 4 (SI1) of parameter No.13115.

< > ¥ % $! ~ : " '

- Setting that inputting ( ) ? * & @ _ AB/ab become effective

Soft key input of the characters shown below and switching between the uppercase and lowercase input modes by a soft key are enabled by setting 1 in bit 5 (SI2) of parameter No.13115.

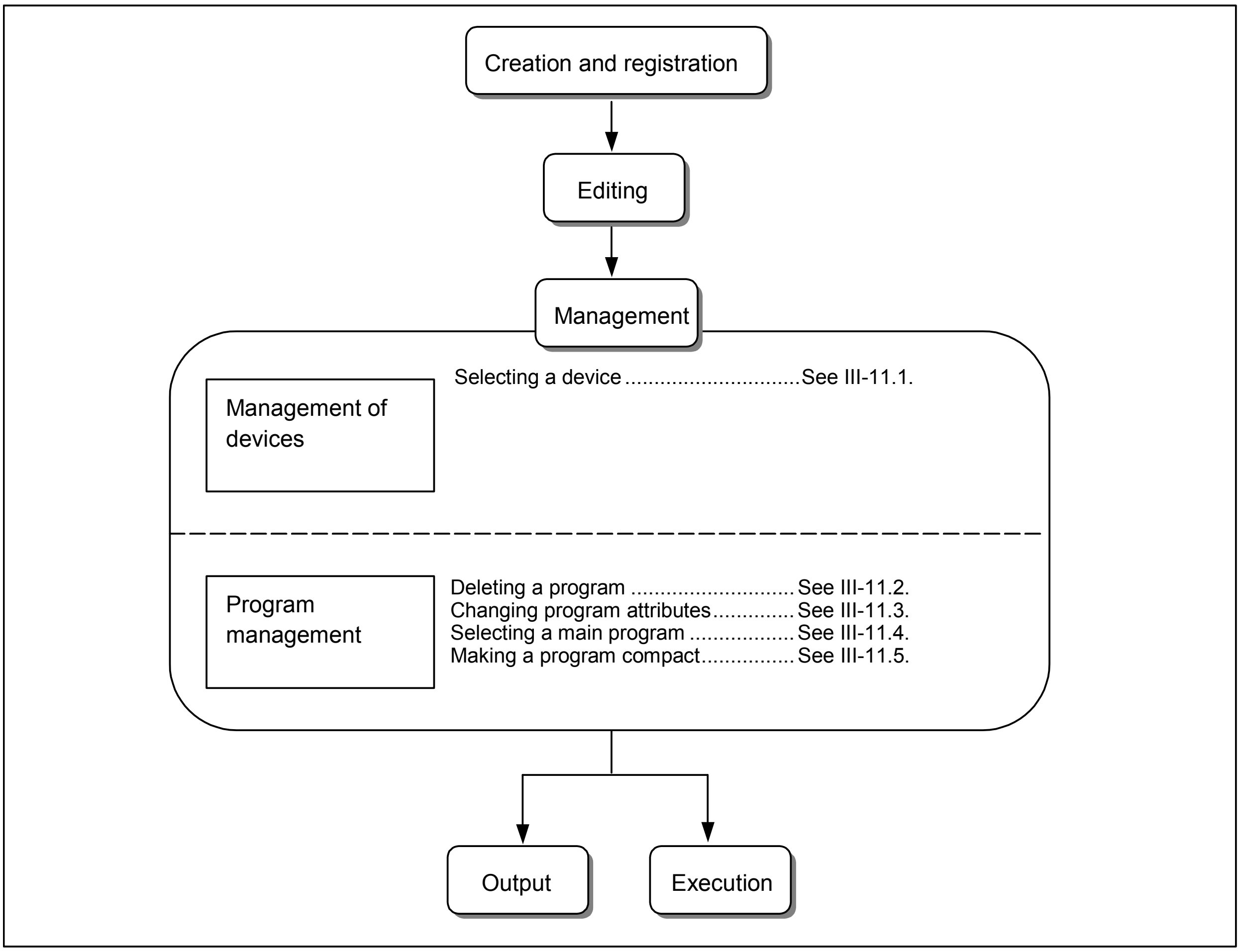
( ) ? * & @ _

11 PROGRAM MANAGEMENT

Program management functions are classified into the following two types:

- Functions for devices
- Functions for programs

The functions for devices include selection and so on.
The functions for programs include main program selection, deletion, change of names and attributes, program compaction, and so on.



11.1 SELECTING A DEVICE

When the fast data server function (option) is provided, a program storage device can be selected. This section explains the selection procedure.

Procedure for selecting a device

1 Press the function key 

.

2 Press the soft key [DIR].

3 Press the operation selection soft key [(OPRT)].

4 Hold down the continuous menu key  until the soft key [DEVICECHANGE] (or [DEVICE] when an 8.4-inch display unit is used) is displayed.

5 Press the soft key [DEVICECHANGE].
6 Press the soft key for the desired device.

11.1.1 Selecting a Memory Card Program as a Device

Overview

By selecting a memory card including a program storage file (named "FANUCPRG.BIN") as a device, memory operation can be performed with the program in the program storage file selected as the main program.
In addition, the content of a program storage file can be displayed on the program list screen or a program in a program storage file can be edited on the program edit screen.
A program storage file can be created using a memory card program tool (A08B-9010-J700#ZZ11) on a commercially available personal computer. To be used, the program storage file must be written to a memory card formatted in FAT16 format.
(A program held in a program storage file is hereinafter referred to as a memory card program. Moreover, a memory card storing a program storage file is referred to as a program storage memory card.)

Procedure for selecting a device

1 Press the function key .
2 Press the soft key [DIR].
3 Press the operation selection soft key [(OPRT)].
4 Hold down the continuous menu key  until the soft key [DEVICECHANGE] (or [DEVICE] when an 8.4-inch display unit is used) is displayed.
5 Press the soft key [DEVICECHANGE].
6 Press the soft key [MEMCARD].

> **NOTE**
> A FAT16-formatted memory card containing the program storage file FANUCPRG.BIN is recognized as a program storage memory card.

Procedure for removing a device

When a program storage memory card is replaced or a memory card is used for normal usage such as data input/output, clear the recognition of the program storage memory card with removal operation.

1 Press the function key .
2 Press the soft key [DIR].
3 Press the operation selection soft key [(OPRT)].
4 Hold down the continuous menu key  until the soft key [DEVICECHANGE] (or [DEVICE] when an 8.4-inch display unit is used) is displayed.
5 Press the soft key [DEVICECHANGE].
6 Hold down the continuous menu key  until the soft key [UMOUNT] is displayed.
7 Press the soft key [UMOUNT].

> **NOTE**
> 1 The soft key [UMOUNT] is displayed after the device selected by the CNC is recognized as a "program storage memory card" as the result of device change operation.

> **NOTE**
> 2 This operation is enabled only in EDIT mode or MEM mode.
> When a memory card program is selected in the main programs of two paths in a 2-path control system, set the modes of both paths to EDIT mode or MEM mode.
> 3 If the default folder is a folder in the program storage file stored on the memory card, the default folder is changed to "//CNC_MEM/" as the result of detach operation.
> 4 When the main program is a memory card program, the main program enters the unselected state by a removal operation.

Explanation

- About operation

A memory card program can be selected as a main program to perform memory operation.
Memory operation has the following features:

- Subprogram call nesting is allowed.
- Macro program call nesting is allowed.
- In a custom macro, a control command using a GOTO statement/WHILE statement can be specified.

- Selection as a main program

As a main program to be automatically executed in the memory mode, a memory card program can be selected.
However, information of having selected the memory card program as the main program is lost with the power disconnection. (If the power disconnection and re-turning on is performed from the state of above information, the selections are as follows.)
 - The device selection is CNC_MEM.
 - The main program selection is none.

- Sub program (call using M98)
- Macro program (call using G65/G66/M96)

The following subprogram/macro program is called:

- Sub program call (M98)
- Macro call (Simple macro call G65 / Modal call G66)
- Macro interrupt (M96)

- Sub program (call using M code/S code/T code/particular address/the second auxiliary function)
- Macro program (call using G code/M code) (one touch macro call)

The following subprogram/macro program calls a program from the CNC_MEM device (CNC program storage memory):

- Subprogram call using M code/S code/T code/particular addresses/the second auxiliary function
- Macro call using G code/M code
- One touch macro call

> **NOTE**
> For a memory card program, subprogram call using M code/S code/T code/particular addresses/the second auxiliary function or macro call using G code/M code can be specified. However, a program on the CNC_MEM device (CNC program storage memory) is called.

- External program number search / External workpiece number search

A program on a program storage memory card can be searched for with the external program number search function or external workpiece number search function.

- Main program search

The current main program is searched for. The found main program is displayed at the top of the list. The cursor is placed on the main program.

1 Press the soft key [PROGRMSEARCH].
2 Press the soft key [MAIN SRCH].

> **NOTE**
> If a program on another device is set as the main program, executing a main program search causes automatic switching to the device.

Limitation

For a memory card program, M198 cannot be specified. Moreover, no memory card program can be called from a program on the CNC_MEM device (CNC program storage memory) by specifying M198.

When a setting is made to enable an external device subprogram call from a memory card (M198) or DNC operation from a memory card (bit 7 (MNC) of parameter No. 0138 = 1), the content of program storage file cannot be displayed during automatic operation.

When a program storage memory card is selected, the memory card cannot be used for the ordinary purposes listed below. To use a memory card in such a case, perform a "removal" operation to cancel the recognition of the program storage memory card.

- ALL I/O screen
  Display of the contents of a memory card, and inputting/outputting data to and from a memory card
- PMC data I/O screen
  Display of the contents of a memory card, and inputting/outputting to and from a memory card
- Program directory screen
  Inputting/outputting program data to and from a memory card
- External device subprogram call (M198) operation
  Subprogram call (M198) with a memory card set as an external device
- DNC operation
  DNC operation from a memory card

> ⚠ **CAUTION**
> 1 Do not remove the memory card when a program that specifies a write to the memory card is being edited. The data can be destructed.
> 2 If an editing operation is completed, the results of editing are preserved even when the power to the CNC is turned off.
> 3 When removing the memory card, be sure to perform a "removal" operation. If the memory card is removed without performing a "removal" operation and an attempt is made to access the memory card, the alarm (SR1964) or alarm (IO1030) is issued.
> If the card is removed inadvertently, insert the card again and perform a "removal" operation.
> When an alarm is issued, perform the following operation:
> - The power of CNC is turned off and on.
> The alarm can be reset only by turning off the power to the CNC.

> ⚠ **CAUTION**
> 4 There are cases in which when a memory card is replaced with another, the CNC cannot detect the replacement. Thus, it is risky to replace a memory card without performing a "removal" operation, and this should never be attempted.

- Operation of creation, edition, and management of a program

When "memory card program as a device" is selected, operation of creation, edition, and management of a program is below:

Item	Usable
Creation of a program	Unusable
Inserting, alteration, and deletion a Word	Usable
Deletion of a block	Usable
Program search	Usable
Sequence number search	Usable
Deletion of a program	Unusable
Editing a custom macro	Usable
Password function	Unusable
Selecting a device	Usable
Selecting a main program	Usable
Making a program compact	Unusable
Input/output of program	Unusable

11.2 DELETING A PROGRAM

This section explains the procedure for deleting a program.

Procedure for deleting a file

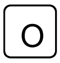
1 Select EDIT mode.

2 Press the function key 

.

3 Press the soft key [DIR].

4 Key in the program number of a program to be deleted.

(Press the address key 

then key in a desired program number.)

5 Press the edit key 

.

When the 8-level data protection function is used, the cursor appears on the program list.
In this case, select a program not with step 4 above but with the cursor key  or .

> **NOTE**
> 1 Depending on the operation status and protection status, a file cannot sometimes be deleted.
> 2 The 8-level data protection function is optional.

11.3 CHANGING PROGRAM ATTRIBUTES

This section explains the procedure for changing the attribute of a program (edit disable, edit/display disable, or protection of data at eight levels).

Procedure for selecting the attribute of a program

1 Select EDIT mode.

2 Press the function key PROG.

3 Press the soft key [DIR].

4 Press the soft key [DIR+] to display a detailed program list. (Each time the soft key [DIR+] is pressed, the program list display switches between detailed display and normal display.)

5 Select the program of which attribute is to be changed.
Move the cursor with the cursor key ⬇ or ⬆ to a program whose attribute is to be changed.

6 Press the soft key [(OPRT)].

7 Press the soft key ['CHANGEATTRIB] (or [ATTRIB] when an 8.4-inch display unit is used).

8 Press the continuous menu key ▷ until the desired soft key (step 8 below) appears.

9
- To disable editing, press the soft key [EDIT DISABL] (or [EDT OFF] when an 8.4-inch display unit is used).
- To enable editing, press the soft key [EDIT ENABLE] (or [EDT ON] when an 8.4-inch display unit is used).
- To disable editing and display, press the soft key [DISP DISABL] (or [DSP OFF] when an 8.4-inch display unit is used).
- To enable editing and display, press the soft key [DISP ENABLE] (or [DSP ON] when an 8.4-inch display unit is used).
- To change the change protection level, type a change protection level, then press the soft key [CHANGELEVEL] (or [CHG LEV] when an 8.4-inch display unit is used).
- To change the output protection level, type an output protection level, then press soft key [OUT LEVEL] (or [OUT LEV] when an 8.4-inch display unit is used).

10 Press the soft key [END].

NOTE

1 Depending on the operation status and protection status, a file cannot sometimes be deleted.

2 Depending on the settings of the following parameters, the settable items vary:
- Program protection (parameter No. 3210 and No. 3211)
- Eight-level data protection (Optional function)

11.4 SELECTING A MAIN PROGRAM

This section explains the procedure for selecting a main program.

Procedure for selecting a main program

1 Select EDIT mode.

2 Press the function key PROG.

3 Press the soft key [DIR].

4 Key in the program number of a program to be selected as a main program.
(Press the address key O then key in a desired program number.)

5 Press the soft key [O SEARCH].
A selection can also be made by pressing the cursor key ⬇.

> **NOTE**
> 1 Depending on the operation status and protection status, the main program cannot sometimes be selected.
> 2 The 8-level data protection function is optional.

11.5 MAKING A PROGRAM COMPACT

This section explains the procedure for making a program compact.

Procedure for making a program compact

1 Select EDIT mode.
2 Press the function key PROG.
3 Press the soft key [DIR].
4 Key in the program number of a program to be made compact.
(Press the address key O then key in a desired program number.)
5 Press the soft key [PROGRMCNDENS] (or [CONDENS] when an 8.4-inch display unit is used).

When the 8-level data protection function is used, the cursor appears on the program list.
In this case, select a program not with step 4 above but with the cursor key ⬇ or ⬆.

> **NOTE**
> 1 Depending on the operation status and protection status, a program cannot sometimes be made compact.
> 2 Only programs on the CNC_MEM device can be made compact.
> 3 The 8-level data protection function is optional.

12 SETTING AND DISPLAYING DATA

To operate a CNC machine tool, various data must be set on the MDI panel for the CNC. The operator can monitor the state of operation with data displayed during operation.
This chapter describes how to display and set data for each function.
Chapter 12, "SETTING AND DISPLAYING DATA", consists of the following sections:

Explanation

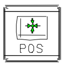
- Screen transition chart

The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

See Chapter 7 for the screen that appears when function key MESSAGE is pressed. In general, function key CUSTOM1 or CUSTOM2 is prepared by the machine tool builder and used for macros. Refer to the manual issued by the machine tool builder for the screen that appears when function key CUSTOM1 or CUSTOM2 is pressed.

- Data protection key

The machine may have a data protection key to protect part programs, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

Screen displayed when the function key  is pressed (for 8.4/10.4-inch display unit)

Page 1

	(1)	(2)	(3)	(4)	(5)	
	ABS	REL	ALL	HNDL	(OPRT)	+

(1) ABS	(2) REL	(3) ALL	(4) HNDL
Position display in the workpiece coordinate system ⇨ See III-12.1.1	Position display in the workpiece coordinate system ⇨ See III-12.1.1	Position display in the workpiece coordinate system ⇨ See III-12.1.1	Manual handle interruption ⇨ See III-4.6
Actual feedrate display ⇨ See III-12.1.5	Actual feedrate display ⇨ See III-12.1.5	Actual feedrate display ⇨ See III-12.1.5	
Display of run time and parts count ⇨ See III-12.1.6	Display of run time and parts count ⇨ See III-12.1.6	Display of run time and parts count ⇨ See III-12.1.6	

Page 2

	(6)	(7)	(8)	(9)	(10)	
	MONI				(OPRT)	+

(6) MONI ⇩
Operating monitor display ⇨ See III-12.1.7

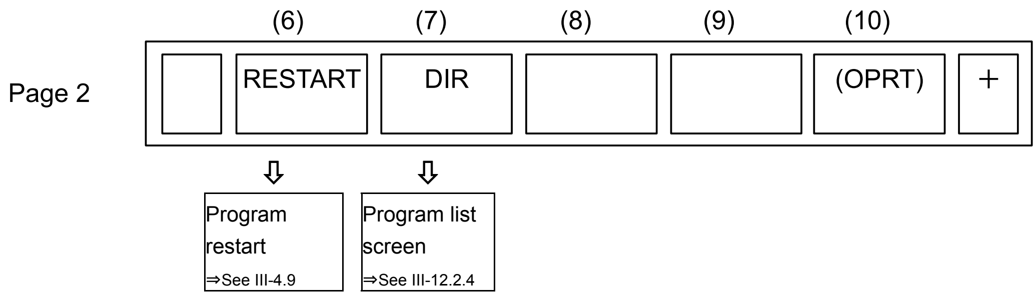
Screen displayed when the function key  is pressed (for 8.4-inch display unit)

In the MEM/RMT mode

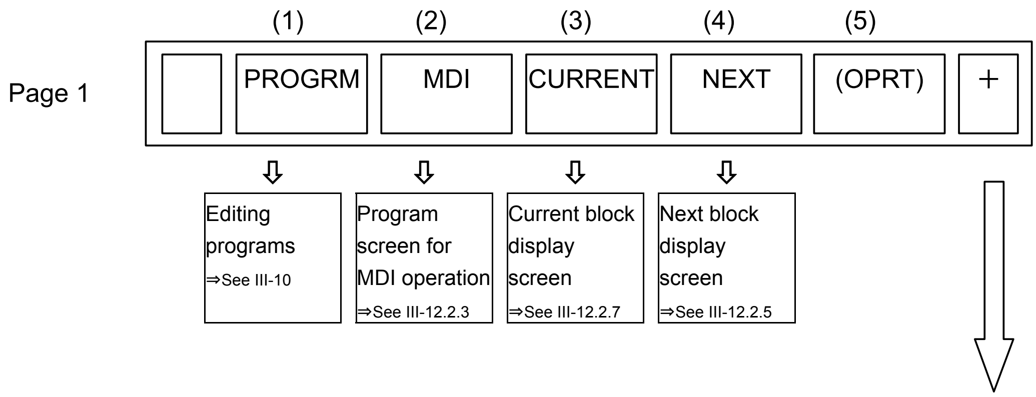
Page 1

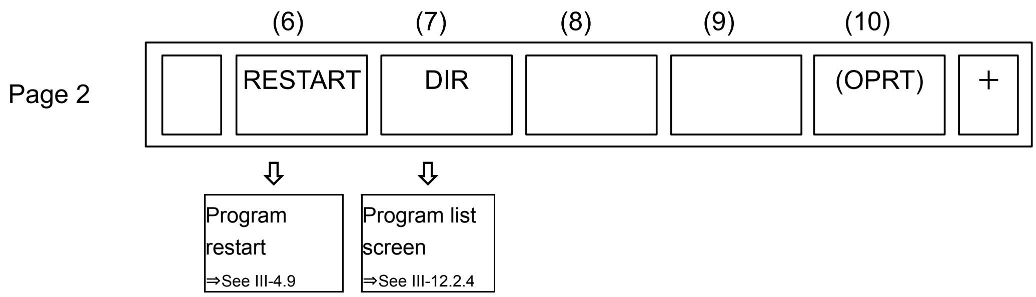
	(1)	(2)	(3)	(4)	(5)	
	PROGRM	CHECK	CURRENT	NEXT	(OPRT)	+

(1) PROGRM	(2) CHECK	(3) CURRENT	(4) NEXT
Editing programs ⇒See III-10	Program check screen ⇒See III-12.2.6	Current block display screen ⇒See III-12.2.7	Next block display screen ⇒See III-12.2.5

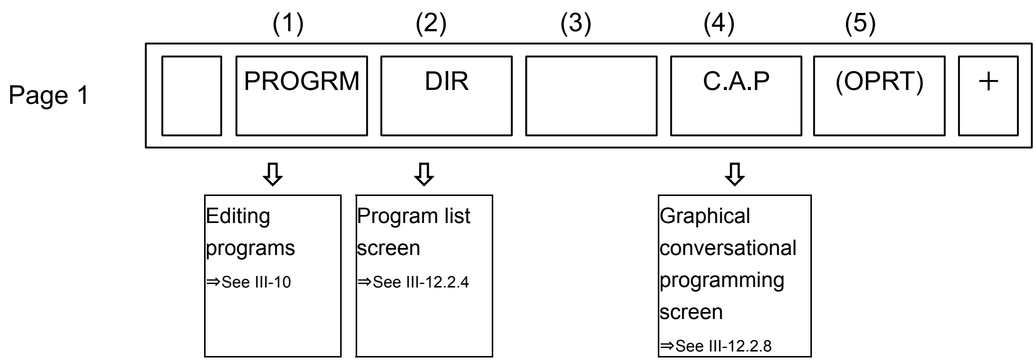


In the MDI mode





In the EDIT/TJOG/THND mode



In the JOG/HND/REF mode

Page 1

	(1)	(2)	(3)	(4)	(5)	
	PROGRM		CURRENT	NEXT	(OPRT)	+
	⇩ Editing programs ⇒See III-10		⇩ Current block display screen ⇒See III-12.2.7	⇩ Next block display screen ⇒See III-12.2.5		

Page 2

	(6)	(7)	(8)	(9)	(10)	
	RESTART	DIR			(OPRT)	+
	⇩ Program restart ⇒See III-4.9	⇩ Program list screen ⇒See III-12.2.4				

Screen displayed when the function key  **is pressed (for 10.4-inch display unit)**

In the MEM/JOG/HND/RMT mode

Page 1

	(1)	(2)	(3)	(4)	(5)	
	PROGRA M			NEXT BLOCK	(OPRT)	+
	⇩ Editing programs ⇒See III-10			⇩ Next block display screen ⇒See III-12.2.5		

Page 2

	(6)	(7)	(8)	(9)	(10)	
	RESTART	DIR			(OPRT)	+
	⇩ Program restart ⇒See III-4.9	⇩ Program list screen ⇒See III-12.2.4				

In the MDI mode

Page 1

	(1)	(2)	(3)	(4)	(5)	
	PROGRAM	MDI		NEXT BLOCK	(OPRT)	+

⇩ (1) Editing programs ⇒See III-10

⇩ (2) Program screen for MDI operation ⇒See III-12.2.3

⇩ (4) Next block display screen ⇒See III-12.2.5

Page 2

	(6)	(7)	(8)	(9)	(10)	
	RESTART	DIR			(OPRT)	+

⇩ (6) Program restart ⇒See III-4.9

⇩ (7) Program list screen ⇒See III-12.2.4

In the EDIT/TJOG/THND mode

Page 1

	(1)	(2)	(3)	(4)	(5)	
	PROGRAM	DIR		C.A.P	(OPRT)	+

⇩ (1) Editing programs ⇒See III-10

⇩ (2) Program list screen ⇒See III-12.2.4

⇩ (4) Graphical conversational programming screen ⇒See III-12.2.8

Screen displayed when the function key OFS/SET is pressed (for 8.4-inch display unit)

Page 1

	(1)	(2)	(3)	(4)	(5)	
	OFFSET	SETTING	WORK		(OPRT)	+
	Setting and displaying the tool offset value ⇒See III-2.1.1[*1]	Displaying and entering setting data ⇒See III-12.3.1	Displaying and setting the workpiece origin offset value ⇒See III-12.3.4			

Page 2

	(6)	(7)	(8)	(9)	(10)	
	MACRO	MENU	OPR	TOOLLF	(OPRT)	+
	Displaying and setting custom macro common variables ⇒See III-12.3.6	Displaying and setting pattern data inputs ⇒See III-12.3.12	Displaying and setting the software operator's panel ⇒See III-12.3.7	Displaying and setting tool life management data ⇒See III-12.3.11		

Page 3

	(11)	(12)	(13)	(14)	(15)	
	OFST.2	W.SHFT		BARRIER	(OPRT)	+
	Setting the Y-Axis offset ⇒See III-2.1.6[*1]	Setting the workpiece coordinate system shift value ⇒See III-2.1.5[*1]		Chuck and tail stock barriers ⇒See III-2.1.7[*1]		

Page 4

	(16)	(17)	(18)	(19)	(20)	
		PR-LEV			(OPRT)	+
		Precision level selection ⇒See III-12.3.10				

Page 5

	(21)	(22)	(23)	(24)	(25)	
		LANG.	PROT.	GUARD	(OPRT)	+

(22) Displaying and switching the display language ⇒See III-12.3.8

(23) Protection of data at eight levels ⇒See III-12.3.9

(24) Wrong operation prevention functions ⇒See III-6.5

Screen displayed when the function key OFS/SET is pressed (for 10.4-inch display unit)

Page 1

	(1)	(2)	(3)	(4)	(5)	
	OFFSET	SETTING	WORK		(OPRT)	+

(1) Setting and displaying the tool offset value ⇒See III-2.1.1 [*1]

(2) Displaying and entering setting data ⇒See III-12.3.1

(3) Displaying and setting the workpiece origin offset value ⇒See III-12.3.4

Page 2

	(6)	(7)	(8)	(9)	(10)	
	MACRO	PATTERN MENU	OPERAT PANEL	TOOL LIFE	(OPRT)	+

(6) Displaying and setting custom macro common variables ⇒See III-12.3.6

(7) Displaying and setting pattern data inputs ⇒See III-12.3.12

(8) Displaying and setting the software operator's panel ⇒See III-12.3.7

(9) Displaying and setting tool life management data ⇒See III-12.3.11

Page 3

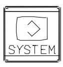
	(11)	(12)	(13)	(14)	(15)	
	Y OFFSET	WORK SHIFT		BARRIER	(OPRT)	+
	Setting the Y-Axis offset ⇒See III-2.1.6[*1]	Setting the workpiece coordinate system shift value ⇒See III-2.1.5[*1]		Chuck and tail stock barriers ⇒See III-2.1.7[*1]		⇩

Page 4

	(16)	(17)	(18)	(19)	(20)	
		PRECI LEVEL			(OPRT)	+
		Precision level selection ⇒See III-12.3.10				⇩

Page 5

	(21)	(22)	(23)	(24)	(25)	
		LANGUA GE	PROTEC T	GUARD	(OPRT)	+
		Displaying and switching the display language ⇒See III-12.3.8	Protection of data at eight levels ⇒See III-12.3.9	Wrong operation prevention functions ⇒See III-6.5		

Screen displayed when the function key  **is pressed (for 8.4-inch display unit)**

Page 1

	(1)	(2)	(3)	(4)	(5)	
	PARAM	DGNOS		SYSTEM	(OPRT)	+
	Displaying and setting parameters ⇒See III-12.4.1	Checking by diagnostic display ⇒See III-7.3		System configuration screen ⇒See III-12.4.12		⇩

Page 2

	(6)	(7)	(8)	(9)	(10)	
		PITCH	SV.SET	SP.SET	(OPRT)	+
		Displaying and setting pitch error compensation data ⇒See III-12.4.2	Servo setting ⇒See III-12.4.3	Spindle setting ⇒See III-12.4.5		

Page 3

	(11)	(12)	(13)	(14)	(15)	
	W.DGNS	ALL IO		OPEHIS	(OPRT)	+
	Waveform diagnosis display ⇒Maintenance manual B-64305EN	Input/output on the ALL/IO screen ⇒See III-8.3		Overview of the history function ⇒See III-12.4.13		

Page 4

	(16)	(17)	(18)	(19)	(20)	
	PMCMNT	PMCLAD	PMCCNF	PM.MGR	(OPRT)	+
	PMC diagnosis and maintenance screens ⇒Maintenance manual B-64305EN ⇒PMC Ladder Language Programming manual B-64393EN	Ladder diagram monitor and editor screens ⇒Maintenance manual B-64305EN ⇒PMC Ladder Language Programming manual B-64393EN	PMC configuration data setting screens ⇒PMC Ladder Language Programming manual B-64393EN	Power Mate CNC manager functions ⇒Maintenance manual B-64305EN		

Page 5

	(21)	(22)	(23)	(24)	(25)	
	COLOR	MAINTE	M-INFO		(OPRT)	+
	Color setting screen ⇒See III-12.4.8	Periodic maintenance screen ⇒See III-12.4.11	Maintenance information screen ⇒Maintenance manual B-64305EN			

Page 6

	(26)	(27)	(28)	(29)	(30)	
		FSSB	PRMSET		(OPRT)	+
		FSSB display and setting screen ⇒Maintenance manual B-64305EN	Parameter setting support screen ⇒See III-12.4.10			

Page 7

	(31)	(32)	(33)	(34)	(35)	
	EMBED	PCMCIA	ETHBRD	FL-net	(OPRT)	+
	Embedded Ethernet functions ⇒Maintenance manual B-64305EN	Embedded Ethernet functions ⇒Maintenance manual B-64305EN	Ethernet functions ⇒Fast Ethernet/Fast Data Server operator's manual B-64414EN	FL-net functions ⇒FL-net Board connection manual B-64453EN		

Page 8

	(36)	(37)	(38)	(39)	(40)	
	RMTDIAG	M-TUN			(OPRT)	+

(36) RMTDIAG ⇩
Machine remote diagnosis
⇒Fast Ethernet/Fast Data Server operator's manual B-64414EN
⇒Machine remote diagnosis package operator's manual B-63734EN

(37) M-TUN ⇩
Machining parameter tuning
⇒See III-12.4.9

Page 9

	(41)	(42)	(43)	(44)	(45)	
	ID-INF		MEMORY		(OPRT)	+

(43) MEMORY ⇩
Displaying the memory contents
⇒Maintenance manual B-64305EN

Page 10

	(46)	(47)	(48)	(49)	(50)	
	PROF.M	PROF.S	DNET M	DNET S	(OPRT)	+

(46) PROF.M ⇩
PROFIBUS-DP master functions
⇒PROFIBUS-DP Board connection manual B-64403EN

(47) PROF.S ⇩
PROFIBUS-DP slave functions
⇒PROFIBUS-DP Board connection manual B-64403EN

(48) DNET M ⇩
DeviceNet master functions
⇒DeviceNet Board connection manual B-64443EN

(49) DNET S ⇩
DeviceNet slave functions
⇒DeviceNet Board connection manual B-64443EN

Screen displayed when the function key SYSTEM is pressed (for 10.4-inch display unit)

Page 1

	(1)	(2)	(3)	(4)	(5)	
	PARAMETER	DIAGNOSIS		SYSTEM	(OPRT)	+
	Displaying and setting parameters ⇒See III-12.4.1	Checking by diagnostic display ⇒See III-7.3		System configuration screen ⇒See III-12.4.12		

Page 2

	(6)	(7)	(8)	(9)	(10)	
		PITCH ERROR	SERVO SETTING	SPINDLE SETTING	(OPRT)	+
		Displaying and setting pitch error compensation data ⇒See III-12.4.2	Servo setting ⇒See III-12.4.3	Spindle setting ⇒See III-12.4.5		

Page 3

	(11)	(12)	(13)	(14)	(15)	
	WAVE DIAG	ALL IO		OPERAT HISTRY	(OPRT)	+
	Waveform diagnosis display ⇒Maintenance manual B-64305EN	Input/output on the ALL/IO screen ⇒See III-8.3		Overview of the history function ⇒See III-12.4.13		

Page 4

	(16)	(17)	(18)	(19)	(20)	
	PMC MAINTE	PMC LADDER	PMC CONFIG	P.MATE MGR.	(OPRT)	+
	⇩	⇩	⇩	⇩		⇩
	PMC diagnosis and maintenance screens ⇒Maintenance manual B-64305EN ⇒PMC Ladder Language Programming manual B-64393EN	Ladder diagram monitor and editor screens ⇒Maintenance manual B-64305EN ⇒PMC Ladder Language Programming manual B-64393EN	PMC configuration data setting screens ⇒PMC Ladder Language Programming manual B-64393EN	Power Mate CNC manager functions ⇒Maintenance manual B-64305EN		

Page 5

	(21)	(22)	(23)	(24)	(25)	
	COLOR	PERIOD MAINTE	MAINTE INFO		(OPRT)	+
	⇩	⇩	⇩			⇩
	Color setting screen ⇒See III-12.4.8	Periodic maintenance screen ⇒See III-12.4.11	Maintenance information screen ⇒Maintenance manual B-64305EN			

Page 6

	(26)	(27)	(28)	(29)	(30)	
		FSSB	PARAMETER		(OPRT)	+
		⇩	⇩			⇩
		FSSB display and setting screen ⇒Maintenance manual B-64305EN	Parameter setting support screen ⇒See III-12.4.10			

Page 7

	(31)	(32)	(33)	(34)	(35)	
	EMBED PORT	PCMCIA LAN	ETHER BOARD	FL-net	(OPRT)	+

- (31) Embedded Ethernet functions ⇒Maintenance manual B-64305EN
- (32) Embedded Ethernet functions ⇒Maintenance manual B-64305EN
- (33) Ethernet functions ⇒Fast Ethernet/Fast Data Server operator's manual B-64414EN
- (34) FL-net functions ⇒FL-net Board connection manual B-64453EN

Page 8

	(36)	(37)	(38)	(39)	(40)	
	FINE TORQUE				(OPRT)	+

Page 9

	(41)	(42)	(43)	(44)	(45)	
	REMOTE DIAG	MCHN TUNING			(OPRT)	+

- (41) Machine remote diagnosis ⇒Fast Ethernet/Fast Data Server operator's manual B-64414EN ⇒Machine remote diagnosis package operator's manual B-63734EN
- (42) Machining parameter tuning ⇒See III-12.4.9

Page 10

	(46)	(47)	(48)	(49)	(50)	
	ID-INF		MEMORY		(OPRT)	+

- (48) Displaying the memory contents ⇒Maintenance manual B-64305EN

		(51)	(52)	(53)	(54)	(55)	
Page 11		PROFI MASTER	PROFI SLAVE	DEVNET MASTER	DEVNET SLAVE	(OPRT)	+
		⇩	⇩	⇩	⇩		
		PROFIBUS-DP master functions ⇒PROFIBUS-DP Board connection manual B-64403EN	PROFIBUS-DP slave functions ⇒PROFIBUS-DP Board connection manual B-64403EN	DeviceNet master functions ⇒DeviceNet Board connection manual B-64443EN	DeviceNet slave functions ⇒DeviceNet Board connection manual B-64443EN		

NOTE

For information about a dedicated screen for each control type in the T series/M series, refer to the manuals:
*1: Operator's manual (T series) (B-64304EN-1)
*2: Operator's manual (M series) (B-64304EN-2)

12.1 SCREENS DISPLAYED BY FUNCTION KEY

Section 12.1, "SCREENS DISPLAYED BY FUNCTION KEY ", consists of the following subsections:

Press function key to display the current position of the tool.

The following three screens are used to display the current position of the tool:

- Current position display screen for the workpiece coordinate system.
- Current position display screen for the relative coordinate system.
- Current overall position display screen.

The above screens can also display the feedrate, run time, and the number of parts.

Function key can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key can also be used to display the screen for displaying the distance moved by handle interruption. See III- 4.6 for details on this screen.

On any of the position display screens, the status (such as D, I, L, S, *, or M) of an axis is indicated on the left of the axis name to prevent wrong operations. See "Axis status display" in III-6.5, "WRONG OPERATION PREVENTION FUNCTIONS" for details.

12.1.1 Position Display in the Workpiece Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate

Procedure

1 Press function key .

2 Press chapter selection key [ABSOLUTE].

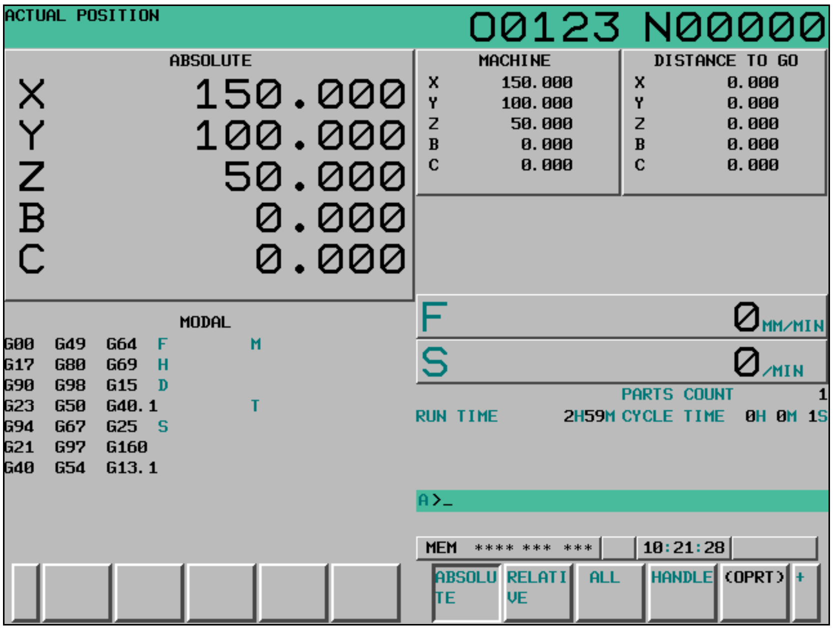


Fig. 12.1.1 (a) Current position (absolute) screen (10.4-inch)

Explanation

- Presetting the workpiece coordinate system

A workpiece coordinate system shifted by manual intervention or other operations can be preset by MDI operation to a workpiece coordinate system that is offset by a workpiece origin offset from the machine zero point before shifting.
For the procedure, see Subsection 12.1.4, "Workpiece Coordinate System Preset".

- Display including compensation values

M

Bits 6 (DAL) and 7 (DAC) of parameter No. 3104 can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

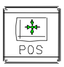
Bit 1 (DAP) parameter No. 3129 and bit 7 (DAC) of parameter No. 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

12.1.2 Position Display in the Relative Coordinate System

Displays the current position of the tool in a relative coordinate system based on the coordinates (see Explanation) set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values.
The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

Procedure

1 Press function key .

2 Press chapter selection key [RELATIVE].

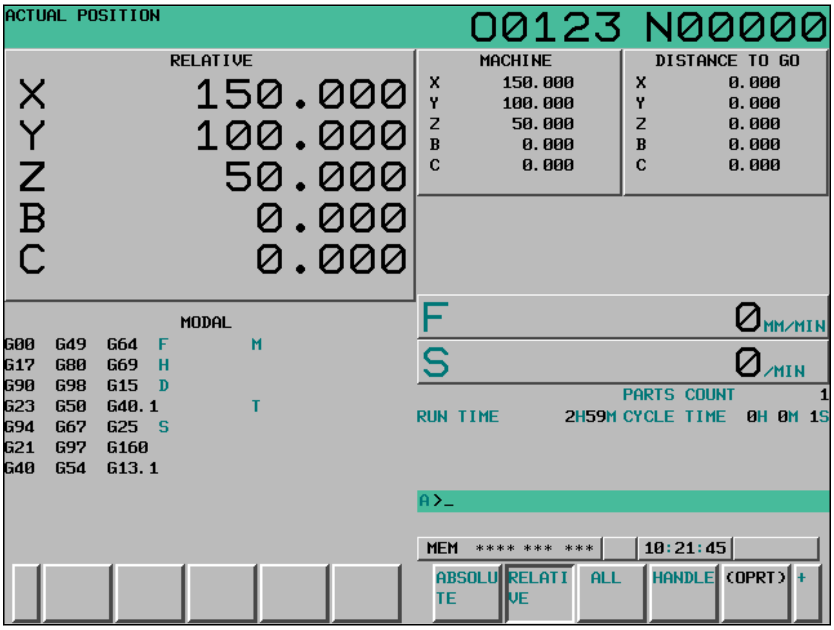


Fig. 12.1.2 (a) Current position (relative) screen (10.4-inch)

See Explanation for the procedure for setting the coordinates.

Explanation

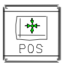
- Setting the relative coordinates

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows:

Resetting relative coordinates to 0

Procedure

- When all axes are reset to 0

1 Press function key .

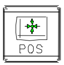
2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.

3 Press soft key [(OPRT)].

4 Press soft key [ORIGIN].

5 Press soft key [ALL AXIS].
At this time, the current position of all axes represented in relative coordinates are reset to 0.

- When a specified axis is reset to 0

1 Press function key .

2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.

3 Press soft key [(OPRT)].

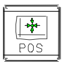
4 Press soft key [ORIGIN].

5 Input the axis name with keys (the axis name blinks) and press soft key [EXEC].

At this time, the current position of the specified axis represented in relative coordinates are reset to 0.

Presetting relative coordinates

Procedure

1 Press function key .

2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.

3 Press soft key [(OPRT)].

4 Input the axis name. At this time, the axis name blinks.

5 Input the coordinates, and press soft key [PRESET].
At this time, the current position of all axes represented in relative coordinates are set to specified value.

- Display including compensation values

M

Bits 4 (DRL) and 5 (DRC) of parameter No. 3104 can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

Bit 0 (DRP) parameter No. 3129 and bit 5 (DRC) of parameter No. 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

- Presetting by setting a coordinate system

M

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G92 command.

T

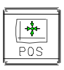
Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G50 command (for G code system A) or G92 command (for G code system B or C).

12.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance.
The relative coordinates can also be set on this screen. See III-12.1.2 for the procedure.

Procedure for displaying overall position display screen

Procedure

1 Press function key .

2 Press chapter selection key [ALL].

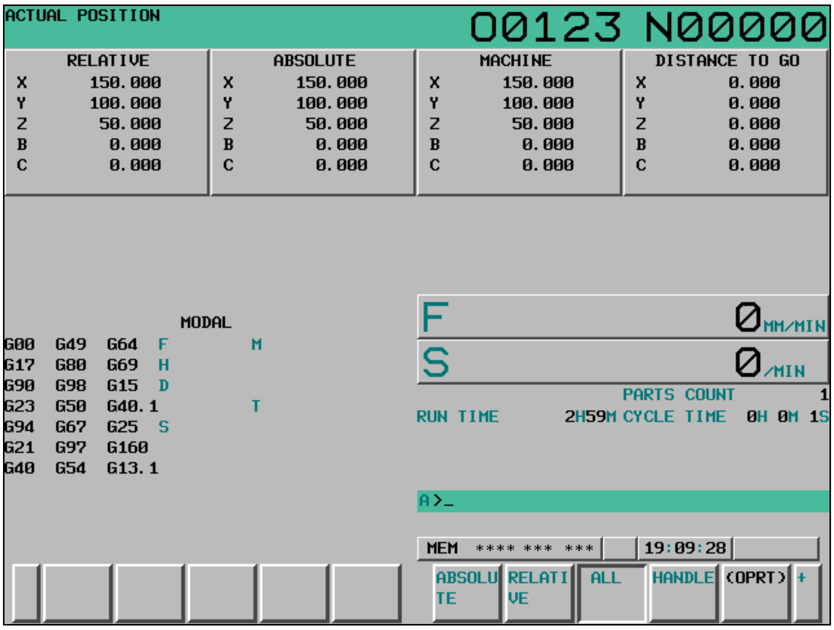


Fig. 12.1.3 (a) Current position (overall) screen (10.4-inch)

Explanation

- Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:

- Current position in the relative coordinate system
  (relative coordinate)
- Current position in the workpiece coordinate system
  (absolute coordinate)
- Current position in the machine coordinate system
  (machine coordinate)
- Distance to go (distance to go)

- Distance to go

The distance remaining is displayed in the MEM or MDI mode.
The distance the tool is yet to be moved in the current block is displayed.

- Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter No. 3104.

- Resetting the relative coordinates

The total position display screen also supports the resetting of the relative coordinates to 0 or presetting of them to specified values. See the procedure for resetting the relative coordinates described in Subsection III-12.1.2

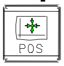
12.1.4 Workpiece Coordinate System Preset

If a workpiece coordinate system has been shifted with manual intervention or any other operation, an MDI operation can be performed to preset the system to a workpiece coordinate system that is offset by a workpiece origin offset, from the machine zero point that has been set before the shifting.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See II-7.2.4.)

Procedure for the workpiece coordinate system preset

Procedure

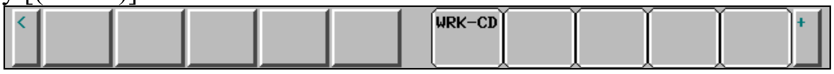
- When all axes are preset

1 Press function key 

.

2 Press chapter selection key [ABSOLUTE] to display the absolute coordinate screen.

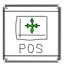
3 Press soft key [(OPRT)].



4 Press soft key [WRK-CD].

5 Press soft key [ALL AXIS].

- When a specified axis is preset

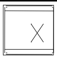
1 Press function key 

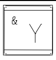
.

2 Press chapter selection key [ABSOLUTE] to display the absolute coordinate screen.

3 Press soft key [(OPRT)].

4 Press soft key [WRK-CD].

5 Enter the name of the axis (

, 

, ...) to be preset with keys.

At this time, the axis name blinks.

6 Enter 0 with corresponding numeric key and press soft key [EXEC].

Explanation

- Operation mode

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- Presetting relative coordinates

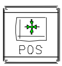
As with absolute coordinates, bit 3 (PPD) of parameter No. 3104 is used to specify whether to preset relative coordinates.

12.1.5 Actual Feedrate Display

If bit 0 (DPF) of parameter No. 3105 is set to 1, the actual machine feed per minute (actual feedrate) or per revolution can be indicated on the current position display screen (and the program check screen for the 8.4-inch display unit).

Display procedure for the actual feedrate on the current position display screen

Procedure

1 Press the function key 

to display a current position display screen. At the location indicated by , an actual feedrate is displayed.

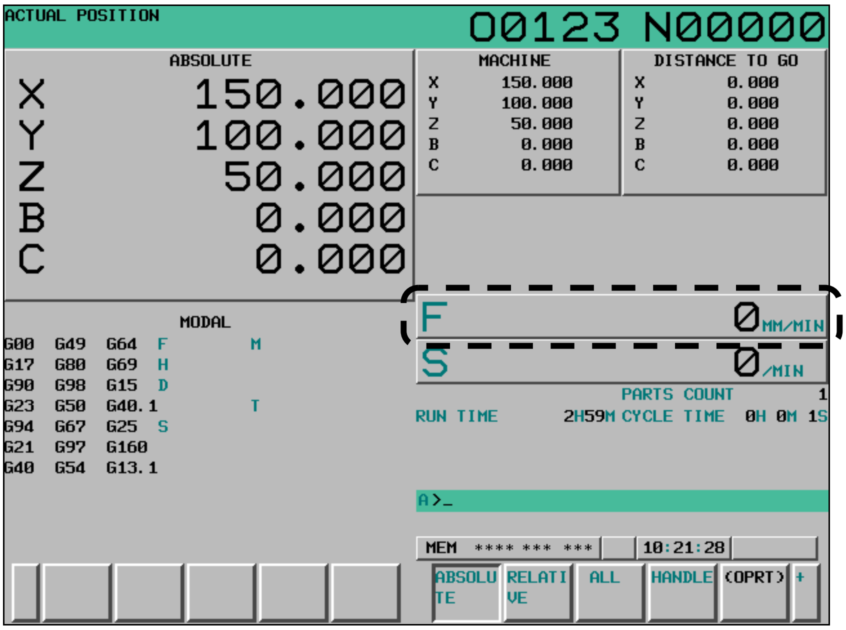


Fig. 12.1.5 (a) Current position (absolute) screen (10.4-inch)

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanation

- Actual feedrate value

The actual rate is calculated by the following expression:

$$\mathrm{Fact} = \sqrt{\sum_{i=1}^{n} (\mathrm{fi})^2}$$

n : Number of axes

fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate

Fact : Actual feedrate displayed

The display unit:

mm/min (metric input).

inch/min (Inch input, two digits below the decimal point are displayed.)

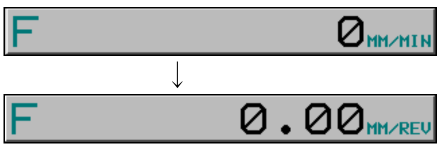
- Actual feedrate display of feed per revolution

In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- Actual feed per revolution

Switching between feed per minute display and actual feed per revolution display in the actual feedrate table can be performed depending on bit 3 (GSC) of parameter No. 3107, bit 5 (FSS) of parameter No. No.3191, the G code modal, and the operation state.

For feed per minute display, the unit is millimeter per minute. For actual feed per revolution, the unit is millimeter per revolution.



Input increment parameter INI (bit 2 of parameter No. 0) is used to switch between inch display and millimeter display. The relationship between input increment parameter INI (bit 2 of parameter No. 0) and unit display of actual feedrate and actual feed per revolution is shown below.

Table 12.1.5 (a) Unit display

	Actual feedrate	Actual feed per revolution
Inch (INI=0)	MM/MIN	MM/R
Millimeter (INI=1)	INCH/MIN	INCH/R

- Actual feedrate display switching condition

Actual feedrate display is switched as shown in the table below depending on the bit 3 (GSC) of parameter No. 3107, bit 5 (FSS) of parameter No. No.3191, the G code modal, and the operation state.

Table 12.1.5 (b) Displayed information switching condition

GSC	FSS	Operation state	G code modal	Actual feedrate
0	-	-	-	Feed per minute
1	0	Manual feed Rapid traverse Dry run	-	Feed per minute
		Other than Manual feed, Rapid traverse, and Dry run	M series: G93, G94 T series: G98(G code system A) G93, G94(G code system B, C)	Feed per minute
			M series: G95 T series: G99(G code system A) G95(G code system B, C)	Feed per revolution
	1	-	-	Feed per revolution

- Actual feedrate display of rotary axis

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 50 mm/min (in metric input) or 0.50 inch/min (in inch input).

- Actual feedrate display on the other screen

For the 8.4-inch display unit, the actual feedrate is displayed on the program check screen.
It is also displayed on the graphic screen.

- Displaying the number of decimal places

For actual feedrate display, the number of decimal places is indicated as described below depending on the setting of parameter No. 3135 and the input unit.

Setting 0: Millimeter input   With no decimal point
1: Millimeter input   One digit to the right of the decimal point
2: Millimeter input   Two digits to the right of the decimal point
3: Millimeter input   Three digits to the right of the decimal point

For inch input, the number of decimal places is the setting plus 2.

For feed per revolution display, the number of decimal places is described below.
When the input unit is millimeter, the number of displayed decimal places is two.
When the input unit is inch, the number of displayed decimal places is three.

* The number of decimal places is fixed, so it cannot be changed by a parameter or the like.

12.1.6 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

Procedure

1 Press the function key POS to display a current position display screen. "PARTS COUNT", "RUN TIME", and "CYCLE TIME" are displayed in the area enclosed by .

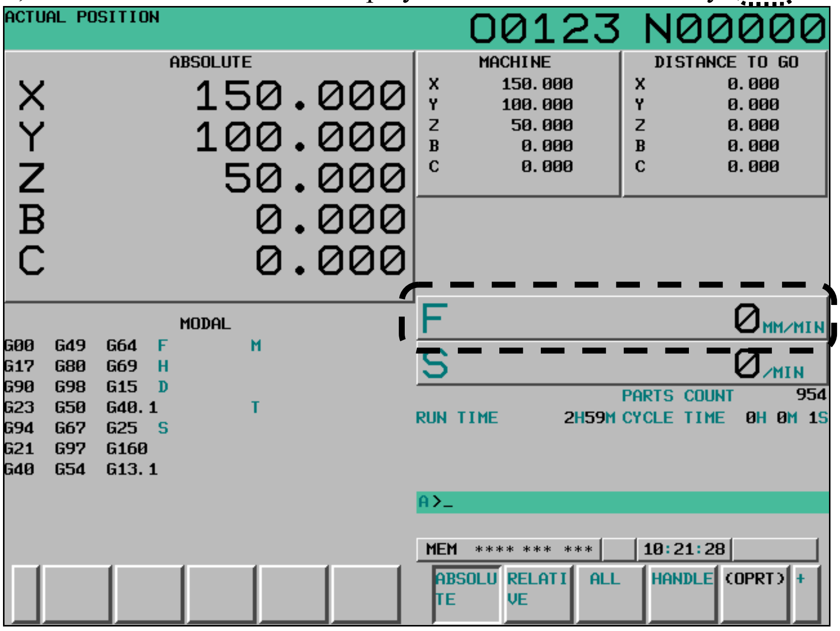


Fig. 12.1.6 (a) Current position (relative) screen (10.4-inch)

Explanation

- PART COUNT

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed.

- Incrementing the number of machined parts

Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- Display on the other screen

Details of the run time and the number of machined parts are displayed on the setting screen. See III-12.3.3.

- Parameter setting

The number of machined parts and run time cannot be set on current position display screens.
The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters Nos. 6711, 6751, and 6752 or on the setting screen.

12.1.7 Operating Monitor Display

The load meter for a servo axis can be displayed. Also, the load meter and speed meter for a serial spindle can be displayed.
To enable this function, bit 5 (OPM) of parameter No. 3111 must be set to 1.

Procedure for displaying the operating monitor

Procedure

1 Press function key .

2 Press continuous menu key .

3 Press soft key [MONITOR].

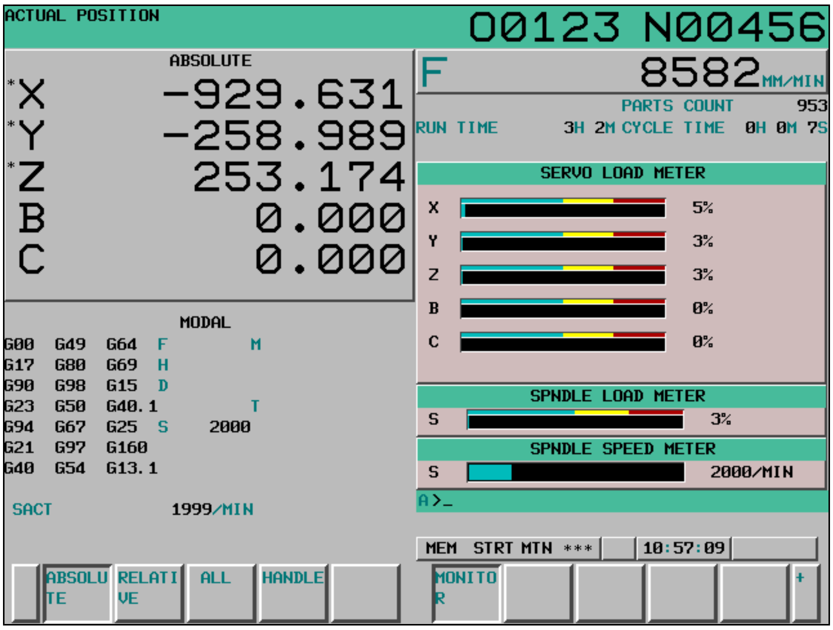


Fig. 12.1.7 (a) Operating monitor (10.4-inch)

Explanation

- Display of the servo axes

Servo axis load meters as many as the maximum number of controlled axes of the path can be displayed.
For the 10.4-inch display unit, up to five axes can be concurrently displayed on one screen. If soft key [MONITOR] is pressed, the load meters of the sixth and subsequent axes are displayed.
For the 8.4-inch display unit, up to four axes can be concurrently displayed on one screen. If soft key [MONITOR] is pressed, the load meters of the fifth and subsequent axes are displayed.
To display up to five axes concurrently on one screen of the 8.4-inch display unit as in the 10.4-inch display unit, set bit 4 (9DE) of parameter No. 11350 to 1.

- Display of the spindle axes

Both a load meter and spindle meter can be displayed for the first spindle when a serial spindle is used.

- Unit of graph

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%).
The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- Load meter

The reading on the load meter depends on servo parameter No. 2086 and spindle parameter No. 4127.

- Speedometer

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter No. 3111 to 1.
The spindle speed to be displayed during operation monitoring is calculated from the speed of the spindle motor (see the formula below).
The spindle speed can therefore be displayed, during operation monitoring, even when no position coder is used.
To display the correct spindle speed, however, the maximum spindle speed for each gear (spindle speed at each gear ratio when the spindle motor rotates at the maximum speed) must be set in parameters No. 3741 to No.3744.
Input of the clutch/gear signal for the first spindle of the serial spindle is used to determine the gear currently selected, so input of signals CTH1A and CTH2A must be controlled according to the gear selection with reference to the table below.
(Formula for calculating the spindle speed to be displayed)

$$\text{Spindle speed displayed during operation monitoring} = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor}} \times \text{Maximum spindle speed with the gear being used}$$

The following table lists the correspondence between clutch and gear selection signals CTH1A and CTH2A, used to determine the gear being used, and parameters:

CTH1A	CTH2A	Parameter	Serial spindle specification
0	0	=No.3741 (Maximum spindle speed with gear 1)	HIGH
0	1	=No.3742 (Maximum spindle speed with gear 2)	MEDIUM HIGH
1	0	=No.3743 (Maximum spindle speed with gear 3)	MEDIUM LOW
1	1	=No.3744 (Maximum spindle speed with gear 4)	LOW

The speed of the spindle motor and spindle can be displayed, during operation monitoring, only for the first serial spindle and the spindle switching axis for the first serial spindle. It cannot be displayed for the second spindle.

- Color of graph

If the value of a load meter exceeds 100%, the bar graph turns yellow.

12.1.8 Display of Axes in 2-path System Simultaneously

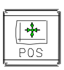
T

In the current position display screen of 8.4-inch display unit, the axes information for 2-path system can be displayed without switching the system.
The display of axes in 2-path system simultaneously becomes effective in the following screens.

- The total position display screen
- The manual handle interruption amount display screen

Procedure for displaying the total position display screen and the manual handle interruption amount display screen

Procedure

1 Press function key .

2 Press chapter selection key [ALL] to display the relative total position display screen.
Press chapter selection key [HNDL] to display the manual handle interruption amount display screen.

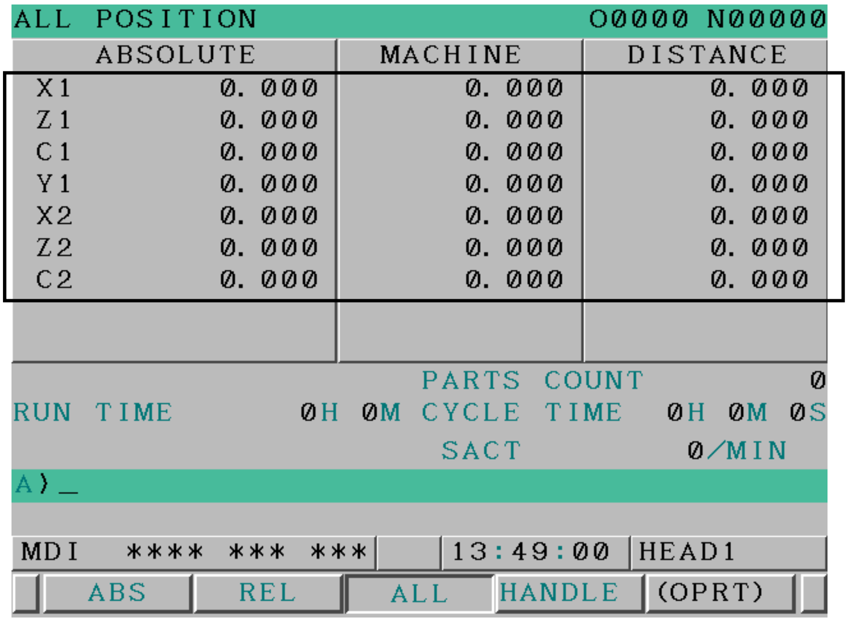


Fig. 12.1.8 (a) The display of axes in 2-path system simultaneously (in the total position display screen)

Explanation

- Condition to display

The display of axes in 2-path system simultaneously becomes effective when all conditions of 1-5 the following is full.

1 2-path system (TT system) is used.
2 8.4-inch display unit is used.
3 The simultaneous display of 2-path is enabled (bit 2 (DOP) of parameter No. 3193 = 0).
4 The number of axes that can be displayed on one screen of the 8.4-inch display unit is up to 5 (bit 4 (9DE) of parameter No. 11350 = 1).
5 The current position display screen displays two paths regardless of path selection signal HEAD<G0063.0> (bit 1 (DIP) of parameter No.3103 = 0).

- The order of displaying each path

In the current position display screen, if the parameter which decides display order of each path (bit 7 (SBA) of parameter No.3101) is set to 1, the axes information in the 2nd path can be displayed ahead.
When 0 is set in it, the axes information in the 1st path is displayed ahead.

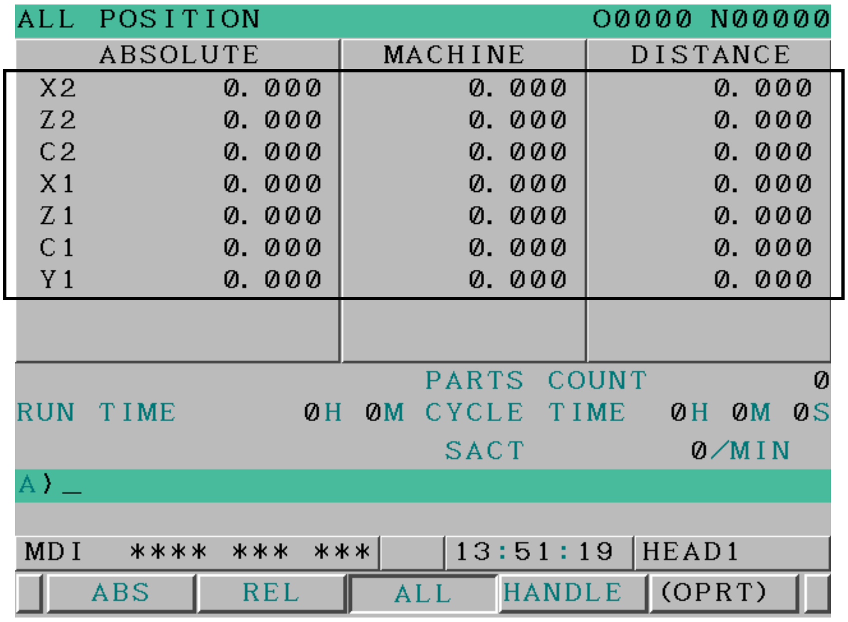


Fig. 12.1.8 (b) The axes information is displayed in order in the 2nd -> 1st path (in the total position display screen)

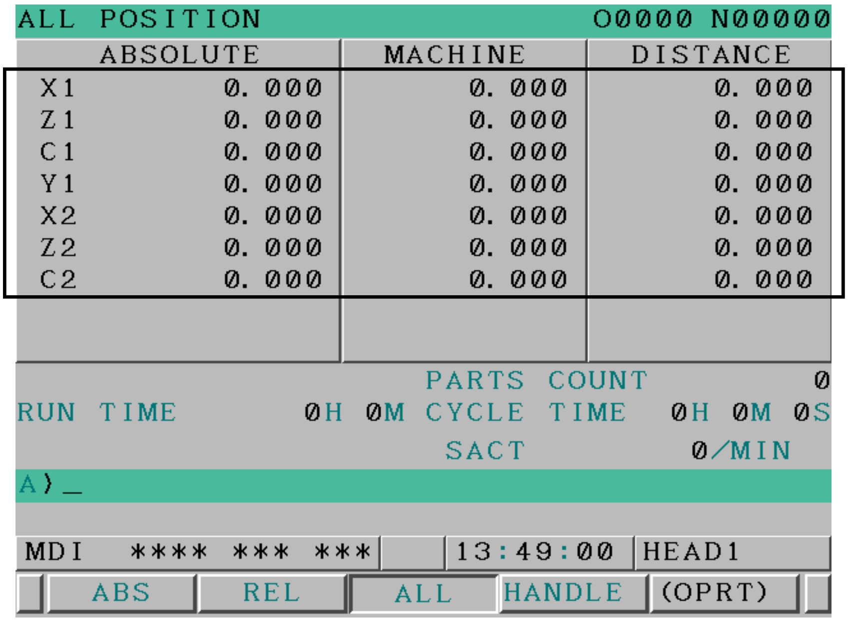


Fig. 12.1.8 (c) The axes information is displayed in order in the 1st -> 2nd path (in the total position display screen)

- The order of displaying each axis

The order of the axes displayed in the current position display screen can be specified (parameter (No.3130)). The order of the axes which can be specified is limited in order in each path. The order of the axes cannot be specified across the axis in the other path.

Before the replacement		Parameter No.3130		After the replacement
X1		4		Y1
Z1		3		C1
C1	→	2	→	Z1
Y1		1		X1
X2		3		C2
Z2		2		Z2
C2		1		X2

Fig. 12.1.8 (d) Specification of the order of displaying each axis (specification with axes in each path)

Before the replacement		Parameter No.3130		After the replacement
X1		7		(Blank)
Z1		6		(Blank)
C1	→	5	→	(Blank)
Y1		4		Y1
X2		3		C2
Z2		2		Z2
C2		1		X2

Fig. 12.1.8 (e) Specification of the order of displaying each axis (specification with axes across the other path)

- Non-display of the axis

When the order of displaying the axes is specified, the axis for which 0 is specified in the order of the display becomes non-display.

Before the replacement		Parameter No.3130		After the replacement
X1		1		X1
Z1		2		Z1
C1	→	0	→	(Blank)
Y1		4		Y1
X2		1		X2
Z2		2		Z2
C2		0		(Blank)

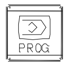
Fig. 12.1.8 (f) Specification of non-display of the axis

- Display of axis with top-alignment

To the area where the current position display becomes blank by being set as non-display, the current position is displayed with top-alignment by another axis for which the current position is displayed by setting 1 to bit 0 (TAD) of parameter No.13102.

Before displaying the axis with top-alignment	Parameter No.13102#0	After displaying the axis with top-alignment
X1	0 -> 1	X1
Z1		Z1
(Blank)		Y1
Y1		X2
X2		Z2
Z2		(Blank)
(Blank)		(Blank)

Fig. 12.1.8 (g) Specification of display of axis with top-alignment

12.2 SCREENS DISPLAYED BY FUNCTION KEY 

Section 12.2, "SCREENS DISPLAYED BY FUNCTION KEY ", consists of the following subsections:

This section describes the screens displayed by pressing function key . There are the program editing screen, the program list screen, and the screen for displaying the status of the command specified by the program currently being executed.

1. Program screen
2. Program list screen
3. Next block display screen
4. Program check screen (8.4-inch LCD)

On the program screen, you edit the program that is currently selected, and display the block that is currently executed during program operation. In MDI mode, you also edit an MDI operation program, and display the block that is currently executed.

12.2.1 Program Contents Display

Displays the program currently being executed in MEM mode.

Displaying the program being executed

Procedure

1 Press function key  to display the program screen.

2 Press chapter selection soft key [PROGRAM].
The cursor is positioned at the block currently being executed.

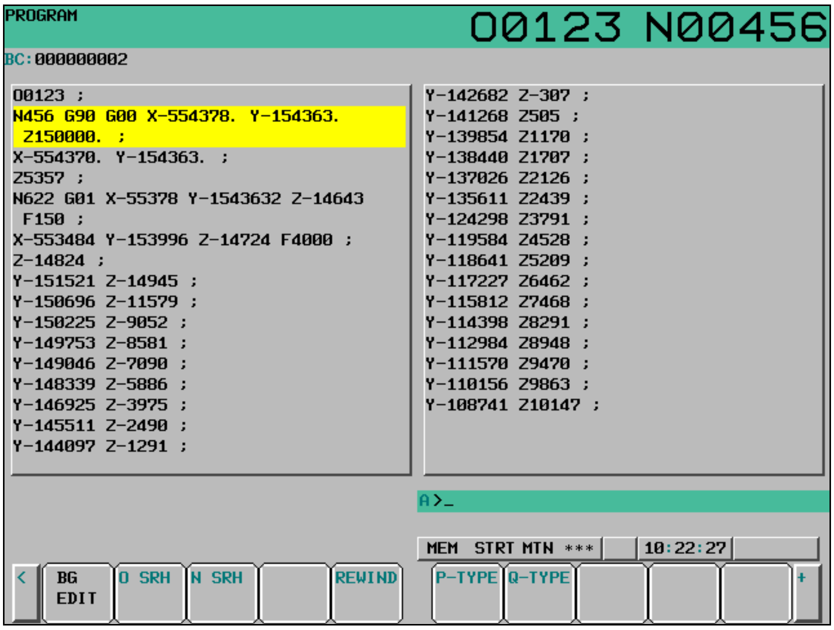


Fig. 12.2.1 (a) Screen for displaying the program being executed (full screen display) (10.4-inch)

For the 10.4-inch display unit, if soft key [PROGRAM] is pressed again to switch screen display to full screen display or small screen display.
In small screen display, position display and modal display are allowed at the same time.
In full screen display, much information about a program can be displayed at a time.

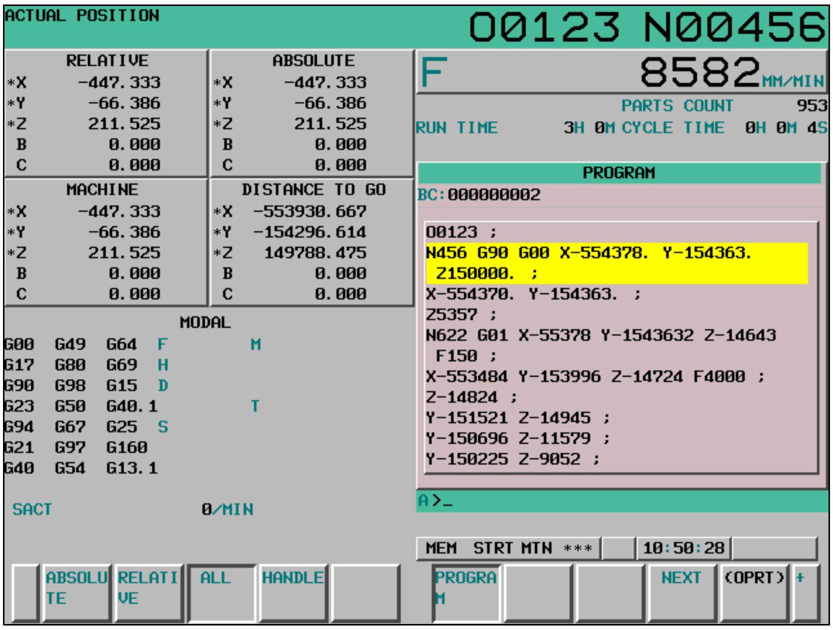


Fig.12.2.1 (b) Screen for displaying the program being executed (small screen display) (10.4-inch)

12.2.2 Editing a Program

A program can be edited in the EDIT mode.
A program can be edited on a word-by-word basis.
For program creation and editing operation, see Chapter III-9, "CREATING PROGRAMS" and Chapter III-10, "EDITING PROGRAMS".

Displaying the program editing screen

Procedure

1 Press function key PROG to display the program screen.

2 Press chapter selection soft key [PROGRAM].

Editing operations such as text insertion, modification, and deletion, and cursor movements are performed on a word-by-word basis.

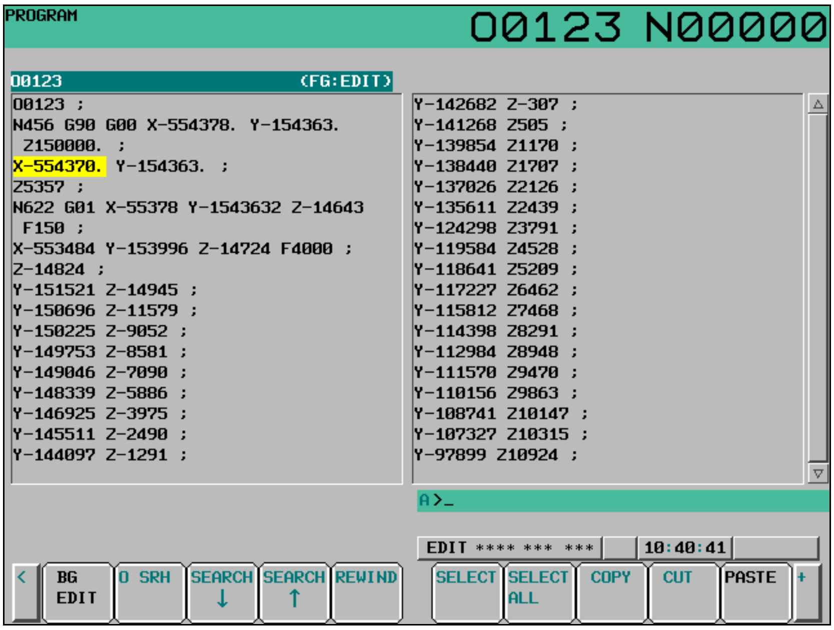


Fig. 12.2.2 (a) Program editing screen (quarter screen display) (10.4-inch)

For the 10.4-inch display unit, if soft key [PROGRAM] is pressed again during editing to switch screen display to full screen display or small screen display.
In small screen display, position display and modal display are allowed at the same time.
In full screen display, much information about a program can be displayed at a time.

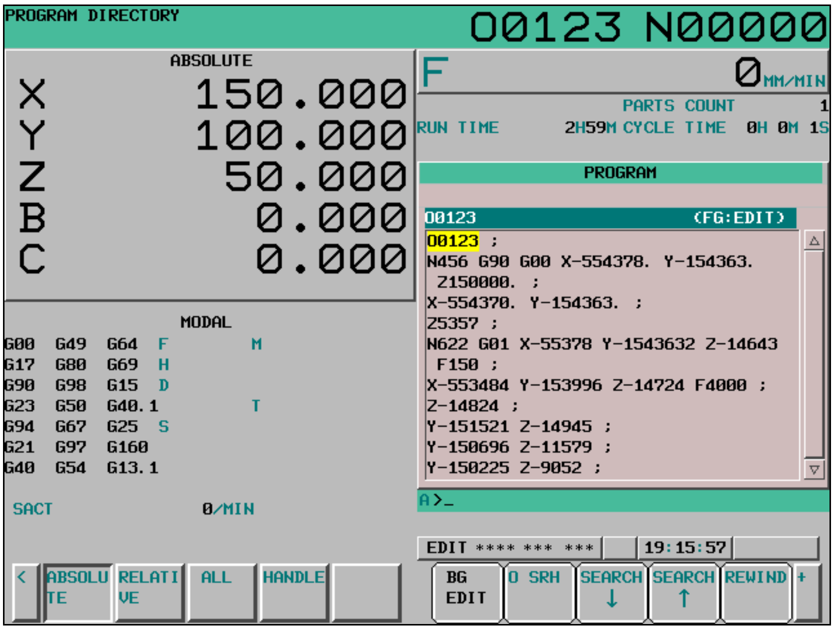


Fig. 12.2.2 (b) Program editing screen (small screen display) (10.4-inch)

12.2.3 Program Screen for MDI Operation

During MDI operation or editing of an MDI operation program in the MDI mode, the program currently being executed mode is displayed.
For MDI operation, see Section III-4.2, “MDI Operation”.

Procedure for displaying the program screen for MDI operation

Procedure

1 Enter the MDI mode.

2 Press function key .

3 Press chapter selection soft key [MDI].
The program input from the MDI is displayed.

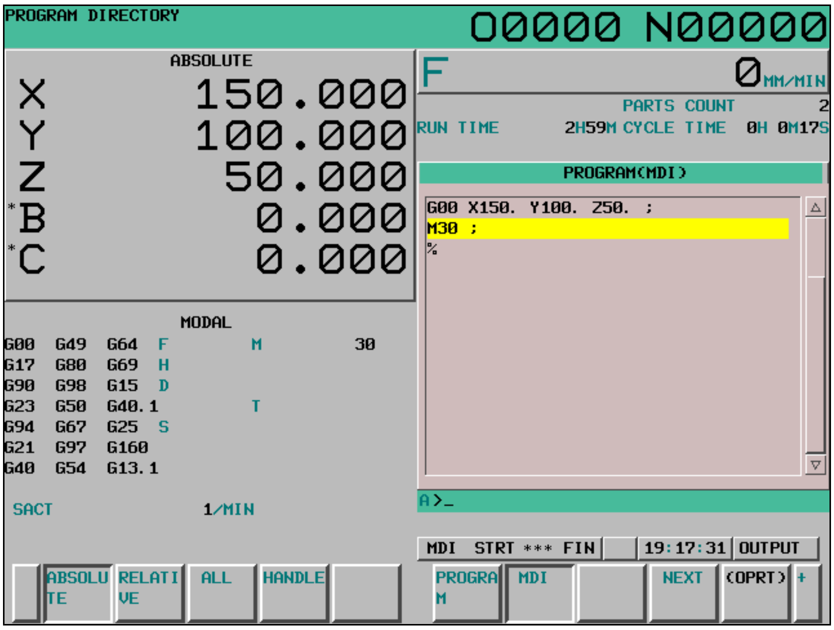


Fig. 12.2.3 (a) MDI operation program screen (10.4-inch)

In case of the 8.4-inch display unit, when bit 7 (MDL) of parameter No.3107 is set 1, the modal information can be displayed in the screen.

12.2.4 Program List Screen

A list of programs registered in the program memory is displayed.
The program list screen can be displayed in any mode.
For the program list screen, see Chapter III-11, “PROGRAM MANAGEMENT”.

Displaying the program list screen

Procedure

1 Press function key .

2 Press continuous menu key .

2 Press chapter selection soft key [LIST].

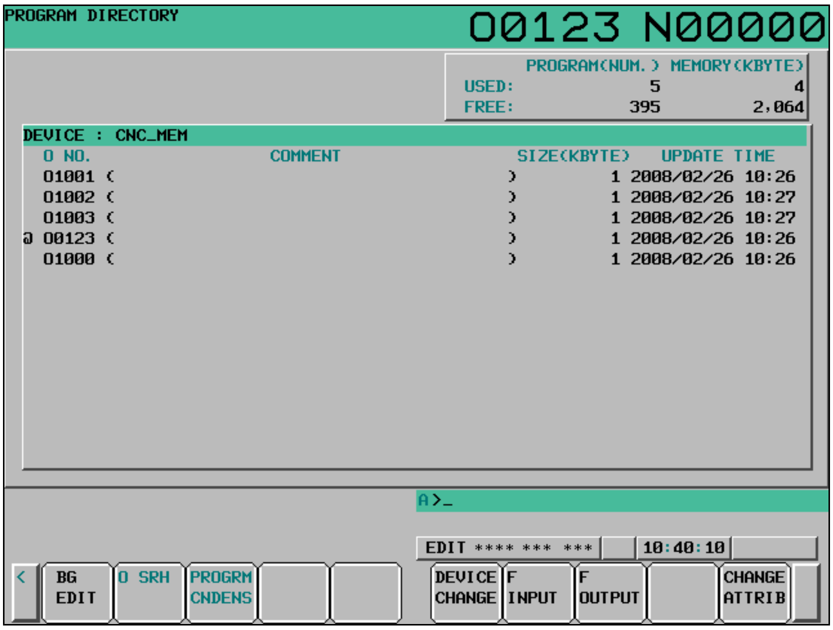


Fig. 12.2.4 (a) Program list screen (10.4-inch)

12.2.5 Next Block Display Screen

The block currently being executed and the block to be executed next are displayed in the MEM mode and MDI model.

Procedure for displaying the next block display screen

Procedure

1 Press function key .

2 Press chapter selection soft key [NEXT].
The G codes, addresses, command values specified in the block currently being executed and the next block are displayed.

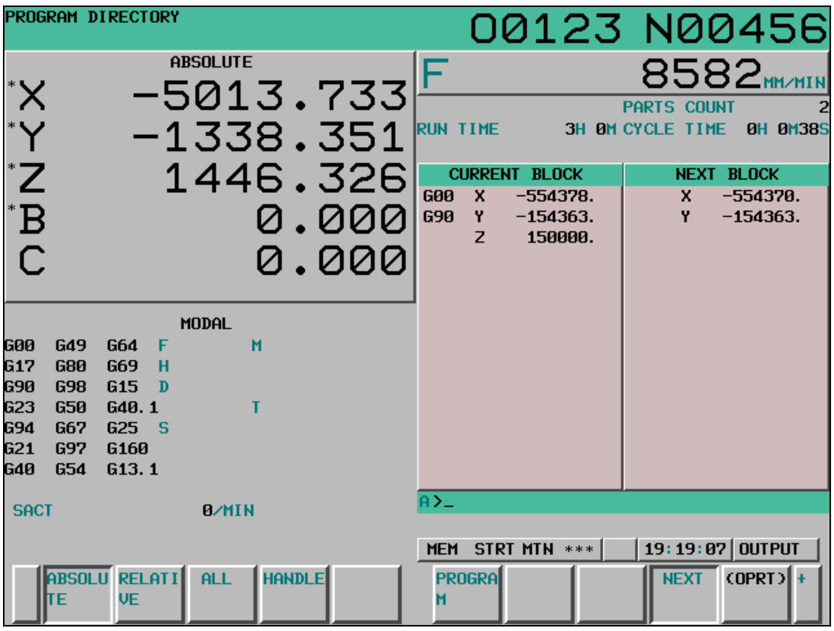


Fig. 12.2.5 (a) Next block display screen (10.4-inch)

NOTE
1 If a reset is made during execution of a program, the display of the current block and next block is cleared.
2 When the feed hold state (HOLD) is caused between the block and the block during the program execution, the next block display is cleared.

12.2.6 Program Check Screen

The program currently being executed, the current position and modal information are displayed in the MEM mode. This screen is displayed when the simultaneous display of 2-path is enabled in the 2-path system.

Procedure for displaying the program check screen

Procedure

1 Press function key 

.

2 Press chapter selection soft key [CHECK].
The program currently being executed, current position of the tool, and modal data are displayed.

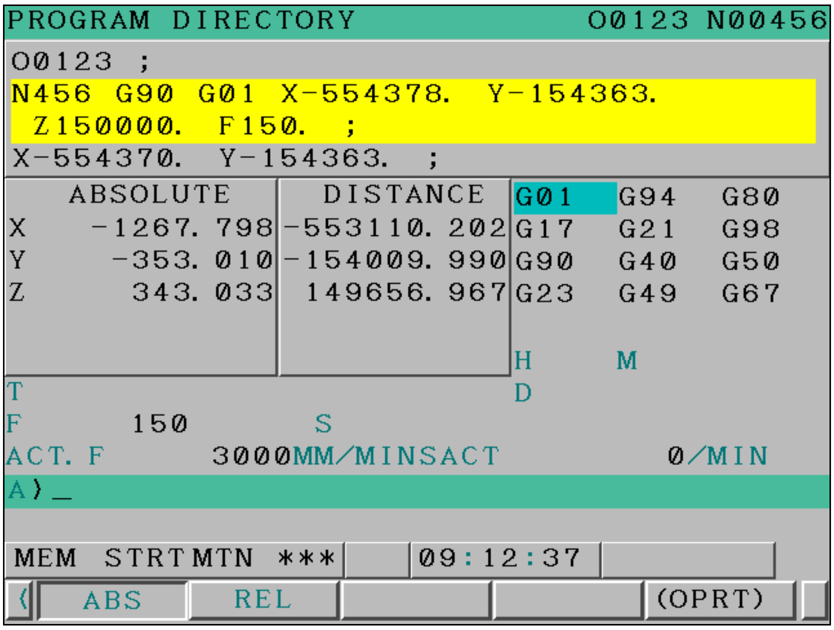


Fig. 12.2.6 (a) Program check screen (8.4-inch)

Explanation

- Program display

Up to four blocks from the beginning of the block being executed in the program currently being executed are displayed.
The block being executed is displayed in reverse video.

- Current position display

Chapter selection soft key [RELATIVE] or [ABSOLUTE] can be pressed to switch between relative coordinate system display and workpiece coordinate system display.

- Modal G codes

Up to 12 modal G codes are displayed.

- Display of the actual speed and SACT

The actual speed is displayed with mm/min or inch/min according to unit of input.
The actual spindle speed (SACT) is displayed.

The simultaneous display of 2-path (for the 10.4-inch display unit)

T

The simultaneous display of 2-path can be done at the TT (2-path) system.
In the 10.4-inch display unit, chapter selection soft key [CHECK] is displayed by making this function effective.
It is necessary to set 1 to bit 0 (DHD) of parameter No.3106 and 0 to bit 2 (DOP) of parameter No.3193 to make this function effective.

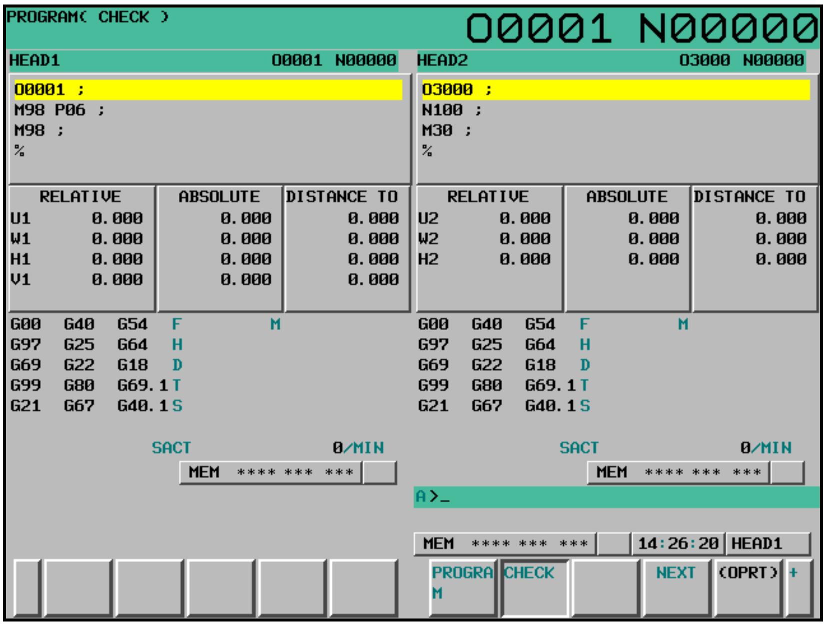


Fig. 12.2.6 (b) The simultaneous display of 2-path at program check screen (for the 10.4-inch display unit)

Switch of the display of 1-path and the simultaneous display of 2-path (for the 8.4-inch display unit)

T

In the 8.4-inch display unit, the display can be switched to the display of 1-path and the simultaneous display of 2-path by pressing operation selection soft key.
It is necessary to set 0 to bit 0 (DHD) of parameter No.3106 and 0 to bit 2 (DOP) of parameter No.3193 to make this function effective.
When the power is turned on, the display of 1-path has been selected.

Procedure

1. Press operation selection soft key [M-PATH].
2. The program check screen changes from the display of 1-path to the simultaneous display of 2-path.
   The display of the soft key changes from [M-PATH] to [S-PATH].
3. Press operation selection soft key [S-PATH].
4. The program check screen changes from the simultaneous display of 2-path to the display of 1-path.
   The display of the soft key changes from [S-PATH] to [M-PATH].

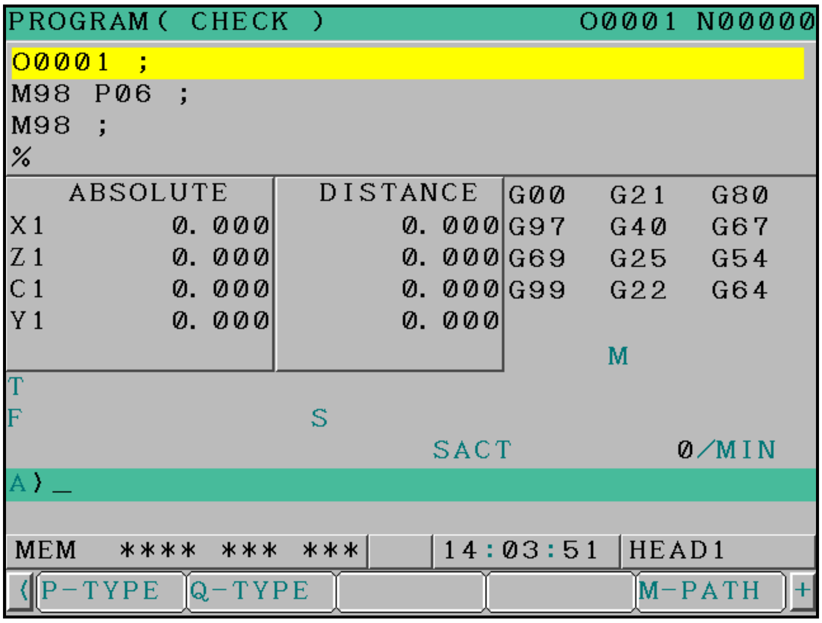


Fig. 12.2.6 (c) The display of 1-path at program check screen (for the 8.4-inch display unit)

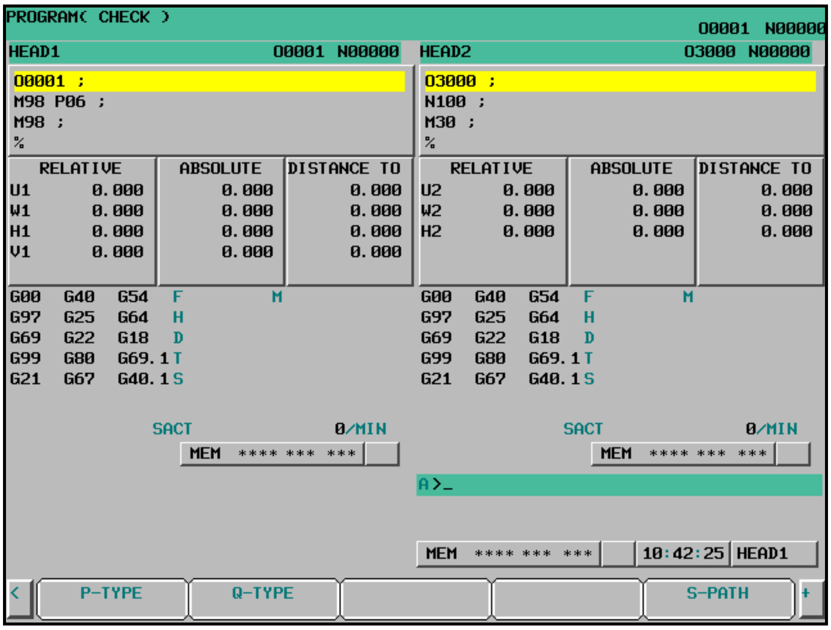


Fig. 12.2.6 (d) The simultaneous display of 2-path at program check screen (for the 8.4-inch display unit)

Load meter and speedometer display (for the 8.4-inch display unit)

In the 8.4-inch display unit, the display can be switched to the display of the amount of the movement to be made and modal information and the display of spindle load meter and spindle speedometer by pressing operation selection soft key.

It is necessary to set 1 to bit 0 (SMS) of parameter No.3117 to make this function effective.

Procedure

1. Press operation selection soft key [MONI].
2. The display of the amount of the movement to be made and modal information changes to the display of spindle load meter and spindle speedometer on the program check screen.
   The display of the soft key changes from [MONI] to [D.GO].
3. Press operation selection soft key [D.GO].

4 The display of spindle load meter and spindle speedometer changes to the display of the amount of the movement to be made and modal information on the program check screen.
The display of the soft key changes from [D.GO] to [MONI].

Explanation

- Target spindle to display

Only load meter and speedometer of the first serial spindle can be displayed.

- Load meter

The display of load meter depends on setting the spindle parameter No.4127.
The graph of load meter is displayed up to 200%.

- Speedometer

Speedometer displays the spindle motor speed, but it is possible to change to the display of spindle speed by setting as bit 6 (OPS) of parameter No.3111 = 1.
Displayed spindle speed is calculated by using the spindle motor speed. (Refer to the expression below)
Therefore, the spindle speed can be displayed even if there is no position coder.
However, It is necessary to set parameter No.3741-3744 (the maximum spindle speed at each gear) correctly to display correct spindle speed.
Moreover, the gear under the selection is judged by using clutch/gear signals to the first serial spindle.

Expression for spindle speed display

$$\text{Displayed spindle speed=} \quad \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed}} \times \begin{array}{c}\text{The maximum spindle speed at gear}\\ \text{under the selection}\end{array}$$

12.2.7 Current Block Display Screen (Only for the 8.4-Inch Display Unit)

The block currently being executed and modal specification value are displayed in the MEM mode and MDI mode.

Displaying the current block screen

Procedure

1 Press function key  .

2 Press chapter selection soft key [CURRENT].
The modal specification values, specification values, addresses, and G codes specified in the block currently being executed are displayed.

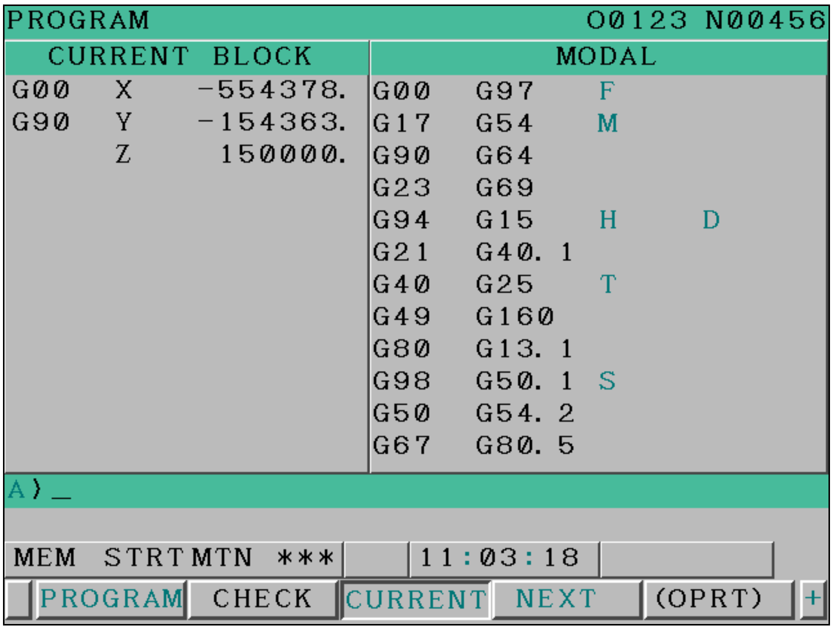


Fig. 12.2.7 (a) Current block screen (8.4-inch)

12.2.8 Graphical Conversational Programming Screen

The G code menu and details on G codes are displayed in the EDIT mode.
A program can be created one block at a time while seeing the G code menu on the screen and the G code detail screen.
For details on the conversational programming with the graphic function, see Chapter III-11, "PROGRAM CREATION".

Displaying the graphic conversational programming screen

Procedure

1 Press function key .

2 Press continuous menu key .

3 Press chapter selection soft key [C.A.P].

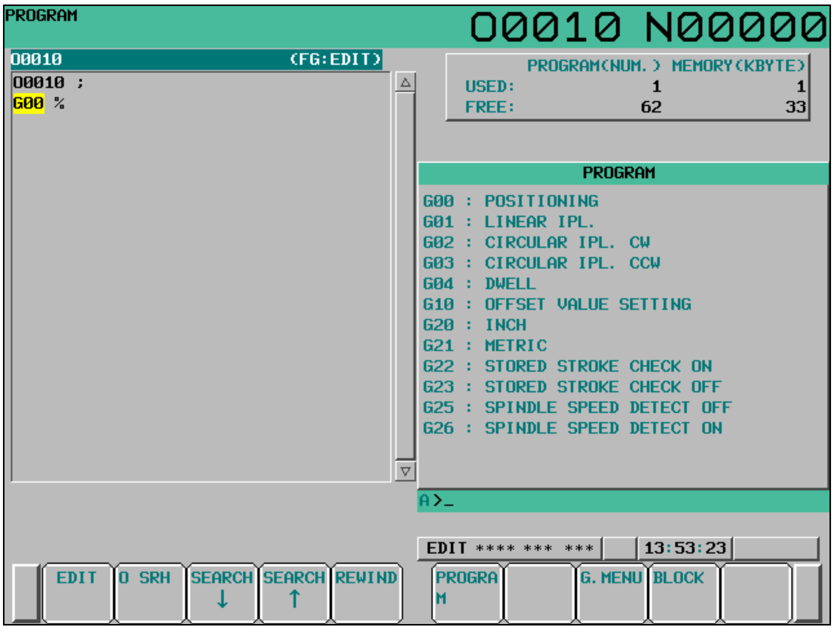


Fig. 12.2.8 (a) Graphical conversational programming screen (G code menu screen) (10.4-inch)

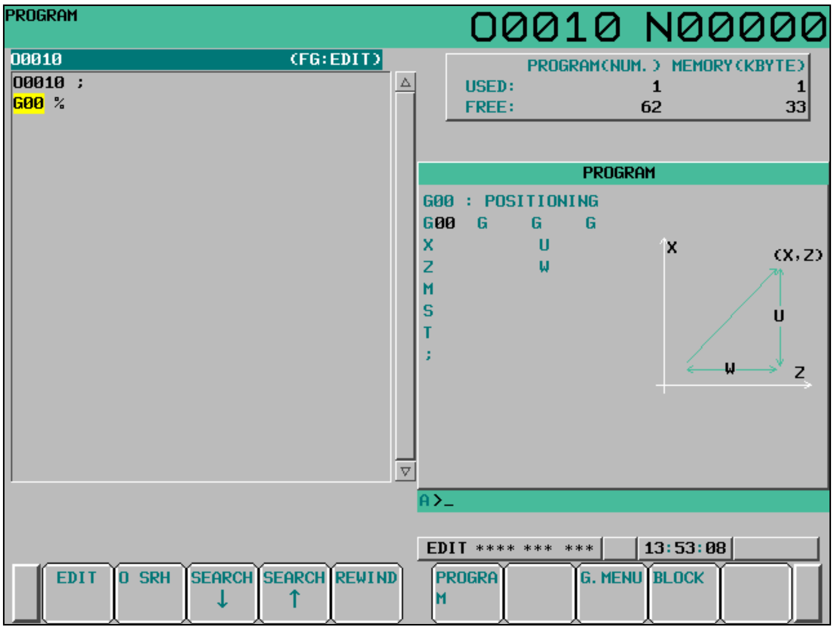


Fig. 12.2.8 (b) Graphical conversational programming screen (G code detail screen) (10.4-inch)

The G code menu screen and G code detail screen are present on the graphical conversational programming screen.
To display the G code detail screen, press soft key [BLOCK]. To display the G code menu screen, press soft key [G. MENU].

G code menu screen

On the G code menu screen, select a G code to be specified from a list of G codes displayed.

G code detail screen

On the G code detail screen, enter the necessary values since the specification form for each G code is displayed.

12.2.9 Background Editing

Editing of a program other than the main program is called background editing. In background editing, a program can be edited during execution of another program and the same editing operations as normal editing (foreground editing) can be performed. There are the editing mode and reference mode in background editing. A program can be edited in the editing mode, but a program cannot be edited in the reference mode. In the reference mode, a program being operated can be referenced.
Background editing can be performed in any mode.

Function

- Editable programs

The editable programs for each device are shown below.

Editable programs for each device

Device	Editable program
CNC built-in memory	Program in the selected path
MEMCARD	All programs
Data server	All programs

- Number of programs editable in the background

The number of programs editable in the background is only one for each path.

- Background editing on a 2-path system

Each path has its own editing status, background or foreground. If the editing status of path 1 is background and that of path 2 is foreground, therefore, switching from path 1 to path 2 causes the program of path 2 being subjected to foreground editing to be displayed on the program editing screen and, by contrast, switching from path 2 to path 1 causes the program of path 1 being subjected to background editing to be displayed.
In addition, if simultaneous editing of programs of the both paths is enabled (bit 0 (DHD) of parameter No. 3106 = 1 and bit 2 (DOP) of parameter No. 3193 = 0), starting background editing for one of the paths causes background editing for the other path to be started; hence simultaneous background editing of programs for both the paths.
A program for one path can be displayed along with one for the other path during background editing. It is possible to switch between the edit target programs displayed side by side for copy and paste; hence efficient program editing.

- Screen display

When background editing starts, the normal editing screen is switched to the background editing screen. The program name and "BG: EDIT", which indicates that background editing is in progress, appear in the program status line above the program.
When background editing is performed in the reference mode, "BG: READ ONLY" appears in the program status line. The characters are displayed in green when the program is displayed in the color mode.

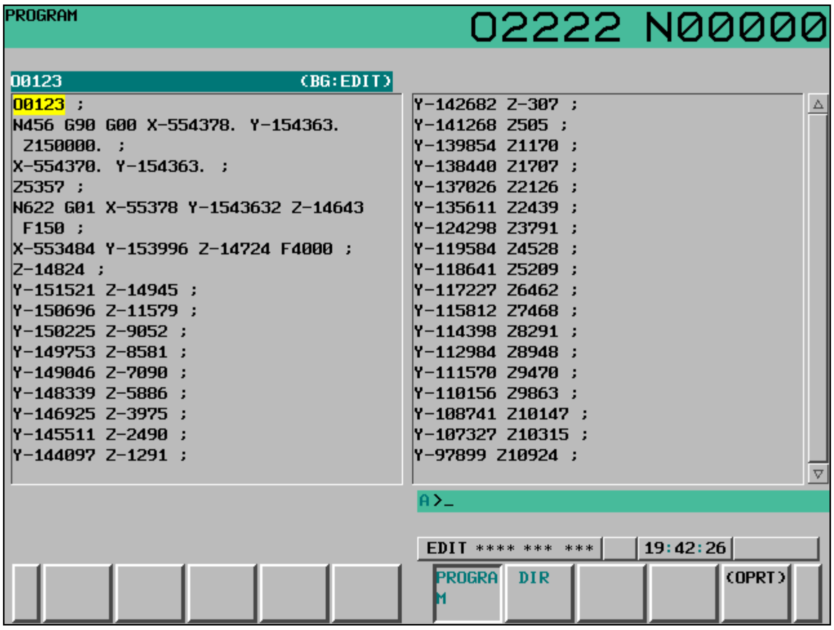


Fig. 12.2.7 (a) Screen being edited in the background (editing mode) (10.4inch)

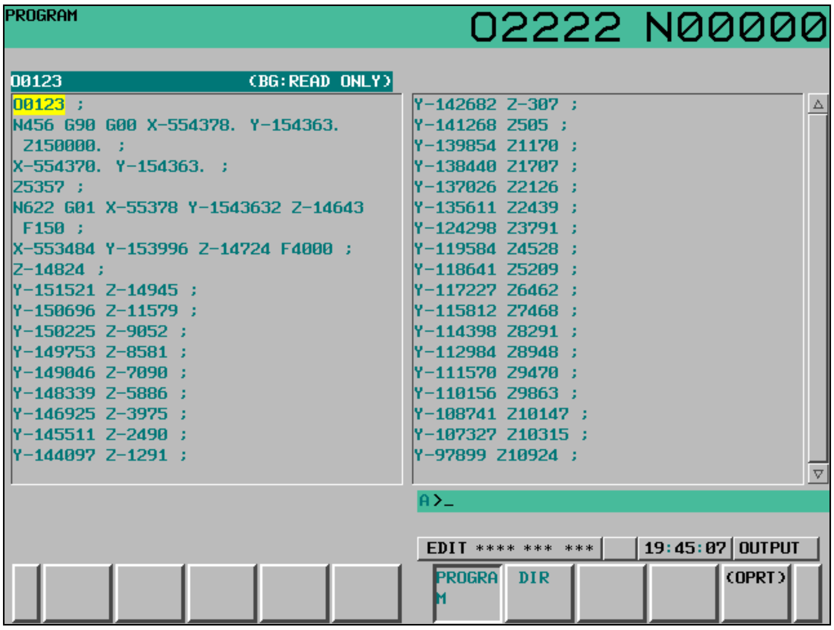


Fig. 12.2.7 (b) Screen being edited in the background (reference mode) (10.4inch)

- Editing status

The following items are displayed on the program status line and program editing area according to the background editing status.

Displayed items in the program status line and editing area

Editing status	Displayed items
No program selected	(BG-EDIT) “NO PROGRAM” is displayed in the editing area.
No program selected Reference mode	(BG:READ ONLY) "NO PROGRAM" is displayed in the editing area.
Program selected	program-name + (BG:EDIT)
Program selected Reference mode, Read only	program-name + (BG:READ ONLY) The contents of the program are displayed in green.

- Operation in the background editing

In the reference mode, a program being subject to background editing can be operated.

- Switching to full display or small display

For the 10.4-inch display unit, if soft key [PROGRAM] is pressed again during the background editing, the screen size is switched to full display or small display.
In small screen display, position display or modal display can be selected concurrently with the program being subject to background editing.
In full screen display, much information about the program being subject to background editing can be displayed at a time.

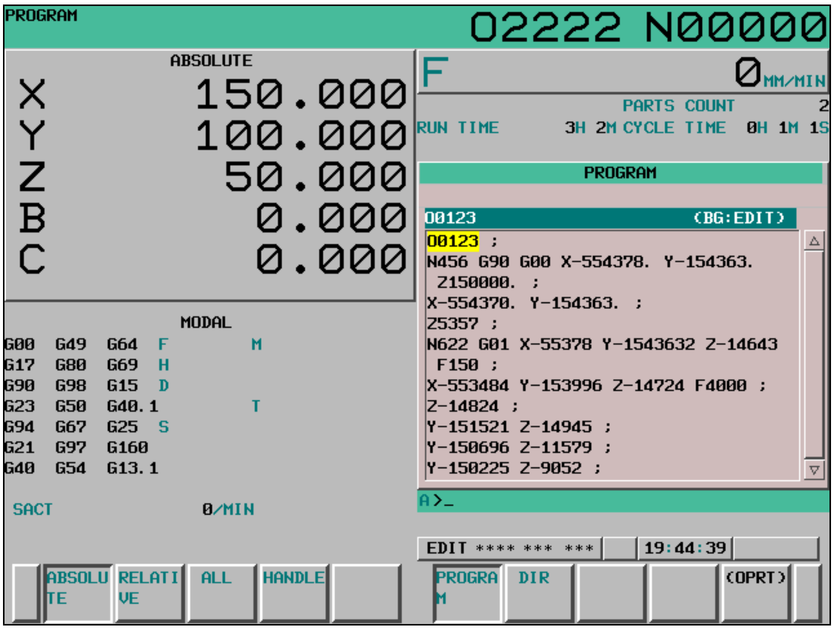


Fig. 12.2.7(c) Program screen (small screen) (10.4-inch)

Starting of background editing

Background editing can be started from the following screens.
- PROGRAM
- PROGRAM LIST
- MDI PROGRAM
- PROGRAM CHECK
- CURRENT BLOCK
- NEXT BLOCK
- RESTART

Procedure

Method 1 When background editing is performed in the editing mode

1. Display a screen on which background editing can be started.
2. Press soft key [(OPRT)].
3. Press soft key [BG EDIT].

4. Enter the number of the program to be opened in the editing mode into the key-in buffer.

5 Press soft key [EDIT EXEC] ([EDIT] for the 8.4-inch display unit).
The program with the entered number is opened during background editing in the editing mode. If the program with the entered number is not present, a new program is created and the program is opened during background editing.

Method 2 When background editing is performed in the reference mode

1 Press function key PROG.
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)].
4 Make sure that there is no character string in the key-in buffer and press soft key [BG EDIT].

5 Enter the number of the program to be opened in the reference mode into the key-in buffer.
6 Press soft key [REF EXEC] ([REF EXE] for the 8.4-inch display unit).
The program with the entered number is opened during background editing in the reference mode. If the program with the entered number is not present, a warning message “SPECIFIED PROGRAM NOT FOUND” appears

NOTE

1 A program in the place where the main program is present can be specified for background editing (or a program in the CNC built-in memory if no main program is present) can be specified for background editing. On the program list screen, however, the program of the displayed device can be opened for background editing. When editing a program not in the place where the main program is present in the background, select the program on the program list screen.
2 When opening an O-number program for background editing, address O can be omitted.
3 The mode selected at the start of background editing is kept until it is finished. To switch between the editing mode and the reference mode, finish background editing and then start background editing in the desired mode again.
4 Even when background editing is performed in the editing mode, if either of the following programs is specified, the program is opened temporarily in the reference mode.
   - Program being operated
   - Main program
5 The device for which a new program is created at the start of background editing is only the CNC built-in memory. For the MEMCARD or data server device, specify an existing program because no program is created.

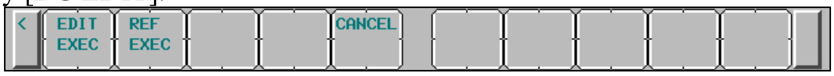
Selecting a program subject to background editing with the cursor

When the device being displayed is the MEMCARD or data server device, the program name at the cursor position can be stored in the key-in buffer to be opened for background editing.

Procedure

1 Press function key PROG.
2 Press soft key [DIR].
(When [LIST] is not displayed, press continuous menu key ▷.)
3 Press soft key [(OPRT)].

4 Change the device being displayed to the MEMCARD or data server.
For changing a device, see III-11.1.
5 Move the cursor to the program to be opened for the background editing.
6 Press soft key [BG EDIT].



7 Press [EDIT EXEC] ([EDIT EXE] for the 8.4-inch display unit) or [REF EXEC] ([REF EXE] for the 8.4-inch display unit) to start background editing.

Ending background editing

Procedure

1 Press function key .

2 Press soft key [PROGRAM] or [DIR].
3 Press soft key [(OPRT)] and press [BG-END].
Background editing is ended and this screen is switched to the screen in the program function last displayed in the current CNC mode before background editing.

12.3 SCREENS DISPLAYED BY FUNCTION KEY

Section 12.3, “SCREENS DISPLAYED BY FUNCTION KEY ”, consists of the following subsections:

Press function key to display or set tool compensation values and other data.

This section describes how to display or set the following data:

1. Tool compensation value
2. Setting data
3. Sequence number comparison and stop
4. Run time and part count
5. Workpiece origin offset value
6. Custom macro common variables
7. Software operator’s panel
8. Display language switching
9. Protection of data at eight levels
10. Precision level selection
11. Tool life management data
12. Pattern data input

For information on setting an offset value for the T series and M series, refer to the Operator’s Manual (T series) (B-64304EN-1) and the Operator’s Manual (M series) (B-64304EN-2), respectively.

The software operator’s panel, display language switching, precision level selection, and pattern data input depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

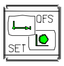
12.3.1 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.
See III-9.2 for automatic insertion of sequence numbers.
See III-12.3.2 for the sequence number comparison and stop function. This subsection describes how to set data.

Procedure for setting the setting data

Procedure

1 Select the MDI mode.

2 Press function key 

.

3 Press soft key [SETTING] to display the setting data screen.

4 This screen consists of several pages.

Press page key 

or 

until the desired screen is displayed.

An example of the setting data screen is shown below.

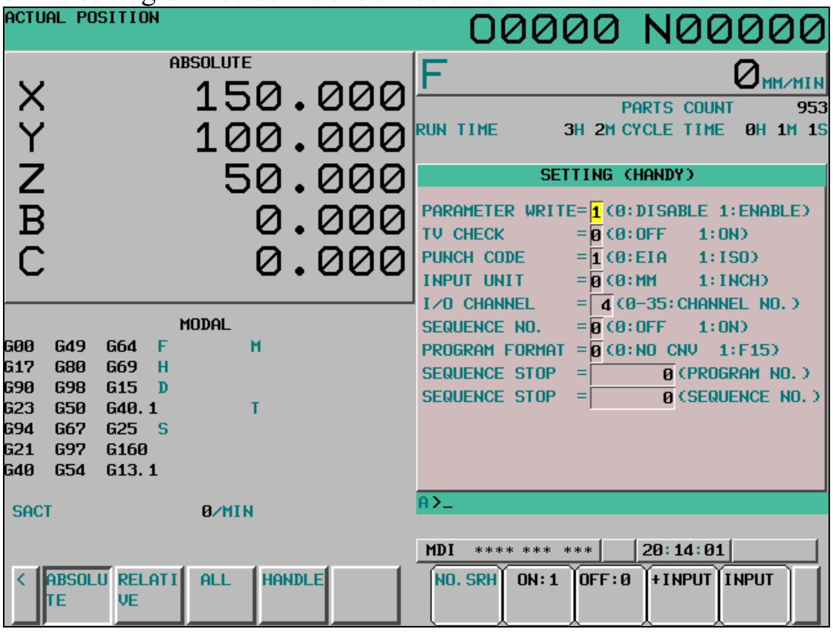


Fig. 12.3.1 (a) SETTING (HANDY) screen (10.4-inch)

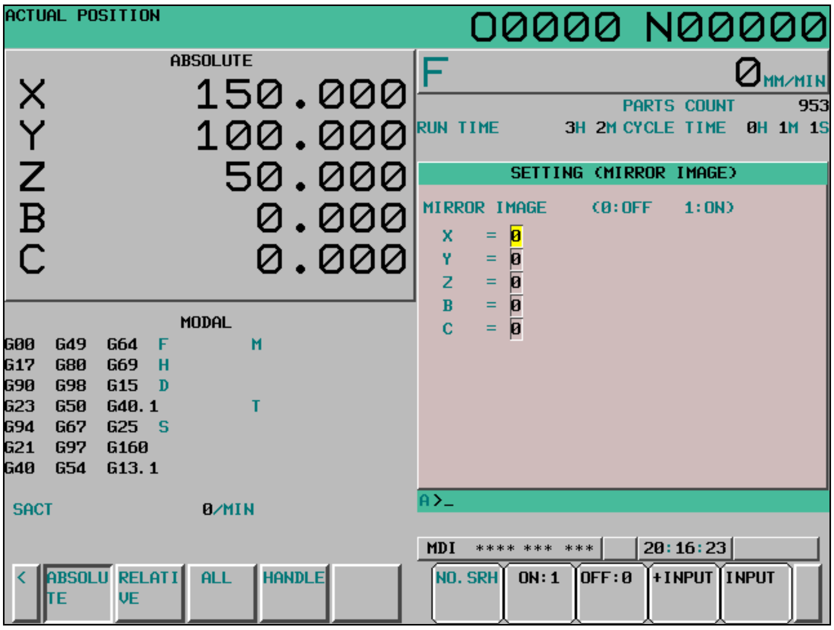
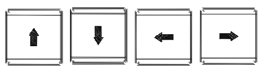


Fig. 12.3.1 (b) SETTING (MIRROR IMAGE) screen (10.4-inch)

5 Move the cursor to the item to be changed by pressing cursor keys  .

6 Enter a new value and press soft key [INPUT].

Explanation

- PARAMETER WRITE

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- TV CHECK

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- PUNCH CODE

Setting code when data is output through reader/puncher interface.
0 : EIA code output
1 : ISO code output

- INPUT UNIT

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- I/O CHANNEL

Using channel of reader/puncher interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2

- SEQUENCE NO.

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- PROGRAM FORMAT

Setting of whether to use the Series 10/11 format.
0: Uses the standard format.
1: Uses the Series 10/11 format.
For the Series 10/11 format, refer to Chapter II-6 in the OPERATOR’S MANUAL (T series) (B-64304EN-1) or Chapter II-7 of Part II in the OPERATOR’S MANUAL (M series) (B-64304EN-2).

- SEQUENCE STOP

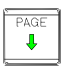
Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs

- MIRROR IMAGE

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- Others

Page key 

or 

can also be pressed to display the SETTING (TIMER) screen. See III-12.3.3 for this screen.

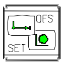
12.3.2 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

Procedure

1 Select the MDI mode.

2 Press function key .

3 Press chapter selection soft key [SETTING].

4 Press page key  or  several times until the following screen is displayed.

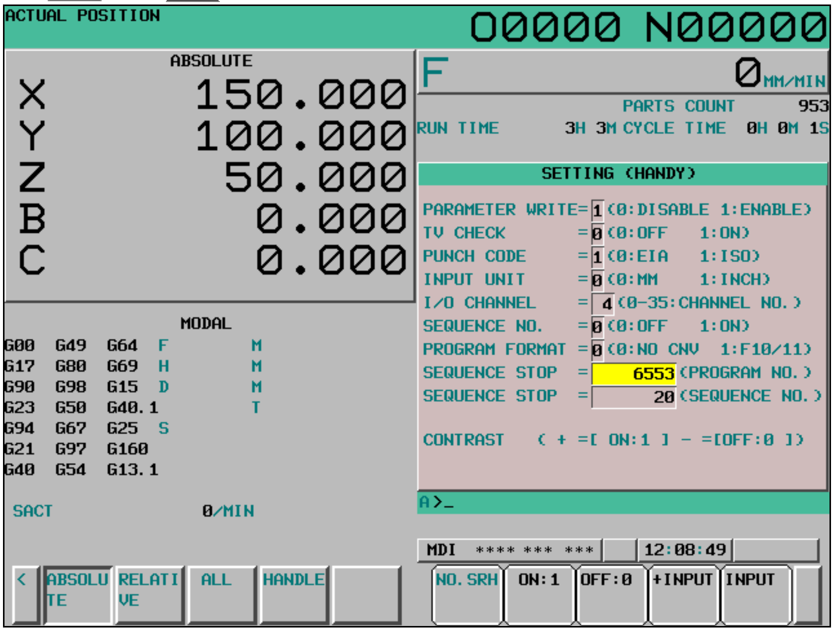


Fig. 12.3.2 (a) SETTING (HANDY) screen (10.4-inch)

5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 9999) of the program containing the sequence number with which operation stops.

6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with five or less digits) the sequence number with which operation is stopped.

7 When the set program is selected and automatic operation is started, the operation makes a single block stop at the block with the set sequence number.

Explanation

- Sequence number after the program is executed

After the specified sequence number is found during the execution of the program, the sequence number setting for sequence number compensation and stop becomes “-1”.

- Exceptional blocks

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

```
[Example]N1 #1=1 ;
         N2 IF[#1 EQ 1]GOTO 08 ;
         N3 GOTO 09 ;
         N4 M98 P1000 ;
         N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- Stop in the canned cycle

When the sequence number of the block in which the canned cycle command is present is found, the program stops after completion of return operation of the canned cycle.

- When the same sequence number is found several times in the program

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- Block to be repeated a specified number of times

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

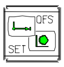
12.3.3 Displaying and Setting Run Time, Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed.
This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

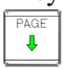
Procedure

1 Select the MDI mode.

2 Press function key 
.

3 Press chapter selection soft key [SETTING].

4 Press page key 

or 

several times until the following screen is displayed.

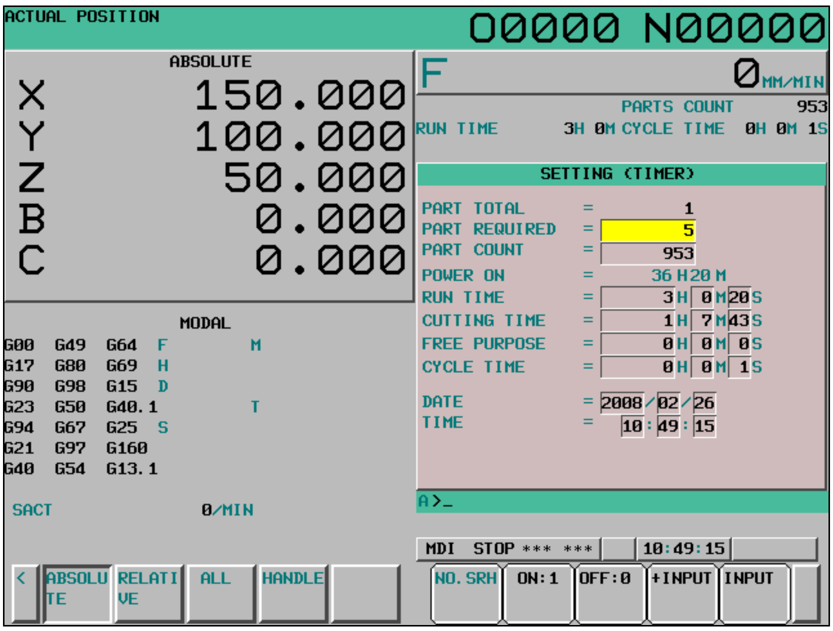


Fig. 12.3.3 (a) SETTING (TIMER) screen (10.4-inch)

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.
6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key [INPUT].

Explanation

- PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. This value cannot be set on this screen. Set the value in parameter No. 6712.

- PARTS REQUIRED

It is used for setting the number of machined parts required.
When the “0” is set to it, there is no limitation to the number of parts.
Also, its setting can be made by the parameter No 6713.

- PARTS COUNT

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. The value can also be set by parameter No. 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- POWER ON

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter No. 6750.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter No. 6751 or No. 6752.

- CUTTING TIME

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter No. 6753 or No. 6754.

- FREE PURPOSE

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- DATA and TIME

Displays the current date and time. The date and time can be set on this screen.

- Usage

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter No. 6710 is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (bit 0 (PCM) of parameter No. 6700 is set to 1). For details, see the manual issued by machine tool builders.

Limitation

- Run time and part count settings

Negative value cannot be set. Also, the setting of "M" and "S" of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- Time settings

Neither negative value nor the value exceeding the value in the following table can be set.

Table 12.3.3 (a)

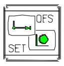
Item	Maximum value	Item	Maximum value
Year	2096	Hour	23
Month	12	Minute	59
Day	31	Second	59

12.3.4 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59) and external workpiece origin offset.
The workpiece origin offset value and external workpiece origin offset value can be set on this screen.

Procedure for displaying and setting the workpiece origin offset value

Procedure

1 Press function key .

2 Press chapter selection soft key [WORK].
The workpiece coordinate system setting screen is displayed.

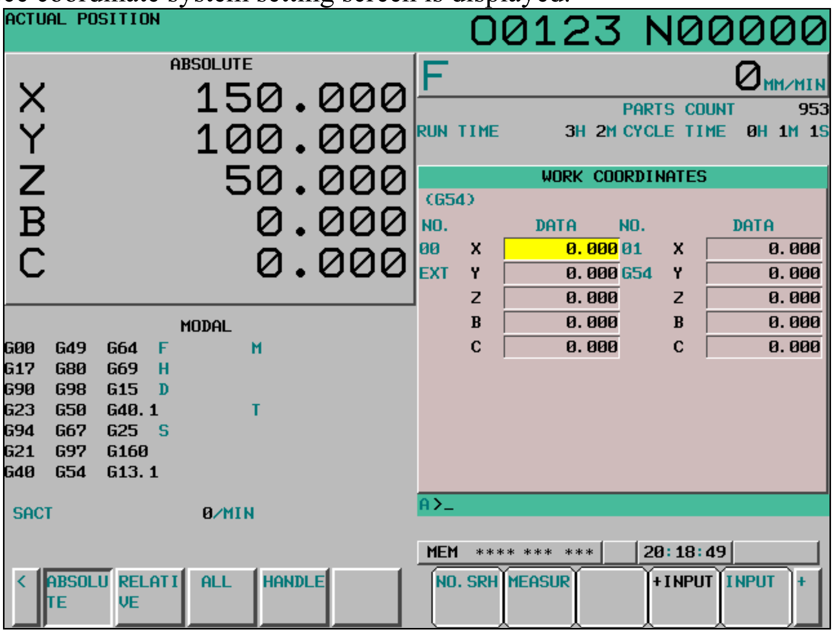


Fig. 12.3.4 (a) WORK COORDINATES screen (10.4-inch)

3 The screen for displaying the workpiece origin offset values consists of two or more pages.
Display a desired page in either of the following two ways:

- Press the page key  or .
- Enter the workpiece coordinate system number (0 : external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59) and press operation selection soft key [NO.SRH].

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

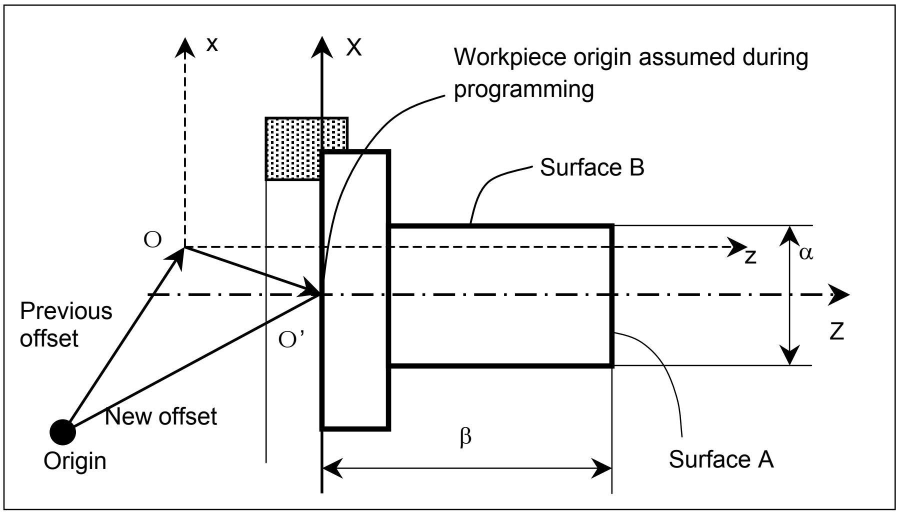
6 Enter a desired value by pressing numeric keys, then press soft key [INPUT]. The entered value is specified in the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing soft key [+INPUT], the entered value can be added to the previous offset value.
7 When performing counter input, enter the axis name in the key-in buffer and press soft key [C INPUT] to set the relative coordinates of the specified axis.
8 Repeat steps 5, 6, and 7 to change other offset values.
9 Turn on the data protection key to disable writing.

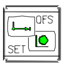
12.3.5 Direct Input of Workpiece Origin Offset Value Measured

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions. Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Procedure for direct input of workpiece origin offset value measured

Procedure



1 For the workpiece shown above, cut surface A in manual operation.
2 Retract the tool only in the X-axis direction without moving the Z-axis and stop the spindle.
3 Measure distance β between surface A and the programmed origin of the workpiece coordinate system as shown above.
4 Press function key 
.
5 To display the WORK COORDINATES screen, press the chapter selection soft key [WORK].

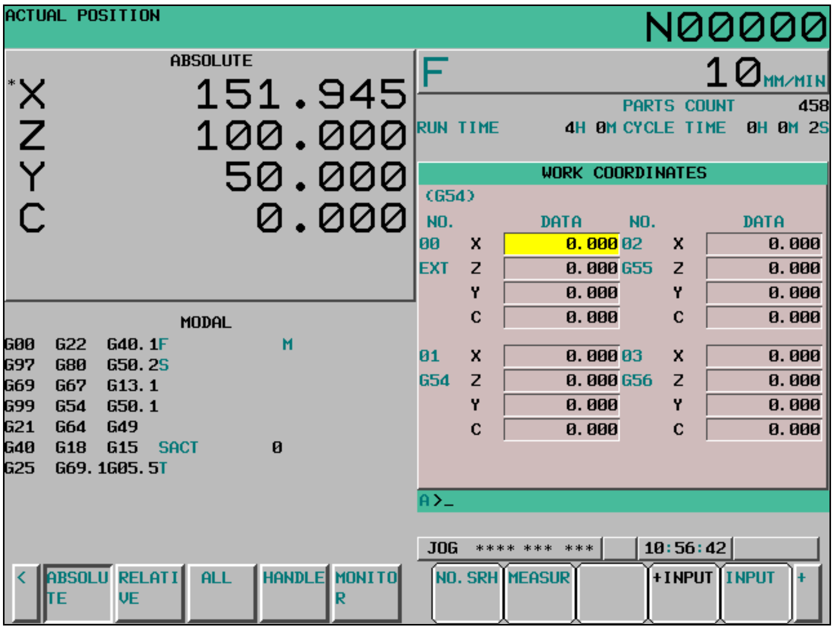


Fig. 12.3.5 (a) WORK COORDINATES screen (10.4-inch)

6 Position the cursor to the workpiece origin offset value to be set.
7 Press the address key for the axis along which the offset is to be set (Z-axis in this example).
8 Enter the measured value (β) then press soft key [MEASUR].
9 Cut surface B in manual operation.
10 Retract the tool only in the Z-axis direction without moving the X-axis and stop the spindle.
11 Measure diameter α of surface B and enter this value directly as the X value as described in Steps 7 and 8.

Limitation

- Consecutive input

Offsets for two or more axes cannot be input at the same time.

- During program execution

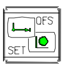
This function cannot be used while a program is being executed.

12.3.6 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the screen.
The values for variables can be set on this screen.
Relative coordinates can also be set to variables.

Procedure for displaying and setting custom macro common variables

Procedure

1 Press function key .

2 Press the continuous menu key , chapter selection soft key [MACRO], and soft key [(OPRT)] in the stated order.
The following screen is displayed.

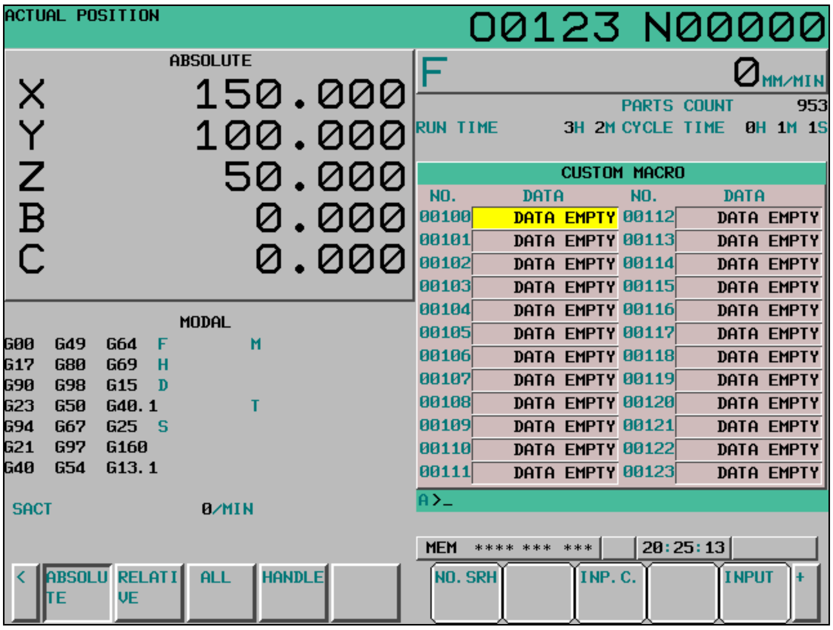
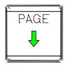
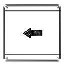
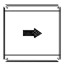
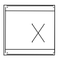
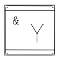


Fig. 12.3.6 (a) CUSTOM MACRO screen (10.4-inch)

3 Move the cursor to the variable number to set using either of the following methods:
- Enter the variable number and press soft key [NO.SRH].
- Move the cursor to the variable number to set by pressing page keys  and/or  and cursor keys , , , and/or .

4 Enter data with numeric keys and press soft key [INPUT].

5 To set a relative coordinate in a variable, press address key , , or , then press soft key [INP.C.].

6 To set a blank in a variable, just press soft key [INPUT].

Explanation

If the value of a variable produced by an operation is not displayable, an indication below is provided.
When the significant number of digits is 12 (with bit 0 (F0C) of parameter No. 6008 set to 0):

Variable value range	Variable value indication
0 < Variable value < +0.0000000001	+Underflow
0 > Variable value > -0.0000000001	-Underflow
Variable value > 999999999999	+Overflow
Variable value < -999999999999	-Overflow

When the significant number of digits is 8 (with bit 0 (F0C) of parameter No. 6008 set to 1):

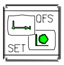
Variable value range	Variable value indication
0 < Variable value < +0.0000001	+Underflow
0 > Variable value > -0.0000001	-Underflow
Variable value > 99999999	+Overflow
Variable value < -99999999	-Overflow

12.3.7 Displaying and Setting the Software Operator's Panel

Operations on the MDI panel can substitute for the functions of switches on the machine operator's panel. This means that a mode selection, jog feed override selection, and so forth can be made with operations on the MDI panel, eliminating the need to use the corresponding switches on the machine operator's panel.
Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator's panel

Procedure

1 Press function key .

2 Press the continuous menu key , then press chapter selection soft key [OPERAT PANEL] ([OPR] for the 8.4-inch display unit).

3 The screen consists of several pages. Press page key  or  until the desired screen is displayed.

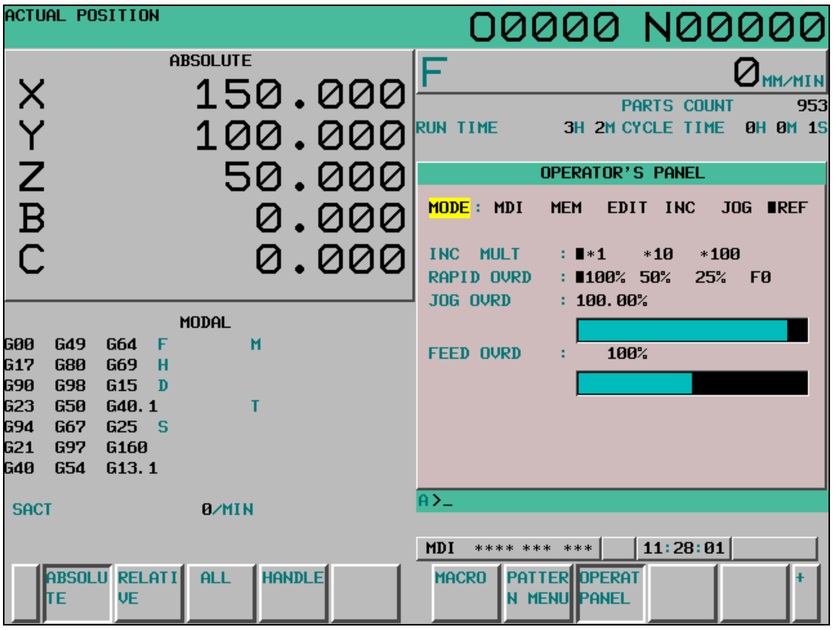


Fig. 12.3.7 (a) Example 1: Without the manual handle feed function (10.4-inch)

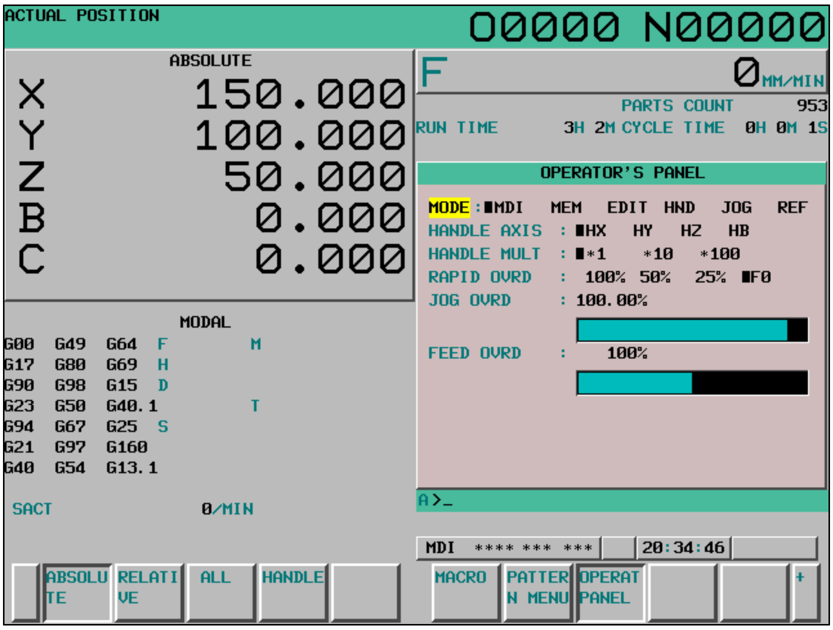


Fig. 12.3.7 (b) Example 2: With the manual handle feed function (10.4-inch)

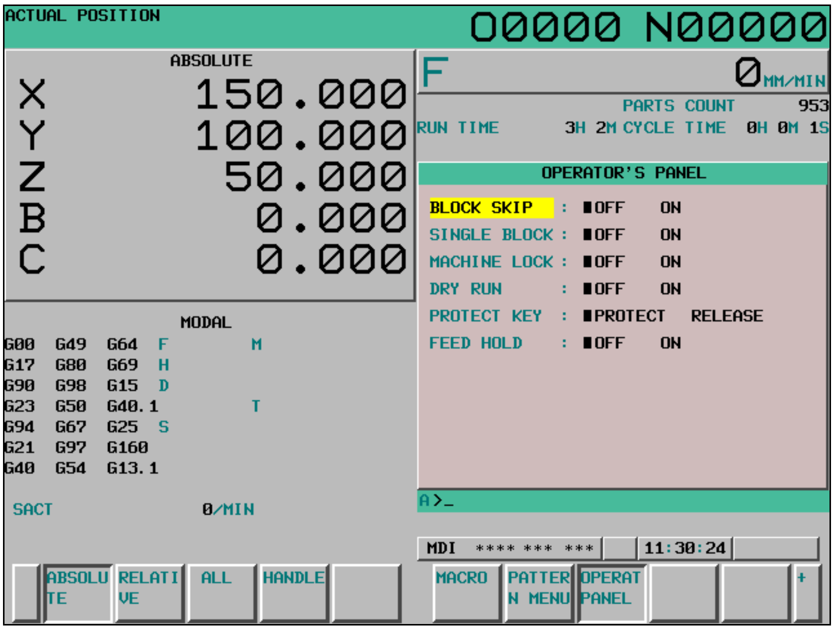
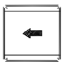
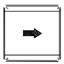


Fig. 12.3.7 (c) Example 3: (10.4-inch)

4 Move the cursor to the desired switch by pressing cursor key  or .

5 Push the cursor key  or  to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the  key together with an arrow key to perform jog rapid traverse.

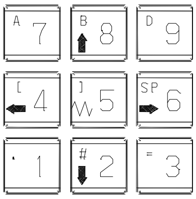


Fig. 12.3.7 (d) MDI arrow keys (T series)

Explanation

- Valid operations

The valid operations on the software operator's panel are shown below. Whether to use each group can be chosen using parameter No. 7200. Those groups that are not used are not displayed on the software operator's panel.

Group1 : Mode selection
Group2 : Selection of jog feed axis, Manual rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification
Group4 : Jog federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

- Screens on which jog feed is valid

When the LCD indicates other than the software operator's panel screen, jog feed is not conducted even if the arrow key is pushed.

- Jog feed and arrow keys

Parameters Nos. 7210 to 7217 are used to specify the correspondence between the arrow keys, axes, and movement directions.

- Feed magnification of incremental feed

The displayed item can be switched depending on whether the manual handle feed function is enabled. When the function is enabled, the feed magnification during incremental feed is switched to the handle magnification.

- General purpose switches

For the meanings of these switches, refer to the manual issued by machine tool builder.

12.3.8 Displaying and Switching the Display Language

The language used for display can be switched to another language.
A display language can be set using a parameter. However, by modifying the setting of the display language on this screen, the display language can be switched without turning off then on the power.

Displaying and setting the display language

Procedure

1 Press function key OFS/SET.
2 Press the continuous menu key several times.
3 Press soft key [LANGUAGE] to display the language screen.

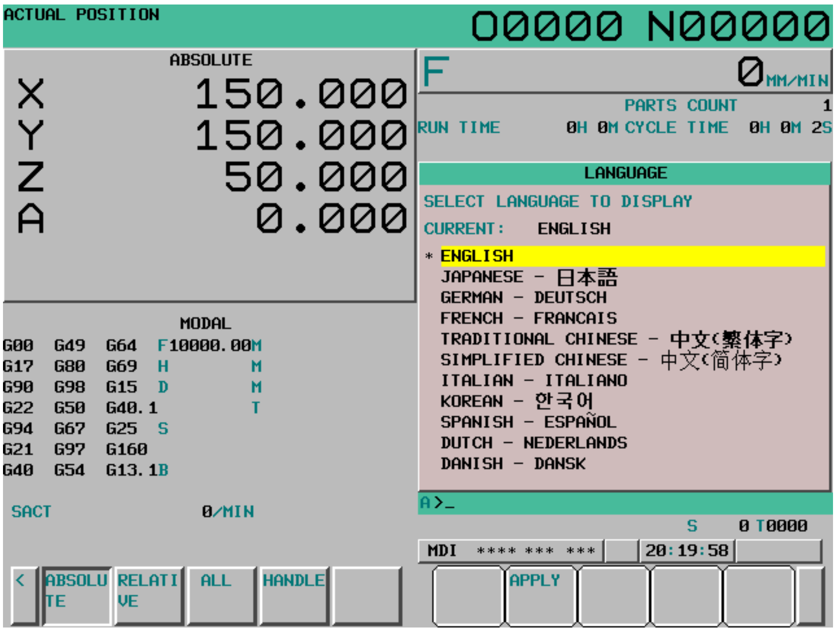


Fig. 12.3.8 (a) LANGUAGE screen (10.4-inch)

4 Press page key PAGE↑ or PAGE↓, then press cursor keys ↑, ↓ to move the cursor to a desired display language.
5 Press operation soft key [APPLY]. The display language is switched to the selected language. The language specified on this screen continues to be used if the power is turned off then back on.

Explanation

- Language switching

The language screen can be displayed if bit 0 (NLC) of parameter No. 3280 is set to 0.

- Selectable languages

The display languages selectable on this screen are as follows:

1. English
2. Japanese
3. German
4. French
5. Chinese (Traditional)
6. Chinese (Simplified)
7. Italian
8. Korean
9. Spanish
10. Dutch
11. Danish
12. Portuguese
13. Polish
14. Hungarian
15. Swedish
16. Czech
17. Russian
18. Turkish
19. Bulgarian

Among the languages listed above, English and other usable languages are displayed on the screen as a list of switchable languages.

> **NOTE**
> The Bulgarian is optional.

Limitation

- Language parameter modification on the parameter screen

Which language to use for display is specified with parameter No. 3281. This parameter can be modified using the parameter screen as well. However, if a modification is made on the parameter screen, the new setting is not reflected until “APPLY” operation is performed on the language screen or the power is turned on again. If an invalid value is set in parameter No. 3281 on the parameter screen, the screen is displayed in English after the power is turned on again.

12.3.9 Protection of Data at Eight Levels

You can set eight CNC and PMC operation levels and one of eight protection levels for each type of CNC and PMC data.
When an attempt is made to change CNC and PMC data or output it to an external unit, the operation level is compared with the protection level to determine whether to allow the change or external output.

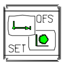
> **NOTE**
> The 8-level data protection function is optional.

12.3.9.1 Operation level setting

You can set eight CNC and PMC operation levels.

Displaying and setting the operation level setting screen

Procedure

1 Press function key .

2 Press the continuous menu key  several times until soft key [PROTECT] ([PROT.]) for the 8.4-inch display unit) is displayed.

3 Press soft key [PROTECT].
The operation level setting screen shown below is displayed.

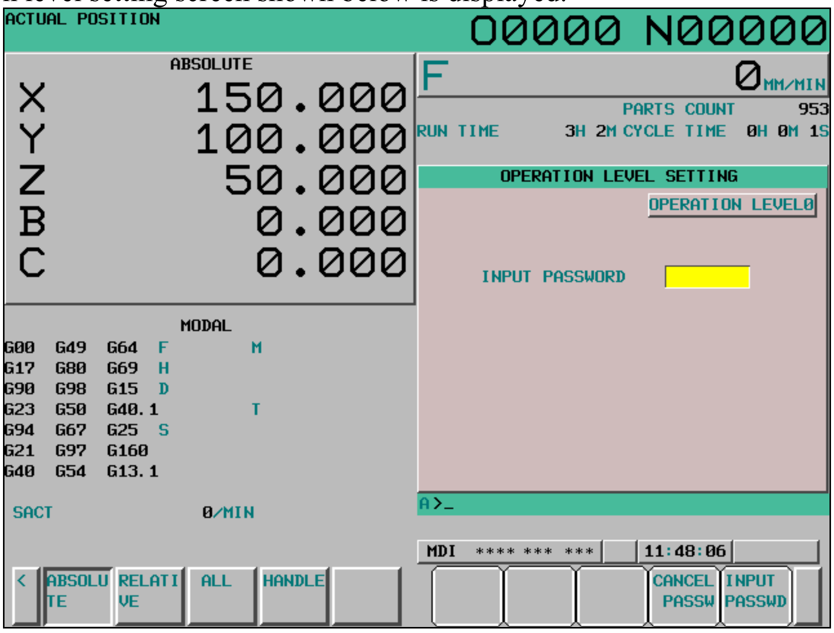


Fig. 12.3.9.1 (a) Operation level setting screen (10.4-inch)

4 Key in the password for an operation level to be set/modified, then press soft key [INPUT PASSWD] ([INPUT]) for the 8.4-inch display unit).

5 To return the operation level to 0, 1, 2, or 3, press soft key [CANCEL PASSWD] ([CANCEL]) for the 8.4-inch display unit).

Explanation

- Operation level setting

To select operation level 0 to 3, use the corresponding memory protection key signal.
To select operation level 4 to 7, use the corresponding password.

Table 12.3.9.1 (a) Operation level setting

Operation level	Setting	Sample grouping
7 (high)	Password	-
6	Password	MTB
5	Password	Dealer and integrator
4	Password	End user
3	Memory protection key signal	User level (level 1)
2	Memory protection key signal	User level (level 2)
1	Memory protection key signal	User level (level 3)
0 (low)	Memory protection key signal	User level (level 4)

When operation level 4 to 7 is set, the operation level remains unchanged until the password is cleared.

(The operation level also remains unchanged if the power is turned off.)
Operation level 7 is reserved for CNC and PMC maintenance.

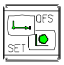
> **NOTE**
> When a password is being entered, an asterisk (*) is displayed instead of each entered character.

12.3.9.2 Password modification

The current operation level is displayed.
The password for each of operation levels 4 to 7 can be modified.

Displaying and setting the password modification screen

Procedure

1 Press function key .

2 Press the continuous menu key  several times until soft key [PROTECT] ([PROT.] for the 8.4-inch display unit) is displayed.

3 Press soft key [PROTECT].

4 Press soft key [PASSWORD]([PASSWD] for the 8.4-inch display unit).
The PASSWORD CHANGE screen shown below is displayed.

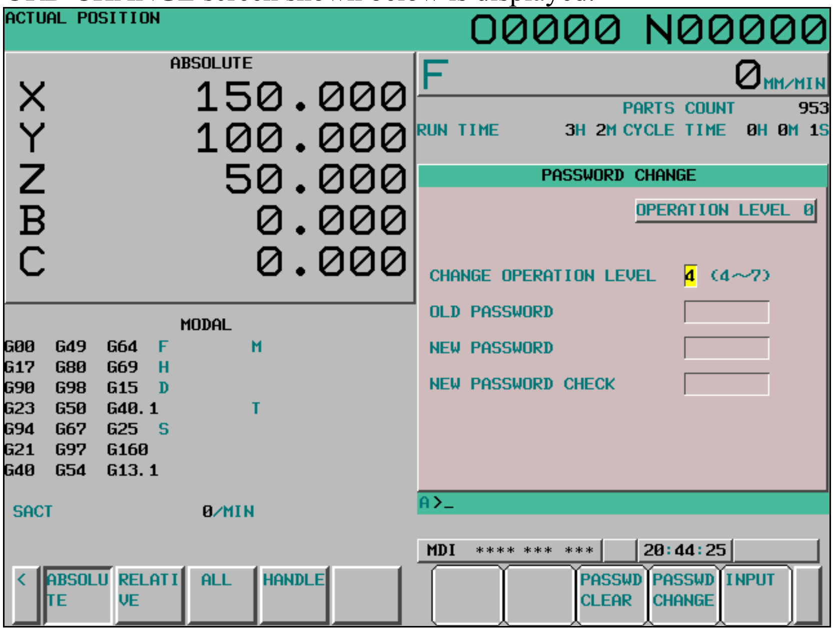


Fig. 12.3.9.2 (a) PASSWORD CHANGE screen (10.4-inch)

5 Key in an operation level whose password is to be modified, then press soft key [INPUT].

6 Key in the current password for the operation level whose password is to be modified, then press soft key [INPUT].

7 Key in a new password, then press soft key [INPUT].

8 Key in the new password again for confirmation, then press soft key [INPUT].

9 Press soft key [PASSWD CHANGE] ([CHANGE] for the 8.4-inch display unit).

10 To clear the password, press soft key [PASSWD CLEAR] ([CLEAR] for the 8.4-inch display unit).

Explanation

Up to eight characters (only uppercase alphabetic characters and numeric characters) can be input.

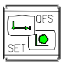
> **NOTE**
> 1 For a password, consisting of three to eight characters, the following characters are available:
>   - Uppercase alphabetic characters
>   - Numeric characters
>
> 2 When a password is being entered, an asterisk (*) is displayed instead of each entered character.
> 3 Whether a password can be changed at the current operation level is determined as follows:
>   - Password of an operation level higher than the current operation level
>   Cannot be changed.
>   - Password of the current operation level
>   Can be changed.
>   - Password of an operation level lower than the current operation level
>   Can be changed (only to the initial password).
>
> 4 The set password is not displayed.
>   Be careful not to forget the password.

12.3.9.3 Protection level setting

The current protection level is displayed.
The change protection level and output protection level of each data item are displayed.
The change protection level and output protection level of each data item can be changed.

Confirmation based on protection level setting

Procedure

1 Press function key .
2 Press the continuous menu key  several times until soft key [PROTECT] ([PROT.] for the 8.4-inch display unit) is displayed.
3 Press soft key [PROTECT].
4 Press soft key [DATA LEVEL] ([PRT LV] for the 8.4-inch display unit)to change the protection level of CNC data or press soft key [PMC LEVEL] ([PMC LV] for the 8.4-inch display unit)to change the protection level of PMC data.

The following protection level change screen is displayed.

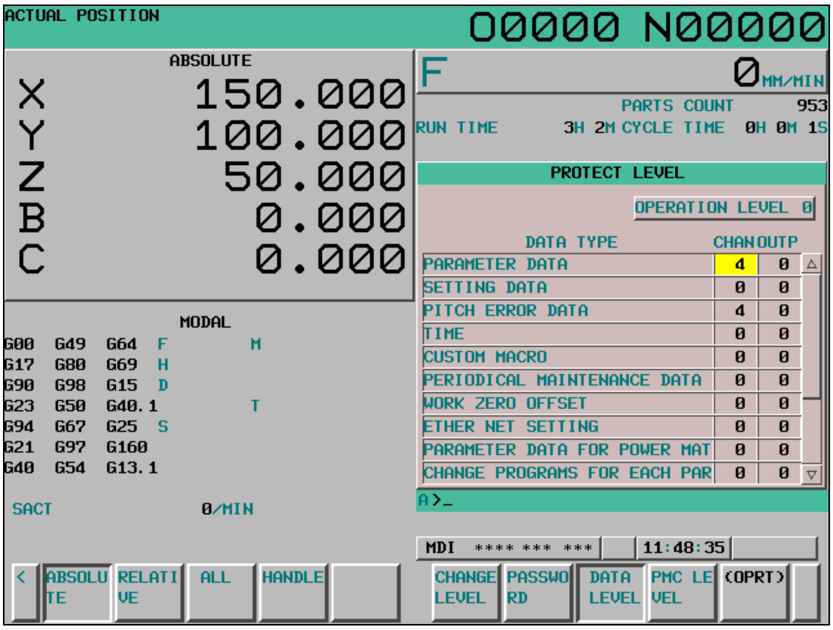


Fig. 12.3.9.3 (a) Protection level change screen (10.4-inch)

5 Move the cursor to the change level or output level of a desired data item.
6 Key in a new desired level, then press soft key [INPUT].

Explanation

When the protection level of a data item is higher than the current operation level, the protection level of the data item cannot be changed.
The protection level of a data item cannot be changed to a protection level higher than the current operation level.
For each of the following types of data, you can set a data protection level. There are the following two types of data protection levels:

- Change protection level

Sets the protection level used when data is changed.

- Output protection level

Sets the protection level used when data is output to an external unit.
As a protection level, you can set a value of 0 (low) to 7 (high).

Table 12.3.9.3 (a) Protection level of each type of data

Type of data	Initial protection level	
	Change	Output
Custom macro variable data <CUSTOM MACRO> (including variable data dedicated to the macro executor)	0	0
Periodical maintenance data <PERIODICAL MAINTENANCE DATA>	0	0
Tool offset data <TOOL OFFSET DATA> (For each type when tool geometry compensation and tool wear compensation are treated differently)	0	0
Clock data <TIME>	0	0
Workpiece origin shift amount data <WORK ZERO SHIFT>	0	0
Workpiece origin offset data <WORK ZERO OFFSET>	0	0
Ethernet setting data <ETHER NET SETTING>	0	0
Parameter data <PARAMETER DATA>	4	0
Settings <SETTING DATA>	0	0
Pitch error compensation data <PITCH ERROR DATA>	4	0
Parameter data for Power Mate CNC manager function <PARAMETER DATA FOR POWER MATE CNC MANAGER>	0	0
Part program editing operation <CHANGE PROGRAMS FOR EACH PARTS>	0	0

Type of data	Initial protection level	
	Change	Output
Absolute coordinate preset operation <PRESET OF ABSOLUTE AXIS DATA>	0	0

Table 12.3.9.3 (b) Protection level of PMC data

Type of data	Initial protection level	
	Change	Output
Composition parameter	0	0
Setting (online)	0	0
Setting (each path)	0	0
Sequence program	0	0
PMC parameter	0	0
Timer	0	0
Counter	0	0
Keep relay	0	0
Keep relay (system)	0	0
Data table	0	0
Data table control	0	0
PMC memory	0	0

NOTE

1. For some types of data, the output function is not provided.
2. When the protection level of data is higher than the current operation level, the protection level cannot be changed.
3. The protection level of data cannot be changed to a level higher than the current operation level.
4. Settable types of data increase or decrease, depending on the option configuration.
5. For details on the protection level of PMC data, refer to “PMC Programming Manual (B-64393EN)”.
6. The type of tool offset data put in effect varies depending on the tool compensation value memory used.
7. To change the protection level for each part program, do so on the PROGRAM FOLDER screen rather than on the PROTECT LEVEL screen.
8. Part program editing includes program editing for the MDI mode.
9. Presetting absolute coordinates causes preset workpiece coordinate system values to be protected.
10. During tool offset data input/output, if any tool offset data type is not allowed to be changed or output to the outside, it is processed as follows:
    - Input : Any data type other than those whose change is not allowed is changed.
    - Output : Any data type other than those whose change is not allowed is output.

12.3.10 Precision Level Selection

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen can be selected. As shown in the figure below, the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

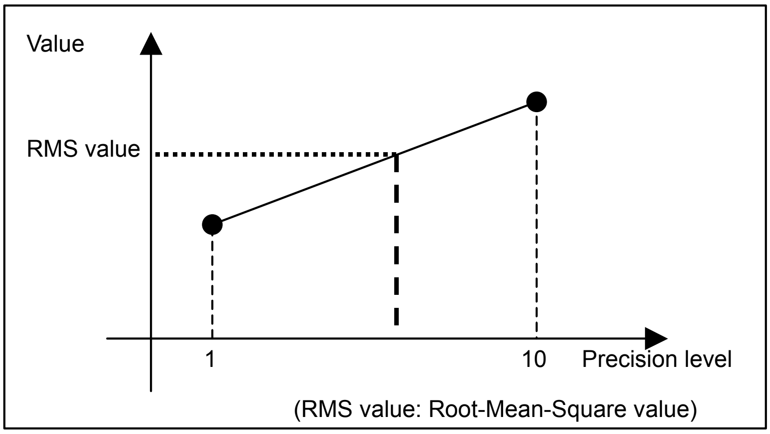


Fig. 12.3.10 (a) Image of "level"

Procedure for precision level selection

1 Select the MDI mode.

2 Press function key [OFS/SET].

3 Press the continuous menu key [▷] several times until soft key [PREC LEV] ([PR-LEV] for the 8.4-inch display unit)is displayed.

4 Press soft key [PREC LEV].

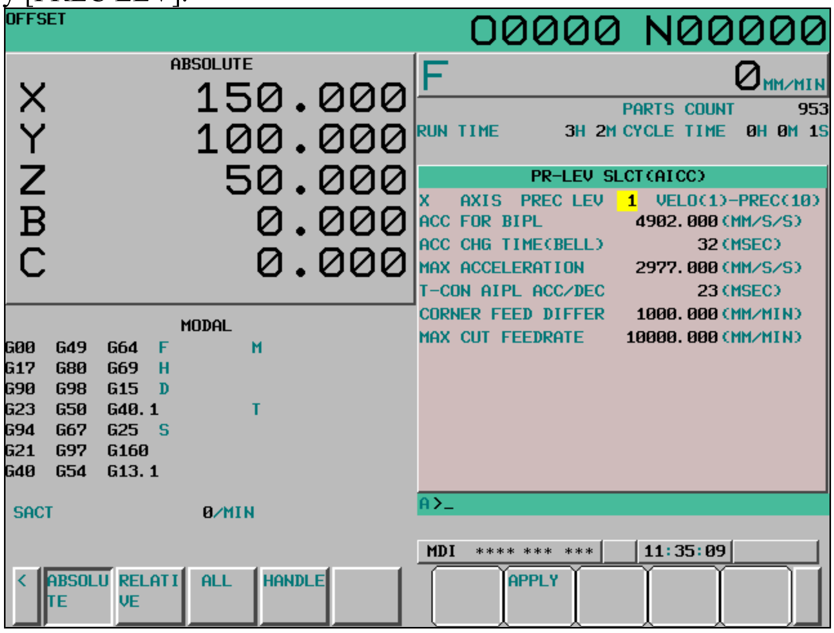


Fig. 12.3.10 (b) Precision level selection screen (10.4-inch)

5 To change the precision level, key in a desired precision level (1 to 10), then press the [INPUT] key on the MDI panel or soft key [APPLY].

6 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

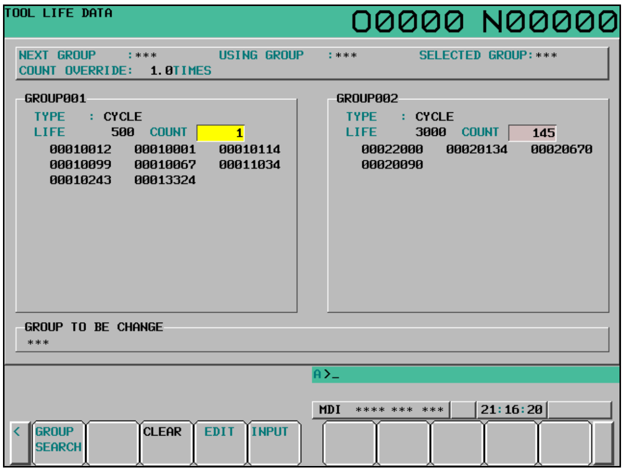
7 If there is an axis in addition to the currently displayed axes, press page key [PAGE ↑] or [PAGE ↓] several times to display the screen for the axis.

12.3.11 Displaying and Setting Tool Life Management Data

Displaying tool life management data on a screen enables the current status of tool life management to be grasped. Also on the screen, tool life management data can be edited. The screen is either:

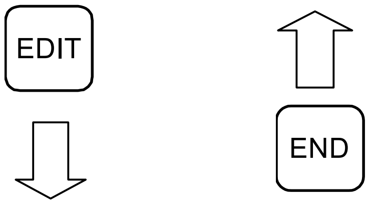
- Tool life management (list screen) or
- Tool life management (group editing screen)

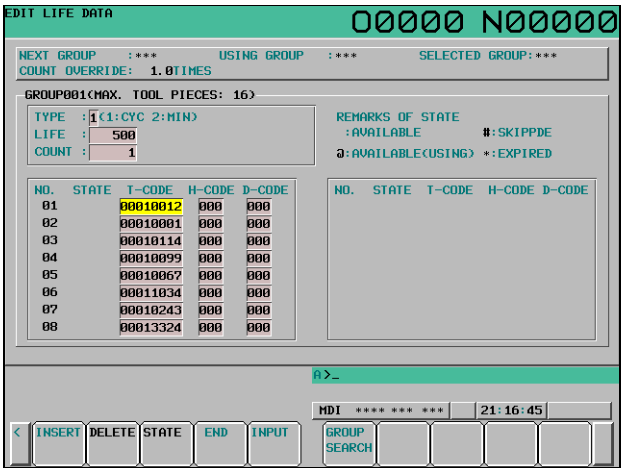
Overview



Tool life management (list screen)
Displayed items:
- NEXT GROUP, USING GROUP, SELECTED GROUP
- COUNT OVERRIDE
- GROUP NO.
- TYPE
- LIFE, COUNT
- TOOL MANAGEMENT STATUS
- TOOL NUMBER
- GROUP TO BE CHANGE

List screen





Tool life management (group editing screen)
Displayed items:
- NEXT GROUP, USING GROUP, SELECTED GROUP
- COUNT OVERRIDE
- MAX.TOOL PIECES
- TYPE
- LIFE, COUNT
- STATE
- T-CODE, H-CODE, and D-CODE

Pressing horizontal soft key [EDIT] or [END] switches from the list screen to the group editing screen or vice versa.

Group editing screen

M

The group editing screen always displays H and D codes.

T

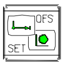
The T series has the turret system to change tools and H and D codes are not used, so these codes are not displayed.

12.3.11.1 Tool life management (list screen)

This screen can display the life management status of all tools in tool groups and whether the life of the tool groups has expired. It also enables you to set tool life counters and clear execution data.

Displays on the list screen

Procedure

1 Press function key  .

2 Press the continuous menu key  several times to display soft key [TOOL LIFE] ([TOOLLF] for the 8.4-inch display unit).

3 Press soft key [TOOL LIFE].

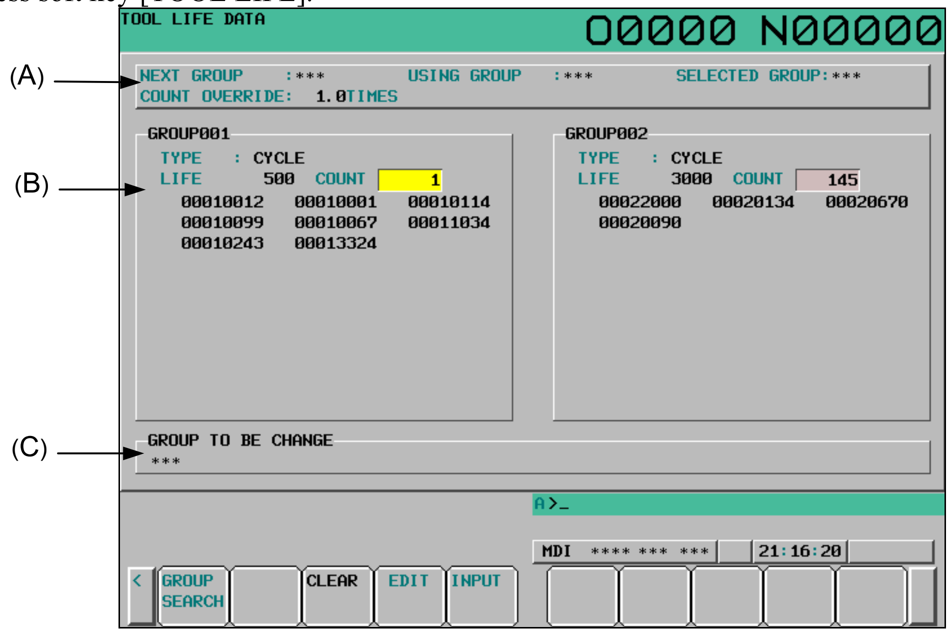


Fig. 12.3.11.1 (a) Displaying tool life management (list screen) (10.4-inch)

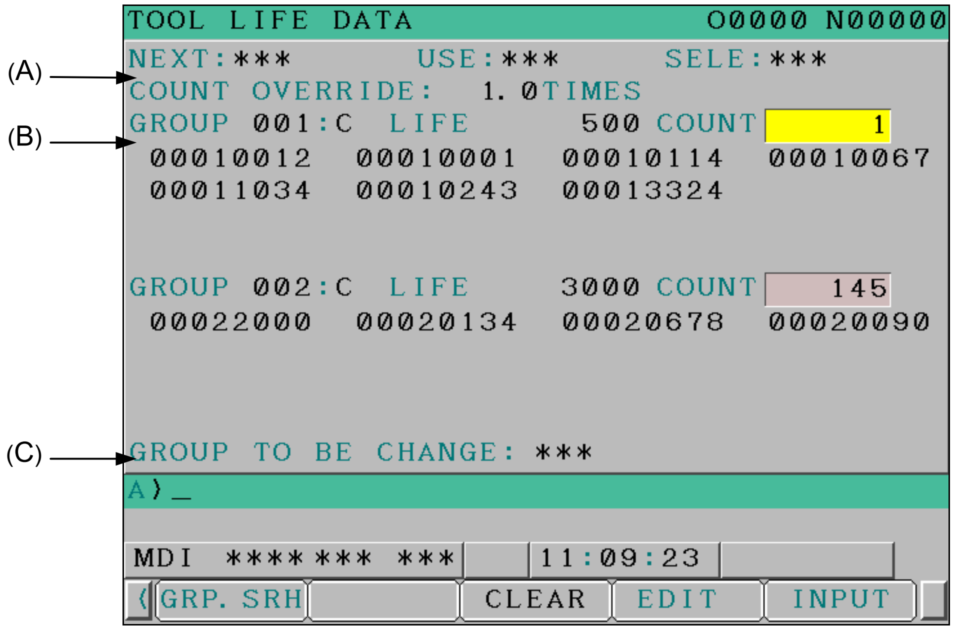


Fig. 12.3.11.1 (b) Displaying tool life management (list screen) (8.4-inch)

- Contents of (A)

(A) displays tool group numbers and an override value. If there is no tool group to display, ***, instead of tool group numbers, is displayed.

NEXT GROUP:

Tool group number for which life counting is started by the next M06 command.

USING GROUP:

Tool group number for which life counting is currently under way.

SELECTED GROUP:

Tool group number for which life counting is currently under way or life counting has been performed most recently.

COUNT OVERRIDE:

The override value by which the life count (time) is multiplied is displayed if the tool life counter override signal is enabled (bit 2 (LFV) of parameter No. 6801 = 1).

"1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

- Contents of (B)

(B) displays the set life value, the current content of the tool life counter, and the registered tool numbers (in the order they are used) for each tool group. If the life count type is specification by duration, the measurement unit used in displaying and specifying set life values and tool life counter values is selected according to the setting of bit 0 (FCO) of parameter No. 6805 as listed below.

Bit 0 (FCO) of parameter No. 6805	0	1
Measurement unit used in displaying and specifying set life values and tool life counter values	1 minute	0.1 minutes

The following table lists the prefixes used with tool numbers.

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

NOTE

1 The tool life counter indicates the count value for the tool indicated with @.

2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.

3 If bit 3 (EMD) of parameter No. 6801 =1, the following differences can occur depending on the type of the life counter in use.

- If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

- Contents of (C)

(C) displays tool group numbers for which a tool change signal has been issued.
If there are so many tool group numbers that all the numbers cannot be displayed, some are omitted, and “>>” is displayed instead.
If there is no tool group number in need of change, “***” is displayed.

Setting data on the list screen

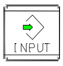
Tool life management data can be specified in the reset state (both the OP and RST signals are “0”). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is “1”).

> **NOTE**
> As for USING GROUP or NEXT GROUP settings:
> 1) During automatic operation (OP signal = “1” and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
> 2) In the reset state (OP signal = “0” and RST signal “0”), the following editing operation stops life management because it cannot be continued.
> - Clearing execution data

Procedure

- Setting the tool life counter

The tool life counter can be set with a value, using the following methods.
Method 1
1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press soft key [INPUT].

Method 2
1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press 

key.

- Clearing execution data

All existing execution data for a tool group selected by the cursor can be cleared as follows:
1 Place the cursor on the tool group whose execution data you want to clear.
2 Press soft key [ERASE].
3 Press soft key [EXEC].

> **NOTE**
> Setting bit 4 (GRS) of parameter No. 6800 to 1 enables execution data for all registered tool groups to be cleared.

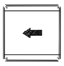
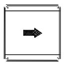
- Selecting tool groups

Tool groups can be selected using the following methods.
Method 1
1 Enter a tool group number from the keypad.
2 Press soft key [GRP.SRH].

> **NOTE**
> If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key  or  to display desired groups.

2 Press cursor movement key  or  to move the cursor to the desired group.

- Switching to the group editing screen

Switch to tool life management (group editing screen).

1 Move the cursor to the tool group you want to edit:

2 Press soft key [EDIT].

12.3.11.2 Tool life management (group editing screen)

On this screen, it is possible to edit tool life management data (such as tool life value, tool life counter, and tool data) for the tool group of interest.

Displaying the group editing screen

Procedure

1 Place the cursor on the tool group you want to edit from the list screen.
2 Press soft key [(OPRT)].
3 Press soft key [EDIT].

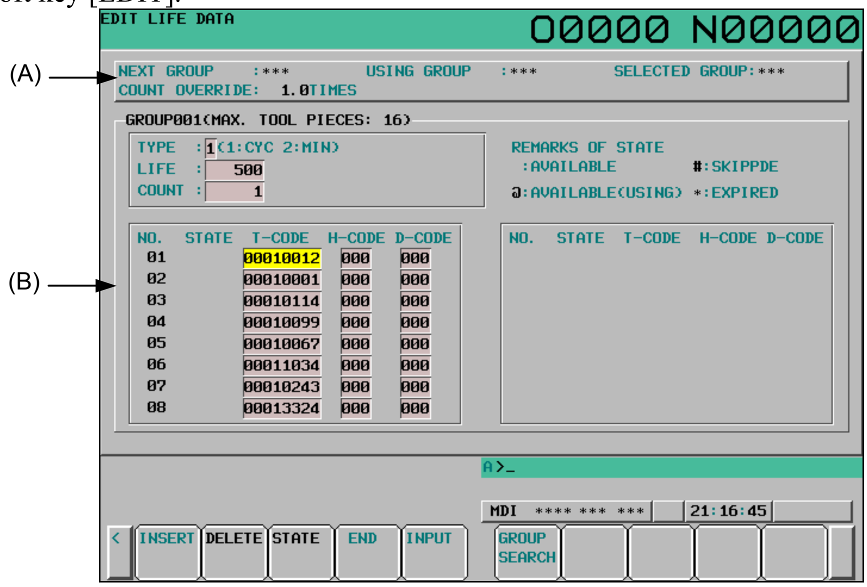


Fig. 12.3.11.2 (a) Displaying tool life management (group editing screen) (10.4-inch)

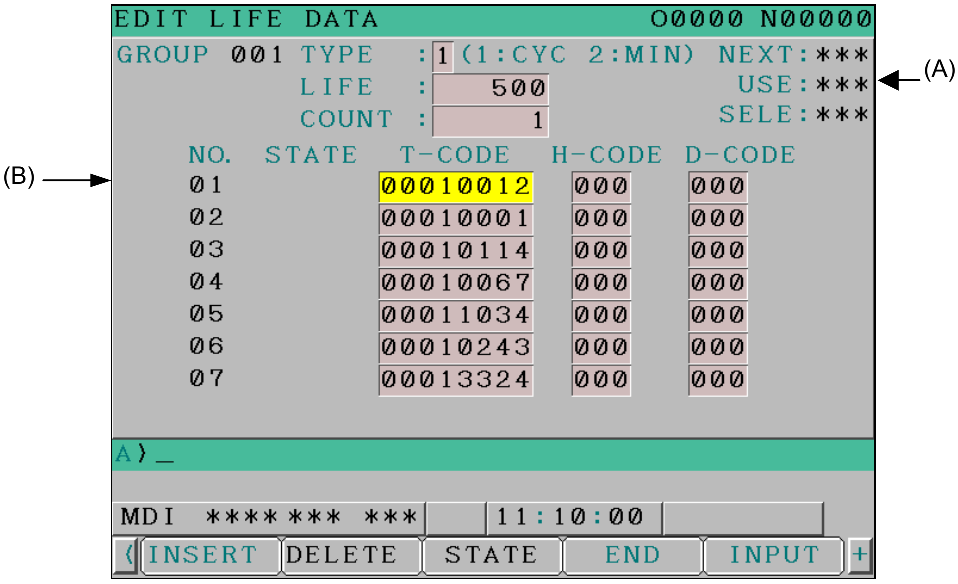


Fig. 12.3.11.2 (b) Displaying tool life management (group editing screen) (8.4-inch)

> **NOTE**
> 1 If no tool is registered with a tool group, none of a life count type, a life value, and a tool life counter value is displayed for the tool group.
> 2 COUNT OVERRIDE is not displayed on the group editing screen for the 8.4-inch display unit.

- Contents of (A)

Like the counterpart of the list screen, (A) of the editing screen displays the next tool group number and override value. If there is no appropriate tool group, *** is displayed instead.

NEXT GROUP:
Tool group for which life counting is started by the next M06 command.

USING GROUP:
Tool group number for which life counting is currently under way.

SELECTED GROUP:
Tool group number for which life counting is currently under way or life counting has been performed most recently.

COUNT OVERRIDE:
The override value by which the life count (time) is multiplied is displayed if the tool life counter override signal is enabled (bit 2 (LFV) of parameter No. 6801 = 1).
"1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

- Contents of (B)

(B) details tool life management data related to a selected tool group as follows:

TYPE :

1	Specification by count
2	Specification by duration

LIFE : Tool life value

COUNT : Tool life counter

STATE :

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

T-CODE : Tool number

M

H-CODE : Tool length compensation specification code
D-CODE : Cutter compensation specification code

T

H-CODE : No display.
D-CODE : No display.

> **NOTE**
> 1 The tool life counter indicates the count value for the tool indicated with @.
> 2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
> 3 If bit 3 (EMD) of parameter No. 6801 = 1, the following differences can occur depending on the type of the life counter in use.
> - If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
> - If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.
>
> 4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

Setting data on the group editing screen

Tool life management data can be specified in the reset state (both the OP and RST signals are "0"). However, setting bit 1 (TCI) of parameter No. 6804 to "1" enables tool life management data to be specified even during automatic operation (the OP signal is "1").
The available editing operations are listed below.

M

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code, H code, and D code)	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, H code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

T

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code)	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status and T code)	MDI
Selecting tool skip	MDI

Items that can be edited	Mode
Specifying to clear tool data (life re-set)	MDI

If no tool is registered with a tool group, none of a life count type, a tool life value, and a tool life counter value can be set for the tool group. First of all, add a tool number (T code).

NOTE

1 As for USING GROUP or NEXT GROUP editing:
  <1> During automatic operation (OP signal = “1” and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
  <2> In the reset state (OP signal = “0” and RST signal “0”), the following editing operation stops life management because it cannot be continued.
   - Adding tool numbers (T code)
   - Deleting all tool group data at a time
   - Deleting tool data (status, T code, H code, and D code)

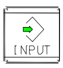
2 The following editing operations may set the tool change signal to ”1”.
  - Selecting tool skip for the last tool.
  - Deleting tool numbers, resulting in any tool other than those whose life has expired or who have been skipped being not found in the tool group of interest.

3 The following editing operations may reset the tool change signal to ”0”.
  - Adding tool numbers, leading to tools whose life has not expired being set in the tool group of interest.
  - Selecting tool clear.

Procedure

- Setting a life count type, tool life value, tool life counter, and tool data

Setting a life count type, tool life value, tool life counter, and tool data
Method 1
1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press soft key [INPUT].

Method 2
1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press 

.

NOTE

1 Changing a tool life value or tool life counter does not affect the tool status or tool change signal.

2 Changing a life count type causes the tool life value and too life counter to be reset to 0.

- Adding tool numbers

Tool numbers can be added to a tool group as follows:
1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) just before a tool number to be added.
3 Enter the tool number from the key pad.
4 Press soft key [INSERT].

(Example)

Adding tool number 1550 between numbers 1 and 2 (for the M series)

1 Move the cursor to the data for number 1, enter “1550”, and press [INSERT].

NO.	STATE	T-CODE	H-CODE	D-CODE
01		00000001	000	000
02	#	00000002	000	000

2 The entered T code 1550 is inserted in the position of number 2. The H and D codes are reset to 0.

NO.	STATE	T-CODE	H-CODE	D-CODE
01		00000001	000	000
02		00001550	000	000
03	#	00000002	000	000

- Deleting all tool group data at a time

All tool group data can be deleted at a time as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) just Select the tool group from which all tool data you want to delete at a time.
3 Press soft key [DELETE].
4 Press soft key [GROUP].
5 Press soft key [EXEC].

- Deleting tool data

Tool data can be deleted from a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) you want to delete.
3 Press soft key [DELETE].
4 Press soft key [<CURS>].

NOTE
1 Deleting all tools from a tool group is equivalent to deleting the tool group itself.
2 Deleting a tool indicated with @ (in use) results in @ being moved to the previous tool whose life has expired or which has been skipped.

- Selecting tool skip

Tool data can be placed in a skip state as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to skip.
3 Press soft key [STATE].
4 Press soft key [SKIP].

- Specifying to clear tool data (life re-set)

Tool data state can be cleared as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to clear.
3 Press soft key [STATE].
4 Press soft key [CLEAR].

- Selecting a tool group

A tool group can be selected as follows:
Method 1
1 Enter a tool group number from the keypad.
2 Press soft key ['GRP.SRH].

- Switching to the list screen

The tool life management (list screen) can be resumed as follows:
1 Press soft key [END].

12.3.12 Displaying and Setting Pattern Data Inputs

Described below are a method for displaying machining menus (pattern menus) created by machine tool builders and a method for setting them. The descriptions are based on an example. For actual pattern menus and pattern data, refer to manuals released by respective machine tool builders.

Displaying pattern data and pattern menus

Given below is a procedure for displaying pattern menus.

1 Press function key .
2 Press the continuous menu key .
3 Press soft key [PATTERN MENU] ([MENU] for the 8.4-inch display unit).
The pattern menu screen shown below appears.

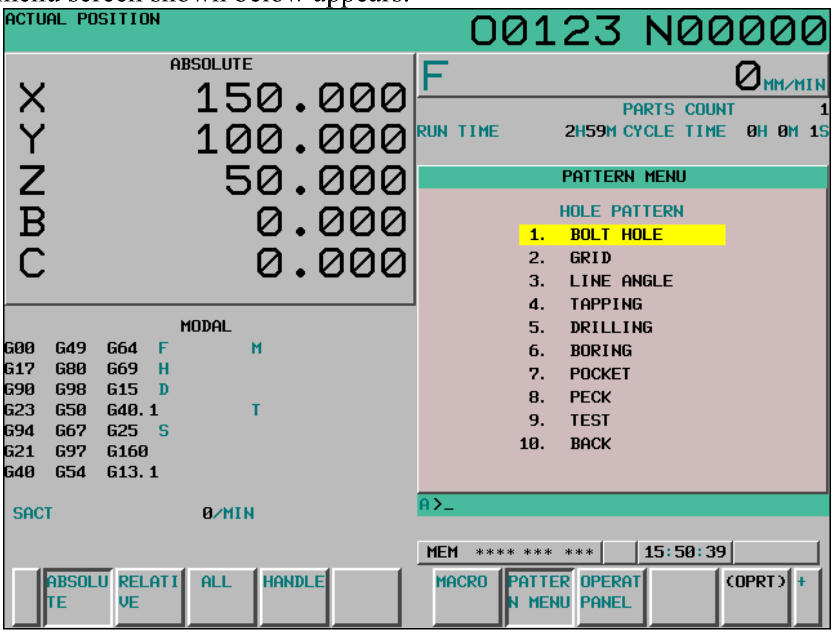


Fig. 12.3.12 (a) Pattern menu screen (10.4-inch)

On this screen, a pattern to be used can be selected.
The following two methods can be used to select patterns.

- Using the cursor
  Move the cursor to a pattern name you want to select, using cursor key or , and then press soft key [SELECT] or .
- Specifying a pattern number
  Enter a number displayed at the left side of a pattern name, and press soft key [SELECT] or .

The custom macro screen (pattern data screen) shown below appears.

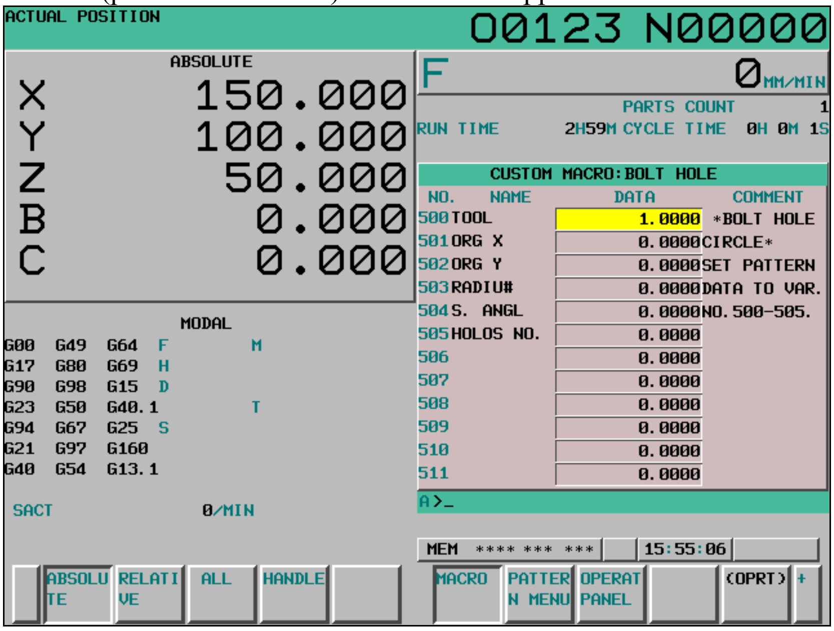
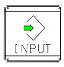


Fig. 12.3.12 (b) Custom macro screen (pattern data) (10.4-inch)

Enter the necessary pattern data, and press  .

After entering all necessary data, select the MEMORY mode and press the cycle start button. Machining begins.

Explanation

- Explanations about the pattern menu screen

HOLE PATTERN

An arbitrary character string consisting of 12 or less characters can be displayed as a menu title.

BOLT HOLE

An arbitrary character string consisting of 10 or less characters can be displayed as a pattern name.

Machine tool builders should program menu title and pattern name character strings, using custom macros, and save them in program memory.
For details on this program, see II-16.

- Explanations about the custom macro screen (pattern data screen)

BOLT HOLE

An arbitrary character string consisting of 12 or less characters can be displayed as a pattern data title.

TOOL.

An arbitrary character string consisting of 10or less characters can be displayed as a variable name.

BOLT HOLE CIRCLE

Up to 12 lines (10.4-inch display unit) or 8 lines (8.4-inch display unit) of comments can be displayed by a comment statement with one block assumed to be 12 characters and with one block to be one line.

Machine tool builders should program variable name and comment text character strings, using custom macros, and save them in program memory.
For details on this program, see II-16.

12.4 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.
This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the memory card (see III-8).
In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed by the operations under function key SYSTEM.

Section 12.4, "SCREENS DISPLAYED BY FUNCTION KEY SYSTEM", consists of the following subsections:

See III-7 for the diagnosis screens displayed by pressing function key SYSTEM.

Screens of a 7.2/8.4/10.4-inch display unit

12.4.1 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.
Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

Procedure

1. Set 1 for PARAMETER WRITE to enable writing. See the procedure for enabling/displaying parameter writing described below.
2. Press function key SYSTEM.
3. Press chapter selection soft key [PARAM] to display the parameter screen.

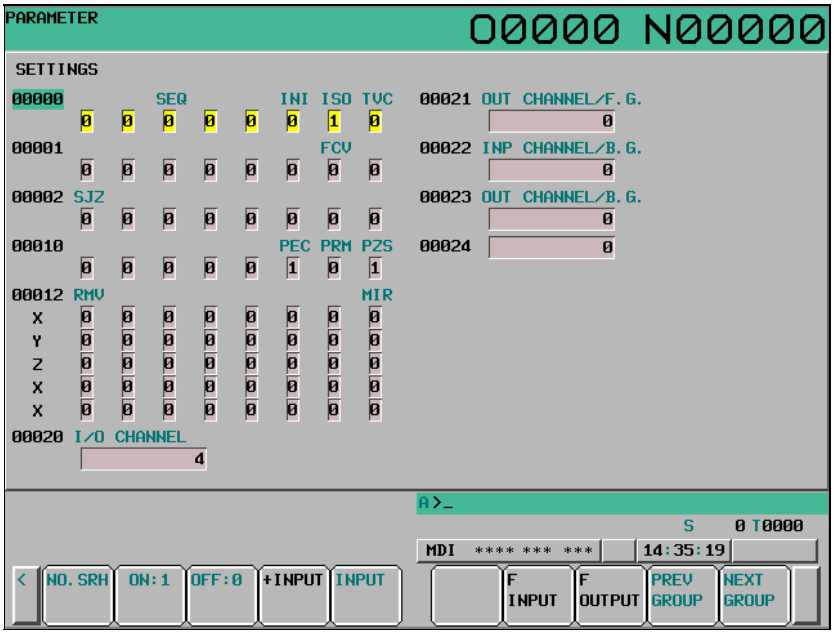


Fig. 12.4.1 (a) PARAMETER screen (10.4-inch)

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:
- Enter the parameter number and press soft key [NO.SRH] .
- Move the cursor to the parameter number using the page keys, and , and cursor keys, , , , and .

5 To set the parameter, enter a new value with numeric keys and press soft key [INPUT] in MDI mode. The parameter is set to the entered value and the value is displayed.

6 Set 0 for PARAMETER WRITE to disable writing.

Procedure for enabling/displaying parameter writing

Procedure

1 Select the MDI mode or enter state emergency stop.

2 Press function key .

3 Press soft key [SETTING] to display the setting screen.

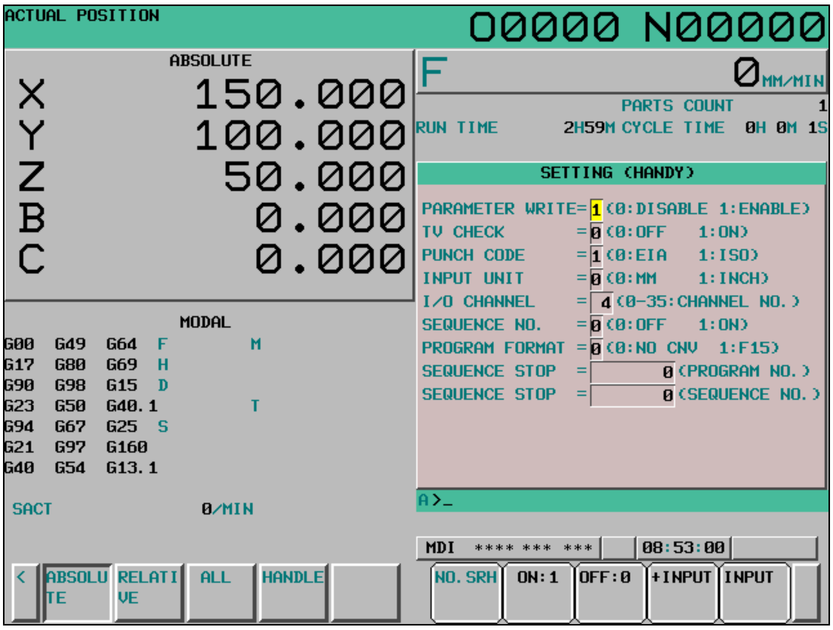


Fig. 12.4.1 (b) SETTING screen (10.4-inch)

4 Move the cursor to PARAMETER WRITE using cursor keys.

5 Press soft key [(OPRT)], then press [ON:1] to enable parameter writing.
At this time, the CNC enters the alarm state SW0100.

6 After setting parameters, return to the setting screen. Move the cursor to PARAMETER WRITE and press soft key [(OPRT)] , then press [OFF:0].

7 Press the  key to release the alarm condition.
When an alarm PW0000 occurred, however, it is not released unless the power is turned off and back on.

Explanation

- Setting parameters with external input/output devices

See III-8 for setting parameters with external input/output devices such as the memory card.

- Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm PW0000. In this case, turn off the power, then turn it on again.

- Parameter list

Refer to the Parameter Manual (B-64310EN) for the parameter list.

- Setting data

Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for PARAMETER WRITE is not necessary when these parameters are set on the setting screen.

12.4.2 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.

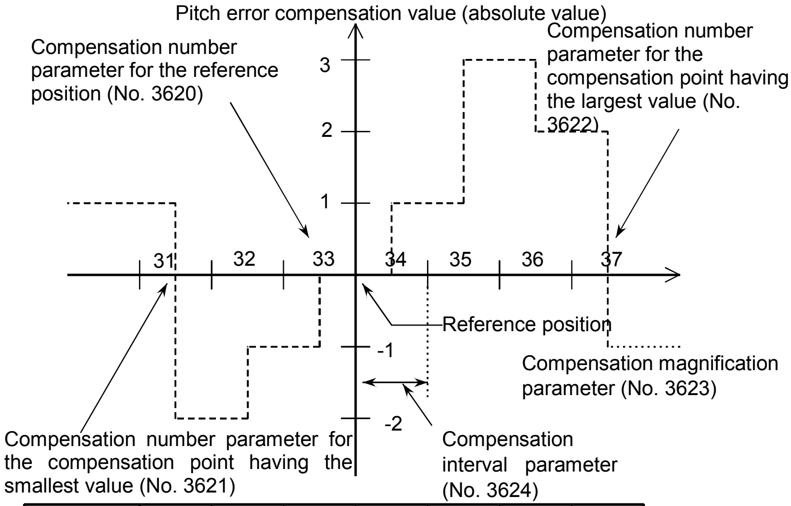
Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.

The pitch error compensation data is set according to the characteristics of the machine connected to the NC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced. In principle, the end user must not alter this data.

Pitch error compensation data can be set with external devices such as the memory card (see Chapter III-8). Compensation data can also be written directly with the MDI panel.

The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.

In the following example, 33 is set for the pitch error compensation point at the reference position.



Compensation position number	31	3	3	3	3	3	3
Compensation value to be set	-3	+1	+1	+1	+2	-1	-3

- Number of the pitch error compensation point at the reference position (for each axis): Parameter No. 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter No. 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter No. 3622
- Pitch error compensation magnification (for each axis): Parameter No. 3623
- Interval of the pitch error compensation points (for each axis): Parameter No. 3624
- Travel distance per revolution of pitch error compensation of the rotary axis type (for each axis): Parameter No. 3625

- Bi-directional pitch error compensation

The bi-directional pitch error compensation function allows independent pitch error compensation in different travel directions. (When the movement is reversed, compensation is automatically carried out as in a backlash.)

To use this function, specify pitch error compensation for each travel direction, that is, separately for the positive and negative directions of a movement.

When using bi-directional pitch error compensation (setting bit 0 (BDPx) of parameter No. 3605 to 1), specify the following parameters in addition to the pitch error compensation parameter.

- Number of the pitch error compensation point at the negative end (for travel in the positive direction, for each axis): Parameter No. 3621
- Number of the pitch error compensation point at the positive end (for travel in the positive direction, for each axis): Parameter No. 3622
- Number of the pitch error compensation point at the negative end (for travel in the negative direction, for each axis): Parameter No. 3626

- Pitch error compensation in the reference position when moving to the reference position from opposite to the reference position return direction (for each axis): Parameter No. 3627

Procedure for displaying and setting the pitch error compensation data

Procedure

1 Set the following parameters:
- Number of the pitch error compensation point at the reference position (for each axis): Parameter No. 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter No. 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter No. 3622
- Pitch error compensation magnification (for each axis): Parameter No. 3623
- Interval of the pitch error compensation points (for each axis): Parameter No. 3624
- Travel distance per revolution of pitch error compensation of the rotary axis type (for each axis): Parameter No. 3625

When using bi-directional pitch error compensation (setting bit 0 (BDPx) of parameter No. 3605 to 1), specify the following parameters in addition to the pitch error compensation parameter.
- Number of the pitch error compensation point at the negative end (for travel in the positive direction, for each axis): Parameter No. 3621
- Number of the pitch error compensation point at the positive end (for travel in the positive direction, for each axis): Parameter No. 3622
- Number of the pitch error compensation point at the negative end (for travel in the negative direction, for each axis): Parameter No. 3626
- Pitch error compensation in the reference position when moving to the reference position from opposite to the reference position return direction (for each axis): Parameter No. 3627

2 Press function key SYSTEM.

3 Press the continuous menu key ▷, then press chapter selection soft key [PITCH ERROR] ([PITCH] for the 8.4-inch display unit). The following screen is displayed:

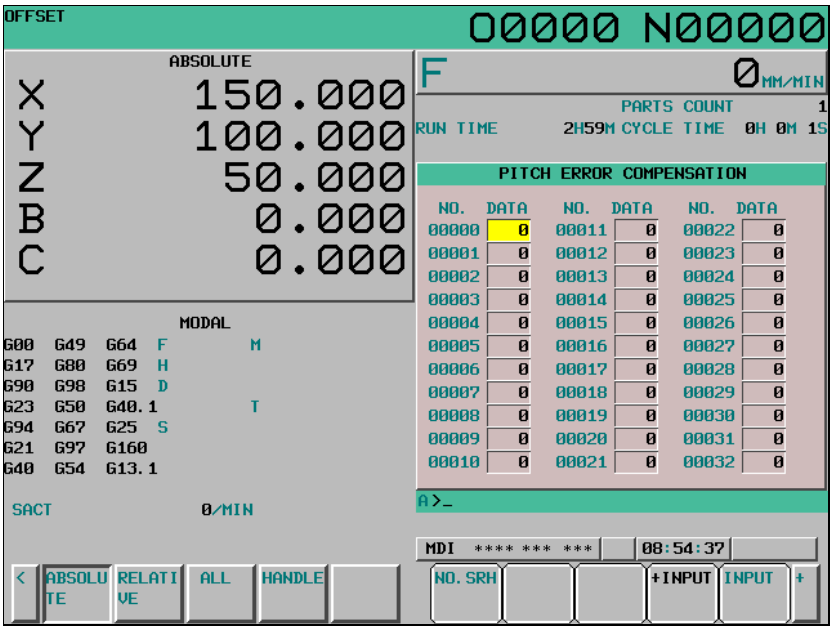


Fig. 12.4.2 (a) PITCH ERROR COMPENSATION screen (10.4-inch)

4 Move the cursor to the compensation point number to be set in either of the following ways:

- Enter the compensation point number and press the soft key [NO.SRH].
- Move the cursor to the compensation point number using the page keys, and , and cursor keys, , , , and .

5 Enter a value with numeric keys and press soft key [INPUT] in MDI mode.

> **NOTE**
> To use pitch error compensation, set bit 0 (NPE) of parameter No. 8135 to 0.

12.4.3 Servo Setting

Enter the machine constants required for servo setting for calculation within the CNC to set the corresponding CNC parameters automatically. Simple help on the item at the cursor position is displayed.

Procedure for servo parameter setting

Procedure

1 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.

2 Press function key , continuous menu key , and soft key [SERVO SETTING] ([SV.SET] for the 8.4-inch display unit) in this order.

4 Press soft key [SERVO SETTING] to select the servo setting screen.
The following screen appears:

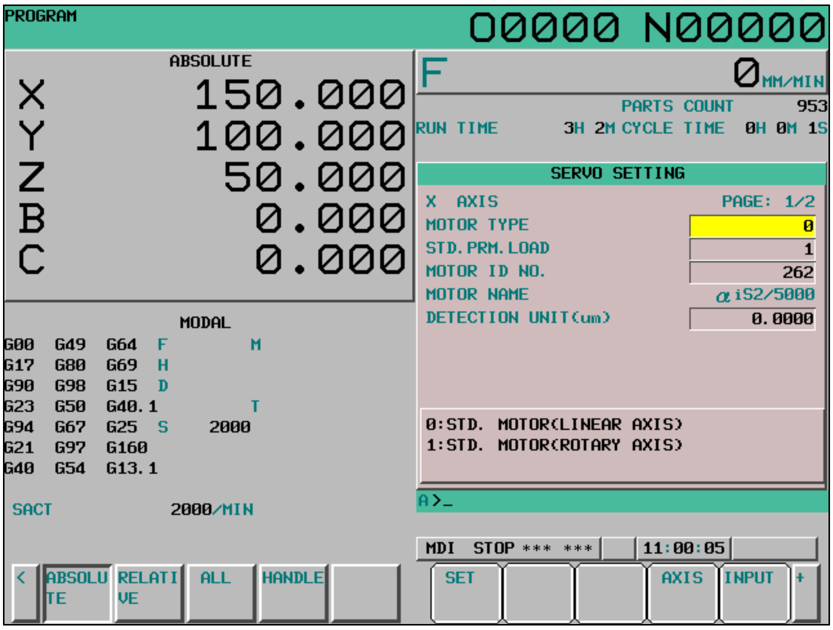


Fig. 12.4.3 (a) Servo parameter setting screen (10.4-inch)

4 Press soft key [AXIS] and select the axis to be set or changed.

5 Move the cursor to the data to be set or changed using the page keys and cursor keys.

6 Key in a setting and press soft key [INPUT] or MDI key [INPUT].

7 Set all items and press soft key [SET].
When data is set successfully, soft key [SET] is hidden. When data is changed, soft key [SET] appears again.

Entering special data

The settings of DIRECTION SET and DIRECTION REVERSE are entered with soft keys.
Move the cursor to the item to be set and press the soft key of the data to be set.
When soft key [(OPRT)] is displayed, press [(OPRT)] to display the soft keys of the data to be set.

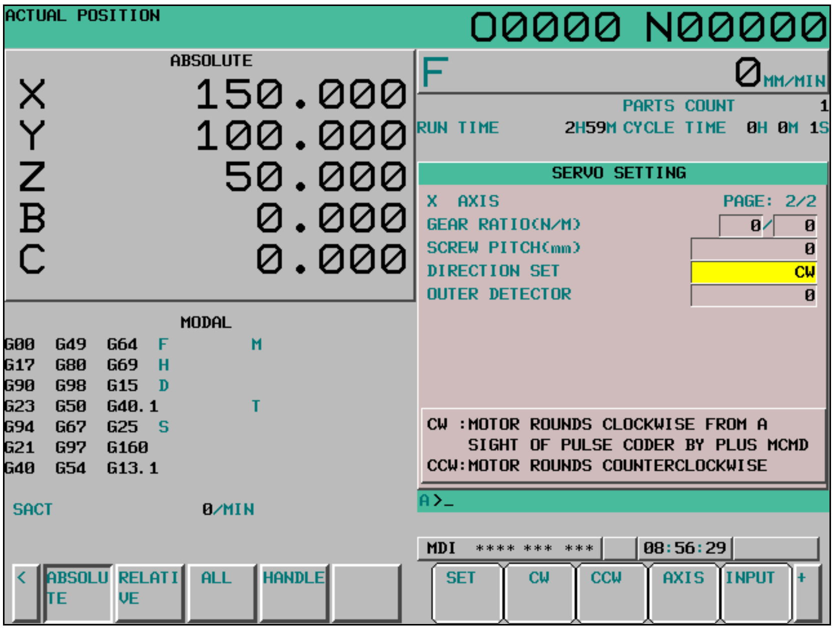


Fig. 12.4.3 (b) Servo setting screen (DIRECTION SET) (10.4-inch)

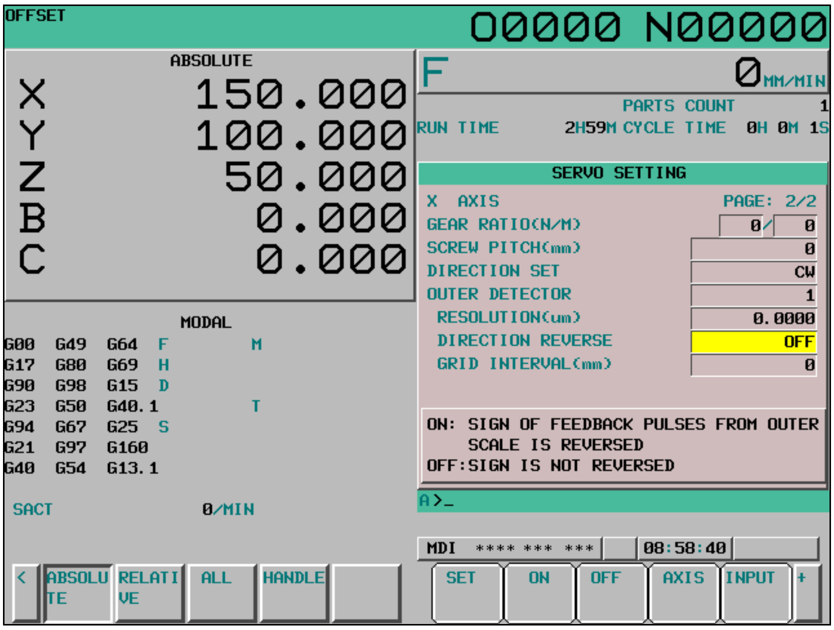


Fig. 12.4.3 (c) Servo setting screen (DIRECTION REVERSE) (10.4-inch)

NOTE

1 The DIRECTION SET item becomes blank "   " if its corresponding parameter is set to an undefined value.

2 It is also possible to input data with numeric keys and then press soft key [INPUT] or MDI key [INPUT].
The soft keys to be displayed and their values are shown below.
- CW: 1, CCW: 0
- ON: 1, OFF: 0

Setting data

When all items are set and soft key [SET] is pressed, the CNC sets the CNC parameters to the calculated results.

When a setting is illegal

If a CNC parameter falls outside the setting range as a result of CNC internal calculation based on each setting, the cursor moves to the DETECTION UNIT item and a warning saying "ILLEGAL SETTING DATA" is issued.
Enter an detection unit that can be set and press soft key [SET] again.

Setting a detection unit automatically

When the cursor is placed on the DETECTION UNIT item, soft key [AUTO] appears. The detection unit can be set automatically by pressing soft key [AUTO].
The detection unit to be set automatically is calculated based on the settings of other items and the values of the parameters fall within the setting ranges.

Fig. 12.4.3 (d) Soft keys for selection of a detection unit (10.4-inch)

> **NOTE**
> 1 When the detection unit setting is 0, the detection unit is set to 1.0000 or 0.1000. If bit 1 (ISC) of parameter No. 1013 is 1, the detection unit is 0.1000.
> 2 For an axis for which bit 3 (DIA, direct specification) of parameter No. 1006 is set to 1, the detection unit is set to half the setting value (detection unit of a radius). (Only for the T series)

Displaying the servo setting screen for inputting parameters

Press soft key [(OPRT)] and press continuous menu key ▷ to display soft key [CHANGE].
Press soft key [CHANGE] to display the servo setting screen for inputting parameters. At this time, the screen for the axis to be set appears and the cursor moves to the first item.

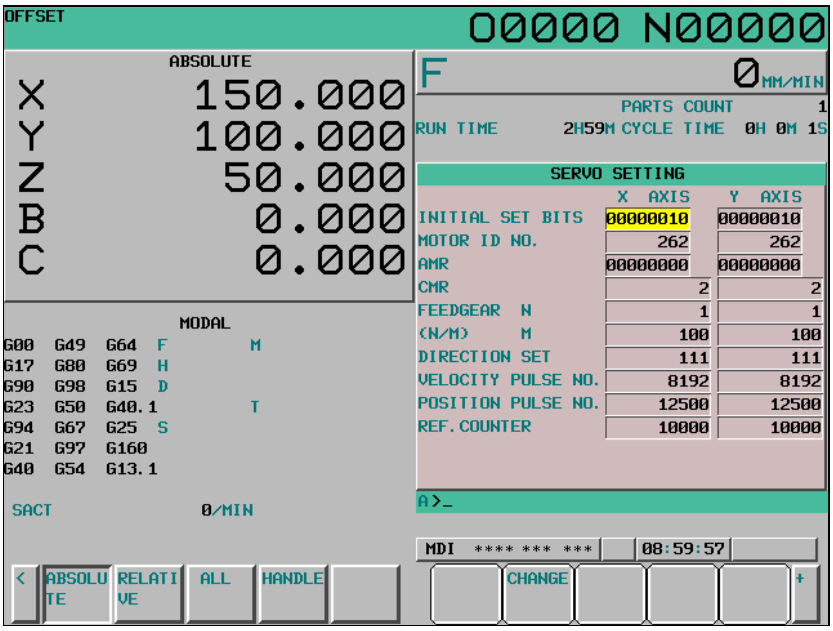


Fig. 12.4.3 (c) Servo setting screen for inputting parameters (10.4-inch)

To display the servo setting screen for inputting machine constants again, press soft key [CHANGE] in the same procedure. At this time, the servo setting screen for inputting machine constants is displayed with the axis selected by the cursor on the servo setting screen for inputting parameters specified as the target axis.
To prevent the servo setting screen for inputting machine constants from being displayed, set bit 2 (SVO) of parameter No. 13117 to 1.

12.4.4 Servo Tuning

Data related to servo tuning is displayed and set.

Procedure for servo tuning

Procedure

1 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
2 Press function key [SYSTEM], continuous menu key [▷], and soft key [SERVO SETTING] ([SV.SET] for the 8.4-inch display unit) in this order.
3 Press soft key [SERVO TUNE] ([SV.TUN] for the 8.4-inch display unit) to select the servo tuning screen.
The following screen appears:

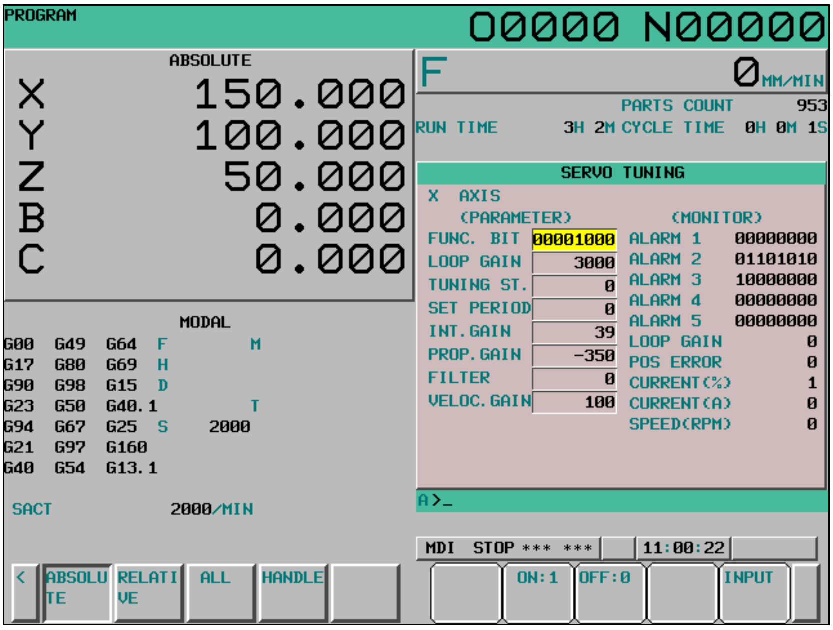


Fig. 12.4.4 (a) Servo tuning screen (10.4-inch)

4 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
5 Key in a desired value then press soft key [INPUT].

12.4.5 Spindle Setting

The machine constants required to start a spindle are input to perform calculation by the CNC. When the CNC is restarted, the parameters required to start the spindle are set.

Setting spindle parameters

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key SYSTEM, continuous menu key ▷, then soft key [SPINDL SETING]([SP.SET] for the 8.4-inch display unit).

3 Press soft key [SPINDL SETING] to select the spindle setting screen.
The following screen appears:

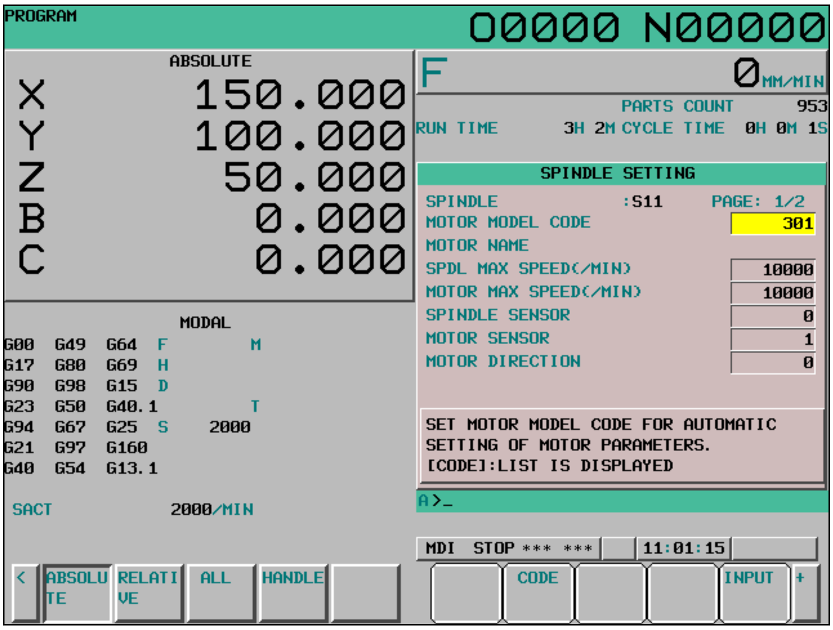


Fig. 12.4.5 (a) Spindle setting screen for entering machine constants (10.4-inch)

4 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.

5 Key in the setting and press soft key [INPUT] or MDI key [INPUT].

6 After the all machine constants required to start a spindle are input, press soft key [SET]. A necessary parameter is calculated by this operation. After the calculation ends, the parameters required to start the spindle are set by the CNC is restarted.

Changing the spindle to be set

Press soft key [(OPRT)] to display soft key [SPINDL CHANGE] ([SP.CHG] for the 8.4-inch display unit). Soft key [SPINDL CHANGE] is used to change the spindle to be set. Press soft key [SPINDL CHANGE] several times to select the spindle to be set.

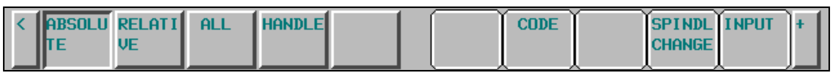


Fig. 12.4.5 (b) Spindle change soft keys (10.4-inch)

> **NOTE**
> When multiple serial spindles are not connected, soft key [SP.CHG] is not displayed.

Input by soft keys

The settings of MOTOR SENSOR, PROXIMITY SWITCH EDGE, MOTOR DIRECTION, and POS. CODER DIRECTION are made by soft keys. When the cursor is moved to the item to be set, the following soft keys are displayed. Press the soft key for the data to be set to input data.
When soft key [(OPRT)] is displayed, press [(OPRT)] to display the soft keys for the data to be set.

Fig. 12.4.5 (c) Soft keys displayed for MOTOR SENSOR and PROXIMITY SWITCH EDGE (10.4-inch)

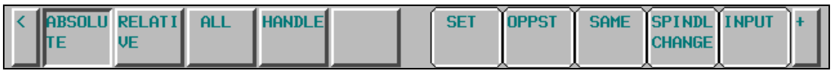


Fig. 12.4.5 (d) Soft keys displayed for MOTOR DIRECTION and POS. CODER DIRECTION (10.4-inch)

> **NOTE**
> It is also possible to input data with numeric keys and press soft key [INPUT] or MDI key [INPUT] to enter them.
> The soft keys to be displayed and their corresponding values are shown below.
> - ON: 1, OFF: 0
> - OPPST: 1, SAME: 0

Input from a list of motor model codes

The data of MOTOR MODEL CODE can be input on the motor model code list screen. The motor model code list screen appears when soft key [CODE] is pressed. Soft key [CODE] appears when the cursor is placed on the MOTOR MODEL CODE item.

To return from the motor model code list screen to the previous screen, press soft key [RETURN].

Fig. 12.4.5 (e) Soft keys displayed for MOTOR MODEL CODE (10.4-inch)

Fig. 12.4.5 (f) Soft keys displayed on the motor model code list screen (10.4-inch)

When the motor model code list screen is displayed, motor model codes and their corresponding motor names and amplifier names are listed. When the cursor is moved to the code number to be set and soft key [SELECT] is pressed, input is completed. Upon completion of input, the previous screen is displayed.

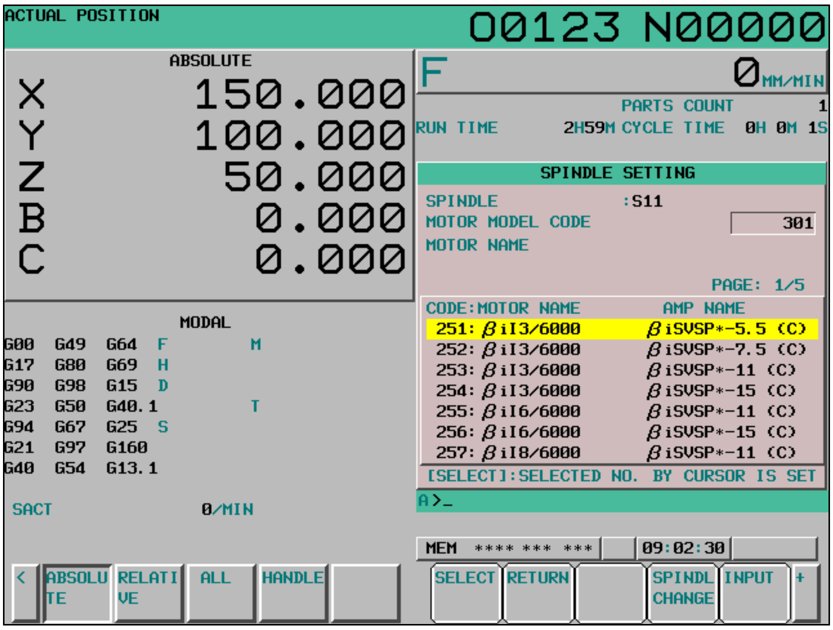
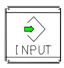


Fig. 12.4.5 (g) Motor model code list screen (10.4-inch)

> **NOTE**
> To input a motor model code not listed, press soft key [INPUT] or  key on the MDI panel and input the motor model code.

Displaying the spindle setting screen for inputting parameters

Press soft key [(OPRT)] and continuous menu key to display soft key [CHANGE].
Press soft key [CHANGE] to display the spindle setting screen for inputting parameters. At this time, the spindle to be set is displayed with the cursor placed at the beginning.

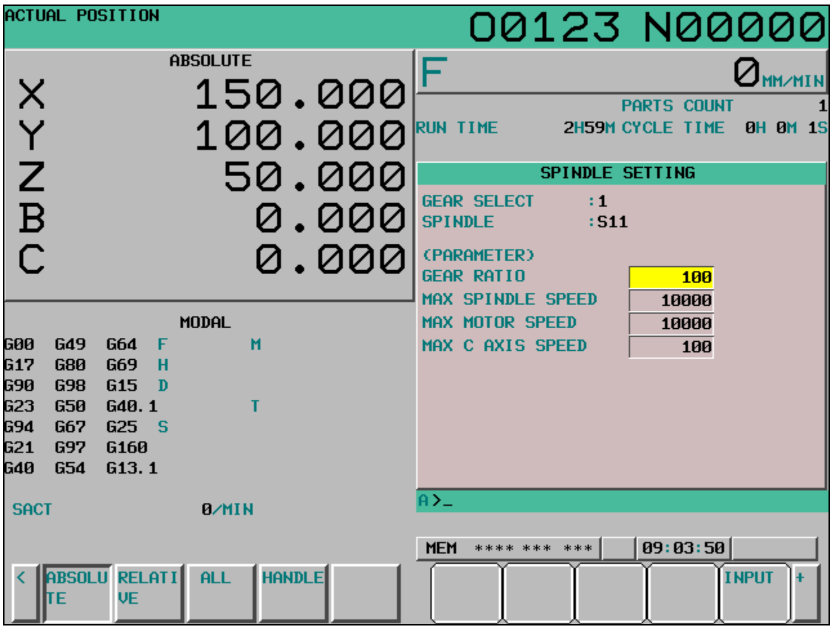


Fig. 12.4.5 (h) Spindle setting screen for inputting parameters (10.4-inch)

To display the spindle setting screen for inputting machine constants again, press soft key [CHANGE] in the same procedure. At this time, the target is the spindle displayed on the spindle setting screen for inputting parameters with the cursor placed at the beginning.
To prevent the spindle setting screen for inputting machine constants from being displayed, set bit 2 (SDO) of parameter No. 13118 to 1.

12.4.6 Spindle Tuning

Spindle tuning data is displayed and set.

Setting for spindle tuning

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.
2 Press function key SYSTEM, continuous menu key , then soft key [SPINDL SETING]([SP.SET] for the 8.4-inch display unit).
3 Press soft key [SPINDL TUNE] ([SP.TUN] for the 8.4-inch display unit) to select the spindle tune screen.
4 The following screen appears:

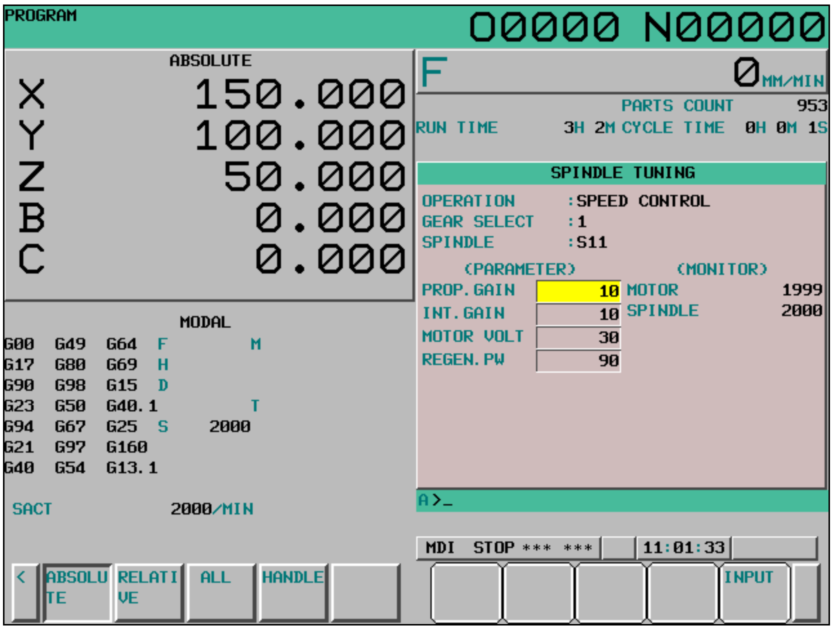


Fig. 12.4.6 (a) Spindle tuning screen (10.4-inch)

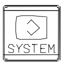
5 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
6 Key in a desired value then press soft key [INPUT].

12.4.7 Spindle Monitor

Spindle-related data is displayed.

Displaying the spindle monitor

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.
2 Press function key , continuous menu key , then soft key [SPINDL SETING]([SP.SET] for the 8.4-inch display unit).
3 Press soft key [SPINDL MONIT.] ([SP.MON] for the 8.4-inch display unit) to select the spindle monitor screen.
4 The following screen appears:

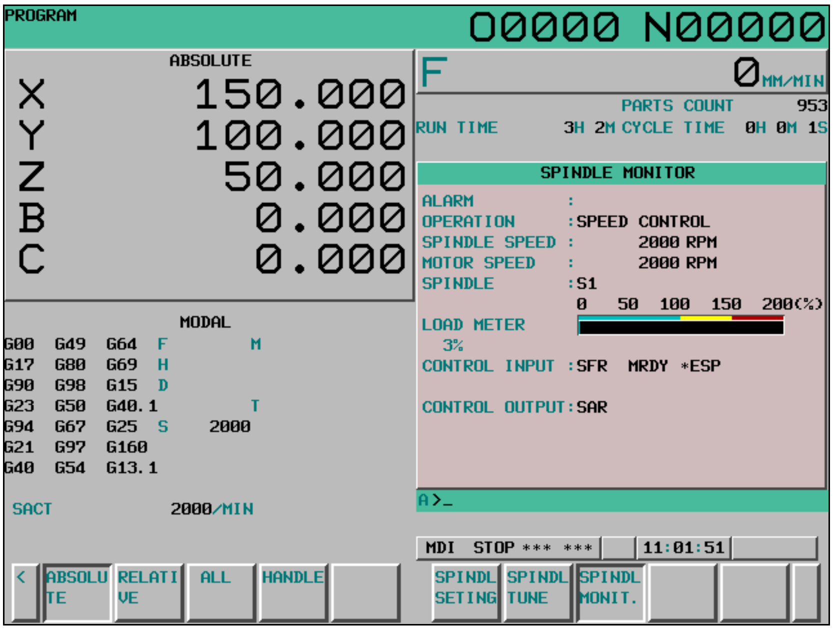


Fig. 12.4.7 (a) Spindle monitor screen (10.4-inch)

12.4.8 Color Setting Screen

Screen colors can be set on the color setting screen.

Displaying the color setting screen

Procedure

1 Press function key SYSTEM.

2 Press the continuous menu key several times to display soft key [COLOR].

3 Press soft key [COLOR] to display the color setting screen.

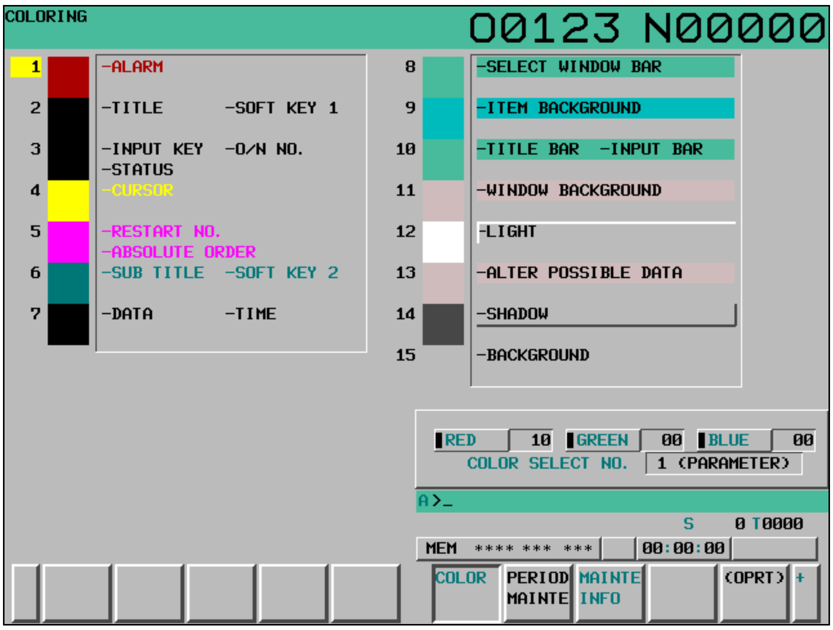
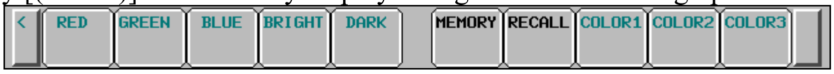


Fig. 12.4.8 (a) Color setting screen (10.4-inch)

Procedure for operating the color setting screen

- Modifying the color (color palette values)

1 Press soft key [(OPRT)]. The soft key display changes to the following operation soft keys:



2 Move the cursor to a color number whose color palette values are to be modified.
The current color palette value set for each primary color is displayed.

3 Select a primary color whose setting is to be modified, with the corresponding operation soft key [RED], [GREEN], or [BLUE].
More than one primary color can be selected at the same time.
Each time operation soft key [RED], [GREEN], or [BLUE] is pressed, the operation soft key toggles between selection and deselection.
(When operation soft keys [RED], [GREEN], and [BLUE] are not displayed, press the rightmost soft key to display the operation soft keys.)

4 Select operation soft key [BRIGHT] or [DARK] to modify the brightness of the selected prime color(s).

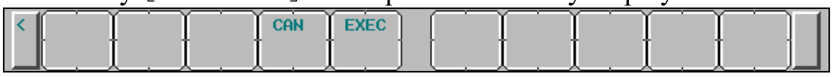
- Storing the color (color palette values)

The set color palette values can be stored.

1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area.
(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)

COLOR1 Standard color data parameters Nos. 6581 to 6595
COLOR2 Parameters Nos. 10421 to 10435
COLOR3 Parameters Nos. 10461 to 10475

2 Press operation soft key [MEMORY]. The operation soft key display switches to the following:

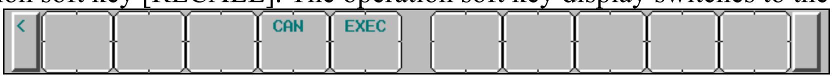


3 Pressing operation soft key [EXEC] stores the current color palette settings in the selected area.
Pressing operation soft key [CAN] or the leftmost key does not store the current color palette settings in the selected area.

- Calling the color (color palette values)

1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area where color palette values are stored.
(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)

2 Press operation soft key [RECALL]. The operation soft key display switches to the following:



3 Pressing operation soft key [EXEC] calls the color palette values from the selected area to enable the color to be modified. This operation is invalid if no color palette values are stored.
Pressing operation soft key [CAN] or the leftmost key does not call the color palette values from the selected area.

NOTE
1 Immediately after the power is turned on, the settings of COLOR1 (parameters) are used for display. If nothing is stored for COLOR1, the default colors are used for display.

NOTE
2 Do not modify the color setting data parameters directly by MDI key input. When modifying the standard color data, be sure to perform a storage operation on the color setting screen.

12.4.9 Machining Parameter Tuning

12.4.9.1 Machining parameter tuning (AI contour)

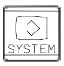
In advanced preview control, AI advanced preview control, and AI contour control, by setting a velocity-emphasized parameter set and precision-emphasized parameter set and setting the precision level matching a machining condition such as rough machining or finish machining on the precision level setting screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on velocity (precision level 1) and emphasis on precision (precision level 10) can be set.
Set the following parameters:

- Acceleration rate of acceleration/deceleration before interpolation
- Acceleration change time (bell-shaped)
- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control
- Allowable acceleration change value for each axis in acceleration change under jerk control in successive linear interpolation operations
- Ratio of the change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation
- Allowable acceleration rate
- Acceleration rate of acceleration/deceleration after interpolation
- Corner speed difference
- Maximum feedrate
- Items that can be set freely (2 items)

For details of each parameter, see the descriptions of AI contour control.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a precision level, see the description of the precision level selection screen in Subsection 12.3.10.

Procedure for machining parameter tuning

Procedure

1 Set the MDI mode.
2 Press function key 

.
3 Press soft key [MCHN TUNING] ([M-TUN] for the 8.4-inch display unit) to display the machining parameter tuning screen.

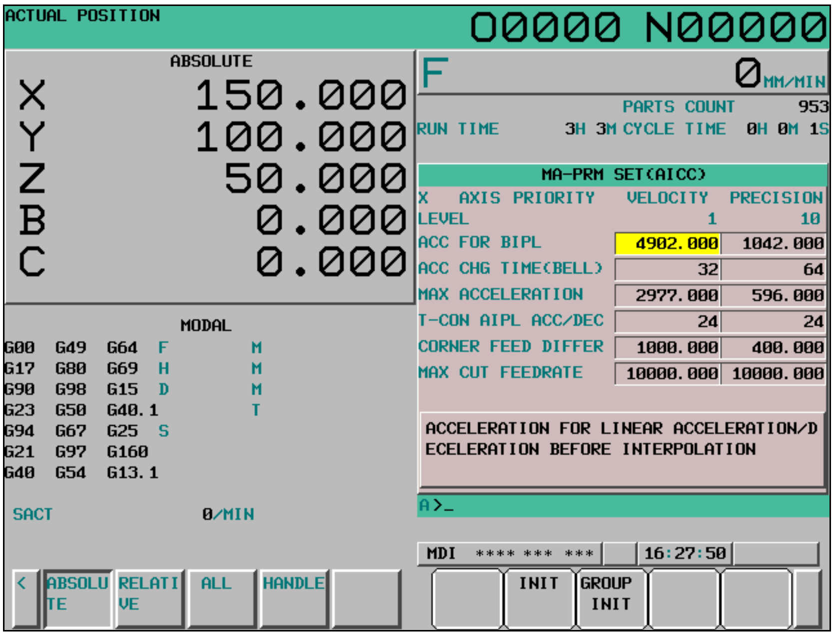


Fig. 12.4.9 (a) Machining parameter tuning screen (10.4-inch)

4 Move the cursor to the position of a parameter to be set, as follows:
Press page key or , and cursor keys , , and /or to move the cursor to the parameter.

5 Key in desired data then press the key on the MDI panel.

6 When data is input, a RMS value is found according to the precision level parameters. If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed The parameters of precision level can be changed on the precision level selection screen or parameter setting screen..

7 Repeat steps 4 and 5 until all machining parameters are set.

8 In addition to the setting method described above, a parameter setting method using soft keys is available. Pressing soft key [INIT] displays the standard value (recommended by FANUC) of the item selected by the cursor in the key input buffer. Pressing soft key [EXEC] initializes the item to the standard value. Pressing soft key [GROUP INIT] ([G_INIT] for the 8.4-inch display unit) initializes all items of a group (emphasis on velocity or emphasis on precision) selected by the cursor to the standard values.

The table below indicates the initial settings.

Table 12.4.9 (a) Initial settings

Setting item	AI contour control		Unit
	Emphasis on velocity (LV1)	Emphasis on precision (LV10)	
Acceleration rate of acceleration/deceleration before interpolation <ACC FOR BIPL>	4902.000	1042.000	mm/sec^2
Acceleration change time (bell-shaped) <ACC CHG TIME(BELL)>	32	64	msec
Allowable acceleration change value <JERK ACC DIFF>	0	0	mm/sec^2
Allowable acceleration change value in successive linear interpolation operations <JERK ACC DIFF(LIN)>	0	0	mm/sec^2
Ratio of the change time of the jerk control <JERK ACC RATIO>	0	0	%

Setting item	AI contour control		Unit
	Emphasis on velocity (LV1)	Emphasis on precision (LV10)	
Allowable acceleration rate <MAX AACELERATION>	2977.000	596.000	mm/sec^2
Time constant for acceleration/deceleration after interpolation <T-CON AIPL ACC/DEC>	24	24	msec
Corner speed difference <CORNER FEED DIFFER>	1000	400	mm/min
Maximum cutting speed <MAX CUT FEEDRATE>	10000	10000	mm/min

Explanation

- Look-ahead acceleration/deceleration before interpolation

Set an acceleration rate for a linear portion in look-ahead acceleration/deceleration before interpolation.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13610 (velocity-emphasized parameter)
Parameter No. 13611 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1660: Maximum allowable acceleration rate for each axis in acceleration/deceleration before interpolation

- Acceleration change time (bell-shaped)

Set a time constant for a bell-shaped portion in acceleration/ deceleration before look-ahead interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13612 (velocity-emphasized parameter)
Parameter No. 13613 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1772: Time constant for bell-shaped look-ahead acceleration/deceleration before interpolation of constant acceleration time type

> ⚠ **CAUTION**
> A set time constant is applied to all axes. So, a modification made to this item changes the settings for all axes.

- Allowable acceleration change value in velocity control based on acceleration change under jerk control

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13614 (velocity-emphasized parameter)
Parameter No. 13615 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1788: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13616 (velocity-emphasized parameter)
Parameter No. 13617 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1789: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

⚠ CAUTION
1 For an axis with 0 set in this parameter, the parameters (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615) are valid.
2 For an axis with 0 set in the parameter (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615), velocity control based on acceleration change is disabled, so that this parameter has no effect.

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

Unit of data: %

Set the ratio (in %) of the change time of jerk control to the change time of acceleration in smooth bell-shaped acceleration/deceleration before interpolation.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13618 (velocity-emphasized parameter)
Parameter No. 13619 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1790: Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration rate

Set an allowable acceleration rate in acceleration-based speed determination.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13620 (velocity-emphasized parameter)
Parameter No. 13621 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1735: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in circular interpolation
Parameter No. 1737: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in AI contour control

> ⚠ **CAUTION**
> When bit 0 (MCR) of parameter No. 13600 is set to 1, the deceleration function based on acceleration in circular interpolation is not set.

- Time constant for acceleration/deceleration after interpolation

Set a time constant for acceleration/deceleration after interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13622 (velocity-emphasized parameter)
Parameter No. 13623 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1769: Time constant for acceleration/deceleration after cutting feed interpolation

- Corner speed difference

Set an allowable corner speed difference used for speed determination.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13624 (velocity-emphasized parameter)
Parameter No. 13625 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1783: Allowable speed difference for each axis in automatic corner deceleration based on speed difference

- Maximum cutting speed

Set a maximum cutting speed for each axis.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13626 (velocity-emphasized parameter)
Parameter No. 13627 (precision-emphasized parameter)

Moreover, the following parameter is also set from the precision level:
Parameter No. 1432: Maximum cutting feedrate for each axis in the AI contour control mode

- Arbitrary items

Two arbitrary parameters can be registered. Each item can correspond to a CNC parameter or servo parameter. A parameter number corresponding to each item is to be specified with parameters.

As indicated below, set the parameters for corresponding parameter numbers, velocity-emphasized parameters (precision level 1), and precision-emphasized parameters (precision level 10).

Table 12.4.9 (b) Parameters related to arbitrary items

	Corresponding parameter number	Setting of velocity-emphasized (precision level 1) value	Setting of precision-emphasized (precision level 10) value
Arbitrary item 1	No.13628	No.13630	No.13632
Arbitrary item 2	No.13629	No.13631	No.13633

- Display
  Tuning target parameter numbers are displayed.

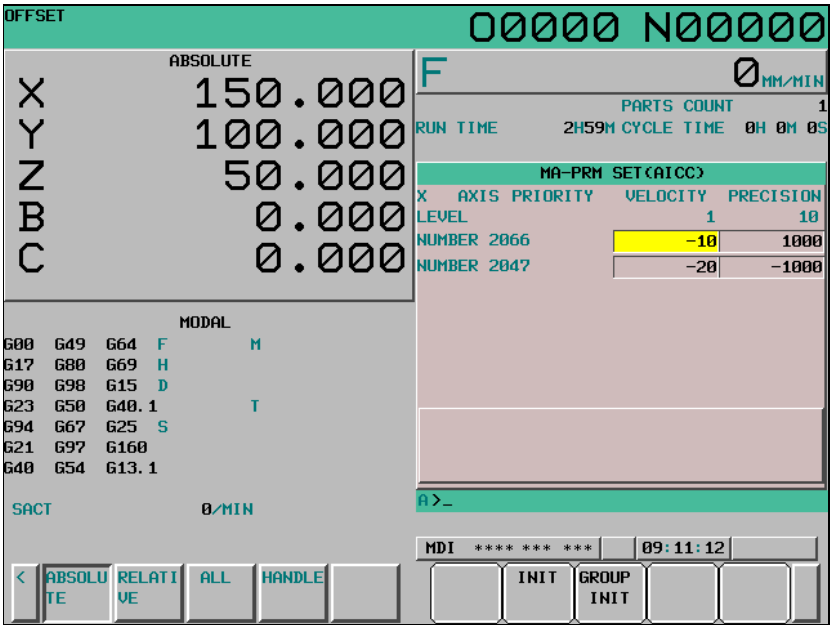


Fig. 12.4.9 (b) Machining parameter tuning screen (10.4inch)
Example of using arbitrary items

> **NOTE**
> As arbitrary items, the numbers of the following parameters cannot be specified:
> - Bit parameter
> - Spindle parameters (Parameter Nos. 4000 to 4799)
> - Real-type parameter
> - Power-off parameter
> - Nonexistent parameter

12.4.9.2 Machining parameter tuning (nano smoothing) (M Series)

M

In nano smoothing, by setting a parameter set and setting the smoothing level matching a machining condition on the smoothing level selection screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.

On this screen, the parameter sets for emphasis on precision (smoothing level 1) and emphasis on surface smoothing (smoothing level 10) can be set.
Set the following parameters:

- Tolerance

For details of each parameter, see the descriptions of nano smoothing.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a smoothing level, see the description of the smoothing level selection screen.

NOTE
These setting items are displayed only when machining quality level adjustment is enabled.

Procedure for machining parameter tuning

1 Set the MDI mode.
2 Press function key SYSTEM.
3 Press soft key [MCHN TUNING].
4 Press soft key [NANO SMOOTH] to display the machining parameter tuning screen.

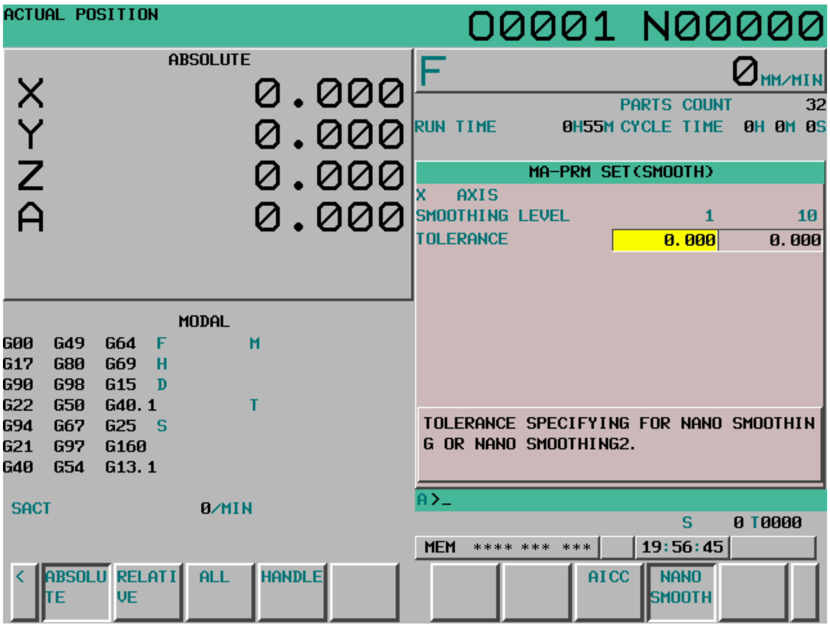


Fig. 12.4.9.2 (a) Machining parameter tuning screen (nano smoothing)

5 Move the cursor to the position of a parameter to be set, as follows:
Press page key PAGE ↑ or PAGE ↓, and cursor keys ↑, ↓, ← and /or → to move the cursor to the parameter.
6 Key in desired data then press the INPUT key on the MDI panel.
7 When data is input, a RMS value is found according to the smoothing level parameters. (The smoothing level can be changed on the smoothing level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.
8 Repeat steps 5 and 6 until all machining parameters are set.

Explanation

- Tolerance

Set the value specified for the tolerance for nano smoothing.

Unit of data: mm, inch, degree (input unit)

The parameter set on the machining parameter tuning screen (smoothing) is reflected in the following parameters:
Parameter No. 11682 (smoothing level 1)
Parameter No. 11683 (smoothing level 10)

Moreover, the following parameter is also set according to the smoothing level:
Parameter No. 19541: Tolerance specified for nano smoothing

> ⚠ **CAUTION**
> Since the tolerance specified for nano smoothing is common to all axes, changing this item changes the setting for all axes.

12.4.10 Parameter Setting Support Screen

The parameter setting support screen is a screen for parameter setting and tuning designed to achieve the following:

1 The minimum required parameters that must be set when the machine is started up are collectively displayed to facilitate start-up of the machine.
2 The servo tuning screen, spindle tuning screen, and machining parameter setting support screen are displayed for smooth tuning.

The parameter setting support screen consists of a menu screen and several setting screens.

12.4.10.1 Displaying the menu screen and selecting a menu item

The parameter setting support menu screen displays the following items:
[START UP]

- AXIS SETTING
- FSSB (AMP)
- FSSB (AXIS)
- SERVO SETTING
- SERVO PARAMETER
- SERVO GAIN TUNING
- HIGH-PRECISION
- SPINDLE SETTING
- MISCELLANY

[TUNING]

- SERVO TUNING
- SPINDLE TUNING
- AICC TUNING

On the parameter setting support menu screen, one of the displayed items can be selected to display the corresponding screen. From each setting screen, you can return to the menu screen by performing a soft key operation.

> **NOTE**
> Some items may not be displayed, depending on the system configuration.

Displaying the menu screen and selecting a setting screen

Procedure

1 Set the MDI mode.

2 Switch the setting of "PARAMETER WRITE" to "ENABLED". For details, see the procedure for "PARAMETER WRITE" in Subsection III-12.4.1.

3 Press function key SYSTEM.

4 Press the continuous menu key several times.

5 Press soft key [PARAMETER] ([PRMSET] for the 8.4-inch display unit) to display the menu screen for parameter setting support.

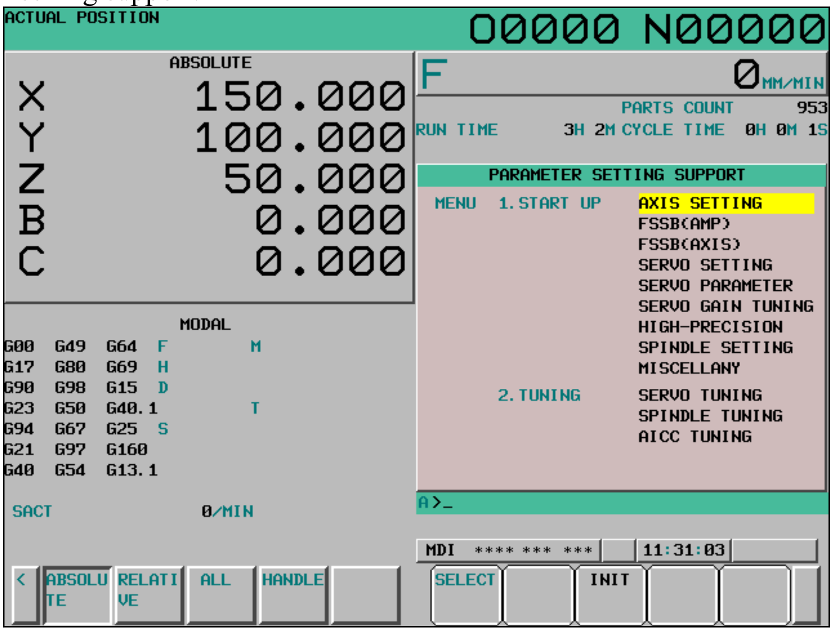


Fig. 12.4.10.1 (a) Menu screen for parameter setting support (10.4-inch)

6 Move the cursor to a desired item by pressing cursor key or .

7 Press soft key [SELECT]. The screen display switches to the selected screen.

Returning to the menu screen

Procedure

1 Press soft key [SELECT] on the parameter setting support menu screen. The screen and soft keys shown below are displayed. (The screen below is displayed when "AXIS SETTING" is selected.)

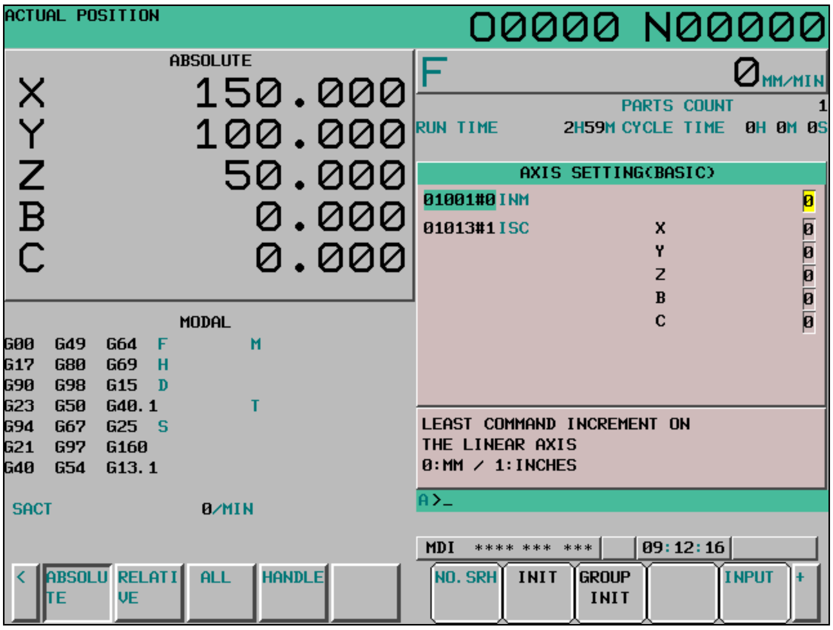


Fig. 12.4.10.1 (b) Axis setting screen (10.4-inch)

2 Press the continuous menu key ▷ several times.

3 Press soft key [MENU].
The screen display returns to the parameter setting support menu screen.

4 Upon completion of parameter setting, switch the setting of "PARAMETER WRITE" to "DISABLED".

> **NOTE**
> Some setting screens can also be displayed by a chapter selection soft key. If a screen is selected using a chapter selection soft key, however, you cannot return to the parameter setting support menu screen.

Explanation

- Items displayed with [START UP]

The items of [START UP] indicate the screens for setting the minimum required parameters for starting up the machine.

Table 12.4.10.1 (a) Items displayed with [START UP]

Display item	Description
AXIS SETTING	Sets the CNC parameters of axes, spindles, coordinates, feedrates, and acceleration/deceleration.
FSSB (AMP)	Displays the FSSB amplifier setting screen.
FSSB (AXIS)	Displays the FSSB axis setting screen.
SERVO SETTING	Displays the servo setting screen.
SERVO PARAMETER	Sets the CNC parameters of servo current control, speed control, position control, and backlash acceleration.
SERVO GAIN TUNING	Adjusts the speed loop gain automatically.
HIGH-PRECISION	Sets the CNC parameters of servo time constants and automatic acceleration/deceleration.
SPINDLE SETTING	Displays the spindle setting screen.
MISCELLANY	Sets the CNC parameters of axis path assignment, DI/DO, and serial spindles.

- Items displayed with [TUNING]

The items of [TUNING] indicate the screens for servo, spindle, and high-speed high-precision machining tuning.

Fig. 12.4.10.1 (b) Items displayed with [TUNING]

Display item	Description
SERVO TUNING	Servo tuning screen
SPINDLE TUNING	Spindle tuning screen
AICC TUNING	Machining parameter tuning screen

NOTE
Some items may not be displayed, depending on the system configuration.

12.4.10.2 Displaying and setting the axis setting screen

This screen enables the CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration to be displayed and set. The parameters displayed can be divided into four groups:

(Basic) group :
The parameters related to basic settings are displayed.

(SPINDLE) group :
The parameters related to spindles are displayed.

(Coordinate) group :
The parameters related to coordinates are displayed.

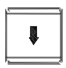
(Feedrate) group :
The parameters related to feedrate are displayed.

(Acceleration/deceleration) group:
The parameters related to acceleration/deceleration are displayed.

The parameters can be initialized to the standard values (recommended by FANUC).

Display and setting

Procedure

1 Move the cursor to [AXIS SETTING] by pressing cursor key  or  on the parameter setting support menu screen.

2 Press soft key [SELECT]. The screen display switches to the screen and soft keys shown below.

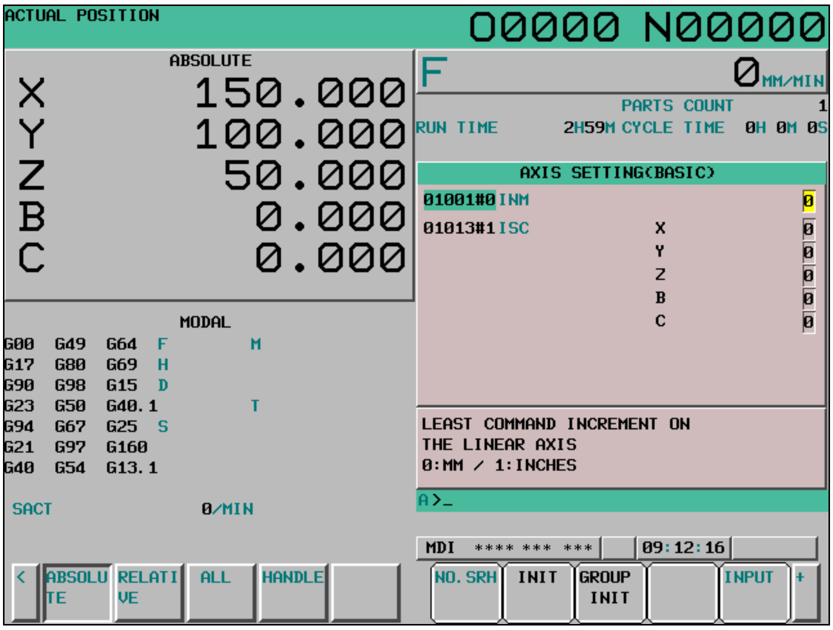


Fig. 12.4.10.2 (a) Parameter setting support screen (axis setting) (10.4-inch)

3 Move the cursor to a parameter number to be set or displayed, according to one of the methods below.
- Enter the parameter number and press soft key [NO.SRH] .
- Move the cursor to the desired parameter number by pressing page key or , and cursor keys , , , and/or .

  A brief description of the parameter where the cursor is placed is provided at the bottom of the screen. However, no description is provided when the cursor is placed on multiple bits for bit parameters.

4 Input desired data then press the key on the MDI panel to set the parameter.

5 Press soft key [INIT]. The standard value (recommended by FANUC) for the item selected by the cursor is displayed in the key input buffer. Pressing soft key [EXEC] in this state initializes the item to the standard value.

6 Press soft key [GROUP INIT]([G_INIT] for the 8.4-inch display unit). A message asking whether to set the group standard values is displayed on the screen. Pressing soft key [EXEC] in this state inputs all of the standard values of the group.

NOTE

1 If the cursor is placed on a parameter that has no standard value assigned, no standard value is input even when [INIT] is pressed.

2 When the cursor is placed on multiple bits for bit parameters, the multiple bits can be input simultaneously. When [INIT] is pressed in this state, the key input buffer displays the standard values for the bits where the cursor is placed. If a bit has no standard value assigned, "*" is displayed for the bit, and no value is input for the bit.

3 When [GROUP INIT]([G_INIT] for the 8.4-inch display unit) is pressed, those parameters that have no standard values assigned are not initialized.

12.4.10.3 Displaying and setting the FSSB amplifier setting screen

From the parameter setting support screen, the FSSB amplifier setting screen can be displayed. For details of the FSSB amplifier setting screen, see the description of the FSSB amplifier setting screen in Subsection 1.4.4 in the Connection Manual (Function) (B-64303EN-1).

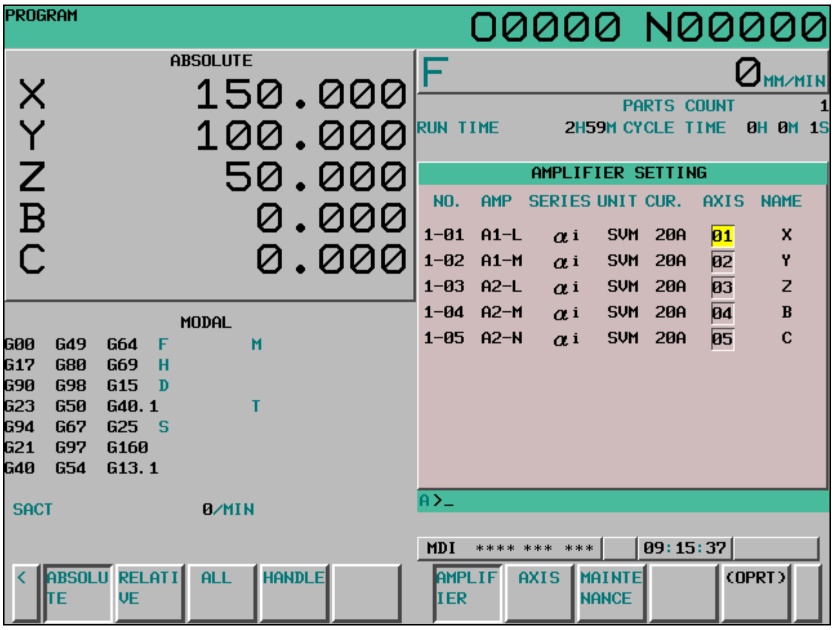


Fig. 12.4.10.3 (a) FSSB amplifier setting screen (10.4-inch)

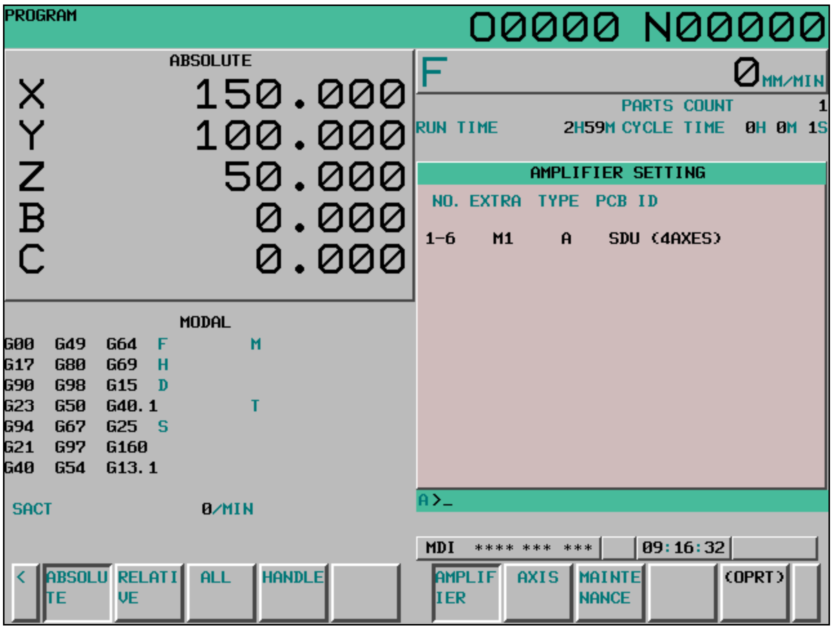


Fig. 12.4.10.3 (b) FSSB amplifier setting screen 2 (10.4-inch)

12.4.10.4 Displaying and setting the FSSB axis setting screen

From the parameter setting support screen, the FSSB axis setting screen can be displayed. For details of the FSSB axis setting screen, see the description of the FSSB axis setting screen Subsection 1.4.4 in the Connection Manual (Function) (B-64303EN-1).

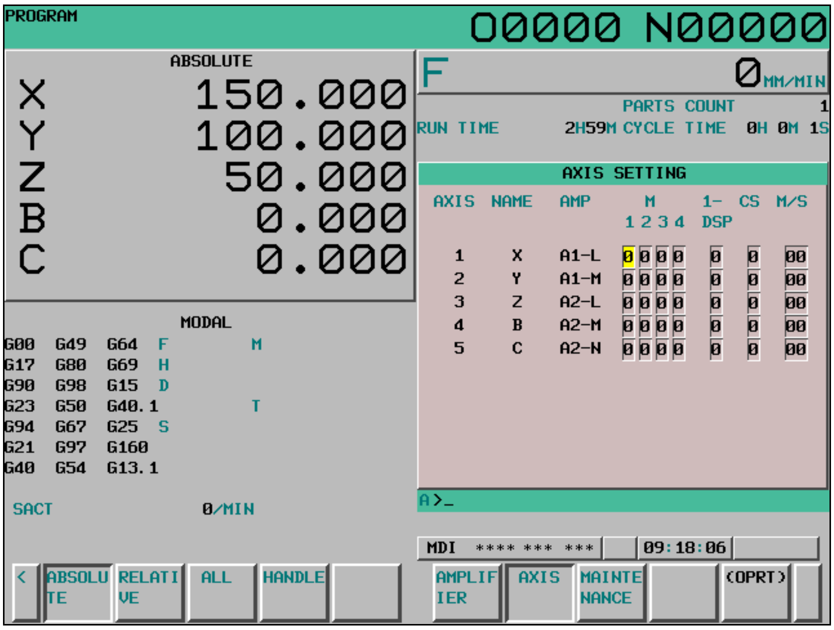


Fig. 12.4.10.4 (a) FSSB axis setting screen (10.4-inch)

12.4.10.5 Displaying and setting the servo setting screen

From the parameter setting support screen, the servo setting screen can be displayed. For details of the servo setting screen, see the description of the servo setting in Subsection III-12.4.4.

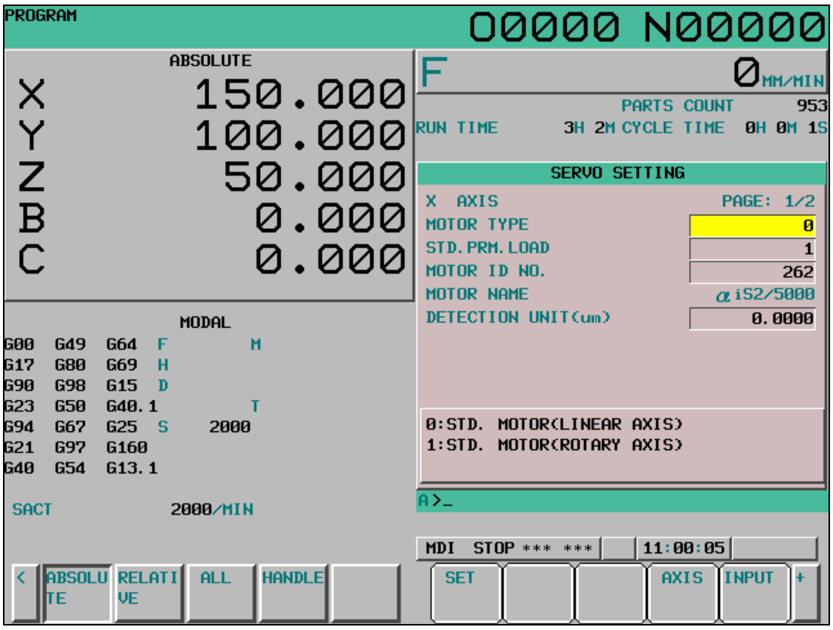


Fig. 12.4.10.5 (a) Servo setting screen (10.4-inch)

12.4.10.6 Displaying and setting the servo setting screen

Servo-related parameters can be displayed or changed.
The CNC parameters of servo current control, speed control, position control, and backlash acceleration can be displayed or changed.

Displaying the servo parameter screen

The servo parameter screen can be displayed from the parameter setting support screen. For the display procedure, see the parameter setting support screen (axis setting) described earlier.

Switching screen display

There are two types of screen display for the servo parameter screen.

1. Display for each axis
   The items to be set are displayed for each axis.
2. Display for each item
   Display is performed for each item to be set.
   (Data of all axes is displayed for each item.)

Screen display is switched between display for each axis and display for each item each time soft key [CHANGE] is pressed. To display soft key [CHANGE], follow the procedure below.

1. Display the servo parameter screen.
2. Press continuous menu key ▷ to display soft key [CHANGE].

Fig. 12.4.10.6 (a) Servo parameter screen (10.4-inch)

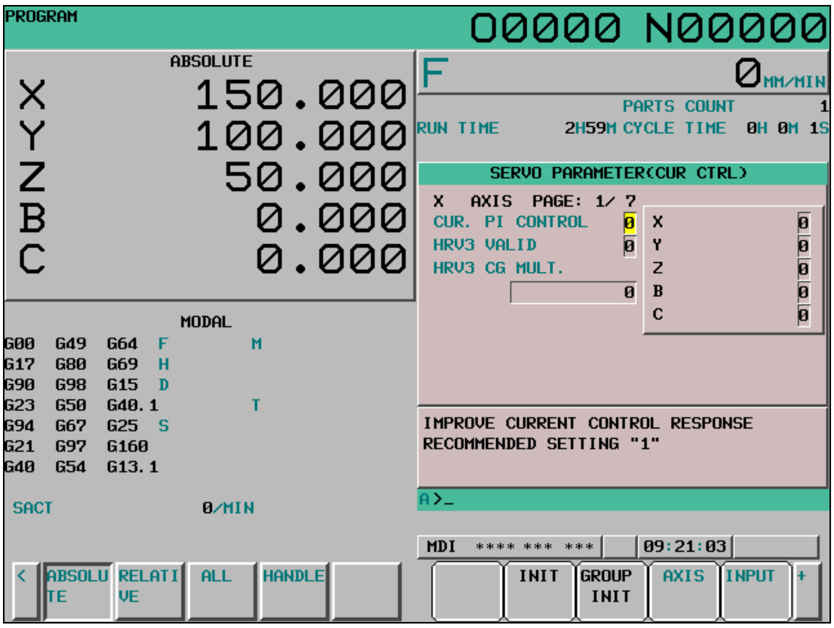


Fig. 12.4.10.6 (b) Servo parameter screen (display for each axis) (10.4-inch)

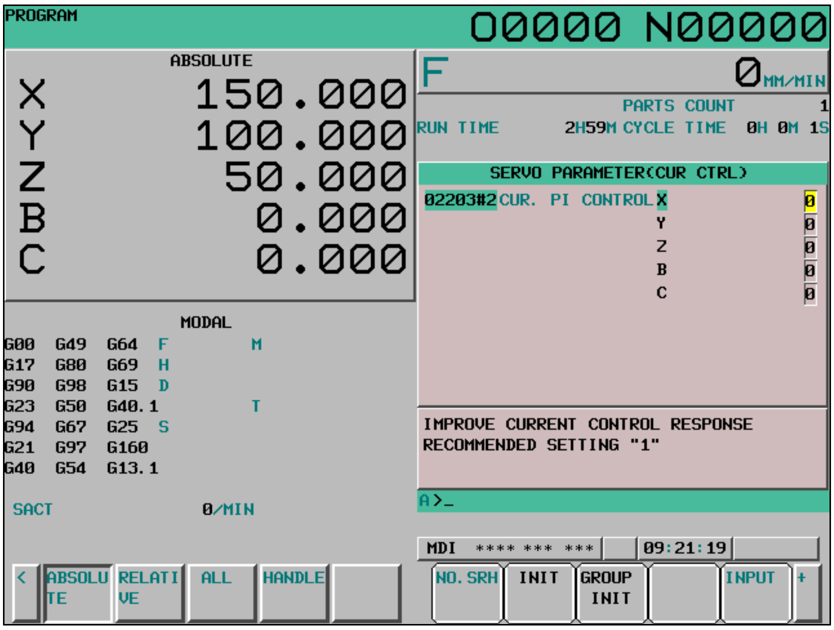


Fig. 12.4.10.6 (c) Servo parameter screen (display for each item) (10.4-inch)

Setting the servo parameter screen

Procedure

1. On the setting screen, confirm PARAMETER ENABLE SWITCH ON.
2. Press soft key [AXIS] several times to select the axis to be set.
3. Enter a value with numeric keys and press soft key [INPUT] or MDI key [INPUT].

The parameters can be initialized to their default values (FANUC recommended values). (For this procedure, see this subsection or "Parameter setting support screen (axis setting)".)

12.4.10.7 Displaying and setting the servo gain tuning screen

On the servo gain tuning screen, it is possible to adjust the following parameters automatically so that optimum velocity gain can be set easily according to the machine characteristics.

- Load inertia ratio (parameter No. 2021)
- Velocity gain multiplier during cutting (parameter No. 2107)
- Velocity gain multiplier during high speed HRV current control (parameter No. 2335)

Displaying and setting the servo gain tuning screen

The servo gain tuning screen can be displayed from the parameter setting support screen. For the display procedure, see "Parameter setting support screen (axis setting)" described earlier.

There are two types of screen display for the servo gain tuning screen.
On the automatic tuning screen, servo software calculates the optimum servo loop gain of all axes or a selected axis and automatically sets VELOCITY GAIN, CUT OVR, and H. SP HRV.
On the manual tuning screen, VELOCITY GAIN, CUT OVR, and H. SP HRV of the desired axis can be set directly with MDI keys.

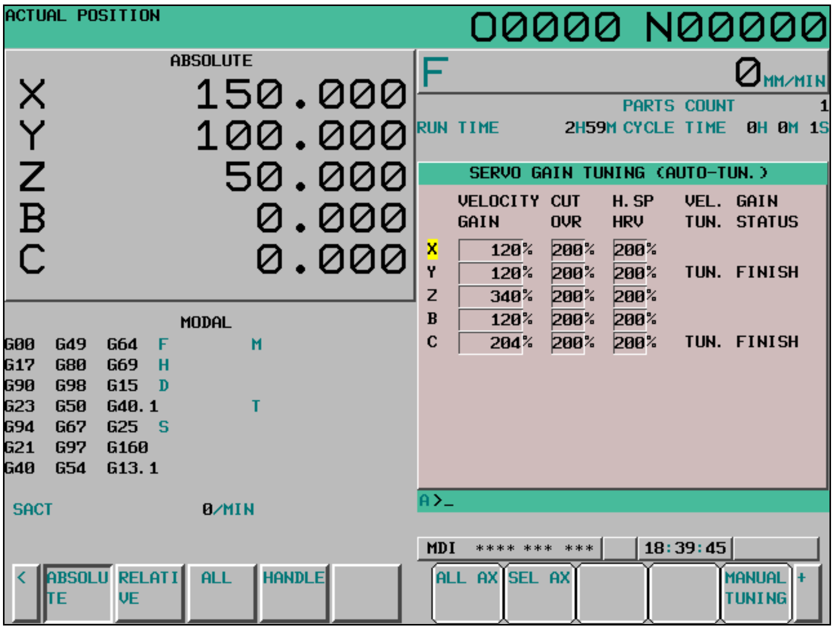


Fig. 12.4.10.7 (a) Servo gain tuning screen (automatic tuning screen) (10.4-inch)

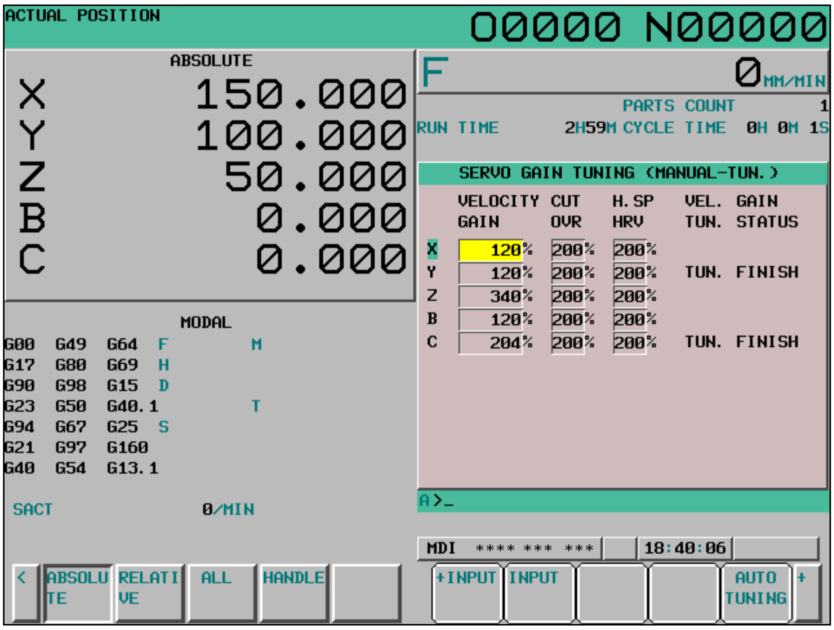


Fig. 12.4.10.7 (b) Servo gain tuning screen (manual tuning screen) (10.4-inch)

Display item

VELOCITY GAIN

The value calculated by the CNC with the following expression on the basis of parameter No. 2021 is displayed.
(Expression):

VELOCITY GAIN = (256 + No. 2021) / 256 × 100

CUT OVR

The setting of parameter No. 2107 is displayed.

H. SP HRV

The setting of parameter No. 2335 is displayed.

VEL. GAIN TUN. STATUS

The automatic tuning status is displayed. The automatic tuning status is indicated by one of the following four states: "tuning finished", which indicates that automatic tuning is completed, "tuning not finished", which indicates that automatic tuning is not completed, "tuning in progress", which indicates that automatic tuning is in progress, and "initial status error", which indicates that automatic tuning is not completed successfully.

For the axis for which automatic tuning is completed, TUN. FINISH is displayed under VEL. GAIN TUN. STATUS. For the axis for which automatic tuning is in progress, ON TUNING is displayed. For the axis for which automatic tuning is not completed successfully, INIT. ERR is displayed. For the axis for which automatic tuning is not completed, no message is displayed under VEL. GAIN TUN. STATUS.

Even when automatic tuning of an axis is completed, however, if automatic tuning is performed again, the previous message is cleared. Therefore, if automatic tuning is interrupted, the indication becomes blank.

CNC mode when the velocity gain is set

When setting the velocity gain on the servo gain tuning screen, set PARAMETER WRITE to 1 (enabled) on the setting screen and set the CNC in the MDI mode.

If the velocity gain is set when PARAMETER WRITE is 0, a warning saying "WRITE PROTECT" is issued. If the velocity gain is set in other than the MDI mode, a warning saying "WRONG MODE" is issued.

Servo gain automatic tuning screen

On the automatic tuning screen, servo software calculates the optimum servo loop gain of all axes or a selected axis and automatically sets VELOCITY GAIN, CUT OVR, and H. SP HRV. This process is called automatic tuning.

When soft key [ALL AX] is pressed on the automatic tuning screen, the servo loop gain of all axes is automatically set. When [SEL AX] is pressed, the servo loop gain of the axis selected by the cursor is automatically set. These processes are called all axis tuning and selected axis tuning, respectively.

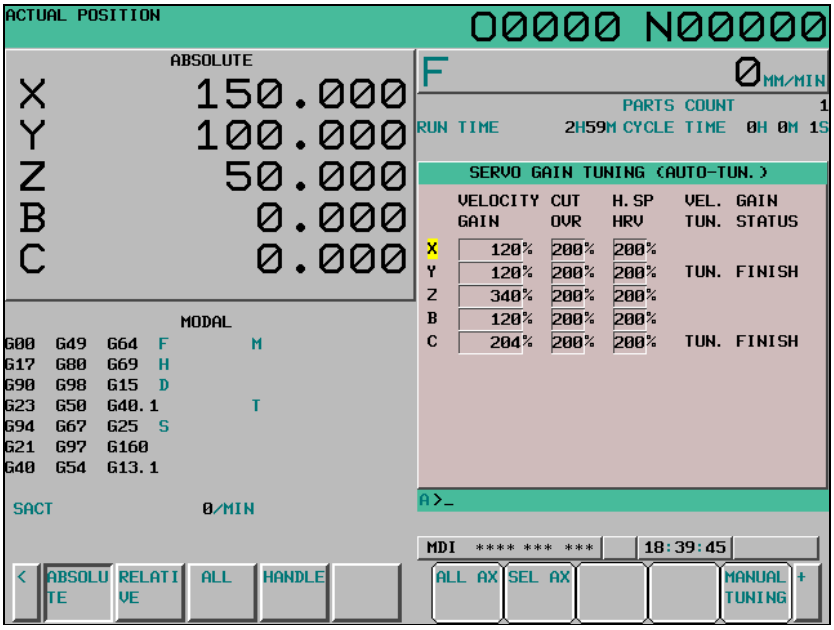


Fig. 12.4.10.7 (c) Servo gain tuning screen (all axis tuning) (10.4-inch)

- All axis tuning

When [ALL AX] is pressed on the automatic tuning screen, the following soft keys are displayed and SERVO GAIN TUNING (AUTO-TUN. ALL AXES) is indicated on the title bar of the screen.

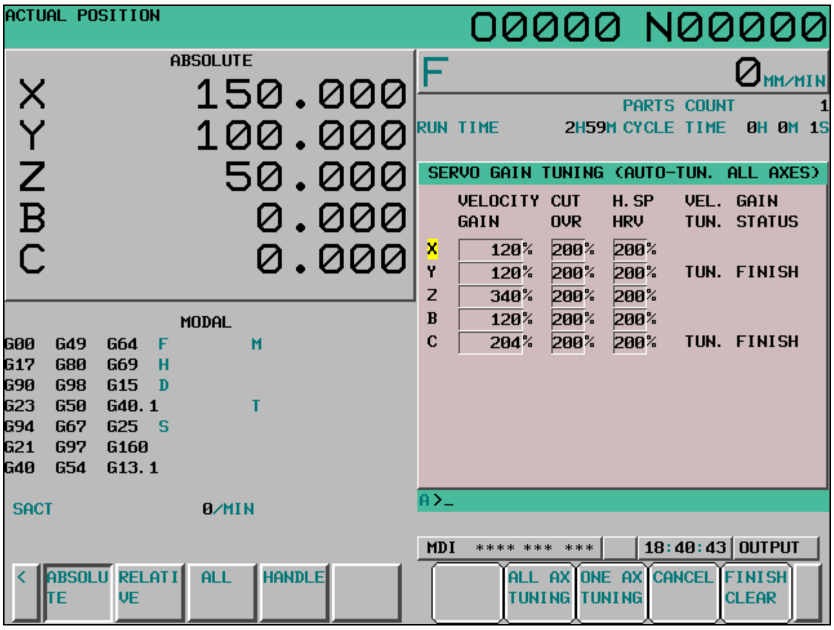


Fig. 12.4.10.7 (d) Automatic tuning screen (all axis tuning) (10.4-inch)

- All axis batch tuning

When no axis in a path is being subjected to automatic tuning, soft key [ALL AX TUNING] ([EXEC ALL] for the 8.4-inch display unit) is displayed.
When soft key [ALL AX TUNING] is pressed during all axis tuning, all axes are subjected to automatic tuning one by one, beginning with the first axis. However, the targets of automatic tuning are only the axes for which automatic tuning is not completed. For example, in the conditions shown in the above figure, automatic tuning starts with the X-axis and, when the X-axis is completed, proceeds to the Z-axis. The Y-axis and C-axis are not subjected to automatic tuning.
The cursor keys are disabled during all axis batch tuning. Upon completion of automatic tuning of all axes, the cursor returns to the first axis and the cursor keys are enabled.
If the screen is switched to another during all axis batch tuning, automatic tuning is canceled when automatic tuning of the current axis is completed. To start automatic tuning again, display this screen and press soft key [ALL AX TUNING].

NOTE

1. The axis for which TUN. FINISH or INIT. ERR is indicated under VEL. GAIN TUN. STATUS is not subjected to automatic tuning.
2. If the screen is switched to another during all axis batch tuning, all axis batch tuning is aborted.

- All axis step tuning

When no axis in a path is being subjected to automatic tuning, soft key [ONE AX TUNING] ([ONE EX] for the 8.4-inch display unit) is displayed.
When soft key [ONE AX TUNING] is pressed, as in all axis batch tuning, the axes for which automatic tuning is not completed are subjected to automatic tuning one by one, beginning with the first axis.
Each time an axis is subjected to automatic tuning, however, automatic tuning stops. At this time, the cursor automatically moves to the next axis for which automatic tuning is not completed.

For example, in the conditions shown in the above figure, automatic tuning starts with the X-axis and, when the X-axis is completed, the cursor moves to the Z-axis; then automatic tuning stops. If soft key [ONE AX TUNING] is pressed again at this time, the Z-axis is subjected to automatic tuning.
The cursor keys are disabled when any axis is being subjected to automatic tuning and, if automatic tuning is completed, the cursor keys are enabled. When automatic tuning of all axes is completed, the cursor returns to the first axis.

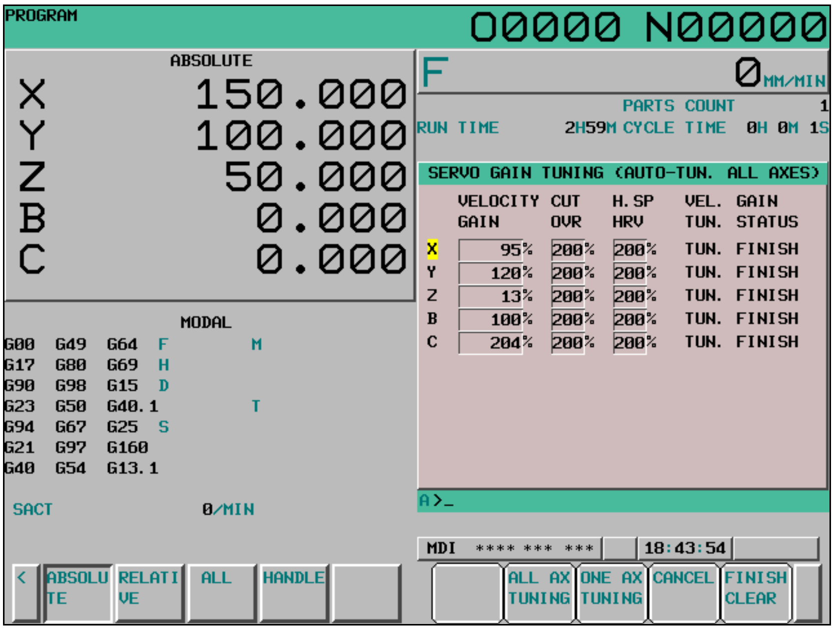
> **NOTE**
> Even when the cursor positions at other than the first axis, automatic tuning starts with the first axis.

- Aborting automatic tuning

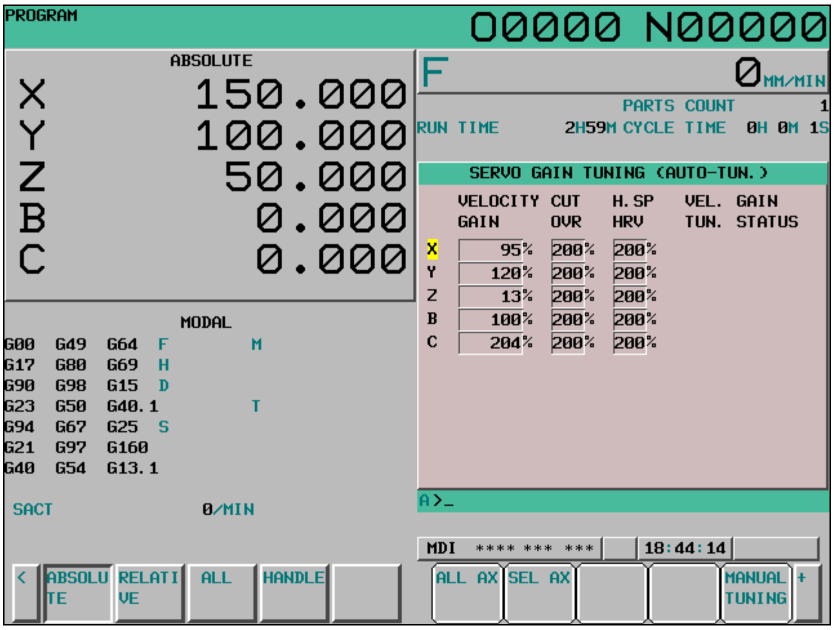
When no axis in a path is being subjected to automatic tuning, soft key [CANCEL] is displayed. When soft key [CANCEL] is pressed, the servo gain tuning screen (automatic tuning screen) is displayed again without automatic tuning being started.

- Clearing tuning completion

When no axis in a path is not being subjected to automatic tuning, soft key [FINISH CLEAR] ([FIN.CL] for the 8.4-inch display unit) is displayed.
When [FINISH CLEAR] is pressed during all axis tuning, the VEL. GAIN TUN. STATUS indication of the axes in the tuning completion state is cleared to the incompletion state (blank) as shown in the right half of the lower figure. However, the set VELOCITY GAIN, CUT DVR, and H. SP HRV are not cleared.



Cleared



> **NOTE**
>
> A clear operation in the tuning completion state does not clear the VEL. GAIN TUN. STATUS indication of the axis in the INIT. ERR state.
> To clear the INIT. ERR state, change the setting of VELOCITY GAIN, CUT DVR, or H. SP HRV on the manual tuning screen.

- Select axis tuning

When soft key [SEL AX] is pressed on the automatic tuning screen, the following soft keys are displayed and the SERVO GAIN TUNING (AUTO-TUN SEL AXES) is indicated on the title bar of the screen.

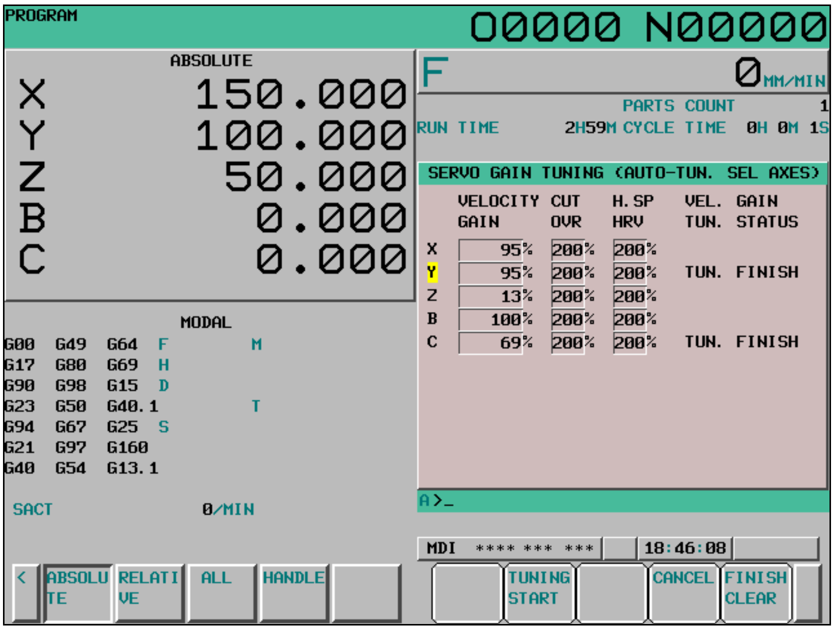


Fig. 12.4.10.7 (e) Automatic tuning screen (select axis tuning) (10.4-inch)

- Selected axis tuning

When no axis in a path is being subjected to automatic tuning, soft key [TUNING START] ([START] for the 8.4-inch display unit) is displayed.

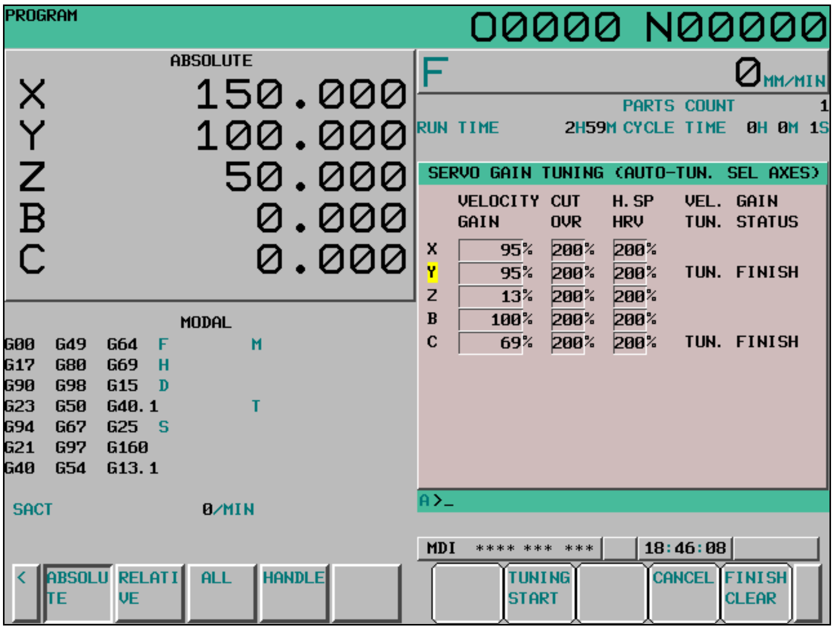
When soft key [TUNING START] is pressed during selected axis tuning, the axis selected by the cursor is subjected to automatic tuning. At this time, automatic tuning is performed again regardless of the tuning status of the selected axis. For example, in the state shown above, automatic tuning of the Y-axis was finished, but automatic tuning is performed again.
The cursor keys are disabled during selected axis tuning. Upon completion of automatic tuning, the cursor keys are enabled.

- Aborting automatic tuning

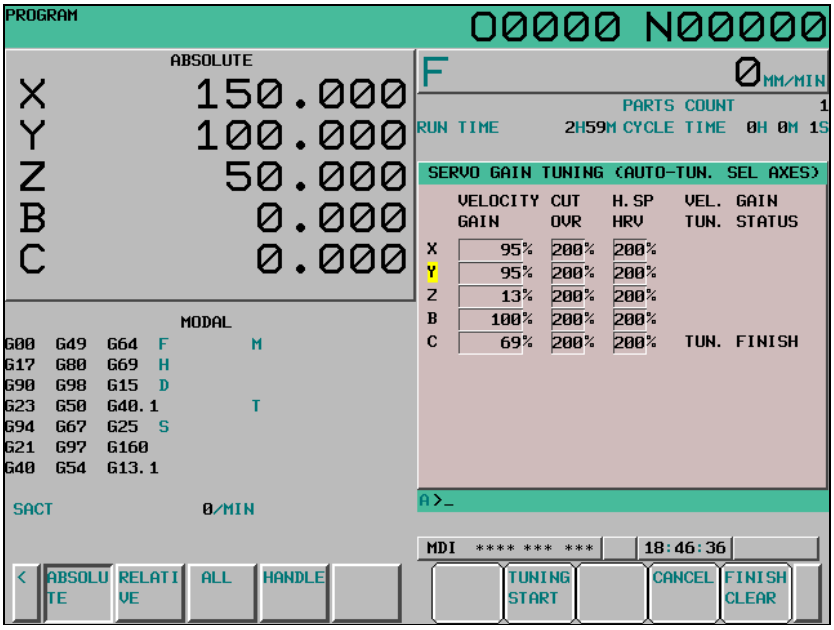
When no axis in a path is being subjected to automatic tuning, soft key [CANCEL] is displayed. When soft key [CANCEL] is pressed, the servo gain tuning screen (automatic tuning screen) is displayed again without automatic tuning being started.

- Clearing the automatic tuning status

When no axis in a path is not being subjected to automatic tuning, soft key [FINISH CLEAR] ([FIN.CL] for the 8.4-inch display unit) is displayed. When [FINISH CLEAR] is pressed during selected axis tuning, if the axis selected by the cursor is in the tuning completion state, the indication of VEL. GAIN TUN. STATUS is cleared to the incompletion state (blank). However, the set VELOCITY GAIN, CUT DVR, and H. SP HRV are not cleared.



Cleared



> **NOTE**
>
> A clear operation in the tuning completion state does not clear the VEL. GAIN TUN. STATUS indication of the axis in the INIT. ERR state.
> To clear the INIT. ERR state, change the setting of VELOCITY GAIN, CUT DVR, or H. SP HRV on the manual tuning screen.

- Forcibly stopping automatic tuning

When one axis in a path is being subjected to automatic tuning, the following screen and soft key [TUNING STOP] ([STOP] for the 8.4-inch display unit) is displayed.

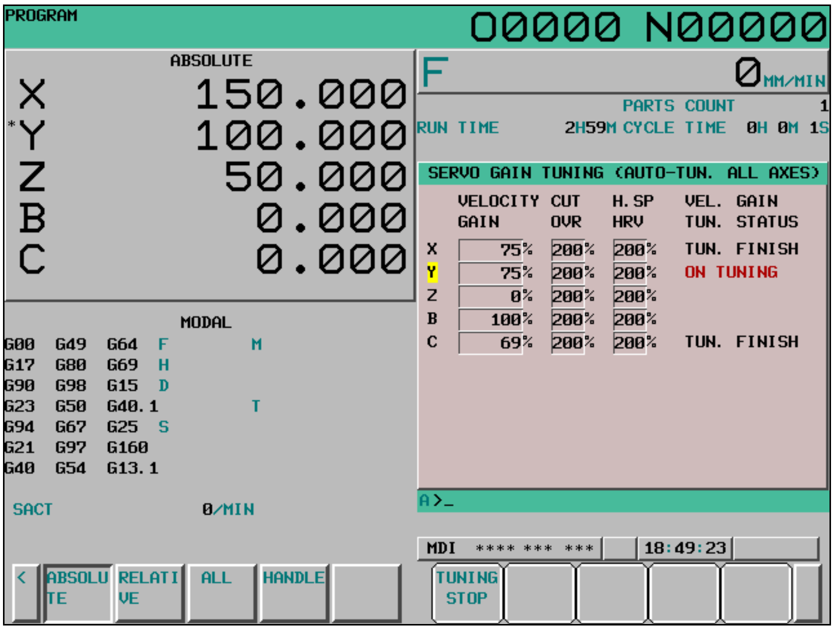


Fig. 12.4.10.7 (f) All axes being tuned (10.4-inch)

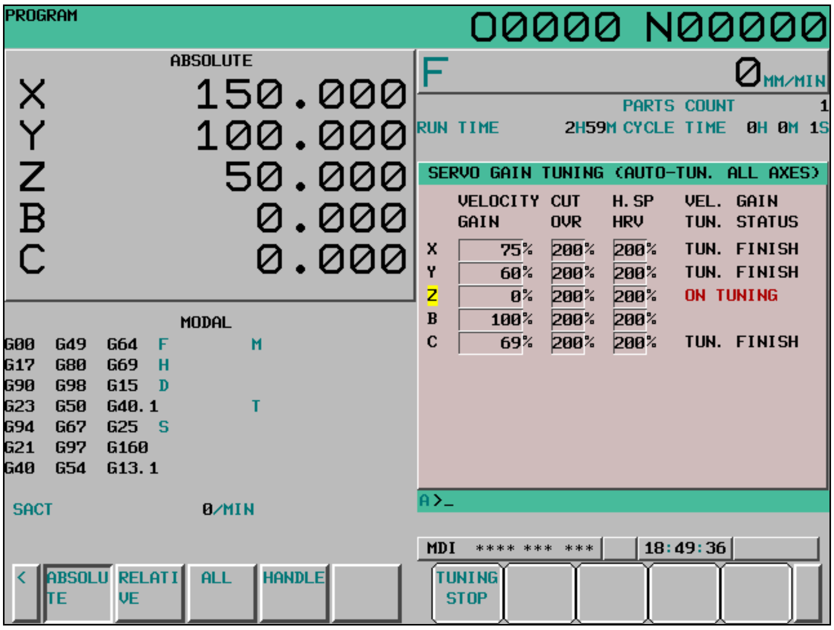


Fig. 12.4.10.7 (g) Selected axis being tuned (10.4-inch)

When soft key [TUNING STOP] is pressed at this time, automatic tuning is forcibly stopped even during automatic tuning. It is also possible to forcibly stop automatic tuning by pressing the RESET key or set the CNC in the emergency stop state.
The VELOCITY GAIN, CUT DVR, and H. SP HRV settings of the axis for which automatic tuning was stopped midway are returned to the settings before automatic tuning. For all axis tuning, the VELOCITY GAIN, CUT DVR, and H. SP HRV settings of the axis for which automatic tuning was completed remain at the values set during automatic tuning.
However, automatic tuning is stopped midway, so the indication of VEL. GAIN TUN. STATUS becomes blank (incompletion).

NOTE
Automatic tuning can be stopped regardless of the cursor position.

- Servo gain manual tuning screen

On the manual tuning screen, it is possible to directly enter a value in VELOCITY GAIN, CUT DVR, and H. SP HRV of an arbitrary axis with MDI keys. This operation is called manual tuning.

- Displaying the manual tuning screen

When soft key [MANUAL TUNING] ([MANUAL] for the 8.4-inch display unit) is pressed on the automatic tuning screen, the manual tuning screen is displayed.

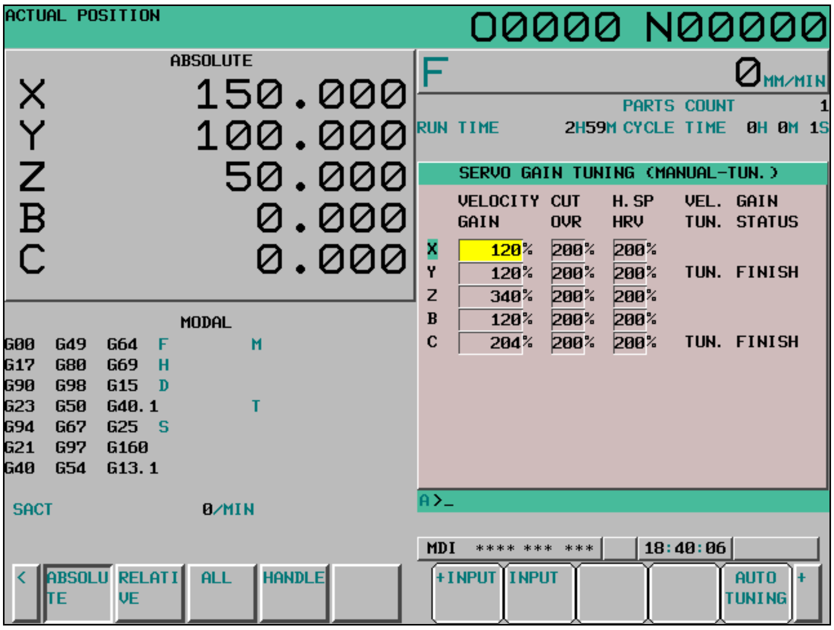


Fig. 12.4.10.7 (h) Servo gain tuning screen (manual tuning screen) (10.4-inch)

> **NOTE**
> If any axis is being subjected to automatic tuning, [MANUAL TUNING] is not displayed on the automatic tuning screen. Accordingly, the manual tuning screen cannot be displayed during automatic tuning.

- Setting parameters

Move the cursor to the desired item of the desired axis and enter a value directly with MDI keys.
When soft key [+INPUT], soft key [INPUT], or MDI key [INPUT] is pressed, the value is set.

Soft key [+ INPUT]

The value already set plus the value entered by MDI keys are set.
Example) When the current setting is 500 and the value entered by MDI keys is 50
The value to be entered is 550.

MDI key [INPUT], soft key [INPUT]

The value entered by the MDI key is set as is.

> **NOTE**
> During input with soft key [INPUT], if a value preceded by the + symbol such as "+1" is input, a warning saying "FORMAT ERROR" is issued.
> If a value falling outside the set range is input, the value is changed so as to fall within the range.

- Automatic tuning state

TUN. FINISH is indicated under VEL. GAIN TUN. STATUS for the axis that was subjected to automatic tuning and INIT. ERR is indicated for the axis for which the initial status is illegal. If any of VELOCITY GAIN, CUT OVR, and H.SP HRV of the axis that was subjected to automatic tuning or the axis for which the initial status is illegal is set to a value, the VEL. GAIN TUN. STATUS indication is cleared to become blank (incompletion state).

- Switching to the automatic tuning screen

To return to the automatic tuning screen from the manual tuning screen, press soft key [AUTO TUNING] ([AUTO] for the 8.4-inch screen) on the manual tuning screen. The automatic tuning screen appears.

- Caution

Reset

If the RESET key is pressed or external reset signal ERS<Gn008.7> or reset & rewind signal RRW<Gn008.6> is input during automatic tuning, automatic tuning is stopped.

VEL. GAIN TUN. STATUS "INIT. ERR"

When INIT. ERR is indicated under VEL. GAIN TUN. STATUS, automatic tuning cannot be performed because the setting of VELOCITY GAIN is illegal.
Change the setting of VELOCITY GAIN.

Changing parameters

If the setting of VELOCITY GAIN, CUT OVR, or H.SP HRV of the axis being subjected to automatic tuning is changed on the parameter screen or by an external application, automatic tuning is stopped. At this time, automatic tuning is incomplete and VEL. GAIN TUN. STATUS becomes blank.

Re-execution of automatic tuning

To re-execute automatic tuning of the axis for which TUN. FINISH or INIT. ERR is indicated in all axis tuning, clear the indication of VEL. GAIN TUN. STATUS.
If selected axis tuning is used, automatic tuning of an axis that was subjected to automatic tuning can be re-executed without the indication of VEL. GAIN TUN. STATUS being cleared.

MDI operation and automatic tuning

Do not perform automatic tuning during MDI operation. Otherwise, a warning saying "CNC RUNNING" is issued.
Similarly, do not perform MDI operation during automatic tuning. Otherwise, an alarm DS2005 "NOW GAIN TUNING" is issued.

Automatic tuning between paths

Automatic tuning cannot be performed on multiple paths at the same time. If an attempt is made to perform automatic tuning on one path while automatic tuning is performed on another path, a warning saying "NOW TUNING OTHER AXIS" is issued. At this time, automatic tuning of the path on which automatic tuning is performed continues (not stopped).

Servo alarm

If the positional deviation exceeds the positional deviation limit, servo alarm SV0411 is issued. Set the parameter No. 1828 (positional deviation limit during movement) again.

12.4.10.8 Displaying and setting the high-precision setting screen

The CNC parameters of servo time constants and automatic acceleration/deceleration can be displayed and changed.

Displaying the high-precision setting screen

The servo parameter screen can be displayed from the parameter setting support screen. (For the display procedure, see "Parameter setting support screen (axis setting)" earlier.)

Switching screen display

There are the following two display methods for high-precision setting.

1. Display for each axis
   The items to be set are displayed for each axis.
2. Display for each item
   Display is performed for each item to be set. (Data of all axes is displayed for each item.)

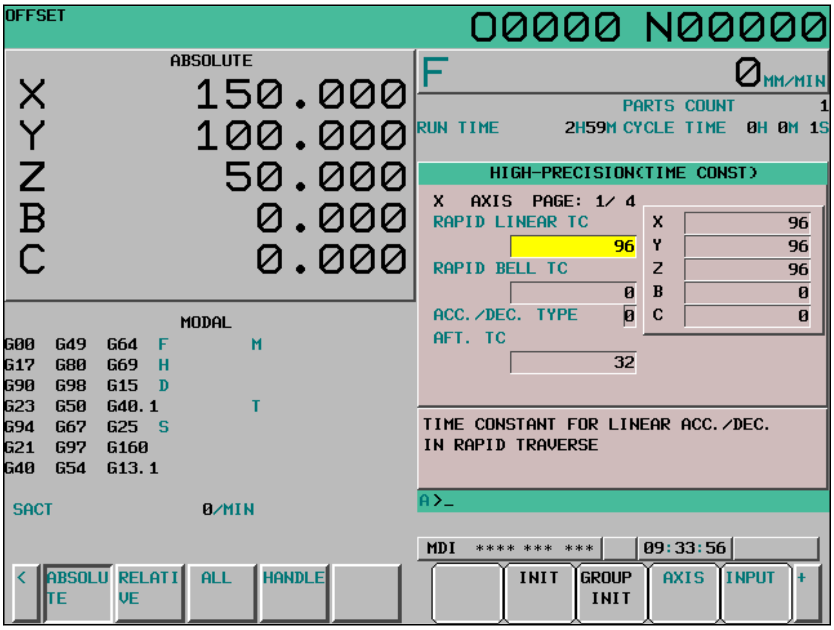


Fig. 12.4.10.8 (a) High-precision setting screen (display for each axis) (10.4-inch)

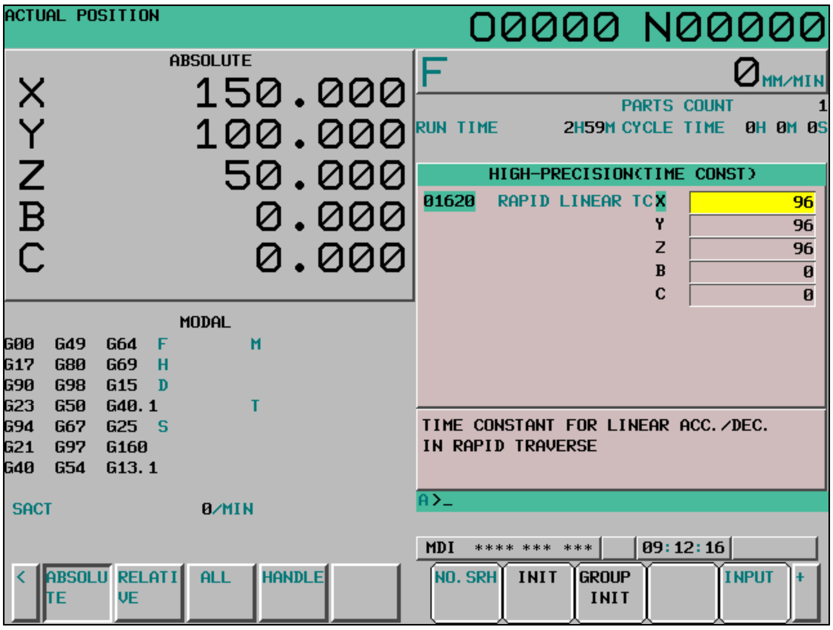


Fig. 12.4.10.8 (b) High-precision setting screen (display for each item) (10.4-inch)

Screen display is switched between display for each axis and display for each item each time soft key [CHANGE] is pressed. (For the switching method, see "Parameter setting support screen (servo parameter)" earlier.)

Setting the high-precision setting screen

For the setting procedure, see "Parameter setting support screen (servo parameter)" earlier.

12.4.10.9 Displaying and setting the spindle setting screen

The parameters related to spindles can be displayed or changed.
For the display and setting methods, see "Parameter setting support screen (axis setting)" earlier.

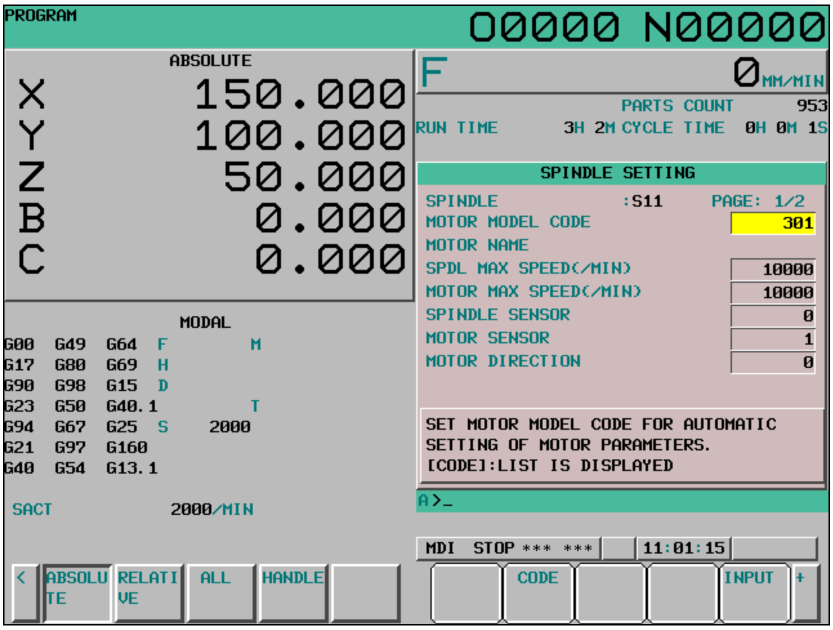


Fig. 12.4.10.9 (a) Parameter setting support screen (spindle setting) (10.4-inch)

12.4.10.10 Displaying and setting the miscellaneous setting screen

Allocation of axis paths and setting of DI/DO and serial spindles can be performed. It is also possible to initialize parameters to the default settings (FANUC recommended values).
For the display and setting procedures, see "Parameter setting support screen (axis setting)" earlier.

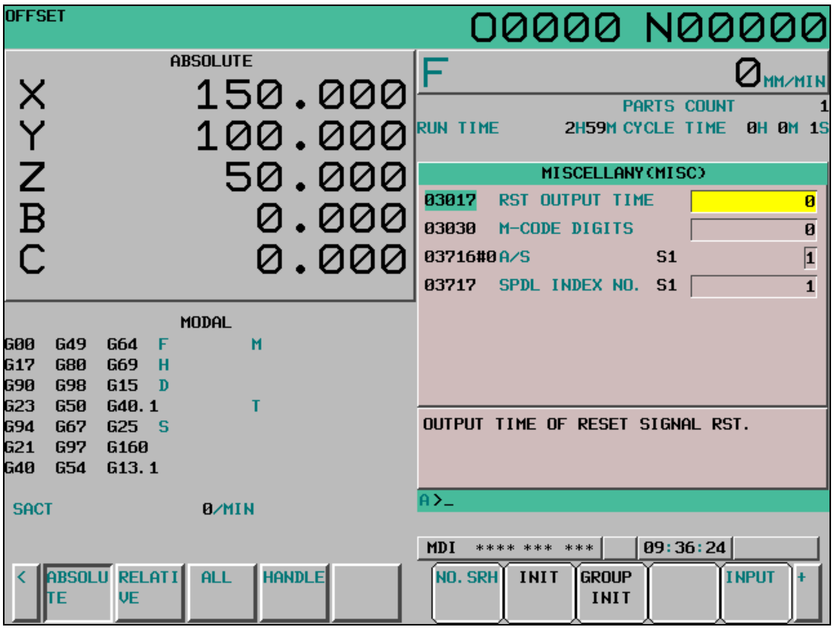


Fig. 12.4.10.10 (a) Parameter setting support screen (miscellaneous) (10.4-inch)

12.4.10.11 Displaying and setting the servo tuning screen

From the parameter setting support screen, the servo tuning screen can be displayed. For details of the servo tuning screen, see the description of the servo tuning screen in Subsection III-12.4.4.

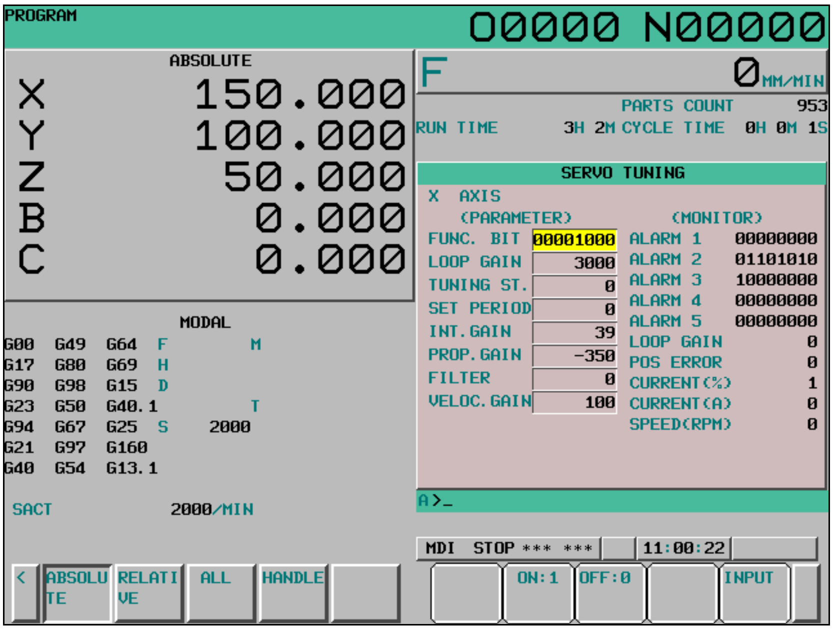


Fig. 12.4.10.11 (a) Servo tuning screen (10.4-inch)

12.4.10.12 Displaying and setting the spindle tuning screen

From the parameter setting support screen, the spindle tuning screen can be displayed. For details of the spindle tuning screen, see the description of the spindle tuning screen in Subsection III-12.4.6.

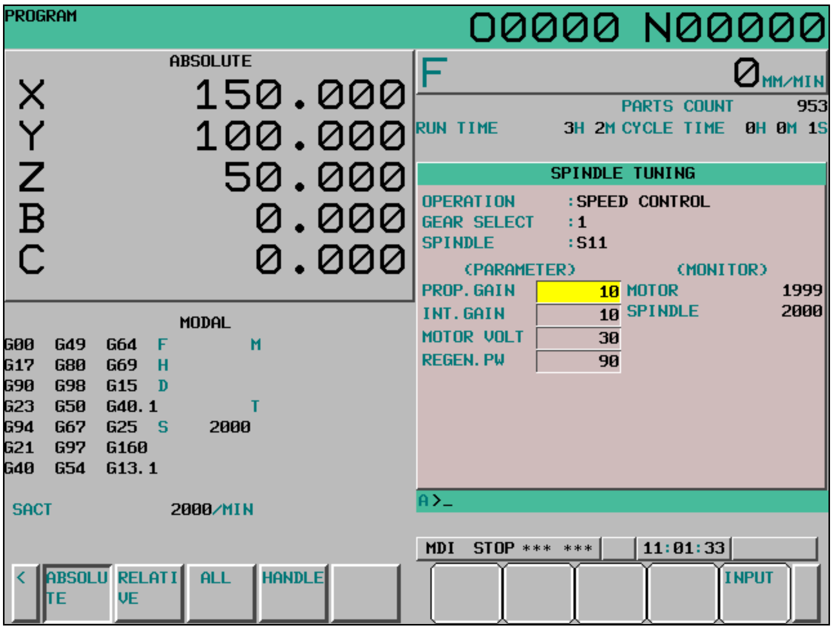


Fig. 12.4.10.12 (a) Spindle tuning screen (10.4-inch)

12.4.10.13 Displaying and setting the machining parameter tuning screen

From the parameter setting support screen, the machining parameter tuning screen can be displayed. For details of the machining parameter tuning screen, see the description of the machining parameter tuning screen in Subsection III-12.4.9.

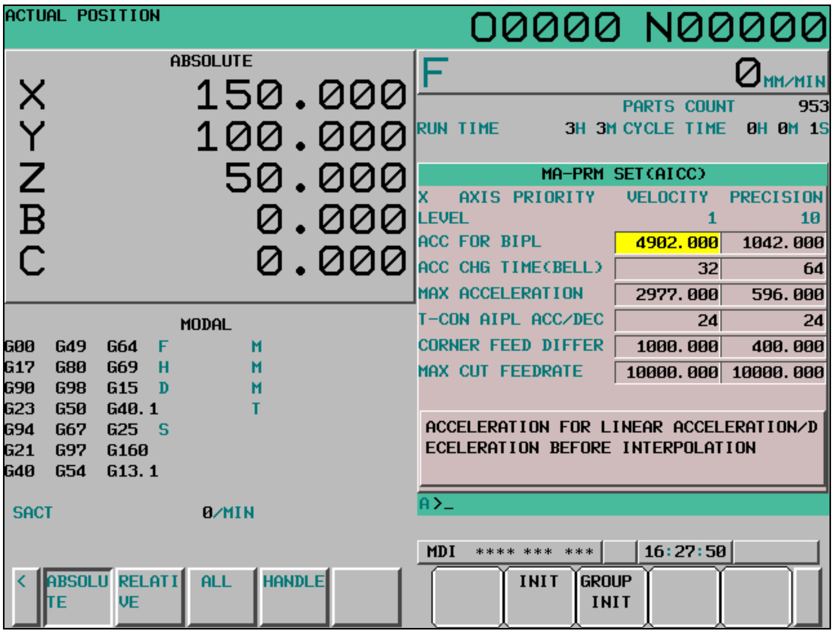


Fig. 12.4.10.13 (a) Machining parameter tuning screen (10.4-inch)

Explanation

- Parameters displayed for parameter setting support

Table 12.4.10.13 (a) Parameters displayed for parameter setting support (1)

Menu	Group	Parameter No.	Name	Brief description
SPINDLE SETTING	SPINDLE SETTING	3741		Maximum spindle speed
		4000#0		Motor rotation direction
		4001#4		Position coder rotation direction
		4002#3,2,1,0		Spindle sensor switch
		4004#3,2		Proximity switch
		4005#0		Speed feedback method
		4006#1		Gear ratio resolution
		4010#2,1,0		Motor sensor type
		4019#7		The parameters of the serial spindle are: 0: Set automatically. 1: Not set automatically. (Not necessary for the analog spindle.)
		4020		Maximum motor rotation speed (rpm)
		4056		Motor rotation count per one revolution of spindle Maximum motor speed/maximum spindle speed × 100 (rounding off)
		4133		Motor model code of the serial spindle (Not necessary for the analog spindle.)
		4171		Number of gears on the spindle side
		4172		Number of gears on the motor side
		4334		Speed detector arbitrary pulse count

Table 12.4.10.13 (b) Parameters displayed for parameter setting support (2)

Menu	Group	Parameter No.	Name	Brief description
AXIS SETTING	BASIC	1001#0	INM	Least command increment on linear axes: 0:Metric (millimeter machine) 1:Inch (inch machine)
		1005#0	ZRNx	When an automatic operation (other than G28) is executed before reference position return: 0:Alarm is issued (PS0224) 1:No alarm is issued.
		1005#1	DLZx	Reference position return without dogs: 0:Disabled 1:Enabled
		1006#0	ROTx	Setting of linear axes or rotary axis: 0:Linear axis 1:Rotation axis
		1006#3	DIAx	Setting of the amount of travel: 0:Radius specification 1: Diameter specification
		1006#5	ZMIx	Reference position return direction: 0:Plus direction 1:Minus direction
		1008#0	ROAx	Rotation axis roll-over function: 0:Disabled 1:Enabled
		1008#2	RRLx	With the amount of travel per revolution, relative coordinates are: 0:Not rounded 1:Rounded
		1013#1	ISCx	Sets the least input increment and least command increment: 0:IS-B 1:IS-C
		1020		Program name
		1022		Sets each axis in the basic coordinate system.
		1023		Servo axis number
		1815#1	OPTx	A separate pulse coder is: 0:Not used 1:Used
		1815#4	APZx	The correspondence between machine positions and absolute-position detector positions is: 0:Not established 1:Established
		1815#5	APCx	The position detector used is: 0:Other than an absolute-position detector 1:Absolute-position detector
		1825		Servo loop gain
		1826		In-position width
		1828		Positional deviation limit during travel
		1829		Positional deviation limit during stop

Table 12.4.10.13 (c) Parameters displayed for parameter setting support (3)

Menu	Group	Parameter No.	Name	Brief description
AXIS SETTING	SPINDLE	3716#0	A/S	Sets the type of spindle motor: 0:Analaog / 1:Serial.
		3717		Spindle amplifier number
	COORDI-NATE	1240		Machine coordinate of the first reference position
		1241		Machine coordinate of the second reference position
		1260		Amount of travel per revolution of a rotary axis
		1320		Positive direction border coordinates of stored stroke check 1
		1321		Negative direction border coordinates of stored stroke check 1
	FEED RATE	1401#6	RDR	For a rapid traverse command, dry run is: 0:Disabled 1:Enabled
		1410		Dry run feedrate
		1420		Rapid traverse rate
		1421		Rapid traverse override F0 rate
		1423		Jog feedrate
		1424		Manual rapid traverse rate
		1425		FL feedrate for reference position return
		1428		Reference position return feedrate
		1430		Maximum cutting feedrate
	ACC./DEC.	1610#0	CTL	Acceleration/deceleration for cutting feed is: 0:Exponential acceleration/deceleration 1:Linear acceleration/deceleration after interpolation
		1610#4	JGL	Acceleration/deceleration for jog feed is: 0:Exponential acceleration/deceleration 1:Same as acceleration/deceleration for cutting feed (The settings of bit 1 (CTBx) and bit 0 (CTLx) of parameter No. 1610 are followed.)
		1620		Time constant for linear acceleration/deceleration for rapid traverse
		1622		Time constant for acceleration/deceleration for cutting feed
		1623		FL feedrate for acceleration/deceleration after interpolation for cutting feed
		1624		Time constant for acceleration/deceleration for jog feed
		1625		FL feedrate for exponential acceleration/deceleration for jog feed
MISCELLA-NY	MISC	981		Sets the path of each axis.
		982		Sets the path of each spindle.
		3017		Output time of reset signal
		3030		Allowable number of digits of an M code
		3716#0	A/Ss	Sets the type of spindle motor: 0:Analaog / 1:Serial.
		3717		Spindle amplifier number

12.4.11 Periodic Maintenance Screen

Periodic maintenance screens are used for managing consumables (such as the backlight of a LCD unit and backup batteries). By setting the name of a consumable, its life time, and the method for counting down the life time, the life of the consumable can be counted down in the appropriate method set for the consumable, and the remaining life time can be displayed.
With these screens, the user can easily manage consumables that require periodic replacement.

Explanation

There are four periodic maintenance screens: the status screen, the setting screen, the machine menu screen, and the NC menu screen.

- Status screen:
  Item names, remaining times, and count statuses are displayed, and item names are set.
- Setting screen:
  Life times, remaining times, and count types (count-down method) are set.
- Machine menu screen:
  The names of consumables in the machine can be registered.
- NC menu screen:
  The names of consumables in the NC are already registered.

Using periodic maintenance screens

<1> Referencing a periodic maintenance screen
Display the status screen. For the meanings of items on the status screen, see **Status screen**.

<2> Adding the name of a new consumable item to a periodic maintenance screen or editing an existing consumable item on the screen
The name of a consumable item can be added or edited on the machine menu screen. For details, see **Machine menu screen**.

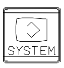
<3> Adding or editing a life time, remaining time, and a method of counting down the remaining time for a consumable on a periodic maintenance screen
A life time and remaining time can be added or edited on the setting screen. For details, see **Setting screen**.

<4> Newly displaying the item name and remaining time of a consumable on a periodic maintenance screen.

1 Setting an item name
Select the item name of a consumable to be displayed from the machine menu screen or NC menu screen, or enter the name using the MDI keys. For the procedure, see **Item name** in **Status screen**.

2 Setting a life time, remaining time, and count type
Select the life time, remaining time, and count type of a consumable to be displayed from the setting screen. For the procedure, see **Remaining time** in **Status screen**.

Procedure for displaying a periodic maintenance screen

1 Press function key .

2 Press the continuous menu key  several times to display [PERIOD MAINTE] ([MAINTE] for the 8.4-inch display unit).

3 Press soft key [PERIOD MAINTE] to display the periodic maintenance screen.

Status screen

When soft key [STATUS] is pressed, the status screen is displayed. The status screen shows the item names, count statuses, and remaining times of managed consumables.

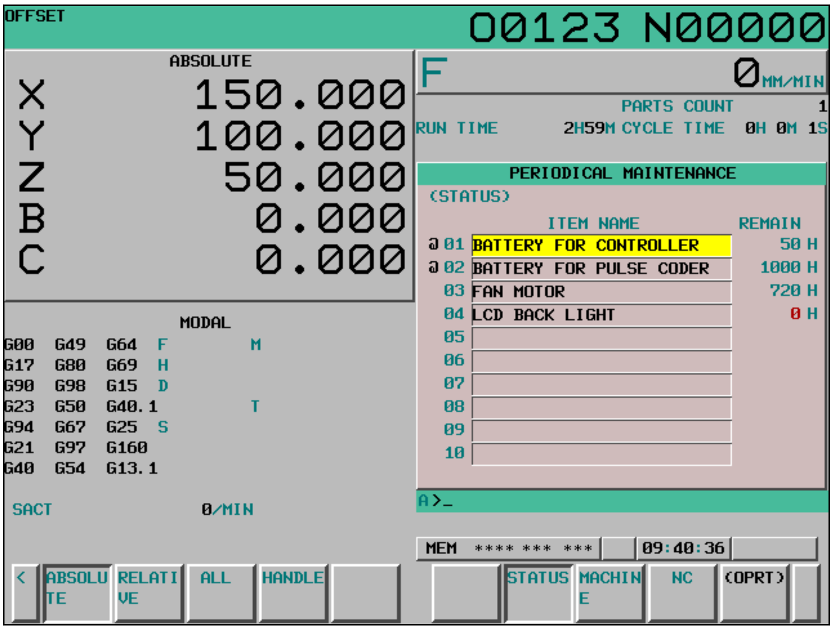


Fig. 12.4.11 (a) Status screen (10.4-inch)

- Item name

As the item name, set the name of a consumable to be managed by periodic maintenance. To set an item name, select a name from the machine menu screen or NC menu screen, or directly enter the name using the MDI keys.

Setting an item name from a menu screen

1 On the status screen, move the cursor to a target item name, press soft key [(OPRT)], then press soft key [ENTRY].
2 Press soft key [MACHINE] or [NC] to display the machine menu screen or NC menu screen.
3 Move the cursor to an existing item name on the menu screen, press soft key [(OPRT)], and press soft key [SELECT] then [EXEC] successively.
4 The screen display returns to the status screen, and the item name selected on the menu screen is added to the status screen.

Initially, there is no item name set on the machine menu screen, so item names must be registered in advance. For the registration method, see the description of the procedure for registering item names on the machine menu screen.

Setting an item name using the MDI keys

1 Press soft key [(OPRT)].
2 Type the alphanumeric characters to be input, then press soft key [INPUT].
3 The entered item name is registered on the status screen.

When soft key [+INPUT] is pressed instead of soft key [INPUT], the entered characters can be added to an existing item name.
When typing 2-byte characters, type "*" before and after the character codes. The character codes must conform to FANUC codes. (See Appendix G, "FANUC 2-BYTE CHARACTER CODE TABLE".)
An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.
Example: To register "LCD バックライト", enter the following:

>LCD＊110E10F410CC114010B610FE＊_

NOTE

1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning "DATA OUT OF RANGE" may be output.

Deleting an item name

To delete a registered item name, move the cursor to the item name, press soft key [ERASE], then press soft key [EXEC].
When an item name is deleted, its life time, remaining time, and count type are deleted at the same time.

- Remaining time

As the remaining time, the period of time left until the time for replacement is reached by count down operation is displayed.
If the percentage of the remaining time to the life time has reached to the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.

Setting the remaining time

1 On the status screen, place the cursor on an item for which the remaining time is to be set (the item name must have been set in advance).
2 Press soft key [(OPRT)], then press soft key [CHANGE].
3 The screen display changes to the setting screen.
4 Set the life time, remaining time, and count type. For the setting method and other information, see **Setting screen**.

NOTE

On the status screen, the remaining time and life time cannot be set.
These items must be set on the setting screen.

- Count status

The count status is indicated on the left side of the item number as follows:

Indication	Count status
Blank	Counting stopped
@	Counting in progress
*	Life expired

Setting screen

On the setting screen, the life time, remaining time, and count type of a managed consumable are set.

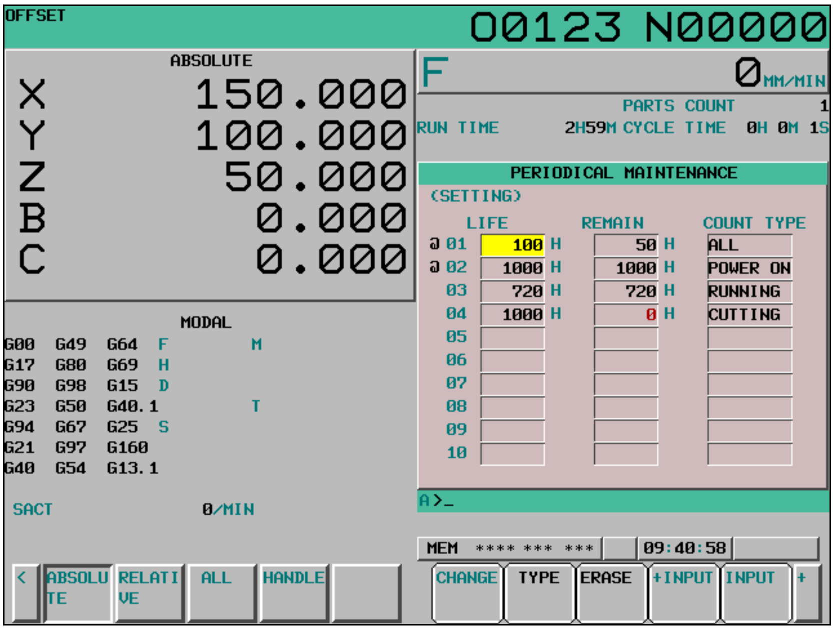


Fig. 12.4.11 (b) Setting screen

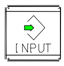
Display procedure

1 When the status screen is displayed, press soft key [(OPRT)].

2 Press soft key [CHANGE].

- Life time

Set the life time of a consumable.

Move the cursor to an existing item, type a life time, then press soft key [INPUT] (or the  key). The life time is then set, and the same value is set also as the remaining time.

At this time, "------" is indicated in the count type field.

When soft key [+INPUT] is pressed, the entered value can be added to the life time already set. The same value as the added value is added also to the remaining time.

A value ranging from 0 to 65535 (in hours) can be set.

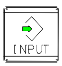
> **NOTE**
> 1 If a setting operation is attempted when the item name is not registered, the warning "EDIT REJECTED" is issued.
> 2 If a value exceeding the valid range is entered, the warning "DATA IS OUT OF RANGE" is issued.
> 3 If soft key [ERASE] or [TYPE] is pressed, the warning "EDIT REJECTED" is issued.

- Remaining time

The period of time left until the time for replacement is reached by count down operation is indicated.

If the percentage of the remaining time to the life time has reached the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.

Even after the life time has expired, count down operation continues.

Move the cursor to the remaining time of a target registered number, type a remaining time, then press soft key [INPUT] (or the  key). The remaining time is then set.

When soft key [+INPUT] is pressed, the entered value can be added to the remaining time already set.

A value ranging from 0 to (life time) can be set.

When soft key [ERASE] then soft key [EXEC] are pressed, the same value as the life time is set.

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 When a value exceeding the valid range is entered, the warning “DATA IS OUT OF RANGE” is issued.
3 If soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.

- Count type

As the count type, select the way of counting.
Place the cursor on the count type of a target registration number, then press soft key [TYPE]. Count types are displayed as soft keys as shown below. Select one of these soft keys, then press soft key [EXEC].

Soft key	Meaning	Indication
[NO CNT]	Counting is not performed (stopped)	——————
[ALL]	Counting is performed at all times	All times
[POWER ON]	Counting is performed when power is on.	When power is on
[RUN]	Counting is performed when operation is in progress.	When operation is in progress
[CUT]	Counting is performed when cutting is being performed.	When cutting is being performed

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 Soft keys [INPUT] and [+INPUT] have no effect.
3 When counting is performed at all times, a 24-hour error is generated in leap year.
4 If soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued.

Machine menu screen

On the machine menu screen, the names of consumables of the machine are registered. From this screen, item names can be added to the status screen. For the method of addition to the status screen, see the description of the status screen.

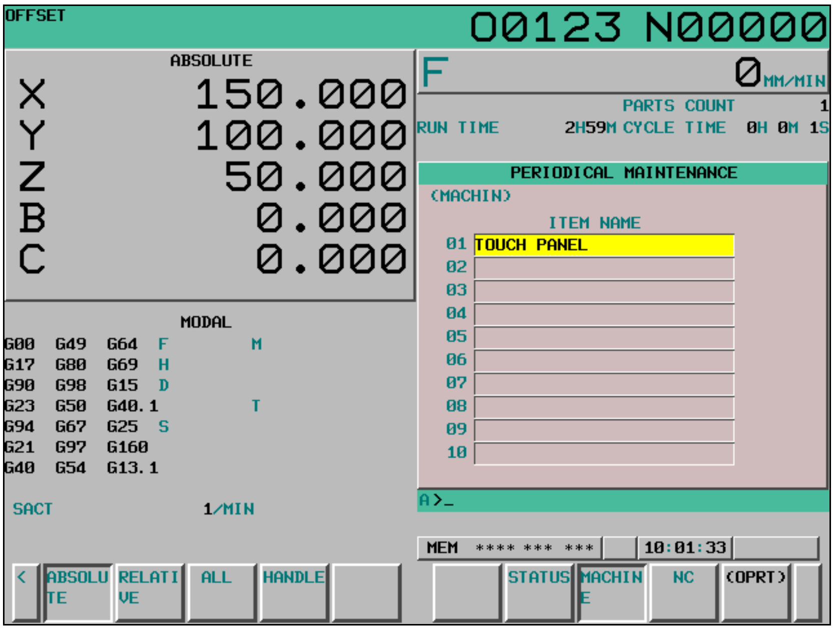


Fig. 12.4.11 (c) Machine menu screen

- Displaying the screen

1 When the status screen is displayed, press soft key [MACHINE].

On the machine menu screen, item names can be registered using one of the following two methods:

- Registration from a program
- Registration using the MDI keys

- Registration from a program

By executing a program having the following format, an item name can be registered in the menu of the machine:

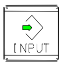
Format

G10 L61 Px [n]

X Registration number

n Item name, format: [alphanumeric-characters*2-byte-characters*alphanumeric-characters]

- Registration using the MDI keys

When the cursor is moved on the machine menu screen, an item name is entered in the following format, and soft key [INPUT] (or the  key) is pressed, then the item name can be registered on the machine menu screen.

When soft key [+INPUT] is pressed, the typed characters can be added to an already registered item name.

Format

Alphanumeric-characters*2-byte-characters*alphanumeric-characters

Description of the format

Alpha numeric characters must be entered directly.
Two-byte codes must conform to FANUC codes. (See Appendix G, "FANUC 2-BYTE CHARACTER CODE TABLE".)
To register a two-byte character, enclose the two-byte code with asterisks (*). An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.
Example) To register "LCD バックライト", enter the following:

Registration with a program
G10 L61 P1 [LCD＊110E10F410CC114010B610FE＊]
Registration with MDI keys
>LCD＊110E10F410CC114010B610FE＊_

NOTE

1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning “DATA IS OUT OF RANGE” may be output.
3 When a blank item name is selected on the machine screen, the warning “EDIT REJECTED” is issued.

To delete a registered item name, move the cursor to the item name, press soft key [ERASE], then press soft key [EXEC].

NC menu screen

On the NC menu screen, the names of NC consumables are registered. From this screen, an item name can be registered on the status screen. For the method of registration to the status screen, see the description of the status screen.

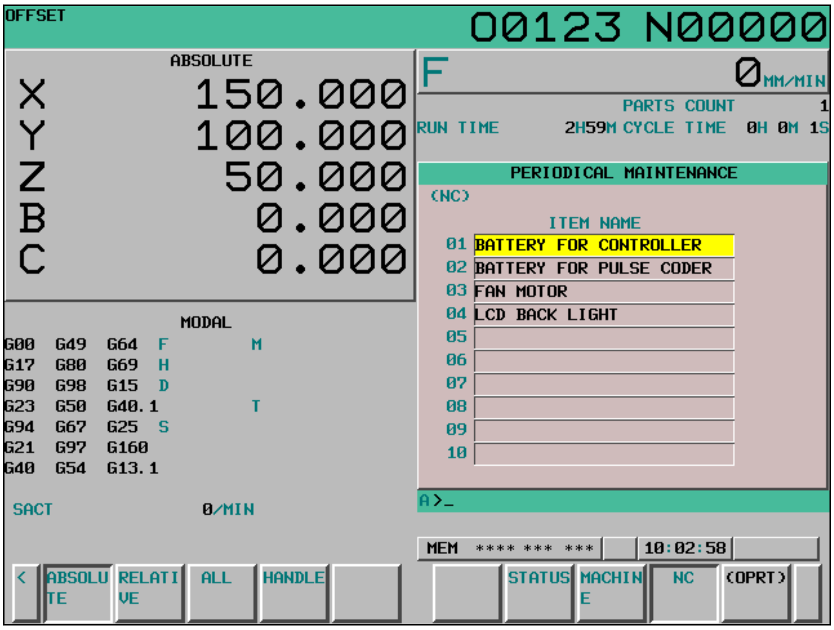


Fig. 12.4.11 (d) NC menu screen

- Displaying the screen

1 When the status screen is displayed, press soft key [NC].

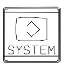
> **NOTE**
> On the NC menu screen, the registration, deletion, and I/O of item names cannot be performed.
> When a blank item name is selected, a blank is set.

12.4.12 System Configuration Screen

The system configuration screen provides information about the types of installed hardware and software.

Procedure for displaying the screen

Procedure

1 Press the  key to display a screen that shows parameters and other information.

2 Press soft key [SYSTEM]. The system configuration screen is displayed.

There are two types of system configuration screens: the hardware configuration screen and the software configuration screen. The screen display can switch between these screens by using  and . When soft key [SERVO INFO] or [SPINDLE INFO] is pressed, information about the connected servo system or spindles is displayed.

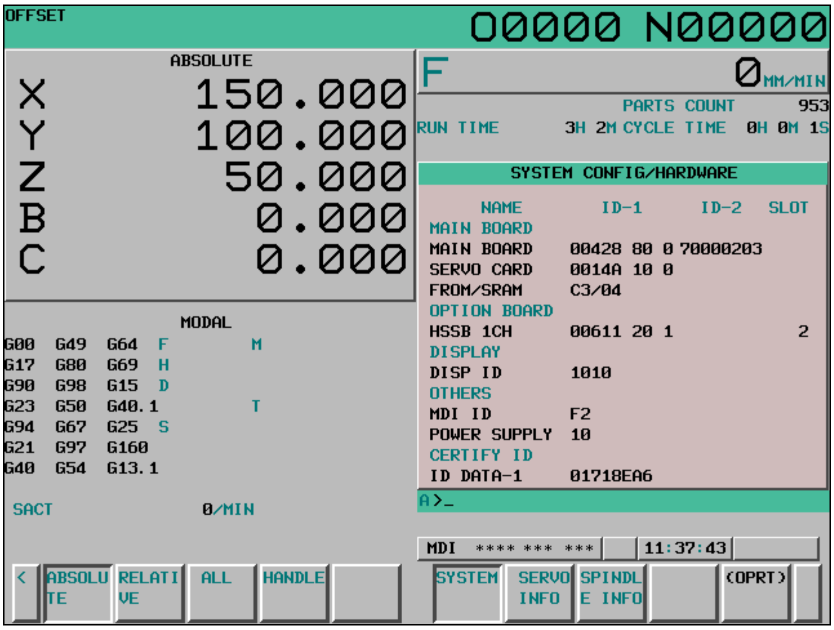


Fig. 12.4.12 (a) System configuration screen

Hardware configuration screen

This screen shows the names and IDs of the hardware used by the NC.

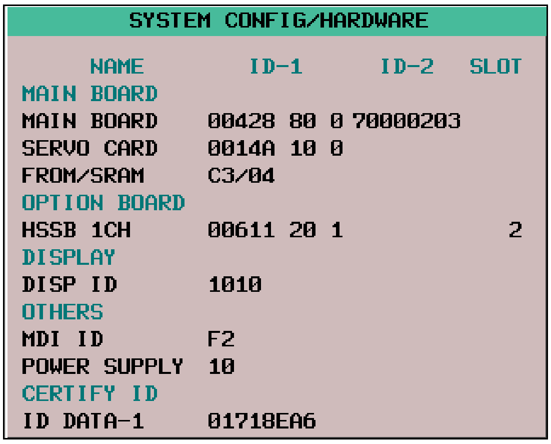
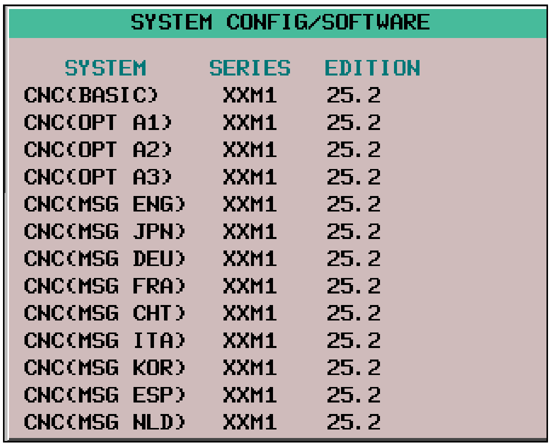


Fig. 12.4.12 (b) Hardware configuration screen

Software configuration screen

This screen shows the names and series/editions of the software used by the NC.



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

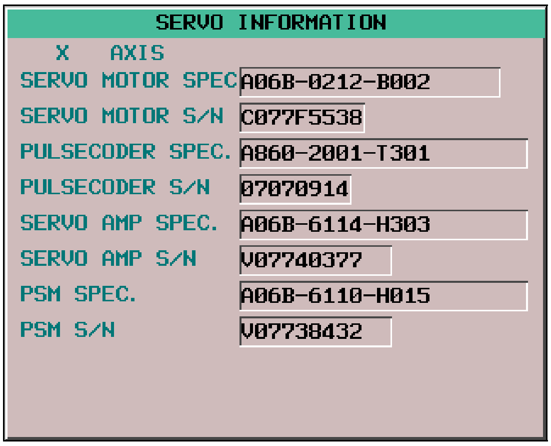
Fig. 12.4.12 (c) Software configuration screen

Servo information screen

When a servo system is connected to the NC, ID information of the connected servo devices (servo motors and servo amplifier modules) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press soft key [SERVO INFO].
2 The servo information screen is displayed.



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

Fig. 12.4.12 (d) Servo information screen

Spindle information screen

When a spindle system is connected to the NC, the ID information of the connected spindle devices (spindle motors and spindle amplifier modules) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press soft key [SPINDLE INFO] ([SPINDLE] for the 8.4-inch display unit).
2 The spindle information screen is displayed.

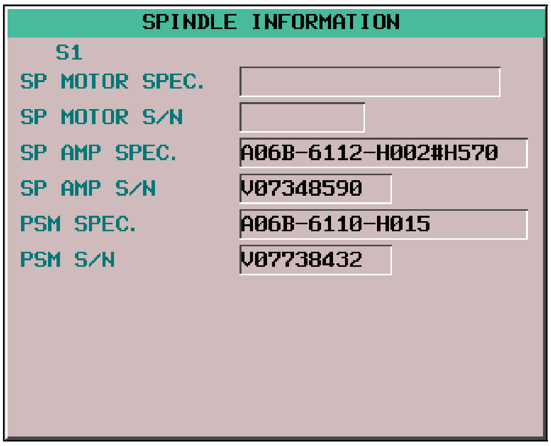


Fig. 12.4.12 (e) Spindle information screen

12.4.13 Overview of the History Function

The history function records the operations performed by the operator, alarms that occurred, and external operator messages and checks their history or outputs them as history data.

Conditions for recording history

a Display screen
A history of screens other than the operation history screen is recorded.

b Signal selection
Up to 60 I/O signals can be selected so that they can be recorded in history data.

c Parameter setting
With parameters, it is possible to separately set whether to record an MDI key operation history and an external operator message history, add external alarms and messages, and record data modification histories such as operation histories of parameters, tool offsets, workpiece offsets (workpiece shift amounts), custom macro common variables, and I/O signals.

Data output

All history data stored can be output to external input/output devices. (See Subsection 12.4.15.5, "Outputting all history data".)

> **NOTE**
> 1 All history data remains even after the power is turned off.
> Clearing memory also deletes these history data items.
> 2 Set the time and date correctly on the setting screen.

12.4.13.1 Alarm history

The alarm generated in CNC is recorded. The alarm of 50 times is recorded, and it is displayed from the new one sequentially. If the amount of alarm history data exceeds 50 items, alarm history data is automatically deleted in sequence from the oldest one.

Fig. 12.4.13.1 (a) Alarm history screen

Screen display

Issued alarms are displayed sequentially from the latest alarm.
The following information is displayed for each alarm:

- Path name (only when 2-path control is performed)
- Time and date of alarm occurrence
- Type and number of the alarm
- Alarm message

> **NOTE**
> To record also external alarm and macro alarm messages as history data, set bit 3 (EAH) of parameter No. 3112 to 1, and at the same time set bit 7 (HAL) of parameter No. 3196 to 0.

When a name such as a path name, axis name, or spindle name has been changed after the issuance of an alarm, a newly assigned name is displayed on the alarm history screen.

Procedure

1 Press function key .

2 Press continuous menu key  several times until soft key [HISTRY] is displayed.

3 Press soft key [HISTRY]. The alarm history screen is then displayed.

4 The screen display can be changed to the previous page and the next page by using page keys  and .

Erasing alarm history data from the alarm history screen

Procedure

1 Display the alarm history screen.
2 Press soft key [(OPRT)].
3 Press soft key [CLEAR]. Alarm history data is then erased.

Display of external alarms and macro alarms

The following parameter can be used to record a message as well as the corresponding alarm number in alarm history when an external alarm or macro alarm occurs.

	#7	#6	#5	#4	#3	#2	#1	#0
3112					EAH			

[Data type] Bit

#3 EAH Messages of the external alarm/macro alarm in alarm or operation history:
0: Not recorded
1: Recorded

> **NOTE**
> This parameter is valid when bit 7 (HAL) of parameter No. 3196 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3195	EKE							

[Data type] Bit

#7 EKE The contents of operation history and alarm history
0: cannot be deleted.
1: can be deleted.

	#7	#6	#5	#4	#3	#2	#1	#0
3196	HAL							

[Data type] Bit

#7 HAL When an alarm is issued, additional information (modal data, absolute coordinates, and machine coordinates present at the issuance of the alarm) is:
0: Recorded in the operation history.
1: Not recorded in the operation history.

12.4.13.2 External operator message history

External operator message can be stored as history. And, stored history can be seen on the external operator message history screen.

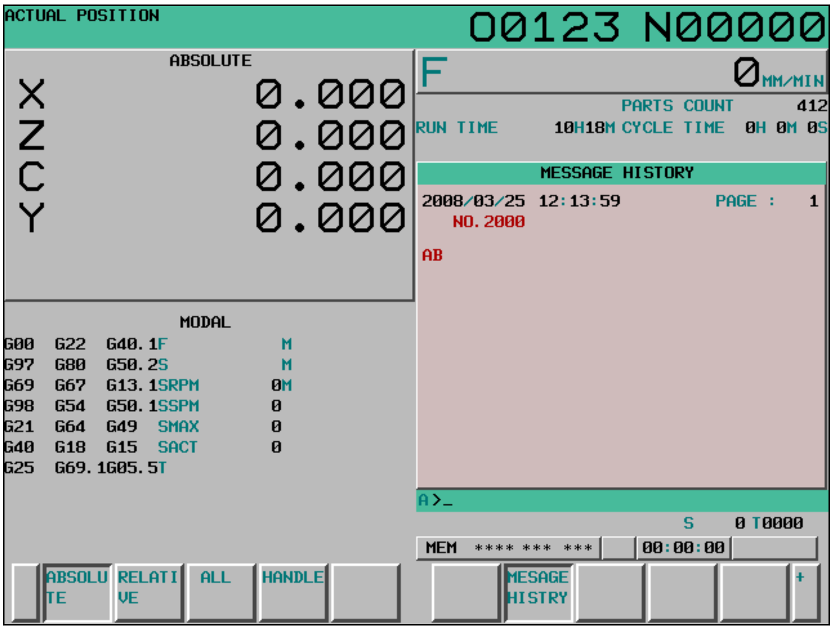


Fig. 12.4.13.2 (a) External operator message history screen

Screen display

To display the external operator message history screen, set bit 2 (OMH) of parameter No. 3112 to 1.

Procedure

1 Press function key .

2 Press continuous menu key  several times until soft key [MESSAGE HISTRY] is displayed.

3 Press soft key [MESSAGE HISTRY]. The external operator message history screen is displayed.

4 The screen display can be changed to the previous page and the next page by using page keys  and .

Erasing history data from the external operator message history screen

Procedure

1 Display the external operator message history screen.

2 Press soft key [(OPRT)].

3 Press soft key [CLEAR]. The external operator message history data is then erased.

Parameter setting

	#7	#6	#5	#4	#3	#2	#1	#0
3112						OMH		

[Data type] Bit

#2 OMH The external operator message history screen is:

0: Not displayed.

1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3113	MS1	MS0						HMC

[Data type] Bit

#0 HMC The contents of the external operator message history:
0: Cannot be erased.
1: Can be erased.

#6 MS0
#7 MS1 Set the combination of the number of characters and the number of messages to be preserved in the external operator message history.

Parameter		Maximum number of characters	Number of messages
MS1=0	MS0=0	255	8
MS1=0	MS0=1	200	10
MS1=1	MS0=0	100	18
MS1=1	MS0=1	50	32

NOTE
1 Although up to 255 characters can be specified for each external operator message, you can use the combination of bits 6 (MS0) and 7 (MS1) of parameter No. 3113 to limit the number of characters and select the number of messages to be preserved in the external operator message history.
2 The settings of bits 6 (MS0) and 7 (MS1) of parameter No. 3113 take effect the next time the power is turned on. The external operator message history is erased at that time.
3 Even though you change the settings of bits 6 (MS0) and 7 (MS1) of parameter No. 3113, the alarm PW0000, "POWER MUST BE OFF" is not issued. You must however turn on the power again before the new settings can take effect.
4 If text (such as single-byte katakana or kanji characters) is entered in character code, the number of characters recorded in the external operator message history may be smaller than the maximum number of characters set by bits 6 (MS0) and 7 (MS1) of parameter No. 3113.

12.4.13.3 Operation history

This function displays a history of the operator's key operations and signal operations made when a failure occurred or an alarm was issued, and also information about alarms.
The following data is recorded:

a Operation history
  i MDI key operations made by the operator
  ii I/O signal (X,Y,G,F) on/off switching
b Alarm history
  i Alarms issued
  ii Modal information in a block executed and coordinates observed when an alarm was issued (Not displayed on the screen)

c Data modification history
  i Modification of tool offset data
    (When bit 0 (HTO) of parameter No. 3196 is set to 1)
  ii Modification of workpiece offset data/extended workpiece offset data/workpiece shift (T series)
    (When bit 1 (HWO) of parameter No. 3196 is set to 1)
  iii Modification of parameters
    (When bit 2 (HPM) of parameter No. 3196 is set to 1)
  iv Modification of custom macro common variable data
    (When bit 3 (HMV) of parameter No. 3196 is set to 1)
d External operator message history and macro message history
  (When bit 6 (HOM) of parameter No. 3196 is set to 0)
e Time stamp (time and date)

With some exceptions, history data of the operation history and alarm history can be viewed on the operation history screen. (The data modification history, external operator message history, and alarm messages are not displayed.)
All recorded history data can be output to external input/output devices.

NOTE
Up to about 8000 items of history data can be recorded if the data includes history data of MDI key operations only. However, because history data varies in size, the maximum number of history data items that can be recorded is not fixed.

Parameter setting

	#7	#6	#5	#4	#3	#2	#1	#0
3106				OPH				

[Data type] Bit

 # 4 OPH The operation history screen is:
0: Not displayed.
1: Displayed.

3122	Time interval used to record time data in operation history

[Input type] Parameter input
[Data type] Word path
[Unit of data] min
[Valid data range] 0 to 1440

When history data is recorded within a set time period, the time for each set time period is recorded in the history data.
When 0 is set, the specification of a time period of 10 minutes is assumed.
If no data is recorded within a set time period, the time for that period is not recorded.

NOTE
When two paths are present, set this parameter to the same value for both path.

	#7	#6	#5	#4	#3	#2	#1	#0
3195	EKE	HDE	HKE					

[Input type] Parameter input
[Data type] Bit

#5 HKE A key operation history is:
0: Recorded.
1: Not recorded.

#6 HDE A DI/DO history is:
0: Recorded.
1: Not recorded.

#7 EKE The contents of operation history and alarm history
0: cannot be deleted.
1: can be deleted.

	#7	#6	#5	#4	#3	#2	#1	#0
3196	HAL	HOM			HMV	HPM	HWO	HTO

[Data type] Bit

#0 HTO A modification history of tool offset data is:
0: Not recorded.
1: Recorded.

#1 HWO A modification history of workpiece offset data/extended workpiece offset data/workpiece shift (T series) is:
0: Not recorded.
1: Recorded.

#2 HPM A modification history of parameters is:
0: Not recorded.
1: Recorded.

#3 HMV A modification history of custom macro common variables is:
0: Not recorded.
1: Recorded.

#6 HOM A history of external operator messages and macro messages (#3006) is:
0: Recorded.
1: Not recorded.

#7 HAL When an alarm is issued, additional information (modal data, absolute coordinates, and machine coordinates present at the issuance of the alarm) is:
0: Recorded in the operation history and alarm history.
1: Not recorded in the operation history and alarm history.
To record as many alarm history items as possible, rather than detailed alarm information, set 1. The numbers of ten G code modal groups to be recorded are set in parameter Nos. 12990 to 12999.

12990	(1st) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input

[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups
Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.
* If a value beyond the valid data range is set, the status of group 01 is recorded.

12991	(2nd) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups
Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.
* If a value beyond the valid data range is set, the status of group 02 is recorded.

12992	(3rd) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups
Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.
* If a value beyond the valid data range is set, the status of group 03 is recorded.

12993	(4th) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups
Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.
* If a value beyond the valid data range is set, the status of group 04 is recorded.

12994	(5th) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups
Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.
* If a value beyond the valid data range is set, the status of group 05 is recorded.

12995	(6th) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups
Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.
* If a value beyond the valid data range is set, the status of group 06 is recorded.

12996	(7th) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups

Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.

* If a value beyond the valid data range is set, the status of group 07 is recorded.

12997	(8th) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups

Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.

* If a value beyond the valid data range is set, the status of group 08 is recorded.

12998	(9th) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups

Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.

* If a value beyond the valid data range is set, the status of group 09 is recorded.

12999	(10tht) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups

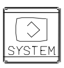
Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.

* If a value beyond the valid data range is set, the status of group 10 is recorded.

Screen display

To display the operation history screen, set bit 4 (OPH) of parameter No. 3106 to 1.

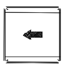
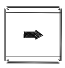
Procedure

1 Press function key .

2 Press continuous menu key  several times until soft key [OPERAT HISTRY] ([OPEHIS] for the 8.4-inch display unit) is displayed.

3 Press soft key [OPERAT HISTRY], then press newly displayed soft key [OPERAT HISTRY]. The operation history screen is then displayed.

4 To display the previous page and next page of the operation history, use page keys  and .

To display a part across two pages, use the cursor keys  and . The screen display is shifted by half page.

(With the 8.4-inch display devices, the screen display is shifted by one column.)

By pressing soft key [(OPRT)] on the operation history screen, it becomes possible to perform the following soft key operations:

a [TOP] displays the starting page (the oldest data).
b [BOTTOM] displays the end page (the latest data).
c [NO.SRH] displays specified operation history data.
(Example) Specifying 50 then [NO.SRH] displays the 50th data.

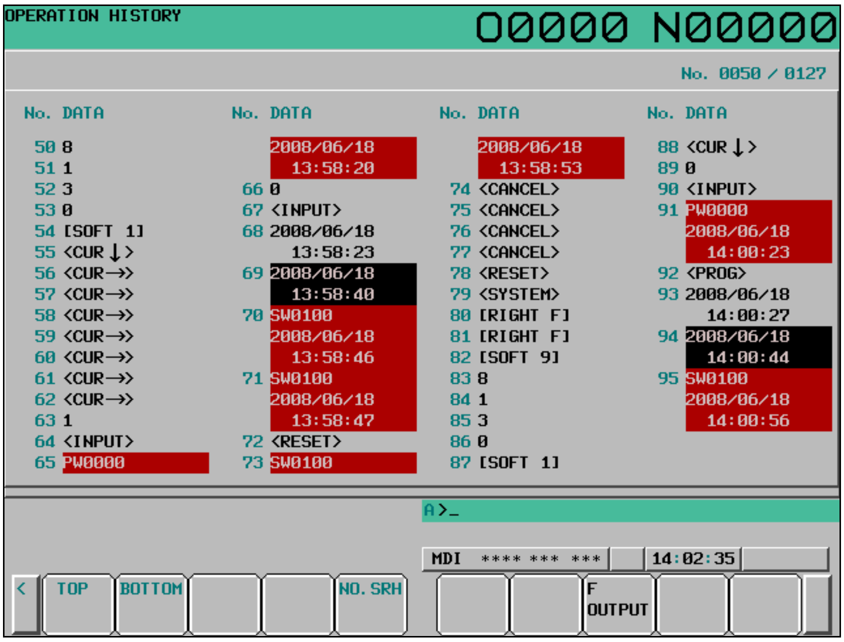


Fig. 12.4.13.3 (a) Operation history screen

Displayed information

1 Serial number and display start history number/total number of history data items
A serial number is indicated on the left side of each recorded history data item. A smaller serial number indicates an older data item.
In the upper right part of the screen, the display start history number and the total number of history data items are indicated. The total number of history data items does not include history data items not displayed on the screen.

2 Data
- MDI key
When bit 5 (HKE) of parameter No. 3195 is set to 0, key operations are recorded.
A key operation is indicated following a path number (for example, "1_[LEFT F]", and "2_[LEFT F]"). (When only one path is used, the path number is not indicated.)
P_ used in "P_[LEFT F]", for example, indicates a key operation made from the outside.
i Address keys and numeric keys
Characters such as A to Z, 0 to 9, ;, +, and - are indicated directly.
These characters are displayed in black.
ii Function menu keys, operation menu keys, and soft keys
These keys are enclosed in brackets [] (for example, "[LEFT F]", "[SOFT 1]" to "[SOFT 10]", and "[RIGHT F]"). These keys are displayed with green characters.
iii Function keys, page keys, cursor keys, and so on
These keys are enclosed in angle brackets <> (for example, "<POS>", "<SYSTEM>", "<PAGE ↑>", "<CUR →>", "<RESET>", and "<INPUT>"). These keys are displayed with green characters.
iv Power-on key
This key is displayed with white characters in the green background.

- I/O signals
  When bit 6 (HDE) of parameter No. 3195 is set to 0, I/O signals specified on the operation history signal selection screen are recorded. Recorded signals are indicated on a bit-by-bit basis with information about the signal address and a change in bit. These signals are displayed with purple characters.

G000.2 ↓

Change from 0 to 1 is indicated by ↑.
Change from 1 to 0 is indicated by ↓.

Indicates a bit.

Indicates the address of a signal.

NOTE
1 When more than one bit at the same address changes at the same time, the changes of these bits are treated as one history data item.
2 A signal fluctuation for less than 8 msec is not recorded as history data.

- Alarms
  Alarm numbers and the issuance times are displayed on the operation history screen.
  This alarm information is displayed with white characters in the red background.

Indicates the path number.
Indicates the alarm number.
For the servo axis type, indicates the servo axis name.
For the spindle type, indicates the spindle name or spindle number.

1_OT0506 (XA1)
2008/01/11 Indicates the date and time of issuance in two lines.
11:22:33

When a path name, axis name, or spindle name was changed after the issuance of an alarm, the alarm is indicated with a newly assigned name.

- Time and date
  The time and date of the following are displayed in two lines:
  i The time and date of power-on. These are displayed with white characters in the green background.
  ii The time and date of power-off. These are displayed with green characters.
  iii Date on which the date changed. This is displayed with black characters.
  iv Date and time at regular intervals of the period set in parameter No. 3122. These are displayed with black characters.
  v Date and time when history data was erased. These are displayed with black characters.

NOTE
1 For item iv (recording time at regular intervals) above, if there is no operation to be recorded within a certain period, the time is not recorded. For item iii (date on which the date changed) above, however, it is assumed that there is a data item to be recorded and item iv (recording time at regular intervals) above is also recorded.
2 When the date and time in the CNC system are changed, the date and time at which the change is made may be recorded as the date and time in iii or iv above.

History data not displayed on the screen

In addition to history data of MDI keys, I/O signal status, alarms issued, external operator messages (not displayed on the operation history screen), and time stamps, data described below can be recorded with time. Such history data cannot be displayed on the screen but can be output to external input/output devices. (See Subsection 12.4.15.5, "Outputting all history data".)

1 Detailed data at alarm issuance
If bit 7 (HAL) of parameter No. 3196 is set to 0, 10 modal G codes, auxiliary function codes D, E, F, H, M, N, O, S, and T, absolute coordinates, and machine coordinates in the block being executed when an alarm was issued are recorded together with the alarm number and the time of alarm issuance.
The group numbers of the 10 modal G codes to be recorded are set in parameter Nos. 12990 to 12999. If these parameters are not specified, modal G codes of groups 01 to 10 are recorded.

NOTE
To record as many items as possible instead of recording detailed data at alarm issuance, set bit 7 (HAL) of parameter No. 3196 to 1.

2 External alarm messages and macro alarm messages
If bit 3 (EAH) of parameter No. 3112 is set to 1, external alarm messages and macro alarm messages can also be recorded as history data.

NOTE
To record also external alarm and macro alarm messages as history data, set bit 3 (EAH) of parameter No. 3112 to 1, and at the same time set bit 7 (HAL) of parameter No. 3196 to 0.

3 Modification of tool offset data
If bit 0 (HTO) of parameter No. 3196 is set to 1, when tool offset data is modified, the number and type of the tool offset are recorded as well as the tool offset data before modification, the tool offset data after modification, and the time of modification.

4 Modification of workpiece offset/workpiece shift (T series) data
If bit 1 (HWO) of parameter No. 3196 is set to 1, when workpiece offset data is modified, the number of the modified workpiece offset is recorded as well as the workpiece offset data before modification, the workpiece offset data after modification, and the time of modification. Similar data is recorded also when workpiece shift amounts (T series) are modified.

5 Modification of parameters
If bit 2 (HPM) of parameter No. 3196 is set to 1, when a parameter is modified, the number and type (axis type, spindle type, path type, or machine group type) of the parameter are recorded as well as the parameter data before modification, the parameter data after modification, and the time of modification.

NOTE
Modifications made at power-on and modifications of passwords and keys are not recorded as history data.

6 Modification of custom macro common variables (#100 to #999)
If bit 3 (HMV) of parameter No. 3196 is set to 1, when a custom macro common variable is modified, the number of the common variable is recorded as well as the common variable value before modification, the common variable value after modification, and the time of modification.

Erasing history data from the operation history screen

Procedure

1 Display the operation history screen.
2 Press soft key [(OPRT)].
3 Press soft key [ALL CLEAR].
4 Press soft key [EXEC]. Operation history data is erased.

12.4.13.4 Selecting operation history signals

I/O signals to be recorded as history data can be selected. Up to 60 signals can be set to select signals.

Setting data

1 Press function key SYSTEM.
2 Press continuous menu key ▷ several times until soft key [OPERAT HISTRY] is displayed.
3 Press soft key [OPERAT HISTRY].
4 Press soft key [SIGNAL SELECT] to display the operation history signal selection screen.
5 Press soft key [(OPRT)].
6 Move the cursor to a desired position by using cursor keys ↑ and ↓.
7 Type a signal type (X, G, F, or Y) and an address, then press INPUT.
Example:
When G0004 is entered then INPUT is pressed:
The entered signal address, G0004, is set in ADDRESS, and initial value 00000000 is set in SIGNAL.
8 Select the bits to be recorded in the history.
To change the status of all bits of the specified signal address, place the cursor in all bits so that the bits are highlighted (for example, "00000000"), then press soft key [ON:1] or [OFF:0]. Then, the bits are set to 11111111 or 00000000.
To change the status of only a certain bit, move the cursor to the bit by using cursor keys ← and →, and press [ON:1] or [OFF:0]. The selected bit is then set to 1 or 0.
9 Up to 60 addresses can be set to select signals. Addresses need not necessarily be set sequentially starting from No. 1.

NOTE
1 While the operation history signal selection screen is being displayed, recording of history data is not performed.
2 Only X, Y, G, and F can be set for I/O signals.
For data not set, "********" is displayed.
3 Even when an address is set, history data is not recorded if all bits are set to 0.
4 When the ON/OFF width of an input signal is less than 8 msec, recording of history data is not performed. Also, there are some signals that are not recorded.
5 When many signals are selected, the processing speed may lower.

Clearing the selection of each signal

1 Display the operation history signal selection screen.
2 Move the cursor to the data to be cleared.
3 Press soft key [DELETE].

4 Press soft key [EXEC].

Clearing the selection of all signals

1 Display the operation history signal selection screen.
2 Press soft key [ALLDEL].
3 Press soft key [EXEC].

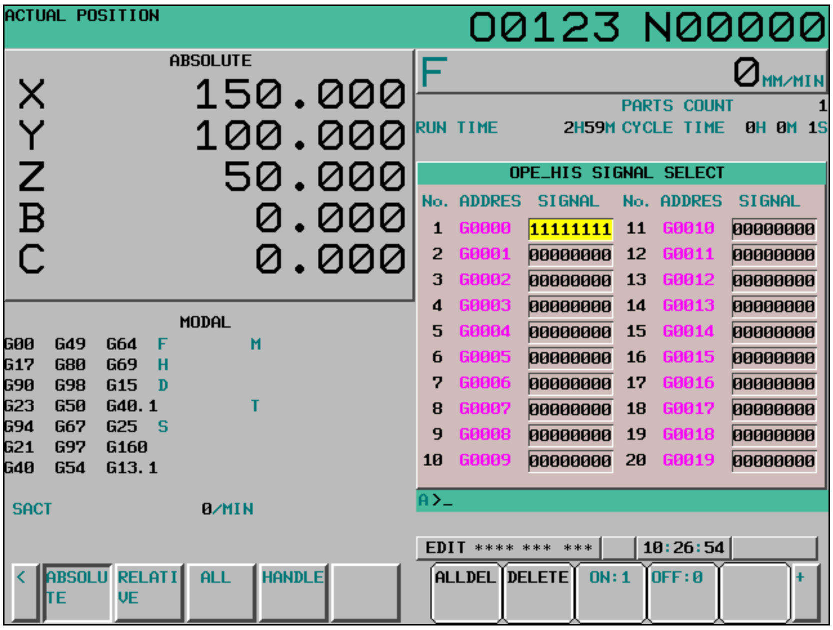


Fig. 12.4.13.4 (a) Operation history signal selection screen

Selection with parameters

If bit 4 (PHS) of parameter No. 3206 is 1, selection of input/output signals to be recorded as history data can be made with parameters.
In this case, if a signal is selected or deselected on the operation history signal selection screen, the value of the corresponding parameter is automatically changed. If the value of a parameter is changed on the parameter screen, the display on the operation history signal selection screen is also changed.
The same applies to input operations. If operation history signal selection data is input on the operation history signal selection screen, the value of the corresponding parameter is automatically changed. If a parameter is input on the parameter screen, the display on the operation history signal selection screen is also changed.
Only the first 20 of the 60 data items can be selected with parameters.

Parameter setting

	#7	#6	#5	#4	#3	#2	#1	#0
3206				**PHS**				

[Input type] Parameter input
[Data type] Bit

#4 PHS Operation history signal selection:
0: Does not interact with parameters.
Operation history signal selection is added or deleted on the operation history signal selection screen.
Changing the settings of parameters Nos. 12801 to 12820, Nos. 12841 to 12860, or Nos. 12881 to 12900 has no effect on operation history signal selection.
Changes to the signals of the addresses specified by parameters Nos. 12801 to 12820, Nos. 12841 to 12860, or Nos. 12881 to 12900 are not recorded in the history.

1: Interacts with parameters.
Operation history signal selection can be performed either on the operation history signal selection screen or by setting parameters.

> **NOTE**
> Setting this parameter to 1 reflects the current operation history signal selection data on parameters Nos. 12801 to 12900.

12801	**Operation history signal selection address type (No.01)**
to	**to**
12820	**Operation history signal selection address type (No.20)**

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 4
These parameters set operation history signal selection address types Nos. 1 to 20.
The correspondence between address types and settings is as given in the table below.

Address type	Parameter value
Not selected.	0
X	1
G	2
Y	3
F	4

Nos. 1 to 20 correspond to Nos. 1 to 20 on the operation history signal selection screen.
These parameters are paired with other parameters as given below.

No.	Address type	Address number	Bit number
01	No. 12801	No. 12841	No. 12881
02	No. 12802	No. 12842	No. 12882
03	No. 12803	No. 12843	No. 12883
…	…	…	…
20	No. 12820	No. 12860	No. 12900

> **NOTE**
> 1 Operation history signals that can be selected and deselected with parameters are for the first 20 of 60 sets.
> 2 To deselect a signal, set 0.
> At this time, 0 is set as the initial value in the address number (Nos. 12841 to 12860) and the bit number (Nos. 12881 to 12900) corresponding to that signal.
> 3 When an address type is set, 0 is set as the initial value in the address number (Nos. 12841 to 12860) and the bit number (Nos. 12881 to 12900).
> [Example]
> If parameter No. 12801 is set to 2, the parameters are initialized as follows:
> No. 12841=0 Address number
> No. 12881=00000000 Bit number
> 4 If an attempt is made to set a value that cannot be set, a warning, "DATA IS OUT OF RANGE" appears; retry setting a value.

12841	Operation history signal selection address number (No.01)
to	to
12860	Operation history signal selection address number (No.20)

[Input type] Parameter input
[Data type] Word
[Valid data range] For an explanation of the address ranges of the G, F, X, and Y signals, refer to the PMC Programming Manual (B-64393EN).

These parameters set operation history signal selection address numbers Nos. 1 to 20.
Nos. 1 to 20 correspond to Nos. 1 to 20 on the operation history signal selection screen.
These parameters are paired with other parameters as given below.

No.	Address type	Address number	Bit number
01	No. 12801	No. 12841	No. 12881
02	No. 12802	No. 12842	No. 12882
03	No. 12803	No. 12843	No. 12883
…	…	…	…
20	No. 12820	No. 12860	No. 12900

NOTE
1 Operation history signals that can be selected and deselected with parameters are for the first 20 of 60 sets.
2 When an address number is set, 0 is set as the initial value in the bit number (Nos. 12881 to 12900) corresponding to that signal.
3 If an attempt is made to set a value that cannot be set or if the address type (Nos. 12801 to 12820) corresponding to that signal is 0, a warning, "DATA IS OUT OF RANGE" appears; retry setting a value.

	#7	#6	#5	#4	#3	#2	#1	#0
12881	RB7	RB6	RB5	RB4	RB3	RB2	RB1	RB0
to				to				
12900	RB7	RB6	RB5	RB4	RB3	RB2	RB1	RB0

[Input type] Parameter input
[Data type] Bit

RB7 - RB0 History of the respective operation history signal selection bits Nos. 1 to 20 (RB7 to RB0) corresponding to the operation history signal selection addresses set in parameters Nos. 12801 to 12860 is:
0 : Not retained. (History of the bit is not recorded.)
1 : Retained. (History of the bit is recorded.)

These parameters are paired with other parameters as given below.

No.	Address type	Address number	Bit number
01	No. 12801	No. 12841	No. 12881
02	No. 12802	No. 12842	No. 12882
03	No. 12803	No. 12843	No. 12883
…	…	…	…
20	No. 12820	No. 12860	No. 12900

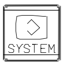
NOTE
1 Operation history signals that can be selected and deselected with parameters are for the first 20 of 60 sets.

> **NOTE**
> 2 If the value of the address type (Nos. 12801 to 12820) corresponding to that signal is 0, a warning, "DATA IS OUT OF RANGE" appears; retry setting a value.

12.4.13.5 Outputting all history data

All history data (operation history data, alarm history data, and operator message history data) can be output to external input/output devices.
It is impossible, however, to output history data individually.

Procedure

1 Make an output device ready for output.
2 Set the EDIT mode.
3 Press function key 

.
4 Press continuous menu key  several times until soft key [OPERAT HISTRY] ([OPEHIS] for the 8.4-inch display unit) is displayed.
5 Press soft key [OPERAT HISTRY], then press newly displayed soft key [OPERAT HISTRY]. The operation history screen is displayed.
6 Press soft key [(OPRT)].
7 Press soft key [F OUTPUT].
8 Enter a file name, and press soft key [EXEC].
When soft key [EXEC] is pressed without entering a file name, the output file name is assumed to be OPRT_HIS.TXT.

Output format

History data is output as an ASCII file in the following format:

1 MDI keys
After "MDI", "path-number_", "key-data", and "input-time" are output in this order.
(Key data input at power-on is indicated as "Power on MDI".)
<Example>
MDI 01_A 12:23:34
MDI 02_<CAN> 12:23:34
MDI 02_[SOFT HF1] 12:23:35
MDI P_<RESET> 12:34:56
Power on MDI 01_<RESET> 12:34:56

2 I/O signals
After "DI/DO", "PMC-number_", "signal-address_bit-status", and "time-of-change" are output in this order.
<Example>
DI/DO 1_F0002.2_on 12:34:56
DI/DO 1_ G0043.0_off G0043.1_off 12:35:00
(For multiple bits at the same address)

3 Alarms
After "Alarm", "path-number_", "type", "alarm-number", "G-code-modal-data", "modal-data-other-than-G-code", "absolute-coordinate-value" and "machine-coordinate-value" for each axis, and "date-and-time-of-alarm-issuance" are output in this order.
An asterisk "*" is output before the modal data that was specified in a block executed when the alarm was issued.

<Example>

- Alarm 01_SR01973
  *G0. G97. G69. G99. G21. G50.2 G25. G13.1 B0. D0. E0. *F100. H0. M10. *N123. Test_ S1000. T1010.
  X1 ABS 197.999 MCN 197.999 Y1 ABS -199806.00 MCN -199806.00 Z1 ABS 297.009 MCN 0.123 C1 ABS 10395.999 MCN 0.000 at 2007/09/01 19:03:28
- Alarm 02_ OT00506(ZA2)
  *G1. G17. G90. G22. G94. G20. *G42. G49. G80. G12.1 B0. *D12. E0. *F100. H34. M0. *N123. O123 S0. T0.
  X2 ABS 123.999 MCN 234.000 Y2 ABS -123.00 MCN -234.00 ZA2 ABS 1234.567 MCN –1234.567 at 2007/09/01 12:34:56
- When additional information is not to be recorded when an alarm is issued (bit 7 (HAL) of parameter No. 3196 is set to 1), only "path-number_", "alarm-number", and "date-and-time-of-alarm-issuance" are output.
  Alarm 01_OT00506(XC1) at 2007/09/01 22:08:32
  Alarm 02_SW00100 at 2007/09/01 19:07:52
- When external alarm/macro alarm messages are to be recorded (bit 7 (HAL) of parameter No. 3196 is set to 0) and bit 3 (EAH) of parameter No. 3112 is set to 1, the messages are also output.
  Alarm 01_MC00001 Message ATC ALARM
  G0. G97. G69. G99. G21. G40. G25. G22. G80. D0. E0. F0. H0. M0. N0. O9999 S0. T0.
  X2 ABS 10.000 MCN 0.000Y2 ABS 123.000 MCN 0.000Z2 ABS 0.000 MCN 0.000 at 2007/09/01 10:06:43

4 External operator messages
After "EXT_Message", "message-number", "message", and "date-and-time-of-issuance" are output in this order.
<Example>
EXT_Message 01234 OIL PRESSURE DECREASE at 2007/09/01 2:38:43

5 Modification of tool offset data
After "Tool Offset", "path-number_", "type", "offset-number", "offset-data-before-modification", "offset-data-after-modification", and "time-of-modification" are output in this order. The following types are provided:

Common to M/T	:	G=Geometric compensation	W=Wear compensation
M series	:	H=Tool length compensation	D=Cutter compensation
T series	:	R=Tool-nose radius compensation	T=Tool nose direction

<Example>
Tool Offset 01_X0002 0.000 → 1 at 12:15:43
Tool Offset 02_XW0001 -9999.999 → 9999.999 at 12:15:46
Tool Offset 01_RG0032 0.000 → 0.003 at 12:15:52
Tool Offset 02_T0001 5. → 2. at 19:34:11
Tool Offset 02_W0123 -10.000 → 123.456 at 10:28:58
Tool Offset 01_HG0456 0.000 → 999.999 at 11:37:40
Tool Offset 01_ 0064 12.340 → 12.569 at 11:39:42

6 Modification of workpiece offset/workpiece shift (T series) data
After "Work Offset", "EXT Work Offset", or "Work Shift", "path-number_(axis-name)", "type", "offset-number", "offset-data-before-modification", "offset-data-after-modification", and "time-of-modification" are output in this order.
<Example>
Work Offset 01_G55(XA1) 15.000 → 0.007 at 09:23:03
Work Offset 02_EXT(Z2) 0.000 → 300.003 at 09:22:50
EXT Work Offset 02_G54.1P300 (Y2) 123.456 → 9999.999 at 12:15:46

Work Shift (X) 02_999999.999 → 999999.999 at 10:22:37

7 Modification of parameters After "Parameter", "type", "parameter-number", "parameter-before-modification", "parameter-after-modification", and "time-of-modification" are output in this order.
The following types are provided:

Path type	: L is added before the path number.
Axis type	: A is added before the axis number.
Spindle type	: S is added before the spindle number.
Machine group type	: Indicated as the machine type. T is added before the machine group number.
Others	: No type is output.

<Example>

Parameter N03112 00000100 → 00001100 at 11:18:40
Parameter Path type N01410 L02 0.000 → 1000.000 at 18:58:48
Parameter Axis type N01423 A04(B2) 0.000 → 10000.000 at 18:58:48
Parameter Spindle type N04011 S1(S) 10011010 → 10011010 at 18:58:53
Parameter Machine type N06310 T01 0 → -32768 at 19:21:13

8 Modification of custom macro common variables (#100 to #999)
After "Macro variable", "path-number_", "#variable-number", "common-variable-value-before-modification", "common-variable-value-after-modification", and "time-of-modification" are output in this order.
Variable values are output in the data format M × (10**(-E)).
<Example>

- When #149 on the first path was modified from <empty> to 12.345
  Macro variable 01_#149 Empty → 123450000*(10**-7) at 15:02:35
- When #549 on the second path was modified from -12.345 to 123456789012
  Macro variable 02_#549 -123450000*(10**-7) →123456789*(10**3) at 15:03:27

9 Date and time
Power on at 2008/02/01 17:11:17 (Date and time of power-on)
Power off at 2008/02/01 17:49:17 (Date and time of power-off)
Date 2008/02/01 00:00:00 (Record indicating a change in date)
Time stamp at 2008/02/01 15:51:00 (Record at regular intervals)
Data delete at 2008/02/01 10:56:18 (Date and time when history data was deleted)

Example of output

```
%
=============== OPERATION HISTORY ===============
Data delete at 2000/01/23 12:34:01
MDI 01_<RESET> 12:34:02
MCR_Message MACRO MESSAGE  at 2000/01/23 12:34:03
Alarm 01_MC03001 Message MACRO ALRM
G0.G97.G69.G99.G21.G40.G25.G22.G80. D0.E0.F0.H0.M0.N0.O1234 S0.T0.
X1 ABS 0.005 MCN 0.000 Z1 ABS 0.010 MCN 0.000 at 2000/01/23 12:34:04
MDI 01_<SYSTEM> 12:34:05
MDI 01_[RIGHT F] 12:34:06
MDI 01_[RIGHT F] 12:34:07
MDI 01_[RIGHT F] 12:34:08
MDI 01_[SOFT HF9] 12:34:09
DI/DO  1_ G0043.1_on 12:34:10
Alarm 01_SR01973
G0.G97.G69.G99.G21.G40.G25.G22.G80. D0.E0.F0.H0.M0.N0.O1234 S0.T0.
X1 ABS 0.005 MCN 0.000 Z1 ABS 0.010 MCN 0.000 at 2000/01/23 12:34:11
MDI 01_<RESET> 12:34:12
EXT_Message 02001 EXT MESSAGE at 2000/01/23 12:34:13

=============== ALARM HISTORY ===============
Alarm 01_MC03001 Message MACRO ALRM at 2000/01/23 12:34:04
Alarm 01_SR01973 at 2000/01/23 12:34:11

========= OPERATION MESSAGE HISTORY =========
MCR_Message MACRO MESSAGE  at 2000/01/23 12:34:03
EXT_Message 02001 EXT MESSAGE at 2000/01/23 12:34:13
%
```

12.5 SCREENS DISPLAYED BY FUNCTION KEY 

Function key  can be pressed to display alarms, alarm history, operator messages, or external operator message history, etc.

For alarms, see III-7.1. For alarm history and external operator message history, see III-12.4.13.

For operator messages, refer to the manual provided by the machine tool builder.

12.6 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION

The program number, sequence number, and current CNC status are always displayed on the screen except when the power is turned on, a system alarm occurs.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and displays a warning message.
This section describes the display of the program number, sequence number, and status, and warning messages displayed for incorrect data setting or input/output operation.

12.6.1 Displaying the Program Number and Sequence Number

The number of the program currently selected or currently executed and the current sequence number are indicated in the upper right part as shown below.

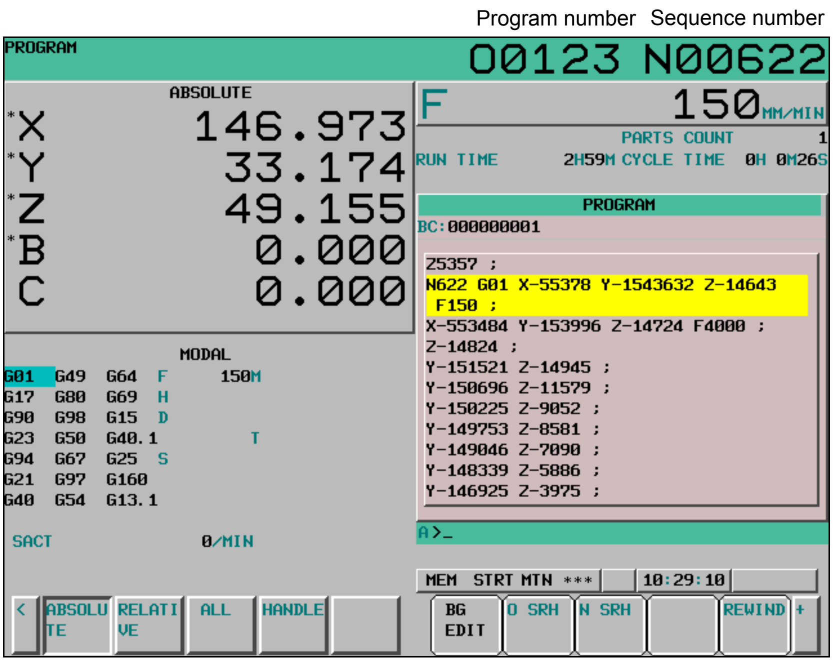


Fig. 12.6.1 (a) Program number and sequence number (10.4-inch)

In the EDIT mode, the number of the program currently edited in the foreground are indicated.

12.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line 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 on the next to last line of the screen. This prevents invalid data setting and input/output errors.

Explanation

- Description of each display

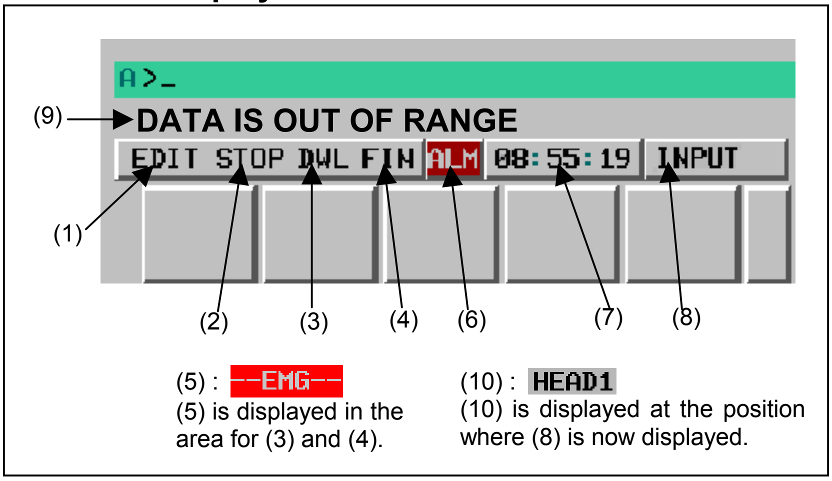


Fig. 12.6.2 (a) Positions of status indications

(1) Current mode

MDI : Manual data input, MDI operation
MEM : Automatic operation (memory operation)
RMT : Automatic operation (DNC operation, or such like)
EDIT : Memory editing
HND : Manual handle feed
JOG : Jog feed
INC : Manual incremental feed
REF : Manual reference position return
**** : Modes other than the above.

(2) Automatic operation status

**** : Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP : Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD : Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT : Automatic operation start-up (The state in which the system operates automatically)

(3) Axis moving status/dwell status

MTN : Indicates that the axis is moving.
DWL : Indicates the dwell state.
*** : Indicates a state other than the above.

(4) State in which an auxiliary function is being executed

FIN : Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)

*** : Indicates a state other than the above.

(5) Emergency stop or reset status

--EMG-- : Indicates emergency stop.(Blinks in reversed display.)

-RESET- : Indicates that the reset signal is being received.

(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.)

NOTE

When the FAN in the α*i* servo amplifier or the α*i* spindle amplifier falls into the warning state or the number of rotations of the FAN built into the stand-alone type control unit decreases, FAN is displayed blinking.
Refer to "Chapter of α*i* SERVO WARNING INTERFACE in MAINTENANCE MANUAL(B-64305EN)" or "Chapter of SERVO WARNING INTERFACE in CONNECTION MANUAL(FUNCTION) (B-64303EN-1)" for details and measures concerning the warning state of FAN in the α*i* servo amplifier.
Refer to "Chapter of WARNING INTERFACE in MAINTENANCE MANUAL (B-64305EN)" or " Chapter of SPINDLE WARNING INTERFACE in CONNECTION MANUAL(FUNCTION) (B-64303EN-1)" for details and measures concerning the warning state of FAN in the α*i* spindle amplifier.
When not corresponding to the above, it is thought that the number of rotations of the FAN built into the stand-alone type control unit decreased.
Therefore, replace the FAN according to "Chapter of REPLACING A FAN UNIT in MAINTENANCE MANUAL(B-64305EN)".

Space : Indicates a state other than the above.

(7) Current time

hh : mm : ss - Hours, minutes, and seconds

(8) Program editing status and program operation status

INPUT : Indicates that data is being input.

OUTPUT : Indicates that data is being output.

SEARCH : Indicates that a search is being performed.

EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)

LSK : Indicates that labels are skipped when data is input.

RSTR : Indicates that the program is being restarted

COMPARE : Indicates that a data comparison is being made.

OFST : Indicates that the tool length compensation amount measurement mode is set (for the M series) or that the tool length compensation amount write mode is set (for the T series).

WOFS : Indicates that the workpiece origin offset amount measurement mode is set.

AICC 1 : Indicates that operation is being performed in the AI contour control mode.
(M series only, parameters Nos.3241 to 3247)

AI APC : Indicates that operation is being performed in the AI advanced preview control mode.
(M series only, parameters Nos.3241 to 3247)

APC : Indicates that operation is being performed in the advanced preview control mode.
(T series only, parameters Nos.3251 to 3257)

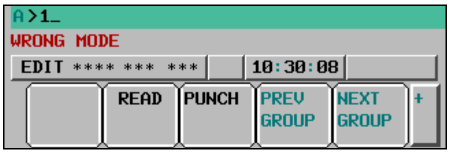
WSFT : Indicates that the workpiece shift amount write mode is set.

Space : Indicates other states.

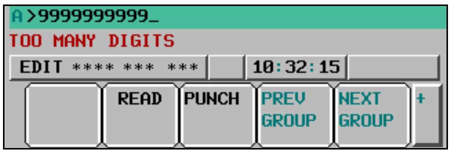
(9) Warning for data setting or input/output operation

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. In this case, the CNC does not accept the setting or input/output operation (retry the operation according to the message).

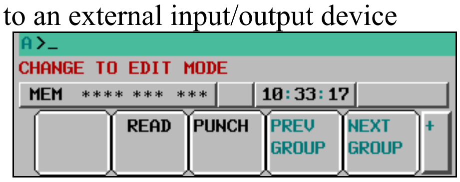
Example 1)
When a parameter is entered



Example 2)
When a parameter is entered



Example 3)
When a parameter is output to an external input/output device



(10) Path name

The number of a path whose status is indicated is displayed.

HEAD1 : Indicates that the status being indicated is for path 1.
Other names can be used depending on the settings of parameters 3141 to 3147. The path name is displayed at the position where (8) is now displayed. When a program is being edited or operated, (8) is displayed depending on the situation.

12.7 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION

Overview

Keeping the same characters displayed in the same positions on the screen for a long time will shorten the life of the LCD.
To prevent this, the CNC screen can be erased. The screen erasure function allows the user to perform a key operation to erase the screen. The automatic screen erasure function erases the screen automatically when there has been no key operation for a parameter-set period of time.

Screen erasure function

When 0 is set in parameter No. 3123, the CNC screen can be erased by pressing the CAN key and any function key (such as POS or PROG) at the same time. The CNC screen can be displayed again by pressing any function key.

Automatic screen erasure function

When there has been no key operation for a time (in minutes) set in parameter No. 3123, the CNC screen is erased automatically. The CNC screen is displayed again by pressing a key.

- Screen erasure by the automatic screen erasure function

If the following conditions are all satisfied for the time (in minutes) set in parameter No. 3123, the CNC screen is erased.
Conditions for automatically erasing the CNC screen

- Parameter No. 3123 ≠ 0
- None of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- No alarm is issued.

- Redisplay of the screen by the automatic screen erasure function

If one of the following conditions is satisfied when the CNC screen is off, the CNC screen is displayed again:
Conditions for redisplaying the CNC screen

- One of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- An alarm is issued.

- Screen erasure by using the CAN key + function key

When a non-zero value is set in parameter No. 3123, the screen is not erased with the CAN key and a function key.

- Set time

Only the time set in parameter No. 3123 for path 1 is valid.

- Alarm in another path

When an alarm is issued in any of the paths, the screen is not erased.

Parameter

3123	Time required before a screen saver is activated

[Input type] Setting input
[Data type] Byte path
[Unit of data] min
[Valid data range] 0 to 127

After a time (in minutes) set in parameter No. 3123 passes without key operation, the NC screen is erased automatically. Pressing a key causes the NC screen to reappear.

> **NOTE**
> 1 Setting 0 disables automatic screen erasure.
> 2 This function cannot be used together with manual screen erasure. If 1 or a larger value is set in this parameter, manual screen erasure is disabled.

12.8 LOAD METER SCREEN

Overview

On the following screens, the servo or spindle load meter and spindle speed meter can be displayed in the display area of the remaining distance and the display area of modal information.

- 8.4-inch LCD: Program check screen
  → Displaying the spindle load meter and spindle speed meter

T

- 10.4-inch LCD: Screen having current position display on its left half
  (Displaying the current position and the program)
  → Switching between the servo load meter and the spindle load meter

12.8.1 For the 8.4-Inch Display Unit

To display the spindle load meter and spindle speed meter on the 8.4-inch display unit, set bit 0 (SMS) of parameter No. 3117 to 1.

Description

The spindle load meter and spindle speed meter are displayed in the display area of the remaining distance and modal information on the program check screen.

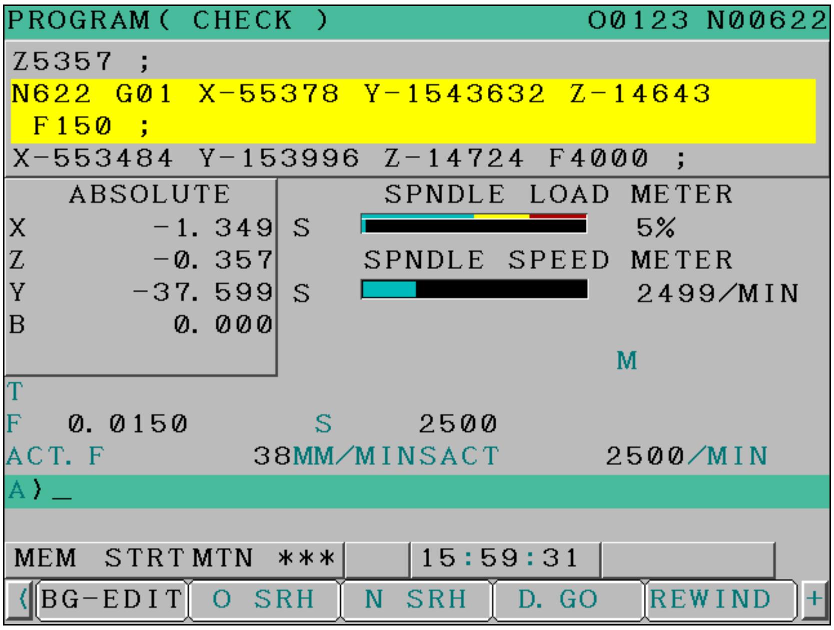
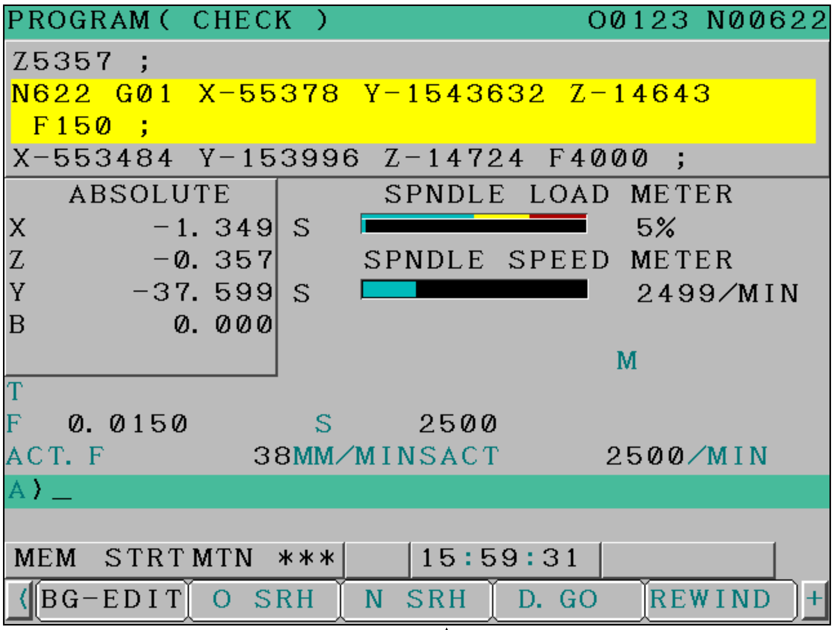


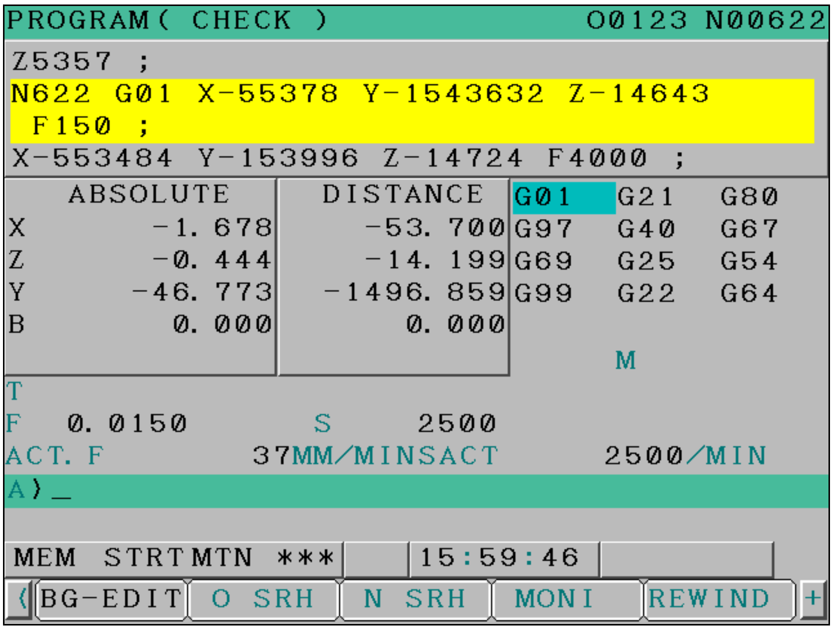
Fig. 12.8.1.(a) Spindle load meter and spindle speed meter

Switching between screens

To display the spindle load meter and spindle speed meter, press soft key [MONI].
To switch between the remaining distance and modal information, press soft key [D. GO].



Switching with [MONI]/[D. GO]



> **NOTE**
> To use spindle load meter display and spindle speed meter display, the serial spindle is necessary.

T

12.8.2 For the 10.4-Inch Display Unit

To display the servo load meter and spindle load meter on the screen of the 10.4-inch display unit on the left half of which the current position is displayed, set bit 7 (PLD) of parameter No. 3192 to 1.

Screen layout

Either the servo load meter or the spindle load meter is displayed in the position of remaining distance display of all position display.

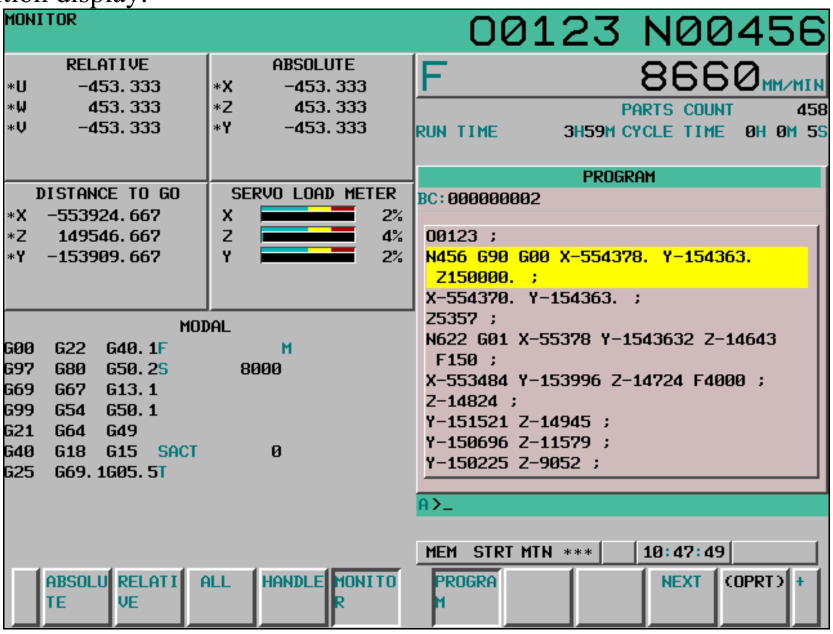


Fig. 12.8.2.(a) Servo load meter

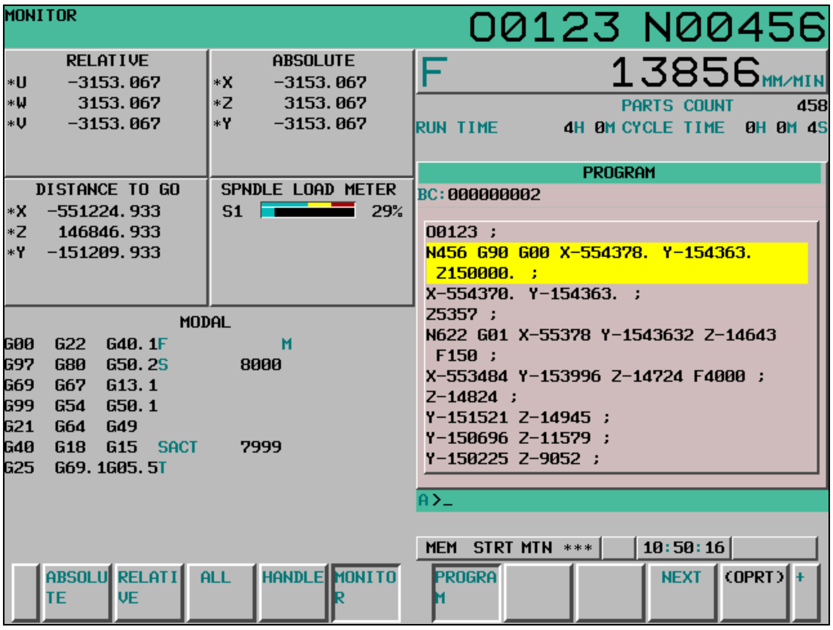
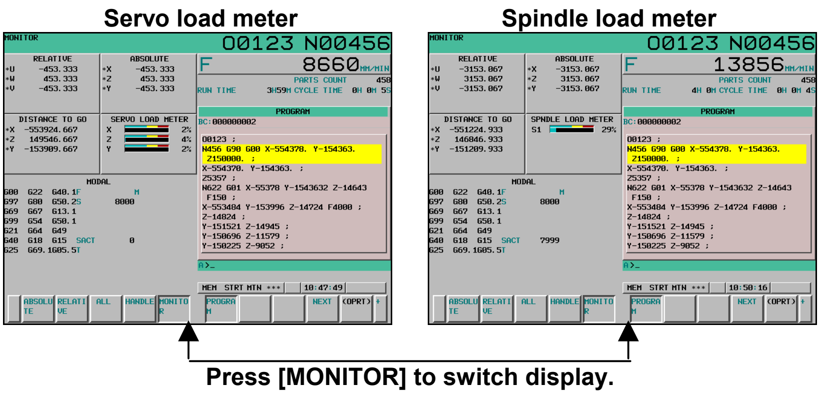


Fig. 12.8.2 (b) Spindle load meter

Switching between screens

To display the servo load meter or spindle load meter, press soft key [MONITOR] at the bottom of the screen. The default is the servo load meter. Pressing soft key [MONITOR] switches between the servo load meter and the spindle load meter.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3117								SMS

[Input type] Parameter input
[Data type] Bit path

 # 0 SMS On the program check screen of the 8.4-inch display unit, the function for displaying the spindle load meter and spindle speed meter in the remaining movement amount display position and modal information display position is:

0: Disabled.
1: Enabled.

T

	#7	#6	#5	#4	#3	#2	#1	#0
3192	PLD							

[Input type] Parameter input
[Data type] Bit

 # 7 PLD On the screen of the 10.4-inch display unit where positional display is performed on the left half, the function for displaying the servo axis load meter and spindle load meter is:
0: Disabled.
1: Enabled.

13140	First character in spindle load meter display
13141	Second character in spindle load meter display

[Input type] Setting input
[Data type] Byte spindle
[Valid data range] These parameters set character codes to set the name of each spindle that appears in spindle load meter display. Any character string consisting of numeric characters, alphabetical characters, katakana characters, and symbols with a maximum length of two characters can be displayed as a spindle name.
If 0 is set, the following is displayed:
1st spindle S1
2nd spindle S2
3rd spindle S3

13 GRAPHIC FUNCTION

Chapter 13, "GRAPHIC FUNCTION", consists of the following sections:

13.1 GRAPHIC DISPLAY

The graphic display functions enable drawing of the tool path of the program currently used for machining.
These functions are intended to display the movement of the tool during automatic operation or during manual operation.

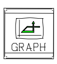
T

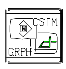
In case of a two-path lathe system, the tool paths of two turrets are drawn at the same time in the right and left views on one screen.

This enables the operator to check the progress of machining and the current tool position.
These functions include the following:

- 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.

13.1.1 Graphic Parameter Screen

Press the function key 

(or 

when a small MDI unit is used) and then press the [PARAM] soft key to display the GRAPHIC PARAMETER screen.
On this screen, make necessary settings for drawing of a tool path.
The graphic parameter screen consists of three pages.

M

- Graphic parameter screen (first page)

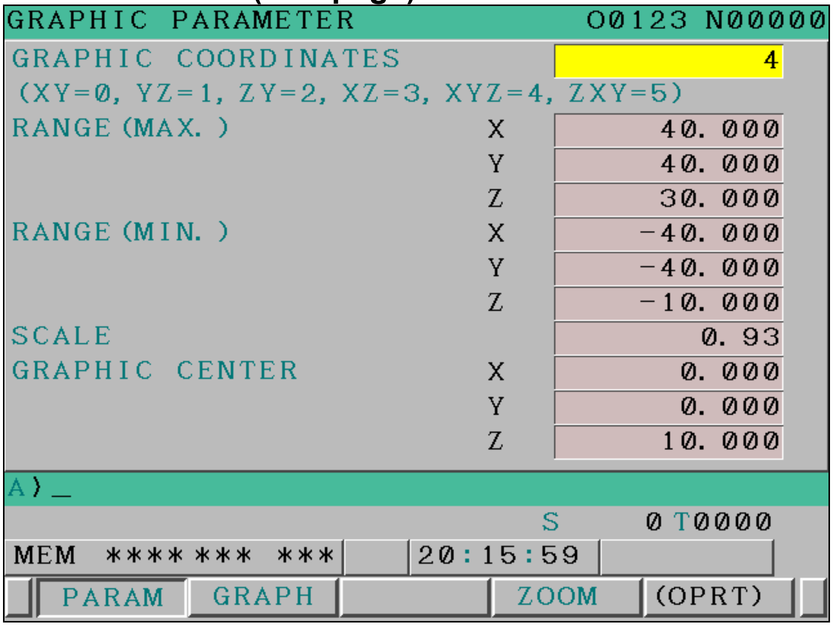


Fig. 13.1.1 (a) Graphic parameter screen (first page) (8.4-inch LCD)

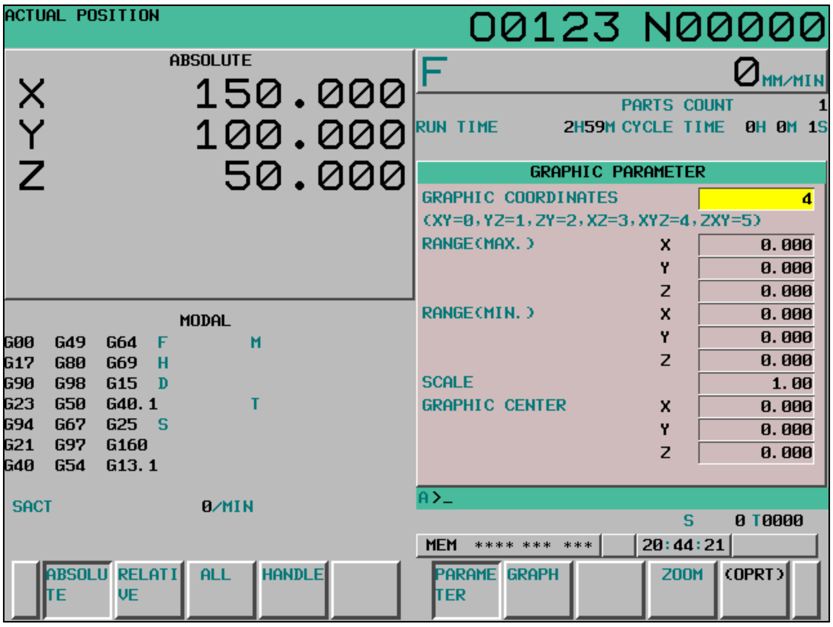


Fig. 13.1.1 (b) Graphic parameter screen (first page) (10.4.4-inch LCD)

On graphic parameter screen (first page) , a graphic coordinate system, graphic range, and so forth are set. In the setting of a graphic coordinate system, the coordinate axes and axis names of the set coordinate system are displayed. When a three-dimensional coordinate system is displayed, a rotation angle is also displayed.

The graphic range can be set by one of the two methods: one method by setting the maximum and minimum values of coordinate positions and the other method by setting the graphic scale factor and graphic range center.

- Graphic parameter screen (second page)

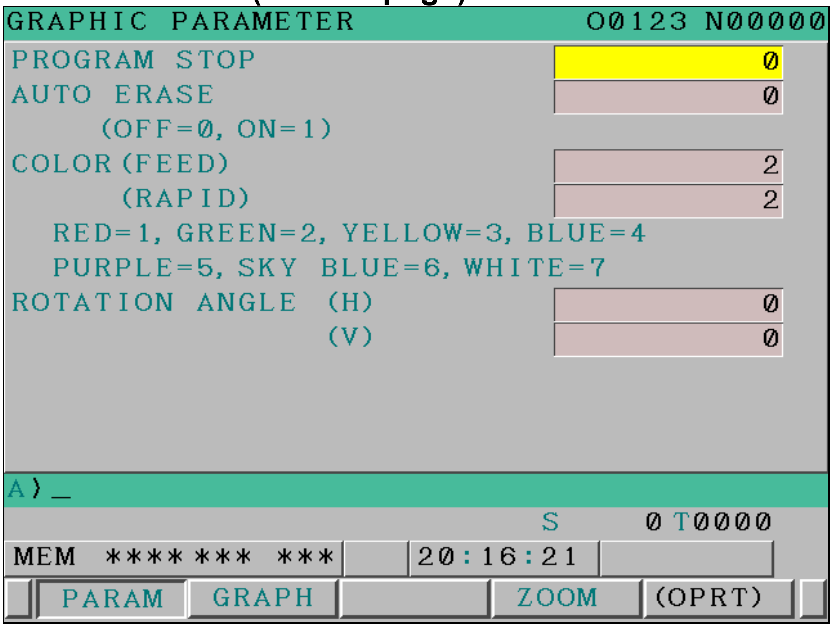


Fig. 13.1.1 (c) Graphic parameter screen (second page) (8.4-inch LCD)

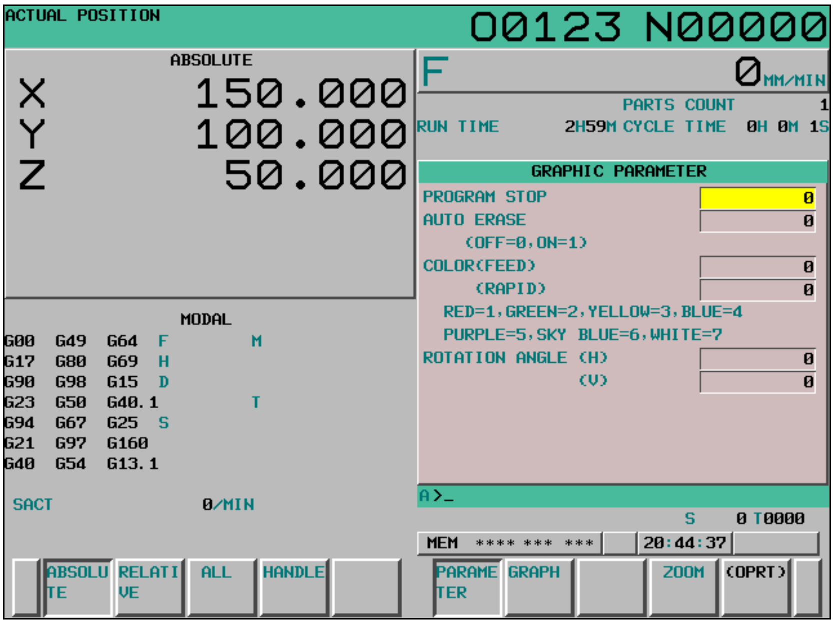


Fig. 13.1.1 (d) Graphic parameter screen (second page) (10..4-inch LCD)

On the graphic parameter screen (second page), a drawing end block, automatic erasure, graphic colors, and rotation angles are set.

- Graphic parameter screen (third page)

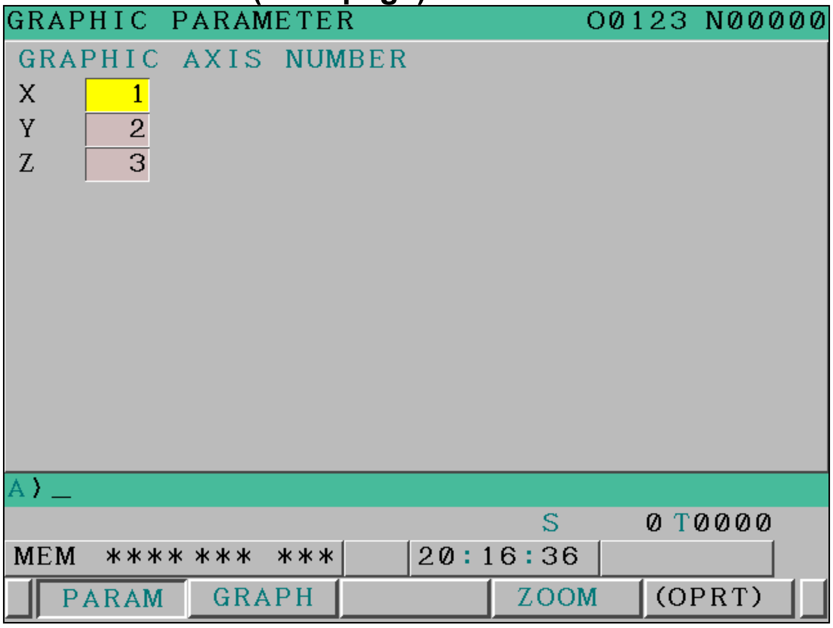


Fig. 13.1.1 (e) Graphic parameter screen (third page) (8.4-inch LCD)

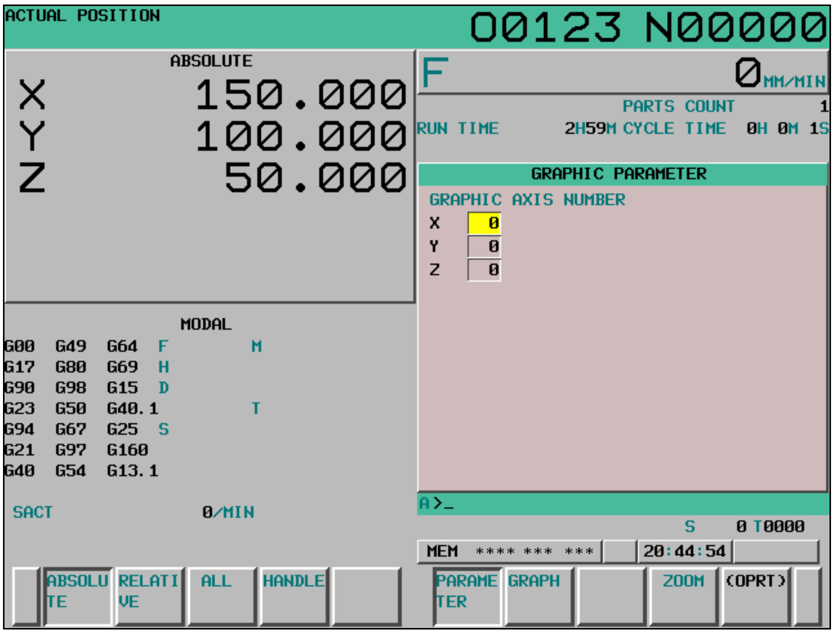


Fig. 13.1.1 (f) Graphic parameter screen (third page) (10.4-inch LCD)

On graphic parameter screen (third page), coordinate axes to be used for drawing are set.

T

- **Graphic parameter screen (first page)**

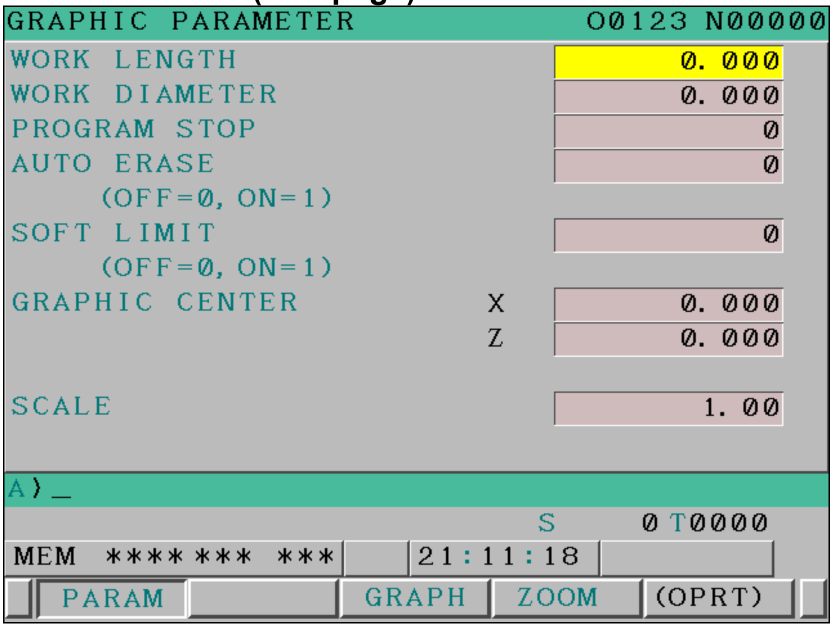


Fig. 13.1.1 (g) Graphic parameter screen (first page) (8.4-inch LCD)

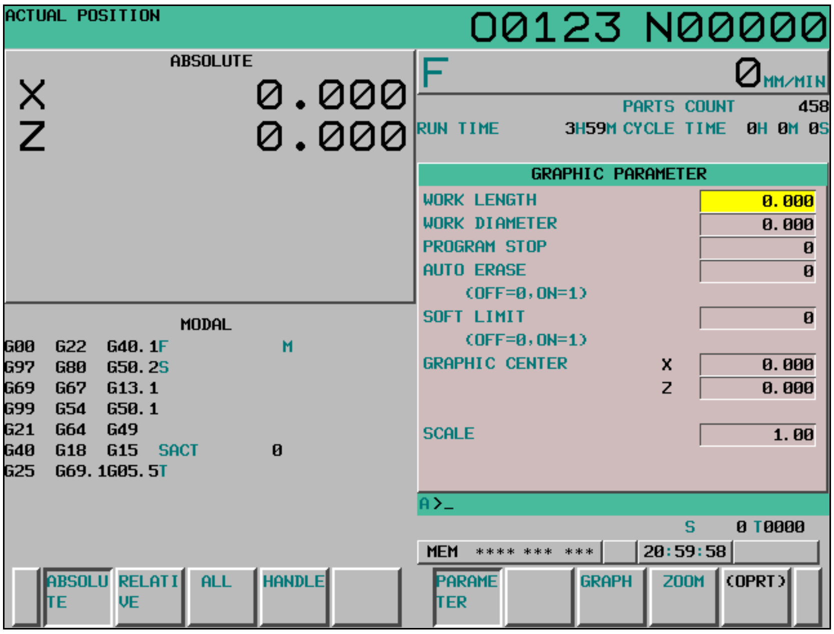


Fig. 13.1.1 (h) Graphic parameter screen (first page) (10.4-inch LCD)

On the graphic parameter screen (first page), blank dimensions (length and diameter), drawing end block, automatic erasure, soft limit, graphic range, and so forth are set.

The graphic range can be set by one of the two methods: one method by setting the blank dimensions (length and diameter) and the other method by setting the graphic scale factor and graphic range center.

- Graphic parameter screen (second page)

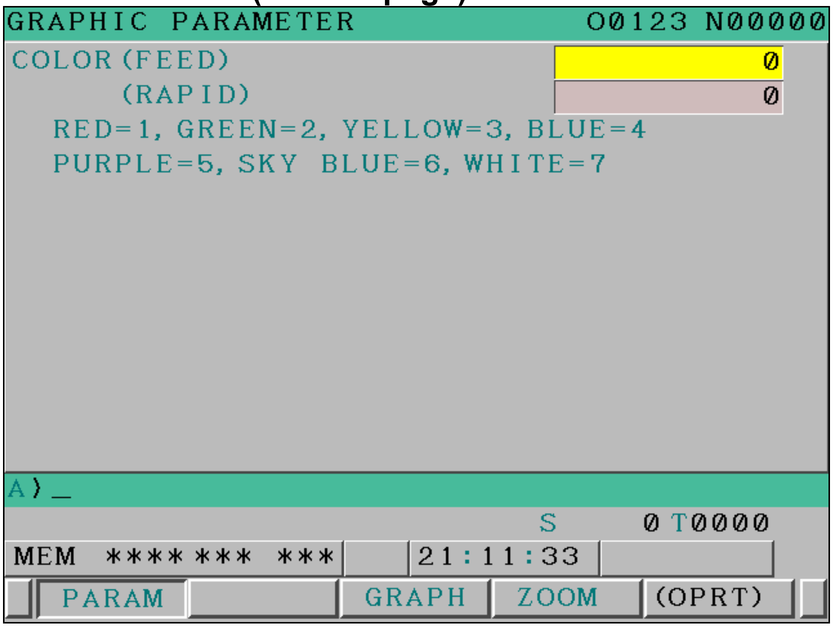


Fig. 13.1.1 (i) Graphic parameter screen (second page) (8.4-inch LCD)

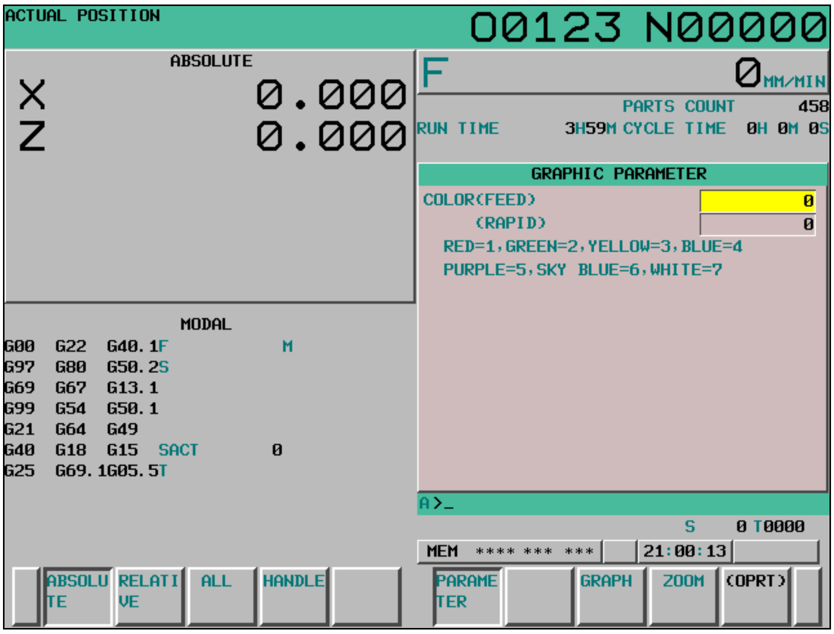


Fig. 13.1.1 (j) Graphic parameter screen (second page) (10.4-inch LCD)

On graphic parameter screen (second page), graphic colors are set.

- Graphic parameter screen (third page)

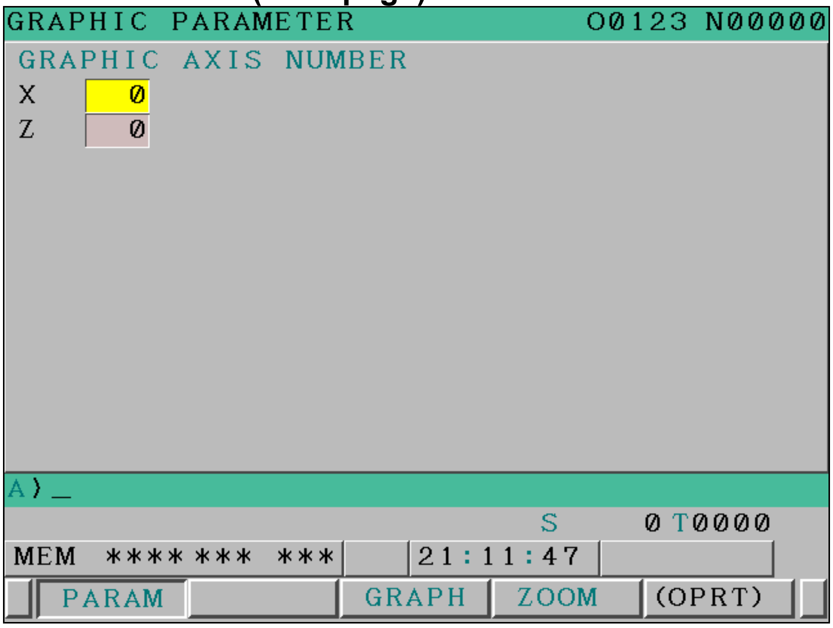


Fig. 13.1.1 (k) Graphic parameter screen (third page) (8.4-inch LCD)

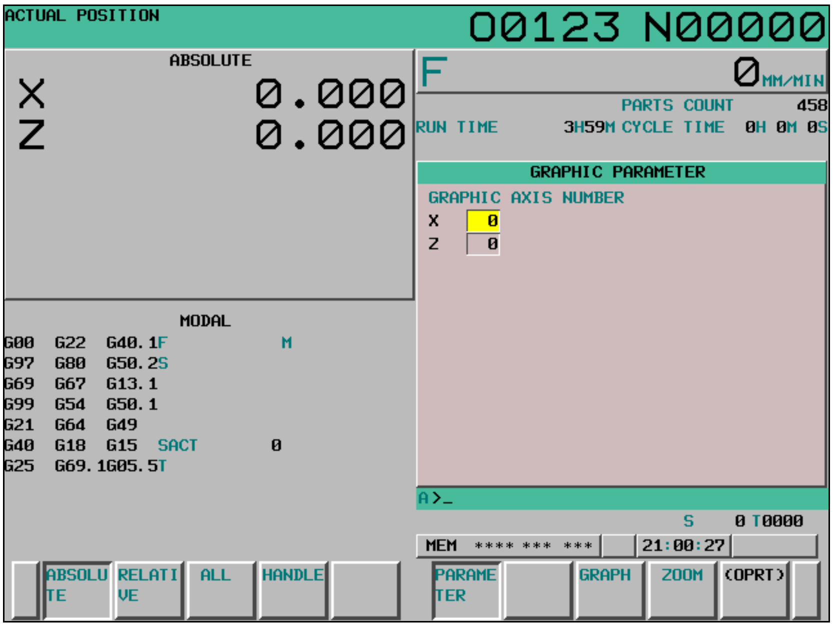


Fig. 13.1.1 (l) Graphic parameter screen (third page) (10.4-inch LCD)

On graphic parameter screen (third page), coordinate axes to be used for drawing are set.

Graphic parameter setting: Operating procedure

Procedure

To display the soft keys for input operations on the graphic parameter screen, do the following:

1 Press the function key (or when a small MDI unit is used) to display the GRAPHIC PARAMETER screen.

2 Press the [(OPRT)] soft key or a numeric key.

- Moving the cursor

The cursor can be moved to a desired parameter by the page key or and the cursor key , , , or .

With the cursor keys, however, you cannot move from page 1 or 2 to page 3.

- Input of settings (absolute input)

Method 1

(1) Key in a value to be set.

(2) Press the [INPUT] soft key.

Method 2

(1) Key in a value to be set.

(2) Press the key.

- Input of settings (incremental input)

Method 1

(1) Key in a value to be incremented to or decremented from the current setting.

(2) Press the [+INPUT] soft key.

Setting standard values

M

After you change the graphic range center and scale factor in a graphic range enlargement/reduction operation or input operation, you can press the [NORMAL] soft key to restore these values to the settings obtained by automatic operation based on the maximum and minimum values in the graphic range.

T

After you change the graphic range center and scale factor in a graphic range enlargement/reduction operation or input operation, you can press the [NORMAL] soft key to restore these values to the settings obtained by automatic operation based on the blank length and diameter.

Explanation

For tool path drawing, a graphic coordinate system and graphic range need to be set on the graphic parameter screen.

The graphic parameters to be set on the graphic parameter screen are described below.

If you change any of the graphic parameters and switch to the PATH GRAPHIC screen, the tool path already drawn is erased.

The graphic parameters are preserved even if the power is turned off.

- Graphic coordinate system

M

Select a desired graphic coordinate system for tool path drawing then set the corresponding number.

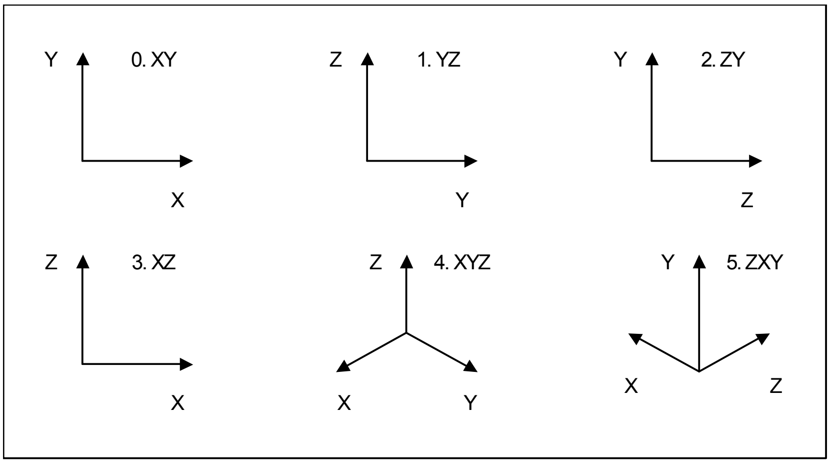


Fig. 13.1.1 (m) Graphic coordinate system

T

Select a desired graphic coordinate system for tool path drawing from the following and set its number in parameter No. 6510.

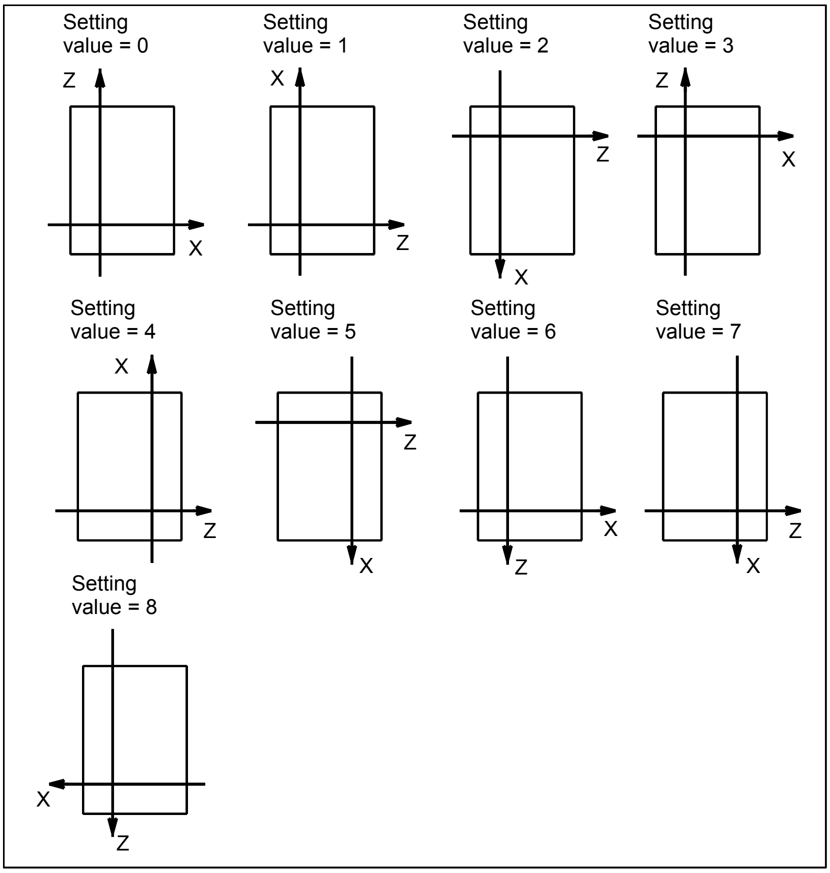


Fig. 13.1.1 (n) Graphic coordinate system

T

For a two-path lathe system with one spindle and two turrets (bit 1 (SPC) of parameter No. 6500 is 1), select a desired coordinate system from the following and set its number in parameter No. 6509.

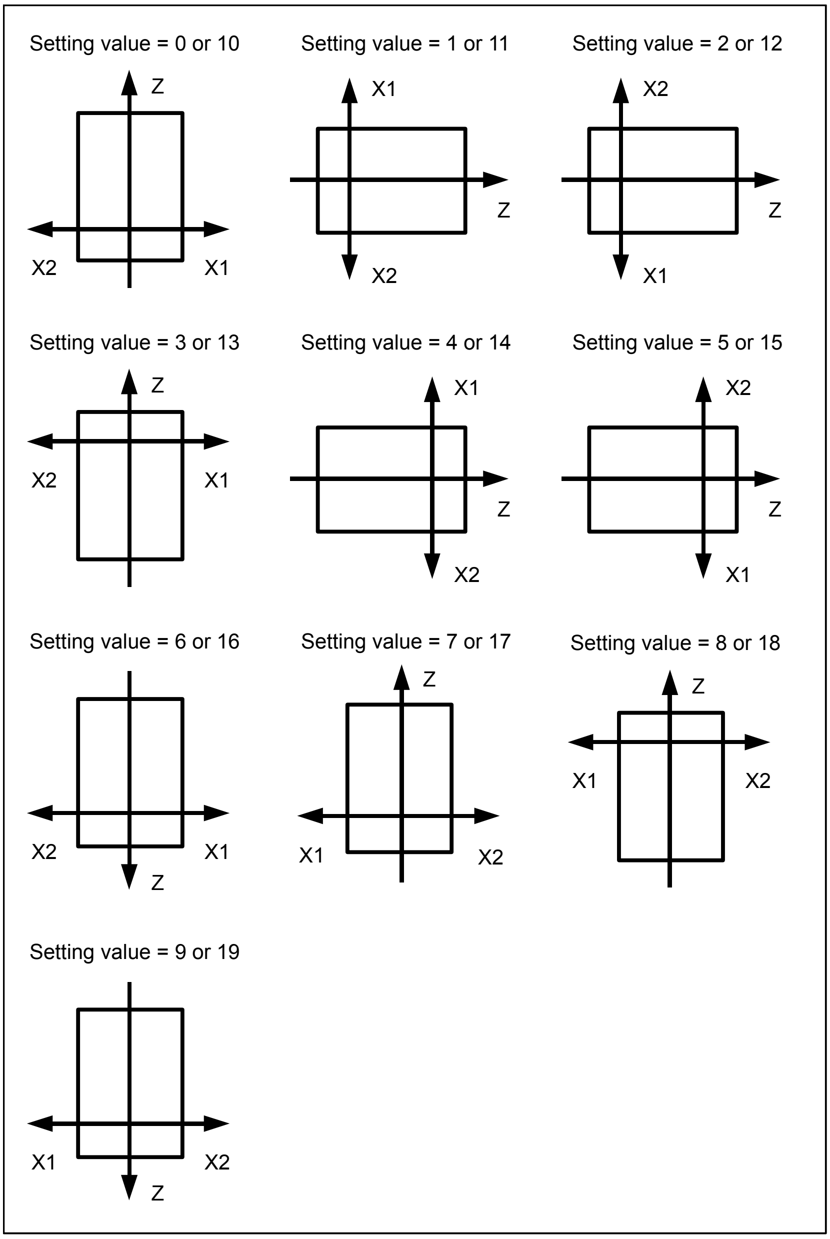


Fig. 13.1.1 (o) Graphic coordinate system (one spindle and two turrets)

> **NOTE**
> The above graphic coordinate system for one spindle and two turrets is enabled when two paths are displayed at the same time.

M

- Graphic range setting

For tool path drawing in a drawing area on the PATH GRAPHIC screen, set a graphic range. Two methods are available:

1. Method by setting the maximum and minimum values of coordinate positions
2. Method by setting the graphic range center and scale factor

Whether method 1 or method 2 is used is determined by which parameters were set most recently. A set graphic range is preserved even if the power is turned off.

1. Method by setting the maximum and minimum values of coordinate positions

Set the desired graphic range by the maximum and minimum coordinate values in the workpiece coordinate system.
Drawing is performed so that the entire set range falls within the drawing area.

The graphic range center and scale factor are automatically calculated from the set maximum and minimum values, and then the settings of the graphic range center and scale factor on the graphic parameter screen are updated.
Even when the scale factor is automatically determined, the value is clamped to the range of 0.01 to 100.
The maximum value must be larger than the minimum value.

> **NOTE**
> When you set the graphic parameters for the maximum and minimum values in the graphic range, the graphic parameters for the graphic range center and scale factor are automatically updated. When you change the graphic range center and scale factor, however, the maximum and minimum values in the graphic range are not updated.

2. Method by setting the graphic range center and scale factor

Set the central coordinates of the drawing area by coordinate values in the workpiece coordinate system.
Next, set the scale factor to make the graphic range fall within the drawing area.
Set a value of 0.01 to 100 as the scale factor.
With a smaller scale factor, you can perform tool path drawing in a wider range.
With a larger scale factor, you can perform drawing while enlarging the vicinity of the graphic range center.

T

- Graphic range setting

For tool path drawing in a drawing area on the PATH GRAPHIC screen, set a graphic range. Two methods are available:

1. Method by setting the blank length and diameter
2. Method by setting the graphic range center and scale factor

Whether method 1 or method 2 is used is determined by which parameters were set most recently. A set graphic range is preserved even if the power is turned off.

1. Method by setting the blank length and diameter

Set the length and diameter of a blank in a machining program for drawing.
Drawing is performed so that the entire set range falls within the drawing area.

The graphic range center and scale factor are automatically calculated from the set the length and diameter of a blank, and then the settings of the graphic range center and scale factor on the graphic parameter screen are updated.
Even when the scale factor is automatically determined, the value is clamped to the range of 0.01 to 100.

> **NOTE**
> When you set the graphic parameters for the blank length and diameter, the graphic range center and scale factor are automatically updated. When you change the graphic range center and scale factor, however, the blank length and diameter are not updated.

2. Method by setting the graphic range center and scale factor

Set the central coordinates of the drawing area by coordinate values in the workpiece coordinate system.
Next, set the scale factor to make the graphic range fall within the drawing area.
Set a value of 0.01 to 100 as the scale factor.
With a smaller scale factor, you can perform tool path drawing in a wider range.
With a larger scale factor, you can perform drawing while enlarging the vicinity of the graphic range center.

- Drawing end block

To perform drawing for a part of a program, set the sequence number of a block at which to end the drawing. Once the drawing has completed, this value is automatically canceled and change to -1.

- Automatic erasure

The previous graphic is automatically erased before drawing is started.
1 : The previous graphic is automatically erased immediately before drawing is started.
0 : The previous graphic is not automatically erased.

T

- Soft limit

When 1 is set, an area of stored stroke limit 1 is drawn with a dashed-two dotted line.

- Graphic color

Set a graphic color number for a tool path for each of cutting feed and rapid traverse.

1: Red	2: Green	3: Yellow	4: Blue
5: Purple	6: Sky blue	7: White	

> **NOTE**
> For any value other than the above settings, the graphic color of a tool path is green.

M

- Horizontal rotation angle

When a three-dimensional graphic coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with the horizontal plane used as the rotation plane. Set a rotation angle from -360° to +360°.

In Fig. 13.1.1 (p) below, the graphic coordinate system XYZ is converted to X”Y”Z” by the following settings:

Horizontal rotation angle: 210°

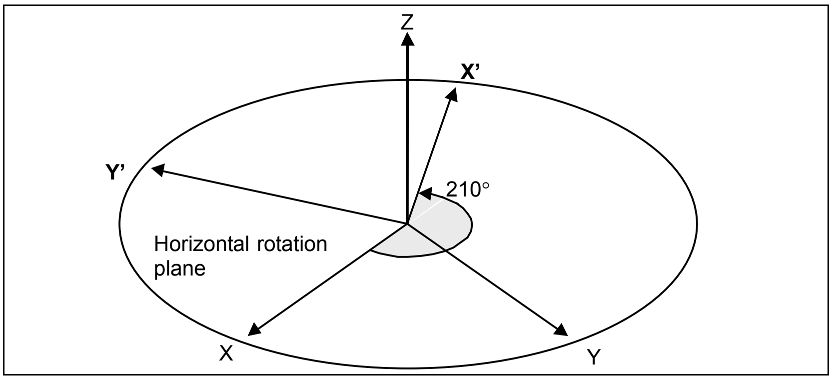


Fig. 13.1.1 (p) Coordinate system rotation in horizontal direction

- Vertical rotation angle

When a three-dimensional coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with an axis on the horizontal plane specified as a vertical rotation axis. Set a rotation angle from -360° to +360°.

In Fig. 13.1.1 (q) below, the graphic coordinate system XYZ is converted to X'Y'Z' by the following settings:

Vertical rotation angle: 20°

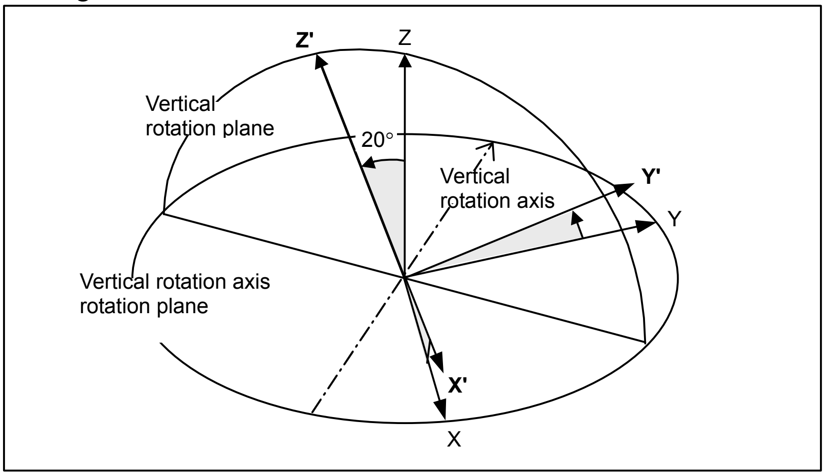


Fig. 13.1.1 (q) Coordinate system rotation in vertical direction

- Graphic axis number

Set which controlled axis to be assigned to which graphic axis.
For each controlled axis, set one of the following graphic axis numbers:

First graphic axis:	1
Second graphic axis:	2
Third graphic axis:	3
Axis not used for drawing:	0

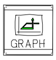
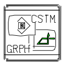
NOTE

1 When 0 is set for all controlled axes, it is assumed that 1, 2, and 3 are set sequentially for the first to the third controlled axes.

2 With the T series, a tool path is drawn along the first and second graphic axes. No tool path is drawn along the third graphic axis.

13.1.2 Path Graphic Screen

Explanation

Press the function key  (or  when a small MDI unit is used) and then press the [GRAPH] soft key to display the PATH GRAPHIC screen.

The PATH GRAPHIC screen mainly consists of three parts.

- Drawing area part for tool path drawing
- Part for display of machining information including a tool position
- Part for display of the graphic coordinate system

M

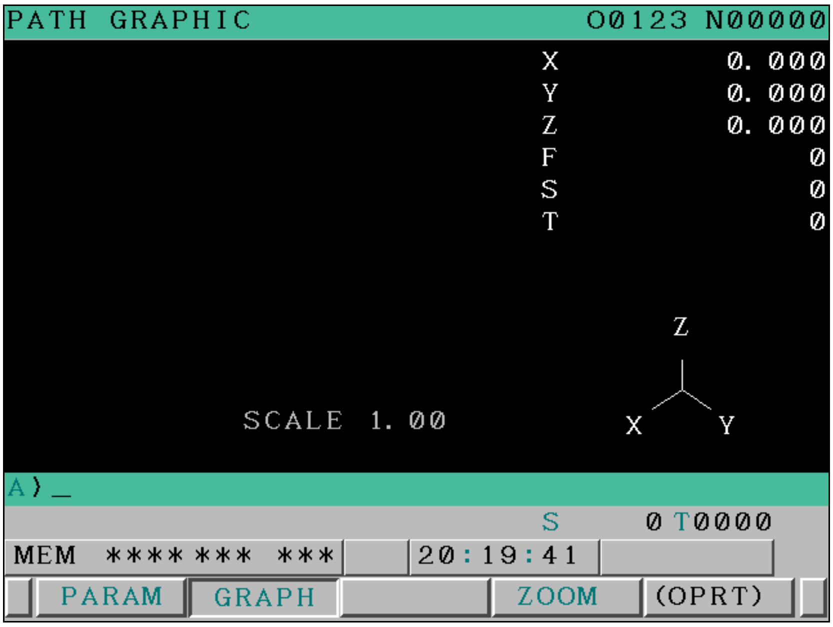


Fig. 13.1.2 (a) PATH GRAPHIC screen (8.4-inch LCD)

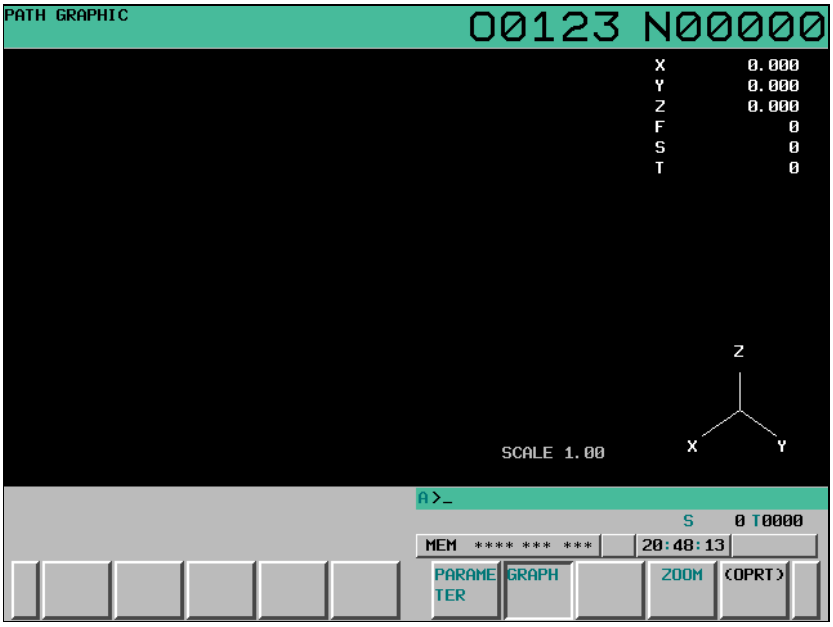


Fig. 13.1.2 (b) PATH GRAPHIC screen (10.4-inch LCD)

T

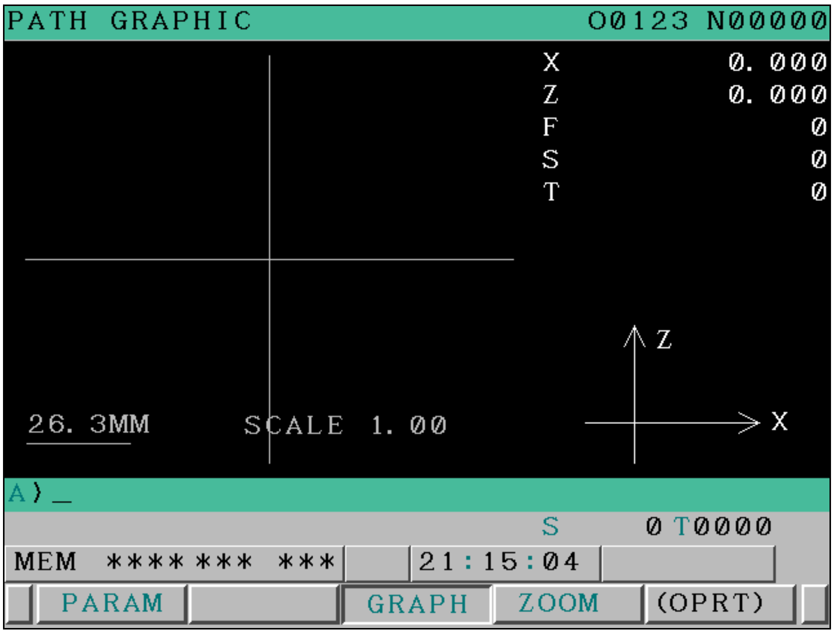


Fig. 13.1.2 (c) PATH GRAPHIC screen (8.4-inch LCD)

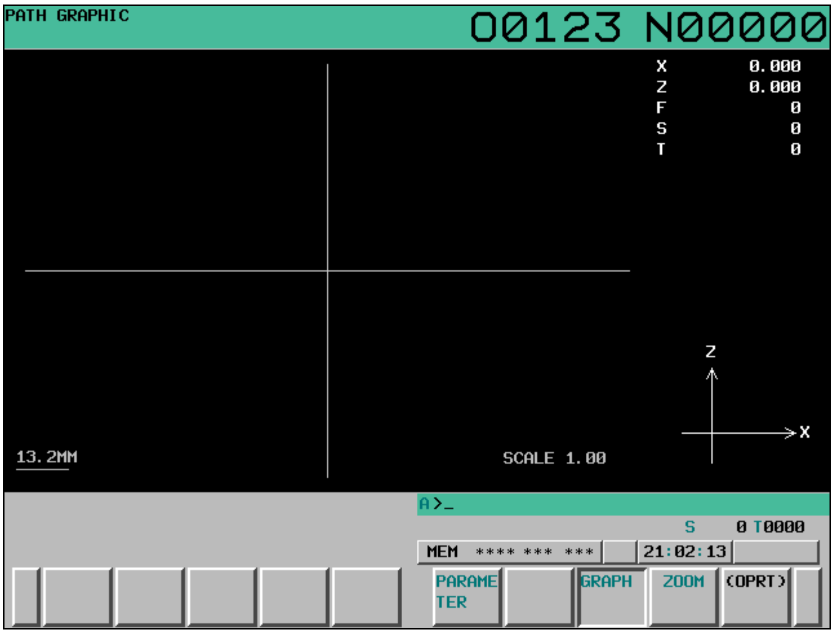


Fig. 13.1.2 (d) PATH GRAPHIC screen (10.4-inch LCD)

T

- Screen for simultaneously displaying two paths (Two-path lathe system)

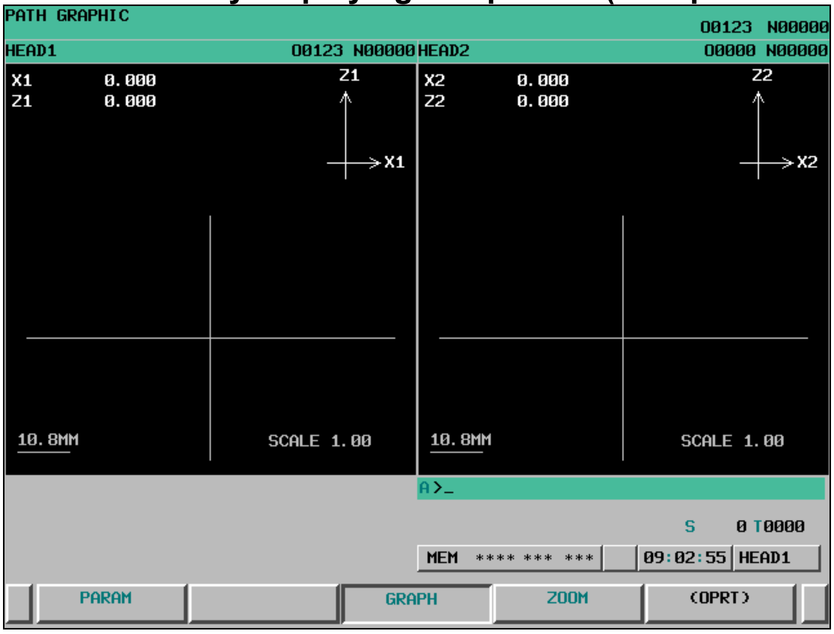


Fig. 13.1.2 (e) PATH GRAPHIC screen (8.4-inch LCD)

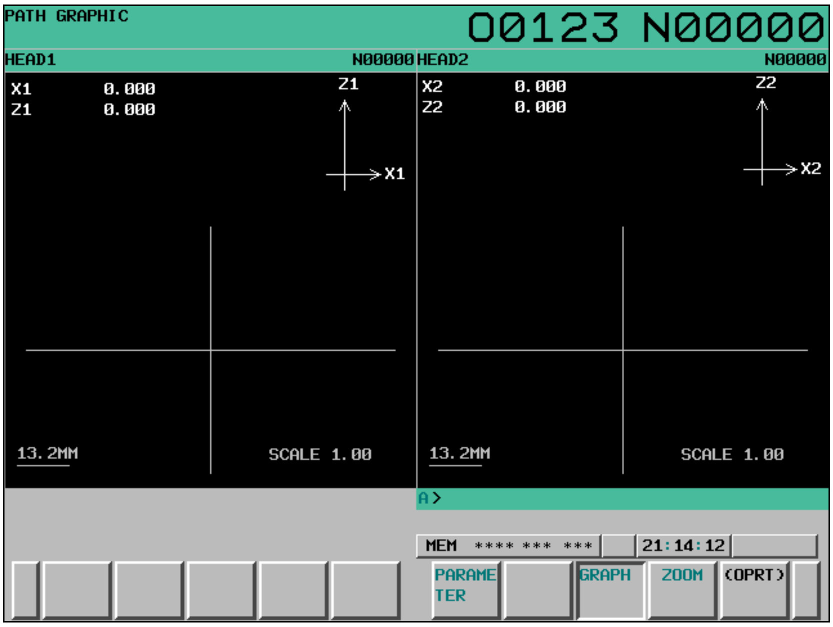


Fig. 13.1.2 (f) PATH GRAPHIC screen (10.4-inch LCD)

- Screen for displaying a single path

When you set bit 2 (DOP) of parameter No. 3193 to disable the simultaneous display of two paths, each path is displayed in the same way as on the screen for one-path system.

- Tool path

In the graphic coordinate system set with a graphic parameter, a tool path is drawn in the workpiece coordinate system. A distinction can be made between travel paths for rapid traverse and cutting feed by setting graphic colors.

Even if a tool position changes discontinuously due to origin setting and switching of the workpiece coordinate system, the tool path is drawn assuming that the tool moves.
Tool path drawing continues even after you switch to another screen.

> **NOTE**
> Drawing does not continue and a drawn path is not retained when:
> - You switch to the conversational macro screen.
> - You switch to a screen displayed by the C language executor.
> - You switch to the Manual Guide *i* screen.
> - You start or stop the CNC screen display function.

- Machining information

On the right side of the screen, you see a position in the workpiece coordinate system as well as the feedrate (F), spindle speed (S), and tool number (T).

> **NOTE**
> Up to three graphic axes are supported for the M series and up to two graphic axes are supported for the T series.

- Graphic coordinate system

The coordinate axes and axis names are displayed in the lower right corner of the screen.

- Scale factor and dimensions

When the graphic coordinate system is on a plane, the scale factor for a graphic range and the dimensional values of measures are displayed.

T

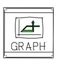
- Coordinate axes in the workpiece coordinate system

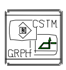
In a drawing area, graphic axes in the workpiece coordinate system are displayed.
Note that you see the position of the workpiece coordinate system set at the time when drawing starts. During automatic operation, the positions of the coordinate axes are not changed even if the workpiece coordinate system is changed.

Procedure for drawing a path on the path graphic screen

Procedure

- Starting drawing

1 Press the function key 

(or 

when a small MDI unit is used) and set necessary graphic parameters on the GRAPHIC PARAMETER screen.
2 Press the [GRAPH] soft key to display the PATH GRAPHIC screen.
3 Start automatic or manual operation. Movements of the machine are drawn on the screen.

> **NOTE**
> 1 Set the machine lock state to perform drawing only without moving the tool.
> 2 When the feedrate is high, the tool path may not be drawn correctly. In such a case, decrease the feedrate by performing, for example, a dry run.

- Stopping drawing

1 Press the function key (or when a small MDI unit is used).
If the PATH GRAPHIC screen does not appear, press the [GRAPH] soft key to display the screen.
2 Tool path drawing stops when automatic operation has completed or is stopped halfway.

- Erasure of the drawing

Press the [CLEAR] soft key. The tool path drawn so far is erased.

Procedure for enlarging or reducing a drawn path on the path graphic screen

On the tool path graphic screen, you can move the center position of the tool path drawing or enlarge the tool path drawing while viewing the drawn tool path.
If any of these operations is executed, the tool path already drawn is cleared.

Procedure

1 Press the function key (or when a small MDI unit is used).
2 Press the [GRAPH] soft key to display the PATH GRAPHIC screen, and then draw a tool path.
3 Press the [LARGE] soft key

- Procedure for changing the graphic range by setting a graphic center and magnification

The center position of drawing can be moved. At the same time, the scale can also be changed. So, the tool path can be enlarged or reduced at a desired new center position.

4 After step 3 described above, press the [CENTER] soft key.
A yellow cursor appears at the center of the screen, and the soft key display is changed.
5 Move the yellow cursor to a new graphic center position by using the cursor key ⬆, ⬇, ⬅, or ➡.
6 When changing the scale, key in a value from 0.01 to 100 (magnification) then press the [INPUT] soft key. An input value is displayed at "SCALE" in the lower-right corner of the screen.
When you press the [+INPUT] soft key, the current magnification is incremented by an input value.
7 Press the [EXEC] soft key to end the operation.
After this step, the setting for graphic movement is effective to enable drawing with the new setting.

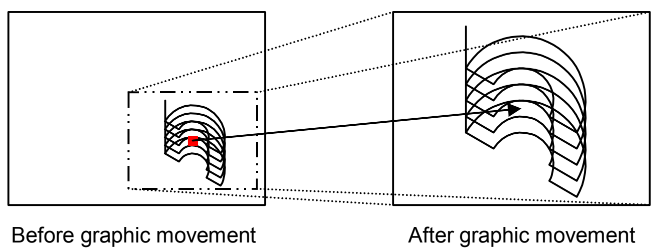


Fig. 13.1.2 (g) Graphic movement (magnification = 2.00)

- Procedure for changing the graphic range with a rectangle

A tool path can be drawn by enlarging a specified rectangular area.

4 After step 3 described above, press the [AREA] soft key.
Two cursors, one in red and the other in yellow, appear at the center of the screen, and the soft key display is changed.

5 Move the yellow cursor by using the cursor key [↑], [↓], [←], or [→]. The cursor to be moved can be switched by pressing the [HI/LO] soft key.
Move the two cursors to the diagonal points of a new rectangular graphic range. A tool path is drawn next time so that the drawn tool path is contained in this rectangular range.

6 Press the [EXEC] soft key to end the operation.
After this step, the setting made in the above steps is effective to enable drawing in the new graphic range.

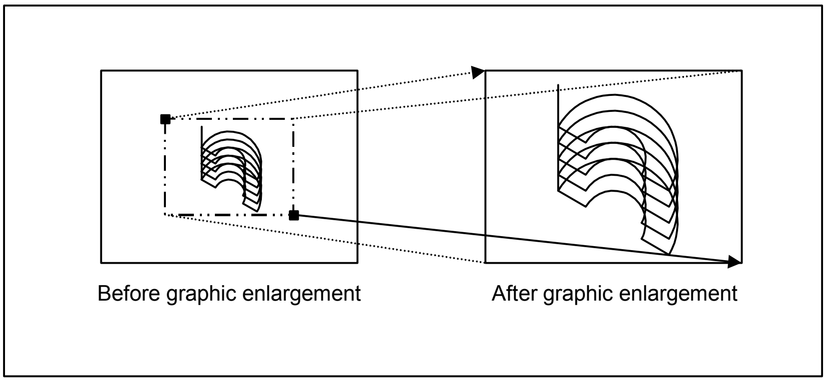


Fig. 13.1.2 (h) Graphic enlargement

> **NOTE**
> 1 To stop an enlargement/reduction operation, press the [CANCEL] soft key.
> 2 Even if you perform an enlargement/reduction operation, the tool path already drawn on the screen is neither moved nor enlarged. The enlargement/reduction setting is effective next time drawing is performed.

13.2 DYNAMIC GRAPHIC DISPLAY (M SERIES)

Overview

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.
This function is distinguished from automatic program operation as indicated by the following terminology:

Automatic operation	Operation performed for actual machining
Background operation	Virtual operation performed for drawing

13.2.1 Path Drawing

Overview

The following tool path drawing screens are used to make various settings and execute drawing:

- PATH GRAPHIC (SETTING) screen
  This screen is used to set data needed for tool path drawing.

- PATH GRAPHIC (EXECUTION) screen
  This screen is used for tool path drawing.
  Enlarge or reduce a graphic range, or rotate the graphic coordinate system.

- PATH GRAPHIC (POSITION) screen
  This screen is used to indicate the current tool position during automatic operation by showing the graphic cursor on the path drawn on the PATH GRAPHIC (EXECUTION) screen.

13.2.1.1 PATH GRAPHIC (SETTING) screen

This screen is used to set graphic parameters needed for tool path drawing.
Data set using this screen is made valid by displaying the PATH GRAPHIC (EXECUTION) screen or executing drawing. If a tool path is already drawn, the path is erased.
Graphic parameter data once set is preserved even if the power is turned off.

Path Graphic (Setting) screen: Operating procedure

Procedure

1 Press the function key GRAPH (or CSTM GRPH when a small MDI unit is used) to display the PATH GRAPHIC (SETTING-1) screen.

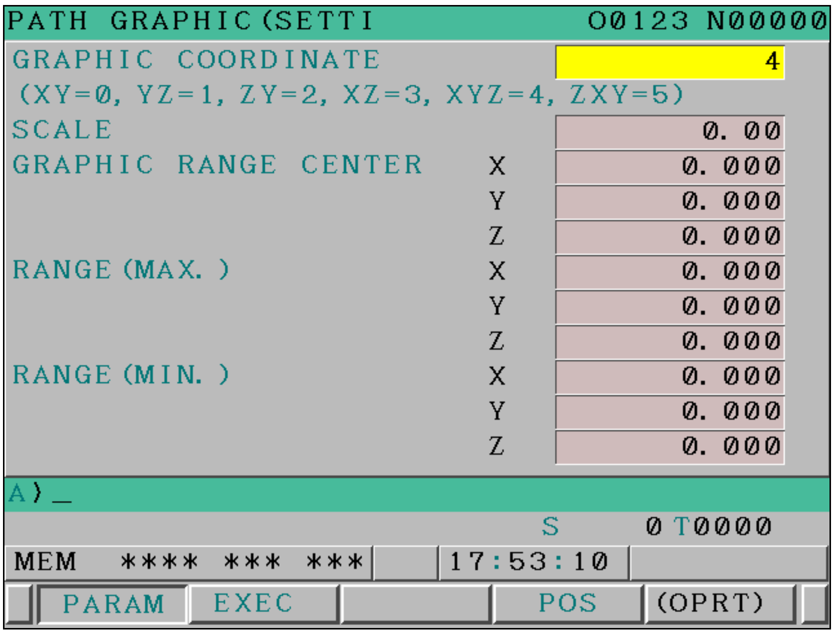


Fig. 13.2.1.1 (a) PATH GRAPHIC (SETTING-1) screen (first page) (8.4-inch LCD)

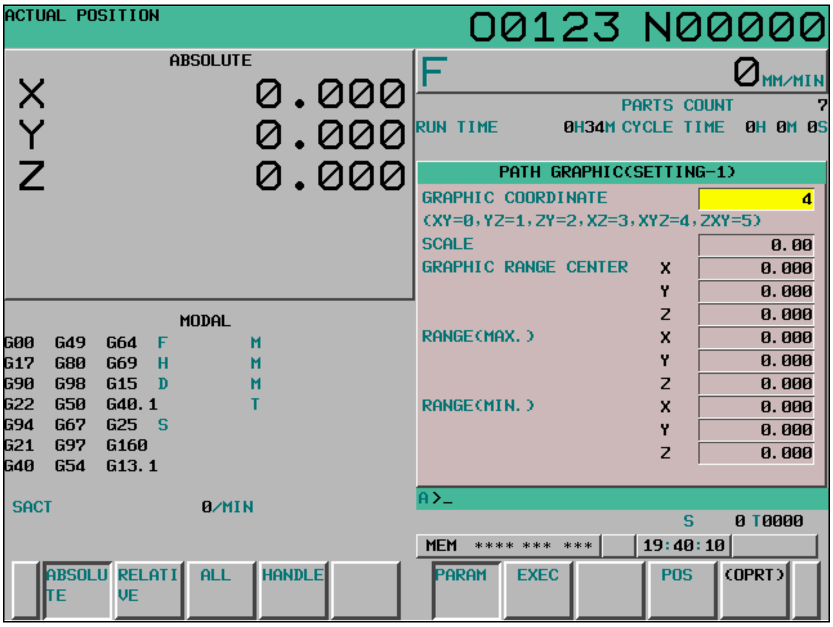


Fig. 13.2.1.1 (b) PATH GRAPHIC (SETTING-1) screen (first page) (10.4-inch LCD)

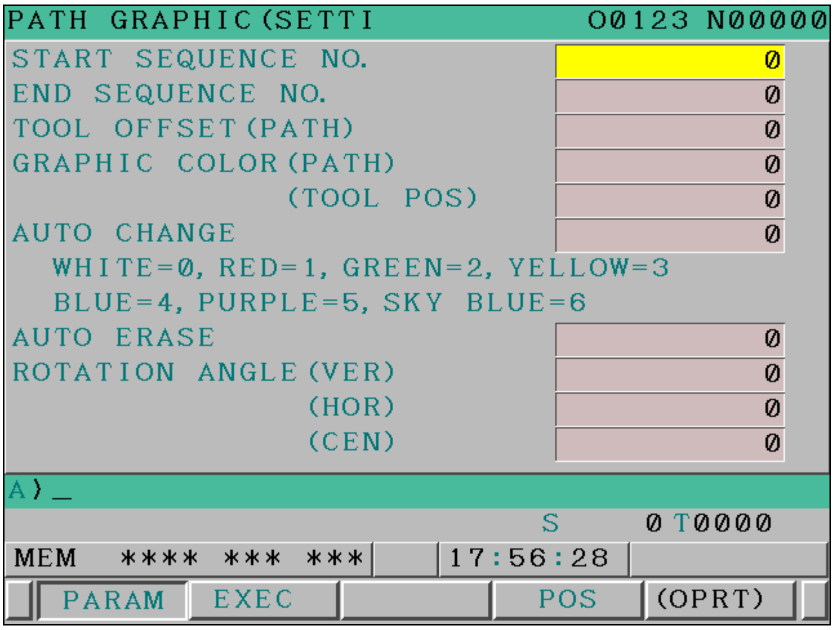


Fig. 13.2.1.1 (c) PATH GRAPHIC (SETTING-2) screen (first page) (8.4-inch LCD)

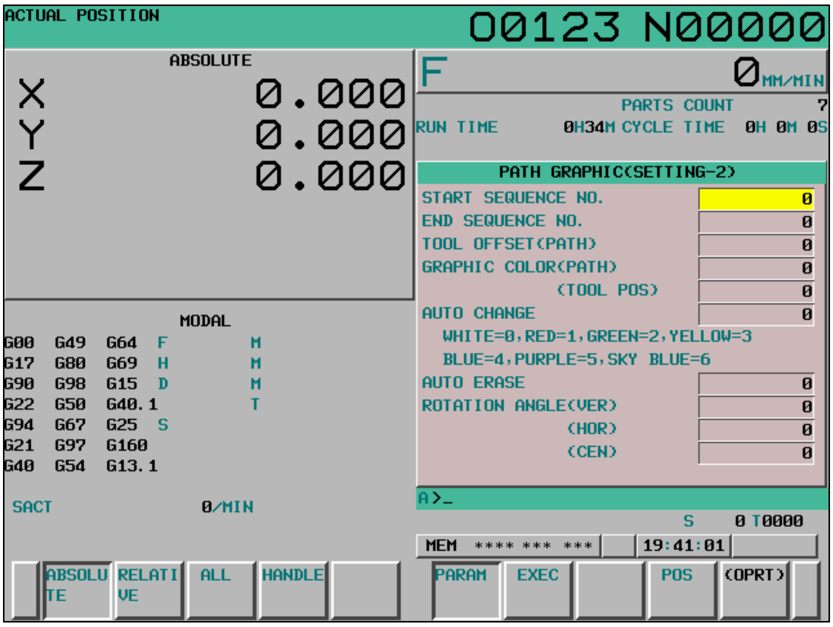


Fig. 13.2.1.1 (d) PATH GRAPHIC (SETTING-2) screen (first page) (10.4-inch LCD)

2 Two screens are used for the PATH GRAPHIC (SETTING) screen.
Use the MDI page keys to switch between the screens for display of a desired setting item.

3 Use the MDI cursor keys to place the cursor at the value of the desired item. Use the numeric keys to type a numeric value to be set. (The typed numeric value is stored in the key-in buffer.)

4 To directly set the numeric value typed in step 3, press the INPUT key or the [INPUT] soft key.
To increment the current value by the numeric value typed in step 3, press the [+INPUT] soft key.

For each setting item, see the explanation.

Explanation

The setting items on the PATH GRAPHIC (SETTING) screen are described below.

- Graphic coordinate system (GRAPHIC COORDINATE)

Select a graphic coordinate system for drawing from the following and set its number.

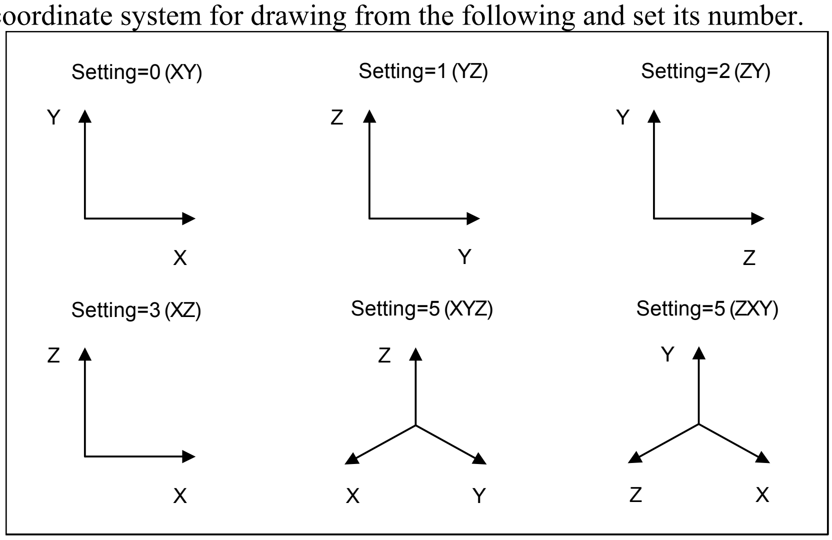


Fig. 13.2.1.1 (e) Graphic coordinate system

- Scale (SCALE)

Set the scale factor for drawing in the range of 0.01 to 100.00 (times).
With a small scale factor, it is possible to draw within a wide range.
With a large scale factor, it is possible to draw in the vicinity of the graphic center being enlarged.

> **NOTE**
> When 0 is set, the scale factor and central coordinate positions for drawing are determined according to graphic parameters for the graphic range (maximum and minimum values).

- Graphic range center (GRAPHIC RANGE CENTER)

To specify the center coordinates of a graphic range, set a coordinate on each axis in the workpiece coordinate system of the program executed for drawing.

> **NOTE**
> 1 Set a coordinate on each axis in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.
> 2 This setting is effective when a nonzero value is set as the graphic parameter for the scale factor.

- Graphic range (maximum value)/(minimum value) (RANGE(MAX.)/(MIN.))

Set the desired graphic range by the maximum and minimum coordinate values in the workpiece coordinate system.
The scale factor and central coordinate positions for drawing are automatically calculated from the set maximum and minimum values, and drawing is performed so that the entire set range falls within the drawing area.

> **NOTE**
> 1 This setting is effective when 0 is set as the graphic parameter for the scale factor.
> 2 The scale factor for drawing is clamped to the range of 0.01 to 100.00.
> 3 Set the position of the maximum value larger than the minimum value.
> 4 When bit 3 (BGM) of parameter No. 11329 is 1, set a coordinate value on each axis in the machine coordinate system.

- Start/end sequence numbers (START/END SEQUENCE NO.)

Set start and end sequence numbers for drawing. A drawing target program is executed from the beginning, but drawing is performed only for the portion between the start and end sequence numbers.
When 0 is specified as the start sequence number, drawing is performed from the beginning of the program.
When 0 is specified as the end sequence number, drawing is performed until the end of the program is reached. The sequence numbers are checked with no distinction between the main program and subprogram.

- Tool offset (Path) (TOOL OFFSET(PATH))

For tool path drawing, whether to enable or disable the tool offset function (tool length compensation, cutter compensation) can be selected.

Setting 0: The tool offset function is enabled for drawing.
1: The tool offset function is disabled for drawing.

- Graphic color (GRAPHIC COLOR)

Set colors to be used for tool path drawing.
The colors that can be set are indicated below together with their setting values:

Graphic color	White	Red	Green	Yellow	Blue	Purple	Light blue
Setting value	0	1	2	3	4	5	6

Path (PATH)

Set colors to be used for a drawn tool path.

Tool position (TOOL POS)

Set a graphic cursor color to be used for PATH GRAPHIC (POSITION) screen.

- Automatic change (AUTO CHANGE)

T codes specified in a drawing target program can be used to automatically change the color of a tool path during drawing.

Setting 0: The automatic change is not performed.
1: The automatic change is performed.

When 1 is set, the setting for the tool path color is incremented by 1 each time a T code is executed; the tool path color changes accordingly. When the setting reaches 6, it is reset to 0.

- Auto erase (AUTO ERASE)

When drawing is started with the [AUTO] or [START] soft key from a state where drawing is not being executed or is not stopped temporarily, the previously drawn path can be erased.

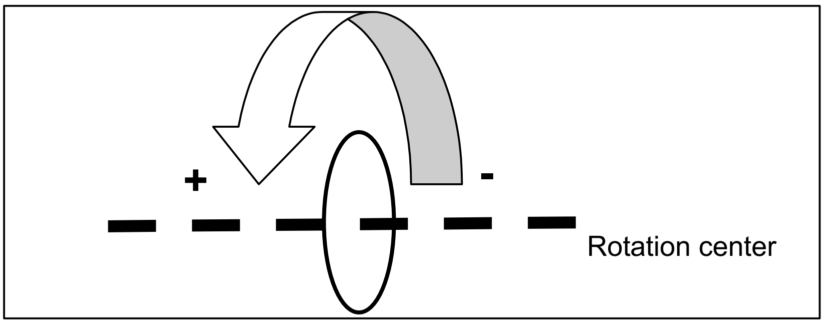
Setting 0: The previously drawn path is not erased.
1: The previously drawn path is erased.

- Rotation angle

Set a rotating angle of the graphic coordinate system that centers on the graphic range center. The rotating angle is a range of -360°-+360°.
Set a rotating angle as a reference position (position of the rotating angle 0°) on the indicated direction of each graphic coordinate system.

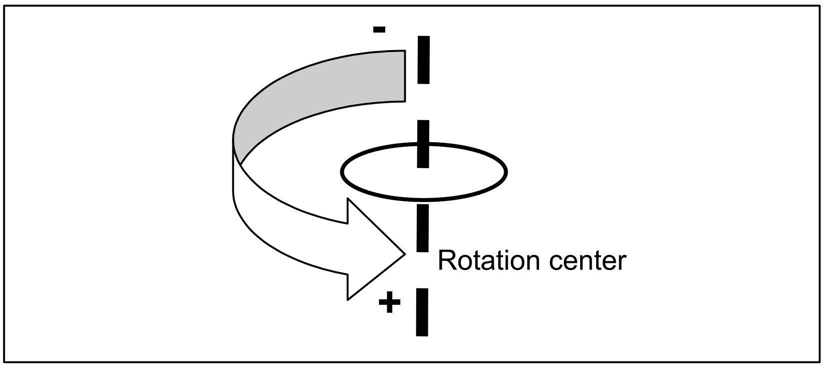
Vertical plane rotation angle

Set a rotating angle at the horizontal direction center in front of the screen.
The rotation direction is as follows.



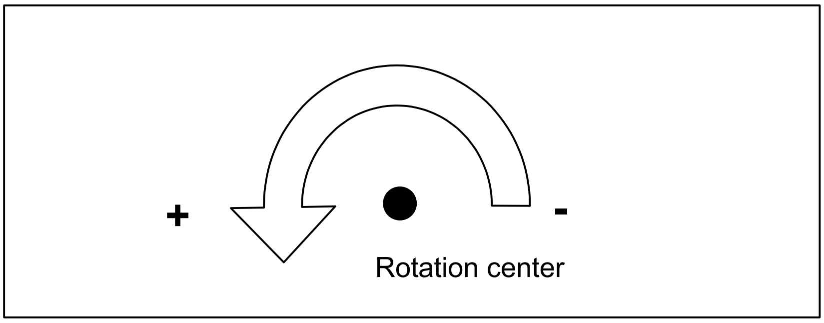
Horizontal plane rotation angle

Set a rotating angle at the vertical direction center in front of the screen.
The rotation direction is as follows.



Screen center rotation angle

Set a rotating angle at the vertical direction center of the screen plane.
The rotation direction is as follows.



13.2.1.2 PATH GRAPHIC (EXECUTION) screen

The PATH GRAPHIC screen is used to draw a tool path. The following operations can be performed:

- Starting/ending tool path drawing
- Rewind of a drawing target program
- Erasing a drawn tool path
- Enlarging/reducing/moving the graphic range
- Changing/rotating the graphic coordinate system

The screen is composed of the following items:
(1) Drawing area
(2) Status display of Background operation
(3) Program number and sequence number for drawing execution
(4) Current coordinates
(5) Feedrate and M/S/T/D code instruction information
(6) Graphic coordinate system
(7) Actual dimension line

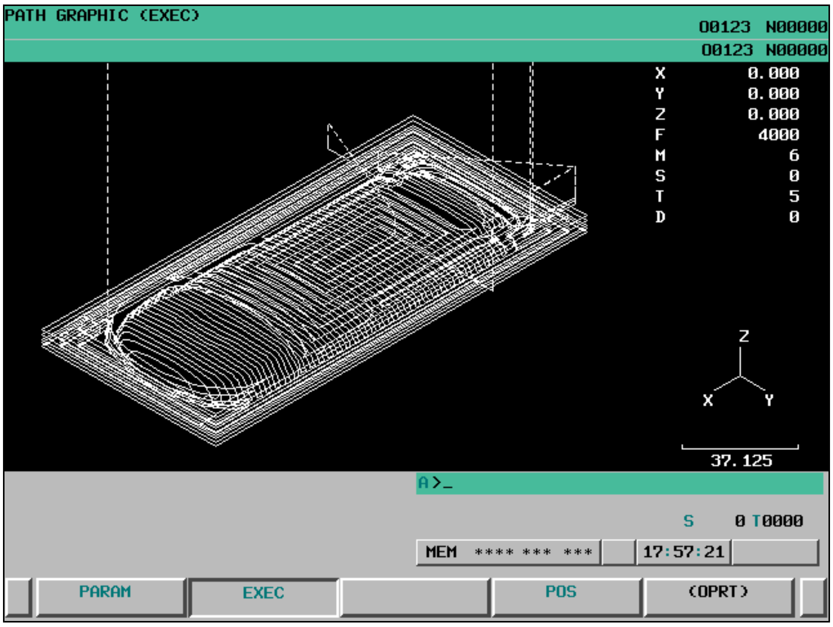


Fig. 13.2.1.2 (a) PATH GRAPHIC (EXECUTION) screen (8.4-inch LCD)

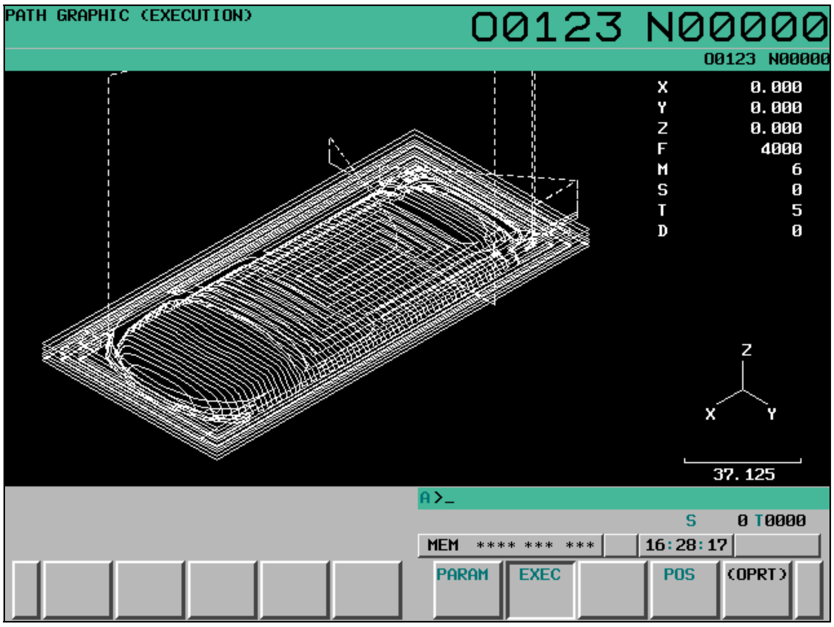
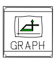
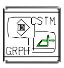


Fig. 13.2.1.2 (b) PATH GRAPHIC (EXECUTION) screen (10.4-inch LCD)

PATH GRAPHIC (EXECUTION) screen: Procedure

Procedure

1 Press the function key  (or  when a small MDI unit is used) to display the PATH GRAPHIC (SETTING-1) screen.
2 Press the [EXEC] soft key. The PATH GRAPHIC screen is displayed.

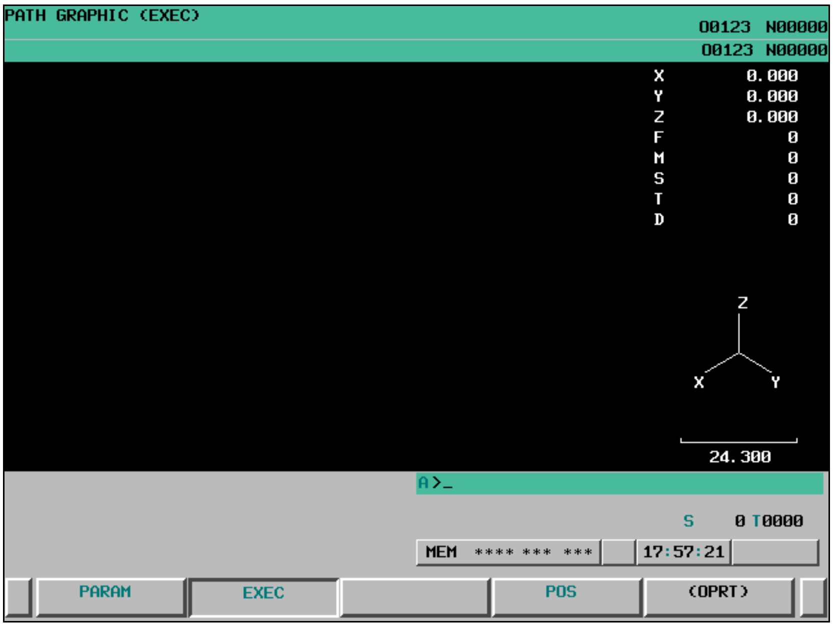


Fig. 13.2.1.2 (c) PATH GRAPHIC (EXECUTION) screen (8.4-inch LCD)

Fig. 13.2.1.2 (d) PATH GRAPHIC (EXECUTION) screen (10.4-inch LCD)

3 Press the [(OPRT)] soft key. The soft keys for tool path drawing are displayed.

Fig. 13.2.1.2 (e) PATH GRAPHIC (EXECUTION) screen (operation) (8.4-inch LCD)

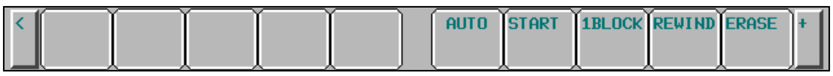


Fig. 13.2.1.2 (f) PATH GRAPHIC (EXECUTION) screen (operation) (10.4-inch LCD)

4 Press the continuous menu key  to display the soft keys for enlarging/reducing/moving the graphic range.

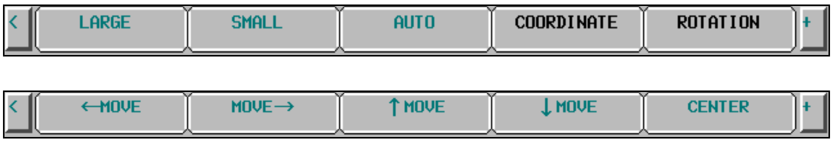


Fig. 13.2.1.2 (g) PATH GRAPHIC (EXECUTION) screen (enlarging/reducing/moving the graphic range) (8.4-inch LCD)

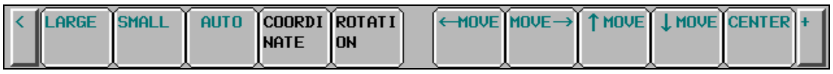


Fig. 13.2.1.2 (h) PATH GRAPHIC (EXECUTION) screen (enlarging/reducing/moving the graphic range) (10.4-inch LCD)

5 Press the [COORDINATE] soft key to display the soft keys for changing the graphic coordinate system.

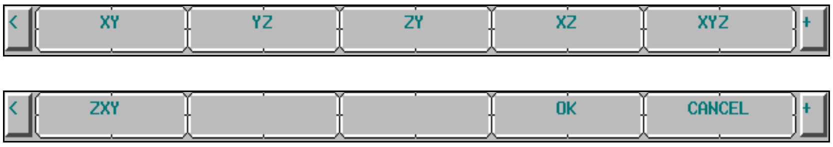


Fig. 13.2.1.2 (i) PATH GRAPHIC (EXECUTION) screen (changing the graphic coordinate system) (8.4-inch LCD)

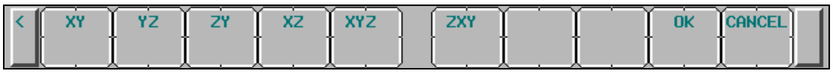


Fig. 13.2.1.2 (j) PATH GRAPHIC (EXECUTION) screen (changing the graphic coordinate system) (10.4-inch LCD)

6 Press the [ROTATION] soft key to display the soft keys for rotating the graphic coordinate system.

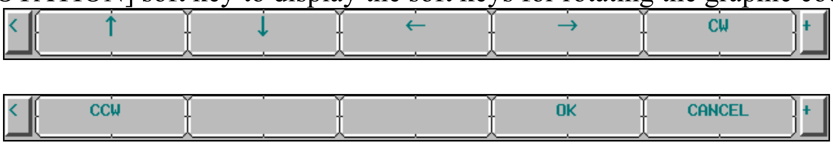


Fig. 13.2.1.2 (k) PATH GRAPHIC (EXECUTION) screen (rotating the graphic coordinate system) (8.4-inch LCD)

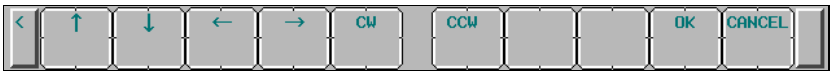


Fig. 13.2.1.2 (l) PATH GRAPHIC (EXECUTION) screen (rotating the graphic coordinate system) (10.4-inch LCD)

For the operation of each soft key, see the explanation.

Explanation

- Graphic program selection

Drawing is performed for the program selected as the main program. On the program list screen, however, you can select another program only for drawing.
The procedure is as follows.

1 Press the [(OPRT)] soft key on the program list screen, and then press the continuous menu key ▷ several times to display the [DRAW SELECT] soft key.

Fig. 13.2.1.2 (m) Program list screen ([DRAW SELECT] soft key) (8.4-inch LCD)

Fig. 13.2.1.2 (n) Program list screen ([DRAW SELECT] soft key) (10.4-inch LCD)

2 Use the MDI keys to type the number of a program for drawing.
3 Press the [DRAW SELECT] soft key.

The number of the program selected in the above steps is prefixed with "#" indicating that the program is selected for drawing.

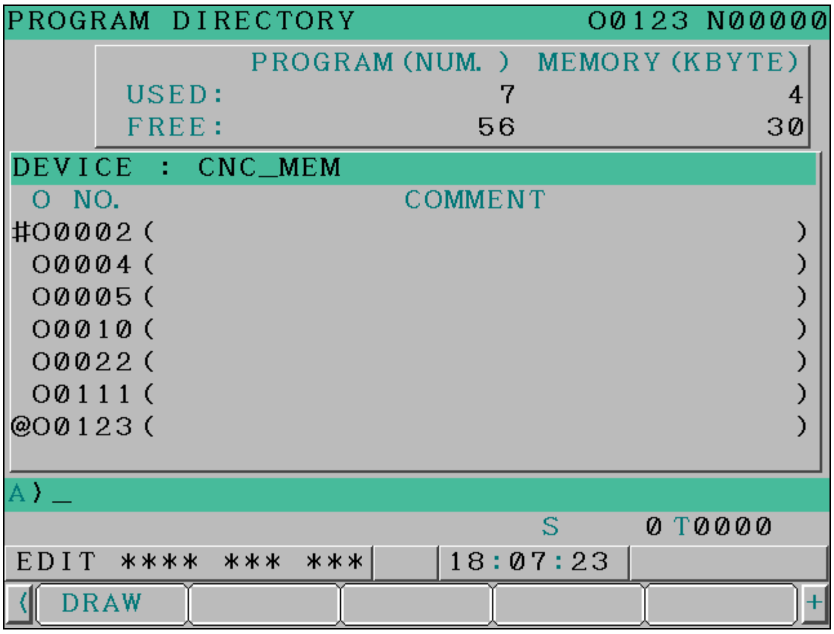


Fig. 13.2.1.2 (o) Program list screen (drawing target program selection state) (8.4-inch LCD)

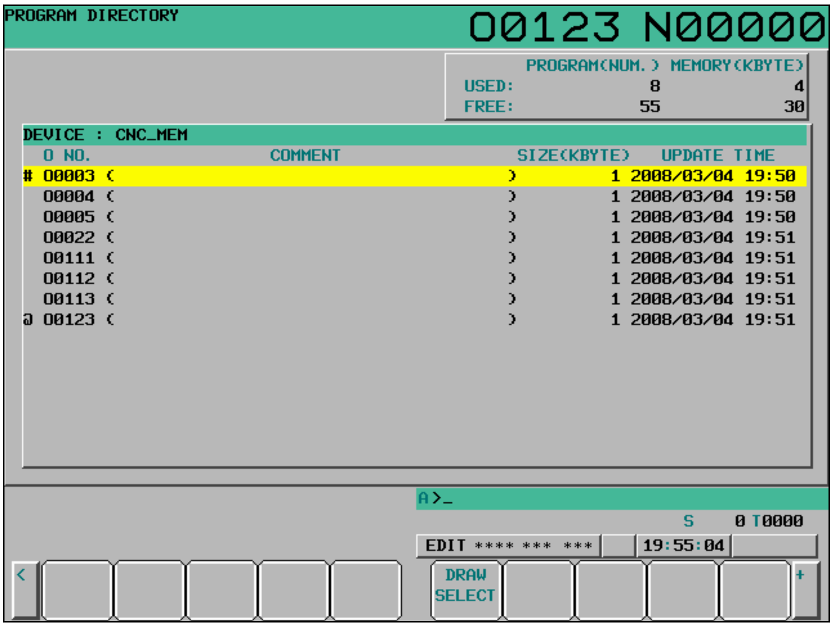


Fig. 13.2.1.2 (p) Program list screen (drawing target program selection state) (10.4-inch LCD)

> **NOTE**
> The file that can be selected as the drawing target program is only a file that can be selected as the main program.

- Starting / Stopping drawing

To draw the tool path of a program selected for drawing, press one of the following soft keys displayed by step 3 mentioned above:

- [AUTO] soft key
  This soft key performs automatic scaling. Before drawing is started, the maximum and minimum coordinate values for the drawing target program are determined and set as the graphic parameters for the maximum and minimum values in the graphic range, and 0 is set for the scale factor and graphic range center. Drawing is then started. The tool path is drawn properly on the screen.
- [START] soft key
  This soft key starts drawing from the head of the program.
- [1BLOCK] soft key
  This soft key executes the program for drawing and stops temporarily on a block-by-block basis as with ordinary single block stop operation.

If a program is executed with a soft key above, the soft key display is changed to the following:

Fig. 13.2.1.2 (q) Soft keys displayed during drawing execution (8.4-inch LCD)

Fig. 13.2.1.2 (r) Soft keys displayed during drawing execution (10.4-inch LCD)

The operations of these soft keys are as follows:

- [END] soft key
  This soft key terminates the execution of the drawing target program to stop drawing.
- [PAUSE] soft key
  This soft key temporarily stops the execution of the drawing target program to stop drawing temporarily.
- [1BLOCK] soft key
  This soft key executes the program for drawing and stops drawing temporarily on a block-by-block basis as with ordinary single block stop operation.
- [RESTART] soft key
  If the [RESTART] soft key is pressed in the stop state set by the [PAUSE] soft key or the [1BLOCK] soft key, the execution of drawing can be restarted from the block where drawing is stopped.

NOTE
When the new path was drawn by the operation of starting drawing without erasing of the old path before the operation, it is impossible to redraw the old path by each operation of enlarging/reducing/moving the graphic range and changing/rotating the graphic coordinate system.

The state of drawing is indicated as follows:

DRAWING: Indicates that drawing is being executed.

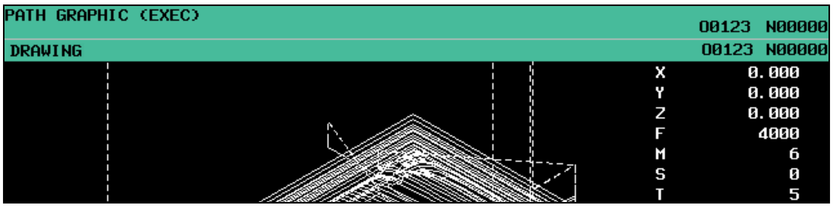


Fig. 13.2.1.2 (s) State indication during drawing

STOP: Indicates that drawing is temporarily stopped.

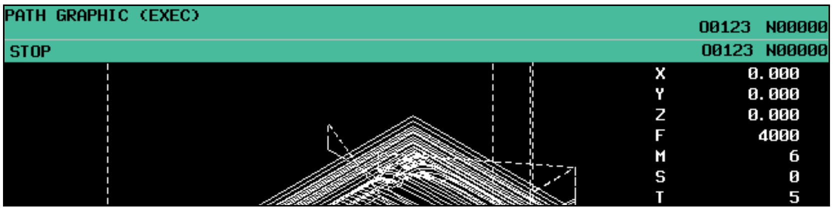


Fig. 13.2.1.2 (t) State indication during temporary stop

ALM: Indicates that occurring of alarm in the Background operation.

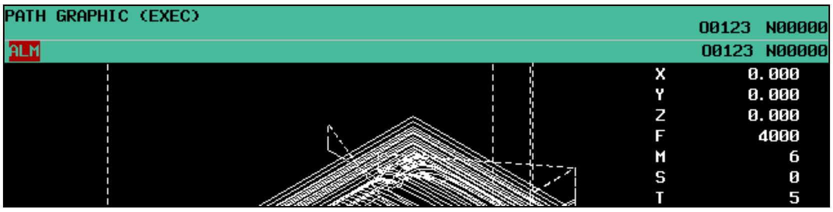


Fig. 13.2.1.2 (u) State indication during occurring of alarm

- End of drawing

When M02 or M30 is executed, the program executed for drawing terminates drawing.
Upon program termination, the soft key display returns to the soft keys (Fig. 13.2.1.2 (e)/(f)) displayed before drawing is started.

- Rewind of a drawing target program

If the execution of drawing of a selected program has ended or is stopped halfway, press the [REWIND] soft key to restart drawing from the beginning of the program.

- Erasing a drawn tool path

Press the [ERASE] soft key to erase a drawn tool path.

NOTE

1 If the screen display is switched during tool path drawing, the background operation is stopped to end drawing.
2 A tool path once drawn is erased when the screen display is switched.

- Enlarging/reducing the graphic range

The following soft keys displayed by step 4 are used:

- [LARGE] soft key
  This soft key increases the scale to enlarge the graphic range.
- [SMALL] soft key
  This soft key decreases the scale to reduce the graphic range.
- [AUTO] soft key
  When the graphic parameters for the graphic range (maximum and minimum values) are set, this soft

key automatically scales the set graphic range so that the range falls within the drawing area. When the graphic range (maximum and minimum values) is not set (0 is set), this operation is disabled.

NOTE
1 Set the unit of scale for one enlargement/reduction operation in parameter No. 14713.
2 An enlargement/reduction scale used here is set in the graphic parameter for scale.

- Moving the graphic range

The following soft keys displayed by step 4 are used:

- [←MOVE] soft key
  This soft key moves the graphic range to left.
- [MOVE→] soft key
  This soft key moves the graphic range to right.
- [↑MOVE] soft key
  This soft key moves the graphic range upward.
- [↓MOVE] soft key
  This soft key moves the graphic range downward.
- [CENTER] soft key
  This soft key returns the graphic range to the original position.

NOTE
1 Set the travel increment made by one horizontal move operation in parameter No. 14714.
2 Set the travel increment made by one vertical move operation in parameter No. 14715.
3 The graphic range modified here is not set in the graphic parameter for graphic range center.

- Changing the graphic coordinate system

The following soft keys displayed by step 5 are used.
A graphic coordinate system selected here is the same one as set in the graphic parameter for the graphic coordinate system.

- [XY] soft key
  This soft key selects the graphic coordinate system of XY (with a setting of 0).
- [YZ] soft key
  This soft key selects the graphic coordinate system of YZ (with a setting of 1).
- [ZY] soft key
  This soft key selects the graphic coordinate system of ZY (with a setting of 2).
- [XZ] soft key
  This soft key selects the graphic coordinate system of XZ (with a setting of 3).
- [XYZ] soft key
  This soft key selects the graphic coordinate system of XYZ (with a setting of 4).
- [ZXY] soft key
  This soft key selects the graphic coordinate system of ZXY (with a setting of 5).
- [OK] soft key
  This soft key changes the current graphic coordinate system to the graphic coordinate system selected by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the graphic coordinate system selected by one of the soft keys above to return to the original graphic coordinate system.

NOTE

The graphic coordinate system selected here is set in the graphic parameter for the graphic coordinate system.

- Rotating the graphic coordinate system

The following soft keys displayed by step 6 are used.

- [↑] soft key
  This soft key rotates the graphic coordinate system upward.
- [↓] soft key
  This soft key rotates the graphic coordinate system downward.
- [←] soft key
  This soft key rotates the graphic coordinate system to left.
- [→] soft key
  This soft key rotates the graphic coordinate system to right.
- [CW] soft key
  This soft key rotates the graphic coordinate system clockwise.
- [CCW] soft key
  This soft key rotates the graphic coordinate system counterclockwise.
- [OK] soft key
  This soft key changes the rotation angle of the current graphic coordinate system to the one set by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the rotation of the graphic coordinate system performed by one of the soft keys above to return to the original graphic coordinate system.

NOTE

1 Set the travel increment made by one rotation operation in parameter No. 14716.
2 The rotation angle of the graphic coordinate system set here is not set in the graphic parameter for rotation angle.

13.2.1.3 PATH GRAPHIC (POSITION) screen

The PATH GRAPHIC (POSITION) screen indicates the current position during operation by the cursor on a tool path drawn on the PATH GRAPHIC (EXECUTION) screen; you can check a programmed tool movement during automatic operation.

The screen is composed of the following items:
(1) Drawing area
(2) Current coordinates
(3) Feedrate and M/S/T/D code instruction information
(4) Graphic coordinate system
(5) Actual dimension line
(6) Cursor to show current position

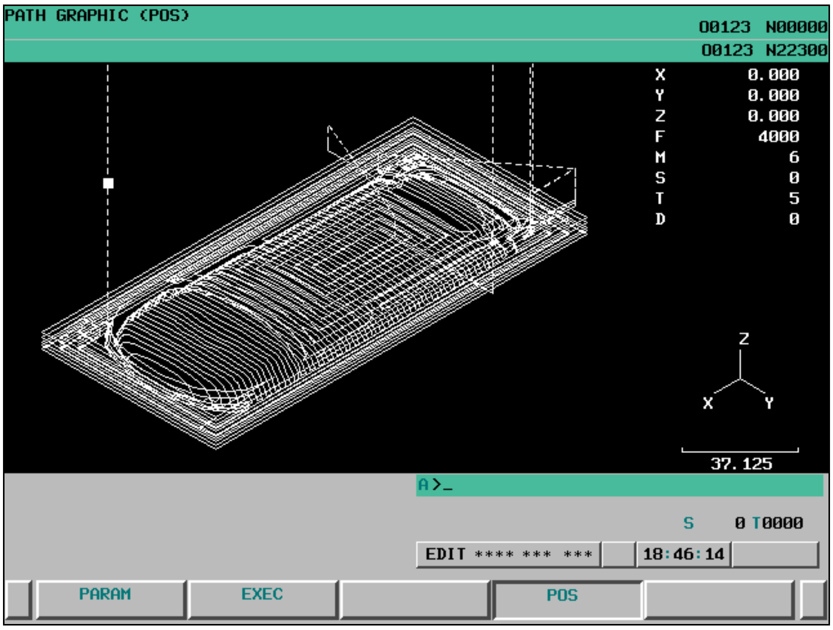


Fig. 13.2.1.3 (a) PATH GRAPHIC (POSITION) screen (8.4-inch LCD)

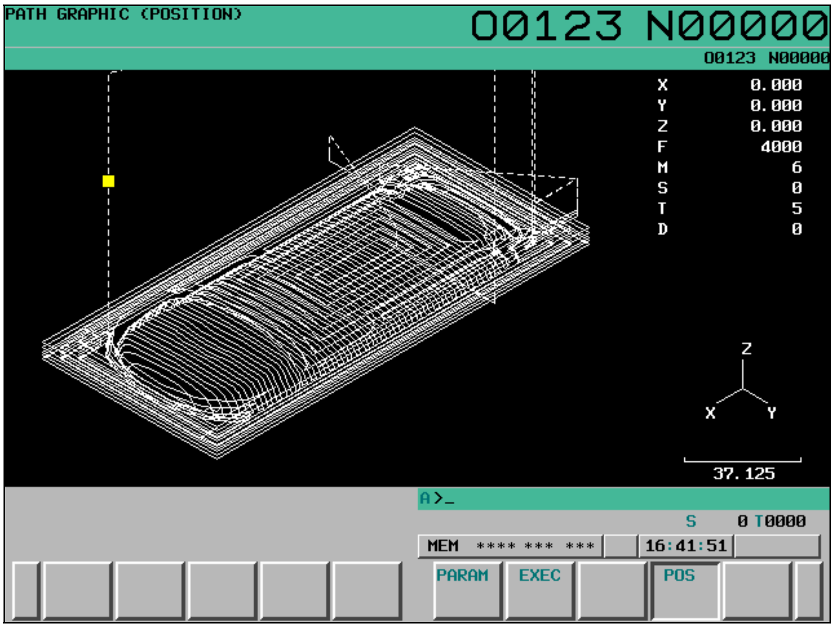
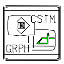


Fig. 13.2.1.3 (b) PATH GRAPHIC (POSITION) screen (10.4-inch LCD)

PATH GRAPHIC (POSITION) screen: Procedure

Procedure

1 Press the function key  (or  when a small MDI unit is used) to display the PATH GRAPHIC (SETTING-1) screen.

2 Press the [POS] soft key.
The screen display changes to the PATH GRAPHIC (POSITION) screen and the cursor indicating the tool position is displayed.

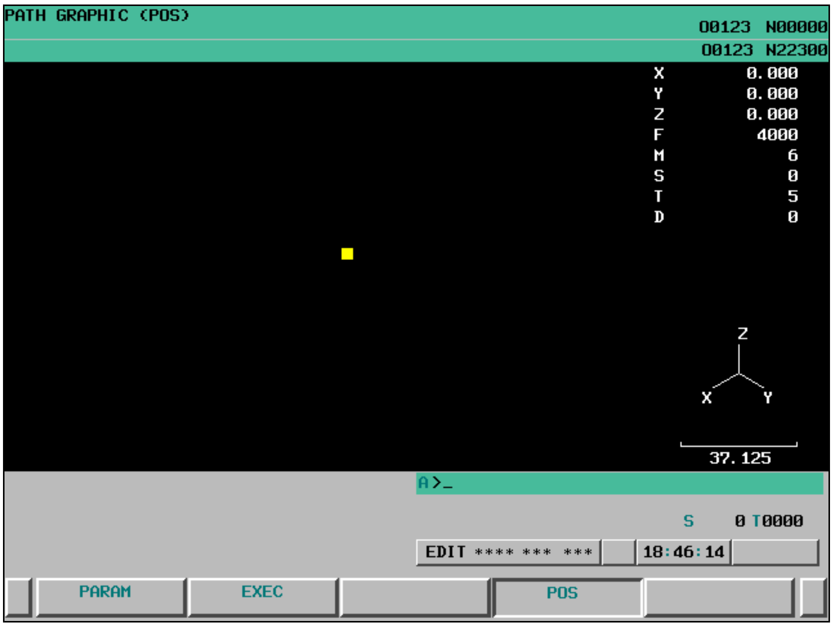


Fig. 13.2.1.3 (c) PATH GRAPHIC (POSITION) screen (8.4-inch LCD)

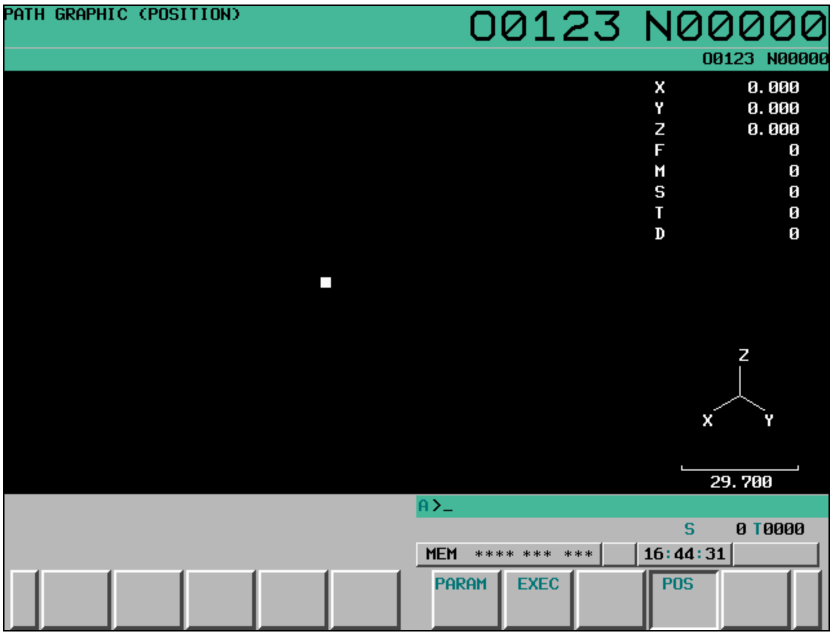


Fig. 13.2.1.3 (d) PATH GRAPHIC (POSITION) screen (10.4-inch LCD)

For the method of checking the current tool position, see the explanation.
Pressing a soft key other than the [POS] soft key displays the corresponding screen.

Explanation

Use the following procedure to check the tool position during operation on the PATH GRAPHIC (POSITION) screen:

(1) Draw the tool path of the selected program on the PATH GRAPHIC (EXECUTION) screen.
(2) After the end of drawing, switch the screen display to the PATH GRAPHIC (POSITION) screen.
(3) Start automatic operation with the program executed for tool path drawing.

After automatic operation is started, the blinking cursor moves along the drawn tool path as the tool moves. The cursor blinking rate is fast when the tool is moving, and the cursor blinking rate is slow when the tool is stopped.

The following items displayed on the screen are provided for the program under automatic operation:
- Current coordinates
- Feedrate and M/S/T/D code specification information

> **NOTE**
> 1 A tool path drawn by setting the tool offset parameter to 1 (to disable the tool offset function) is different from the actual tool path. In this case, the cursor indicating the tool position may not move along the drawn tool path.
> 2 The tool path cannot be drawn correctly if a command not supporting drawing or a command that specifies operation different from operation performed during drawing is specified or if an operation dependent on the operation state or setting on the machine side is performed. So, the tool path in actual operation may differ from a drawn tool path. In this case, the cursor indicating the tool position does not move along the drawn tool path.
> 3 When the graphic parameter of the graphic coordinate system, graphic range (maximum and minimum values), scale, graphic range center, and rotation angle is changed, the drawn tool path drawn is erased.
> Therefore, please draw the tool path again on the PATH GRAPHIC (EXECUTION) screen to display the tool position by setting the changed graphic parameter.

13.2.2 Animation

Overview

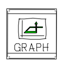
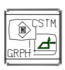
For animation drawing, make necessary settings and perform operations for drawing execution on the following screens:
- ANIME GRAPHIC (SETTING) screen
  On this screen, data required to execute animation drawing can be set.
- ANIMATION GRAPHIC (EXECUTION) screen
  This screen is used for animation drawing.
  On this screen, the graphic range can be reduced/enlarged, and the graphic coordinate system can be rotated.
- ANIMATION GRAPHIC (3-PLANE) screen
  On this screen, a three-plane diagram can be drawn for a machining profile that has been drawn.
  In addition, the positions of side views and cross sections can be changed.

13.2.2.1 ANIME GRAPHIC (SETTING) screen

This screen is used to set graphic parameters needed for animation drawing.
Data set using this screen is made valid by displaying the ANIMATION GRAPHIC (EXECUTION) screen or executing drawing.
Graphic parameter data once set is preserved even if the power is turned off.

ANIME GRAPHIC (SETTING) screen: Procedure

Procedure

1 Press the function key  (or  when a small MDI unit is used) to display the ANIME GRAPHIC (SETTING-1) screen.

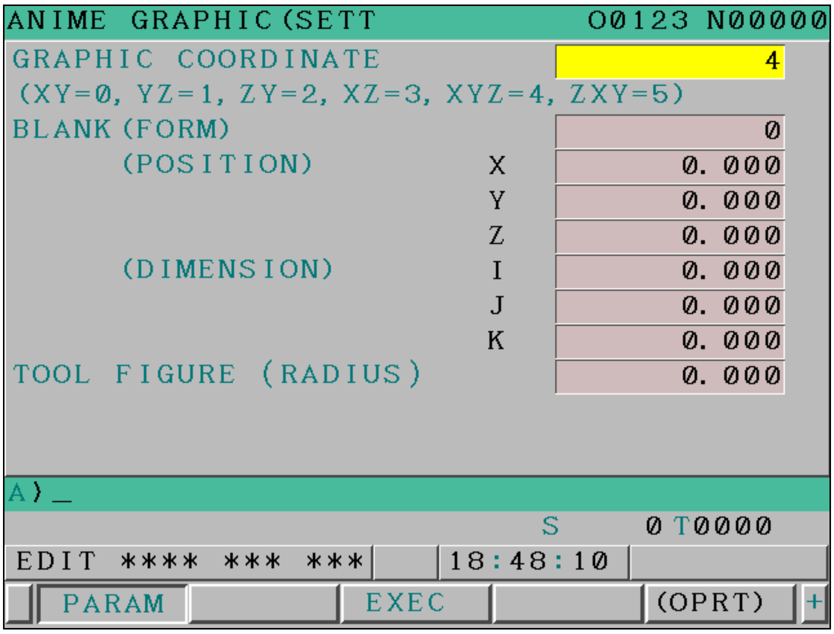


Fig. 13.2.2.1 (a) ANIME GRAPHIC (SETTING-1) screen (8.4-inch LCD)

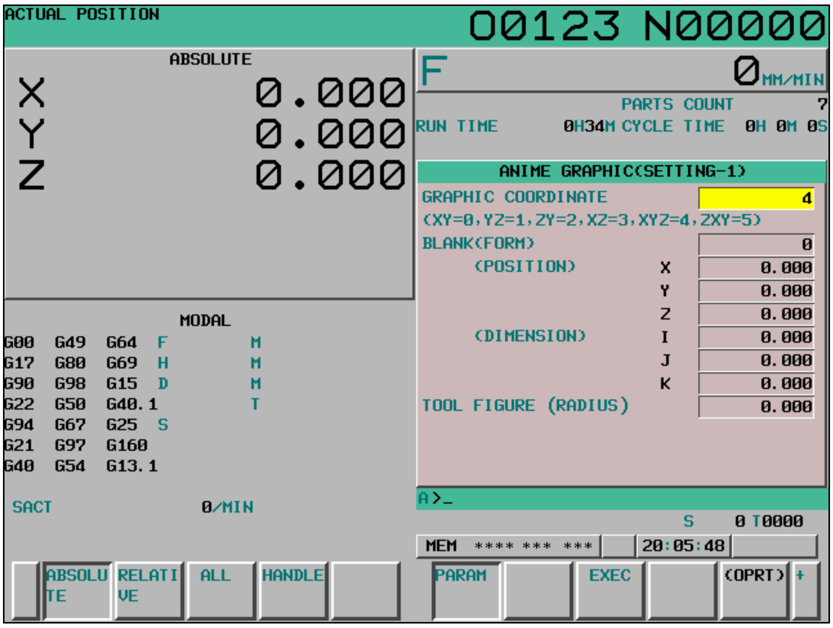


Fig. 13.2.2.1 (b) ANIME GRAPHIC (SETTING-1) screen (10.4-inch LCD)

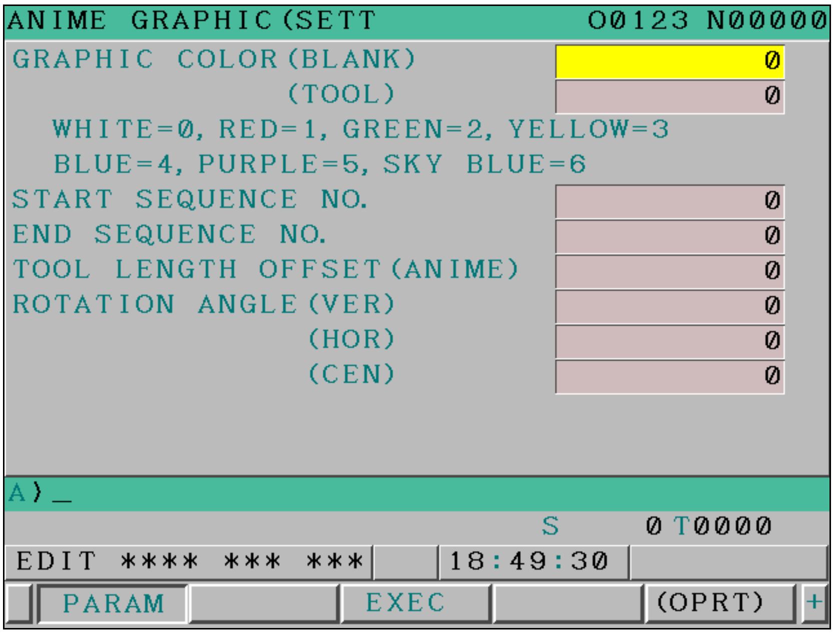


Fig. 13.2.2.1 (c) ANIME GRAPHIC (SETTING-2) screen (8.4-inch LCD)

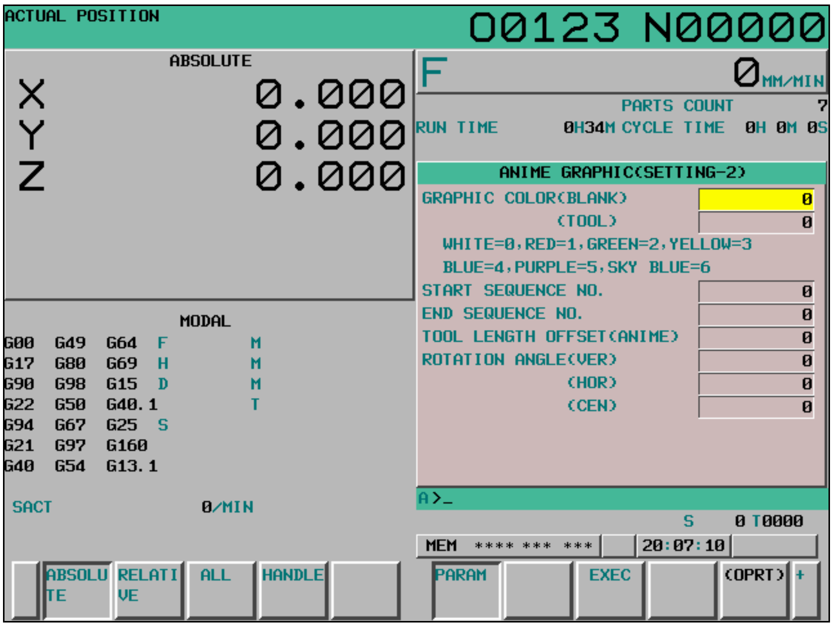
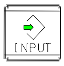


Fig. 13.2.2.1 (d) ANIME GRAPHIC (SETTING-2) screen (10.4-inch LCD)

2 Two screens are used for the ANIME GRAPHIC (SETTING) screen.
Use the MDI page keys to switch between the screens for display of a desired setting item.

3 Use the MDI cursor keys to place the cursor at the value of the desired item. Use the numeric keys to type a numeric value to be set. (The typed numeric value is stored in the key-in buffer.)

4 To directly set the numeric value typed in step 3, press the  key or the [INPUT] soft key.
To increment the current value by the numeric value typed in step 3, press the [+INPUT] soft key.

For each setting item, see the explanation.

Explanation

The setting items on the ANIME GRAPHIC (SETTING) screen are described below.
However, the graphic parameters listed below are shared for tool path drawing. So, refer to the explanation of the PATH GRAPHIC (SETTING) screen for tool path drawing.

- Graphic coordinate system
- Start/end sequence numbers
- Rotation angles (vertical plane, horizontal plane, screen center)

- Blank figure (BLANK(FORM))

With a drawing program, set the figure, position, and dimensions of a blank to be machined.

> **NOTE**
> To draw blank figures and tool figures, be sure to set the graphic parameters for the blank (figure/position/dimensions) and tool figure (radius).

Figure (FIGURE)

Select a type of blank figure from the following and set the corresponding value:

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

Position (POSITION)

Set the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

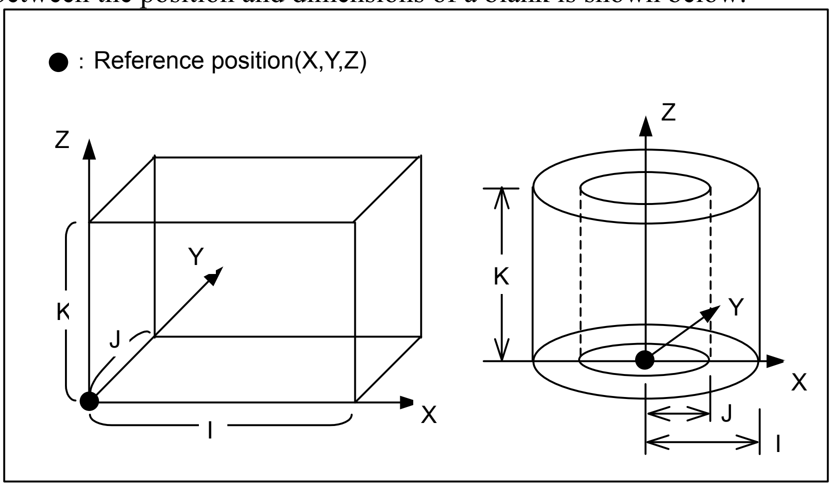
> **NOTE**
> Set the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

Dimensions (DIMENSION)

Set the dimensions of each type of blank figure as indicated below.

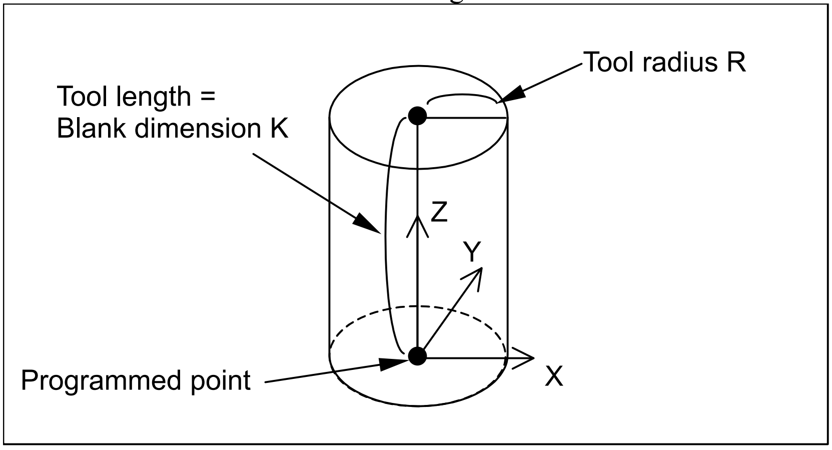
Type of blank figure	Dimension I	Dimension J	Dimension K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Radius of column	0	Length of column
Cylinder	Radius of outer circle of cylinder	Radius of inner circle of cylinder	Length of cylinder

The relationship between the position and dimensions of a blank is shown below.



- Tool figure (radius) (TOOL FIGURE (RADIUS))

Set the radius of a tool figure to be drawn.
The tool length is the same as dimension K of a blank figure in the Z-axis direction.



- Graphic color

Set colors to be used for animation drawing.
The colors that can be set are indicated below together with their setting values:

Graphic color	White	Red	Green	Yellow	Blue	Purple	Light blue
Setting value	0	1	2	3	4	5	6

Blank

Set the color of a blank to be drawn in animation drawing.

Tool

Set the color of a tool to be drawn in animation drawing.

- Tool length offset (Anime)

For animation drawing, whether to enable or disable the tool length offset can be selected.
Setting 0: The tool length offset is disabled for drawing.
1: The tool length offset is enabled for drawing.

> **NOTE**
> In animation drawing, the cutter radius compensation is always enabled for drawing.

13.2.2.2 ANIMATION GRAPHIC (EXECUTION) screen

The ANIMATION GRAPHIC screen is used to draw a animation. The following operations can be performed:

- Starting/ending animation drawing
- Rewind of a drawing target program
- Initializing a blank
- Enlarging/reducing/moving the graphic range
- Changing/rotating the graphic coordinate system

The screen configuration is as follows:
(1) Drawing area
(2) Status display of Background operation
(3) Program name and sequence number for drawing execution
(4) Current coordinates

(5) Feedrate and M/S/T/D code instruction information
(6) Graphic coordinate system

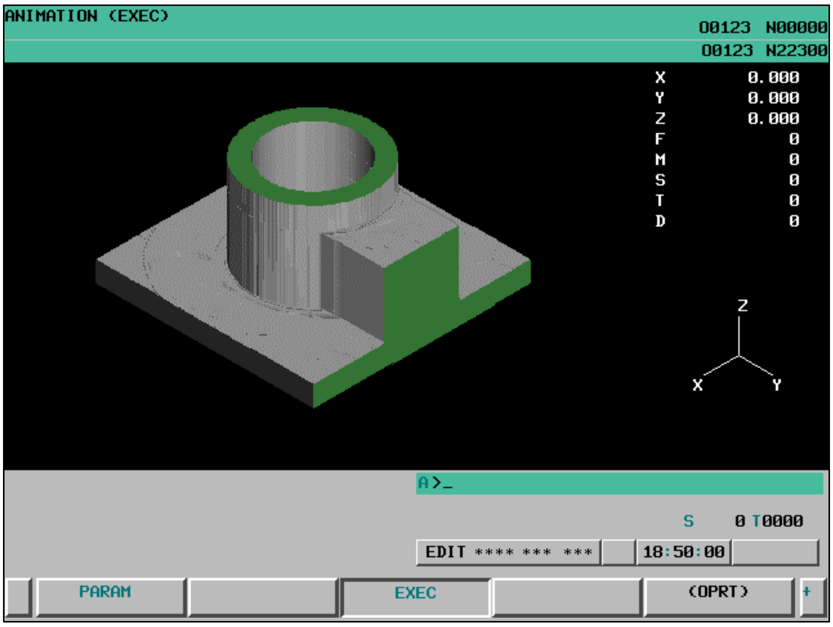


Fig. 13.2.2.2 (a) ANIMATION GRAPHIC (EXECUTION) screen (8.4-inch LCD)

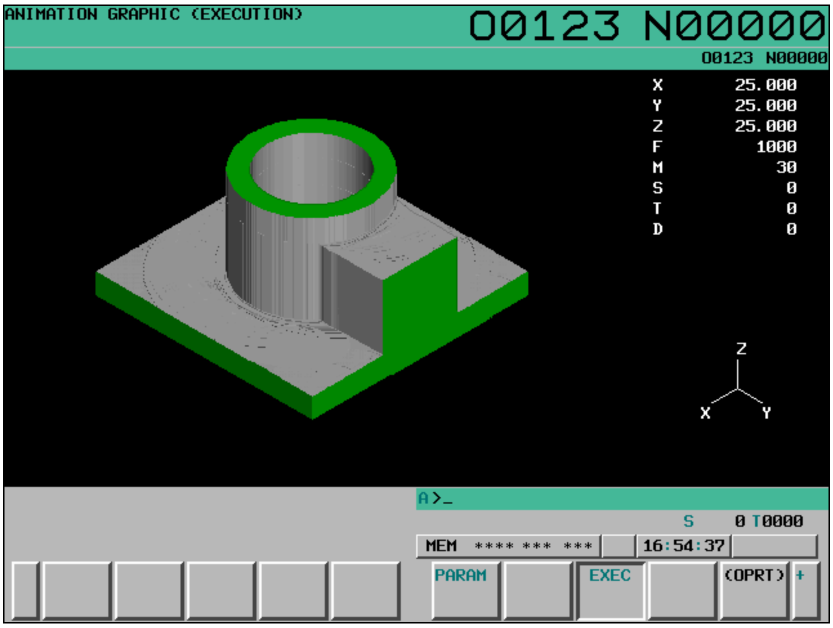
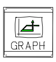
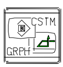


Fig. 13.2.2.2 (b) ANIMATION GRAPHIC (EXECUTION) screen (10.4-inch LCD)

ANIMATION GRAPHIC (EXECUTION) screen: Procedure

Procedure

1 Press the function key  (or  when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

2 Press the [EXEC] soft key. The ANIMATION GRAPHIC (EXECUTION) screen is displayed.

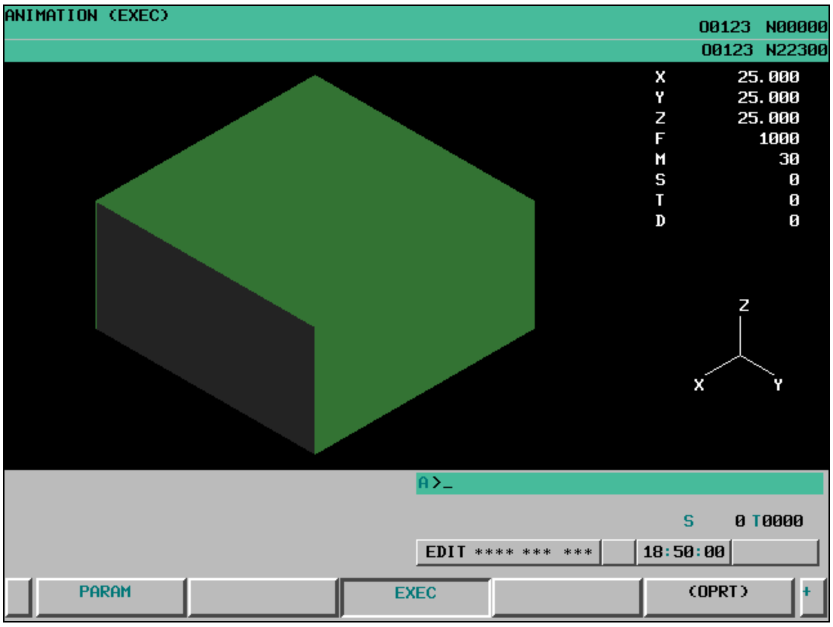


Fig. 13.2.2.2 (c) ANIMATION GRAPHIC (EXECUTION) screen (8.4-inch LCD)

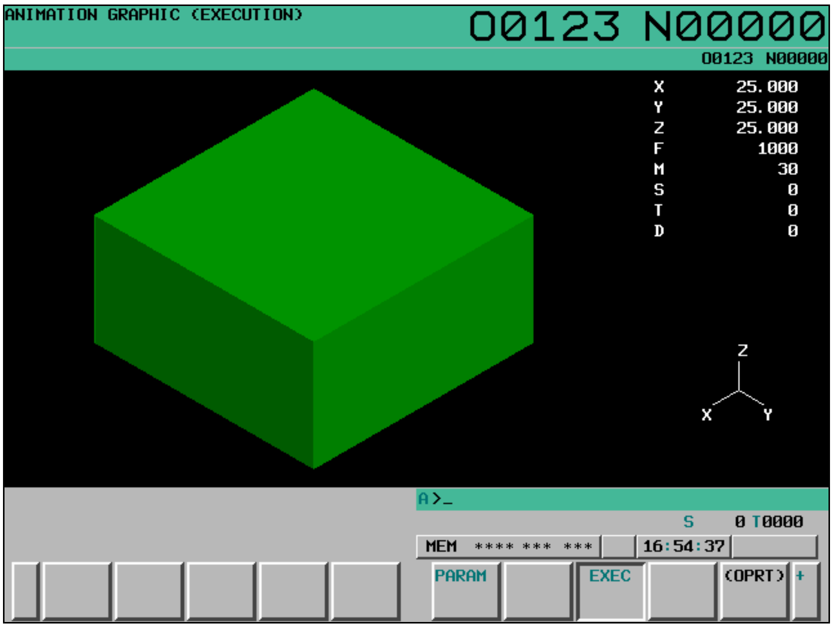


Fig. 13.2.2.2 (d) ANIMATION GRAPHIC (EXECUTION) screen (10.4-inch LCD)

3 Press the [(OPRT)] soft key. The soft keys for tool path drawing are displayed.

Fig. 13.2.2.2 (e) ANIMATION GRAPHIC (EXECUTION) screen (operation) (8.4-inch LCD)

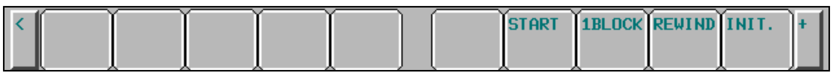


Fig. 13.2.2.2 (f) ANIMATION GRAPHIC (EXECUTION) screen (operation) (10.4-inch LCD)

4 Press the continuous menu key ▷ to display the soft keys for enlarging/reducing/moving the graphic range.

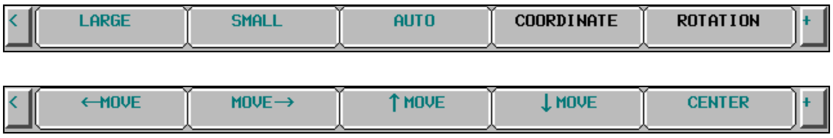


Fig. 13.2.2.2 (g) ANIMATION GRAPHIC (EXECUTION) screen (enlarging/reducing/moving the graphic range) (8.4-inch LCD)

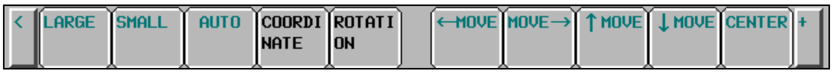


Fig. 13.2.2.2 (h) ANIMATION GRAPHIC (EXECUTION) screen (enlarging/reducing/moving the graphic range) (10.4-inch LCD)

5 Press the [COORDINATE] soft key to display the soft keys for changing the graphic coordinate system.

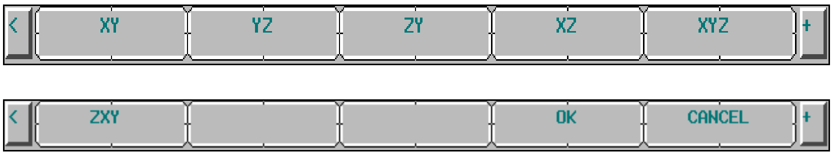


Fig. 13.2.2.2 (i) ANIMATION GRAPHIC (EXECUTION) screen (changing the graphic coordinate system) (8.4-inch LCD)

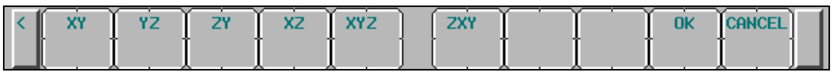


Fig. 13.2.2.2 (j) ANIMATION GRAPHIC (EXECUTION) screen (changing the graphic coordinate system) (10.4-inch LCD)

6 Press the [ROTATION] soft key to display the soft keys for rotating the graphic coordinate system.

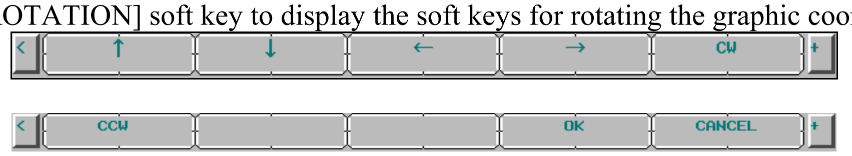


Fig. 13.2.2.2 (k) ANIMATION GRAPHIC (EXECUTION) screen (rotating the graphic coordinate system) (8.4-inch LCD)

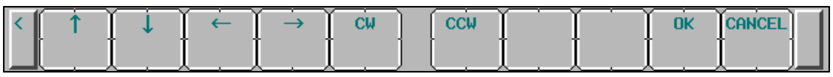


Fig. 13.2.2.2 (l) ANIMATION GRAPHIC (EXECUTION) screen (rotating the graphic coordinate system) (10.4-inch LCD)

For the operation of each soft key, see the explanation.

Explanation

The operations listed below are the same operations as for the PATH GRAPHIC (EXECUTION) screen. See the explanation of the PATH GRAPHIC (EXECUTION) screen.

- Graphic program selection
- Starting / Stopping of drawing
- End of drawing
- Rewind of a drawing target program

- Initializing a blank

Press the [INIT.] soft key to initialize and return a drawn machining profile to the original blank figure.

NOTE
1 Blank initialization can also be performed by any of the following operations:
- Start of drawing
- Changing of the graphic coordinate system and graphic range by performing enlargement/reduction/movement/rotation operations
- Change of screen

2 A machining profile once drawn for a blank is erased when the screen is switched.

- Enlarging/reducing the graphic range

The following soft keys displayed by step 4 are used:

- [LARGE] soft key
  This soft key increases the scale to enlarge the graphic range.
- [SMALL] soft key
  This soft key decreases the scale to reduce the graphic range.
- [AUTO] soft key
  When the blank figure specified in a graphic parameter is input, this soft key automatically scales the graphic range so that the blank figure is contained within the graphic range.
  If no blank figure is input, this soft key has no effect.

NOTE
Set the unit of scale for one enlargement/reduction operation in parameter No. 14713.

- Moving the graphic range

The following soft keys displayed by step 4 are used:

- [←MOVE] soft key
  This soft key moves the graphic range to left.
- [MOVE→] soft key
  This soft key moves the graphic range to right.
- [↑MOVE] soft key
  This soft key moves the graphic range upward.
- [↓MOVE] soft key
  This soft key moves the graphic range downward.
- [CENTER] soft key
  This soft key returns the graphic range to the original position.

NOTE
1 Set the travel increment made by one horizontal move operation in parameter No. 14714.
2 Set the travel increment made by one vertical move operation in parameter No. 14715.

- Changing the graphic coordinate system

The following soft keys displayed by step 5 are used.
A graphic coordinate system selected here is the same one as set in the graphic parameter for the graphic coordinate system.

- [XY] soft key
  This soft key selects the graphic coordinate system of XY (with a setting of 0).
- [YZ] soft key
  This soft key selects the graphic coordinate system of YZ (with a setting of 1).

- [ZY] soft key
  This soft key selects the graphic coordinate system of ZY (with a setting of 2).
- [XZ] soft key
  This soft key selects the graphic coordinate system of XZ (with a setting of 3).
- [XYZ] soft key
  This soft key selects the graphic coordinate system of XYZ (with a setting of 4).
- [ZXY] soft key
  This soft key selects the graphic coordinate system of ZXY (with a setting of 5).
- [OK] soft key
  This soft key changes the current graphic coordinate system to the graphic coordinate system selected by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the graphic coordinate system selected by one of the soft keys above to return to the original graphic coordinate system.

NOTE
The graphic coordinate system selected here is set in the graphic parameter for the graphic coordinate system.

- Rotating the graphic coordinate system

The following soft keys displayed by step 6 are used.

- [↑] soft key
  This soft key rotates the graphic coordinate system upward.
- [↓] soft key
  This soft key rotates the graphic coordinate system downward.
- [←] soft key
  This soft key rotates the graphic coordinate system to left.
- [→] soft key
  This soft key rotates the graphic coordinate system to right.
- [CW] soft key
  This soft key rotates the graphic coordinate system clockwise.
- [CCW] soft key
  This soft key rotates the graphic coordinate system counterclockwise.
- [OK] soft key
  This soft key changes the rotation angle of the current graphic coordinate system to the one set by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the rotation of the graphic coordinate system performed by one of the soft keys above to return to the original graphic coordinate system.

NOTE
1 Set the travel increment made by one rotation operation in parameter No. 14716.
2 The rotation angle of the graphic coordinate system set here is not set in the graphic parameter for rotation angle.

13.2.2.3 ANIMATION GRAPHIC (3-PLANE) screen

For a three-dimensional machining profile drawn on the ANIMATION GRAPHIC (EXECUTION) screen, a three-plane diagram including one plan view and two side views can be drawn.
You can select one of four pairs of side view display positions. You can also arbitrarily change the position of a cross section to be drawn in a side view.

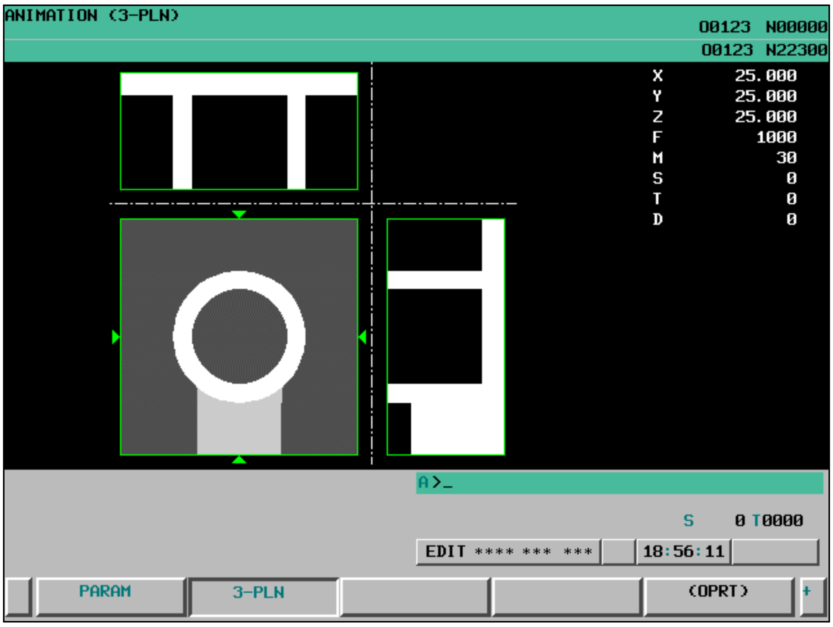


Fig. 13.2.2.3 (a) ANIMATION GRAPHIC (3-PLANE) screen (8.4-inch LCD)

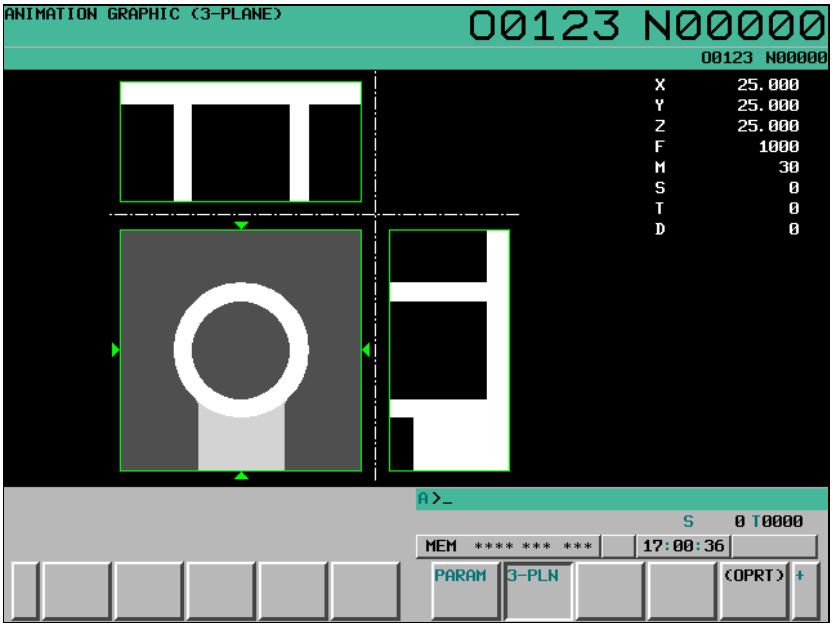


Fig. 13.2.2.3 (b) ANIMATION GRAPHIC (3-PLANE) screen (10.4-inch LCD)

ANIMATION GRAPHIC (3-PLANE) screen: Procedure

Procedure

1 Press the function key [GRAPH] (or [CSTM/GRPH] when a small MDI unit is used) to display the ANIME GRAPHIC (SETTING-1) screen.
If drawing is executed on the ANIMATION GRAPHIC (EXECUTION) screen before this operation, machining profile is displayed.
If drawing is not executed on the ANIMATION GRAPHIC (EXECUTION) screen before this operation, original blank figure is displayed.

2 Press the continuous menu key [▷].

3 Press the [3-PLN] soft key. The ANIMATION GRAPHIC (3-PLANE) screen is displayed.

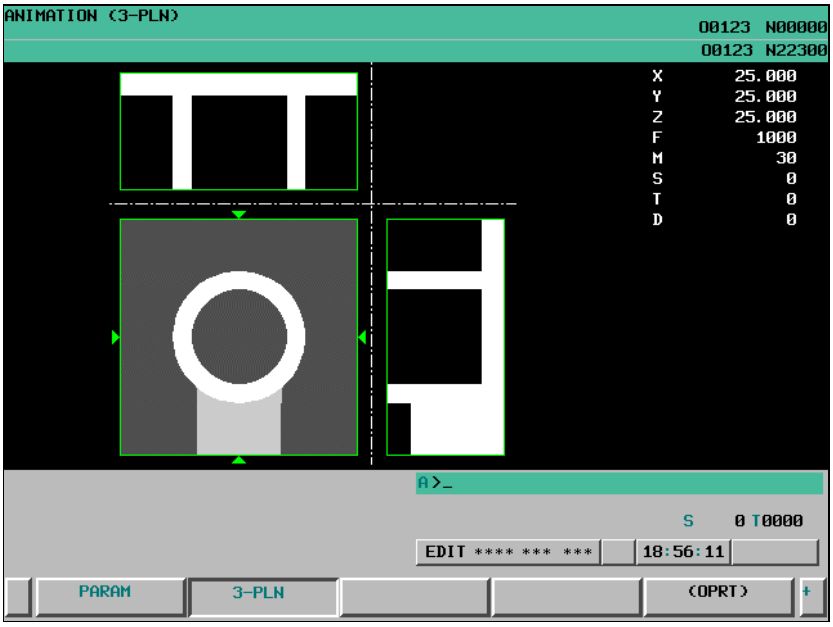


Fig. 13.2.2.3 (c) ANIMATION GRAPHIC (3-PLANE) screen (8.4-inch LCD)

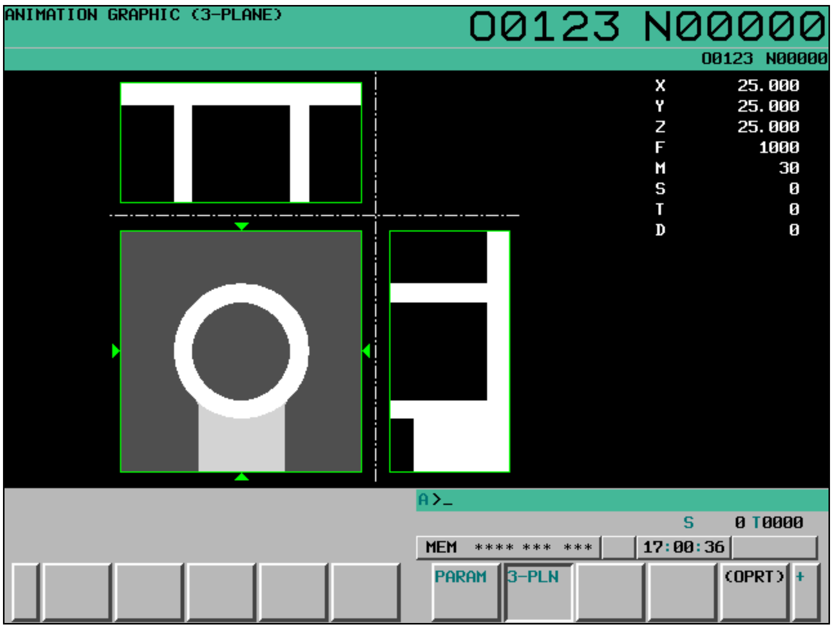


Fig. 13.2.2.3 (d) ANIMATION GRAPHIC (3-PLANE) screen (10.4-inch LCD)

4 Press the [(OPRT)] soft key. The soft keys for three-plane diagram drawing are displayed.

Fig. 13.2.2.3 (e) ANIMATION GRAPHIC (3-PLANE) screen (three-plane diagram operation) (8.4-inch LCD)

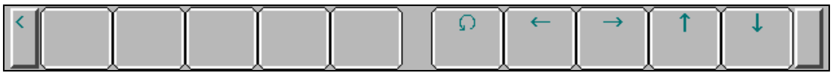


Fig. 13.2.2.3 (f) ANIMATION GRAPHIC (3-PLANE) screen (three-plane diagram operation) (10.4-inch LCD)

Explanation

- Switching the side view display

Use the soft key [↺] displayed in step 4 to perform this operation.
Each time you press the key, the side view display changes as shown below.

Display of the right and top side views
↓
Display of the top and left side views
↓
Display of the left and bottom side views
↓
Display of the bottom and right side views

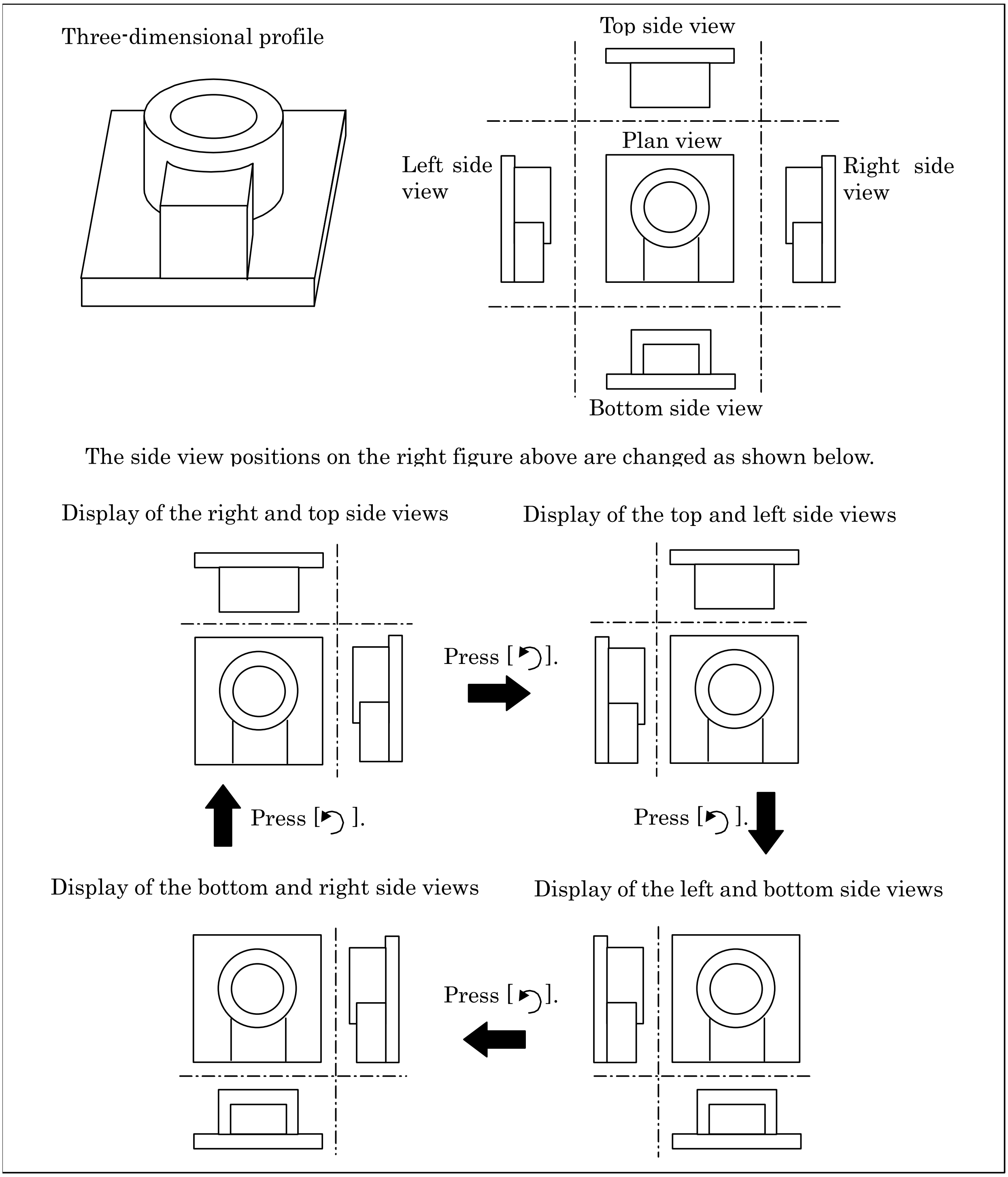


Fig. 13.2.2.3 (g) Display example of three-plane diagram drawing

- Changing the position of a cross section in a side view

Use the following soft keys displayed in step 4 or the MDI cursor keys to specify the position of a cross section in a side view.

- Soft keys [←] and [→], cursor keys [←] and [→]
  Use these keys to horizontally move the marks ▲▼ that indicate the position of a cross section in a left or right side view.
- Soft keys [↑] and [↓], cursor keys [↑] and [↓]
  Use these keys to vertically move the marks ▶◀ that indicate the position of a cross section in a top or bottom side view.

After the above operation, a cross section at the marked position is drawn in a side view.

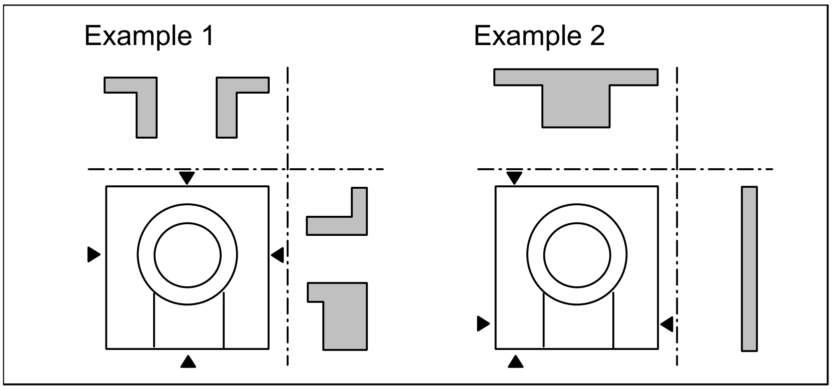


Fig. 13.2.2.3 (h) Display example of cross section position

You can change the cross section position continuously by holding down any of the MDI cursor keys. The amount of change of the cross section position can be modified in the range of 1 to 10 dots by setting parameter No. 6515.

- Method of drawing a three-plane diagram

You can select one of the following methods of drawing a three-plane diagram by setting bit 2 (3PL) of parameter No. 6501.

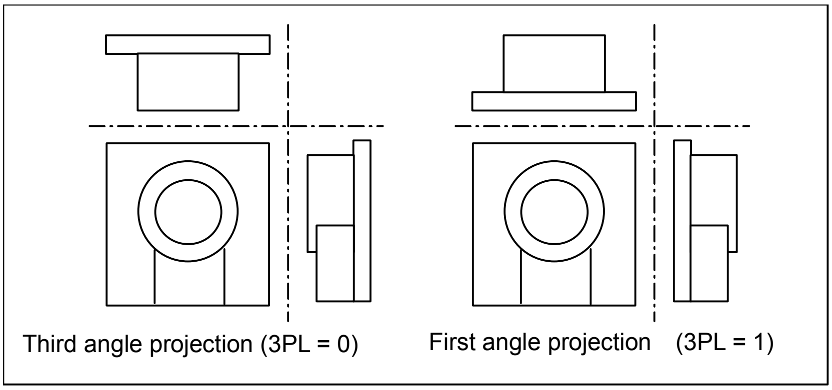


Fig. 13.2.2.3 (i) Selection of method of drawing three-plane diagram

13.2.2.4 Programmable Data Input (G10) for Blank Figure Drawing Parameters

Overview

For animation drawing by the dynamic graphic display function, a programmable data input (G10) command can be used to set the graphic parameters for the blank figure (figure/position/dimensions) in an NC program.

Format

G10 L90 P_ IP_ I_ J_ K_ ;

P_	: Blank figure
IP_	: Addresses of the basic three axes and the reference position of a blank
I_,J_,K_	: Dimensions of a blank

NOTE

1 This command is valid only during animation drawing. It is ignored during normal automatic operation.

NOTE
2 This command is a one-shot G code.
3 This command must be specified in a single block.

Explanation

- Blank figure (P_)

Specify the type of a blank figure with either of the following settings for shapes.

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

The specified value is set in parameter No. 11343.

- Reference position of a blank (IP_)

Specify the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

NOTE
Specify the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

The specified value is set in parameter No. 11344.

- Dimensions of a blank (I_,J_,K_)

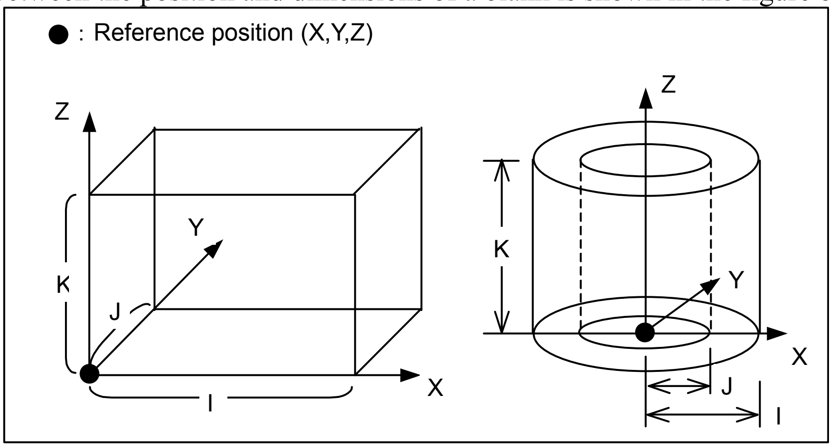
For the shape of each blank, specify the dimensions of the blank as follows:

Blank figure	Address I	Address J	Address K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Radius of column	0	Length of column
Cylinder	Radius of outer circle of cylinder	Radius of inner circle of cylinder	Length of cylinder

The specified value are set in parameter No.11345 (address I), parameter No.11346 (address J), and parameter No.11347 (address K).

- Relationship between the position and dimensions of a blank

The relationship between the position and dimensions of a blank is shown in the figure below.



- Operation to be performed when this command is issued

When this command is executed in the animation drawing operation, the specified values are set in the drawing parameters for the blank figure, reference position, and dimensions that correspond to the specified arguments, and the drawing area is automatically set again with the values set.
This causes this command and subsequent commands to perform drawing in the new drawing area.

> **NOTE**
> When this command is executed, the following occurs for drawing that has been previously performed.
> The command should therefore be executed at the beginning of a program for drawing.
> - The blank figure is initialized.
> - The tool figure is erased.
>   Therefore, be sure to set the radius value of the tool again with G10L91 command (Refer to the next section) after G10L90 is instructed.
> - The drawing start position is made invalid.

13.2.2.5 Programmable Data Input (G10) for Tool Figure Drawing Parameters

Overview

For animation drawing by the dynamic graphic display function, a programmable data input (G10) command can be used to set the graphic parameter for the tool figure (radius) in an NC program.

Format

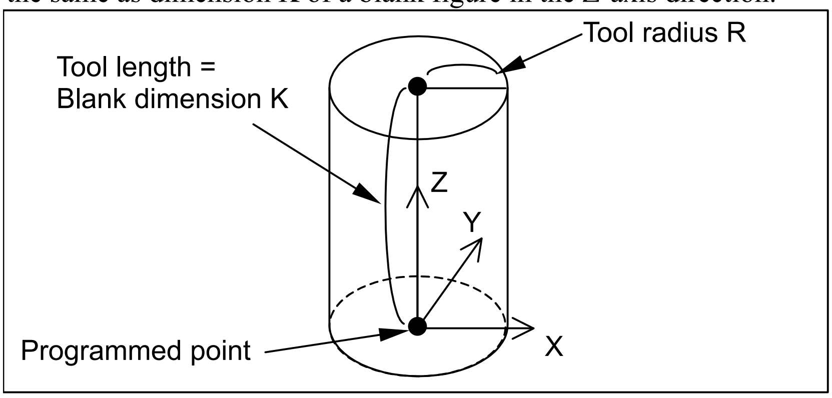
G10 L91 R_ ;
R_ : Tool radius

> **NOTE**
> 1 This command is valid only during animation drawing. It is ignored during normal automatic operation.
> 2 This command is a one-shot G code.
> 3 This command must be specified in a single block.

Explanation

- Tool radius (R_)

Set the radius of a tool figure to be drawn.
The tool length is the same as dimension K of a blank figure in the Z-axis direction.



- Operation to be performed when this command is issued

When this command is executed in the animation drawing operation, the specified value is set in the graphic parameter for the tool figure (radius) that corresponds to the specified argument, and the tool figure is redrawn with the set value.

NOTE

When this command is executed, the following occurs for drawing that has been previously performed.

- The blank figure is updated.

13.2.3 Warning Messages

Warning message	Content
START REJECTED	This program cannot be drawn.
NO PROGRAM SELECTED	No drawing target program is selected.
UNAVAILABLE COMMAND IS IN DRAWING	An NC statement/macro statement that can not execute drawing is instructed.
ILLEGAL SETTING OF GRAPHIC PARAMETERS	Graphic parameters are not set correctly.

13.2.4 Note

NOTE

1 The drawing target axes are the basic three axes.
The basic three axes are the X-axis, Y-axis, and Z-axis on the basic coordinate system set with parameter No.1022.

2 In a drawing program having a sequence of contiguous small blocks, drawing requires longer processing time, possibly causing machining time to become longer than actual machining time.

> **NOTE**
>
> 3 When drawing is executed, data is treated as described below.
>
> (1) Parameters
> The same parameters as for automatic operation are used. However, parameters cannot be rewritten with a command such as the G10 command. If an attempt is made to rewrite a parameter with programmable parameter input (G10L52), the command is ignored.
> However, by setting bit 7 (GST) of parameter No.11329, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING".
>
> (2) Tool compensation value/workpiece origin offset/extended workpiece origin offset/macro variable, etc.
> For each of drawing execution and automatic operation, separate data is used. At the start of drawing, data for drawing is produced by copying data for automatic operation. In subsequent operations, the data for drawing and the data for automatic operation are treated independently of each other.
> So, even if data is rewritten with a command such as G10, drawing execution and automatic operation do not affect each other. Note, however, that data rewritten in drawing execution is not reflected in the data for automatic operation but is deleted.
>
> (3) Tool life management data
> These data items are not used for drawing execution. So, if an attempt is made to rewrite data with G10, the command is ignored.
> However, by setting bit 7 (GST) of parameter No.11329, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING".

13.2.5 Restrictions

- Functions that operate differently in drawing execution and automatic operation

The operations of the following functions in drawing execution differ from the operations in automatic operation:

1. Operations that differ, depending on the custom macro
   1) Interface signal
      #1000 to #1035 are assumed to be 0 at all times.
   2) Message output
      A message to be output with #3006 is not displayed but is ignored.
   3) clock
      #3001 and #3002 are ignored. So, note that drawing is not continued, for example, if the following is specified:
      #3001=0 ;
      WHILE [#3001 LE 100] DO1 ;
      END1 ;
   4) Mirror image
      #3007 is assumed to be 0 at all times.
   5) Program-restarted state
      #3008 is assumed to be 0 at all times.
   6) External output command
      BPRNT, DPRNT, POPEN, and PCLOS are ignored.

2 Functions that perform partly different operations
   1) When G28 (automatic reference position return) is specified, up to the intermediate point is drawn.
   2) When G29 (automatic return from the reference position) is specified, drawing is performed from the intermediate point.
   3) When G27 (reference position return check) is specified, no reference position return check is made.
   4) No stored stroke limit check is made.
   5) When G31 (skip function), or G31.1, G31.2, or G31.3 (multistage skip function) is specified, drawing is performed up to the specified position, regardless of the skip signal.
   6) When G60 (single direction positioning) is specified, drawing is performed directly up to the specified position at all times even if the positioning direction is opposite.

3. Functions that perform different operations
   If the following functions are specified, the operations described below result:
   1) G07.1 (cylindrical interpolation)
      Linear interpolation is performed only with linear axes.

- Functions that do not support drawing

The following functions are ignored in drawing execution:
1) G04 (dwell)
2) G20, G21 (inch/metric switch)
3) Auxiliary function (M, S, T, B)
4) G22, G23 (stored stroke limit check on/off)
5) G10 (programmable data input for other than the graphic parameters for the blank figure/tool figure)

> **NOTE**
> If G10 (programmable data input) is specified, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING" by setting bit 7 (GST) of parameter No.11329.

The following functions are warning in drawing execution:
1) M198 (external subprogram call)

- Functions that support drawing

The operations performed with the following program commands can be drawn:
1) G00 (Positioning)
2) G01 (Linear interpolation)
3) G02/G03 (Circular interpolation)
   However, helical interpolation with the instruction axis other than basic three axes cannot be drawn.
4) G17/G18/G19 (Plane selection)
5) G33 (Threading)
   However, it is drawn as linear interpolation.
6) G40/G41/G42 (Cutter compensation/cancel)
7) G52 (Local coordinate system)
8) G53 (Machine coordinate system selection)
9) G54 to G59 (Workpiece coordinate system selection)
10) G54.1 (Extended workpiece coordinate system selection)
11) G65 (Macro call)
12) G68/G69 (Coordinate system rotation/cancel)
13) G90/G91 (Absolute/incremental command)
14) G92 (Workpiece coordinate system change)
15) G92.1 (Workpiece coordinate system preset)
16) G94/G95 (Feed per minute/Feed per rotation)

17) G96/G97 (Constant surface speed control/cancel)
18) M98 (Subprogram call)
19) G73/G74/G76/G81/G82/G83/G84/G85/G86/G87/G88/G89/G80
(Canned cycle for drilling)

NOTE

1 It is possible to draw with the G68 (Coordinate system rotation) instruction only in the tool path drawing.
And, the display of coordinates when instructing in G68 is a coordinate value on the workpiece coordinate system.
2 In animation drawing, shape by the movement of back boring cycle command is different from actual shape.
3 In animation drawing, movement to shift amount at the bottom of a hole in the fine boring cycle and back boring cycle command is not drawn by setting bit 1 (ABC) of parameter No.11349=0.

- Operations that do not support drawing

The following operations cannot be drawn:

1) Backward movement based on the retrace function
2) Forward movement/backward movement/resumed forward movement based on the manual handle retrace function
3) Movement made with skip signal input for a skip command or multistage skip command
4) Movement on an axis based on PMC axis control
5) Operation based on manual interrupt, manual handle interrupt, etc.
6) Operation based on the high-speed, high-precision function (AI advanced preview control, AI contour control, AI contour control II)

- Drawing start position

In tool path drawing, if G92, G52, or G92.1 is specified at the start of a drawing target program, the position specified with the G code is the drawing start position. If none of these G codes is specified, the end point of the first moving command is the drawing start position.
In animation drawing, the current position when the tool figure is displayed is the drawing start position.
If the value where to be drawn is absolute coordinate value (bit 3 (BGM) of parameter No.11329=0), the origin setting by modal coordinate system command (work coordinate system and local coordinate system) on the automatic operation side is not reflected in the value where to be drawn.

- Use of this function with other functions

When this function is specified, the following functions cannot be used:

- Graphic display
- MANUAL GUIDE *i*

- VGA display based on the C language executor

If VGA display based on the C language executor is used, the VGA window is erased when the screen display is switched to a screen of this function.
So, when using the VGA window, determine the screen of this function by screen number and close the VGA window before switching the screen display.

- Use of the CNC screen display function

The following restriction exists when the screen of dynamic graphic display function is displayed by the CNC screen display function.

1) The drawing screen is not displayed in the CNC screen display function by way of via embedded Ethernet. Please use the CNC screen display function by way of the FAST Ethernet board.
2) The drawing screen is not displayed in the CNC screen dual display function.

3) Please do not start / shut down the CNC screen display function when you display the drawing screen. Please start / shut down the CNC screen display function after it switches to other screens.

- Use of the Screen hard copy function

Drawing is temporarily stopped when the drawing screen is copied with the screen hard copy function. The drawing can be restarted by pushing [RESTART] soft key if necessary after the copy operation is finished.

13.3 DYNAMIC GRAPHIC DISPLAY (T SERIES)

Overview

The dynamic graphic display function enables the display of a machining travel path without actually running the machine.
With the dynamic graphic display function, the machine need not be actually run as with the graphic display function. Before starting path drawing, however, you should select MEM mode by the mode switch on the machine operator's panel and set conditions that can be used to start actual NC operation of the machine.

13.3.1 Graphic Parameter Screen

The graphic parameter screen used to make necessary settings for tool path drawing and the methods of making these settings are the same as those for the graphic display function.
See Section 13.1, "GRAPHIC DISPLAY".

13.3.2 Path Drawing

Explanation

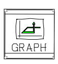
The PATH GRAPHIC screen is used to draw a tool path. The following operations can be performed:

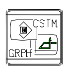
- Starting/ending tool path drawing
- Rewinding a drawing target machining program
- Erasing a drawn tool path
- Enlarging/reducing the graphic range

Procedure for drawing a path on the path graphic screen

To draw a tool path, select MEM mode by the mode switch on the machine operator's panel and set conditions that can be used to start actual NC operation of the machine.
Then, use the following procedure to perform drawing.

Procedure

1 Press the function key 

(or 

when a small MDI unit is used) and set necessary graphic parameters on the GRAPHIC PARAMETER screen.

2 Press the [PATH EXEC] soft key. The PATH GRAPHIC screen is displayed.

3 Press the [(OPRT)] soft key. The soft keys for tool path drawing are displayed.

Fig. 13.3.2 (a) PATH GRAPHIC (EXECUTION) screen (operation) (8.4-inch LCD)

Fig. 13.3.2 (b) PATH GRAPHIC (EXECUTION) screen (operation) (10.4-inch LCD)

> **NOTE**
> A travel path cannot be drawn if axis movement is disabled due to the start lock or interlock state. Release the lock state before starting drawing.

- Starting drawing

- [EXEC] soft key
  Drawing is performed continuously until the M02 or M30 block in the program is reached.
  When the single block switch on the machine operator's panel is turned on and drawing is started, a single block stop operation is performed after drawing only for a single block.
- [PROCESS] soft key
  If drawing is started by pressing the [PROCESS] soft key, a single block stop operation is performed at the M00 or M01 block in the machining program when that block is executed.
  To restart the drawing, press the [EXEC] or [PROCESS] soft key again.

- Stopping drawing

- [STOP] soft key
  During drawing started by pressing the [EXEC] or [PROCESS] soft key, a single block stop operation can be performed by pressing the [STOP] soft key.
  To restart the drawing, press the [EXEC] or [PROCESS] soft key again.

- Erasing a drawn tool path

- [ERASE] soft key
  A drawn path can be erased by pressing the [ERASE] soft key.

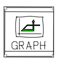
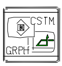
- Rewind of a drawing target program

- [REWIND] soft key
  The machining program can be rewound by pressing the [REWIND] soft key.
  This soft key is enabled when drawing is stopped.

Procedure for enlarging or reducing a drawn path on the path graphic screen

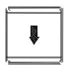
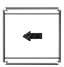
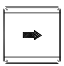
On the tool path graphic screen, you can move the center position of the tool path drawing or enlarge or reduce the tool path drawing while viewing the drawn tool path.
If any of these operations is executed, the tool path already drawn is cleared.

Procedure

1 Press the function key  (or  when a small MDI unit is used).
2 Press the [GRAPH] soft key to display the PATH GRAPHIC screen, and then draw a tool path.
3 Press the return menu key .
4 Press the [LARGE] soft key.

- Procedure for changing the graphic range by setting a graphic center and magnification

The graphic center position can be moved. At the same time, the scale can also be changed. So, the tool path can be enlarged or reduced at a desired new center position.

5 After step 4 described above, press the [CENTER] soft key.
A yellow cursor appears at the center of the screen, and the soft key display is changed.
6 Move the yellow cursor to a new graphic center position by using the cursor key , , , or .

7 When changing the scale, key in a value from 0.01 to 100 (magnification) then press the [INPUT] soft key. An input value is displayed at "SCALE" in the lower-right corner of the screen.
When you press the [+INPUT] soft key, the current magnification is incremented by an input value.

8 Press the [EXEC] soft key to end the operation.
After this step, the setting made in the above steps is effective to enable drawing in the new graphic range.

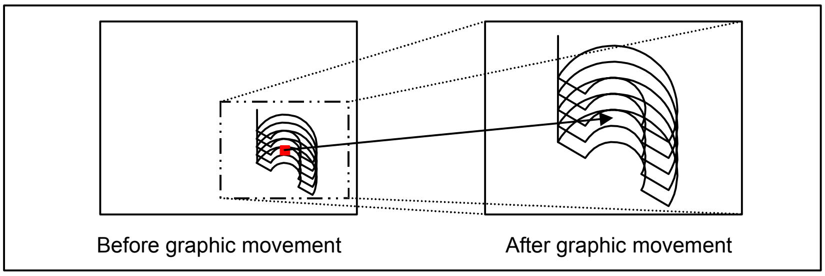


Fig. 13.3.2 (c) Graphic movement (magnification = 2.00)

- Procedure for changing the graphic range with a rectangle

A tool path can be drawn by enlarging a specified rectangular area.

5 After step 4 described above, press the [AREA] soft key.
Two cursors, one in red and the other in yellow, appear at the center of the screen, and the soft key display is changed.

6 Move the yellow cursor by using the cursor key [↑], [↓], [←], or [→]. The cursor to be moved can be switched by pressing the [HI/LO] soft key.
Move the two cursors to the diagonal points of a new rectangular graphic range. A tool path is drawn next time so that the drawn tool path is contained in this rectangular range.

7 Press the [EXEC] soft key to end the operation.
After this step, the setting made in the above steps is effective to enable drawing in the new graphic range.

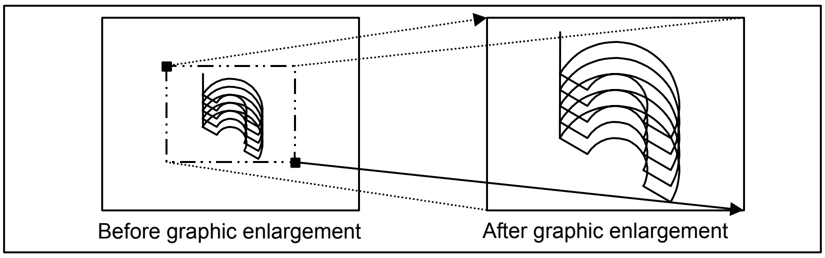


Fig. 13.3.2 (d) Graphic enlargement

NOTE

1 To stop an enlargement/reduction operation, press the [CANCEL] soft key.

2 Even if you perform an enlargement/reduction operation, the tool path already drawn on the screen is neither moved nor enlarged. The enlargement/reduction setting is effective next time drawing is performed.

13.3.3 Restrictions

- Tool registration command in the tool life management function

During drawing, tool registration (G10L3 command) in the tool life management function is disabled. You can therefore use the G10L3 command to register a new tool only during normal automatic operation.

14 VIRTUAL MDI KEY FUNCTION

Chapter 14, "VIRTUAL MDI KEY FUNCTION", consists of the following sections:

14.1 VIRTUAL MDI KEY

Overview

This function is used to perform program editing and changing of various data using the keyboard displayed on the LCD with a touch panel.

Screen on which a CNC screen is displayed in the upper left 1/4 area

Explanation

A CNC screen equivalent to that displayed on 8.4-inch LCD is displayed in the upper left area of the screen, and the keyboard is displayed in the remaining area.

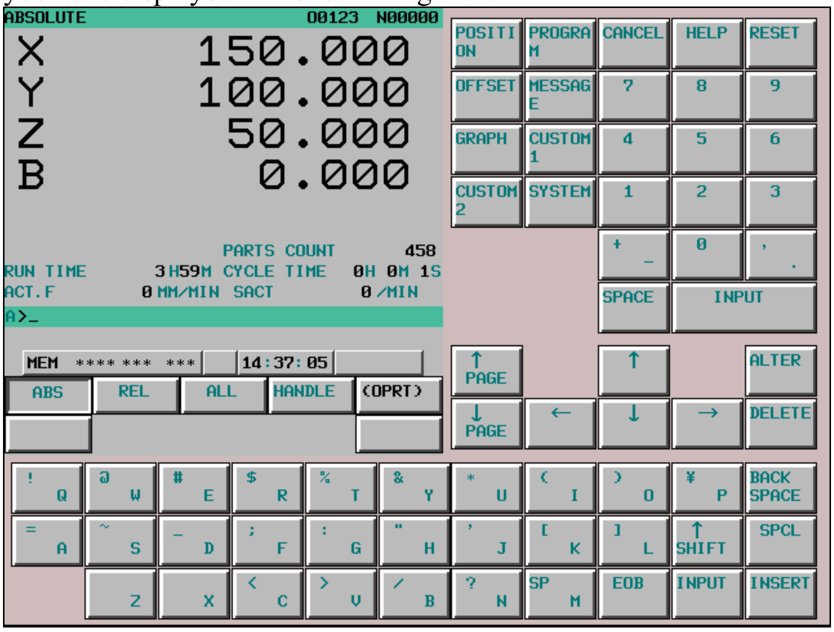


Figure 14.1 (a) Screen on which a CNC screen is displayed in the upper left area

Operation

- Input key

The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.

- Cancel key

The displays "BACK SPACE" and "CANCEL" on the virtual MDI keyboard are equivalent to the cancel key.

- Shift key

The display "↑ SHIFT" on the virtual MDI keyboard is equivalent to the shift key.
Pressing the "↑ SHIFT" key once places the system in the shift state, and pressing the "↑ SHIFT" key again or another key releases the system from the state.
Pressing a key in the shift state causes the character indicated in the upper left corner of the key to be entered.

- Simultaneous pressing of two keys

The operation to be performed for pressing two key simultaneously, such as the "CANCEL" and "RESET" keys to erase alarm SW0100, is as follows:

(1) Press the "SPCL" key. The "SPCL" key is held down, and places the system in SPCL mode.
(2) Press the keys to be pressed simultaneously, one at a time.
(3) Press the "INPUT" key. Of the keys pressed in SPCL mode, the last two are assumed to have been pressed simultaneously. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.
Example: "SPCL" → "CANCEL" → "RESET" → "INPUT"

NOTE

1 In SPCL mode, other keys are disabled until the "SPCL" or "INPUT" key is pressed.
2 Pressing the "SPCL" key in "SPCL" mode causes all keys pressed in SPCL mode to be disabled. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.

Screen on which a CNC screen is displayed in the entire area

Explanation

A CNC screen equivalent to that displayed on 8.4-inch LCD is displayed in the entire area. Function keys and the ON/OFF button for virtual keys are displayed at the bottom of the screen.

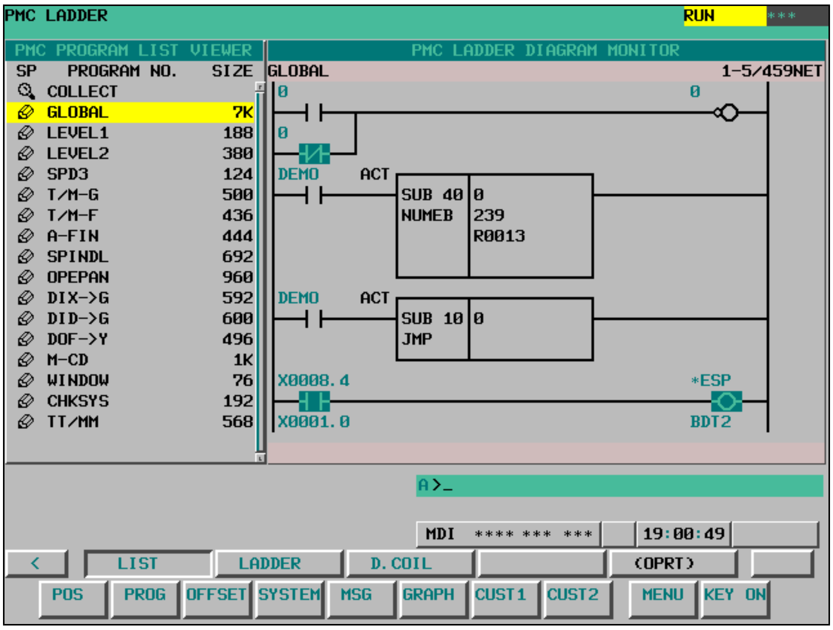


Figure 14.1 (b) Screen on which a CNC screen is displayed in the entire area

Operation

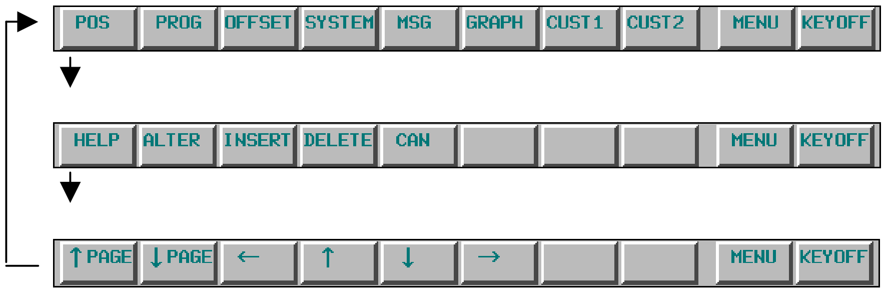
- Function key page switching

Pressing "MENU" located near the lower right corner of the screen switches the screen to page 1, page 2, page 3, and back to page 1 in this order.

Function keys on page 1

Function keys on page 2

Function keys on page 3



- Display of virtual keys

Pressing "KEY ON" located at the lower right corner of the screen displays virtual MDI keys. The character string on the key top changes to "KEYOFF".
Pressing "KEYOFF" hides virtual MDI keys.

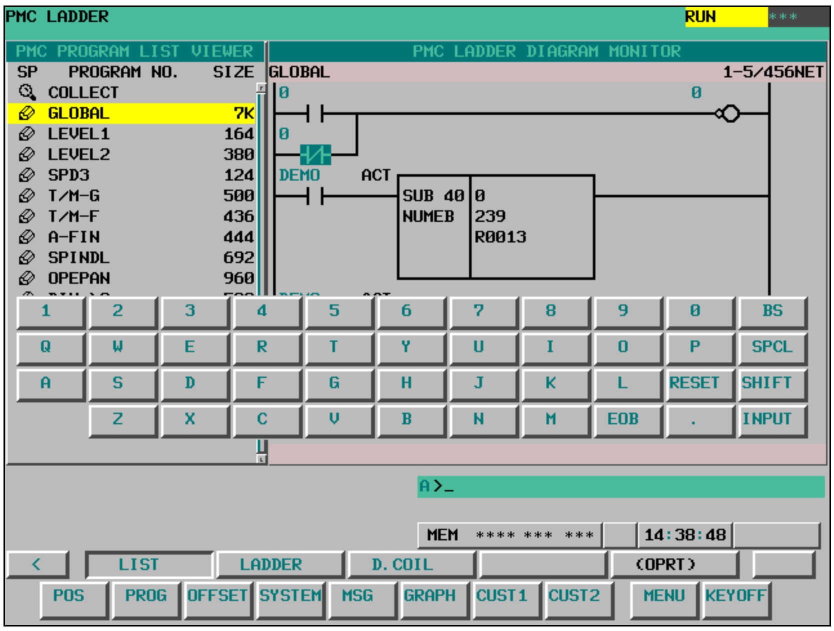


Figure 14.1 (c) State in which virtual keys are ON

- Input key

The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.

- Cancel key

The displays "BS" on the virtual MDI keyboard are equivalent to the cancel key. (The function key "CAN" is also equivalent to the cancel key.)

- Shift key

The display "SHIFT" on the virtual MDI keyboard is equivalent to the shift key. The characters on the key tops change each time the shift key is pressed. (The characters that can be input are displayed.)

Figure 14.1 (d) Key tops in the shift state

- Simultaneous pressing of two keys

The operation to be performed for pressing two key simultaneously, such as the "CAN" and "RESET" keys to erase alarm SW0100, is as follows:

(1) Press the "SPCL" key. The "SPCL" key is held down, and places the system in SPCL mode.
(2) Press the keys to be pressed simultaneously, one at a time.
(3) Press the "INPUT" key.
Example: "SPCL" → "CAN" → "RESET" → "INPUT"

NOTE

1 In SPCL mode, other keys are disabled until the "SPCL" or "INPUT" key is pressed.
2 Pressing the "SPCL" key in "SPCL" mode causes all keys pressed in SPCL mode to be disabled. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.

14.1.1 Limitations

- Display of VGA windows on the C language executor

This function uses one VGA window, so that the number of VGA windows that can be used on the C language executor is reduced by one.

- LCDs

LCDs on which this function can be used are 10.4-inch LCDs with a touch panel.

IV. MAINTENANCE

1 ROUTINE MAINTENANCE

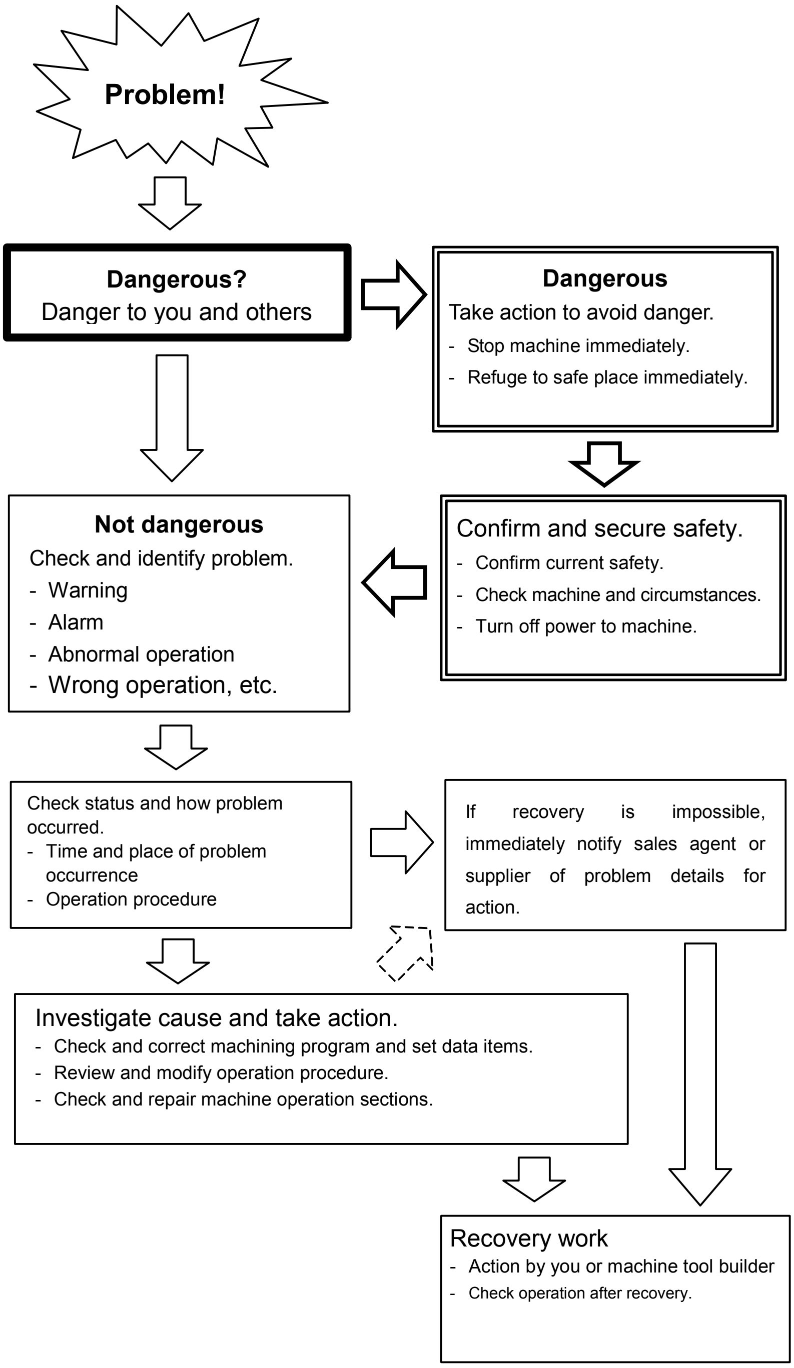
This chapter describes routine maintenance work that the operator can perform when using the CNC.

> ⚠ **WARNING**
> Only those persons who have been educated for maintenance and safety may perform maintenance work not described in this chapter.

Chapter 1, "ROUTINE MAINTENANCE", consists of the following sections:

1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED

If an unexpected operation occurs or an alarm or warning is output when the CNC and machine are used, the problem needs to be solved quickly. For this purpose, the status of the problem must be identified correctly, and a proper action must be taken.
The procedure for taking an action for a problem is shown below.



For details of investigation and action on problems arising from the CNC, refer to "TROUBLESHOOTING PROCEDURE" in the Maintenance Manual (B-64305EN) issued by FANUC.

1.2 BACKING UP VARIOUS DATA ITEMS

With the CNC, various data items such as offset data and system parameters are stored in the SRAM of the control unit and are protected by a backup battery. However, an accident can erase the data. By storing the data at another location (outside the CNC), the data, when lost, can be restored.
So, when the machine is started up or data is updated, for example, the data should be backed up (stored outside the CNC).

- Data backup operation

The data items listed below should be backed up. For the method of data output operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

<1> Machining programs
→ See III-8.2.1.
<2> System parameters
→ See III-8.2.2.
<3> Tool offset data
→ See III-8.2.3.
<4> PMC data
→ See PMC PROGRAMMING MANUAL (B-64393EN).
<5> Pitch error compensation data (when the pitch error compensation function is selected.)
→ See III-8.2.4.
<6> Custom macro variables (when the custom macro function is selected.)
→ See III-8.2.5.
<7> Workpiece coordinate system setting data (when the workpiece coordinate system function is selected.)
→ See III-8.2.6.
<8> Operation history data
→ See III-8.2.7.

It is recommended that recording media (such as floppy disks and memory cards) daily used with the machine be used to store data. Stored data should be managed properly so that the data can be restored quickly if a problem occurs.

- Data restoration work

In order to restore lost data to the state of the stored data, input the data backed up according to the previous item into the CNC. For the method of data input operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

⚠ **WARNING**

After inputting stored data, do not start an operation immediately. Instead, check that the data is input correctly and that settings are made to meet a desired operation.
If an operation is executed without making this check, the machine and workpiece can be damaged and personal injury can occur due to an unexpected machine movement. Use sufficient care.

⚠ **CAUTION**

Before recovery of the following data items, consult with the machine tool builder of the machine used:
- System parameters
- PMC data
- Macro programs and custom macro variables
- Pitch error compensation values

> **NOTE**
> The method of recovery described in this section is intended just to restore the state of the backed up data, and does not guarantee recovery of the state that was present when the data was lost.

1.3 METHOD OF REPLACING BATTERY

This chapter describes how to replace the CNC backup battery and absolute Pulsecoder battery. This section consists of the following subsections::

1.3.1 Replacing Battery for CNC Control Unit

Battery for memory backup

Offset data, and system parameters are stored in SRAM in the control unit. The power to the SRAM is backed up by a lithium battery mounted on the front panel of the control unit. The above data is not lost even when the main battery goes dead. The backup battery is mounted on the control unit at shipping. This battery can maintain the contents of memory for about a year.
When the voltage of the battery becomes low, alarm message "BAT" blinks on the display and the battery alarm signal is output to the PMC. When this alarm is displayed, replace the battery as soon as possible. In general, the battery can be replaced within two or three weeks, however, this depends on the system configuration.
If the voltage of the battery becomes any lower, memory can no longer be backed up. Turning on the power to the control unit in this state causes system alarm to occur because the contents of memory are lost. Clear the entire memory and reenter data after replacing the battery.
FANUC thus recommends that the battery be replaced periodically, once a year, regardless of whether a battery alarm is issued.
The following two kinds of batteries can be used.

- Lithium battery, incorporated into the CNC control unit.
- Two alkaline dry cells (size D) in an external battery case.

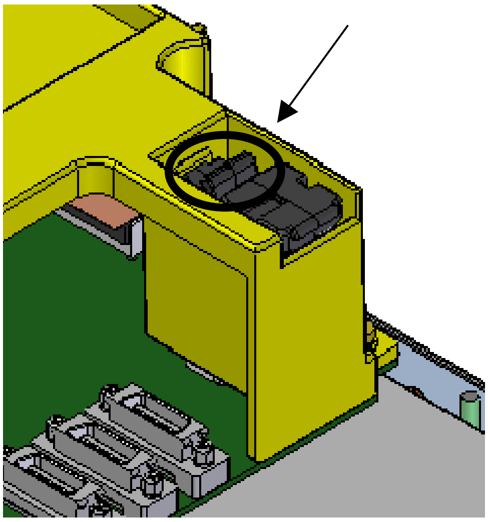
> **NOTE**
> A lithium battery is installed as standard at the factory.

When a lithium battery is used (LCD mounted type)

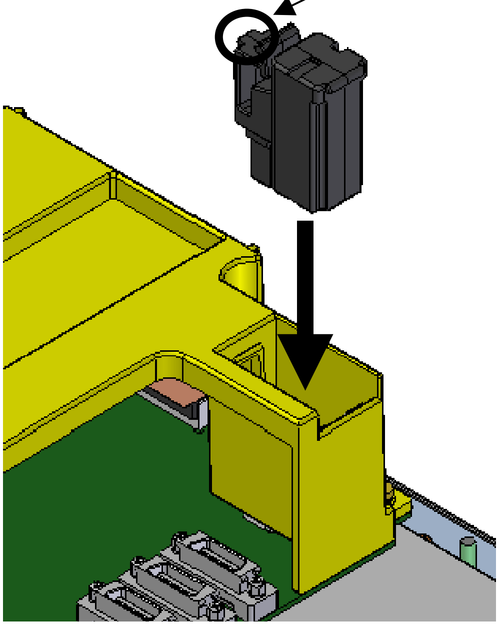
- Replacement procedure

Prepare a new battery unit (ordering code: A02B–0309–K102).

(1) Turn on the power to the CNC. After about 30 seconds, turn off the power.
(2) Extract the old battery unit from the lower right of the rear of the CNC unit. (Hold the latch of the battery unit, and extract the unit upward while releasing the claw from the case.)



(3) Mount the new battery unit. (Push the battery unit in until the claw is latched into the case.) Ensure that the latch is engaged securely.



> ⚠ **WARNING**
> Using other than the recommended battery may result in the battery exploding. Replace the battery only with the specified battery (A02B-0309-K102).

> ⚠ **CAUTION**
> Steps (1) to (3) should be completed within 30 minutes. Do not leave the control unit without a battery for any longer than the specified period. Otherwise, the contents of memory may be lost.
> If steps (1) to (3) may not be completed within 30 minutes, save all contents of the SRAM memory to the memory card beforehand. Thus, if the contents of the SRAM memory are lost, the contents can be restored easily.
> See Chapter 5, "INPUT AND OUTPUT OF DATA" or Appendix C, "BOOT SYSTEM" of maintenance manual (B-64305EN) for explanations about how to save the contents of the SRAM memory.

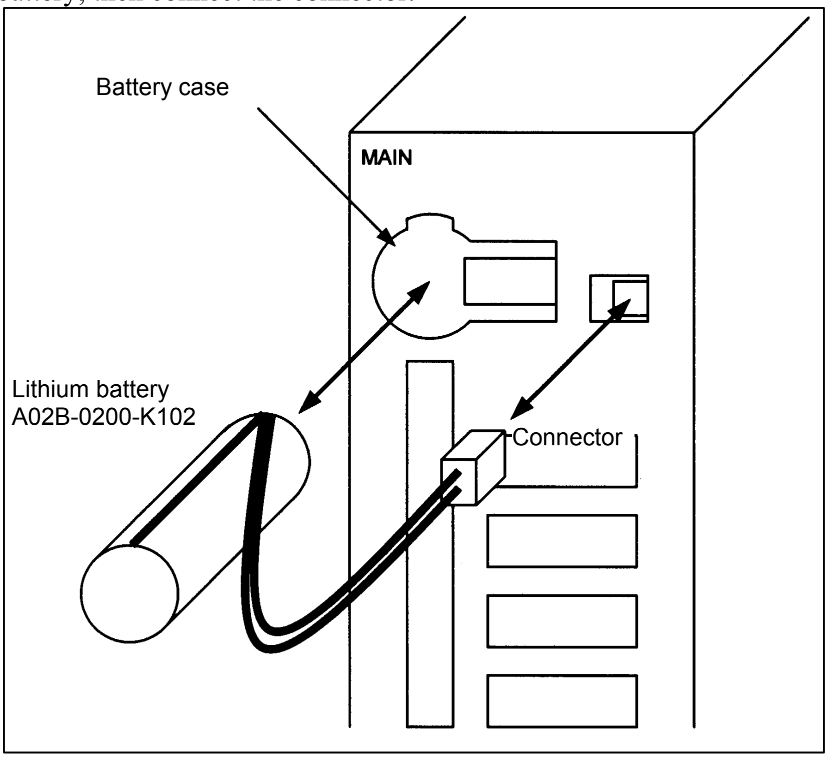
When discarding a battery, observe the applicable ordinances or other rules of your local government. In addition, cover the exposed pins with tape or other insulation materials to prevent a short circuit before discarding the battery.

When a lithium battery is used (Stand-alone type)

- Replacement procedure

If a lithium battery is used, have A02B-0200-K102 (FANUC internal code: A98L-0031-0012) handy.

1 Turn the CNC on. About 30 seconds later, turn the CNC off.
2 Remove the battery from the top area of the CNC unit.
  Disconnect the connector first. Then, remove the battery from the battery case. The connector is not latched. Simply pulling the cable detaches the connector.
  The battery case is provided in the top area of the face plate of the main board.
3 Replace the battery, then connect the connector.



> ⚠ **WARNING**
> The incorrect mounting of the battery may cause an explosion. Avoid using any battery other than the one specified here (A02B-0200-K102).

> ⚠ **CAUTION**
> Complete steps 1 to 3 within 30 minutes.
> If the battery is left removed for a long time, the SRAM would lose the contents.
> If there is a danger that the replacement cannot be completed within 30 minutes, save the whole contents of the SRAM to a memory card. The contents of the memory can be easily restored with the memory card in case the memory loses the contents.
> See Chapter 5, "INPUT AND OUTPUT OF DATA" or Appendix C, "BOOT SYSTEM" of maintenance manual (B-64305EN) for explanations about how to save the contents of the SRAM memory.

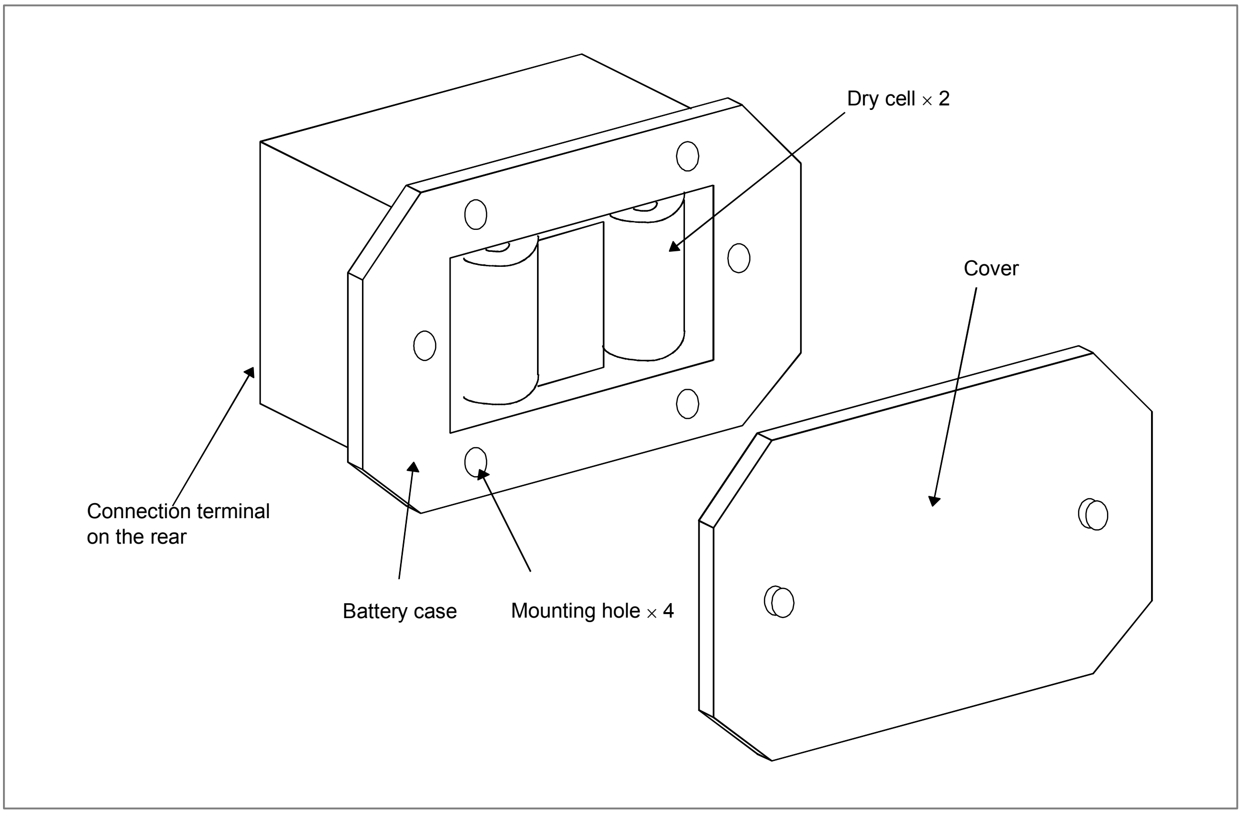
Discard the dead battery, observing appropriate municipal rules and regulations. When discarding the battery, insulate the terminal with a tape so that no short-circuit would occur.

When alkaline dray cells (size D) are used

- Replacing the battery

(1) Prepare two new alkaline dry cells (size D).
(2) Turn on the power of the control unit.
(3) Remove the battery case cover.
(4) Replace the batteries, paying careful attention to their orientation.
(5) Replace the battery case cover.

> ⚠ **CAUTION**
> To replace the battery when the power is off, follow the same procedure as that for the replacement of a lithium battery, described above.



1.3.2 Battery for Absolute Pulsecoders

- When the voltage of the batteries for absolute Pulsecoders becomes low, alarm 307 or 306 occurs, with the following indication in the CNC state display at the bottom of the CNC screen.

  Alarm 307 (alarm indicating the voltage of the battery becomes low) :
  The indication "APC" blinks in reversed display.
  Alarm 306 (battery zero alarm) :
  The indication "ALM" blinks in reversed display.

- When alarm 307 (alarm indicating the voltage of the battery becomes low) occurs, replace the battery as soon as possible. In general, the battery should be replaced within one or two weeks, however, this depends on the number of Pulsecoders used.
- When alarm 306 (battery zero alarm) occurs, Pulsecoders are reset to the initial state, in which absolute positions are not held. Alarm 300 (reference position return request alarm) also occurs, indicating that reference position return is required.
- In general, replace the batteries periodically within the service life listed below.

  - A06B-6050-K061 or D-size alkaline dry cells (LR20) :
    Two years (for each six-axis configuration)
  - A06B-6073-K001 :
    Two years (for each three-axis configuration)
  - A06B-6114-K504 :
    One year (for each three-axis configuration)

NOTE
The above values indicate the estimated service life of batteries used with FANUC absolute Pulsecoders. The actual battery service life depends on the machine configuration based on, for example, detector types. For details, contact the machine tool builder.

- Replacing batteries

To prevent absolute position information in absolute Pulsecoders from being lost, turn on the machine power before replacing the battery. The replacement procedure is described below.

(1) Ensure that the power to the servo amplifier is turned on.
(2) Ensure that the machine is in the emergency stop state (the motor is inactive).
(3) Ensure that the DC link charge LED of the servo amplifier is off.
(4) Detach the old batteries and attach new ones.

The replacement of the batteries in a separate battery case and the replacement of the battery built into the servo amplifier are described below in detail.

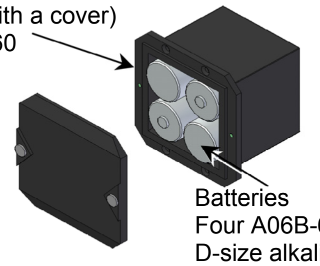
⚠ **WARNING**

- The absolute Pulsecoder of each of the αi/αi S series servo motors and the βi S series servo motors (βi S0.4 to βi S22) has a built-in backup capacitor. Therefore, even when the power to the servo amplifier is off and the batteries are replaced, reference position return is not required if the replacement completes within less than 10 minutes. Turn the power on and replace the batteries if the replacement will take 10 minutes or more.
- To prevent electric shock, be careful not to touch metal parts in the power magnetics cabinet when replacing the batteries.
- Because the servo amplifier uses a large-capacitance electrolytic capacitor internally, the servo amplifier remains charged for a while even after the power is turned off. Before touching the servo amplifier for maintenance or other purposes, ensure your safety by measuring the residual voltage in the DC link with a tester and confirming that the charge indication LED (red) is off.
- Be sure to replace the batteries with specified ones. Pay attention to the battery polarity. If a wrong type of battery is used or a battery is installed with incorrect polarity, the battery may overheat, blow out, or catch fire, or the absolute position information in the absolute Pulsecoders may be lost.
- Ensure that the battery connector is inserted in the correct position.

- Replacing the Batteries in a Separate Battery Case

Use the following procedure to replace the batteries in the battery case.

(1) Loosen the screws on the battery case and detach the cover.
(2) Replace the batteries in the case (pay attention to the polarity).
(3) Attach the cover to the battery case.



⚠ **CAUTION**

- Four D-size alkaline dry cells (LR20) that are commercially available can be used as batteries. A set of four A06B-6050-K061 batteries is optionally available from FANUC.
- Replace all the four batteries with new ones. If old and new batteries are mixed, the absolute position information in the absolute Pulsecoders may be lost.

- Replacing the Battery Built into the Servo Amplifier

Use the following procedure to replace the special lithium battery.

(1) Detach the battery cover.
(2) Replace the special lithium battery.
(3) Attach the battery cover.

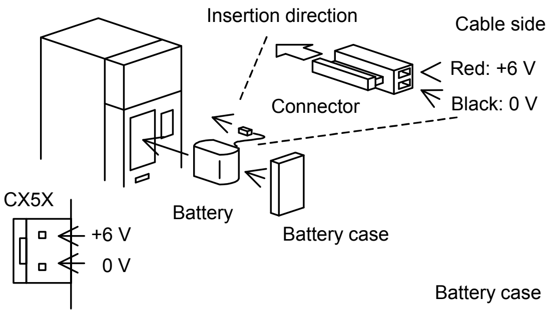
> ⚠ **CAUTION**
> - Purchase the battery from FANUC because it is not commercially available. It is therefore recommended that you have a backup battery.
> - When the built-in battery is used, do not connect BATL (B3) of connector CXA2A/CXA2B. Also, do not connect two or more batteries to the same BATL (B3) line. These connections are dangerous because battery output voltages may be short-circuited, causing the batteries to overheat.
> - Install the battery in the servo amplifier in a direction that allows slack in the cable. If the battery cable is under tension, a bad connection may occur.
> - If the +6 V pin and 0 V pin are short-circuited, the battery may overheat, blow out, or catch fire, or the absolute position information in the absolute Pulsecoders may be lost.
> - When inserting the connector, align it to the connector pins.

[Connecting the battery]

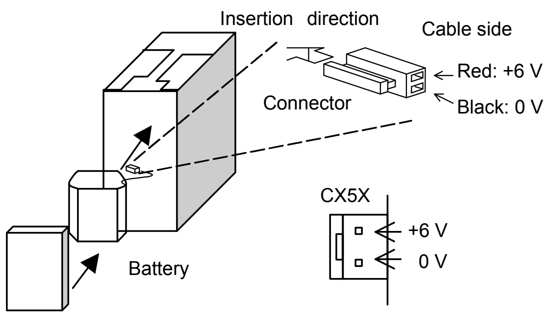
The battery for the βiSV4 and βiSV20 series amplifiers is mounted in the battery case on the underside of each of the amplifiers.

The battery for the other βi series amplifiers and the αi series amplifiers is mounted at the front of each of the amplifiers.

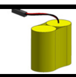
[α*i* series][β*i* series β*i* SV40, β*i* SV80, β*i* SVSP]



[β*i* series β*i* SV4, β*i* SV20]



[Battery sets and outlines]

Battery ordering drawing number	Manufacturer model number	Applicable servo amplifier	Battery case ordering drawing number	Outline
A06B-6114-K504 (Note)	BR-2/3AGCT4A (Panasonic)	α*i* series 60/90 mm width	A06B-6114-K505	
		α*i* series 150/300 mm width	A06B-6114-K506	
		β*i* series β*i* SV (two-axis model)	A06B-6114-K505	
		β*i* series β*i* SVSP	A06B-6114-K506	
A06B-6093-K001	BR-AGCF2W (Panasonic)	β*i* series β*i*SV4, β*i*SV20	A06B-6093-K002	
		β*i* series β*i*SV40, β*i*SV80	A06B-6093-K002	

> **NOTE**
> When an old type BR-CCF2TH battery is used, order a battery case that accommodates battery A06B-6114-K504.

- Used batteries

Old batteries should be disposed as "INDUSTRIAL WASTES" according to the regulations of the country or autonomy where your machine has been installed.

APPENDIX

A PARAMETERS

This manual describes all parameters indicated in this manual.
For those parameters that are not indicated in this manual and other parameters, refer to the parameter manual.

> **NOTE**
> A parameter that is valid with only one of the path control types for the lathe system (T series) and machining center system (M series) is indicated in the upper or lower row as described below. A blank represents an unusable parameter.
>
> [Example 1]
> The parameter HTG is common to the T series and M series, and RTV and ROC are parameters used with the T series only.
>
> | | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 | |
> |---|---|---|---|---|---|---|---|---|---|
> | 1403 | RTV | | HTG | ROC | | | | | T series |
> | | | | HTG | | | | | | M series |
>
> [Example 2]
> The following parameter is used with the M series only:
>
> | | | |
> |---|---|---|
> | 1411 | | T series |
> | | Cutting feedrate | M series |

A.1 DESCRIPTION OF PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
0000							ISO	TVC

[Input type] Setting input
[Data type] Bit path

#0 TVC TV check
0: Not performed
1: Performed

#1 ISO Code used for data output
0: EIA code
1: ISO code

> **NOTE**
> 1 The I/O setting of a memory card is made by bit 0 (ISO) of parameter No. 0139.
> 2 The I/O setting of data server is made by bit 0 (ISO) of parameter No. 0908.

	#7	#6	#5	#4	#3	#2	#1	#0
0001							**FCV**	

[Input type] Setting input
[Data type] Bit path

#1 FCV Program format
0: Series 0 standard format
(This format is compliant with the Series 0*i*-C.)
1: Series 10/11 format

> **NOTE**
> 1 Programs created in the Series 10/11 program format can be used for operation on the following functions:
> 1 Subprogram call M98,M198
> 2 Thread cutting with equal leads G32 (T series)
> 3 Canned cycle G90, G92, G94 (T series)
> 4 Multiple repetitive canned cycle G71 to G76 (T series)
> 5 Drilling canned cycle G80 to G89 (T series)
> G73, G74, G76, G80 to G89(M series)
> 2 When the program format used in the Series 10/11 is used for this CNC, some limits may add. Refer to the OPERATOR'S MANUAL.

	#7	#6	#5	#4	#3	#2	#1	#0
0010							**PRM**	

[Input type] Setting input
[Data type] Bit path

#1 PRM Whether the parameter whose setting is 0 is output or not:
0: It is selected with soft key [ALL] or [NON-0].
1: It is not selected with soft key [ALL] or [NON-0]. The parameter whose setting is 0 is not output.

	#7	#6	#5	#4	#3	#2	#1	#0
0012								**MIRx**

[Input type] Setting input
[Data type] Bit axis

#0 MIRx Mirror image for each axis
0: Mirror image is off. (Normal)
1: Mirror image is on. (Mirror)

0020	**I/O CHANNEL : Input/output device selection, or interface number for a foreground input device**

[Input type] Setting input
[Data type] Byte
[Valid data range] 0 to 9

The CNC has the following interfaces for transferring data to and from an external input/output device and the host computer:

- Input/output device interface (RS-232-C serial ports 1 and 2)
- Memory card interface

- Data server interface
- Embedded Ethernet interface

By setting bit 0 (IO4) of parameter No. 0110, data input/output can be controlled separately. When IO4 is not set, data input/output is performed using the channel set in parameter No. 0020. When IO4 is set, a channel can be assigned to each of foreground input, foreground output, background input, and background output.
In these parameters, specify the interface connected to each input/output device to and from which data is to be transferred. See the table below for these settings.
To execute the DNC operation or M198 command with FOCAS2/Ethernet, set this parameter to 6.

Correspondence between settings and input/output devices	
Setting	**Description**
0,1	RS-232-C serial port 1
2	RS-232-C serial port 2
4	Memory card interface
5	Data server interface
6	Execution of the DNC operation or M198 command with FOCAS2/Ethernet
9	Embedded Ethernet interface

	#7	#6	#5	#4	#3	#2	#1	#0
0100					NCR		CTV	

[Input type] Setting input
[Data type] Bit

#1 CTV Character counting for TV check in the comment section of a program.
0: Performed
1: Not performed

#3 NCR Output of the end of block (EOB) in ISO code
0: LF, CR, CR are output.
1: Only LF is output.

	#7	#6	#5	#4	#3	#2	#1	#0
0138	MNC							

[Input type] Parameter input
[Data type] Bit

#7 MNC DNC operation from the memory card and external device subprogram call from the memory card are:
0: Not performed.
1: Performed.

0980	Machine group number to which each path belongs

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1
Set the machine group number to which each path belongs.

For the 0*i*-D/0*i* Mate-D, be sure to set this parameter to 1.

> **NOTE**
> If this parameter is set to 0, a setting of 1 is assumed.

0981	Absolute path number to which each axis belongs

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 1, 2

Set the path to which each axis belongs.

> **NOTE**
> 1 When 0 is set for all axes, the parameter is automatically set according to the number of controlled axes of each path.
> 2 When the setting falls outside the range, the axis is assumed to belong to the first path.

0982	Absolute path number to which each spindle belongs

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 1, 2

Set the path to which each spindle belongs.

> **NOTE**
> 1 When 0 is set for all axes, the parameter is automatically set according to the number of controlled axes of each path.
> 2 When the setting falls outside the range, the axis is assumed to belong to the first path.
> 3 When spindle control with servo motor is enabled, the servo motor used as the spindle controlled axis is treated as a spindle. Therefore, it is necessary to set the path to which the axis subject to spindle control with servo motor.

0983	Path control type of each path

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 For the 0*i* -D/0*i* Mate-D, this parameter does not need to be set because it is set automatically.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1

Set the path control type of each path.
The following two path control types are available:
T series (lathe system) : 0
M series (machining system): 1

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 INM Least command increment on the linear axis
0: In mm (metric system machine)
1: In inches (inch system machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1002	IDG			XIK	AZR			JAX

[Input type] Parameter input
[Data type] Bit path

#0 JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return:
0: 1 axis
1: 3 axes

#3 AZR When no reference position is set, the G28 command causes:
0: Reference position return using deceleration dogs (as during manual reference position return) to be executed.
1: Alarm (PS0304) “G28 was specified when no reference position is set” to be displayed.

> **NOTE**
> When reference position return without dogs is specified, (when bit 1 (DLZ) of parameter No.1005 is set to 1) the G28 command specified before a reference position is set causes an alarm PS0304 to be issued, regardless of the setting of AZR.

#4 XIK When LRP, bit 1 of parameter No.1401, is set to 0, namely, when positioning is performed using non-linear type positioning, if an interlock is applied to the machine along one of axes in positioning,
0: The machine stops moving along the axis for which the interlock is applied and continues to move along the other axes.
1: The machine stops moving along all the axes.

#7 IDG When the reference position is set without dogs, automatic setting of the IDGx parameter (bit 0 of parameter No.1012) to prevent the reference position from being set again is:
0: Not performed.
1: Performed.

		#7	#6	#5	#4	#3	#2	#1	#0
1004		IPR							

[Input type] Parameter input
[Data type] Bit path

#7 IPR Whether the least input increment for each axis is set to a value 10 times as large as the least command increment is specified, in increment systems of IS-B or IS-C at setting mm.
0: The least input increment is not set to a value 10 times as large as the least command increment.
1: The least input increment is set to a value 10 times as large as the least command increment.
If IPR is set to 1, the least input increment is set as follows:

Input increment	Least input increment
IS-B	0.01 mm, 0.01 deg, or 0.0001 inch
IS-C	0.001 mm, 0.001 deg, or 0.00001 inch

NOTE
For IS-A, the least input increment cannot be set to a value 10 times as large as the least command increment.
The least input increment is not multiplied by 10 also when the calculator-type decimal point input (bit 0 (DPI) of parameter No. 3401) is used.

		#7	#6	#5	#4	#3	#2	#1	#0
1005				EDMx	EDPx				ZRNx

[Input type] Parameter input
[Data type] Bit axis

#0 ZRNx If a move command other than G28 is specified by automatic operation when no reference position return is performed yet after the power is turned on:
0: The alarm (PS0224) "PERFORM REFERENCE POSITION RETURN." is issued.
1: Operation is performed without issuing an alarm.

NOTE
1 The state in which a reference position has not been established refers to the following state:
- When an absolute position detector is not used and reference position return has not been performed even once after power-up
- When an absolute position detector is used and the association of the machine position with the position detected with the absolute position detector has not been completed (See the description of bit 4 (APZx) of parameter No. 1815.)

2 When the Cs axis coordinates are to be set up, set ZRN to 0.

#4 EDPx In cutting feed, an external deceleration signal in the + direction for each axis is:
0: Invalid
1: Valid

> **NOTE**
> Be sure to set "1" to this parameter if bit 5 (EDR) of parameter No.1405 is set to 0 when positioning linear interpolation type is used.

#5 EDMx In cutting feed, an external deceleration signal in the - direction for each axis is:
0: Invalid
1: Valid

> **NOTE**
> Be sure to set "1" to this parameter if bit 5 (EDR) of parameter No.1405 is set to 0 when positioning linear interpolation type is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx		DIAx		ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx Setting linear or rotation axis.
#1 ROSx Setting linear or rotation axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624)
0	1	Rotation axis (A type) (1) Inch/metric conversion is not done. Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by parameter No.1008#0(ROAx) and #2(RRLx). (2) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (3) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 (Refer to parameter No.3624) (4) Cannot be used with the rotation axis roll over function and the index table indexing function (M series)

ROSx	ROTx	Meaning
Except for the above.		Setting is invalid (unused)

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

> **NOTE**
> For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
> - Halve the command multiplication (the detection unit is not changed).
> - Halve the detection unit and double the flexible feed gear (DMR).
>
> For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROAx The roll-over function of a rotation axis is
0: Invalid
1: Valid

> **NOTE**
> ROAx specifies the function only for a rotation axis (for which bit 0 (ROTx) of parameter No.1006, is set to 1)

#1 RABx In the absolute commands, the axis rotates in the direction
0: In which the distance to the target is shorter.
1: Specified by the sign of command value.

> **NOTE**
> RABx is valid only when ROAx is 1.

#2 RRLx Relative coordinates are
0: Not rounded by the amount of the shift per one rotation
1: Rounded by the amount of the shift per one rotation

> **NOTE**
> 1 RRLx is valid only when ROAx is 1.
> 2 Assign the amount of the shift per one rotation in parameter No.1260.

	#7	#6	#5	#4	#3	#2	#1	#0
1013							ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ISAx
#1 ISCx Increment system of each axis

Increment system	#1 ISCx	#0 ISAx
IS-A	0	1
IS-B	0	0
IS-C	1	0

	#7	#6	#5	#4	#3	#2	#1	#0
1015	DWT							

[Input type] Parameter input
[Data type] Bit path

#7 DWT When time for dwell per second is specified by P, the increment system:
0: Depends on the increment system
1: Does not depend on the increment system (1 ms)

1020	**Program axis name for each axis**

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 65 to 67,85 to 90

An axis name (parameter No. 1020) can be arbitrarily selected from 'A', 'B', 'C', 'U', 'V', 'W', 'X', 'Y', and 'Z'. (When G code system A is used with the T series, however, 'U', 'V', and 'W' are not selectable.)

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

For the axes with axis names of 'X', 'Y', 'Z', and 'C' in G code system A of the T series, the 'U', 'V', 'W', and 'H' commands are the incremental commands of these axes.

> **NOTE**
> 1 When G code system A is used in the T series, U, V, or W cannot be used as an axis name.
> 2 The same axis name cannot be set for multiple axes.
> 3 When the second auxiliary function is provided (when bit 2 (BCD) of parameter No. 8132 is 1), if the address (parameter No. 3460) that specifies the second auxiliary function is used as an axis name, the second auxiliary function is disabled.
> 4 When address C or A is used during chamfering/corner rounding or direct drawing dimension programming (when bit 4 (CCR) of parameter No. 3405 is 1) in the T series, address C or A cannot be used as an axis name.
> 5 When the multiple repetitive turning canned cycle (T series) is used, only 'X', 'Y', and 'Z' can be used for the address of the target axis.

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, tool radius/tool nose radius compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotation axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1023	Number of the servo axis for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to Number of controlled axes

Set the servo axis for each control axis.
Usually set to same number as the control axis number.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

- With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.
  Example)
  When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.
- For tandem controlled axes or electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
  Tandem axis:
  For a master axis, set an odd (1, 3, 5, 7, ...) servo axis number. For a slave axis to be paired, set a value obtained by adding 1 to the value set for the master axis.
  EGB axis:
  For a slave axis, set an odd (1, 3, 5, 7, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

1031	Reference axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

The unit of some parameters common to all axes such as those for dry run feedrate and one-digit F code feed may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.

Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL		ZPR

[Input type] Parameter input
[Data type] Bit path

#0 ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0: Not set automatically
1: Set automatically

> **NOTE**
> ZPR is valid when the workpiece coordinate system is not used (when bit 0 (NWZ) of parameter No. 8136 is 1). When the workpiece coordinate system is used, the workpiece coordinate system is established based on the workpiece origin offset (parameters No. 1220 to 1226) during a manual reference position return, regardless of the setting of this parameter.

#2 ZCL Local coordinate system when the manual reference position return is performed
0: The local coordinate system is not canceled.
1: The local coordinate system is canceled.

> **NOTE**
> ZCL is valid when the workpiece coordinate system is used (when bit 0 (NWZ) of parameter No. 8136 is 0). To use the local coordinate system (G52), set bit 0 (NWZ) of parameter No. 8136 to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
1202						G92		

[Input type] Parameter input
[Data type] Bit path

#2 G92 When the workpiece coordinate system is used (when bit 0 (NWZ) of parameter No. 8136 is 0), if the G code (M series: G92, T series: G50) for coordinate system setting is specified:
0: G command is executed and no alarm is issued.
1: G command is not executed and an alarm (PS0010) is issued.

1240	Coordinate value of the reference position in the machine coordinate system

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

1241	Coordinate value of the second reference position in the machine coordinate system

1242	Coordinate value of the third reference position in the machine coordinate system

1243	Coordinate value of the fourth reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the second to fourth reference positions in the machine coordinate system.

1250	Coordinate system of the reference position used when automatic coordinate system setting is performed

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate system of the reference position on each axis to be used for setting a coordinate system automatically.

1260	The shift amount per one rotation of a rotation axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the shift amount per one rotation of a rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA						NAL	OUT

[Input type] Setting input
[Data type] Bit path

#0 OUT The area inside or outside of the stored stroke check 2 is set as an inhibition area
0: Inside
1: Outside

#1 NAL If the tool enters the inhibition area of stored stroke limit 1 during manual operation:
0: An alarm is issued and the tool is stopped.
1: An alarm is not issued, the stroke limit reach signal is output to the PMC, and the tool is stopped.

> **NOTE**
> When the tool enters the inhibition area of stored stroke limit 1 due to the move command issued during automatic operation, even if this parameter is set to 1, an alarm is issued and the tool is stopped. Even in this case, the stroke limit reach signal is output to the PMC.

#7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (T series), or a chuck/tail stock barrier (T series) alarm is issued:
0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

	#7	#6	#5	#4	#3	#2	#1	#0
1301		OTS				NPC		

[Input type] Setting input
[Data type] Bit path

#2 NPC As part of the stroke limit check performed before movement, the movement specified in G31 (skip) and G37 (automatic tool length measurement (M series) or automatic tool compensation (T series)) blocks is:
0: Checked
1: Not checked

#6 OTS When the overtravel alarm is issued:
0: The overtravel alarm signal is not output to the PMC.
1: The overtravel alarm signal is output to the PMC.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis

1321	Coordinate value I of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area outside the area set by parameter No. 1320 and No. 1321 is a prohibited area.

1322	Coordinate value of stored stroke check 2 in the positive direction on each axis

1323	Coordinate value of stored stroke check 2 in the negative direction on each axis

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 2 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 Whether the inside area or outside area is a prohibited area is set using bit 0 (OUT) of parameter No. 1300.

1324	Coordinate value of stored stroke check 3 in the positive direction on each axis

1325	Coordinate value of stored stroke check 3 in the negative direction on each axis

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 3 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area inside the area set by parameter No. 1324 and No. 1325 is a prohibited area.

1326	Coordinate value II of stored stroke check 1 in the negative direction on each axis

1327	Coordinate value II of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.
When the stored stroke check switch signal EXLM is set to 1, or the stored stroke check switch signal for each axis direction +EXLx is set to 1, parameter No. 1326 and No. 1327 are used for stroke check instead of parameter No.1320 and No. 1321.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area outside the area set by parameter No. 1326 and No. 1327 is a prohibited area.
3 The EXLM signal is valid only when bit 2 (LMS) of parameter No. 1300 is set to 1.
4 The +EXLx signal is valid only when bit 0 (DLM) of parameter No. 1301 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0			LRP	RPD

[Input type] Parameter input
[Data type] Bit path

#0 RPD Manual rapid traverse during the period from power-on time to the completion of the reference position return.
0: Disabled (Jog feed is performed.)
1: Enabled

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.

#4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				NPC

[Input type] Parameter input
[Data type] Bit path

#0 NPC Feed per revolution without the position coder (function for converting feed per revolution F to feed per minute F in the feed per revolution mode (G95)) is:
0: Not used
1: Used

> **NOTE**
> 1 When using the position coder, set this parameter to 0.
> 2 While this parameter is set to 1, threading cannot be performed even if a position coder is provided.

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

> **NOTE**
> Specify a feedrate in parameter No.1423.

	#7	#6	#5	#4	#3	#2	#1	#0
1403			HTG					

[Input type] Parameter input
[Data type] Bit path

#5 HTG The feedrate for helical interpolation is:
0: Specified using the feedrate along the tangent to an arc
1: Specified using the feedrate along axes including a linear axis

	#7	#6	#5	#4	#3	#2	#1	#0
1404						FM3		

[Input type] Parameter input
[Data type] Bit path

#2 FM3 The increment system of an F command without a decimal point in feed per minute is:
0: 1 mm/min (0.01 inch/min for inch input)
1: 0.001 mm/min (0.00001 inch/min for inch input)

	#7	#6	#5	#4	#3	#2	#1	#0
1405			EDR				FR3	

[Input type] Parameter input
[Data type] Bit path

#1 FR3 The increment system of an F command without a decimal point in feed per revolution is:
0: 0.01 mm/rev (0.0001 inch/rev for inch input)
1: 0.001 mm/rev (0.00001 inch/rev for inch input)

#5 EDR As the external deceleration rate for positioning of linear interpolation type:
0: The external deceleration rate for cutting feed is used.
1: The external deceleration rate in rapid traverse for the first axis of path 1 is used.
Let us use external deceleration 1 as an example.
When this parameter bit is set to 0, the value of parameter No. 1426 is used as the external deceleration rate for external deceleration 1.
When this parameter bit is set to 1, the value of axis 1 of parameter No. 1427 is used as the external deceleration rate for external deceleration 1.

> **NOTE**
> Be sure to set "1" to bit 4 (EDPx) of parameter No. 1005 and bit 5 (EDMx) of parameter No. 1005 if this parameter is set to 0 when positioning linear interpolation type is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1408								RFDx

[Input type] Parameter input
[Data type] Bit axis

#0 RFDx Feedrate control on a rotation axis is exercised using:
0: Conventional method
1: Method that specifies a feedrate on the virtual circle of the rotation axis

1410	Dry run rate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the dry run rate at the 100% position on the jog feedrate specification dial. The unit of data depends on the increment system of the reference axis.

NOTE

When the operation is begun, alarm PS5009 is issued if the setting of this parameter is set to "0.0".
Even if the operation which is not dry run is performed, this alarm is issued.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1421	F0 rate of rapid traverse override for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the F0 rate of the rapid traverse override for each axis.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

(1) When JRV, bit 4 of parameter No.1402, is set to 0 (feed per minute), specify a jog feedrate (feed per minute) under an override of 100%.
(2) When JRV, bit 4 of parameter No.1402, is set to 1 (feed per revolution), specify a jog feedrate (feed per revolution) under an override of 100%.

NOTE

This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rate of manual rapid traverse when the manual rapid traverse override is 100% for each axis.

> **NOTE**
> 1 If 0 is set, the rate set in parameter 1420 (rapid traverse rate for each axis) is assumed.
> 2 When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

1425	FL rate of the reference position return for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

1427	External deceleration rate of rapid traverse for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the external deceleration rate of rapid traverse for each axis.

1428	Reference position return feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a rapid traverse rate for reference position return operation using deceleration dogs, or for reference position return operation before a reference position is set.
This parameter is also used to set a feedrate for the rapid traverse command (G00) in automatic operation before a reference position is set.

NOTE

1. To this feedrate setting (100%), a rapid traverse override (F0, 25, 50, or 100%) is applicable.
2. For automatic return after completion of reference position return and machine coordinate system establishment, the normal rapid traverse rate is used.
3. As a manual rapid traverse rate before machine coordinate system establishment by reference position return, the jog feedrate or manual rapid traverse rate can be selected with bit 0 (RPD) of parameter No. 1401.

	Before coordinate system establishment	After coordinate system establishment
Automatic reference position return (G28)	No.1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return *1	No.1428	No.1428 *3
Manual rapid traverse	No.1423 *2	No.1424

4. When parameter No. 1428 is set to 0, the following parameter-set feedrates are applied.

	Before coordinate system establishment	After coordinate system establishment
Automatic reference position return (G28)	No.1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return *1	No.1424	No.1424 *3
Manual rapid traverse	No.1423 *2	No.1424

1420: rapid traverse rate
1423: Jog feedrate
1424: Manual rapid traverse rate
*1 : By using bit 2 (JZR) of parameter No. 1401, the jog feedrate can be used for manual reference position return at all times.
*2 : When bit 0 (RPD) of parameter No. 1401 is set to 1, the setting of parameter No. 1424 is used.
*3 : When rapid traverse is used for reference position return without dogs or manual reference position return after reference position establishment, regardless of the deceleration dog, the feedrate for manual reference position return based on these functions is used (the setting of bit 1 (DLF) of parameter No. 1404 is followed).

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

1432	Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set a maximum cutting feedrate for each axis in the acceleration/deceleration before interpolation mode such as advanced preview control, AI advanced preview control, or AI contour control. When the acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.
Moreover, this parameter is valid in optimum acceleration/
deceleration for rigid tapping. Be sure to set this parameter for tapping axis.

1434	Maximum manual handle feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set a maximum manual handle feedrate for each axis in case of maximum manual handle feedrate switch signal HNDLF<Gn023.3>=1.

1441	External deceleration rate setting 2 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set external deceleration rate 2 for each axis in rapid traverse.

1444	External deceleration rate setting 3 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set external deceleration rate 3 for each axis in rapid traverse.

1450	Change of feedrate for one graduation on the manual pulse generator during one-digit F feed code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 127

Set the constant that determines the change in feedrate as the manual pulse generator is rotated one graduation during one-digit F feed code.

$$\Delta F = \frac{F\max i}{100n} \quad \text{(where, i=1 or 2)}$$

In the above equation, set n. That is, the number of revolutions
of the manual pulse generator, required to reach feedrate Fmaxi is obtained. Fmaxi refers to the upper limit of the feedrate for a one-digit F code feed command, and set it in parameters No. 1460 or No. 1461.
Fmax1: Upper limit of the feedrate for F1 to F4 (parameter No. 1460)
Fmax2: Upper limit of the feedrate for F5 to F9 (parameter No. 1461)

1451	Feedrate for F1
to	to
1459	Feedrate for F9

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
These parameters set the feedrates for one-digit F code feed commands F1 to F9. When a one-digit F code feed command is specified, and the feedrate is changed by turning the manual pulse generator, the parameter-set value also changes accordingly.

1460	Upper limit of feedrate for F1 to F4

1461	Upper limit of feedrate for F5 to F9

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the upper limit of feedrate for the one-digit F code feed command.
As the feedrate increases by turning the manual pulse generator, the feedrate is clamped when it reaches the upper limit set. If a one-digit F feed command F1 to F4 is executed, the upper limit is that set in parameter No. 1460. If a one-digit F code feed command F5 to F9 is executed, the upper limit is that set in parameter No. 1461.

1465	Radius of a virtual circle when a feedrate is specified on the virtual circle of a rotation axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (B)
Set the radius of a virtual circle when a feedrate on the virtual circle of a rotation axis is specified.
If 0 is set for a rotation axis, the axis is excluded from feedrate calculation.
If the input unit is the inch, enter a value in inches.

The data is then converted to a millimeter value and displayed.

> **NOTE**
> 1 This parameter is valid when bit 0 (ROTx) of parameter No. 1006 and bit 0 (RFDx) of parameter No. 1408 are 1.
> 2 Be careful to set bit 0 (RFDx) of parameter No. 1408 and parameter No. 1465 for the virtual radius. If the virtual radius is set to a small value and a feedrate on the virtual circle of the rotation axis is specified, the movement of the axis becomes faster.

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Input type] Parameter input
[Data type] Bit path

#5 NCI An in-position check:
0: Confirms that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time and that the machine position has reached a specified position (the servo positional deviation is within the in-position width set by parameter No. 1826).
1: Confirms only that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time.

	#7	#6	#5	#4	#3	#2	#1	#0
1606								MNJx

[Input type] Parameter input
[Data type] Bit axis

#0 MNJx In manual handle interrupt :
0: Only cutting feed acceleration/deceleration is enabled, and jog feed acceleration/deceleration is disabled.
1: Both cutting feed acceleration/deceleration and jog feed acceleration/deceleration are applied.

	#7	#6	#5	#4	#3	#2	#1	#0
1610							CTBx	CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

NOTE
When using bell-shaped acceleration/deceleration after interpolation, set this parameter to 0 and set bit 1 (CTBx) of parameter No. 1610 to select bell-shaped acceleration/deceleration after interpolation.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration after interpolation
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell-shaped acceleration/deceleration after interpolation

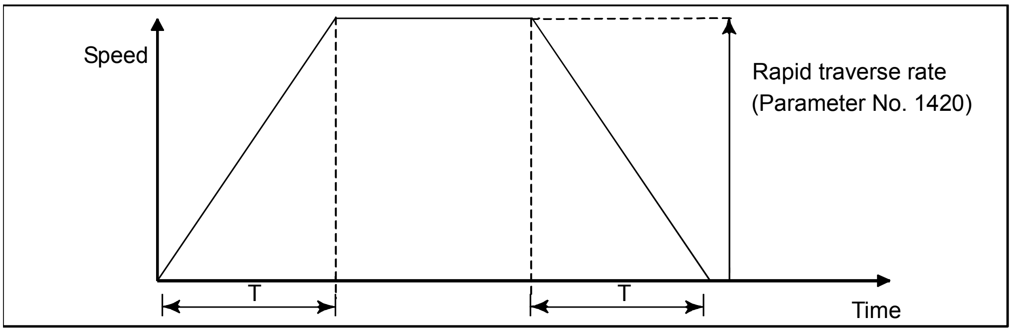
#1 CTBx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (depending on the setting in CTLx, bit 0 of parameter No.1610)
1: Bell-shaped acceleration/deceleration is applied.

NOTE
This parameter is valid only when the bell-shaped acceleration/deceleration after cutting feed interpolation function is used. When this function is not used, the acceleration/deceleration is determined according to bit 0 (CTLx) of parameter No. 1610 regardless of the setting of this parameter.

1620	Time constant T or T_1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

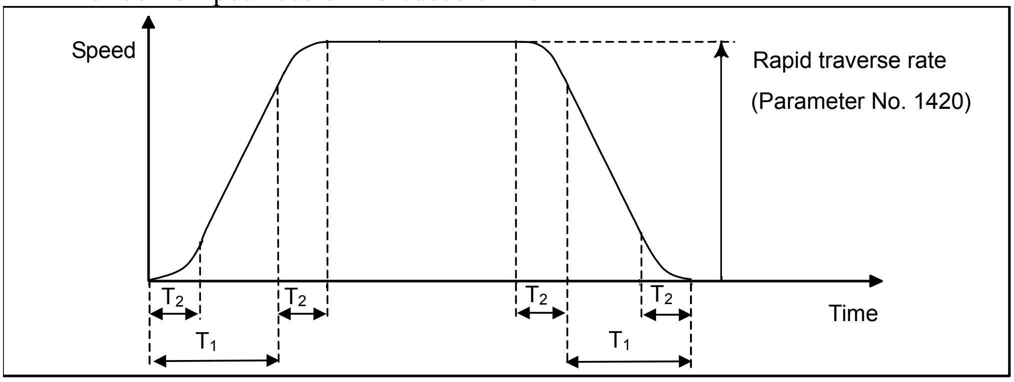
[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.
[Example]
For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621
(However, $T_1 \geq T_2$ must be satisfied.)
Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1622	Time constant of acceleration/deceleration in cutting feed for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in cutting feed for each axis. Which type to use is selected with bits 1(CTBx) and 0(CTLx) of parameter No.1610. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1624	Time constant of acceleration/deceleration in jog feed for each axis.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for acceleration/deceleration in jog feed for each axis.

1660	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)

(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for each axis.
If a value greater than 100000.0 is set, the value is clamped to 100000.0.

If 0 is set, the specification of 100000.0 is assumed. If 0 is set for all axes, however, acceleration/deceleration before interpolation is not performed.

If a maximum allowable acceleration rate set for one axis is greater than a maximum allowable acceleration rate set for another axis by a factor or 2 or more, the feedrate at a corner where the direction of travel abruptly changes can decrease temporarily.

1671	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for linear rapid traverse.
If a value greater than 100000.0, the value is clamped to 100000.0.
If 0 is set, the specification of the following is assumed:

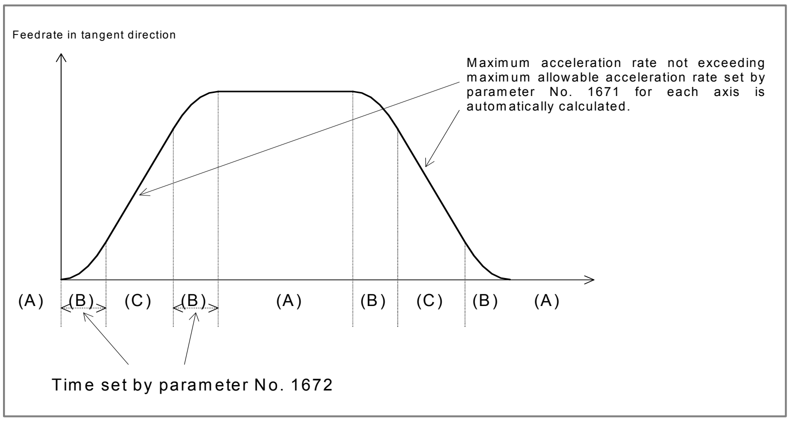
1000.0 mm/sec^2
100.0 inch/sec^2
100.0 degrees/sec^2

If 0 is specified for all axes, however, acceleration/deceleration before interpolation is not performed.

1672	Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

Set an acceleration change time of bell-shaped acceleration/ deceleration for linear rapid traverse (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1671: time of (B) in the figure below).



1710	Minimum deceleration ratio (MDR) for inner circular cutting feedrate change by automatic corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100

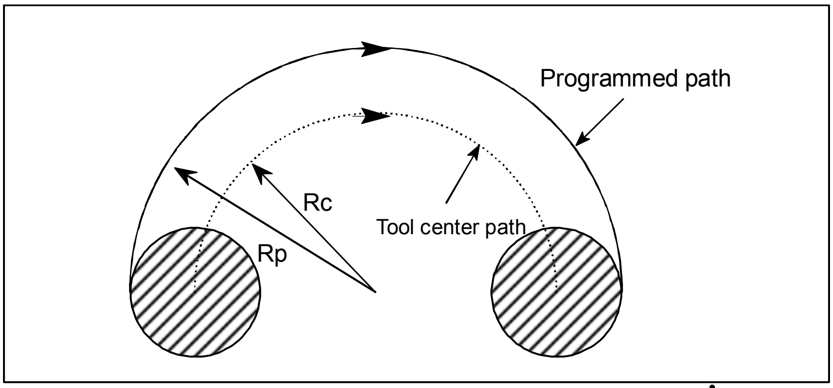
Set a minimum deceleration ratio (MDR) for an inner circular cutting feedrate change by automatic corner override.

In the case of circular cutting offset inward, the actual feedrate is determined by a specified feedrate (F) as follows:

$$F \times \frac{Rc}{Rp}$$

(Rc:Radius of tool center path
Rp:Programmed radius)

Thus, the feedrate along the programmed path satisfies the specified value of F.



However, if Rc is too small when compared with Rp, Rc/Rp ≒ 0 results to stop the tool. So, a minimum deceleration ratio (MDR) is set, and the feedrate of the tool is set to F×(MDR) when Rc/Rp ≤ MDR.

> **NOTE**
> When this parameter is set to 0, the minimum deceleration ratio (MDR) is 100%.

1711	Inner determination angle (θp) for inner corner override

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 2 to 178

Set an inner determination angle for inner corner override in automatic corner overriding.

1712	Override value for inner corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 1 to 100

Set an inner corner override value in automatic corner overriding.

1713	Start distance (Le) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set a start distance for inner corner override in automatic corner overriding.

1714	End distance (Ls) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set an end distance for inner corner override in automatic corner overriding.

When $\theta \leq \theta p$, an inner corner is assumed. (Parameter No. 1711 is used to set θp.)
When a corner is determined to be an inner corner, an override is applied to the feedrate in the range of Le in the previous block from the intersection of the corner and in the range of Ls in the next block from the intersection of the corner.
Distances Le and Ls represent linear distances from the intersection of a corner to points on the tool center path.
Le and Ls are set in parameter No. 1713 and No. 1714.

1732	Minimum allowable feedrate for the deceleration function based on acceleration in circular interpolation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
With the deceleration function based on acceleration in circular interpolation, an optimum feedrate is automatically calculated so that acceleration produced by changing the move direction in circular interpolation does not exceed the maximum allowable acceleration rate specified in parameter No. 1735.
If the radius of an arc is very small, a calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

1735	Maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation.
Feedrate is controlled so that acceleration produced by changing the move direction in circular interpolation does not exceed the value specified in this parameter.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled.
If a different value is set in this parameter for each axis, a feedrate is determined from the smaller of the acceleration rates specified for the two circular axes.

1737	Maximum allowable acceleration rate for the deceleration function based on acceleration in AI contour control for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate produced by changing the tool move direction.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled. If 0 is set for all axes, the deceleration function based on acceleration is not performed.
In circular interpolation, however, the deceleration function based on feedrate control using acceleration in circular interpolation (parameter No. 1735) is enabled.

1738	Minimum allowable feedrate for the deceleration function based on acceleration in AI contour control

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
With the deceleration function based on acceleration in AI advanced preview control or AI contour control, a feedrate most suitable for a desired figure is automatically calculated.
Depending on the figure, however, the calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

1769	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode

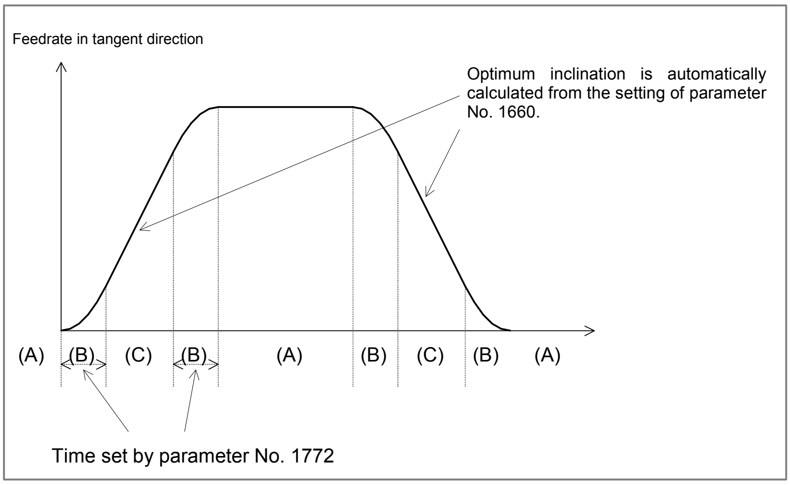
[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

In the acceleration/deceleration before interpolation mode as in advanced preview control, AI advanced preview control, or AI contour control, not the ordinary time constant (parameter No. 1622) but the value of this parameter is used.
Be sure to specify the same time constant value for all axes except for a special application. If different values are set, correct linear and circular figures cannot be obtained.

1772	Acceleration change time of bell-shaped acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

Set an acceleration change time of bell-shaped acceleration/ deceleration before interpolation (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1660: time of (B) in the figure below).



NOTE
The option of bell-shaped acceleration/deceleration before look-ahead interpolation is required. This parameter is valid only in the AI contour control mode.

1783	Maximum allowable feedrate difference for feedrate determination based on corner feedrate difference

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

If a feedrate component change for each axis exceeding the value set in this parameter occurs at the joint of blocks, the feedrate determination function based on corner feedrate difference finds a feedrate not exceeding the set value and performs deceleration by using acceleration/deceleration before interpolation. Thus, a shock to the machine and machining error at a corner can be reduced.

	#7	#6	#5	#4	#3	#2	#1	#0
1802						DC2x	DC4x	

[Input type] Parameter input
[Data type] Bit axis

#1 DC4x When the reference position is established on the linear scale with reference marks:
0: An absolute position is established by detecting three reference marks.
1: An absolute position is established by detecting four reference marks.

#2 DC2x Reference position establishment operation for a linear scale with reference marks is performed as follows:
0: The setting of bit 1 (DC4) of parameter No. 1802 is followed.
1: An absolute position is established by detecting two reference marks.

NOTE
1 When this parameter is set to 1, specify the direction of the scale zero point by setting bit 4 (SCP) of parameter No. 1817.
2 When a rotary encoder with absolute address reference marks is used, this parameter is invalid. Even when this parameter is set to 1, the setting of bit 1 (DC4) of parameter No. 1802 is followed.

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx	DCRx		OPTx	

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx Position detector
0: A separate pulse coder is not used.
1: A separate pulse coder is used.

NOTE
Set this parameter to 1 when using a linear scale with reference marks or a linear scale with an absolute address zero point (full-closed system).

#3 DCRx As a scale with absolute address reference marks:
0: A rotary encoder with absolute address reference marks is not used.
1: A rotary encoder with absolute address reference marks is used.

NOTE
When using a rotary encoder with absolute address reference marks, set also bit 2 (DCLx) of parameter No. 1815 to 1.

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used
0: Not corresponding
1: Corresponding
When an absolute position detector is used, after primary adjustment is performed or after the absolute position detector is replaced, this parameter must be set to 0, power must be turned off and on, then manual reference position return must be performed. This completes the positional correspondence between the machine position and the position on the absolute position detector, and sets this parameter to 1 automatically.

#5 APCx Position detector
0: Other than absolute position detector
1: Absolute position detector (absolute pulse coder)

	#7	#6	#5	#4	#3	#2	#1	#0
1817		TANx						

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#6 TANx Tandem control
0: Not used
1: Used

> **NOTE**
> Set this parameter to both master axis and slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1818					SDCx		RF2x	RFSx

[Input type] Parameter input
[Data type] Bit axis

#0 RFSx If G28 is specified for an axis for which a reference position is not established (ZRF = 0) when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:
0: A movement is made to the reference position after reference position establishment operation.
1: No movement is made after reference position establishment operation, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

#1 RF2x If G28 is specified for an axis for which a reference position is already established (ZRF = 1) when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:
0: A movement is made to the reference position.
1: No movement is made to the intermediate position and reference position, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

#3 SDCx A linear scale with an absolute address zero point is:
0: Not used.
1: Used.

> **NOTE**
> 1 After setting parameter SDCx, be sure to turn the power off and back on again. Note that the power-off alarm (PW0000) is not issued.
> 2 For the full-closed system, set bit 1 (OPTx) of parameter No. 1815 to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1819						DATx		

[Input type] Parameter input
[Data type] Bit axis

#2 DATx When a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used, the automatic setting of parameter No. 1883 and No. 1884 at manual reference position return time is:
0: Not performed.
1: Performed.

The automatic setting procedure is as follows:
<1> Set an appropriate value in parameter No. 1815, No. 1821, and No. 1882.
<2> Position the machine at the reference position by manual operation.
<3> Set this parameter to 1.
<4> Perform a manual reference position return operation. Upon completion of manual reference position return operation, parameter No. 1883 and No. 1884 are set, and this parameter is automatically set to 0.

1820	Command multiplier for each axis (CMR)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] See below :
Set a command multiplier indicating the ratio of the least command increment to the detection unit for each axis.

Least command increment = detection unit × command multiplier

Relationship between the increment system and the least command increment
(1) T series

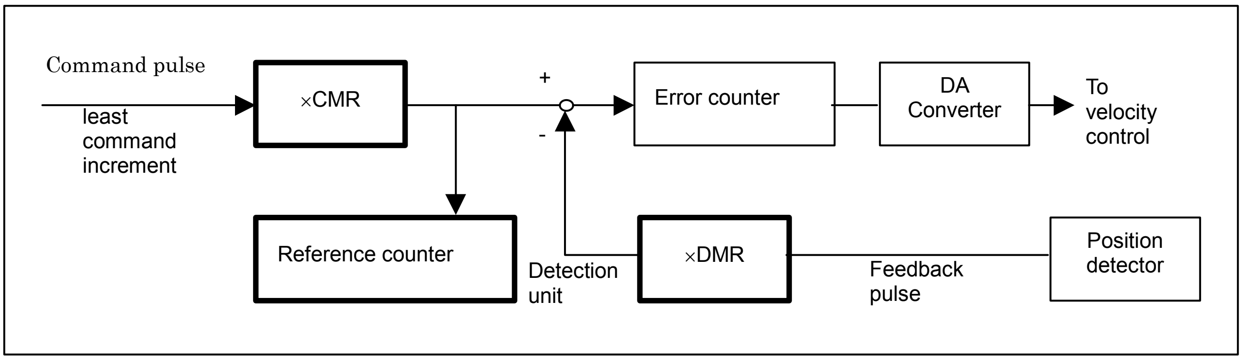
			Least input increment	Least command increment
IS-B	Millimeter machine	Millimeter input	0.001 mm (diameter specification)	0.0005 mm
			0.001 mm (radius specification)	0.001 mm
		Inch input	0.0001 inch (diameter specification)	0.0005 mm
			0.0001 inch (radius specification)	0.001 mm
	Inch machine	Millimeter input	0.001 mm (diameter specification)	0.00005 inch
			0.001 mm (radius specification)	0.0001 inch
		Inch input	0.0001 inch (diameter specification)	0.00005 inch
			0.0001 inch (radius specification)	0.0001 inch
	Rotation axis		0.001 deg	0.001 deg

			Least input increment	Least command increment
IS-C	Millimeter machine	Millimeter input	0.0001 mm (diameter specification)	0.00005 mm
			0.0001 mm (radius specification)	0.0001 mm
		Inch input	0.00001 inch (diameter specification)	0.00005 mm
			0.00001 inch (radius specification)	0.0001 mm
	Inch machine	Millimeter input	0.0001 mm (diameter specification)	0.000005 inch
			0.0001 mm (radius specification)	0.00001 inch
		Inch input	0.00001 inch (diameter specification)	0.000005 inch
			0.00001 inch (radius specification)	0.00001 inch
	Rotation axis		0.0001 deg	0.0001 deg

(2) M series

Increment system	Least input increment and least command increment			
	IS-A	IS-B	IS-C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Millimeter input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

Setting command multiply (CMR), detection multiply (DMR), and the capacity of the reference counter



Set CMR and DMR so that the pulse weight of + input (command from the CNC) into the error counter matches the pulse weight of -input (feedback from the position detector).
[Least command increment]/CMR=[Detection unit]=
[Feedback pulse unit]/DMR

[Least command increment]:
Minimum unit of commands issued from the CNC to the machine

[Detection unit]: Minimum unit for machine position detection

The unit of feedback pulses varies, depending on the type of detector.
[Feedback pulse unit]=[Amount of travel per rotation of the pulse coder]/[Number of pulses per rotation of the pulse coder]

As the size of the reference counter, specify the grid interval for the reference position return in the grid method.
[Size of the reference counter]=[Grid interval]/[Detection unit]
[Grid interval]=[Amount of travel per rotation of the pulse coder]

The setting of a command multiplier is as follows:
(1) When command multiplier is 1 to 1/27
Set value = 1 / command multiplier + 100
Valid data range : 101 to 127
(2) When command multiply is 0.5 to 48
Set value = 2 × command multiplier
Valid data range : 1 to 96

NOTE
1 If a feedrate exceeding the feedrate found by the expression below is used, an incorrect travel amount may result or a servo alarm may be issued. Be sure to use a feedrate not exceeding the feedrate found by the following expression:
Fmax[mm/min] = 196602 $\times 10^4 \times$ least command increment / CMR
2 For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

1821	Reference counter size for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999
Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.

When a linear scale with absolute address reference marks is used, set the interval of mark 1.

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.
If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm (SV0411) is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in the stopped state for each axis.
If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm (SV0410) is generated, and operation is stopped immediately (as in emergency stop).

1851	Backlash compensating value for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value for each axis.
When the machine moves in a direction opposite to the reference position return direction after the power is turned on, the first backlash compensation is performed.

1882	Interval of mark 2 of a linear scale with absolute address reference marks

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set the interval of mark 2 of a linear scale with absolute address reference marks.

1883	Distance 1 from the scale zero point to reference position (linear scale with absolute address reference marks) or distance 1 from the base point to reference position (linear scale with an absolute address zero point)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

1884	Distance 2 from the scale zero point to reference position (linear scale with absolute address reference marks) or distance 2 from the base point to reference position (linear scale with an absolute address zero point)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

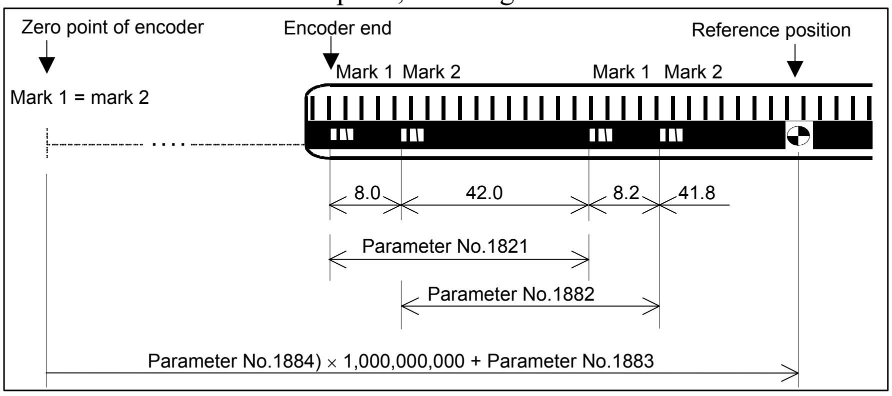
[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999

When a linear scale with absolute address reference marks is used, set the distance from the scale zero point to reference position in parameter Nos. 1883 and 1884).

Distance from the zero point to the reference position of a linear scale
= No. 1884 × 1,000,000,000 + No. 1883

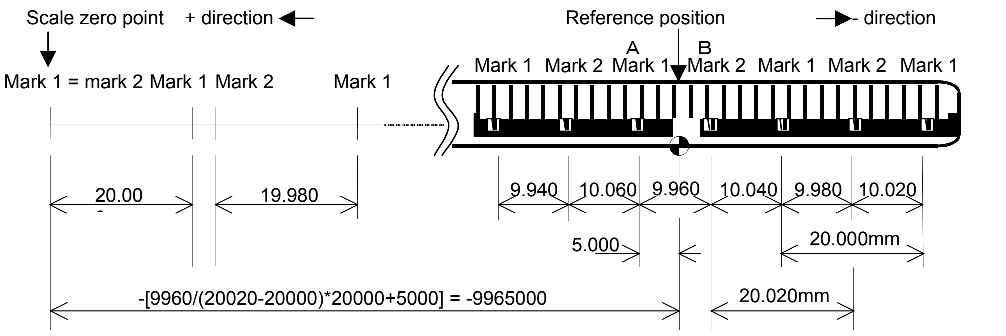
The scale zero point represents a point where mark 1 and mark 2 match. Usually, this point is a virtual point that does not physically exist on the scale. (See the figure below.)

If the reference position is placed in the + direction when viewed from the scale zero point, set a positive value. If the reference position is placed in the - direction when viewed from the scale zero point, set a negative value.



[Example of parameter settings]

When an encoder as shown below is used with an IS-B, millimeter machine:



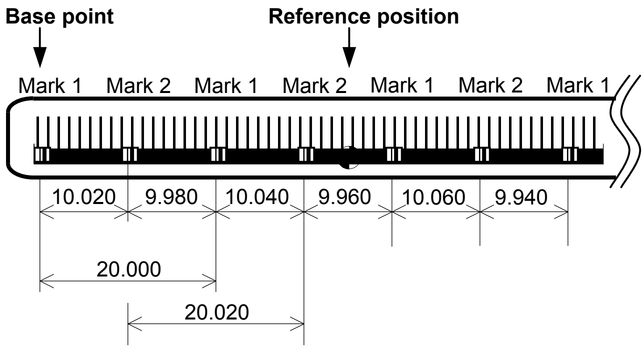
Parameters No.1821 (interval of mark 1) = 20000
No.1882 (interval of mark 2) = 20020
No.1883 (reference position) = position of point A + 5.000
= distance between A and B/(mark 2 - mark 1) × mark 1+ 5000
= 9960/ (20020 - 20000) × 20000＋5000
= 9965000
= -9965000 (the reference position is on the negative side)

[Setting parameter No. 1883]

When it is difficult to measure the distance from the scale zero point to the reference position (parameter No. 1883), the method described below can be used to find the distance.

<1> Set parameter No. 1815 to enable this function.
Set an appropriate value in parameter No. 1821 and No. 1882.
Set 0 in parameter No. 1240.
Set 0 in parameter No. 1883 and No. 1884.
<2> At an appropriate position, establish a reference position.
(As a result, the machine coordinate represents the distance from the scale zero point to the current position.)
<3> By jog feed or handle feed, place the machine at the accurate reference position.
<4> In parameter No. 1883, set the machine coordinate of that time converted to the detection unit (machine coordinate × CMR).
<5> If necessary, set parameter No. 1240.

When a linear scale with an absolute address zero point is used, set the distance from the base point to the reference position in parameter Nos. 1883 and 1884. The base point is a point at a scale end as shown below.



If the reference position is located in the positive direction when viewed from the base point, set a positive value; if the reference position is located in the negative direction, set a negative value. Set the value by following the steps explained below.

<1> Set bit 1 (OPT) of parameter No. 1815 , bit 2 (DCL) of parameter No. 1815, and bit 3 (SDC) of parameter No. 1818 to enable this function.
Set 0 in parameter No. 1240.
Set 0 in parameter No. 1883 and No. 1884.
<2> At an appropriate position, establish a reference position.
(Consequently, the machine coordinate value indicates the distance from the base point to current position.)
<3> By jog feed or handle feed, place the machine at the accurate reference position.
<4> In parameters Nos. 1883 and 1884, set the machine coordinate of that time converted to the detection unit (machine coordinate × CMR).
If necessary, set parameter No. 1240.

NOTE
1 Set parameter Nos. 1883 and 1884 so that the distance from the scale zero point (for a linear scale with absolute address reference marks) or the base point (for a linear scale with an absolute address zero point) to the reference position is within the range from -999,999,999,999 to +999,999,999,999. If a value beyond this range is set, an alarm (PS 5325) is issued.
2 The scale area on the scale cannot be extended across the scale zero point or base point. Make parameter settings not to cause the scale area to extend beyond the scale zero point or base point.

	#7	#6	#5	#4	#3	#2	#1	#0
1902							ASE	FMD

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 FMD The FSSB setting mode is:
0: Automatic setting mode.
(When bit 0 (DFS) of parameter No. 14476 is 0:
If the relationship between the axis and the amplifier and the like are defined on the FSSB setting screen, parameters No. 1023, No. 1905, Nos. 1936 and 1937, Nos. 14340 to 14357, and Nos. 14376 to 14391 are automatically set.)
(When bit 0 (DFS) of parameter No. 14476 is 1:
If the relationship between the axis and the amplifier and the like are defined on the FSSB setting screen, parameters No. 1023, No. 1905, Nos. 1910 to 1919, and Nos. 1936 and 1937 are automatically set.)
1: Manual setting 2 mode.
(When bit 0 (DFS) of parameter No. 14476 is 0:
Manually set parameters No.1023, No.1905, Nos.1936 and 1937, Nos.14340 to 14357, and Nos.14376 to 14391.)
(When bit 0 (DFS) of parameter No. 14476 is 1:
Manually set parameters No.1023, No.1905, Nos.1910 to 1919, and Nos. 1936 and 1937.)

#1 ASE When automatic setting mode is selected for FSSB setting (when the FMD parameter (bit 0 of parameter No.1902) is set to 0), automatic setting is:
0: Not completed.
1: Completed.
This bit is automatically set to 1 upon the completion of automatic setting.

	#7	#6	#5	#4	#3	#2	#1	#0
1905	PM2x	PM1x						

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#6 PM1x The first separate detector interface unit is:
0: Not used.
1: Used.

#7 PM2x The second separate detector interface unit is:
0: Not used.
1: Used.

> **NOTE**
> When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), this parameter is automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), this parameter must be set directly. When a separate detector interface unit is used, a connector number must be set in the corresponding parameter (No.1936 or No.1937).

1936	Connector number of the first separate detector interface unit

1937	Connector number of the second separate detector interface unit

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

This parameter sets the connector number corresponding to the connector connected when using the separator detector interface unit set by bits 6 and 7 of parameter No. 1905 minus 1. That is, set 0 to 7 for connector numbers 1 to 8, respectively. Set 0 for the axis for which the separator detector interface unit is not used.
Use successive numbers for one separator detector interface unit. Do not omit a intermediate number.

Example)

Controlled axis	Connector number for the first separate detector interface unit	Connector number for the second separate detector interface unit	No.1936	No.1937	PM2x, PM1x (No.1905#7, #6)
X	1	Not used	0	0	0, 1
Y	Not used	2	0	1	1, 0
Z	Not used	1	0	0	1, 0
A	Not used	Not used	0	0	0, 0

NOTE
When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), these parameters are automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), these parameters must be set directly.

Parameters No.2000 to 2999 are for digital servo. Refer to FANUC AC SERVO MOTOR αi series PARAMETER MANUAL (B-65270EN)

	#7	#6	#5	#4	#3	#2	#1	#0
2011	XIAx							

[Input type] Parameter input
[Data type] Bit axis

#7 XIAx Temporary absolute coordinate setting is:
0: Not used.
1: Used.

NOTE
1 When temporary absolute coordinate setting is used, bit 1 (OPTx) of parameter No. 1815, bit 5 (APCx) of parameter No. 1815, parameter No. 1874, and parameter No. 1875 must be set.
2 The setting of this parameter becomes effective after the power is turned off then back on.

2031	Torque command difference threshold of torque difference alarm

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 14564

If the absolute value of the torque command difference between two axes exceeds the value set in this parameter, an alarm is issued.
Set the same value for two axes that are placed under axis synchronous control.
The servo axis numbers of the synchronized master axis and slave axis must be assigned so that an odd number is assigned to the master axis and the next axis number is assigned to the slave axis. Examples are (1,2) and (3,4).

	#7	#6	#5	#4	#3	#2	#1	#0
3003	MVG							

[Input type] Parameter input
[Data type] Bit path

#7 MVG During drawing with the dynamic graphic display function, the axis movement signal is:
0: Output.
1: Not output.

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

NOTE
When this parameter is set to 1, set parameter No. 3013, No. 3014, No. 3012, and No. 3019. If parameter No. 3013 and No. 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of X0000. If parameter No. 3012 and No. 3019 are not set, the skip signal, the PMC axis control skip signal, the measurement position arrival signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to X0000.

3012	Skip signal assignment address

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 327
Set an X address to which the skip signal (SKIPn) is to be assigned.

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
> The X addresses that can be actually used are shown below, but they depend on the configuration of I/O Link point count expansion options.
> X0 to X127, X200 to X327

3013	X address to which the deceleration signal for reference position return is assigned

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 327

Set an address to which the deceleration signal (*DECn) for reference position return for each axis is to be assigned.

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
> The X addresses that can be actually used are shown below, but they depend on the configuration of I/O Link point count expansion options.
> X0 to X127, X200 to X327

3019	Address to which the PMC axis control skip signal and the measurement position arrival signal are assigned

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 327

Sets addresses to which X address PMC axis control skip signal ESKIP, measurement position arrival signals (XAE1, XAE2, and XAE3 (M series); XAE1 and XAE2 (T series)), and tool compensation write signals (±MIT1 and ±MIT2 (T series)) are allocated.

Example 1. When No.3012 is set to 5 and No.3019 is set to 6

When XSG (bit 2 of parameter No. 3008) is 1, the PMC axis control skip signal, and measurement position arrival signal are allocated to X0006 and the skip signal is allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(T series)
	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X006		ESKIP	-MIT2	+MIT2	-MIT1	+MIT1	XAE2	XAE1	(T series)

	#7	#6	#5	#4	#3	#2	#1	#0	
		ESKIP				XAE3	XAE2	XAE1	(M series)

Example 2. When No.3012 is set to 5 and No.3019 is set to 5

When XSG (bit 2 of parameter No. 3008) is 1, the PMC axis control skip signal, measurement position arrival signal, and skip signal are allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X005	SKIP	ESKIP SKIP6	-MIT2 SKIP5	+MIT2 SKIP4	-MIT1 SKIP3	+MIT1 SKIP2	XAE2 SKIP8	XAE1 SKIP7	(T series)

	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	ESKIP SKIP6	SKIP5	SKIP4	SKIP3	XAE3 SKIP2	XAE2 SKIP8	XAE1 SKIP7	(M series)

NOTE

This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
The X addresses that can be actually used are shown below, but they depend on the configuration of I/O Link point count expansion options.
X0 to X127, X200 to X327

3030	Allowable number of digits for the M code

3031	Allowable number of digits for the S code

3032	Allowable number of digits for the T code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8

Set the allowable numbers of digits for the M, S, and T codes.
When 0 is set, the allowable number of digits is assumed to be 8.

NOTE

Up to 5 digits can be specified in the S code.

3033	Allowable number of digits for the B code (second auxiliary function)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8

Set the allowable number of digits for the second auxiliary function.
When 0 is set, the allowable number of digits is assumed to be 8.
To enable a decimal point to be specified, bit 0 (AUP) of parameter No. 3450 must be set to 1. In this case, the allowable number of digits set in this parameter includes the number of decimal places.
If a value exceeding the allowable number of digits is specified, the alarm (PS0003) is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC		DRC		PPD			MCN
	DAC	DAL	DRC	DRL	PPD			MCN

[Input type] Parameter input
[Data type] Bit path

#0 MCN Machine position
0: Regardless of whether input is made in mm or inches, the machine position is displayed in mm for millimeter machines, or in inches for inch machines.
1: When input is made in mm, the machine position is displayed in mm, and when input is made in inches, the machine position is displayed in inches accordingly.

#3 PPD Relative position display when a coordinate system is set
0: Not preset
1: Preset

NOTE
If any of the following is executed when PPD is set to 1, the relative position display is preset to the same value as the absolute position display:
(1) Manual reference position return
(2) Coordinate system setting based on G92 (G50 for G code system A on the lathe system)
(3) Workpiece coordinate system presetting based on G92.1 (G50.3 for G code system A on the lath system)
(4) When a T code for the T series is specified.

#4 DRL Relative position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

NOTE
In the T series, whether to exclude a tool offset when displaying the relative position is determined by the setting of bit 0 (DRP) of parameter No. 3129.

#5 DRC When relative positions are displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

#6 DAL Absolute position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

NOTE
In T series, whether to exclude a tool offset when displaying the absolute position is determined by the setting of bit 1 (DAP) of parameter No. 3129.

#7 DAC When an absolute position are displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3106				OPH				

[Input type] Setting input
[Data type] Bit

#4 OPH The operation history screen is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3107	MDL				GSC			

[Input type] Parameter input
[Data type] Bit path

#3 GSC The feedrate to be displayed is:
0: Feedrate per minute.
1: Determined by bit 5 (FSS) of parameter No. 3191.

#7 MDL The modal state on the program(MDI) screen of the 8.4 inch screen is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3111		OPS	OPM			SVP	SPS	SVS

[Input type] Setting input
[Data type] Bit path

#0 SVS The soft key for displaying the servo setting screen is:
0: Not displayed.
1: Displayed.

#1 SPS The soft key for displaying the spindle setting screen is:
0: Not displayed.
1: Displayed.

#2 SVP Spindle synchronization errors displayed on the spindle tuning screen
0: Instantaneous values are displayed.
1: Peak-hold values are displayed.
Spindle synchronization errors are displayed on the side of the spindle that functions as a slave axis in spindle synchronization control.

#5 OPM Operating monitor
0: Not displayed
1: Displayed

#6 OPS The speedometer on the operating monitor screen indicates:
0: Spindle motor speed
1: Spindle speed

	#7	#6	#5	#4	#3	#2	#1	#0
3112					EAH	OMH		

[Input type] Parameter input
[Data type] Bit

#2 OMH The external operator message history screen is:
0: Not displayed.
1: Displayed.

#3 EAH Messages of the external alarm/macro alarm in alarm or operation history:
0: Not recorded
1: Recorded

> **NOTE**
> This parameter is valid when bit 7 (HAL) of parameter No. 3196 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3115							NDAx	NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

> **NOTE**
> When using the electric gear box (EGB) function (M series), set 1 for the EGB dummy axis to disable current position display.

#1 NDAx The current position and the amount of the movement to be made in absolute and relative coordinates are:
0: Displayed.
1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3117								SMS

[Input type] Parameter input
[Data type] Bit path

#0 SMS On the program check screen of the 8.4-inch display unit, the function for displaying the spindle load meter and spindle speed meter in the remaining movement amount display position and modal information display position is:
0: Disabled.
1: Enabled.

3122	Time interval used to record time data in operation history

[Input type] Parameter input
[Data type] Word path
[Unit of data] min
[Valid data range] 0 to 1440

When history data is recorded within a set time period, the time for each set time period is recorded in the history data.
When 0 is set, the specification of a time period of 10 minutes is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
3129							DAP	DRP

[Input type] Parameter input
[Data type] Bit path

#0 DRP For relative position display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

> **NOTE**
> In the M series, whether to exclude tool length compensation when displaying the relative position is determined by bit 4 (DRL) of parameter No. 3104.

#1 DAP For absolute position display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

> **NOTE**
> In M series, whether to exclude the tool length offset when displaying the absolute position is determined according to the setting of bit 6 (DAL) of parameter No. 3104.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 32, 65 to 90

In order to distinguish axes under parallel operation, synchronization control, and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than a synchronization control axis and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a subscript.

Example)
When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

When the subscription of an axis name is not set in a 2-path system, the subscription of an axis name is automatically set to the path number. To hide the subscription of an axis name, set the parameter of the subscription of an axis name to the ASCII code (32) of a space.

3141	Path name (1st character)
3142	Path name (2nd character)
3143	Path name (3rd character)
3144	Path name (4th character)
3145	Path name (5th character)
3146	Path name (6th character)
3147	Path name (7th character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] See the character-code correspondence table.
Specify a path name with codes.
Any character string consisting of alphanumeric characters, katakana characters, and special characters with a maximum length of seven characters can be displayed as a series name.

When 0 is set in parameter No. 3141, the path name is displayed according to the following table.

Language to display in CNC	Path name
English	HEAD1 (HEAD2)
Japanese	刃物台 1 (刃物台 2)
German	KANAL1 (KANAL2)
French	TETE1 (TETE2)
Traditional Chinese	HEAD1 (HEAD2)
Simplified Chinese	路径 1 (路径 2)
Italian	TEST1 (TEST2)
Korean	HEAD1 (HEAD2)
Spanish	CAB.1 (CAB.2)
Dutch	KAN.1 (KAN.2)
Danish	HOVED1 (HOVED2)

Language to display in CNC	Path name
Portuguese	CABEC.1 (CABEC.2))
Polish	GLOWIC1 (GLOWIC2)
Hungarian	FEJ1 (FEJ2)
Swedish	HUVUD1 (HUVUD2)
Czech	KANAL1 (KANAL2)
Russian	П У Т Ь 1 (П У Т Ь 2)
Turkish	HEAD1 (HEAD2)
Bulgarian	Г Л В А 1 (Г Л В А 2)

NOTE
For characters and codes, see Appendix G, “CHARACTER CODE LIST”.

	#7	#6	#5	#4	#3	#2	#1	#0
3193						DOP		

[Input type] Parameter input
[Data type] Bit

#2 DOP In 2-path control, on the POSITION screen (absolute, relative, all, manual handle interruption), PROGRAM CHECK screen, and ALARM screen, two paths' information is:
0: Displayed at the same time.
1: Not displayed at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3195	EKE	HDE	HKE			CPR		

[Input type] Parameter input
[Data type] Common to the bit system

#2 CPR Displaying of the parameter setting support screen by function key [SYSTEM] is:
0: Performed.
1: Not performed.

#5 HKE A key operation history is:
0: Recorded.
1: Not recorded.

#6 HDE A DI/DO history is:
0: Recorded.
1: Not recorded.

#7 EKE The [ALL CLEAR] soft key for clearing all history data is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3196	HAL	HOM			HMV	HPM	HWO	HTO

[Input type] Parameter input
[Data type] Bit

#0 HTO A modification history of tool offset data is:
0: Not recorded.
1: Recorded.

#1 HWO A modification history of workpiece offset data/extended workpiece offset data/workpiece shift (T series) is:
0: Not recorded.
1: Recorded.

#2 HPM A modification history of parameters is:
0: Not recorded.
1: Recorded.

#3 HMV A modification history of custom macro common variables is:
0: Not recorded.
1: Recorded.

#6 HOM The operation history is:
0: Recorded.
1: Not recorded.

#7 HAL When an alarm is issued, additional information (modal data, absolute coordinates, and machine coordinates present at the issuance of the alarm) is:
0: Recorded in the operation history and alarm history.
1: Not recorded in the operation history and alarm history.
To record as many alarm history items as possible, rather than detailed alarm information, set 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE						

[Input type] Parameter input
[Data type] Bit path

#6 NPE With an M02, M30, or M99 block, program registration is assumed to be:
0: Completed
1: Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9	OSR			NE8

[Input type] Parameter input
[Data type] Bit path

#0 NE8 Editing of subprograms with program numbers 8000 to 8999
0: Not inhibited
1: Inhibited
When this parameter is set to 1, the following editing operations are disabled:
(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 8000 to 8999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 8000 to 8999 are not output.)
(3) Program number search
(4) Program editing of registered programs
(5) Program registration
(6) Program collation
(7) Displaying programs

> **NOTE**
> This parameter setting does not affect the following programs:
> (1) Programs on the Data Server
> (2) Programs for running and editing memory card programs on a memory card

#3 OSR Pressing the [O SEARCH] soft key without entering a program number with keys in a program number search:
0: Searches for the next program number (order of registration).
1: Disables the search.

#4 NE9 Editing of subprograms with program numbers 9000 to 9999
0: Not inhibited
1: Inhibited
When this parameter is set to 1, the following editing operations are disabled:
(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 9000 to 9999 are not output.)
(3) Program number search
(4) Program editing of registered programs
(5) Program registration
(6) Program collation
(7) Displaying programs

> **NOTE**
> This parameter setting does not affect the following programs:
> (1) Programs on the Data Server
> (2) Programs for running and editing memory card programs on a memory card

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL	MER	MZE					

[Input type] Parameter input
[Data type] Bit path

#5 MZE After MDI operation is started, program editing during operation is:
0: Enabled
1: Disabled

#6 MER When the last block of a program has been executed at single block operation in the MDI mode, the executed block is:
0: Not deleted
1: Deleted

> **NOTE**
> When MER is set to 0, the program is deleted if the end-of-record mark (%) is read and executed. (The mark % is automatically inserted at the end of a program.)

#7 MCL Whether a program prepared in the MDI mode is cleared by reset
0: Not deleted
1: Deleted

	#7	#6	#5	#4	#3	#2	#1	#0
3204		MKP						

[Input type] Parameter input
[Data type] Bit path

#6 MKP When M02, M30, or EOR(%) is executed during MDI operation, the created MDI program is:
0: Erased automatically.
1: Not erased automatically.

> **NOTE**
> If the bit 6 (MER) of parameter No. 3203 is 1, executing the last block provides a choice of whether to automatically erase a created program.

	#7	#6	#5	#4	#3	#2	#1	#0
3205					PNS	TOK		

[Input type] Parameter input
[Data type] Bit

#2 TOK A WORD COPY or WORD MOVE on the program screen:
0: Is performed as usual.
1: Can also be performed on a record-by-record basis from a program to the key-in buffer.

#3 PNS On the program screen, a search with the cursor keys is:
0: Performed.
1: Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
3207			VRN					

[Input type] Parameter input
[Data type] Bit

#5 VRN On the custom macro variable screen, the variable names of common variables #500 to #549 are:
0: Not displayed.
1: Displayed.

3210	Program protection

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999

This parameter sets a password for protecting program Nos. 9000 to 9999. When a value other than zero is set in this parameter and this value differs from the keyword set in parameter No.3211, bit 4 (NE9) of parameter No.3202 for protecting program Nos. 9000 to 9999 is automatically set to 1.

This disables the editing of program Nos. 9000 to 9999. Until the value set as the password is set as a keyword, NE9 cannot be set to 0 and the password cannot be modified.

> **NOTE**
> 1 The state where password ≠ 0 and password ≠ keyword is referred to as the locked state. When an attempt is made to modify the password by MDI input operation in this state, the warning message "WRITE PROTECTED" is displayed to indicate that the password cannot be modified. When an attempt is made to modify the password with G10 (programmable parameter input), alarm (PS0231) is issued.
> 2 When the value of the password is not 0, the parameter screen does not display the password. Care must be taken in setting a password.

3211	Program protection key

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999

When the value set as the password (set in parameter No.3210) is set in this parameter, the locked state is released and the user can now modify the password and the value set in bit 4 (NE9) of parameter No.3202.

> **NOTE**
> The value set in this parameter is not displayed. When the power is turned off, this parameter is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3280								NLC

[Input type] Parameter input
[Data type] Bit

#0 NLC Dynamic display language switching is:
0: Enabled.
1: Disabled.
When dynamic display language switching is disabled, the language setting screen is not displayed. In this case, change the setting of parameter No. 3281 on the parameter screen then turn on the power again to switch the display language.

3281	Display language

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 18

Select a display language from the following:
0 : English
1 : Japanese
2 : German
3 : French
4 : Chinese(traditional characters)
5 : Italian
6 : Korean
7 : Spanish
8 : Dutch

9 : Danish
10 : Portuguese
11 : Polish
12 : Hungarian
13 : Swedish
14 : Czech
15 : Chinese(simplified characters)
16 : Russian
17 : Turkish
18 : Bulgarian
If a number not indicated above is set, English is selected.

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB	ABS	MAB				DPI
			ABS	MAB				DPI

[Input type] Parameter input
[Data type] Bit path

#0 DPI When a decimal point is omitted in an address that can include a decimal point
0: The least input increment is assumed. (Normal decimal point input)
1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point input)

#4 MAB Switching between the absolute and incremental commands in MDI operation
0: Performed by G90 or G91
1: Depending on the setting of bit 5 (ABS) of parameter No.3401

NOTE
When G code system A of the T series is used, this parameter is invalid.

#5 ABS Program command in MDI operation
0: Assumed as an incremental command
1: Assumed as an absolute command

NOTE
ABS is valid when bit 4 (MAB) of parameter No.3401 is set to 1.
When G code system A of the T series is used, this parameter is invalid.

#6 GSB The G code system is set.
#7 GSC

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM	G91			G01
	G23	CLR			G91	G19	G18	G01

[Input type] Parameter input

[Data type] Bit path

#0 G01 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

#1 G18 Plane selected when power is turned on or when the control is cleared
0: G17 mode (plane XY)
1: G18 mode (plane ZX)

#2 G19 Plane selected when power is turned on or when the control is cleared
0: The setting of bit 1 (G18) of parameter No.3402 is followed.
1: G19 mode (plane YZ)
When this bit is set to 1, set bit 1 (G18) of parameter No.3402 to 0.

G19	G18	G17, G18, or G19 mode
0	0	G17 mode (X-Y plane)
0	1	G18 mode (Z-X plane)
1	0	G19 mode (Y-Z plane)

#4 FPM At power-on time or in the cleared state:
0: G99 or G95 mode (feed per revolution) is set.
1: G98 or G94 mode (feed per minute) is set.

#6 CLR Reset button on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the OPERATOR'S MANUAL.

#7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B		M02	M30		SBP		

[Input type] Parameter input
[Data type] Bit path

#2 SBP In an external device subprogram call (M198), the address P format is based on:
0: File number specification
1: Program number specification

> **NOTE**
> In memory card operation, the program number specification format is used, regardless of the setting of this parameter.

#4 M30 When M30 is specified in a memory operation:
0: M30 is sent to the machine, and the head of the program is automatically searched for. So, when the ready signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1: M30 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

#5 M02 When M02 is specified in memory operation

0: M02 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.

1: M02 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

#7 M3B The number of M codes that can be specified in one block

0: One

1: Up to three

	#7	#6	#5	#4	#3	#2	#1	#0
3405				CCR			DWL	AUX
							DWL	AUX

[Input type] Parameter input
[Data type] Bit path

#0 AUX When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the multiplication factor for a value output (onto the code signal) relative to a specified value is such that:

0: The same multiplication factor is used for both of metric input and inch input.

1: A multiplication factor used for inch input is 10 times greater than that used for metric input.

When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the value output onto the code signal is a specified value multiplied by a value indicated below.

Increment system		Parameter AUX=0	Parameter AUX=1
Metric input system	IS-A for reference axis	100 times	100 times
	IS-B for reference axis	1000 times	1000 times
	IS-C for reference axis	10000 times	10000 times
Inch input system	IS-A for reference axis	100 times	1000 times
	IS-B for reference axis	1000 times	10000 times
	IS-C for reference axis	10000 times	100000 times

#1 DWL The dwell time (G04) is:

0: Always dwell per second.

1: Dwell per second in the feed per minute mode, or dwell per rotation in the feed per rotation mode.

#4 CCR Addresses used for chamfering

0: Address is “I”, “J”, or “K”.
In direct drawing dimension programming, addresses ",C", ",R", and ",A" (with comma) are used in stead of "C", "R", and "A".

1: Address is “C”.
Addresses used for direct drawing dimension programming are "C", "R", and "A" without comma.

> **NOTE**
> If this bit (CCR) is set to 0, the function for changing the compensation direction by specifying I, J, or K in a G01 block in the tool nose radius compensation mode cannot be used.
> If this bit (CCR) is set to 1 when address C is used as an axis name, the chamfer function cannot be used.

	#7	#6	#5	#4	#3	#2	#1	#0
3406	C07	C06	C05	C04	C03	C02	C01	

	#7	#6	#5	#4	#3	#2	#1	#0
3407	C15	C14	C13	C12	C11	C10	C09	C08

	#7	#6	#5	#4	#3	#2	#1	#0
3408	C23	C22	C21	C20	C19	C18	C17	C16

	#7	#6	#5	#4	#3	#2	#1	#0
3409		C30	C29	C28	C27	C26	C25	C24

[Input type] Parameter input
[Data type] Bit

C01 to C30 If bit 6 (CLR) of parameter No.3402 is set to 1, set a group of G codes to be placed in the cleared state when the CNC is reset by the 

key of the MDI panel, the external reset signal, the reset & rewind signal, or the emergency stop signal.
The table below indicates the correspondence between bits and G code groups
The setting of a bit has the following meaning:
0: Places the G code group in the cleared state.
1: Does not place G code group in the cleared state.

Parameter	G code group
C01	01
C02	02
C03	03
:	:
C30	30

3410	Tolerance of arc radius

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
When a circular interpolation command is executed, the tolerance for the radius between the start point and the end point is set.

> **NOTE**
> When the setting is 0, the difference between the arc radius values is not checked.

3411	M code preventing buffering 1
3412	M code preventing buffering 2
:	
3420	M code preventing buffering 10

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.
M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

3421	Range specification 1 of M codes that do not perform buffering (lower limit)
3422	Range specification 1 of M codes that do not perform buffering (upper limit)
3423	Range specification 2 of M codes that do not perform buffering (lower limit)
3424	Range specification 2 of M codes that do not perform buffering (upper limit)
3425	Range specification 3 of M codes that do not perform buffering (lower limit)
3426	Range specification 3 of M codes that do not perform buffering (upper limit)
3427	Range specification 4 of M codes that do not perform buffering (lower limit)
3428	Range specification 4 of M codes that do not perform buffering (upper limit)
3429	Range specification 5 of M codes that do not perform buffering (lower limit)
3430	Range specification 5 of M codes that do not perform buffering (upper limit)
3431	Range specification 6 of M codes that do not perform buffering (lower limit)
3432	Range specification 6 of M codes that do not perform buffering (upper limit)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

When a specified M code is within the range specified with parameter Nos.3421 and 3422, 3423 and 3424, 3425 and 3426, 3427 and 3428, 3429 and 3430, or 3431 and 3432, buffering for the next block is not performed until the execution of the block is completed.

NOTE

1 M00, M01, M02, and M30 are M codes that do not perform buffering, regardless of parameter setting.
M98, M99, M codes for calling subprograms, and M codes for calling custom macros are M codes that performs buffering, regardless of parameter setting.

> **NOTE**
> 2 If the minimum value is greater than the maximum value, the setting is invalid.
> 3 If there is only one data item, the minimum value must be equal to the maximum value.

	#7	#6	#5	#4	#3	#2	#1	#0
3450	BDX							AUP

[Input type] Parameter input
[Data type] Bit path

#0 AUP The second auxiliary function specified in the calculator-type decimal point input format, with a decimal point, or with a negative value is:
0: Disabled.
1: Enabled.
If the second auxiliary function is specified after setting this bit to 0, the following operation results:
1. When a value is specified without a decimal point
   A specified value is output onto the code signal without modification, regardless of the setting of the calculator-type decimal point input format (with bit 0 (DPI) of parameter No. 3401).
2. When a value is specified with a decimal point
   The alarm (PS0007) is issued.
3. When a negative value is specified
   The alarm (PS0006) is issued.

#7 BDX This parameter prevents the unit of the argument from depending on the setting of bit 2 (BCD) of parameter No. 8132 when a subprogram call by an ASCII code is performed with the address (specified by parameter No. 3460) of the second auxiliary function.
0: When bit 0 (AUP) of parameter No. 3450 is 1, the unit of the argument depends on the setting of bit 2 (BCD) of parameter No. 3450.
1: The same unit of the argument is used. The unit when bit 2 (BCD) of parameter No. 8132 is 1 is specified.
[Example]
A setting is made so that address B is used to call O9004, and the program O1 below is executed with parameter No.3460 = 66.

```
O1      O9004
B2      #500 = #146
M30     M99
```

When the increment system is IS-B, and metric input is used, #500 assumes a value indicated in the table below.

Parameter DPI (No.3401#0)	Parameter AUP (No.3450#0)	BDX=0		BDX=1
		Parameter BCD(No.8132#2)=0	Parameter BCD(No.8132#2)=1	
0	0	2.000	2.000	2.000
	1	2.000	0.002	0.002
1	0	2.000	2.000	2.000
	1	2.000	2.000	2.000

	#7	#6	#5	#4	#3	#2	#1	#0
3451								
								GQS

[Input type] Parameter input
[Data type] Bit path

#0 GQS When threading is specified, the threading start angle shift function (Q) is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3452	EAP							

[Input type] Parameter input
[Data type] Bit path

#7 EAP When bit 0 (ADX) of parameter No.3455 is set to 1, calculator-type decimal point input at a macro calling argument address is:
0: Enabled.
1: Disabled.

> **NOTE**
> This parameter is valid when bit 0 (DPI) of parameter No.3401 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3454				G1B				

[Input type] Parameter input
[Data type] Bit path

#4 G1B In programmable parameter input, specifying a change to a specific bit parameter is:
0: Disabled.
1: Enabled. (A bit number is specified with Q_.)

	#7	#6	#5	#4	#3	#2	#1	#0
3455								AXDx

[Input type] Parameter input
[Data type] Bit axis

#0 AXDx If a decimal point is omitted for an axis address with which a decimal point can be used, the value is determined:
0: In accordance with the least input increment. (Normal decimal point input)
1: In millimeters, inches, or seconds. (calculator-type decimal point input)

> **NOTE**
> This parameter specifies the calculator-type decimal point input function for each axis.
> For the same axis name, be sure to make the same setting.

3460	Second auxiliary function specification address

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65to67, 85to87

Specify which of A, B, C, U, V, and W is to be used as the address for specifying the second auxiliary function. If an address used as an axis name is specified, the second auxiliary function is disabled.

Name	A	B	C	U	V	W
Setting value	65	66	67	85	86	87

Address B is assumed when a value other than the above is set.
However, the name U, V, or W can be used with the T series only when G code system B or C is used. When a value from 85 to 87 is specified with G code system A, the specification address for the second auxiliary function is B.

	#7	#6	#5	#4	#3	#2	#1	#0
3605								BDPx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 BDPx Both-direction pitch error compensation is:
0: Not used.
1: Used.

> **NOTE**
> The both-direction pitch error compensation option is required.

3620	Number of the pitch error compensation position for the reference position for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023

Set the number of the pitch error compensation position for the reference position for each axis.

3621	Number of the pitch error compensation position at extremely negative position for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023

Set the number of the pitch error compensation position at the extremely negative position for each axis.

3622	Number of the pitch error compensation position at extremely positive position for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023

Set the number of the pitch error compensation position at the extremely positive position for each axis.
This value must be larger than set value of parameter (No.3620).

3623	Magnification for pitch error compensation for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100

Set the magnification for pitch error compensation for each axis.
If the magnification is set to 1, the same unit as the detection unit is used for the compensation data.
If 0 is set, compensation is not performed.

3624	Interval between pitch error compensation positions for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

The pitch error compensation positions are arranged with equal spacing. The space between two adjacent positions is set for each axis. The minimum interval between pitch error compensation positions is limited and obtained from the following equation:
Minimum interval between pitch error compensation positions = maximum feedrate/7500
Unit : Minimum interval between pitch error compensation positions: mm, inch, deg
Maximum feedrate: mm/min, inch/min, deg/min

Example:

When the maximum feedrate is 15000 mm/min, the minimum interval between pitch error compensation positions is 2 mm.

3625	Travel distance per revolution in pitch error compensation of rotation axis type

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

If the pitch error compensation of rotation axis type is performed (bit 1 (ROSx) of parameter No.1006 is set to 0 and bit 0 (ROTx) of parameter No.1006 is set to 1), set the travel distance per revolution. The travel distance per revolution does not have to be 360 degrees, and a cycle of pitch error compensation of rotation axis type can be set.
However, the travel distance per revolution, compensation interval, and number of compensation points must satisfy the following condition:
(Travel distance per revolution)
= (Compensation interval) × (Number of compensation points)
The compensation at each compensation point must be set so that the total compensation per revolution equals 0.

> **NOTE**
> If 0 is set, the travel distance per revolution becomes 360 degrees.

3626	Number of the both-direction pitch error compensation position at extremely negative position (for movement in the negative direction)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023, 3000 to 4023

When using both-direction pitch error compensation, set the number of compensation point at the farthest end in the negative direction for a movement in the negative direction.

> **NOTE**
> 1 For a movement in the positive direction, set the compensation point number at the farthest end in the negative direction in parameter No.3621.
> 2 A set of compensation data items for a single axis should not be set to lie astride 1023 to 3000.

3627	Pitch error compensation at reference position when a movement to the reference position is made from the direction opposite to the direction of reference position return

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -32768 to 32767

Set the absolute value of pitch error compensation at reference position when a movement to the reference position is made from the negative direction if the direction of reference position return (bit 5 (ZMI) of parameter No.1006) is positive or from the positive direction if the direction of reference position return is negative.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Input type] Parameter input
[Data type] Bit path

#1 NRF With the first move command (G00) after switching the serial spindle to Cs contour control axis:
0: A reference position return operation is once performed then positioning is performed.
1: A normal positioning operation is performed.

NOTE
1 When using the Cs axis establishment function, this parameter is recommended to be set to 1.
2 The setting of this parameter is valid for G00. The first rapid traverse of a canned cycle is normal positioning regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

#1 EMS The multi-spindle control function is:
0: Used.
1: Not used.

NOTE
Make the setting on the side of the path in which multi-spindle control is unnecessary in 2-path control.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

> **NOTE**
> 1 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
> 2 A maximum of one analog spindle can be controlled.
> 3 When using an analog spindle, set it at the end of the spindle configuration.

3717	Motor number to each spindle

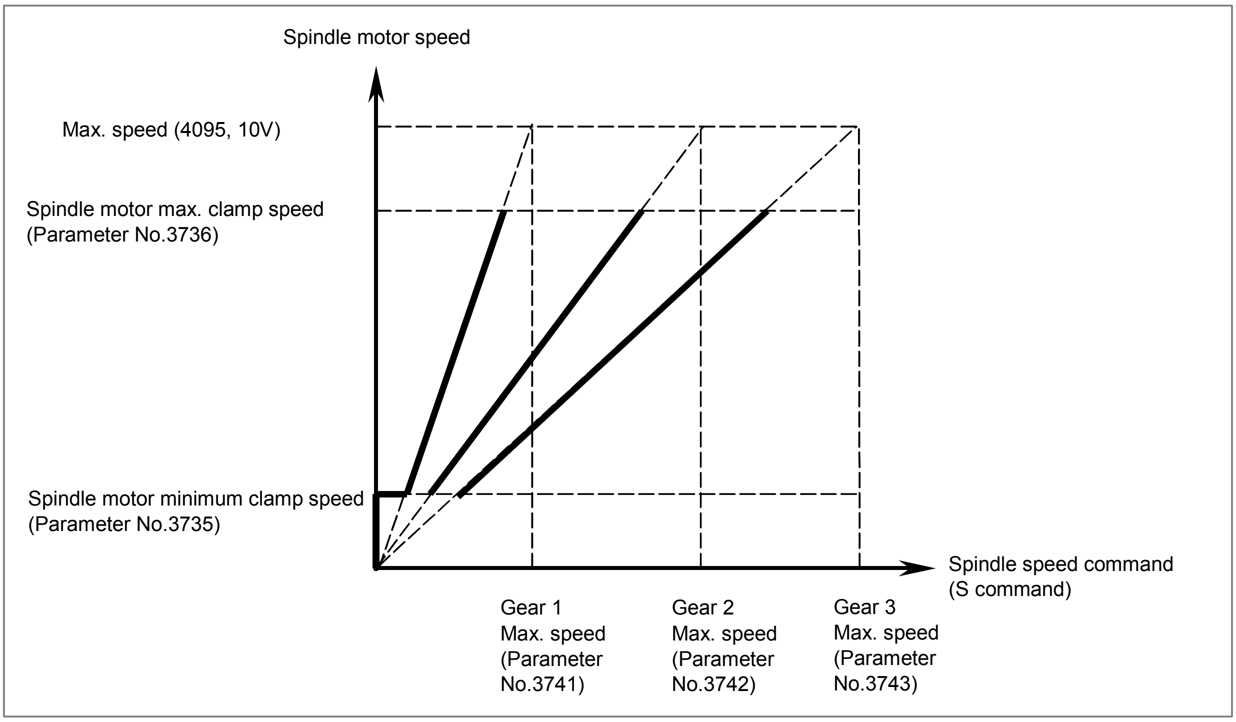
> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes
Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
3: Spindle motor connected to amplifier number 3 is used.

> **NOTE**
> When using an analog spindle, set it at the end of the spindle configuration.
> (Example)
> When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4
	(Note)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
Set the maximum spindle speed corresponding to each gear.



NOTE
If a type-T gear shift scheme is selected for the M series (with the constant surface speed control option installed or bit 4 (GTT) of parameter No. 3706 = 1), parameter No. 3744 is usable also in the M series.
Note, however, that, even in this case, only up to three main gear stages are usable for rigid tapping.

3770	Axis as the calculation reference in constant surface speed control

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis as the calculation reference in constant surface speed control.

NOTE
When 0 is set, constant surface speed control is always applied to the X-axis. In this case, specifying P in a G96 block has no effect on the constant surface speed control.

3781	P code for selecting the spindle in multi-spindle control

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767

If bit 3 (MPP) of parameter No. 3703 is set to 1, set the P code to select each spindle under multi-spindle control. Specify the P code in a block containing the S command.

Example)
If the P code value for selecting the second spindle is set to 3,
S1000 P3;
causes the second spindle to rotate at S1000.

NOTE
1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
3 Under 2-path control, the P code specified here is valid for each path.
For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6 To use this parameter, enable multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1).

Parameters Nos. 4000 to 4799 below are basically used with the serial spindle amplifier. For details of these parameters, refer to either of the following manuals and other related documents, depending on the spindle that is actually connected.

- FANUC AC SPINDLE MOTOR α*i* series Parameter Manual (B-65280EN)

	#7	#6	#5	#4	#3	#2	#1	#0
4900								FLRs

[Input type] Parameter input
[Data type] Bit spindle

#0 FLRs When the spindle speed fluctuation detection function (T series) is used, the unit of an allowable ratio (q) and fluctuation ratio (r) set by parameter No. 4911 and No. 4912 is:
0: 1%
1: 0.1%

4911	Allowable speed ratio (q) used to assume that the spindle has reached a specified speed

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000
When the spindle speed fluctuation detection function is used, set an allowable speed ratio (q) used to assume that the spindle has reached a specified speed.

> **NOTE**
> The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4912	Spindle variation ratio (r) for not issuing a spindle speed fluctuation detection alarm

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, set a spindle fluctuation ratio (r) for not issuing an alarm.

> **NOTE**
> The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4913	Spindle speed fluctuation width (i) for not issuing a spindle speed fluctuation detection alarm

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999

When the spindle speed fluctuation detection function is used, set an allowable fluctuation width (i) for not issuing an alarm.

4914	Time (p) from the change of a specified speed until spindle speed fluctuation detection is started

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] msec
[Valid data range] 0 to 999999

When the spindle speed fluctuation detection function is used, set a time (p) from the change of a specified speed until spindle speed fluctuation detection is started. In other words, spindle speed fluctuation detection is not performed until a set time has elapsed after a specified speed is changed. However, when the actual spindle speed is assumed to have reached a specified value within a set time (p), spindle speed fluctuation detection is started.

	#7	#6	#5	#4	#3	#2	#1	#0
4950						ISZs	IDMs	IORs

[Input type] Parameter input
[Data type] Bit spindle

#0 IORs Resetting the system in the spindle positioning mode
0: Does not release the mode.
1: Releases the mode

#1 IDMs The direction of spindle positioning (half-fixed angle positioning based on M codes) is:
0: Plus direction.
1: Minus direction.

#2 ISZs When an M code for spindle orientation is specified in spindle positioning:
0: The spindle is switched to the spindle positioning mode, and spindle orientation operation is performed.
1: Only the switching of the spindle to the spindle positioning mode is performed. (Spindle orientation operation is not performed.)

4960	M code specifying the spindle orientation

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97
Set an M code for switching to the spindle positioning mode.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4961	M code releasing the spindle positioning mode

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97
Set an M code for canceling the spindle positioning mode on the spindle positioning axis.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4962	M code for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 9999999
Two methods are available for specifying spindle positioning. One method uses axis address for arbitrary-angle positioning. The other use an M code for half-fixed angle positioning. This parameter sets an M code for the latter method.
In this parameter, set an M code to be used for half-fixed angle positioning based on M codes.
Six M code from Mα to M(α+5) are used for half-fixed angle positioning, when α is the value of this parameter.

- When the number of M codes is set in parameter No. 4964, let α be the value set in parameter No. 4962, and let β be the value set in parameter No. 4964. Then, β M codes from Mα to M(α+β-1) are used as M codes for half-fixed angle positioning based on M codes.

The table below indicates the relationship between the M codes and positioning angles.

M code	Positioning angle	Example: Positioning angle when θ = 30°
Mα	θ	30°
M(α+1)	2θ	60°
M(α+2)	3θ	90°
M(α+3)	4θ	120°
M(α+4)	5θ	150°
M(α+5)	6θ	180°
:	:	:
M(α+β-1)	β×θ	β×30°

β represents the number of M codes set in parameter No. 4964.
(When parameter No. 4964 is set to 0, β = 6.)
θ represents the basic angular displacement set in parameter No.4963.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4963	Basic angle for half-fixed angle positioning

[Input type] Parameter input
[Data type] Real spindle
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 to 60

This parameter sets a basic angular displacement used for half-fixed angle positioning using M codes.

4964	Number of M codes for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 255

This parameter sets the number of M codes used for Half-fixed angle positioning using M codes.
As many M codes as the number specified in this parameter, starting with the M code specified in parameter No.4962, are used to specify half-fixed angle positioning.
Let α be the value of parameter No.4962, and let β be the value of parameter No.4964. That is, M codes from Mα to M(α+β-1) are used for half-fixed angle positioning.
Setting this parameter to 0 has the same effect as setting 6. That is, M code from Mα to M(α+5) are used for half-fixed angle positioning.

NOTE

1 Make sure that M codes from Mα to M (α+β-1) do not duplicate other M codes.
2 Do not set an M code that duplicates other M codes used for spindle positioning.
3 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

	#7	#6	#5	#4	#3	#2	#1	#0
5001								
		EVO			TAL		TLB	TLC

[Input type] Parameter input
[Data type] Bit path

#0 TLC
#1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.
Tool length compensation A :
Z-axis at all times
Tool length compensation B :
Axis perpendicular to a specified plane (G17/G18/G19)
Tool length compensation C :
Axis specified in a block that specifies G43/G44

#3 TAL Tool length compensation C
0: Generates an alarm when two or more axes are offset
1: Not generate an alarm even if two or more axes are offset

#6 EVO If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):
0: The new value becomes valid in a block where G43, G44, or an H code is specified next.
1: The new value becomes valid in a block where buffering is performed next.

	#7	#6	#5	#4	#3	#2	#1	#0
5002						LWT	LGN	

[Input type] Parameter input
[Data type] Bit path

#1 LGN Geometry offset number of tool offset
0: Is the same as wear offset number
1: Specifies the geometry offset number by the tool selection number

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

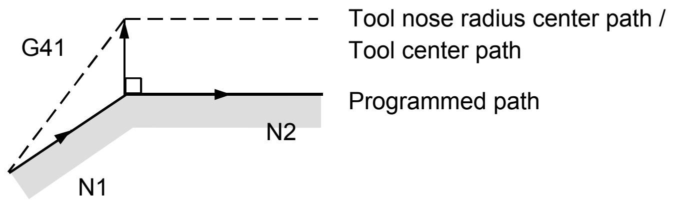
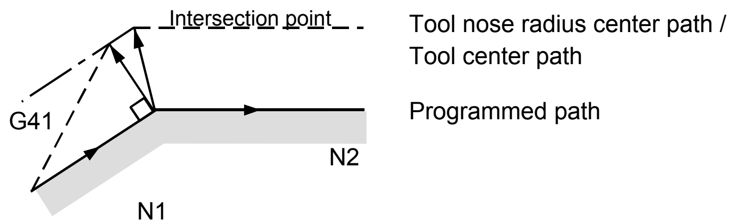
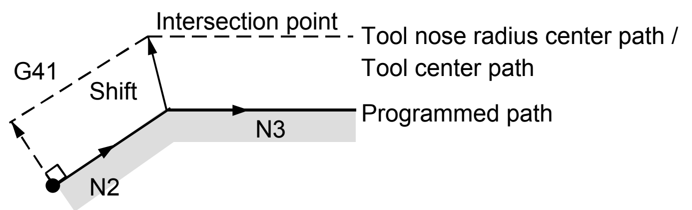
#2 LWT Tool wear compensation is performed by:
0: Moving the tool.
1: Shifting the coordinate system.

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

	#7	#6	#5	#4	#3	#2	#1	#0
5003							**SUV**	**SUP**

[Input type] Parameter input
[Data type] Bit path

#0 SUP
#1 SUV These bits are used to specify the type of startup/cancellation of cutter compensation or tool nose radius compensation.

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output. 
0	1	Type B	A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output. 
1	0 1	Type C	When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.  When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.

NOTE
When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS0*i*-TC is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	
						ODI		

[Input type] Parameter input
[Data type] Bit path

#1 ORC The setting of a tool offset value is corrected as:
0: Diameter value
1: Radius value

NOTE
This parameter is valid only for an axis based on diameter specification. For an axis based on radius specification, specify a radius value, regardless of the setting of this parameter.

#2 ODI The setting of a cutter compensation value is corrected as:
0: Radius value
1: Diameter value

	#7	#6	#5	#4	#3	#2	#1	#0
5008				MCR				

[Input type] Parameter input
[Data type] Bit path

#4 MCR If G41/G42 (cutter compensation or tool nose radius compensation) is specified in the MDI mode, an alarm is:
0: Not raised.
1: Raised. (alarm PS5257)

5028	Number of digits of an offset number used with a T code command

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Specify the number of digits of a T code portion that is used for a tool offset number (wear offset number when the tool geometry/wear compensation function is used).
When 0 is set, the number of digits is determined by the number of tool compensation values.
When the number of tool compensation values is 1 to 9: Lower 1 digit
When the number of tool compensation values is 10 to 99: Lower 2 digits
When the number of tool compensation values is 100 to 200: Lower 3 digits
Example :
When an offset number is specified using the lower 2 digits of a T code, set 2 in parameter No. 5028.
Txxxxxx yy
xxxxxx : Tool selection
yy : Tool offset number

> **NOTE**
> A value longer than the setting of parameter No. 3032 (allowable number of digits of a T code) cannot be set.

5029	Number of tool compensation value memories common to paths

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to number of tool compensation values

When using memories common to paths, set the number of common tool compensation values in this parameter.

Ensure that the setting of this parameter does not exceed the number of tool compensation values set for each path (parameter No. 5024).

[Example 1]

When parameter No. 5029 = 10, parameter No. 5024 (path 1) = 15, and parameter No. 5024 (path 2) = 30 in a 2-path system, tool compensation numbers 1 to 10 of all paths are made common.

[Example 2]

When parameter No. 5029 = 20 and the other conditions are the same as for Example 1, tool compensation numbers 1 to 15 are made common.

> **NOTE**
> 1 Ensure that the setting of parameter No. 5029 does not exceed the number of tool compensation values for each path (parameter No. 5024). If the setting of parameter No. 5029 exceeds the number of compensation values of a path, the least of the numbers of compensation values in all paths is made common.
> 2 When 0 or a negative value is set, memories common to paths are not used.

	#7	#6	#5	#4	#3	#2	#1	#0
5040								OWD

[Input type] Parameter input
[Data type] Bit path

#0 OWD In radius programming (bit 1 (ORC) of parameter No. 5004 is set to 1),

0: Tool offset values of both geometry compensation and wear compensation are specified by radius.

1: Tool offset value of geometry compensation is specified by radius and tool offset value of wear compensation is specified by diameter, for an axis of diameter programming.

NOTE
This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

	#7	#6	#5	#4	#3	#2	#1	#0
5042							OFC	OFA

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OFA
#1 OFC These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFC	OFA	Unit	Valid data range
0	1	0.01mm	±9999.99mm
0	0	0.001mm	±9999.999mm
1	0	0.0001mm	±9999.9999mm

For inch input

OFC	OFA	Unit	Valid data range
0	1	0.001inch	±999.999inch
0	0	0.0001inch	±999.9999inch
1	0	0.00001inch	±999.99999inch

5043	Axis number for which Y-axis offset is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the number of an axis for which the tool offset is corrected.
If 0 or a value beyond the valid data range is set, the Y-axis offset is applied to the Y-axis of the basic three axes. If setting is made for the X- or Z-axis of the basic three axes, the standard tool offset for the X- or Z-axis is not used, and only the Y-axis offset is used.

	#7	#6	#5	#4	#3	#2	#1	#0
5101								FXY

[Input type] Parameter input
[Data type] Bit path

#0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:

0: In case of the Drilling canned cycle:
Z-axis at all times.
In case of the Grinding canned cycle:
- For the T series
  Z-axis at all times.
- For the M series
  G75,G77 command :Y-axis
  G78,G79 command :Z-axis

1: Axis selected by the program

> **NOTE**
> 1 In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 10/11 format.
> 2 When this parameter is 1, the drilling axis determined by plane selection (G17/G18/G19) in the drilling canned cycle in the T series 10/11 format. Therefore, the Y-axis is required to specify G17/G19.

	#7	#6	#5	#4	#3	#2	#1	#0
5106								GFX

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 GFX When grinding canned cycle option is specified, the G71, G72, G73, or G74 command is:

0: A multiple repetitive canned cycle (T series) command.
1: A grinding canned cycle command.

5176	**Grinding axis number in Traverse Grinding Cycle(G71)**
	Grinding axis number in Plunge Grinding Cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

For the Lathe system:
Set the Grinding axis number of Traverse Grinding Cycle(G71).
For the Machining Center system:
Set the Grinding axis number of Plunge Grinding Cycle(G75).

> **NOTE**
> The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5177	Grinding axis number of Traverse direct constant-size Grinding cycle(G72)
	Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

For the Lathe system:
Set the Grinding axis number of Traverse direct constant-size Grinding cycle(G72).
For the Machining Center system:
Set the Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle (G77).

> **NOTE**
> The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5178	Grinding axis number of Oscillation Grinding Cycle(G73)
	Grinding axis number of Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

For the Lathe system:
Set the Grinding axis number of Oscillation Grinding Cycle(G73).
For the Machining Center system:
Set the Grinding axis number of Continuous feed surface grinding cycle(G78).

> **NOTE**
> The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5179	Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74)
	Grinding axis number of Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

For the Lathe system:
Set the Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74).
For the Machining Center system:
Set the Grinding axis number of Intermittent feed surface grinding cycle(G79).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5180	Axis number of dressing axis in Plunge grinding cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Plunge grinding cycle(G75).

NOTE
The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the PS0456 alarm is also issued.

5181	Axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77).

NOTE
The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the PS0456 alarm is also issued.

5182	Axis number of dressing axis in Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Continuous feed surface grinding cycle(G78).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the PS0456 alarm is also issued.

5183	Axis number of dressing axis in Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Intermittent feed surface grinding cycle(G79).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the PS0456 alarm is also issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5200						CRG		G84

[Input type] Parameter input
[Data type] Bit path

#0 G84 Method for specifying rigid tapping:
- 0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
- 1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

#2 CRG Rigid mode when a rigid mode cancel command is specified (G80, G01 group G code, reset, etc.) :
- 0: Canceled after rigid tapping signal RGTAP is set to "0".
- 1: Canceled before rigid tapping signal RGTAP is set to "0".

5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)
5244	Maximum spindle speed in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}

[Valid data range] 0 to 9999
Spindle position coder gear ratio
1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999
Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.
Set the same value for both parameter No.5241 and parameter No.5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200 will be issued. This applies to the M series.

	#7	#6	#5	#4	#3	#2	#1	#0
5400								
	SCR	XSC						

[Input type] Parameter input
[Data type] Bit path

#6 XSC The setting of a scaling magnification (axis-by-axis scaling) is:
0: Disabled.
1: Enabled.

#7 SCR Scaling (G51) magnification unit:
0: 0.00001 times (1/100,000)
1: 0.001 times

	#7	#6	#5	#4	#3	#2	#1	#0
5401								
								SCLx

[Input type] Parameter input
[Data type] Bit axis

#0 SCLx Scaling on this axis:
0: Invalidated
1: Validated

5411	
	Scaling (G51) magnification

[Input type] Setting input
[Data type] 2-word path
[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No.5400)
[Valid data range] 1to999999999
This parameter sets a scaling magnification when axis-by-axis scaling is disabled (with bit 6 (XSC) of parameter No. 5400 set to 0). If no scaling magnification (P) is specified in the program, the setting of this parameter is used as a scaling magnification.

> **NOTE**
> When bit 7 (SCR) of parameter No.5400 is set to 1, the valid data range is 1 to 9999999.

5421	Scaling magnification for each axis

[Input type] Setting input
[Data type] 2-word axis
[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No.5400)
[Valid data range] -999999999 to –1, 1 to 999999999

This parameter sets a scaling magnification for each axis when axis-by-axis scaling is enabled (with bit 6 (XSC) of parameter No. 5400 set to 1). For the first spindle to the third spindle (X-axis to Z-axis), the setting of this parameter is used as a scaling magnification if scaling magnifications (I, J, K) are not specified in the program.

> **NOTE**
> When bit 7 (SCR) of parameter No.5400 is set to 1, the valid data ranges are -9999999 to -1 and 1 to 9999999.

	#7	#6	#5	#4	#3	#2	#1	#0
5431								
								MDL

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 MDL The G60 code (one-direction positioning) is:
0: One-shot G code (group 00).
1: Modal G code (group 01).

5440	Positioning direction and overrun distance in single directional positioning

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the positioning direction and overrun distance in single directional positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.
Overrun distance>0: The positioning direction is positive (+).
Overrun distance<0: The positioning direction is negative (*).
Overrun distance=0: Single directional positioning is not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5450						PLS		

[Input type] Parameter input
[Data type] Bit path

#2 PLS The polar coordinate interpolation shift function is:
0: Not used.
1: Used.
This enables machining using the workpiece coordinate system with a desired point which is not the center of the rotation axis set as the origin of the coordinate system in polar coordinate interpolation.

5460	Axis (linear axis) specification for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of linear axis to execute polar interpolation.

5461	Axis (rotation axis) specification for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of rotation axis to execute polar interpolation.

5463	Automatic override tolerance ratio for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100
Typical setting: 90% (treated as 90% when set to 0)
Set the tolerance ratio of the fastest cutting feedrate to the speed of the rotation axis during automatic override of polar coordinate interpolation.

5464	Compensation for error on hypothetical axis of polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(For IS-B, -999999.999 to +999999.999)
This parameter is used to set the error if the center of the rotation axis on which polar coordinate interpolation is performed is not on the X-axis.
If the setting of the parameter is 0, regular polar coordinate interpolation is performed.

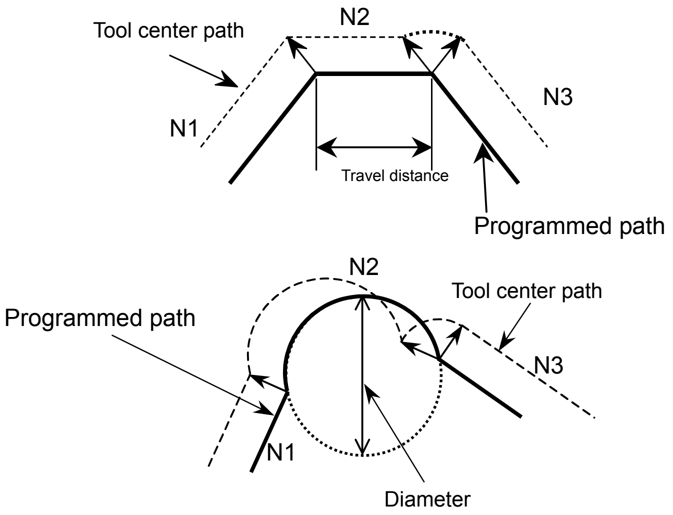
5481	Feedrate of rotation of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/min
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)

This parameter sets the feedrate of the movement along the normal direction controlled axis that is inserted at the start point of a block during normal direction control.

5483	Limit value of movement that is executed at the normal direction angle of a preceding block

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to standard parameter setting table (B)
(For IS-B, -999999.999 to +999999.999)



For straight line
When the travel distance of N2 in the figure on the left does not exceed the setting, block N2 is machined with the tool being normal to block N1.

For arc
When the arc diameter of N2 in the figure on the left does not exceed the setting, arc N2 is machined with the tool being normal to block N1. A normal direction axis is not controlled to move in the normal direction according to the arc movement.

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM	HGO			MGO	G67
			SBM	HGO	V10		MGO	G67

[Input type] Parameter input
[Data type] Bit path

#0 G67 If the macro modal call cancel command (G67) is specified when the macro modal call mode (G66) is not set:
0: Alarm PS0122 is issued.
1: The specification of G67 is ignored.

#1 MGO When a GOTO statement for specifying custom macro control is executed, a high-speed branch to 20 sequence numbers executed from the start of the program is:
0: A high-speed branch is not caused to n sequence numbers from the start of the executed program.
1: A high-speed branch is caused to n sequence numbers from the start of the program.

 # 3 V10 As system variable numbers for tool offset:
0 : The standard system variable numbers for the Series 0 are used.
1 : The same system variable numbers as those used for the Series 10/11 are used.
The tables below indicate the system variables for tool offset numbers 1 to 400. The values for tool offset numbers 1 to 200 can be read from or assigned to the system variables in parentheses.

(1) Tool offset memory A

	System variable number	
	V10 = 0	**V10 = 1**
Wear offset value	#10001 to #10400 (#2001 to #2200)	#10001 to #10400 (#2001 to #2200)

(2) Tool offset memory C

		System variable number	
		V10 = 0	**V10 = 1**
Tool length offset	Wear offset value	#11001 to #11400 (#2201 to #2400)	#10001 to #10400 (#2001 to #2200)
	Geometry offset value	#10001 to #10400 (#2001 to #2200)	#11001 to #11400 (#2201 to #2400)
Tool radius offset	Wear offset value	#13001 to #13400	#12001 to #12400
	Geometry offset value	#12001 to #12400	#13001 to #13400

#4 HGO When a GOTO statement in a custom macro control command is executed, a high-speed branch to the 30 sequence numbers immediately before the executed statement is:

0: Not made.
1: Made.

#5 SBM Custom macro statement

0: Not stop the single block
1: Stops the single block

If you want to disable the single blocks in custom macro statements using system variable #3003, set this parameter to 0. If this parameter is set to 1, the single blocks in custom macro statements cannot be disabled using system variable #3003. To control single blocks in custom macro statements using system variable #3003, use bit 7 (SBV) of parameter No. 6000.

	#7	#6	#5	#4	#3	#2	#1	#0
6001		**CCV**	**TCS**	**CRO**	**PV5**		**PRT**	**MIF**

[Input type] Parameter input
[Data type] Bit path

#0 MIF The custom macro interface signals are based on:

0: Standard specification.
(The signals UI000 to UI015, UO000 to UO015, and UO100 to UO131 are used.)
1: Extended specification.
(The signals UI000 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO000 to UO031, UO100 to UO131, UO200 to UO231, and UO300 to UO331 are used.)

#1 PRT Reading zero when data is output using a DPRINT command

0: Outputs a space
1: Outputs no data

#3 PV5 Custom macro common variables:

0: #500 to #999 are output.
1: #100 to #199 and #500 to 999 are output.

#4 CRO ISO code in BPRWT or DPRNT command

0: Outputs only “LF” after data is output
1: Outputs “LF” and “CR” after data is output

#5 TCS Custom macro (subprogram)
0: Not called using a T code
1: Called using a T code

#6 CCV Common variables #100 to #199 cleared by power-off are:
0: Cleared to <null> by reset
1: Not cleared by reset

	#7	#6	#5	#4	#3	#2	#1	#0
6003			MSB	MPR	TSE	MIN		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#2 MIN Custom macro interrupt
0: Performed by interrupting an in-execution block (Custom macro interrupt type I)
1: Performed after an in-execution block is completed (Custom macro interrupt type II)

#3 TSE Custom macro interrupt signal UINT
0: Edge trigger method (Rising edge)
1: Status trigger method

#4 MPR Custom macro interrupt valid/invalid M code
0: M96/M97
1: M code set using parameters (Nos. 6033 and 6034)

#5 MSB Interrupt program
0: Uses a dedicated local variable (Macro-type interrupt)
1: Uses the same local variable as in the main program (Subprogram- type interrupt)

	#7	#6	#5	#4	#3	#2	#1	#0
6004						VHD		NAT
			D10					NAT

[Input type] Parameter input
[Data type] Bit path

#0 NAT The results of the custom macro functions ATAN (with 2 arguments) and ASIN are specified as follows:
0: The result of ATAN is 0 to 360.0.
The result of ASIN is 270.0 to 0 to 90.0.
1: The result of ATAN is -180.0 to 0 to 180.0.
The result of ASIN is -90.0 to 0 to 90.0.

#2 VHD With system variables #5121 to #5125:
0: The tool offset value (geometry offset value) in the block currently being executed is read. (This parameter is valid only when tool geometry/tool wear compensation memories are available (bit 6 (NGW) of parameter No. 8136 is 0)).
1: An interrupt travel distance based on manual handle interrupt is read.

#5 D10 When tool compensation memory C is used, for reading or writing tool offset values (for up to offset number 200) for D code (tool radius), the same system variables, #2401 through #2800, as Series 10/11 are:
0: Not used.
1: Used.
When bit 3 (V10) of parameter No. 6000 is set to 1

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]]	#2601	[#_OFSDW[1]]
2	#2402	[#_OFSDG[2]]	#2602	[#_OFSDW[2]]
3	#2403	[#_OFSDG[3]]	#2603	[#_OFSDW[3]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]]	#2799	[#_OFSDW[199]]
200	#2600	[#_OFSDG[200]]	#2800	[#_OFSDW[200]]

	#7	#6	#5	#4	#3	#2	#1	#0
6005								SQC

[Input type] Parameter input
[Data type] Bit path

#0 SQC In the subprogram call function, a subprogram sequence number call is:
0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
6007				CVA				

[Input type] Parameter input
[Data type] Bit path

#4 CVA The format for macro call arguments is specified as follows:
0: Arguments are passed in NC format without modifications.
1: Arguments are converted to macro format then passed.
Example)
When G65 P_ X10 ; is specified, the value in local variable #24 in the calling program is set as follows:

Command	CVA=0	CVA=1
#24	0.01	0.01
ADP[#24]	10.0	0.01

NOTE
External operations are the same unless the ADP function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
6008	IJK	GMP		ISO			MCA	F0C

[Input type] Parameter input
[Data type] Bit path

#0 F0C The precision of operation is based on:
0: New specification.
1: FS0*i*-C compatible specification.

> **NOTE**
> For details, refer to the custom macro chapter in the Operator's Manual (B-64304EN).

#1 MCA A macro alarm specification based on system variable #3000 is selected as follows:

0: An alarm number obtained by adding 3000 to a value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 200 can be assigned to variable #3000.)

1: A value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 4095 can be assigned to variable #3000.)

(Example)

Execution of #3000=1 (ALARM MESSAGE);

When bit 1 (MCA) of parameter No. 6008 is set to 0:

The alarm screen displays "MC 3001 ALARM MESSAGE".

When bit 1 (MCA) of parameter No. 6008 is set to 1:

The alarm screen displays "MC0001 ALARM MESSAGE".

#4 ISO

0: When the EIA code is used, the bit patterns of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameter No. 6010 to No. 6018.

1: When the ISO/ASCII code is used, the bit patters of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameter No. 6010 to No. 6018.

#6 GMP The calling of M, T, or a particular code during the calling of a G code, and the calling of a G code during the calling of M, T, or particular code are:

0: Not allowed. (They are executed as an ordinary G, M, T, and NC address.)

1: Allowed.

#7 IJK For addresses I, J, and K specified as arguments:

0: Argument specification I or II is automatically determined.

1: Argument specification I is always used.

> **Example**
> When K_J_I_ is specified:
> - When this parameter is set to 0:
>   Argument specification II is used and K=#6, J=#8, and I=#10 are specified.
> - When this parameter is set to1:
>   Argument specification I is used and I=#4, J=#5, and K=#6 are specified regardless of the specification order.
>   (Argument specification II cannot be used.)

	#7	#6	#5	#4	#3	#2	#1	#0
6010	*7	*6	*5	*4	*3	*2	*1	*0
6011	=7	=6	=5	=4	=3	=2	=1	=0
6012	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0
6014	]7	]6	]5	]4	]3	]2	]1	]0
6015	?7	?6	?5	?4	?3	?2	?1	?0
6016	@7	@6	@5	@4	@3	@2	@1	@0
6017	&7	&6	&5	&4	&3	&2	&1	&0
6018	_7	_6	_5	_4	_3	_2	_1	_0

[Input type] Parameter input
[Data type] Bit path

*0 to *7 : The bit pattern of the EIA or ISO/ASCII code indicating * is set.
=0 to =7 : The bit pattern of the EIA or ISO/ASCII code indicating = is set.
#0 to #7 : The bit pattern of the EIA or ISO/ASCII code indicating # is set.
[0 to [7 : The bit pattern of the EIA or ISO/ASCII code indicating [is set.
]0 to]7 : The bit pattern of the EIA or ISO/ASCII code indicating] is set.
?0 to ?7 : The bit pattern of the EIA or ISO/ASCII code indicating ? is set.
@0 to @7 : The bit pattern of the EIA or ISO/ASCII code indicating @ is set.
&0 to &7 : The bit pattern of the EIA or ISO/ASCII code indicating & is set.
_0 to _7 : The bit pattern of the EIA or ISO/ASCII code indicating _ is set.
0: A corresponding bit is 0.
1: A corresponding bit is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
6019								MCO

[Input type] Parameter input
[Data type] Bit

#0 MCO When data is output, the decimal number value of the macro variable data is
0: Not output as a comment.
1: Output at the same time as a comment.

After the number, data, and the variable name of the macro variable are output when data output operation is performed the variable number and the value of the macro variable data in decimal number are output as a comment.

NOTE
1 Output data by this parameter is "Comment", and this is ignored at the time of reading.

> **NOTE**
> 2 Accuracy of the output data of the comment is up to 15 digits. The range of output data are nine digits above decimal point and eight digits below decimal point. "± OVER FLOW" is output instead of a value when the total digits number is more than 16 and the digit number above the decimal point is ten or more. When the number of digits below the decimal point becomes nine digits or more, the ninth place of the decimal point is rounded off and output. Moreover, the seventh place or the eighth place of the decimal point is rounded off and output when the total digits number is more than 16 and the digit number above decimal point is nine or eight.
> 3 The output becomes "EMPTY" when displayed, the macro variable data is "DATA EMPTY".

6030	M code to execute external device subprogram calls

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 99999999

Set the M code to execute external device subprogram calls. When 0 is set, M198 is used. M01, M02, M30, M98, and M99 cannot be used to execute external device subprogram calls. When a negative number, 1, 2, 30, 98, or 99 is set for this parameter, M198 is used to execute external device subprogram calls.

6031	Start number of common variables to be protected among the common variables (#500 to #999)

6032	End number of common variables to be protected among the common variables (#500 to #999)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 500 to 999

Among the common variables (#500 to #999), the range of common variables specified by this parameter can be protected (by setting their attributes to read-only). If a write attempt (on the left side) is made, an alarm is issued.

> **NOTE**
> Set 0 in both parameter No. 6031 and No. 6032 not to protect common variables.

6033	M code that validates a custom macro interrupt

6034	M code that invalidates a custom macro interrupt

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)

These parameters can be used when bit 4 (MPR) of parameter No.6003, is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPR is 0, irrespective of the state of this parameter.

6036	Number of custom macro variables common to tool path (for #100 to #199)

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 100

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #100 to #199 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 20 is set in parameter No. 6036
#100 to #119: Shared by all paths
#120 to #199: Used by each path independently

NOTE
When 0 or a negative value is set, the memory common to paths is not used.

6037	Number of custom macro variables common to tool path (for #500 to #999)

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 500

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #500 to #999 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 50 is set in parameter No. 6037
#500 to #549: Shared by all paths
#550 to #999: Used by each path independently

NOTE
When 0 or a negative value is set, the memory common to paths is not used.

6038	Start G code used to call a custom macro

[Input type] Parameter input
[Data type] Word path
[Valid data range] -9999 to 9999

6039	Start program number of a custom macro called by G code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6040	Number of G codes used to call custom macros

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255

Set this parameter to define multiple custom macro calls using G codes at a time. With G codes as many as the value set in parameter No. 6040 starting with the G code set in parameter No. 6038, the custom macros of program numbers as many as the value set in parameter No. 6040 starting with the program number set in parameter No. 6039 can be called. Set 0 in parameter No. 6040 to disable this mode of calling.
If a negative value is set in parameter No. 6038, the modal call mode is entered.
Example)
When parameter No. 6038 = 900, parameter No. 6039 = 1000, and parameter No. 6040 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:
G900 → O1000
G901 → O1001
G902 → O1002
:
G999 → O1099
When the setting of parameter No. 6038 is changed to -900, the same set of custom macro calls (modal calls) is defined.

NOTE
1 When the following conditions are satisfied, all calls using these parameters are disabled:
  1) When a value not within the specifiable range is set in each parameter
  2) (Value of parameter No.6039 + value of parameter No.6040 - 1) > 9999
2 The specification of a mixture of simple calls and modal calls is not allowed.
3 If a range of G codes set by these parameters duplicate G codes specified in parameter No.6050 to No.6059, the calls specified by parameter No.6050 to 6059 are made preferentially.

6044	Start M code used to call a subprogram

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6045	Start program number of a subprogram called by M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6046	Number of M codes used to call subprograms (number of subprograms called by M codes)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767

Set this parameter to define multiple subprogram calls using M codes at a time. With M codes as many as the value set in parameter No. 6046 starting with the M code set in parameter No. 6044, the subprograms of program numbers as many as the value set in parameter No. 6046 starting with the program number set in 6045 can be called. Set 0 in parameter No. 6046 to disable this mode of calling.

Example)

When parameter No. 6044 = 80000000, parameter No. 6045 = 3000, and parameter No. 6046 = 100 are set, a set of 100 subprogram calls is defined as follows:

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

NOTE

1 When the following conditions are satisfied, all calls using these parameters are disabled:
  1) When a value not within the specifiable range is set in each parameter
  2) (Value of parameter No. 6045 + value of parameter No. 6046 - 1) > 9999

2 If a range of M codes set by these parameters duplicate M codes specified in parameter No. 6071 to No. 6079, the calls specified by parameter No. 6071 to 6079 are made preferentially.

6047	Start M code used to call a custom macro

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6048	Start program number of a custom macro called by M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6049	Number of M codes used to call custom macros (number of custom macros called by M codes)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767

Set this parameter to define multiple custom macro calls using M codes at a time. With M codes as many as the value set in parameter No. 6049 starting with the M code set in parameter No. 6047, the custom macros of program numbers as many as the value set in parameter No. 6049 starting with the program number set in parameter No. 6048 can be called. Set 0 in parameter No. 6049 to disable this mode of calling.

Example)

When parameter No. 6047 = 90000000, parameter No. 6048 = 4000, and parameter No. 6049 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:

M90000000 → O4000

M90000001 → O4001
M90000002 → O4002
:
M90000099 → O4099

> **NOTE**
> 1 When the following conditions are satisfied, all calls using these parameters are disabled:
> 1) When a value not within the specifiable range is set in each parameter
> 2) (Value of parameter No. 6048 + value of parameter No. 6049 - 1) > 9999
>
> 2 If a range of M codes set by these parameters duplicate M codes specified in parameter No. 6080 through No. 6089, the calls specified by parameter No. 6080 through 6089 are made preferentially.

6050	G code that calls the custom macro of program number 9010
6051	G code that calls the custom macro of program number 9011
6052	G code that calls the custom macro of program number 9012
6053	G code that calls the custom macro of program number 9013
6054	G code that calls the custom macro of program number 9014
6055	G code that calls the custom macro of program number 9015
6056	G code that calls the custom macro of program number 9016
6057	G code that calls the custom macro of program number 9017
6058	G code that calls the custom macro of program number 9018
6059	G code that calls the custom macro of program number 9019

[Input type] Parameter input
[Data type] Word path
[Valid data range] (-9999 to 9999 : excluding 0, 5, 65, 66 and 67)

Set the G codes used to call the custom macros of program numbers 9010 to 9019. However, note that when a negative value is set in this parameter, it becomes a modal call. For example, if this parameter is set to -11, the modal call mode is entered by G11.

6071	M code used to call the subprogram of program number 9001
6072	M code used to call the subprogram of program number 9002
6073	M code used to call the subprogram of program number 9003
6074	M code used to call the subprogram of program number 9004

6075	M code used to call the subprogram of program number 9005

6076	M code used to call the subprogram of program number 9006

6077	M code used to call the subprogram of program number 9007

6078	M code used to call the subprogram of program number 9008

6079	M code used to call the subprogram of program number 9009

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)
These parameters set the M codes that call the subprograms of program numbers 9001 to 9009.

> **NOTE**
> If the same M code is set in these parameters, the younger number is called preferentially. For example, if 100 is set in parameter No. 6071 and 6072, and programs O9001 and O9002 both exist, O9001 is called when M100 is specified.

6080	M code used to call the custom macro of program number 9020

6081	M code used to call the custom macro of program number 9021

6082	M code used to call the custom macro of program number 9022

6083	M code used to call the custom macro of program number 9023

6084	M code used to call the custom macro of program number 9024

6085	M code used to call the custom macro of program number 9025

6086	M code used to call the custom macro of program number 9026

6087	M code used to call the custom macro of program number 9027

6088	M code used to call the custom macro of program number 9028

6089	M code used to call the custom macro of program number 9029

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)
Set the M codes used to call the custom macros of program numbers 9020 to 9029. The simple call mode is set.

> **NOTE**
> 1 If the same M code is set in these parameters, the younger number is called preferentially. For example, if 200 is set in parameter No. 6081 and No. 6082, and programs O9021 and O9022 both exist, O9021 is called when M200 is specified.
> 2 If the same M code is set in a parameter (No. 6071 to No. 6079) used to call subprograms and in a parameter (No. 6080 to No. 6089) used to call custom macros, a custom macro is called preferentially. For example, if 300 is set in parameter No. 6071 and No. 6081, and programs O9001 and O9021 both exist, O9021 is called when M300 is specified.

6090	ASCII code that calls the subprogram of program number 9004

6091	ASCII code that calls the subprogram of program number 9005

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65(A:41H) to 90(Z:5AH)

These parameters set the ASCII codes that call subprograms in decimal.
The settable addresses are indicated below.

Address	Parameter setting value	T series	M series
A	65	O	O
B	66	O	O
D	68	X	O
F	70	O	O
H	72	O	O
I	73	O	O
J	74	O	O
K	75	O	O
L	76	O	O
M	77	O	O
P	80	O	O
Q	81	O	O
R	82	O	O
S	83	O	O
T	84	O	O
V	86	X	O
X	88	X	O
Y	89	X	O
Z	90	X	O

> **NOTE**
> 1 When address L is set, the number of repeats cannot be specified.
> 2 Set 0 when no subprogram is called.

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF	SRE	SLS	HSS			SK0	GSK

[Input type] Parameter input
[Data type] Bit path

#0 GSK As a skip signal, the skip signal SKIPP is:
0: Invalid.
1: Valid.

#1 SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP and the multistage skip signals SKIP2 to SKIP8.
0: Skip signal is valid when these signals are 1.
1: Skip signal is valid when these signals are 0.

#4 HSS
0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

#5 SLS
0: The multi-step skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The multi-step skip function uses high-speed skip signals while skip signals are input.

> **NOTE**
> The skip signals (SKIP and SKIP2 to SKIP8) are valid regardless of the setting of this parameter. They can also be disabled using bit 4 (IGX) of parameter No. 6201.

#6 SRE When a high-speed skip signal is used:
0: The signal is assumed to be input on the rising edge (contact open → close).
1: The signal is assumed to be input on the falling edge (contact close → open).

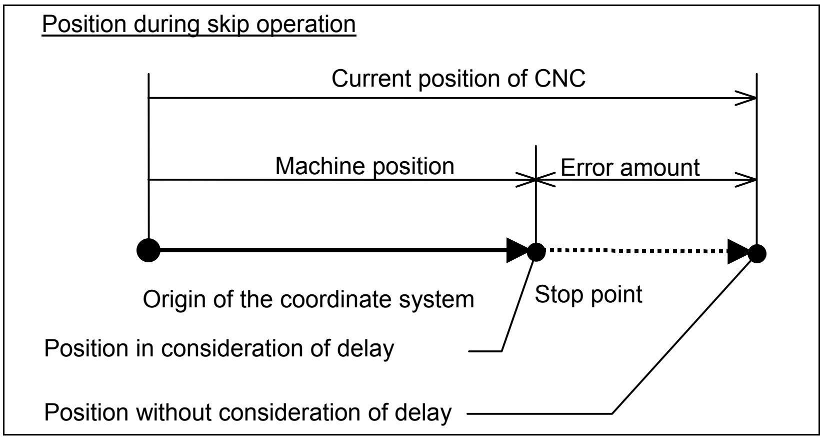
#7 SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6201	SPE			IGX		TSE	SEB	

[Input type] Parameter input
[Data type] Bit path

#1 SEB When a skip signal or measurement position arrival signal goes on while the skip function, or the automatic tool length measurement (M series) or automatic tool compensation (T series) is used, the accumulated pulses and positional deviation due to acceleration/deceleration are:
0: Ignored.
1: Considered and compensated.
The accumulated pulses and positional deviation due to actual acceleration/deceleration when the skip signal or measurement position arrival signal goes on are considered to obtain the position at which the signal is input.

#2 TSE In a skip by the torque limit skip command (G31P98/P99):
0: A servo delay amount (positional deviation) is considered (system variables #5061 to #5065 store positions corrected in consideration of the servo system delay amount).
1: A servo delay amount (positional deviation) is not considered (system variables #5061 to #5065 store positions corrected without consideration of the servo system delay amount).



#4 IGX When the high-speed skip function is used, SKIP, SKIPP, and SKIP2 to SKIP8 are:
0: Enabled as skip signals.
1: Disabled as skip signals.

#7 SPE For the skip function (G31), the skip signal SKIP is:
0: Enabled.
1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	IGX (No.6201 #4)	GSK (No.6200 #0)	SPE (No.6201 #7)	Skip signal SKIPP	Skip signal SKIP	Multistage skip signals SKIP2-SKIP8
Setting	0	0	0	Disabled	Enabled	Enabled
	0	1	0	Enabled	Enabled	Enabled
	0	0	1	Disabled	Disabled	Enabled
	0	1	1	Enabled	Disabled	Enabled
	1	0	0	Disabled	Disabled	Disabled
	1	1	0	Disabled	Disabled	Disabled
	1	0	1	Disabled	Disabled	Disabled
	1	1	1	Disabled	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multistage skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).
To use multistage skip signals, the multistage skip function option is required.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1

[Input type] Parameter input
[Data type] Bit path

1S1 to 1S8 These parameters specify whether to enable or disable each high-speed skip signal when the G31 skip command is issued.
The following table shows the correspondence between the bits, input signals, and commands.
The settings of the bits have the following meaning :
0: The high-speed skip signal corresponding to a bit is disabled.
1: The high-speed skip signal corresponding to a bit is enabled.

Parameter	High-speed skip signals
1S1	HDI0
1S2	HDI1
1S3	HDI2
1S4	HDI3

NOTE
Do not specify the same signal simultaneously for different paths.

	#7	#6	#5	#4	#3	#2	#1	#0
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1
6206	DS8	DS7	DS6	DS5	DS4	DS3	DS2	DS1

[Input type] Parameter input
[Data type] Bit path

1S1to1S8, 2S1to2S8, 3S1to3S8, 4S1to4S8, DS1toDS8

Specify which skip signal is enabled when the skip command (G31, or G31P1 to G31P4) and the dwell command (G04, G04Q1 to G04Q4) are issued with the multi-step skip function.
The following table shows the correspondence between the bits, input signals, and commands.
The setting of the bits have the following meaning :
0: The skip signal corresponding to a bit is invalid.
1: The skip signal corresponding to a bit is enabled.

Multi-step skip function					
Command / Input signal	G31 G31P1 G04Q1	G31P2 G04Q2	G31P3 G04Q3	G31P4 G04Q4	G04
SKIP/HDI0	1S1	2S1	3S1	4S1	DS1
SKIP2/HDI1	1S2	2S2	3S2	4S2	DS2
SKIP3/HDI2	1S3	2S3	3S3	4S3	DS3
SKIP4/HDI3	1S4	2S4	3S4	4S4	DS4
SKIP5	1S5	2S5	3S5	4S5	DS5
SKIP6	1S6	2S6	3S6	4S6	DS6
SKIP7	1S7	2S7	3S7	4S7	DS7
SKIP8	1S8	2S8	3S8	4S8	DS8

NOTE
HDI0 to HDI3 are high-speed skip signals. Do not specify the same signal simultaneously for different paths.

When bit 0 (GSK) of parameter No. 6200 is set to 1, commands to be skipped can be selected by setting the following parameter:

Commands skipped by SKIPP signal <G006.6>

Parameter	Command skipped
When bit 0 (1S1) of parameter No. 6202 is set to 1	G31P1,G04Q1
When bit 0 (2S1) of parameter No. 6203 is set to 1	G31P2,G04Q2

Commands skipped by SKIPP signal <G006.6>

Parameter	Command skipped
When bit 0 (3S1) of parameter No. 6204 is set to 1	G31P3,G04Q3
When bit 0 (4S1) of parameter No. 6205 is set to 1	G31P4,G04Q4
When bit 6 (DS1) of parameter No. 6206 is set to 1	G04,G04Q1,G04Q2,G04Q3,G04Q4

	#7	#6	#5	#4	#3	#2	#1	#0
6207						SFN	SFP	

[Input type] Parameter input
[Data type] Bit path

#1 SFP The feedrate used when the skip function (G31) is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in parameter No. 6281.

> **NOTE**
> For the multi-stage skip function and high-speed skip, see the description of bit 2 (SFN) of parameter No. 6207.

#2 SFN The feedrate used when the skip function based on high-speed skip signals (with bit 4 (HSS) of parameter No. 6200 set to 1) or the multi-skip function is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in a parameter from parameter No. 6282 to No. 6285.

> **NOTE**
> For not the multistage skip function, but the skip function using no high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 0), see the description of bit 1 (SFP) of parameter No. 6207.

6221	Torque limit dead zone time for a torque limit skip command

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 65535

The torque limit skip arrival signal is ignored for a set period of time.
If G31P98 is specified, skip operation is not performed for a set period of time after the torque limit skip arrival signal is set to 1.
If G31P99 is specified, skip operation is not performed for a set period of time after the torque limit skip arrival signal is set to 1.
However, if a skip signal is input, skip operation is performed, regardless of the period of time set in this parameter.

6254	ε value on the X axis during automatic tool compensation (T series)
	ε value during automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)

6255	ε value on the Z axis during automatic tool compensation (T series)
	ε value during automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)

6256	
	ε value during automatic tool length measurement (M series) (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the relevant ε value during automatic tool compensation (T series) or automatic tool length measurement (M series).

NOTE
1 For the M series, when the setting of parameter No. 6252 or 6253 is 0, the setting of parameter No. 6251 is used.
2 Set a radius value regardless of whether diameter or radius programming is specified.

6281	Feedrate for the skip function (G31)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a feedrate for the skip function (G31). This parameter is valid when bit 1 (SFP) of parameter No. 6207 is set to 1.

NOTE
For the multi-stage skip function and high-speed skip, see the description of parameter No. 6282 to No. 6285.

6282	Feedrate for the skip function (G31, G31 P1)
6283	Feedrate for the skip function (G31 P2)
6284	Feedrate for the skip function (G31 P3)
6285	Feedrate for the skip function (G31 P4)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Each of these parameters sets a feedrate for each skip function G code. These parameters are valid when bit 2 (SFN) of parameter No. 6207 is set to 1.

6287	Positional deviation limit in torque limit skip

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 327670

This parameter sets a positional deviation limit for each axis imposed when torque limit skip is specified. When the actual positional deviation exceeds the positional deviation limit, the alarm (SV0004) is issued and an immediate stop takes place.

	#7	#6	#5	#4	#3	#2	#1	#0
6400	MG4	MGO	RVN	HMP	MC8	MC5	FWD	RPO
	MG4	MGO	RVN		MC8	MC5	FWD	RPO

[Input type] Parameter input
[Data type] Bit path

#0 RPO With the manual handle retrace function, the rapid traverse rate is clamped, assuming that:
0: An override of 10% is used.
1: An override of 100% is used.

#1 FWD With the manual handle retrace function, program execution can be performed:
0: In both forward and backward directions.
1: In the forward direction only. Execution in the backward direction is not permitted.

#2 MC5
#3 MC8

These parameters set the number of M code groups and the number of M codes per group. (See explanations of parameters Nos. 6411 to 6490.)

MC5	MC8	M code group setting
0	0	Standard (20 groups of four)
1	0	16 groups of five
0	1	10 groups of eight

When 16 groups of five are used, the meanings of parameters are changed as follows:
Group A No.6411(1) to No.6415(5)
Group B No.6416(1) to No.6420(5)
:
Group P No.6486(1) to No.6490(5)
When 10 groups of eight are used, they are changed as follows:
Group A No.6411(1) to No.6418(8)
Group B No.6419(1) to No.6426(8)
:
Group J No.6483(1) to No.6490(8)

#4 HMP When inversion or backward movement is inhibited in other paths:
0: Inversion or backward movement is not inhibited for the currently executed path.
1: Inversion or backward movement is inhibited also for the currently executed path.

#5 RVN When the manual handle retrace function is used, M codes other than grouped M codes:
0: Do not disable backward movement.
1: Disable backward movement.

When this parameter is set to 1, M codes other than grouped M codes disable backward movement in general. Exceptionally, however, the following M codes allow backward movement:
1. Subprogram call based on M98/M99
2. Subprogram call based on an M code
3. Macro call based on an M code
4. Waiting M code
5. M0

#6 MGO When the manual handle retrace function is used, handle pulses during execution of a G code related to measurement are:
0: Valid.
1: Invalid. A speed with an override of 100% is used for execution at all times.

#7 MG4 In the manual handle retrace function, for blocks for which multi-step skip G04 is enabled (when the multi-step skip software option is used, and the settings of parameter Nos. 6202 to 6206 are valid):
0: Backward movement is not prohibited.
1: Backward movement is prohibited.

	#7	#6	#5	#4	#3	#2	#1	#0
6401	**STO**	**HST**				**CHS**		

[Input type] Parameter input
[Data type] Bit path

#2 CHS In manual handle retrace:
0: The status is displayed if the following conditions are all satisfied:
(1) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.
(2) Check mode output signal MMMOD<Fn091.3> is set to 1.
1: The status is displayed if the following conditions are all satisfied:
(1) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.
(2) Cycle start lamp signal STL<Fn000.5> is set to 1.
(3) Check mode input signal MMOD<Gn067.2> is set to 1.
(4) Handle input signal MCHK<Gn067.3> is set to 1 in the check mode.

#6 HST When the manual handle retrace function is used, the time display field on the status display line of the CNC screen:
0: Does not display status.
1: Displays status.

#7 STO In the manual handle retrace function, the timing for outputting an S code and T code during backward movement is:
0: Different from the timing during forward movement:
1: The same as during forward movement.

6405	Override value (equivalence) for clamping the rapid traverse rate used with the manual handle retrace function

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 0 to 100

This parameter sets an override value (equivalence) for clamping the rapid traverse rate used with the manual handle retrace function. If a value greater than 100 is set in parameter (No.6405), the rapid traverse rate is clamped to an override of 100%. This function is invalid if 0 is set in parameter (No.6405). In this case, the setting of bit 0 (RPO) of parameter No. 6400 is used.

6410	Travel distance per pulse generated from the manual pulse generator

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 0 to 100

Set the travel distance per pulse generated from the manual pulse generator in terms of the override value.

The distance traveled by the machine when the manual handle is actually turned can be found by the following expression:

[Specified speed] × [Handle magnification] × ([Setting of this parameter]/100) × (8/60000) (mm or inch)

[Example]

When a specified feedrate is 30mm/min, the manual handle magnification is 100, and parameter No. 6410 is set to 1, the travel distance per pulse generated from the manual pulse generator is calculated as follows:

[Travel distance per pulse]

=30[mm/min] × 100 × (1/100) × (8/60000)[min]= 0.004mm

6411	M code of group A in manual handle retrace (1)
to	to
6414	M code of group A in manual handle retrace (4)

6415	M code of group B in manual handle retrace (1)
to	to
6418	M code of group B in manual handle retrace (4)

6419	M code of group C in manual handle retrace (1)
to	to
6422	M code of group C in manual handle retrace (4)

6423	M code of group D in manual handle retrace (1)
to	to
6426	M code of group D in manual handle retrace (4)

6427	M code of group E in manual handle retrace (1)
to	to
6430	M code of group E in manual handle retrace (4)

6431	M code of group F in manual handle retrace (1)
to	to
6434	M code of group F in manual handle retrace (4)
6435	M code of group G in manual handle retrace (1)
to	to
6438	M code of group G in manual handle retrace (4)
6439	M code of group H in manual handle retrace (1)
to	to
6442	M code of group H in manual handle retrace (4)
6443	M code of group I in manual handle retrace (1)
to	to
6446	M code of group I in manual handle retrace (4)
6447	M code of group J in manual handle retrace (1)
to	to
6450	M code of group J in manual handle retrace (4)
6451	M code of group K in manual handle retrace (1)
to	to
6454	M code of group K in manual handle retrace (4)
6455	M code of group L in manual handle retrace (1)
to	to
6458	M code of group L in manual handle retrace (4)
6459	M code of group M in manual handle retrace (1)
to	to
6462	M code of group M in manual handle retrace (4)
6463	M code of group N in manual handle retrace (1)
to	to
6466	M code of group N in manual handle retrace (4)
6467	M code of group O in manual handle retrace (1)
to	to
6470	M code of group O in manual handle retrace (4)
6471	M code of group P in manual handle retrace (1)
to	to
6474	M code of group P in manual handle retrace (4)
6475	M code of group O in manual handle retrace (1)
to	to
6478	M code of group Q in manual handle retrace (4)
6479	M code of group R in manual handle retrace (1)
to	to
6482	M code of group R in manual handle retrace (4)
6483	M code of group S in manual handle retrace (1)
to	to
6486	M code of group S in manual handle retrace (4)

6487	M code of group T in manual handle retrace (1)
to	to
6490	M code of group T in manual handle retrace (4)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999

Set a group of M codes output during backward movement.
For backward movement for an M code, the modal M code in the same group set by the parameter is output.
The first M code in each group is set as the default.
When the number of M codes in a group is 3 or less, set the parameter corresponding to an unused M code to 0.
For backward movement for "M0", "M0" is output regardless of which M code is set for the parameter. "0" set in the parameter is ignored.
For an M code which is not set in any group by any of the above parameters, the M code for forward movement is output.
With these parameters, an M code in the same group can be output in backward movement only when the M code is the first M code in each block. When a block contains two or more M codes, the same M codes as output in forward movement are output as a second M code and up.

> **NOTE**
> The above explanation of M code groups applies to the standard settings. The number of M codes in each group and the number of M code groups vary depending on the settings of bit 2 (MC5) and bit 3 (MC8) of parameter No. 6400.

	#7	#6	#5	#4	#3	#2	#1	#0
6500					DPA		SPC	

[Input type] Parameter input
[Data type] Bit

#1 SPC Graphic display in 2-path control includes:
0: Two spindles and two tool posts.
1: One spindle and two tool posts.

> **NOTE**
> This parameter is valid when two paths are displayed at the same time.

#3 DPA The current position display on the graphic display screen displays:
0: The actual position with tool-nose radius compensation considered.
1: The programmed position.

	#7	#6	#5	#4	#3	#2	#1	#0
6501								
			CSR			3PL		ORG

[Input type] Parameter input
[Data type] Bit path

#0 ORG When the coordinate system is changed during tool path drawing by the dynamic graphic display function, drawing is performed:
0: With the same coordinate system.
1: With the current drawing point assumed to be the current position set in the new coordinate system.

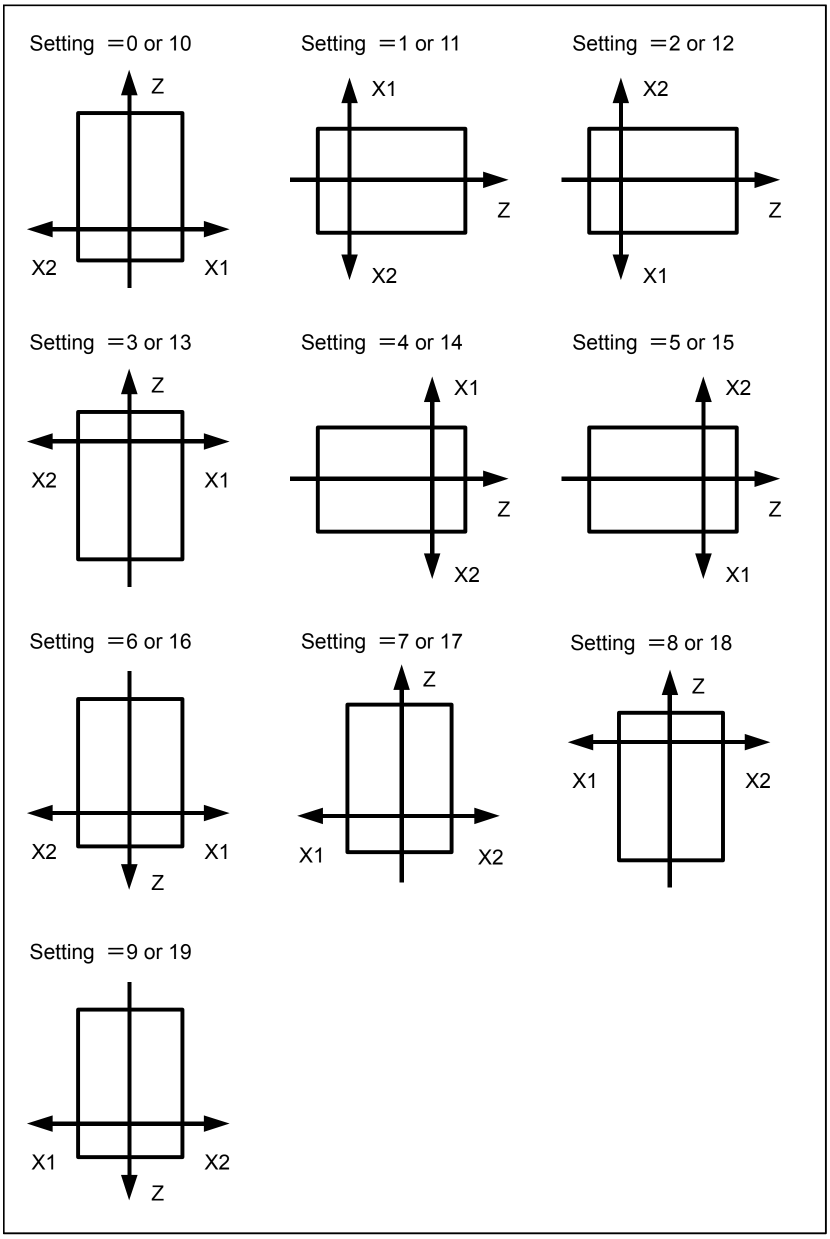
> **NOTE**
> This parameter is valid when bit 3 (BGM) of parameter No. 11329 is 0.

#2 3PL In animated simulation of the dynamic graphic display function, triplane drawing is drawn:
0: In third angle projection.
1: In first angle projection.

#5 CSR On the PATH GRAPHIC (CURRENT POSITION) screen, the shape of the cursor indicating the tool position is:
0: A square (■).
1: An x (×).

6509	Drawing coordinate system for one-spindle graphic (2-path control)

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 9, 10 to 19 (however, a setting of 0 to 9 is the same as that of 10 to 19, respectively.)
This parameter sets the drawing coordinate system for one-spindle graphic (bit 1 (SPC) of parameter No. 6500 is 1) in 2-path control.
The drawing coordinate system is set as shown below.

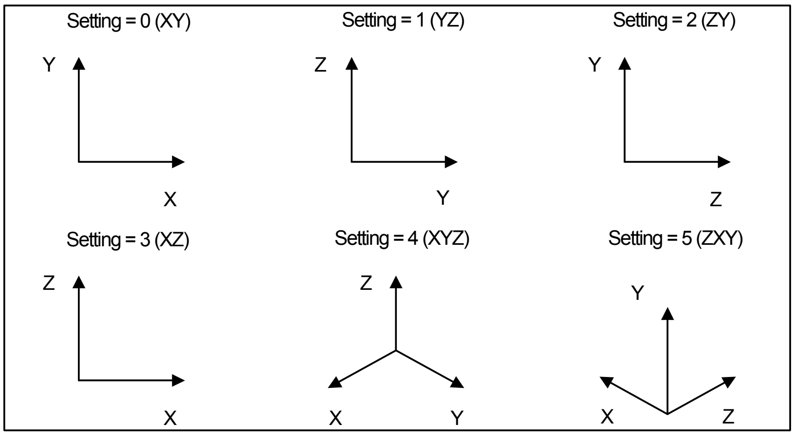


6510	Drawing coordinate system

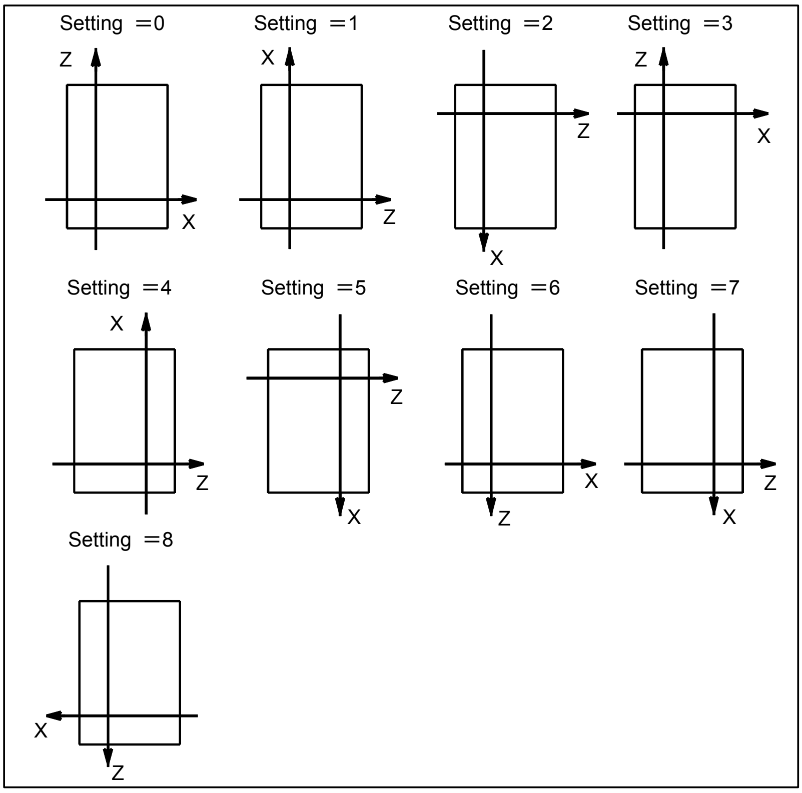
[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 8

This parameter sets the drawing coordinate system for the graphic display function.
The drawing coordinate system is set as shown below.

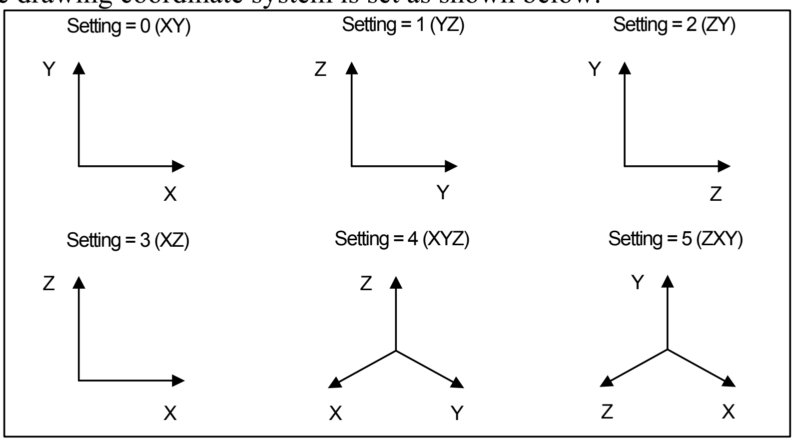
For M series:



For T series:



This parameter sets the drawing coordinate system for the dynamic graphic display function for the machining center system.
The drawing coordinate system is set as shown below.



6515	Change in the cross-sectional position in a triplane drawing in dynamic graphic display

[Input type] Parameter input
[Data type] Byte path
[Unit of data] Dot
[Valid data range] 0 to 10

This parameter sets changes in the cross-sectional position in a triplane drawing in dynamic graphic display, which are made when the soft key is pressed and held.
A setting of 0 is assumed to be 1.

6581	RGB value of color palette 1

6582	RGB value of color palette 2

6583	RGB value of color palette 3

6584	RGB value of color palette 4

6585	RGB value of color palette 5

6586	RGB value of color palette 6

6587	RGB value of color palette 7

6588	RGB value of color palette 8

6589	RGB value of color palette 9

6590	RGB value of color palette 10

6591	RGB value of color palette 11

6592	RGB value of color palette 12

6593	RGB value of color palette 13

6594	RGB value of color palette 14

6595	RGB value of color palette 15

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 151515

Each of these parameters sets the RGB value of each color palette by specifying a 6-digit number as described below.
rrggbb: 6-digit number (rr: red data, gg: green data, bb: blue data)
The valid data range of each color is 0 to 15 (same as the tone levels on the color setting screen). When a number equal to or greater than 16 is specified, the specification of 15 is assumed.
Example)
When the tone level of a color is: red:1 green:2, blue:3, set 10203 in the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
6700								PCM

[Input type] Parameter input
[Data type] Bit path

#0 PCM M code that counts the total number of machined parts and the number of machined parts
0: M02, or M30, or an M code specified by parameter No.6710
1: Only M code specified by parameter No.6710

6710	M code that counts the number of machined parts

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999

The total number of machined parts and the number of machined parts are counted (+1) when the M code set is executed.

> **NOTE**
> The setting of 0 is invalid (no count operation is performed with M00.) Moreover, M98, M99, M198 (external device subprogram calling), and M codes used for subprogram calling and macro calling cannot be set as M codes for count-up operation. (Even when such an M code is set, count-up operation is not performed, ignoring the M code.)

6711	Number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

The number of machined parts is counted (+1) together with the total number of machined parts when the M02, M30, or a M code specified by parameter No.6710 is executed.

> **NOTE**
> The number of parts is not counted for M02, M03, when bit 0 (PCM) of parameter No. 6700 is set to 1.

6712	Total number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

This parameter sets the total number of machined parts.
The total number of machined parts is counted (+1) when M02, M30, or an M code specified by parameter No.6710 is executed.

> **NOTE**
> The number of parts is not counted for M02, M30, when bit 0 (PCM) of parameter No. 6700 is set to 1.

6713	Number of required parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

This parameter sets the number of required machined parts.

Required parts finish signal PRTSF <F0062.7> is output to PMC when the number of machined parts reaches the number of required parts. The number of parts is regarded as infinity when the number of required parts is zero. The PRTSF signal is then not output.

6750	Integrated value of power-on period

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of power-on period.

6751	Operation time (integrated value of time during automatic operation) 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

For details, see the description of parameter No. 6752.

6752	Operation time (integrated value of time during automatic operation) 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of time during automatic operation (neither stop nor hold time included).

The actual time accumulated during operation is the sum of this parameter No. 6751 and parameter No. 6752.

6753	Integrated value of cutting time 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

For details, see the description of parameter No. 6754.

6754	Integrated value of cutting time 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of a cutting time that is performed in cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03).

The actual time accumulated during cutting is the sum of this parameter No. 6753 and parameter No. 6754.

	#7	#6	#5	#4	#3	#2	#1	#0
6800	M6T	IGI	SNG	GRS	SIG	LTM	GS2	GS1

[Input type] Parameter input
[Data type] Bit path

#0 GS1
#1 **GS2** For the maximum number of groups set in parameter No. 6813, up to four tools can be registered per group. The combination of the number of registrable groups and the number of tools per group can be changed by setting GS1 and GS2.

GS2	GS1	Number of groups	Number of tools
0	0	1 to maximum number of groups (No. 6813)/8	1 to 16
0	1	1 to maximum number of groups (No. 6813)/4	1 to 8
1	0	1 to maximum number of groups (No. 6813)/2	1 to 4
1	1	1 to maximum number of groups (No. 6813)	1 to 2

> **NOTE**
> After changing these parameters, set data again by using G10 L3 ;(registration after deletion of data of all groups).

#2 LTM The tool life count is specified by:
0: Count.
1: Duration.

> **NOTE**
> After changing this parameter, set data again by using G10 L3 ;(registration after deletion of data of all groups).

#3 SIG When a tool is skipped by a signal, the group number is:
0: Not input by the tool group number selection signals.
1: Input by the tool group number selection signals.

> **NOTE**
> When this parameter is set to 0, a tool of the currently used group is skipped.

#4 GRS When the tool change reset signal (TLRST) is input:
0: If the life of the group specified by the tool group number selection signals has expired, the execution data of the group is cleared.
1: The execution data of all registered groups is cleared.

If this parameter is set to 1, the execution data of all registered groups is cleared also when the clear operation to clear execution data is performed on the tool life management list screen.

#5 SNG When the tool skip signal (TLSKP) is input while a tool not controlled by the tool life management function is being used:
0: A tool of the most recently used group or a specified group (bit 3 (SIG) of parameter No. 6800) is skipped.
1: The tool skip signal is ignored.

#6 IGI Tool back numbers are:
0: Not ignored.
1: Ignored.

#7 M6T A T code specified in the same block as M06 is:
0: Assumed to be a back number.
1: Assumed to be a command specifying the next tool group.

	#7	#6	#5	#4	#3	#2	#1	#0
6801	M6E				EMD	LVF	TSM	
	M6E				EMD	LVF		

[Input type] Parameter input
[Data type] Bit path

#1 TSM In the tool life management function, life counting is performed as follows when more than one offset is specified:
0: Counting is performed for each tool number.
1: Counting is performed for each tool.

#2 LVF When the life value is counted by duration in the tool life management function, tool life count override signals *TLV0 to *TLV9 <G049.0 to G050.1> are:
0: Not used.
1: Used.

#3 EMD In the tool life management function, the mark "*" indicating that the life has expired is displayed when:
0: The next tool is used.
1: The life has just expired.

NOTE
If this parameter is set to 0, the "@" mark (indicating that the tool is in use) is kept displayed unless the next tool whose life has not expired is used. If this parameter is set to 1, marks are displayed in different ways depending on the life count type.
If the life count type is the duration specification type, the "*" mark (indicating that the life has expired) appears when the life has expired. If the life count type is the count specification type, one count is not assumed until the end of the program (M02, M30, and so on). Therefore, even when the life value and the tool life counter value match, the "*" mark (life has expired) does not appear. The "*" mark (life has expired) appears when the tool is used again by a tool group command (T code) or tool change command (M06) issued after the CNC is reset.

#7 M6E When a T code is specified in the same block as M06:
0: The T code is treated as a back number or the group number to be selected next. Which number is assumed depends on the setting of bit 7 (M6T) of parameter No. 6800.
1: Life counting for the tool group starts immediately.

	#7	#6	#5	#4	#3	#2	#1	#0
6802	RMT	TSK				E17	TCO	T99

[Input type] Parameter input
[Data type] Bit path

#0 T99 When M99 of the main program is executed, and there is a the life was expired tool group:

0: The tool change signal is not output.
1: The tool change signal is output, and the automatic operation becomes a stopped state..

If the life count is specified by use count and this parameter 1, the tool change signal TLCH <Fn064.0> is output and the automatic operation becomes a stopped state if the life of at least one tool group has expired when the M99 command is specified.
If the life count type is the duration specification type, the automatic operation becomes a stopped state if the life of at least one tool group has expired when the M99 command is specified.

M

If the life count is specified by use count, after the M99 command is specified, a tool group command (T code) selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one.

T

If the life count is specified by use count, when a tool group command (T code) is specified after the M99 command is specified, a tool whose life has not expired is selected from a specified group, and the tool life counter is incremented by one.

#1 TCO
#2 E17

Specifies whether to allow the FOCAS2 or PMC window function to write tool information of a group being used or a group to be used next during automatic operation (the OP signal is set to "1").

<table>
<tr><th colspan="3" rowspan="4">Condition</th><th colspan="3">6802#1(TCO)</th></tr>
<tr><th rowspan="3">0</th><th colspan="2">1</th></tr>
<tr><th colspan="2">6802#2(E17)</th></tr>
<tr><th>1</th><th>0</th></tr>
<tr><td rowspan="3">During automatic operation</td><td rowspan="2">Group being used or to be used next</td><td>Tool being used</td><td>×</td><td>□</td><td>○</td></tr>
<tr><td>Tool not being used</td><td>×</td><td>○</td><td>○</td></tr>
<tr><td colspan="2">Group neither being used nor to be used next</td><td>○</td><td>○</td><td>○</td></tr>
<tr><td colspan="3">Not during automatic operation</td><td>○</td><td>○</td><td>○</td></tr>
</table>

○: Tool information can be written from FOCAS2 and PMC window.
×: Tool information cannot be written from FOCAS2 and PMC window.
When an attempt is made to write tool information from PMC window, completion code 13 (REJECT ALARM) is returned.
□: Tool information cannot be cleared.

> **NOTE**
> When tool information of a tool being used (marked with "@") in the group being used or to be used next or tool information of the most recently used tool (marked with "@") in a group that is neither the group being used nor the group to be used next is cleared, the life counter is reset to 0.
> It is possible to modify tool information of a tool in the group to be used next. However, because tool selection is already completed, the selected tool does not change even when the tool information is modified.
> This parameter has no influence on modifications to tool information by edit operations from the tool life management screen.

#6 TSK If the count type in tool life management is the duration type, then when the last tool of a group is skipped by a signal:
0: The count value for the last tool equals the life value.
1: The count value for the last tool remains unchanged.

#7 RMT Tool life arrival notice signal TLCHB is turned on and off as follows:
0: The signal is turned on if the remaining life value (the life value minus the life counter value) is smaller than or equal to the remaining life setting. The signal is turned off if the remaining life value (the life value minus the life counter value) is greater than the remaining life setting.
1: The signal is turned on if the remaining life value (the life value minus the life counter value) is equal to the remaining life setting. The signal is turned off if the remaining life value (the life value minus the life counter value) is not equal to the remaining life setting.

> **NOTE**
> When using the life count override feature, set bit 7 (RMT) of parameter No. 6802 to 0. When the life count is specified by duration, the unit used for determining the result of comparison between the remaining life and the remaining life setting varies depending on the life count interval (bit 0 (FCO) of parameter No. 6805). If the life is counted every second, the comparison is made in units of 1 minute; if the life is counted every 0.1 second, the comparison is made in units of 0.1 minute.

	#7	#6	#5	#4	#3	#2	#1	#0
6804		LFI				ETE	TCI	

[Input type] Parameter input
[Data type] Bit path

#1 TCI During automatic operation (the OP signal is "1"), editing of tool life data is:
0: Disabled.
1: Enabled.

> **NOTE**
> When this parameter is set to 1, tool life data can be edited even during automatic operation (the OP signal is "1"). If the target group for editing is the group being used or the group to be used next, however, only presetting of the life counter is permitted, and other data cannot be modified.

#2 ETE In the tool life management screen, the mark of the tool at the life was expired of the final tool in the group :
0: depends on setting parameter EMD (No.6801#3).
1: is "*" mark.
If bit 2 (ETE) of parameter No. 6804 is set to 1, when the life counter of the final tool in the group becomes equal to the life value, display mark "*" in the final tool of the tool life management screen.
When tool change signal TLCH<Fn064.0> is "1", the state of the life was expired of the tool can be read by reading tool information on the final tool in FOCAS2 or the PMC window.

#6 LFI In tool life management, counting of the life of a selected tool is:
0: Enabled.
1: Enabled or disabled according to the status of tool life counting disable signal LFCIV<G048.2>.

	#7	#6	#5	#4	#3	#2	#1	#0
6805	TAD	TRU	TRS				FGL	FCO

[Input type] Parameter input
[Data type] Bit path

#0 FCO If the life count type is the duration specification type, the life is counted as follows:
0: Every second.
1: Every 0.1 second.
According to the setting of this parameter, the increment system of life values and tool life counter values displayed on the tool life management screen is set as follows:

Parameter FCO	0	1
Increment system for display and setting of life values and life counter values	1-minute increments	0.1-minute increments

> **NOTE**
> After changing the setting of this parameter, set data again by using G10L3;(registration after deletion of data of all groups).

#1 FGL If the life count type is the duration specification type, life data registered by G10 is:
0: In one-minute increments.
1: In 0.1-second increments.

#5 TRS Tool change reset signal TLRST is valid when reset signal RST is not "1" and:

0: The reset state (automatic operation signal OP is "0") is observed.

1: The reset state (automatic operation signal OP is "0"), automatic operation stop state (The STL and SPL signals are "0" and the OP signal is "1"), or the automatic operation pause state (the STL signal is "0" and the SPL signal is "1") is observed. The TLRST signal, however, is invalid when the automatic operation stop state, automatic operation pause state, and automatic operation start state (the STL signal is "1") is observed during execution of a data setting command (G10L3).

#6 TRU When the life count type is the duration specification type, and the life is counted every second (bit 0 (FCO) of parameter No. 6805 is set to 0):

0: Cutting time less than one second is discarded and is not counted.

1: Cutting time less than one second is rounded up and is counted as one second.

NOTE

If the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 is set to 1), cutting time less than 0.1 second is always rounded up and is counted as 0.1 second.

#7 TAD With tool change type D (bit 7 (M6E) of parameter No. 6801 is set to 1), when a block specifying M06 contains no T command:

0: An alarm PS0153 is issued.

1: No alarm is issued.

6810	Tool life management ignore number

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999

This parameter sets the tool life management ignore number.
When the value specified in a T code exceeds the value set in this parameter, the value obtained by subtracting the parameter-set value from the T code value is assumed to be the tool group number for tool life management.

6811	Tool life count restart M code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 127 (except 01, 02, 30, 98, and 99)

When 0 is specified, it is ignored.
When the life is specified by count, the tool change signal (TLCH) is output if the life of at least one tool group has expired when the tool life count restart M code is issued.
The T code (tool life management group command) specified after the tool life count restart M code selects a tool whose life has not expired from a specified group, and the next M06 command increments the tool life counter by one.
When the life is specified by duration, specifying the tool life count restart M code causes nothing. When 0 is set in this parameter, the tool life count restart M code is invalid. When the data of M code exceeds 127 values, set 0 in parameter No.6811, and set the value of M code in parameter No.13221. The data range of parameter No.13221 is from 0 to 255.

6813	Maximum number of groups in tool life management

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Unit of data] Group
[Valid data range] 0, 8, 16 to 128

This parameter sets the maximum number of groups to be used for each path. As the maximum number of groups, set a multiple of eight. When this parameter is 0, 128 groups are set. Up to 128 groups can be set for each path.

> **NOTE**
> If the power is turned on after this parameter is changed, all data in the tool life management file is initialized. Therefore, the life management data of all paths that use the tool life management function must be set.

6844	Remaining tool life (use count)

[Input type] Parameter input
[Data type] Word path
[Valid data range] This range is the same as the tool life range.

This parameter sets a remaining tool life (use count) used to output the tool life arrival notice signal when the tool life is specified by use count. If a value greater than the tool life value or 0 is set in this parameter, the tool life arrival notice signal is not output.

6845	Remaining tool life (use duration)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min
[Valid data range] Not greater than the tool life value

This parameter sets the remaining tool life (use duration) used to output the tool life arrival notice signal when the tool life is specified by use duration. If a value greater than the tool life value or 0 is specified in this parameter, the tool life arrival notice signal is not output.

> **NOTE**
> When the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 = 1), the parameter value is in 0.1-minute increments.

6930	Maximum value of the operating range of the 1st position switch
to	to
6945	Maximum value of the operating range of the 16th position switch

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the maximum value of the operating range of the first to sixteenth position switches.

> **NOTE**
> 1 For a diameter-specified axis, use diameter values to specify the parameters used to set the maximum and minimum values of an operating range.
> 2 The position switch function is enabled upon completion of reference position return.
> 3 The parameter No.6940 to No.6945 is effective when bit 1 (EPW) of parameter No.6901 is set to 1.

6950	**Minimum value of the operating range of the 1st position switch**
to	**to**
6965	**Minimum value of the operating range of the 16th position switch**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the minimum value of the operating range of the first to sixteenth position switches.

> **NOTE**
> 1 For a diameter-specified axis, use diameter values to specify the parameters used to set the maximum and minimum values of an operating range.
> 2 The position switch function is enabled upon completion of reference position return.
> 3 The parameter No.6960 to No.6965 is effective when bit 1 (EPW) of parameter No.6901 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
7055								
					BCG			

[Input type] Parameter input
[Data type] Bit path

#3 BCG The pre-interpolation bell-shaped acceleration/deceleration time constant change function in AI contour control mode is:
0: Disabled.
1: Enabled.

7066	
	Acceleration/deceleration reference speed for the time constant change function of bell-shaped acceleration/deceleration before interpolation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)

[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the acceleration/deceleration reference speed of the time constant change function of the bell-shaped acceleration/deceleration before interpolation in AI contour control.

	#7	#6	#5	#4	#3	#2	#1	#0
7100					HCL		THD	JHD

[Input type] Parameter input
[Data type] Bit path

#0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed
0: Invalid
1: Valid

	When JHD:=0		When JHD:=1	
	JOG feed mode	Manual handle feed mode	JOG feed mode	Manual handle feed mode
JOG feed	○	×	○	×
Manual handle feed	×	○	○	○
Incremental feed	×	×	×	○

#1 THD In the TEACH IN JOG mode, the manual pulse generator is:
0: Disabled.
1: Enabled.

#3 HCL The clearing of handle interruption amount display by soft key [CAN] operation is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Input type] Parameter input
[Data type] Bit axis

#0 HNGx Axis movement direction for rotation direction of manual pulse generator
0: Same in direction
1: Reverse in direction

	#7	#6	#5	#4	#3	#2	#1	#0
7103					HIT	HNT	RTH	

[Input type] Parameter input
[Data type] Bit path

#1 RTH By a reset or emergency stop, the amount of manual handle interruption is:
0: Not canceled.
1: Canceled.

#2 HNT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals) (MP1, MP2), the travel distance magnification for incremental feed/manual handle feed is:

0: Same.
1: 10 times greater.

#3 HIT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals (MP1, MP2), the travel distance magnification for manual handle interrupt is:

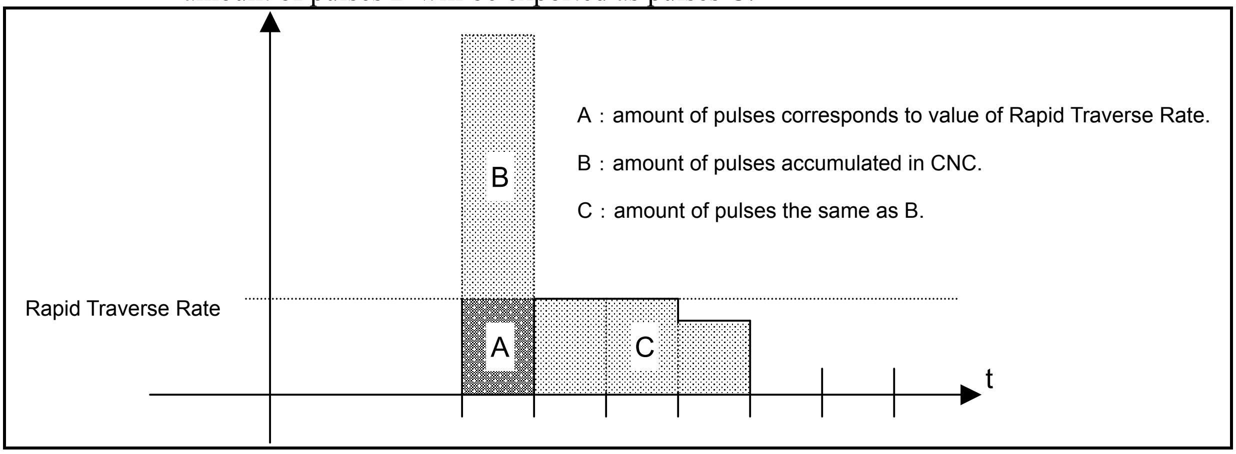
0: Same.
1: 10 times greater.

7117	Allowable number of pulses that can be accumulated during manual handle feed

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Pulse
[Valid data range] 0 to 999999999

This parameter sets the number of pulses from the manual pulse generator that exceed the rapid traverse rate and can be accumulated without being discarded if manual handle feed faster than the rapid traverse rate is specified.

The amount of pulses exceeding the rapid traverse rate can be saved by CNC as B. And amount of pulses B will be exported as pulses C.



Amount of pulses exported by CNC in Manual Handle Feed

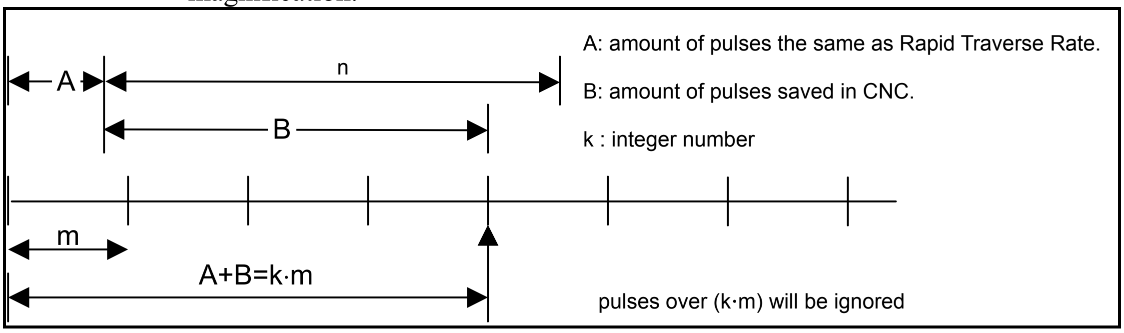
Amount of pulses B is calculated in 2 cases as following:

In case of
1) Parameter No.7117 = 0
The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0)

In case of
2) Parameter No.7117 > 0
The feedrate is clamped as the Rapid Traverse Rate, but the pulses exceeding the Rapid Traverse Rate is not ignored. Amount of pulses accumulated in CNC is calculated as following. (Although stopping the rotation of manual pulse generator, if there is pulses accumulated in CNC, it will be exported and the tool will move as long as amount of it.)

Magnification set by MP1, MP2<Gn019.4,.5> is m, value of parameter No.7117 is n.

n < m: Clamping is set performed at value of parameter No.7117.
n ≥ m: Amount A+B, showed in figure, which's value is multiple of m and small than n. As a result, clamping is performed as an integral multiple of the selected magnification.



Amount of pulses exceeding the Rapid Traverse Rate (n ≥ m)

> **NOTE**
> Due to change of mode, clamping can be performed not as an integral multiple of the selected magnification.
> The distance the tool moves may not match the graduations on the manual pulse generator.

	#7	#6	#5	#4	#3	#2	#1	#0
7200		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OP1 Mode selection on software operator's panel
0: Not performed
1: Performed

#1 OP2 JOG feed axis select and manual rapid traverse select on software operator's panel
0: Not performed
1: Performed

#2 OP3 Manual pulse generator's axis select and manual pulse generator's magnification select on software operator's panel
0: Not performed
1: Performed

#3 OP4 JOG feedrate override select, feedrate override select, and rapid traverse override select on software operator's panel
0: Not performed
1: Performed

#4 OP5 Optional block skip select, single block select, machine lock select, and dry run select on software operator's panel
- 0: Not performed
- 1: Performed

#5 OP6 Protect key on software operator's panel
- 0: Not performed
- 1: Performed

#6 OP7 Feed hold on software operator's panel
- 0: Not performed
- 1: Performed

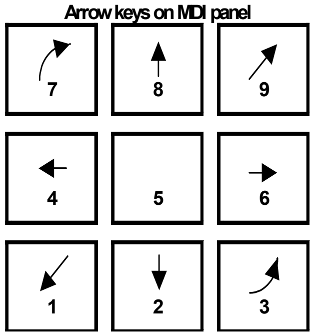
7210	**Jog-movement axis and its direction on software operator's panel "↑"**
7211	**Jog-movement axis and its direction on software operator's panel "↓"**
7212	**Jog-movement axis and its direction on software operator's panel "→"**
7213	**Jog-movement axis and its direction on software operator's panel "←"**
7214	**Jog-movement axis and its direction on software operator's panel "↙"**
7215	**Jog-movement axis and its direction on software operator's panel "↗"**
7216	**Jog-movement axis and its direction on software operator's panel "⤴"**
7217	**Jog-movement axis and its direction on software operator's panel "↷"**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 8

On software operator's panel, set a feed axis corresponding to an arrow key on the MDI panel when jog feed is performed.

Setting value	Feed axis and direction
0	Not moved
1	First axis, positive direction
2	First axis, negative direction
3	Second axis, positive direction
4	Second axis, negative direction
5	Third axis, positive direction
6	Third axis, negative direction
7	Fourth axis, positive direction
8	Fourth axis, negative direction

Arrow keys on MDI panel



[Example] Under X, Y, and Z axis configuration, to set arrow keys to feed the axes in the direction specified as follows, set the parameters to the values given below. [8↑] to the positive direction of the Z axis, [2↓] to the negative direction of the Z axis, [6→] to the positive direction of the X axis [4←] to the negative direction of the X axis, [1↙] to the positive direction of the Y axis, [9↗] to the negative direction of the Y axis

Parameter No.7210 = 5 (Z axis, positive direction)
Parameter No.7211 = 6 (Z axis, negative direction)
Parameter No.7212 = 1 (X axis, positive direction)
Parameter No.7213 = 2 (X axis, negative direction)
Parameter No.7214 = 3 (Y axis, positive direction)
Parameter No.7215 = 4 (Y axis, negative direction)
Parameter No.7216 = 0 (Not used)
Parameter No.7217 = 0 (Not used)

	#7	#6	#5	#4	#3	#2	#1	#0
7300	MOU	MOA						

[Input type] Parameter input
[Data type] Bit path

#6 MOA In program restart operation, before movement to a machining restart point:
0: The last M, S, T, and B codes are output.
1: All M codes and the last S, T, and B codes are output.
This parameter is enabled when the MOU parameter is set to 1.

#7 MOU In program restart operation, before movement to a machining restart point after restart block search:
0: The M, S, T, and B codes are not output.
1: The last M, S, T, and B codes are output.

7310	Ordinal number of an axis along which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to Number of controlled axes
This parameter sets the ordinal number of an axis along which a movement is made in dry run after the program is restarted.

	#7	#6	#5	#4	#3	#2	#1	#0
7600	PLZ							

[Input type] Parameter input
[Data type] Bit path

#7 PLZ Reference position return based on a G28 command on the tool rotation axis for polygon turning is:
0: Performed in the same sequence as manual reference position return.
1: Performed by positioning using the rapid traverse rate.
The synchronous axis returns to the reference position in the same sequence as the manual reference position return when no return-to-reference position is performed after the power is turned on.

7610	Control axis number of tool rotation axis for polygon turning

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes

This parameter sets the control axis number of a rotation tool axis used for polygon turning.
However, when a G51.2 command is executed by setting 0 in this parameter, operation stops with the alarm (PS0314).

7640	Master axis in spindle-spindle polygon turning

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)

This parameter sets the master axis in spindle-spindle polygon turning.

NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 When any one of parameter No. 7640 and No. 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.
3 When an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.

7641	Polygon synchronous axis in spindle-spindle polygon turning

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)

This parameter sets the polygon synchronous (slave) axis in spindle-spindle polygon turning.

NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 When any one of parameter No. 7640 and No. 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.
3 When an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.

7642	Master axis in spindle-spindle polygon turning (spindle number common to the system)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)
This parameter sets the master axis in spindle-spindle polygon turning.

NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3 When an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5 A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

7643	Polygon synchronous axis in spindle-spindle polygon turning (spindle number common to the system)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)
This parameter sets the polygon synchronous (slave) axis in spindle-spindle polygon turning.

NOTE

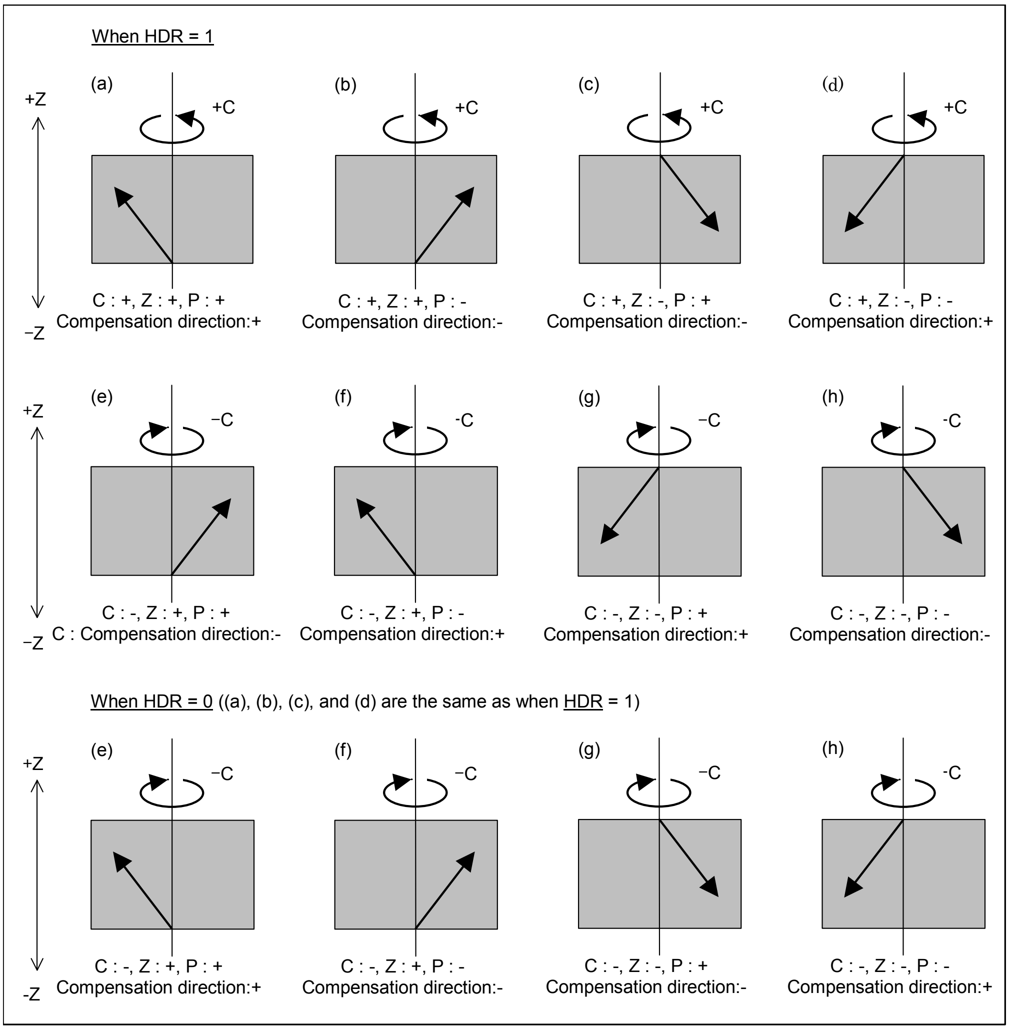
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3 When an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5 A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

	#7	#6	#5	#4	#3	#2	#1	#0
7700								
						HDR		HBR

[Input type] Parameter input
[Data type] Bit path

#0 HBR When the electronic gear box (EGB) function is used, performing a reset:
0: Cancels the synchronous mode (G81).
1: Does not cancel the synchronous mode. The mode is canceled only by the G80 command.

#2 HDR Direction of helical gear compensation (usually, set 1.)
(Example) To cut a left-twisted helical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.



	#7	#6	#5	#4	#3	#2	#1	#0
7701								
					LZR			

[Input type] Parameter input
[Data type] Bit path

#3 LZR When L (number of hob threads) = 0 is specified at the start of EGB synchronization (G81):
0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702								
					ART			TDP

[Input type] Parameter input
[Data type] Bit path

#0 TDP The specifiable number of teeth, T, of the electronic gear box (G81) is:
0: 1 to 1000
1: 0.1 to 100 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 1000 can be specified.

#3 ART The retract function executed when an alarm is issued is:
0: Disabled.
1: Enabled.
When an alarm is issued, a retract operation is performed with a set feedrate and travel distance (parameter Nos. 7740 and 7741).

> **NOTE**
> If a servo alarm is issued for other than the axis along which a retract operation is performed, the servo activating current is maintained until the retract operation is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
7703								
						ARO	ARE	ERV

[Input type] Parameter input
[Data type] Bit path

#0 ERV During EGB synchronization (G81), feed per revolution is performed for:
0: Feedback pulses.
1: Pulses converted to the speed for the workpiece axis.

#1 ARE In the retract function by an alarm, retract operation is:
0: Performed during EGB synchronization or automatic operation (automatic operation signal = 1).
1: Determined by the setting of parameter ARO.

#2 ARO The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization.
1: EGB synchronization and automatic operation (automatic operation signal OP = 1).

> **NOTE**
> This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 1.

The following table lists the parameter settings and corresponding operation.

ARE	ARO	Operation
1	0	During EGB synchronization
1	1	During EGB synchronization and automatic operation
0	0	During EGB synchronization or automatic operation
0	1	

> **NOTE**
> Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when an alarm is issued).

	#7	#6	#5	#4	#3	#2	#1	#0
7731								
					ECN			EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

> **NOTE**
> When this parameter is set to 0, no drilling canned cycle can be used.

#3 ECN During EGB synchronization:
0: G81 cannot be specified again. (An alarm (PS1595) occurs.)
1: G81 can be specified.

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during retraction for each axis.

7772	Number of position detector pulses per rotation about the tool axis

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the tool axis (on the spindle side), for the position detector.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

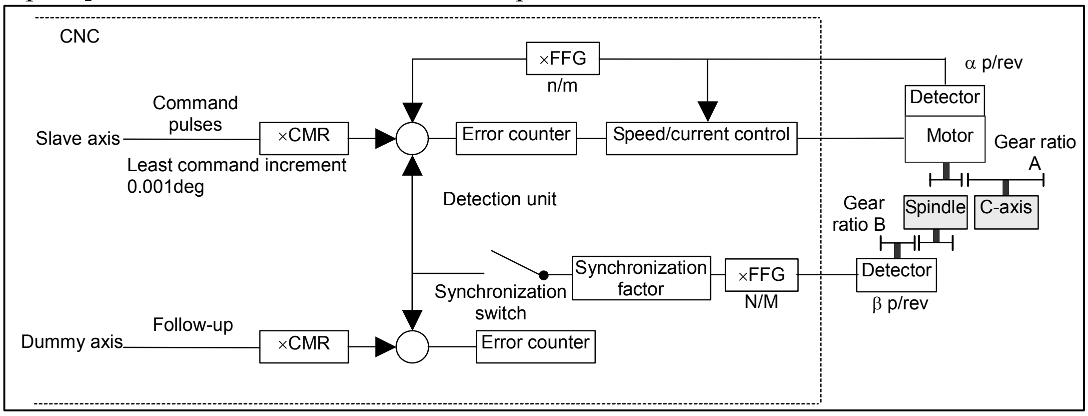
> **NOTE**
> Specify the number of feedback pulses per rotation about the tool axis for the position detector, considering the gear ratio with respect to the position coder.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the workpiece axis (on the slave side), for the position detector.
Set the number of pulses output by the detection unit.

Set parameters Nos. 7772 and 7773 when using the G81 EGB synchronization command.

[Example 1] When the EGB master axis is the spindle and the EGB slave axis is the C-axis



Gear ratio of the spindle to the detector B:
1/1 (The spindle and detector are directly connected to each other.)
Number of detector pulses per spindle rotation β: 80,000 pulses/rev
(Calculated for four pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis: 1/1
Gear ratio of the C-axis A: 1/36 (One rotation about the C-axis to 36 motor rotations)
Number of detector pulses per C-axis rotation α: 1,000,000 pulses/rev
C-axis CMR: 1
C-axis FFG n/m: 1/100

In this case, the number of pulses per spindle rotation is:
$80000 \times 1/1 = 80000$
Therefore, set 80000 for parameter No. 7772.
The number of pulses per C-axis rotation in the detection unit is:
$1000000 \div 1/36 \times 1/100 = 360000$
Therefore, set 360000 for parameter No. 7773.

[Example 2] When the gear ratio of the spindle to the detector B is 2/3 for the above example (When the detector rotates twice for three spindle rotations)
In this case, the number of pulses per spindle rotation is:

$$80000 \times \frac{2}{3} = \frac{160000}{3}$$

160000 cannot be divided by 3 without a remainder. In this case, change the setting of parameter No. 7773 so that the ratio of the settings of parameters Nos. 7772 and 7773 indicates the value you want to set.

$$\frac{\text{No.7772}}{\text{No.7773}} = \frac{160000 / 3}{360000} = \frac{160000}{360000 \times 3} = \frac{160000}{1080000}$$

Therefore, set 160000 for parameter No. 7772 and 1080000 for parameter No. 7773.
As described above, all the settings of parameters Nos. 7772 and 7773 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings. For example, you may set 16 for parameter No. 7772 and 108 for parameter No. 7773 for this case.

	#7	#6	#5	#4	#3	#2	#1	#0
8001					RDE	OVE		MLE

[Input type] Parameter input
[Data type] Bit path

#0 MLE Whether all axis machine lock signal MLK is valid for PMC-controlled axes
0: Valid
1: Invalid
The axis-by-axis machine lock signal MLKx depends on the setting of bit 1 of parameter No. 8006.

#2 OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
1: Signals specific to the PMC
The signals used depend on the settings of these parameter bits as indicated below.

Signals	No.8001#2=0 (same signals as those used for the CNC)		No.8001#2=1 (signals specific to the PMC)	
Feedrate override signals	*FV0 to *FV7	G012	*EFOV0 to *EFOV7	G151
Override cancellation signal	OVC	G006.	EOVC	G150.5
Rapid traverse override signals	ROV1,2	G014.0, .1	EROV1,2	G150.0, .1
Dry run signal	DRN	G46.7	EDRN	G150.7
Rapid traverse selection signal	RT	G19.7	ERT	G150.6

(The signal addresses at PMC selection time are for the group A.)

#3 RDE Whether dry run is valid for rapid traverse in PMC axis control
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10			RPD

[Input type] Parameter input
[Data type] Bit path

#0 RPD Rapid traverse rate for PMC-controlled axes
0: Feedrate specified with parameter No.1420
1: Feedrate specified with the feedrate data in an axis control command by PMC

#3 F10 Least increment for the feedrate for cutting feed (per minute) in PMC axis control
The following settings are applied when bit 4 (PF1) of parameter No. 8002 is set to 0 and bit 5 (PF2) of parameter No. 8002 is set to 0.

	F10	IS-A	IS-B	IS-C
Millimeter input (mm/min)	0	10	1	0.1
	1	100	10	1
Inch input (inch/min)	0	0.1	0.01	0.001
	1	1	0.1	0.01

#4 PF1
#5 PF2 Set the feedrate unit of cutting feedrate (feed per minute) for an axis controlled by the PMC.

Bit 5 (PF2) of parameter No. 8002	Bit 4 (PF1) of parameter No. 8002	Feedrate unit
0	0	1 / 1
0	1	1 / 10
1	0	1 / 100
1	1	1 / 1000

#6 FR1
#7 FR2 Set the feedrate unit for cutting feedrate (feed per rotation) for an axis controlled by the PMC.

Bit 7 (FR2) of parameter No. 8002	Bit 6 (FR1) of parameter No. 8002	Millimeter input (mm/rev)	Inch input (inch/rev)
0	0	0.0001	0.000001
1	1		
0	1	0.001	0.00001
1	0	0.01	0.0001

	#7	#6	#5	#4	#3	#2	#1	#0
8004		NCI						

[Input type] Parameter input
[Data type] Bit path

#6 NCI In axis control by the PMC, a position check at the time of deceleration is:
0: Performed.
1: Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
8005								EDC

[Input type] Setting input
[Data type] Bit path

#0 EDC In axis control by the PMC, an external deceleration function is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
8006		EZR		EFD				

[Input type] Parameter input
[Data type] Bit path

#4 EFD When cutting feed (feed per minute) is used in PMC axis control, the specification unit of feedrate data is:
0: Unchanged (1 times).
1: 100 times greater.

> **NOTE**
> When this parameter is set to 1, bit 3 (F10) of parameter No. 8002 is invalid.

#6 EZR In PMC axis control, bit 0 (ZRNx) of parameter No. 1005 is:

0: Invalid.
With a PMC controlled axis, the alarm (PS0224) is not issued.

1: Valid.
A reference position return state check is made on a PMC controlled axis as with an NC axis according to the setting of bit 0 (ZRNx) of parameter No. 1005.

	#7	#6	#5	#4	#3	#2	#1	#0
8008								EMRx

[Input type] Parameter input
[Data type] Bit axis

#0 EMRx When a PMC axis control command is issued in mirror image state, the mirror image is:

0: Not considered.
1: Considered.

This parameter is valid in the mirror image mode set with the mirror image signals MI1 to MI5 <G106.0 to 4> set to 1 or bit 0 (MIRx) of parameter No. 12 set to 1.
If a movement is made along the same axis by doubly specifying a command with the CNC and PMC axis control when this parameter is set to 0, and the mirror image mode is set, a coordinate shift can occur afterwards. So, do not attempt to make such a movement.

8010	Selection of the DI/DO group for each axis controlled by the PMC

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] T series: 1 to 4 (at 1-path control), 1 to 8 (at 2-path control)
M series: 1 to 4

Specify the DI/DO group to be used to specify a command for each PMC-controlled axis.

P8010	Description
1	Uses path 1 DI/DO group A (G142 to G153)
2	Uses path 1 DI/DO group B (G154 to G165)
3	Uses path 1 DI/DO group C (G166 to G177)
4	Uses path 1 DI/DO group D (G178 to G189)
5	Uses path 2 DI/DO group A (G1142 to G1153)
6	Uses path 2 DI/DO group B (G1154 to G1165)
7	Uses path 2 DI/DO group C (G1166 to G1177)
8	Uses path 2 DI/DO group D (G1178 to G1189)

NOTE
Use path 1 DI/DO (1 to 4) for the axes controlled by path 1.
Use path 2 DI/DO (5 to 8) for the axes controlled by path 2.

8030	Time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

For each axis, this parameter sets a time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control.

NOTE
When 0 is set in this parameter, the value set in parameter No. 1622 is used.
The value set in parameter No. 1622 is used also for linear acceleration/deceleration after cutting interpolation.

	#7	#6	#5	#4	#3	#2	#1	#0
8103								MWT

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 MWT As the signal interface for the waiting M code:
0: The path individual signal interface is used.
1: The path common signal interface is used.
This parameter can be selected only when 2-path control is used.

NOTE
When this parameter is set to 1, the operation equivalent to that of the FS0*i*-C is assumed.

8110	Waiting M code range (minimum value)

8111	Waiting M code range (maximum value)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0, 100 to 99999999
A range of M code values can be set by specifying a minimum waiting M coder value (parameter No. 8110) and a maximum waiting M code value (parameter No. 8111).
(parameter No. 8110) ≤ (waiting M code) ≤ (parameter No. 8111)
Set 0 in these parameters when the waiting M code is not used.

8130	Number of controlled axes

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Maximum number of controlled axes
This parameter sets the number of axes for each path.

> **NOTE**
> When spindle control with servo motor is enabled, set the number of axes including this axis for the axes with a spindle controlled axis with servo motor.

	#7	#6	#5	#4	#3	#2	#1	#0
8131						EDC		HPG
					AOV	EDC	F1D	HPG

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 HPG Manual handle feed is:
0: Not Used.
1: Used.

#1 F1D One-digit F code feed is:
0: Not Used.
1: Used.

#2 EDC External deceleration is:
0: Not Used.
1: Used.

#3 AOV Automatic corner override is:
0: Not Used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8132						BCD	YOF	TLF
			SCL	SPK	IXC	BCD		TLF

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 TLF Tool life management is:
0: Not Used.
1: Used.

#1 YOF Y-axis offset is:
0: Not Used.
1: Used.

#2 BCD Second auxiliary function is:
0: Not Used.
1: Used.

#3 IXC Index table indexing is:
0: Not Used.
1: Used.

> **NOTE**
> When enabling the index table indexing function, set bit 0 (ITI) of parameter No. 5501 to 0 in addition to this parameter. The index table indexing function is enabled only when both ITI and IXC are enabled.

#4 SPK Small diameter peck drilling cycle is:
0: Not Used.
1: Used.

#5 SCL Scaling is:
0: Not Used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8133		SPG	SSN	SYC	MSP	SCS	AXC	SSC
			SSN	SYC	MSP	SCS		SSC

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 SSC Constant surface speed control is:
0: Not Used.
1: Used.

#1 AXC Spindle positioning is:
0: Not Used.
1: Used.

> **NOTE**
> 1 Be sure to set 1 in bit 1 (AXC) of parameter No.8133 and 0 in bit 2 (SCS) of parameter No.8133 to use the spindle positioning function.
> 2 Both serial spindle Cs contour control function and spindle positioning function cannot be made effective at the same time.
> If both are specified as AXC=1 and SCS=1, both functions become invalid.
> Therefore, when a negative value is set in parameter No.1023 with above specification, alarm (SV1026) is generated.

> **NOTE**
> 3 Be sure to set 0 in bit 1 (AXC) of parameter No.8133 and 1 in bit 2 (SCS) of parameter No.8133 to use the serial spindle Cs contour control function.

#2 SCS Cs contour control is:
0: Not Used.
1: Used.

> **NOTE**
> 1 Be sure to set 0 in bit 1 (AXC) of parameter No.8133 and 1 in bit 2 (SCS) of parameter No.8133 to use the serial spindle Cs contour control function.
> 2 Both serial spindle Cs contour control function and spindle positioning function cannot be made effective at the same time.
> If both are specified as AXC=1 and SCS=1, both functions become invalid.
> Therefore, when a negative value is set in parameter No.1023 with above specification, alarm (SV1026) is generated.
> 3 Be sure to set 1 in bit 1 (AXC) of parameter No.8133 and 0 in bit 2 (SCS) of parameter No.8133 to use the spindle positioning function.

#3 MSP Multi-spindle is:
0: Not Used.
1: Used.

#4 SYC Spindle synchronization is:
0: Not Used.
1: Used.

#5 SSN Spindle serial output is:
0: Used.
1: Not Used.

Set this parameter as shown below depending on the spindle configuration.

Spindle configuration	Parameter SSN
When all spindles in the entire system are serial spindles	0
When serial spindles and analog spindles are mixed in the entire system	0
When all spindles in the entire system are analog spindles	1

#6 SPG Polygon turning with two spindles is:
0: Not Used.
1: Used.

> **NOTE**
> Be sure to invalidate the polygon turning with two spindles by this parameter when polygon turning is used. If the polygon turning is performed when 1 is set in this parameter, the alarm is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8134	NCT	NBG			NGR	CCR	BAR	IAP
	NCT	NBG			NGR		BAR	IAP

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 IAP Conversational programming with graphic function is:
0: Not Used.
1: Used.

#1 BAR Chuck and tail stock barrier function (T series) is:
0: Not Used.
1: Used.

> **NOTE**
> 1 The chuck and tail stock barrier function is provided only for the T series.
> 2 When the chuck and tail stock barrier function is selected, stored stroke limits 2 and 3 cannot be used.

That is, this parameter also specifies whether to use stored stroke limits 2 and 3 as shown below.

BAR Stored stroke limits 2 and 3 are:
0: Used.
1: Not Used.

#2 CCR Chamfering / corner R is:
0: Not Used.
1: Used.

#3 NGR Graphic display is:
0: Used.
1: Not Used.

#6 NBG Background editing is:
0: Used.
1: Not Used.

#7 NCT Run hour and parts count display is:
0: Used.
1: Not Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8135	NPD	NCV	NMC	NOR	NRG	NSQ	NHI	NPE

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 NPE Stored pitch error compensation is:
0: Used.
1: Not Used.

#1 NHI Manual handle interruption is:
0: Used.
1: Not Used.

#2 NSQ Program restart is:
0: Used.
1: Not Used.

#3 NRG Rigid tapping is:
0: Used.
1: Not Used.

#4 NOR Spindle orientation is:
0: Used.
1: Not Used.

NOTE
This parameter is valid only when spindle serial output can be used.

#5 NMC Custom macro is:
0: Used.
1: Not Used.

#6 NCV Addition of custom macro common variables is:
0: Used.
1: Not Used.

#7 NPD Pattern data input is:
0: Used.
1: Not Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8136	NCR	NGW	NDO	NOW	NOP		NWC	NWZ
	NTL	NGW	NDO	NOW	NOP	NWN	NWC	NWZ

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 NWZ Workpiece coordinate system is:
0: Used.
1: Not Used.

#1 NWC Workpiece coordinate system preset is:
0: Used.
1: Not Used.

#2 NWN Addition of workpiece coordinate system pair (48 pairs) is:
0: Used.
1: Not Used.

#3 NOP Software operator's panel is:
0: Used.
1: Not Used.

#4 NOW Software operator's panel general purpose switch is:
0: Used.
1: Not Used.

#5 NDO Tool compensation count 400 (M series) or tool compensation count 64(T series 1-path system) / 128(T series 2-path system) is:
0: Used.
1: Not Used.

NOTE
When the option of tool compensation count 99 (T series 1-path system) / 200 (T series 2-path system) is added, this parameter becomes invalid in T series. (Tool compensation count is fixed to 99 (T series 1-path system) / 200 (T series 2-path system).)

#6 NGW Tool offset memory C (M series) or tool geometry/wear compensation (T series) is:
0: Used.
1: Not Used.

#7 NTL Tool length measurement is:
NCR Tool nose radius compensation is:
0: Used.
1: Not Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8137								NVC

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 NVC Balance cutting is:
0: Used.
1: Not Used.

> **NOTE**
> When balance cutting is used (this parameter is 0), the mirror image of facing tool posts cannot be used. To use the mirror image of facing tool posts, set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8162						PKUx		

[Input type] Parameter input
[Data type] Bit axis

#2 PKUx In the parking state,
0: The absolute, relative, and machine coordinates are not updated.
1: The absolute and relative coordinates are updated. The machine coordinates are not updated.

> **NOTE**
> 1 With an axis for which polar coordinate interpolation is specified, set this parameter to 1. If this parameter is set to 0, a coordinate shift can occur when a single block stop or feed hold is performed in the polar coordinate interpolation mode.
> 2 With an axis that is set to function as a synchronous master axis and synchronous slave axis at the same time (with bit 1 (SYWx) of parameter No. 8167), set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8163	NUMx							

[Input type] Parameter input
[Data type] Bit axis

#7 NUMx When neither synchronous control nor composite control is applied, a move command for the axis is:
0: Not disabled.
1: Disabled.

> **NOTE**
> If a move command is specified for an axis with NUMx set to 1 when neither synchronous control nor composite control is applied, alarm PS0353 is issued.

8180	Master axis with which an axis is synchronized under synchronous control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . .)

This parameter sets the path number and intra-path relative axis number of the master axis with which each axis is synchronized. When zero is specified, the axis does not become a slave axis and is not synchronized with another axis. When an identical number is specified in two or more parameters, one master axis has two or more slave axes.

8183	Composite control axis of the other path in composite control for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . .)

This parameter sets with which axis each axis is to be placed under composite control. When zero is specified, control of the axis is not replaced under composite control. An identical number can be specified in two or more parameters, but composite control cannot be exercised for all of them at a time.

> **NOTE**
> When using the conventional 2-path interface (bit 1 (MIX) of parameter No. 8166 is 1, set this parameter on path 2. In this case, use the composite control axis selection signal on path 1.

8186	Master axis under superimposed control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . .)

This parameter sets the path number and intra-path relative axis number of a superimposed master axis for each axis when superimposed control is exercised. When zero is specified, the axis does not become a slave axis under superimposed control and the move pulse of another axis is not superimposed.

An identical number can be specified in two or more parameters to exercise superimposed control simultaneously. This means that superimposed control with one master axis and multiple slave axes is possible.

A slave axis may function as the master axis of another axis to allow three-generation superimposed control: parent (master axis) - child (slave axis/master axis) - grandchild (slave axis).

In this case, a movement along the child is made by its travel distance plus the travel distance of the parent, and a movement along the grandchild is made by its travel distance plus the travel distance of the child plus the travel distance of the parent.

Example of the relationship of parent (X1 of path 1) - child (X2 of path 2) - grandchild (U2 of path 2):
The travel distance of X1 is superimposed on X2, and the travel distances of X1 and X2 are further superimposed on U2.
Parameter No. 8186x of path 2 = 101
Parameter No. 8186u of path 2 = 201

	#7	#6	#5	#4	#3	#2	#1	#0
8200						AZR		AAC

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 AAC
0: Does not perform angular axis control.
1: Performs inclined axis control.

#2 AZR
0: The machine tool is moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.
1: The machine tool is not moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.

	#7	#6	#5	#4	#3	#2	#1	#0
8201	ADG					AO3	AO2	AOT

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 AOT Stored stroke limit 1 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#1 AO2 Stored stroke limit 2 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#2 AO3 Stored stroke limit 3 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#7 ADG The contents of diagnostic data Nos. 306 and 307 are:
0: Not swapped. The slanted axis and Cartesian axis are displayed in this order.
1: Swapped. The Cartesian axis and slanted axis are displayed in this order.

8210	Slant angle of a slanted axis in angular axis control

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] -180.000 to 180.000. However, angular axis control is disabled in the ranges -95.000 to -85.000 and 85.000 to 95.000 (in the case of IS-B).

8211	Axis number of a slanted axis subject to angular axis control

8212	Axis number of a Cartesian axis subject to slanted axis control

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to number of controlled axes
When angular axis control is to be applied to an arbitrary axis, these parameters set the axis numbers of a slanted axis and Cartesian axis. If 0 is set in either of the two parameters, the same number is set in the two parameters, or a number other than the controlled axis numbers is set in either of the two parameters, a slanted axis and Cartesian axis are selected as indicated in the following table:

	Slanted axis	Cartesian axis
M series	Y-axis (axis with 2 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes
T series	X-axis (axis with 1 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes

	#7	#6	#5	#4	#3	#2	#1	#0
8301				SYA				

[Input type] Parameter input
[Data type] Bit path

#4 SYA In the servo-off state in axis synchronous control, the limit of the difference between the positioning deviation of the master axis and that of the slave axis is:
0: Checked.
1: Not checked.

	#7	#6	#5	#4	#3	#2	#1	#0
8302	SMA							

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 SMA When an absolute position detector is attached, and bit 4 (APZ) of parameter No. 1815 for an axis in synchronous operation is set to OFF, APZ of the pairing axis in synchronous operation is:
0: Not set to OFF.
1: Set to OFF.

	#7	#6	#5	#4	#3	#2	#1	#0
8303	SOFx					SAFx	ATSx	ATEx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ATEx In axis synchronous control, automatic setting for grid positioning is:
0: Disabled
1: Enabled
Set this parameter with a slave axis.

#1 ATSx In axis synchronous control, automatic setting for grid positioning is:
0: Not started
1: Started
Set this parameter with a slave axis.

> **NOTE**
> When starting automatic setting for grid positioning, set ATS to 1. Upon the completion of setting, ATS is automatically set to 0.

#2 SAFx In axis synchronous control, a movement along a slave axis is:
0: Not added to actual feedrate display.
1: Added to actual feedrate display.
Set this parameter with a slave axis.

#7 SOFx In axis synchronous control, the synchronization establishment function based on machine coordinates is:
0: Disabled.
1: Enabled.
Set this parameter with a slave axis.
0.

	#7	#6	#5	#4	#3	#2	#1	#0
8304	SYEx		SCAx	MVBx		ADJx		SSAx

[Input type] Parameter input
[Data type] Bit axis

#0 SSAx When the one-direction synchronization establishment function under axis synchronous control is used:
0: The axis with a larger machine coordinate is used as the reference.
1: The axis with a smaller machine coordinate is used as the reference.

> **NOTE**
> 1 When at least one of these parameters is set, the power must be turned off before operation is continued.
> 2 Set this parameter (SSA) to the same value for both the master and slave axes.

#2 ADJx In axis synchronous control, this parameter specifies an axis along which a movement is made in the modification mode.
0: A movement is not made in the modification mode along the axis.
1: A movement is made in the modification mode along the axis.
When this parameter is set to 1, the modification mode is set.
Along an axis with this parameter set to 1, a movement is made by a move command for the master axis.
Set this parameter for one of the master and slave axes.
When there are multiple slave axes for one master axis, set this parameter to 1 for an axis with which a synchronization error excessive alarm is issued for recovery. If an alarm is issued with multiple axes, modify this parameter after recovery of one axis to recover another axis.

#4 MVBx In the modification mode, a move command in a direction that increases a synchronization error is:
0: Ignored.
1: Valid.
When there are multiple slave axes for one master axis, an attempt to reduce the synchronous error of a slave axis by a movement along the master axis can increase the synchronization error of another slave axis. If this parameter is set to 0 in such a case, a movement can be made in neither direction along the master axis. In this case, set bit 2 (ADJ) of parameter No. 8304 to make a movement along a slave axis to perform a corrective operation.

#5 SCAx In axis synchronous control:
0: Synchronous operation is performed when the axis synchronous control manual feed selection signal SYNCJ or the axis synchronous control selection signal SYNC for slave axes is set to 1.
1: Synchronous operation is performed at all times.
Set this parameter with a slave axis.

#7 SYEx When external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control, the slave axis is:
0: Not shifted.
1: Shifted by the same amount as specified for the master axis.
Set this parameter for the slave axis.
This function is disabled during normal operation.

	#7	#6	#5	#4	#3	#2	#1	#0
8305							SSE	SSO

[Input type] Parameter input
[Data type] Bit path

#0 SSO The uni-directional synchronization function in axis synchronous control is:
0: Disabled.
1: Enabled.

#1 SSE After emergency stop, the uni-directional synchronization function in axis synchronous control is:
0: Enabled.
1: Disabled.

8311	Axis number of master axis in axis synchronous control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to Number of controlled axes

Select a master axis in axis synchronous control. In the parameter for the slave axis, set the axis number of the master axis.

Example 1)
When one set of axis synchronous control is used:
When the master axis is the first axis (X-axis), and the slave axis is the third axis (Z-axis), set parameter No. 8311 as follows:
Parameter No.8311 X (first axis) = 0
Parameter No.8311 Y (second axis) = 0
Parameter No.8311 Z (third axis) = 1
Parameter No.8311 A (fourth axis) = 0

Example 2)
When two sets of axis synchronous control is used:
When the master axes are the first axis and second axis, and the slave axes are the fourth axis and third axis, set parameter No. 8311 as follows:
Parameter No.8311 X (first axis) = 0
Parameter No.8311 Y (second axis) = 0
Parameter No.8311 Z (third axis) = 2
Parameter No.8311 A (fourth axis) = 1

8312	Enabling/disabling mirror image in axis synchronous control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] -127 to 128

This parameter sets mirror image for the slave axis. When 100 or a more value is set with this parameter, the mirror image function is applied to synchronous control. Set this parameter to the slave axis.

Example)
For reverse synchronization with the master axis being the third axis and the slave axis being the fourth axis, set parameter No. 8312 as follows:
Parameter No.8312 X (first axis) = 0
Parameter No.8312 Y (second axis) = 0
Parameter No.8312 Z (third axis) = 0
Parameter No.8312 A (fourth axis) = 100

> **NOTE**
> In synchronous operation with mirror image applied, synchronization establishment, synchronization error checking, and modification mode cannot be used.

8314	Maximum allowable error in synchronization error check based on machine coordinates

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets a maximum allowable error in a synchronization error check based on machine coordinates. When the error between the master and slave axes in machine coordinates exceeds the value set in this parameter, the machine stops with the servo alarm (SV0005).
Set this parameter with a slave axis.

> **NOTE**
> Set 0 in this parameter when a synchronization error check is not made.

8323	Limit in positional deviation check in axis synchronous control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

This parameter sets the maximum allowable difference between the master axis and slave axis position deviations. When the absolute value of a positional deviation difference exceeds the value set in this parameter in axis synchronous control, the alarm (DS0001) is issued.
Set this parameter with a slave axis. If 0 is specified in this parameter, no position deviation difference check is made.

8325	Maximum compensation value in synchronization establishment based on machine coordinates

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the maximum compensation value for synchronization. When a compensation value exceeding the value set in this parameter is detected, the servo alarm (SV0001) is issued, and the synchronization establishment is not performed.
Specify a slave axis for this parameter. To enable this parameter, set the parameter SOF (bit 7 of parameter No.8303) to 1. When 0 is set in this parameter, synchronization establishment is not performed.

8326	Difference between master axis and slave axis reference counters

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

The difference between the master axis reference counter and slave axis reference counter (master axis and slave axis grid shift) is automatically set when automatic setting for grid positioning is performed. Then, the difference is transferred together with an ordinary grid shift value to the servo system when the power is turned on. This parameter is set with a slave axis.

8327	Torque difference alarm detection timer

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

This parameter sets a time from the servo preparation completion signal, SA <F000.6>, being set to 1 until torque difference alarm detection is started in axis synchronous control.
When 0 is set in this parameter, the specification of 512 msec is assumed.
Set this parameter with a slave axis.

8337	M code for turning off synchronization in axis synchronous control

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999

This parameter specifies an M code for switching from synchronous operation to normal operation.
The M code set in this parameter is not buffered.

8338	M code for turning on synchronization in axis synchronous control

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999

This parameter specifies an M code for switching from normal operation to synchronous operation.
The M code set in this parameter is not buffered.

8465	Upper limit of the speed of advance preview control/AI advance preview control/AI contour control

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets the upper limit of the speed of advance preview control/AI advance preview control/AI contour control.
If a speed higher than this parameter is specified in the advance preview control/AI advance preview control/AI contour control mode, clamping is performed at the speed specified by this parameter.

When this parameter is 0, clamping is not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
8900								PWE

[Input type] Setting input
[Data type] Bit

#0 PWE The setting, from an external device and MDI panel, of those parameters that cannot be set by setting input is:
0: Disabled.
1: Enabled.

10461	RGB value of color palette 1 for text for color set 3
10462	RGB value of color palette 2 for text for color set 3
	:
10475	RGB value of color palette 15 for text for color set 3

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 151515

Each of these parameters sets the RGB value of each color palette for text by specifying a 6-digit number as described below.
rrggbb: 6-digit number (rr: red data, gg: green data, bb: blue data)
The valid data range of each color is 0 to 15 (same as the tone levels on the color setting screen). When a number equal to or greater than 16 is specified, the specification of 15 is assumed.
Example)
When the tone level of a color is: red:1 green:2, blue:3, set 10203 in the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
11005								SIC

[Input type] Parameter input
[Data type] Bit

#0 SIC Spindle indexing is:
0: Performed based on absolute coordinates.
1: Performed based on machine coordinates.

11090	Path number with which the rotation of each spindle is specified

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 2

When a path is specified for spindle commands, this parameter sets a path number with which the rotation of a spindle can be specified.
0: Spindle commands can be issued from both paths.
1 to 2: Spindle commands can be issued from a set path.

> **NOTE**
> 1 This parameter is valid when SPSP<Gn536.7> is set to 1.
> 2 If the setting is illegal, an alarm (PS5305) is issued when a spindle command is issued from any one of the paths.
> 3 This setting does not apply to spindle commands using the spindle select signals (SWS1 and SWS2<Gn027.0 and 1>).

	#7	#6	#5	#4	#3	#2	#1	#0
11222							CIM	NIM

[Input type] Parameter input
[Data type] Bit path

#0 NIM Automatic conversion of a coordinate system by an inch/metric conversion command (G20 or G21) is:
0: Not performed.
1: Performed.

#1 CIM When an inch/metric conversion command (G20 or G21) is specified, if the workpiece coordinate system is shifted by the shift amount as described below:
0: An alarm (PS1298) is issued.
1: Clearing is performed.

This parameter is valid when bit 0 (NIM) of parameter No. 11222 is 1 or bit 2 (IRF) of parameter No. 14000 is 1 and clears the following items.:
- Manual intervention made when the manual absolute signal is off
- Issuance of a move command with the machine locked
- Movement by handle interrupt
- Operation with a mirror image
- Shifting of a workpiece coordinate system when a local coordinate system or workpiece coordinate system is set up

11307	Display sequence of the coordinates in current position display

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 5

This parameter sets the display sequence of the coordinates of a position displayed on the following screens:
10.4- inch display units
- Total position display screen
- Total position display on each screen

8.4-inch display units
- Total position display screen

The display sequence of coordinates corresponds to the parameter setting as follows:

Setting \ Display sequence of coordinates	1	2	3	4
0	Relative coordinates	Absolute coordinates	Machine coordinates	Remaining travel distance
1	Relative coordinates	Machine coordinates	Absolute coordinates	Remaining travel distance
2	Relative coordinates	Remaining travel distance	Absolute coordinates	Machine coordinates
3	Absolute coordinates	Machine coordinates	Relative coordinates	Remaining travel distance
4	Absolute coordinates	Remaining travel distance	Relative coordinates	Machine coordinates
5	Machine coordinates	Remaining travel distance	Relative coordinates	Absolute coordinates

If the setting is beyond the valid data range, 0 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
11329								
	GST	ACT	AER	GTF	BGM	GTL	DPC	

[Input type] Parameter input
[Data type] Bit path

#1 DPC The current coordinates displayed on each screen of the dynamic graphic display function are:
0: Absolute coordinates.
1: Machine coordinates.

#2 GTL When animated simulation is performed with the dynamic graphic display function, drawing at positions with tool length compensation considered is:
0: Not performed.
1: Performed.

#3 BGM Coordinates used by the dynamic graphic display function are:
0: Absolute coordinates.
1: Machine coordinates.

#4 GTF In tool path drawing with the dynamic graphic display function, drawing at in a position where tool compensation (tool length compensation, cutter compensation) is considered is:
0: Performed.
1: Not performed.

#5 AER When the tool path is drawn with the dynamic graphic display function, automatic erasure at the start of drawing is:
0: Not performed.
1: Performed.

#6 ACT In tool path drawing with the dynamic graphic display function, the drawing color of a tool path is:
0: Not changed automatically.
1: Changed automatically.

#7 GST When drawing cannot be performed for a command with the dynamic graphic display function:
0: The command is ignored, and drawing continues without stopping drawing.
1: Drawing stops.

11330	Magnification of drawing in dynamic graphic display

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01
[Valid data range] 1 to 10000
This parameter sets the magnification of the drawing range in the dynamic graphic display function.

11331	Screen center coordinate value in the drawing range in dynamic graphic display

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the coordinate value of the center of the drawing range in the dynamic graphic display function.

NOTE
If bit 3 (BGM) of parameter No. 11329 is set to 1, set the coordinate value on each axis in the machine coordinate system.

11332	Drawing range of tool path drawing in dynamic graphic display (maximum value)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the maximum coordinates in the drawing range in tool path drawing with dynamic graphic display function.

11333	Drawing range of tool path drawing in dynamic graphic display (minimum value)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the minimum coordinates in the drawing range in tool path drawing with dynamic graphic display function.

11334	Rotation angle of the drawing coordinate system in dynamic graphic display (vertical direction)

[Input type] Parameter input
[Data type] Word path
[Unit of data] degree
[Valid data range] -360 to 360

This parameter sets the rotation angle (vertical direction) of the drawing coordinate system in the dynamic graphic display function.

11335	Rotation angle of the drawing coordinate system in dynamic graphic display (horizontal direction)

[Input type] Parameter input
[Data type] Word path
[Unit of data] degree
[Valid data range] -360 to 360

This parameter sets the rotation angle of the drawing coordinate system in the dynamic graphic display function (the angle of rotation about the vertical axis on the screen, that passes the center position of the blank).

11336	Drawing color of the tool path in tool path drawing in dynamic graphic display

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 6

This parameter sets the color in which the tool path is drawn with the dynamic graphic display function.

11337	Color of the cursor indicating the tool position on the PATH GRAPHIC (TOOL POSITION) screen of dynamic graphic display

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 6

This parameter sets the color of the cursor indicating the tool position on the PATH GRAPHIC (TOOL POSITION) screen of the dynamic graphic display function.

11339	Drawing start sequence number in dynamic graphic display

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999

This parameter sets the sequence number at which drawing is started by the dynamic graphic display function.

11340	Drawing end sequence number in dynamic graphic display

[Input type] Parameter input
[Data type] 2-word path

[Valid data range] 0 to 99999999

This parameter sets the sequence number at which drawing is ended by the dynamic graphic display function.

11341	Drawing color of a blank figure in dynamic graphic display

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 6

This parameter sets the color in which a blank figure is drawn with the dynamic graphic display function.

11342	Rotation angle of the drawing coordinate system of dynamic graphic display (screen center)

[Input type] Parameter input
[Data type] Word path
[Unit of data] degree
[Valid data range] -360 to 360

This parameter sets the rotation angle of the drawing coordinate system in dynamic graphic display function (the angle of rotation about the vertical axis on the screen plane, that passes the center position of the blank).

11343	Blank figure in dynamic graphic display

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1

This parameter sets the type of a blank figure in dynamic graphic display function.

Setting	Figure
0	Cylinder or hollow cylinder (parallel to the Z-axis)
1	Rectangular parallelepiped

11344	Blank reference position in dynamic graphic display

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the reference position of a blank in the dynamic graphic display function by using coordinate values in the workpiece coordinate system.

11345	Blank dimension I in dynamic graphic display

11346	Blank dimension J in dynamic graphic display

11347	Blank dimension K in dynamic graphic display

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
These parameters set the dimensions of a blank in the dynamic graphic display function according to the blank figure as follows:

Blank figure	Address I	Address J	Address K
Rectangular parallelepiped	Length in the X-axis direction	Length in the Y-axis direction	Length in the Z-axis direction
Cylinder	Radius of a cylinder	0	Length of a cylinder
Barrel	Radius of the outer circle of a barrel	Radius of the inner circle of a barrel	Length of a barrel

11348	Drawing color of a tool in animated simulation in dynamic graphic display

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 6
This parameter sets the color in which a tool is drawn during animated simulation in the dynamic graphic display function.

	#7	#6	#5	#4	#3	#2	#1	#0
11349						GSP	ABC	

[Input type] Parameter input
[Data type] Bit

#1 ABC In animated simulation in the dynamic graphic display function, when a fine boring cycle or back boring cycle, which is a hole machining canned cycle, is performed, the movement for a shift at the hole bottom is:
0: Not drawn.
1: Drawn.

#2 GSP In tool path drawing in the dynamic graphic display function, the drawing start position is:
0: The end position of a block that makes a movement for the first time.
1: The current position.

> **NOTE**
> When G92, G52, or G92.1 is specified at the beginning of a program to be drawn, the position specified in this G code is assumed to be the drawing start position.

	#7	#6	#5	#4	#3	#2	#1	#0
11350						**PNE**		

[Input type] Parameter input
[Data type] Bit

#2 PNE A magnified path name is:
0: Not displayed.
1: Displayed.

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.
> The parameter is valid only for the 10.4-inch display.

	#7	#6	#5	#4	#3	#2	#1	#0
11352								**PNI**

[Input type] Parameter input
[Data type] Bit

#0 PNI A magnified path name is displayed in:
0: Normal video.
1: Reverse video.

> **NOTE**
> The parameter is valid only for the 10.4-inch display.

	#7	#6	#5	#4	#3	#2	#1	#0
11353								**SEK**

[Input type] Parameter input
[Data type] Bit

#0 SEK When the power is turned on, or when the clear state is present, sequence numbers are:
0: Not maintained.
1: Maintained.

> **NOTE**
> During a subprogram call, the sequence number of the subprogram is maintained.

11363	**Radius of a tool figure in dynamic graphic display**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
This parameter sets the radius of a tool figure in animated simulation with the dynamic graphic display function.

	#7	#6	#5	#4	#3	#2	#1	#0
11630								
								FRD

[Input type] Parameter input
[Data type] Bit path

#0 FRD The minimum command unit of the rotation angles of coordinate rotation is:
0: 0.001 degree.
1: 0.00001 degree. (1/100,000)

12600	Identification Number for synchronous, composite and superimposed control with program command

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0,1 to 32767
Set identification numbers that can be specified with P,Q addresses.
The axis whose identification number is "0" cannot become under synchronous /composite /superimposed control by CNC program.
The same identification number cannot be set to two or more axes through all paths.
When the same identification number is set, PS alarm (PS5339) occurs at G50.4/G50.5/G50.6/G51.4/G51.5/G51.6 block.

12990	(1st) G code modal group to be recorded in the history when an alarm is issued
12991	(2nd) G code modal group to be recorded in the history when an alarm is issued
12992	(3rd) G code modal group to be recorded in the history when an alarm is issued
12993	(4th) G code modal group to be recorded in the history when an alarm is issued
12994	(5th) G code modal group to be recorded in the history when an alarm is issued
12995	(6th) G code modal group to be recorded in the history when an alarm is issued
12996	(7th) G code modal group to be recorded in the history when an alarm is issued
12997	(8th) G code modal group to be recorded in the history when an alarm is issued
12998	(9th) G code modal group to be recorded in the history when an alarm is issued
12999	(10th) G code modal group to be recorded in the history when an alarm is issued

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum number of G code groups

Set the number of a G code modal group to be recorded in the alarm history and operation history when an alarm is issued.
* If a value beyond the valid data range is set, the status of group 04 is recorded.

13221	M code for tool life count restart

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255 (not including 01, 02, 30, 98, and 99)
When 0 is set, this parameter is ignored.
For the operation of an M code for tool life count restart, see the description of parameter No. 6811.
This parameter is used when an M code for tool life count restart exceeds 127.
Set parameter No. 6811 to 0, and set the value of an M code in this parameter.

13265	H code for using the tool length offset in tool life management

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999
Setting this parameter to H99 generally validates the compensation for the tool currently being used. By setting any H code in this parameter, the H code instead of H99 can be used. If 0 is specified, H99 is assumed.
A value ranging from 0 to 9999 can be set.

13266	D code for enabling cutter compensation in tool life management

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999
Setting this parameter to D99 generally validates the compensation for the tool currently being used. By setting any D code in this parameter, the D code instead of D99 can be used. If 0 is set, D99 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
13600								MCR
	MSA							MCR

[Input type] Parameter input
[Data type] Bit path

#0 MCR When an allowable acceleration rate adjustment is made with the machining condition selection function (machining parameter adjustment screen, precision level selection screen), parameter No. 1735 for the deceleration function based on acceleration in circular interpolation is:
0: Modified.
1: Not modified.

#7 MSA When the machining condition selection function is used, the acceleration rate change time (bell-shaped) (LV1, LV10) is:
0: Set using parameter Nos. 13612 and 13613.
1: Set using parameter Nos. 13662 and 13663.

	#7	#6	#5	#4	#3	#2	#1	#0
13601								MPR

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 MPR The machining parameter adjustment screen is:
0: Displayed.
1: Not displayed.
Even when this parameter is set to 1, the precision level selection screen is displayed.

13610	Acceleration rate for acceleration/deceleration before look-ahead interpolation in advanced preview control/AI advanced preview control/AI contour control (precision level 1)

13611	Acceleration rate for acceleration/deceleration before look-ahead interpolation in advanced preview control/AI advanced preview control/AI contour control (precision level 10)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an acceleration rate for acceleration/ deceleration before interpolation in advanced preview control/AI advanced preview control/AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13612	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 1)

13613	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Byte path
[Unit of data] msec
[Valid data range] 0 to 127
Each of these parameters sets an acceleration rate change time (bell-shaped) in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13620	Allowable acceleration rate when advanced preview control/AI advanced preview control/AI contour control is used (precision level 1)

13621	Allowable acceleration rate when advanced preview control/AI advanced preview control/AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Real axis

[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an allowable acceleration rate in advanced preview control/AI advanced preview control/AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13622	**Time constant for acceleration/deceleration after interpolation when advanced preview control/AI advanced preview control/AI contour control is used (precision level 1)**

13623	**Time constant for acceleration/deceleration after interpolation when advanced preview control/AI advanced preview control/AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 1 to 512
Each of these parameters sets a time constant for acceleration/ deceleration after interpolation when advanced preview control/AI advanced preview control/AI contour control is used. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13624	**Corner speed difference when advanced preview control/AI advanced preview control/AI contour control is used (precision level 1)**

13625	**Corner speed difference when advanced preview control/AI advanced preview control/AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets an allowable speed difference for speed determination based on corner speed difference in advanced preview control/AI advanced preview control/AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13626	**Maximum cutting speed when advanced preview control/AI advanced preview control/AI contour control is used (precision level 1)**

13627	**Maximum cutting speed when advanced preview control/AI advanced preview control/AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Each of these parameters sets a maximum cutting speed in advanced preview control/AI advanced preview control/AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13628	**Parameter number corresponding to arbitrary item 1 when advanced preview control/AI advanced preview control/AI contour control is used**

13629	**Parameter number corresponding to arbitrary item 2 when advanced preview control/AI advanced preview control/AI contour control is used**

NOTE

When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 65535

These parameters set the parameter numbers corresponding to arbitrary items 1 and 2.

NOTE

The parameter numbers corresponding to the following cannot be specified:

- Bit parameters
- Spindle parameters (No. 4000 to No. 4799)
- Parameters of real number type
- Parameters that require power-off (for which the alarm (PW0000) is issued)
- Nonexistent parameters

13630	**Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 1 when advanced preview control/AI advanced preview control/AI contour control is used**

13631	**Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 2 when advanced preview control/AI advanced preview control/AI contour control is used**

13632	**Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 1 when advanced preview control/AI advanced preview control/AI contour control is used**

13633	**Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 2 when advanced preview control/AI advanced preview control/AI contour control is used**

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Depend on the type of parameter for an arbitrary item
[Valid data range] Depend on the type of parameter for an arbitrary item

Each of these parameters sets a value with emphasis placed on speed or precision for a parameter.

13662	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 1), range extended

13663	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 10), range extended

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

Each of these parameters sets an acceleration rate change time (bell-shaped) in AI contour control. Set a value (precision 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

	#7	#6	#5	#4	#3	#2	#1	#0
14000						IRFx		

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit axis

#2 IRFx An inch-metric switch command (G20, G21) at the reference position is:
0: Disabled.
1: Enabled.

When this function is enabled for an axis, if an attempt to switch between the inch and metric unit is made although the tool is not at the reference position on that axis, an alarm (PS5362) is issued, and switching between the inch and metric unit is canceled.
Be sure to move the tool to the reference position by, for example, specifying G28 before switching between the inch and metric unit.

NOTE
1 This function enables the inch/metric switching commands (G20 and G21) at the reference position. It does not enable the switching of the setting input unit (bit 2 (INI) of parameter No. 0000).
2 Switching between inch and metric by setting the setting input unit (bit 2 (INI) of parameter No. 0000) is enabled only when the machine coordinate of the first reference position is 0 (parameter No. 1240 is 0) and presence on the first reference position is assumed.
For a system in which the machine coordinate of the first reference position is not 0, set this parameter to 1 and specify G20/G21 in the first reference position to switch between inch and metric.

14010	Maximum allowable travel distance when the reference position is established for a linear scale with an absolute address reference position

[Input type] Parameter input

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets the maximum allowable travel distance at the FL feedrate when the reference position is established for a linear scale with an absolute address reference position. When the travel distance exceeds the setting of this parameter, the alarm (DS0017) (SCALE WITH REFERENCE POSITION: REFERENCE POSITION ESTABLISHMENT FAILED) is issued. When this parameter is set to 0, the maximum allowable travel distance is not checked.

14340	**ATR value corresponding to slave 01 on FSSB**
to	**to**
14357	**ATR value corresponding to slave 18 on FSSB**

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 7,64,-56,-96

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 18 on FSSB.
The slave is a generic term for servo amplifiers and separate detector interface units connected via an FSSB optical cable to the CNC. Numbers 1 to 18 are assigned to slaves, with younger numbers sequentially assigned to slaves closer to the CNC.
A 2-axis amplifier consists of two slaves, and a 3-axis amplifier consists of three slaves. In each of these parameters, set a value as described below, depending on whether the slave is an amplifier, separate detector, or nonexistent.

- When the slave is an amplifier:
  Set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis to which the amplifier is assigned.
- When the slave is a separate detector interface unit:
  Set 64 for the first separate detector interface unit (connected near the CNC), and set -56 for the second separate detector interface unit (connected far from the CNC).
- When the slave is nonexistent:
  Set -96.

NOTE
1 When the electric gear box (EGB) function is used
Although an amplifier is not actually required for an EGB dummy axis, set this parameter with assuming that a dummy amplifier is connected. That is, as the address conversion table value for a nonexistent slave, set the value obtained by subtracting 1 from the setting of parameter No. 1023 for the EGB dummy axis, instead of -96.
2 When the FSSB is set to the automatic setting mode (when the parameter FMD (No.1902#0) is set to 0), parameter Nos. 14340 to 14357 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the parameter FMD (No.1902#0) is set to 1), be sure to directly set values in parameter Nos. 14340 to 14357.

Example of axis configuration and parameter settings

- Example 1

CNC

Controlled axis number	Program axis name No.1020	Servo axis No.1023
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5

	Slave number	ATR No.14340 to 14357	Axis
Single-axis amplifier	1	0	X
Two-axis amplifier	2	1	A
	3	2	Y
Two-axis amplifier	4	3	Z
	5	4	B
M1	6	64	(M1)
M2	7	-56	(M2)
	8 to 18	-96	(None)

CNC

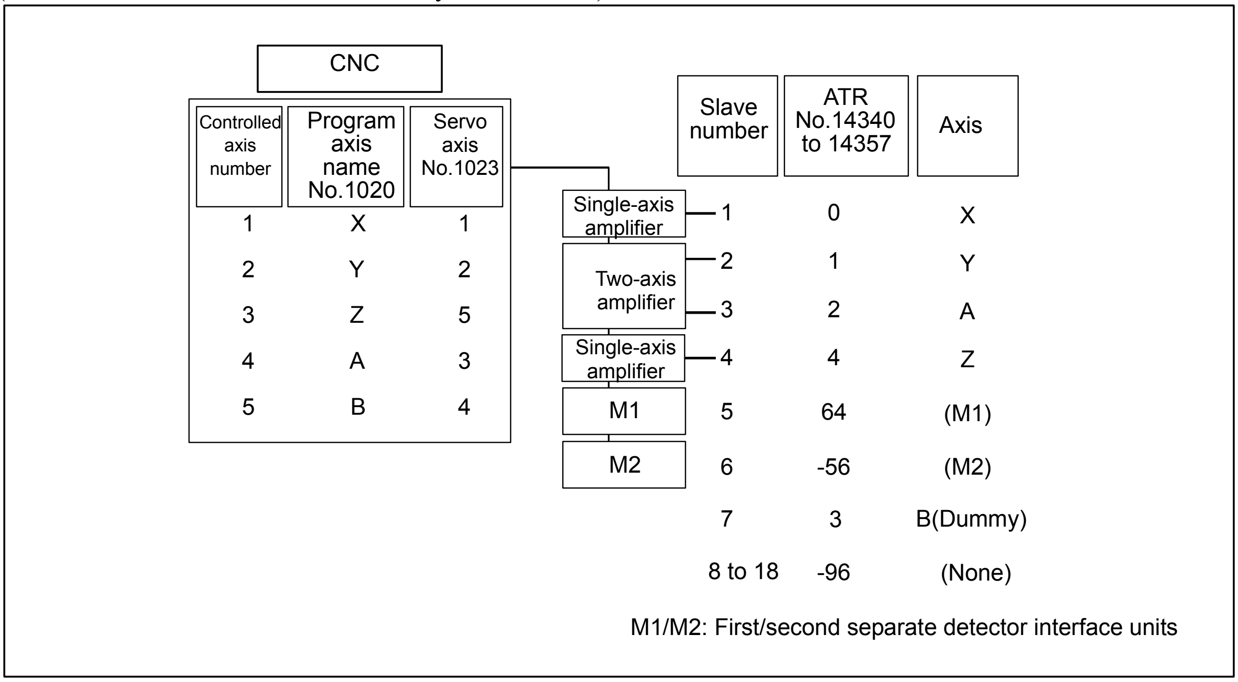
Controlled axis number	Program axis name No.1020	Servo axis No.1023
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5

	Slave number	ATR No.14340 to 14357	Axis
Single-axis amplifier	1	0	X
Two-axis amplifier	2	2	Y
	3	3	Z
Two-axis amplifier	4	1	A
	5	4	B
M1	6	64	(M1)
M2	7	-56	(M2)
	8 to 18	-96	(None)

M1/M2: First/second separate detector interface units

- Example 2

Example of axis configuration and parameter settings when the electronic gear box (EGB) function is used
(EGB slave axis: A-axis, EGB dummy axis: B-axis)



14376	**ATR value corresponding to connector 1 on the first separate detector interface unit**
to	**to**
14383	**ATR value corresponding to connector 8 on the first separate detector interface unit**

14384	**ATR value corresponding to connector 1 on the second separate detector interface unit**
to	**to**
14391	**ATR value corresponding to connector 8 on the third separate detector interface unit**

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 7, 32

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each connector on a separate detector interface unit.
In each of these parameters, set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis connected to a connector on a separate detector interface unit.
When there are axes for which settings are made to use a separate detector interface unit (bit 6 (PM1x) of parameter No. 1905 is set to 1 or bit 7 (PM2x) of parameter No. 1905 is set to 1, set 32 for connectors not used.

> **NOTE**
> When the FSSB is set to the automatic setting mode (when the parameter FMD (No.1902#0) is set to 0), parameter Nos. 14376 to 14391 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the parameter FMD (No.1902#0) is set to 1), be sure to directly set values in parameter Nos. 14376 to 14391.

	#7	#6	#5	#4	#3	#2	#1	#0
14476								DFS

[Input type] Parameter input
[Data type] Bit

#0 DFS The FSSB enters:
0: The FS0*i*-D-specific mode.
1: The FS0*i*-C-compatible mode.

14713	Unit of magnification by which enlargement and reduction is performed with the dynamic graphic display function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 255

This parameter sets the unit of magnification by which enlargement and reduction is performed with the dynamic graphic display function.
Unit of magnification = 64 / setting
If 0 is set, 64 is assumed.

14714	Unit of horizontal movement when a movement is made with the dynamic graphic display function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 255

This parameter sets the unit of horizontal movement (in dots) applied when a movement is made with the dynamic graphic display function.
If 0 is set, 64 is assumed.

14715	Unit of vertical movement when a movement is made with the dynamic graphic display function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 255

This parameter sets the unit of vertical movement (in dots) applied when a movement is made with the dynamic graphic display function.
If 0 is set, 35 is assumed.

14716	Unit of rotation angle when rotation is performed with the dynamic graphic display function

[Input type] Parameter input

[Data type] Word
[Valid data range] 0 to 255
This parameter sets the unit (in degrees) of a rotation angle by which the drawing coordinate system is rotated with the dynamic graphic display function.
If 0 is set, 10 is assumed.

18060	Backward movement prohibition M code that outputs no M code

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999
When an M code that prohibits backward movement is specified during backward movement, backward movement of blocks before the M code is prohibited. In this case, backward movement prohibition signal MRVSP<Fn091.2> is output.

This backward movement prohibition M code is not output to the PMC as an M code. Set an M code that is not used by an auxiliary function or macro as the backward movement prohibition M code.

18065	Backward movement prohibition M code 1 that outputs an M code

18066	Backward movement prohibition M code 2 that outputs an M code

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999
When an M code that prohibits backward movement is specified during backward movement, backward movement of blocks before the M code is prohibited. In this case, backward movement prohibition signal MRVSP<Fn091.2> is output.

These backward movement prohibition M codes are output to the PMC as M codes. Set M codes that are not used by an auxiliary function or macro as the backward movement prohibition M codes.

	#7	#6	#5	#4	#3	#2	#1	#0
19500		FNW						

[Input type] Parameter input
[Data type] Bit path

#6 FNW In the speed determination method by the speed difference of advanced preview control/AI advance preview control/AI contour control and the speed determination method by the acceleration of advanced preview control/AI contour control:
0: The maximum speed that does not exceed the allowable speed difference or allowable acceleration is selected.
1: The feedrate is determined so that the allowable speed difference and allowable acceleration of each axis are not exceeded and that the deceleration speed is constant regardless of the movement direction if the shape is the same.

	#7	#6	#5	#4	#3	#2	#1	#0
19501			FRP					

[Input type] Parameter input
[Data type] Bit path

#5 FRP linear-shaped rapid traverse in the advanced preview control/AI advance preview control/AI contour control mode is:

0: Acceleration/deceleration after interpolation.

1: Acceleration/deceleration before interpolation.

Set a maximum allowable acceleration rate for each axis in parameter No. 1671.
When using bell-shaped acceleration/deceleration before interpolation, set an acceleration rate change time in parameter No. 1672.

When this parameter is set to 1, acceleration/deceleration before interpolation is also applied to rapid traverse if all conditions below are satisfied. At this time, acceleration/deceleration after interpolation is not applied.

- Bit 1 (LRP) of parameter No. 1401 is set to 1: Linear interpolation type positioning
- A value other than 0 is set in parameter No. 1671 for an axis.

If all of these conditions are not satisfied, acceleration/deceleration after interpolation is applied.

> **NOTE**
> To enable blinking display and signal output indicating the advance preview control/AI advance preview control/AI contour control mode even when rapid traverse command is specified, set bit 1 (AIR) of parameter No. 1612 to 1 in addition to the above settings.

	#7	#6	#5	#4	#3	#2	#1	#0
19607			CAV					

[Input type] Parameter input
[Data type] Bit path

#5 CAV When an interference check finds that interference (overcutting) occurred:

0: Machining stops with the alarm (PS0041).
(Interference check alarm function)

1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)

For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit Bit machine group Bit path Bit axis Bit spindle	0 or 1	
Byte Byte machine group Byte path Byte axis Byte spindle	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Word Word machine group Word path Word axis Word spindle	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
2-word 2-word machine group 2-word path 2-word axis 2-word spindle	0 to ±999999999	Some parameters handle these types of data as unsigned data.
Real Real machine group Real path Real axis Real spindle	See the Standard Parameter Setting Tables.	

NOTE

1. Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
2. For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
3. For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
4. For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
5. For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
6. The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

NOTE
1 Values are rounded up or down to the nearest multiples of the minimum data unit.
2 A valid data range means data input limits, and may differ from values representing actual performance.
3 For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value."

(A) Length and angle parameters (type 1)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
inch	IS-A	0.001	-99999.999 to +99999.999
	IS-B	0.0001	-99999.9999 to +99999.9999
	IS-C	0.00001	-9999.99999 to +9999.99999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
inch	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-A	0.01	0.0 to +999000.00
	IS-B	0.001	0.0 to +999000.000
	IS-C	0.0001	0.0 to +99999.9999
inch/min	IS-A	0.001	0.0 to +96000.000
	IS-B	0.0001	0.0 to +9600.0000
	IS-C	0.00001	0.0 to +4000.00000

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

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

(D)Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 deg./sec^2	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
inch/sec^2	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

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

B PROGRAM CODE LIST

Character name	ISO code		EIA code		Custom macro	
	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		
Plus sign	+	2B	+	70		
Minus sign	-	2D	-	40		

Character name	ISO code		EIA code		Custom macro	
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	without custom macro	with custom macro
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)		△	△

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.
○ : 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 Codes not in this table are ignored if their parity is correct.

3 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

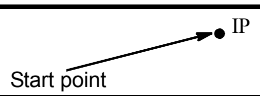
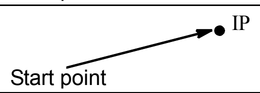
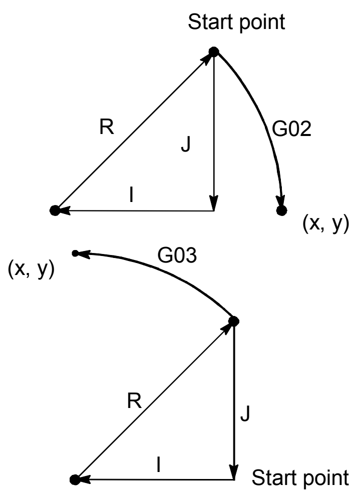
C LIST OF FUNCTIONS AND PROGRAM FORMAT

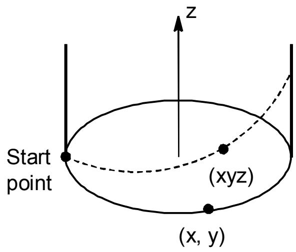
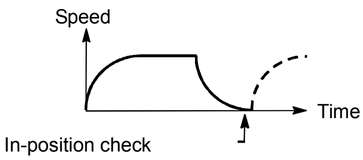
With some functions, the format used for specification on the M series differs from the format used for specification on the T series. Some functions are supported only for either M series or T series.
Some functions cannot be added as options depending on the model.
For details of command formats, see the relevant sections or subsections.

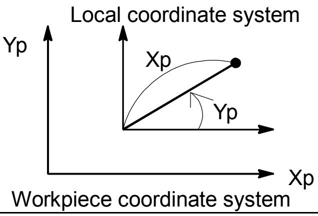
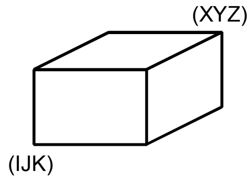
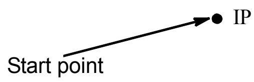
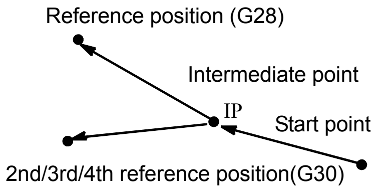
In the list, the following symbols are used:

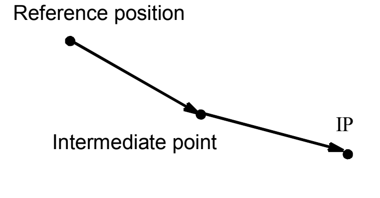
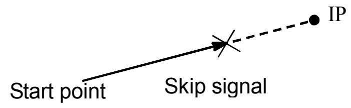
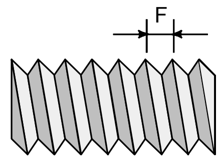
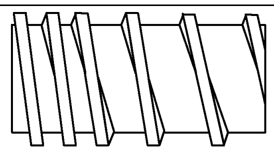
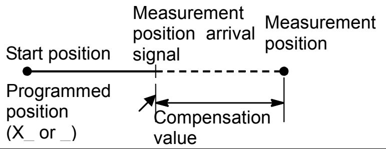
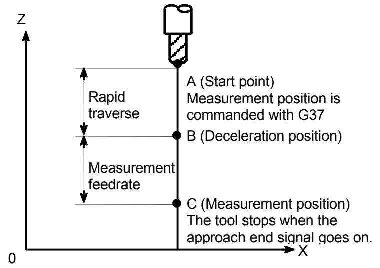
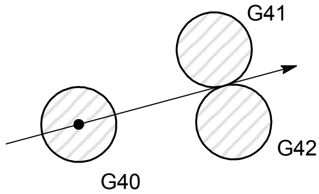
- For M series
  x : 1st basic axis (X),
  y : 2nd basic axis (Y),
  z : 3rd basic axis (Z)
- For T series
  x : 1st basic axis (X),
  z : 2nd basic axis (Z),
  coded using G code system A

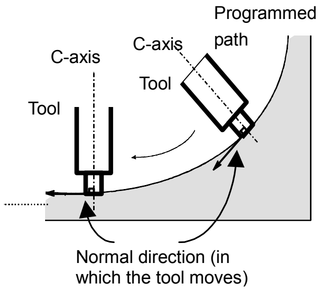
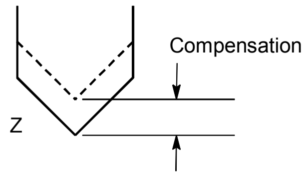
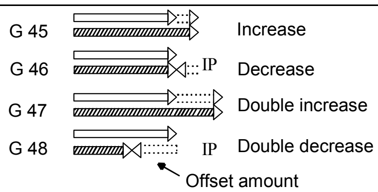
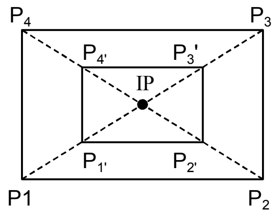
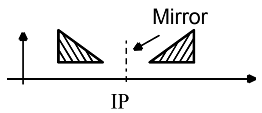
IP_ : presents a combination of arbitrary axis addresses using X, Y, Z, A, B, C, U, V, and W (such as X_ Y_ Z_A_).
α : One of the arbitrary addresses
β : One of the arbitrary addresses
Xp : X-axis or axis parallel to the X-axis
Yp : Y-axis or axis parallel to the Y-axis
Zp : Z-axis or axis parallel to the Z-axis

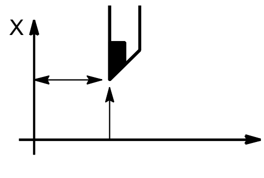
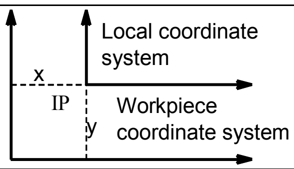
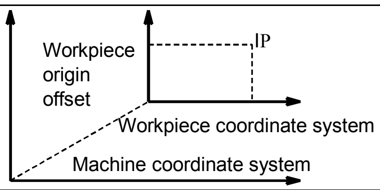
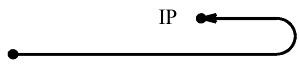
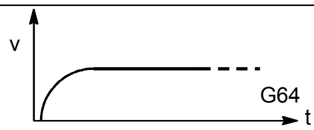
Functions	Illustration	Program format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		G17 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ X_ Y_ $\left\{\begin{matrix}R\_\\I\_\ J\_\end{matrix}\right\}$ F_ ; G18 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ X_ Z_ $\left\{\begin{matrix}R\_\\I\_\ K\_\end{matrix}\right\}$ F_ ; G19 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Y_ Z_ $\left\{\begin{matrix}R\_\\J\_\ K\_\end{matrix}\right\}$ F_ ;

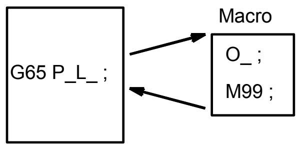
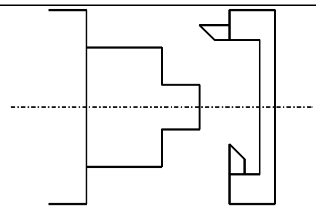
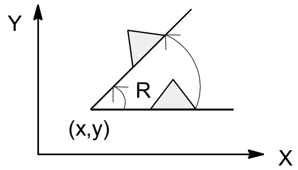
Functions	Illustration	Program format
Helical interpolation (G02, G03)	 In case of G03 on X-Y plane	$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Y\_ \begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix} \alpha\_\ F\_\ ;$ $G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Z\_ \begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix} \alpha\_\ F\_\ ;$ $G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_\ Z\_ \begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix} \alpha\_\ F\_\ ;$ α: Arbitrary address except the circular interpolation axis
Dwell (G04)		M $G04 \begin{Bmatrix} X\_ \\ P\_ \end{Bmatrix} ;$ T $G04 \begin{Bmatrix} X\_ \\ U\_ \\ P\_ \end{Bmatrix} ;$
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)	 	$G09 \begin{Bmatrix} G01 \\ G02 \\ G03 \end{Bmatrix} IP\_\ ;$

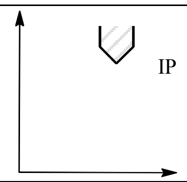
Functions	Illustration	Program format
Programmable data input (G10)		M Tool compensation memory A G10 L01 P_ R_ ; Tool compensation memory C G10 L10 P_ R_ ; (Geometry offset amount/H) G10 L11 P_ R_ ; (Wear offset amount/H) G10 L12 P_ R_ ; (Geometry offset amount/D) G10 L13 P_ R_ ; (Wear offset amount/D) 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
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.

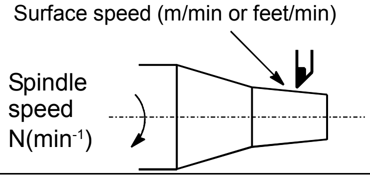
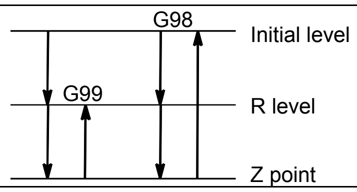
Functions	Illustration	Program format
M Movement from reference position (G29)		G29 IP_ ;
Skip function (G31)		G31 IP_ F_ ;
M Threading (G33) **T** Threading (G32)		**M** G33 IP_ F_ ; F : Lead **T** Equal lead threading G32 IP_ F_ ; F : Lead
T Variable lead thread cutting (G34)		G34 IP_ F_ K_ ; F : Lead 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** $\begin{Bmatrix} G17 \\ G18 \\ G19 \end{Bmatrix} \begin{Bmatrix} G41 \\ G42 \end{Bmatrix}$ D_ ; D : Tool compensation number G40 : Cancel **T** $\begin{Bmatrix} G41 \\ G42 \end{Bmatrix}$ IP_ ; G40 : Cancel

<table>
<tr><th>Functions</th><th>Illustration</th><th>Program format</th></tr>
<tr><td>M
Normal direction control
(G40.1, G41.1, G42.1)</td><td></td><td>G41.1 ; Normal direction control on : right
G42.1 ; Normal direction control on : left
G40.1 ; Normal direction control cancel</td></tr>
<tr><td>M
Tool length compensation
(G43, G44, G49)</td><td></td><td>$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ 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</td></tr>
<tr><td>Tool offset
(G45 to G48)</td><td></td><td>M
$\left\{\begin{matrix}G45\\G46\\G47\\G48\end{matrix}\right\}$ IP_ D_ ;
D : Tool offset number</td></tr>
<tr><td>M
Scaling
(G50, G51)</td><td></td><td>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</td></tr>
<tr><td>M
Programmable mirror image
(G50.1, G51.1)</td><td></td><td>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</td></tr>
<tr><td>T
Polygon turning
(G50.2, G51.2)
(G250, G251)</td><td></td><td>G51.2 (G251) P_ Q_ ; Polygon turning on
P_ Q_ : Rotation ratio between the spindle and rotation axis
G50.2 (G250) ; Polygon turning cancel</td></tr>
</table>

Functions	Illustration	Program format
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
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)		$\left\{\begin{matrix}G54\\ \vdots \\ G59\end{matrix}\right\}$ 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

Functions	Illustration	Program format
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
M Coordinate system rotation (G68, G69)	 In case of X-Y plane	G68 { G17 X_ Y_ / G18 Z_ X_ / G19 Y_ Z_ } R ; R: Angle of rotation in a counterclockwise direction G69 ; Cancel
Canned cycle for drilling M (G73, G74, G76, G80 to G89) T (G80 to G89)		G80 ; Cancel M G73 G74 G76 G81 : G89 } X_Y_Z_P_Q_R_F_K_ ;
T Canned cycle for turning Multiple repetitive canned cycle (G70 to G76) Canned cycle (G90, G92, G94)		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_ ; { G90 / G92 } X_ Z_ I_ F_ ; G94 X_ Z_ I_ F_ ;
Canned grinding cycle (for grinding machine) M (G75 to G79) T (G71 to G74)		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_ ;

<table>
<tr><th>Functions</th><th>Illustration</th><th>Program format</th></tr>
<tr><td></td><td></td><td>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></td><td colspan="2">Parameter EFX（No.7731#0）</td></tr><tr><td></td><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>
<tr><td>Absolute/incremental programming
(G90/G91)</td><td></td><td>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</td></tr>
<tr><td>Maximum incremental command value check (G91.1)</td><td></td><td>G91.1 IP_ ;
IP_ ; Maximum incremental value
Set 0 to cancel maximum incremental value check.</td></tr>
<tr><td>Change of workpiece coordinate system
or
Maximum spindle speed clamp (G92)</td><td></td><td>M
G92 IP_ ; Change of workpiece coordinate system
G92 S_ ; Constant surface speed control : Maximum spindle speed clamp</td></tr>
<tr><td>Workpiece coordinate system preset
M
(G92.1)
T
(G50.3)</td><td></td><td>M
G92.1 IP 0 ;
T
G50.3 IP 0 ;</td></tr>
<tr><td>M
Inverse time feed
(G93)</td><td></td><td>G93 ; Inverse time setting mode</td></tr>
<tr><td>Feed per minute,
Feed per revolution
M
(G94, G95)</td><td rowspan="2">mm/min inch/min
mm/rev inch/rev</td><td rowspan="2">M
G94 F_ ; Feed per minute
G95 F_ ; Feed per revolution
T
G98 F_ ; Feed per minute
G99 F_ ; Feed per revolution</td></tr>
<tr><td>T
(G98, G99)</td></tr>
</table>

Functions	Illustration	Program format
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)
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 ;

D RANGE OF COMMAND VALUE

Linear axis

- 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.

E NOMOGRAPHS

Appendix E, "NOMOGRAPHS", consists of the following sections:

E.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. E.1 (a), due to automatic acceleration and deceleration.

Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

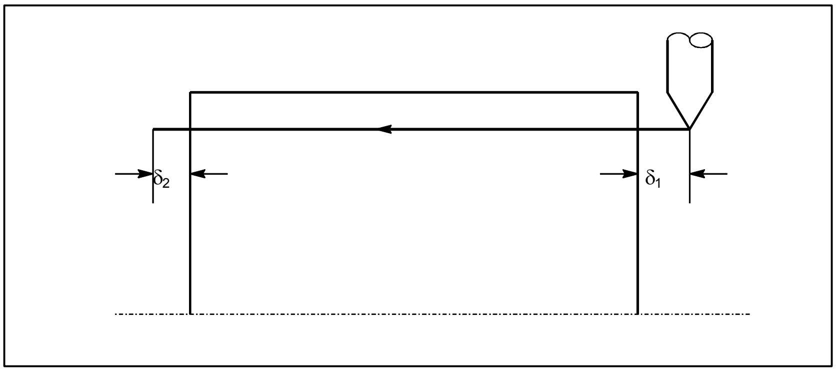


Fig. E.1 (a) Incorrect thread position

Explanation

- How to determine δ_2

$\delta_2 = T_1 V$ (mm). . . . (1)

$V = \frac{1}{60} RL$

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
R : Spindle speed (min^{-1})
L : Thread feed (mm)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

- How to determine δ_1

$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V$ . . . . . (2)

$a = \exp(-\frac{t}{T_1})$ . . . . . (3)

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

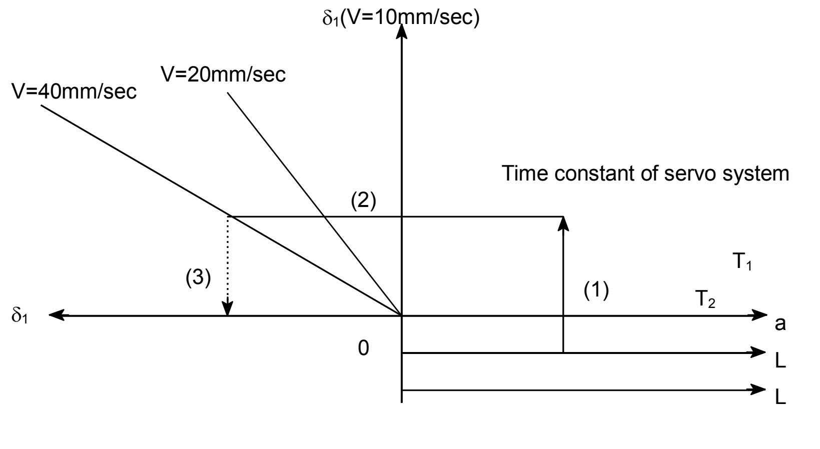
The lead at the beginning of thread cutting is shorter than the specified lead L, and the allowable lead error is ΔL. Then as follows.

$a = \frac{\Delta L}{L}$

When the value of “a” is determined, the time lapse until the thread accuracy is attained. The time “t” is substituted in (2) to determine δ_1: Constants V and T_1 are determined in the same way as for δ_2. Since the calculation of δ_1 is rather complex, a nomography is provided on the following pages.

- How to use nomograph

First specify the class and the lead of a thread. The thread accuracy, a, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm/s will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/s can be obtained at (3).



(Note) See the graph in reference later in the manual for an actual example.

Fig. E.1 (b) Nomograph

> **NOTE**
> The equations for δ_1, and δ_2 are for when the acceleration/ deceleration time constant for cutting feed is 0.

E.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

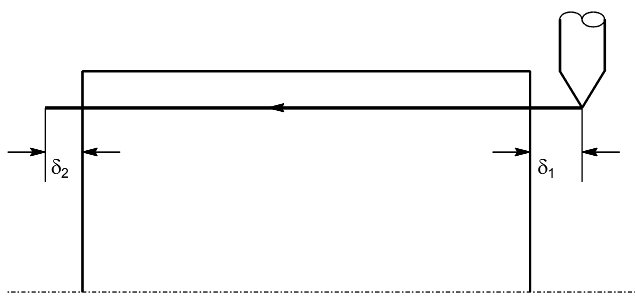


Fig. E.2 (a) Incorrect threaded portion

Explanation

- How to determine δ_2

$a=\frac{LR}{1800*}$(mm)

R : Spindle speed (min^{-1})
L : Thread lead (mm)
* When time constant T_1 of the servo system is 0.033 s.

- How to determine δ_1

$\delta_1 = \frac{LR}{1800*}(-1-\ln a)$ (mm)

$=\delta_2(-1-\ln a)$(mm)

R : Spindle speed (min^{-1})

L : Thread lead (mm)

* When time constant T_1 of the servo system is 0.033 s.

Following a is a permitted value of thread.

a	-1-lna
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

Example

R=350rpm

L=1mm

a=0.01

then

$\delta_2 = \frac{350 \times 1}{1800} = 0.194$(mm)

$\delta_1 = \delta_2 \times 3.605 = 0.701$(mm)

Reference

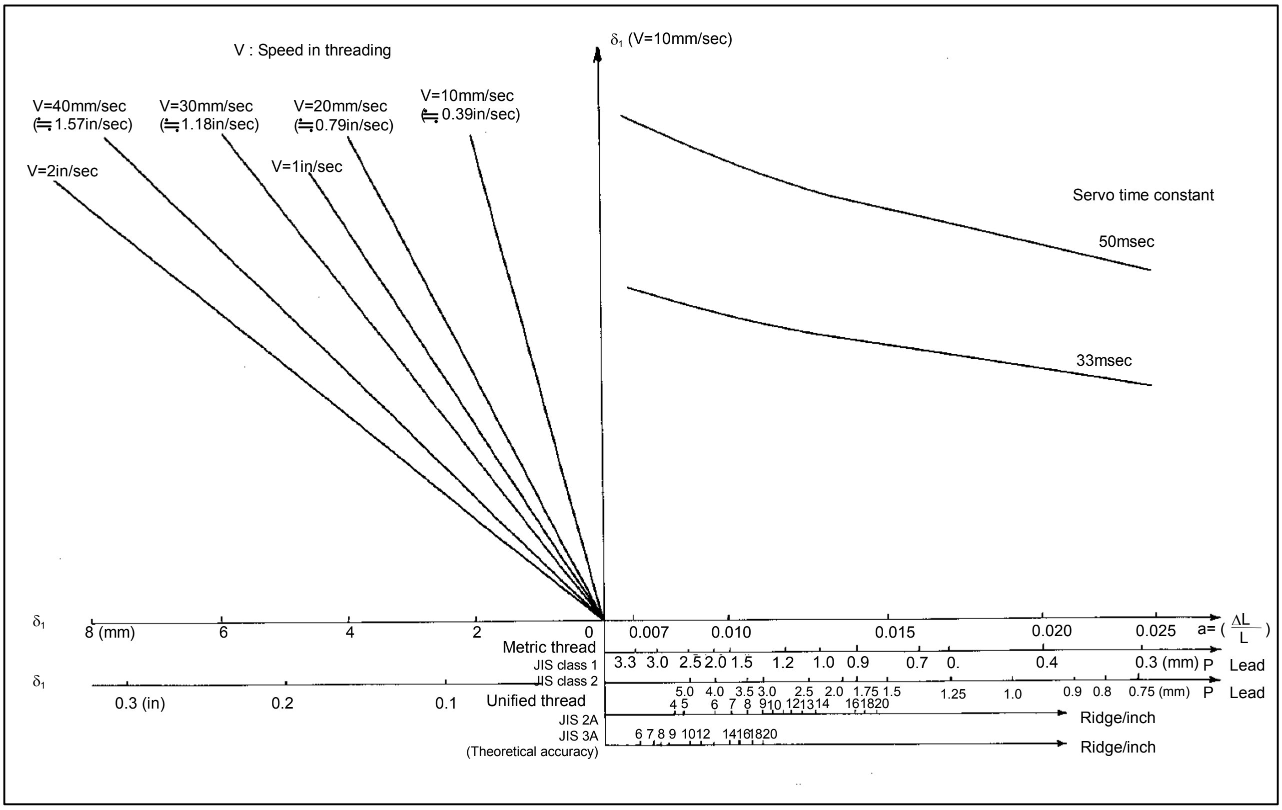


Fig. E.2 (b) Nomograph for obtaining approach distance δ_1

E.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. E.3 (a).

Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

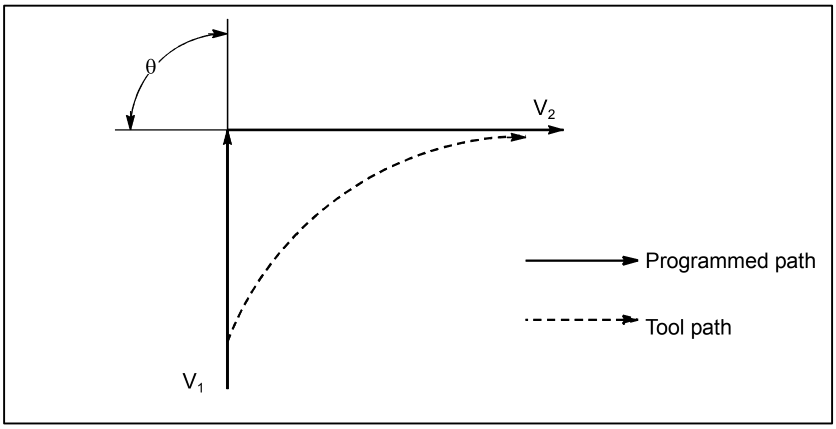


Fig. E.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:

- Feedrate (V_1, V_2)
- Corner angle (θ)
- Exponential acceleration / deceleration time constant (T_1) at cutting ($T_1 = 0$)
- Presence or absence of buffer register.

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example.

When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.

In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Explanation

- Analysis

The tool path shown in Fig. E.3 (b) is analyzed based on the following conditions:

- Feedrate is constant at both blocks before and after cornering.
- The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

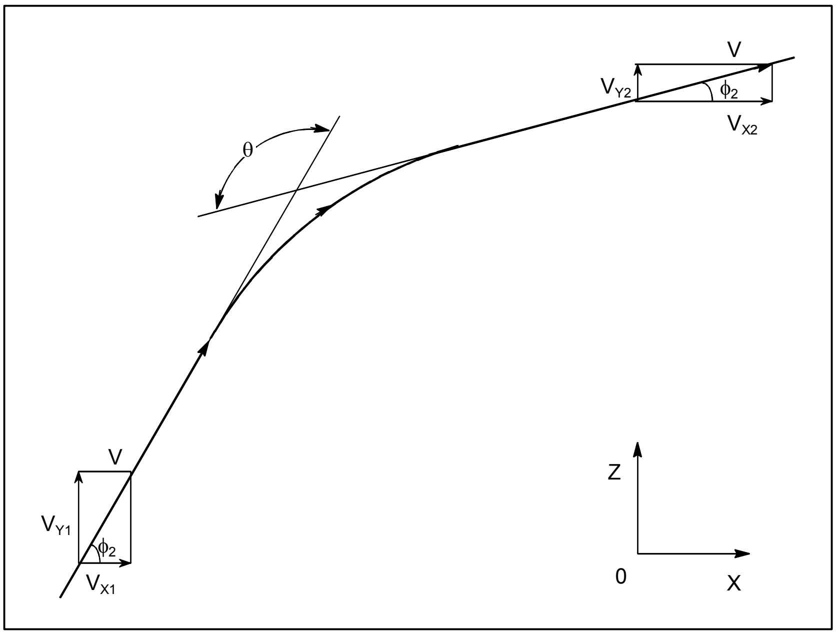


Fig. E.3 (b) Example of tool path

- Description of conditions and symbols

$V_{X1} = V\cos\phi_1$
$V_{Y1} = V\sin\phi_1$
$V_{X2} = V\cos\phi_2$
$V_{Y2} = V\sin\phi_2$

V : Feedrate at both blocks before and after cornering
V_{X1} : X-axis component of feedrate of preceding block
V_{Y1} : Y-axis component of feedrate of preceding block
V_{X2} : X-axis component of feedrate of following block
V_{Y2} : Y-axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and X-axis
ϕ_2 : Angle formed by specified path direction of following block and X-axis

- Initial value calculation

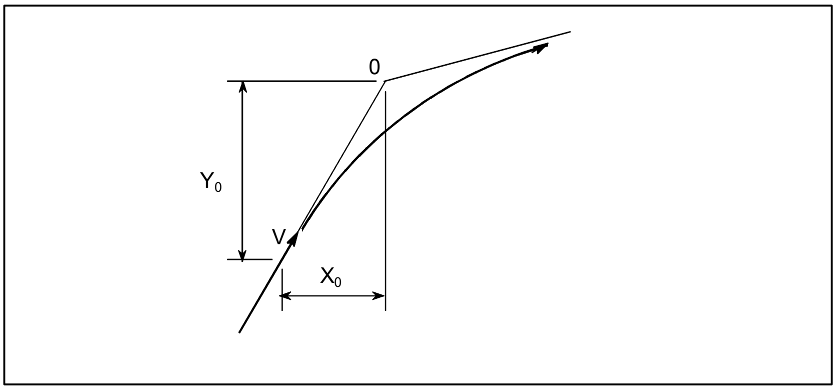


Fig. E.3 (c) Initial value

The initial value when cornering begins, that is, the X and Y coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

$$X_0 = V_{X1}(T_1 + T_2)$$

$$Y_0 = V_{Y1}(T_1 + T_2)$$

T_1: Exponential acceleration/deceleration time constant. (T=0)
T_2: Time constant of positioning system (Inverse of position loop gain)

- Analysis of corner tool path

The equations below represent the feedrate for the corner section in X-axis direction and Y-axis direction.

$$V_X^{(t)} = (V_{X2} - V_{X1})[1 - \frac{V_{X1}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{X1}]$$

$$= V_{X2}[1 - \frac{V_{X1}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\}]$$

$$V_Y^{(t)} = \frac{V_{Y1} - V_{Y2}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{Y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^1 V_X(t)dt - X_0$$

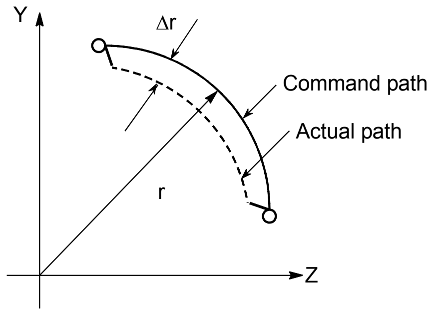
$$= \frac{V_{X2} - V_{X1}}{T_1 - T_2}\{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{X2}(T_1 + T_2 - t)$$

$$Y(t) = \int_0^1 V_Y(t)dt - Y_0$$

$$= \frac{V_{Y2} - V_{Y1}}{T_1 - T_2}\{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{Y2}(T_1 + T_2 - t)$$

E.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, specially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^{\,2} + T_2^{\,2}(1-\alpha^2))\frac{V^2}{r} \quad \ldots\ldots (1)$$

Δr: Maximum radius error (mm)
v : Feedrate (mm/sec)
r : Circle radius (mm)
T_1: Exponential acceleration/deceleration time constant at cutting (sec) (T=0)
T_2: Time constant of positioning system (sec)
(Inverse of position loop gain)
α : Feed forward coefficient (%)

In the case of bell-shaped acceleration/deceleration and linear acceleration/deceleration after cutting feed interpolation, an approximation of this radius error can be obtained with the following expression:

Linear acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{24}T_1^{\,2} + \frac{1}{2}T_2^{\,2}(1-\alpha^2))\frac{V^2}{r}$$

Bell-shaped acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{48}T_1^{\,2} + \frac{1}{2}T_2^{\,2}(1-\alpha^2))\frac{V^2}{r}$$

Thus, the radius error in the case of bell-shaped acceleration/deceleration and linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo loop time constant.

Since the machining radius r (mm) and allowable error Δr (mm) of the workpiece is given in actual machining, the allowable limit feedrate v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

F SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE

Either the clear state or reset state is entered during a reset is set by bit 6 (CLR) of parameter No. 3402 (0: reset state/1: clear state).
The symbols in the figure below have the following meanings.

○ : The setting remains unchanged or the operation continues.
× : The setting is deleted or the operation stops.

Item		Power-on	Clear state	Reset state
Settings	Offset	○	○	○
	Set data	○	○	○
	Parameter	○	○	○
Various data items	Program in memory	○	○	○
	Information of the block read in advance during automatic operation	×	×	×
	N (sequence number)	× (Note 7)	× (Notes 1, 2, 7)	○ (Note 1, 2)
	One-shot G code	×	×	×
	Modal G code	Initial value (Note 3) G20 and G21 are set to the state at the previous power-off.	Initial value (Note 3) G20 and G21 are not changed (Note 2).	○ (Note 2 and 6)
	F	0	0	○ (Note 2)
	S, T, M	×	○ (Note 2)	○ (Note 2)
	K (count specification)	×	×	×
Coordinate	Workpiece coordinates	0 (Note 4)	○	○
Operation during execution	Movement	×	×	×
	Dwell	×	×	×
	M, S, T code output	×	×	×
	Tool position compensation	×	See "Tool Position Compensation at a Reset".	See "Tool Position Compensation at a Reset".
	Tool length compensation	×	Depending on the setting of bit 6 (LVK) of parameter No. 5003	Depending on the setting of bit 6 (LVK) of parameter No. 5003
	Tool radius/tool nose radius compensation	×	×	×
	Storing a called subprogram	×	× (Note 5)	×

Item		Power-on	Clear state	Reset state
Output signals	CNC alarm signal AL	"0"(when no alarm cause is present)	“0” (when no alarm cause is present)	“0” (when no alarm cause is present)
	Reference position return completion signal ZPx	×	○ (or × for emergency stop)	○(or × for emergency stop)
	S, T, B codes	×	○	○
	M code	×	×	×
	M, S, T, B strobe signals	×	×	×
	Spindle rotation signal (S analog signal)	×	○	○
	CNC ready signal MA	“1”	○	○
	Servo ready signal SA	“1” (when a servo alarm does not occur)	“1” (when a servo alarm does not occur)	“1” (when a servo alarm does not occur)
	Cycle start signal STL	×	×	×
	Feed hold signal SPL	×	×	×

Tool position compensation at a reset

○ : Cancelled.

× : Not canceled.

Compensation method		Bit 3 (LVC) of parameter No. 5006 and bit 7 (TGC) of parameter No. 5003			
		LVC=0 TGC=0	LVC=1 TGC=0	LVC=0 TGC=1	LVC=1 TGC=1
Tool movement	Wear compensation	×	○ (Axis movement)	×	○ (Axis movement)
	Geometry compensation				
Coordinate shift	Wear compensation	×	○	×	○
	Geometry compensation	×	×	○	○

NOTES

1 When the beginning position is found, the main program number is displayed.

2 If a reset is made during execution of a block, the states of the modal G code and modal address (such as N, F, S, T, or M) specified in the block are not reflected. The modal information specified in the previous and preceding blocks is kept.

3 The initial value of the modal G code depends on the following parameters. For details, refer to the parameter manual (B-63950).
- Parameter G01 (No.3402#0)
- Parameter G18 (No.3402#1)
- Parameter G19 (No.3402#2)
- Parameter G91 (No.3402#3)
- Parameter FPM (No.3402#4)
- Parameter G23 (No.3402#7)
- Parameters No. 3406 to No. 3409

4 For detection of an absolute position, the value is generated from machine coordinates and a workpiece origin offset.

5 When a reset is made during execution of a subprogram, a return to the main program is performed. Execution at a midpoint in the subprogram is impossible.

6 When one of the following two settings, which hold the modal G code in group 1 by a reset is set:
- Reset state (bit 6 of parameter No. 3402 is 0)
- Clear state (bit 1 of parameter No. 3402 is 1) and
  the modal G code in group 1 is held at a time of a reset (bit 1 of parameter No. 3406 is 1),

if a reset is made during execution of one of the following canned cycles, which cause cycle operation, the mode of the modal G code in group 1 is changed to the G01 mode.
- G90 : Outer surface/inner surface turning cycle
- G92 : Threading cycle
- G94 : Edge turning cycle

7 When bit 0 (SEK) of parameter No. 11353 is set to 1, sequence number N can be held even at power-on or in the clear state.

G CHARACTER-TO-CODES CORRESPONDENCE TABLE

Appendix G, "CHARACTER-TO-CODES CORRESPONDENCE TABLE", consists of the following sections:

G.1 CHARACTER-TO-CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		”	034	Quotation mark
H	072		#	035	Sharp
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		-	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	At mark
1	049		[	091	Left square bracket
2	050		]	093	Right square bracket
3	051		^	094	
4	052		_	095	Underscore
5	053				

G.2 FANUC DOUBLE-BYTE CHARACTER CODE TABLE

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0200	ぁ	あ	ぃ	い	ぅ	う	ぇ	え	ぉ	お	か	が	き	ぎ	く	ぐ
0220	け	げ	こ	ご	さ	ざ	し	じ	す	ず	せ	ぜ	そ	ぞ	た	だ
0240	ち	ぢ	っ	つ	づ	て	で	と	ど	な	に	ぬ	ね	の	は	ば
0260	ぱ	ひ	び	ぴ	ふ	ぶ	ぷ	へ	べ	ぺ	ほ	ぼ	ぽ	ま	み	む
0280	め	も	ゃ	や	ゅ	ゆ	ょ	よ	ら	り	る	れ	ろ	ゎ	わ	素
02A0	材	を	ん	種	類	棒	穴	成	形	質	寸	法	外	径	長	端
02C0	面	最	小	内	大	加	工	切	削	倣	正	途	中	荒	具	番
02E0	号	仕	上	込	点	方	向	速	度	送	量	開	始	深	主	軸
0300	回	転	数	位	置	決	直	線	時	円	反	現	在	指	令	値
0320	領	域	診	断	操	作	手	引	機	械	残	移	動	次	早	電
0340	源	投	入	間	分	秒	自	運	負	荷	実	使	用	寿	命	新
0360	規	除	隅	取	単	補	能	独	終	了	記	角	溝	刃	幅	広
0380	設	定	一	覧	表	部	炭	合	金	鋼	超	硬	先	付	摩	耗
03A0	仮	想	副	行	挿	消	去	山	高	準	備	完	後	弧	助	択
03C0	無	視	器	原	登	録	再	処	理	描	画	過	容	編	集	未
03E0	対	相	座	標	示	名	歯	変	呼	推	馬	力	系	選	達	閉

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0400	禁	復	帰	書	個	桁	稼	由	両	半	逃	底	逆	下	空	四
0420	触	平	代	辺	格	子	周	心	本	群	停	止	巾	微	状	路
0440	範	囲	倍	率	注	側	特	殊	距	離	連	続	増	隔	件	初
0460	期	条	経	握	圧	扱	陰	隠	右	押	横	黄	億	屋	化	何
0480	絵	階	概	該	巻	換	気	起	軌	技	疑	供	共	境	強	教
04A0	掘	繰	係	傾	型	検	権	研	肩	見	験	元	弦	減	孔	巧
04C0	控	更	校	構	根	左	差	雑	参	散	産	算	治	耳	式	失
04E0	修	十	従	勝	商	少	尚	昇	植	色	食	伸	信	侵	振	浸
0500	真	暗	以	意	異	影	鋭	越	価	可	科	果	箇	課	各	拡
0520	核	学	掛	漢	簡	観	関	含	却	客	休	急	業	曲	均	筋
0540	継	計	軽	言	限	互	降	採	済	細	姿	思	写	射	斜	者
0560	車	借	縦	重	出	述	術	渉	照	省	章	証	象	身	進	人
0580	図	違	印	沿	遠	央	奥	往	応	会	解	改	割	活	願	基
05A0	奇	寄	岐	既	近	区	矩	駆	偶	旧	求	球	究	級	欠	結
05C0	口	語	誤	交	厚	項	刻	告	黒	財	策	糸	試	資	事	持
05E0	似	釈	弱	受	収	純	順	所	序	剰	場	常	飾	水	錐	据
0600	制	整	製	前	全	然	則	属	即	他	多	存	谷	探	短	徴
0620	鎖	調	頂	鉄	添	頭	同	導	道	熱	年	濃	箱	発	抜	伴
0640	必	百	複	物	文	間	併	忘	末	密	有	余	与	裏	立	略
0660	青	席	石	積	赤	接	折	粗	創	双	捜	太	打	体	待	態
0680	替	段	知	地	致	遅	追	通	伝	得	読	凸	凹	突	鈍	敗
06A0	杯	背	配	品	不	布	並	頁	別	片	返	勉	弁	保	明	滅
06C0	木	目	歪	揺	様	溶	要	抑	良	輪	和	話	枠	節	説	絶
06E0	千	専	浅	旋	総	走	退	台	第	題	卓	室	着	柱	鋳	丁
0700	低	訂	肉	日	白	薄	比	皮	被	非	美	普	伏	歩	包	門
0720	問	絡	列	万	利	訳	礼	乱	放	枚	約	練	油	劣	例	郭
0740	戻	冷	垂	緑	紫	許	測	精	効	→	↗	↑	↖	←	↙	↓
0760	↘	↻	↺	⌒	◯	▪	板	予	〃	家	装	管	粉	等	▿	▿▿
0780	▿▿▿	∿∿∿	貫	安	α	β	程	抗	張	任	破	損	御	足	守	般
07A0	納	義	丸	汎	固	毎	当	的	詳	鳥	適	論	額	縁	温	給
07C0	界	混	監	締	護	己	称	樹	脂	料	落	確	認	報	排	性
07E0	生	績	判	搬	砥	θ	島	壁						//	)(	■■

	00		02		04		06		08		0A		0C		0E		10		12		14		16		18		1A		1C		1E	
0800	阿		哀		愛		挨		逢		悪		旭		宛		案		闇		鞍		伊		依		偉		委		威	
0820	尉		易		為		維		緯		胃		衣		遺		医		井		育		一		稲		員		因		飲	
0840	院		羽		雨		渦		唄		浦		瓜		噂		雲		営		映		栄		永		泳		洩		英	
0860	衛		液		益		駅		園		延		援		演		炎		煙		鉛		塩		汚		王		岡		沖	
0880	憶		臆		牡		乙		恩		穏		音		佳		夏		暇		架		歌		河		火		花		華	
08A0	菓		貨		我		牙		芽		賀		雅		介		壊		廻		快		怪		海		灰		皆		貝	
08C0	劾		害		慨		街		垣		殻		獲		覚		較		革		楽		笠		括		滑		株		刈	
08E0	乾		寒		刊		勧		喚		官		寛		干		幹		感		慣		敢		歓		汗		環		甘	
0900	看		緩		缶		肝		還		鑑		閑		陥		韓		館		岸		眼		岩		顔		企		危	
0920	喜		希		幾		揮		机		旗		棄		毅		祈		季		稀		徽		輝		騎		擬		犠	
0940	議		菊		詰		脚		丘		久		及		吸		宮		弓		救		泣		牛		居		巨		拒	
0960	挙		虚		魚		亨		享		京		競		協		叫		挟		橋		況		狭		胸		興		郷	
0980	鏡		響		驚		凝		局		極		玉		勤		錦		琴		銀		九		句		苦		駈		屑	
09A0	屈		熊		君		訓		軍		郡		刑		兄		契		掲		敬		景		茎		警		芸		迎	
09C0	劇		激		隙		潔		血		月		倹		健		兼		券		剣		圏		堅		建		憲		拳	
09E0	犬		献		絹		県		謙		軒		鍵		険		幻		古		庫		戸		故		湖		狐		誇	
0A00	顧		五		午		侯		候		光		公		勾		喉		好		孝		幸		康		弘		拘		攻	
0A20	江		港		甲		稿		絞		綱		考		肯		衡		講		購		郊		鉱		香		剛		克	
0A40	国		穀		酷		腰		骨		此		頃		今		困		婚		査		砂		債		妻		彩		才	
0A60	災		砕		祭		菜		裁		載		際		剤		罪		坂		阪		咲		昨		索		錯		桜	
0A80	冊		刷		察		撮		擦		札		殺		皿		三		撒		讃		賛		酸		伺		刺		史	
0AA0	士		姉		市		師		支		枝		死		私		紙		詞		詩		字		寺		磁		辞		七	
0AC0	湿		芝		縞		捨		煮		社		謝		尺		若		酒		首		授		需		秋		習		臭	
0AE0	舟		週		住		柔		宿		祝		縮		熟		春		瞬		盾		巡		暑		女		傷		唱	
0B00	奨		将		床		承		招		昭		焼		焦		笑		紹		衝		賞		障		乗		城		情	
0B20	浄		蒸		錠		職		唇		寝		審		森		申		神		芯		親		辛		針		震		尽	
0B40	陣		須		酢		吹		粋		遂		杉		裾		澄		世		是		勢		征		政		星		晴	
0B60	清		盛		聖		声		西		誓		請		静		税		昔		析		籍		責		跡		雪		舌	
0B80	占		宣		尖		川		戦		扇		栓		泉		洗		染		潜		船		銑		鮮		善		組	
0BA0	訴		倉		層		掃		巣		争		窓		草		騒		像		臓		蔵		贈		造		促		息	
0BC0	束		俗		卒		其		揃		尊		村		詑		堕		妥		耐		帯		怠		滞		袋		貸	
0BE0	隊		滝		宅		拓		濯		託		濁		奪		脱		棚		誰		嘆		担		淡		団		弾	

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0C00	暖	男	談	池	築	畜	竹	筑	秩	茶	昼	虫	駐	貯	帳	庁
0C20	彫	挑	朝	町	脹	腸	跳	沈	珍	賃	墜	痛	塚	爪	吊	釣
0C40	庭	廷	提	釘	泥	摘	滴	笛	典	天	展	店	貼	殿	田	吐
0C60	塗	徒	都	砥	努	土	怒	倒	冬	凍	刀	島	東	湯	灯	答
0C80	筒	統	到	藤	討	踏	透	働	堂	胴	銅	峠	徳	毒	届	曇
0CA0	謎	鍋	縄	南	軟	難	二	匂	乳	尿	念	燃	粘	悩	脳	農
0CC0	把	波	派	廃	拝	肺	買	売	博	拍	泊	舶	麦	肌	畑	八
0CE0	罰	版	犯	班	繁	販	飯	盤	否	彼	悲	扉	批	疲	秘	肥
0D00	費	避	飛	尾	鼻	菱	筆	俵	氷	票	評	病	浜	貧	敏	夫
0D20	婦	富	怖	浮	父	符	腐	武	舞	封	風	服	福	腹	払	沸
0D40	噴	憤	奮	紛	丙	兵	幣	柄	米	壁	癖	偏	便	捕	募	墓
0D60	母	簿	宝	崩	捧	泡	胞	芳	訪	豊	飽	亡	傍	剖	妨	帽
0D80	忙	房	暴	望	紡	肪	膨	防	北	僕	撲	釦	没	翻	磨	魔
0DA0	幕	膜	迄	満	味	魅	脈	妙	民	務	夢	矛	迷	鳴	免	綿
0DC0	模	茂	毛	盲	網	黙	紋	冶	夜	野	矢	役	薬	躍	諭	輸
0DE0	優	友	遊	郵	融	誉	預	幼	揚	曜	洋	葉	陽	養	浴	翼
0E00	螺	来	頼	欄	陸	律	流	留	粒	旅	療	稜	林	臨	隣	涙
0E20	累	励	鈴	暦	歴	烈	裂	労	漏	老	六	脇	惑	詫	湾	腕
0E40	斡	椅	萎	宇	嘘	閲	宴	欧	懐	拐	涯	穫	闇	潟	渇	冠
0E60	患	汽	貴	鬼	偽	戯	欺	喫	窮	糾	拠	漁	恐	狂	脅	仰
0E80	緊	愚	遇	靴	啓	慶	憩	携	撃	傑	嫌	懸	厳	雇	娯	洪
0EA0	紅	耕	航	貢	挫	催	栽	崎	柵	拶	傘	志	施	旨	至	誌
0EC0	識	狩	趣	就	秀	衆	襲	蹴	充	渋	緒	署	諸	叙	掌	訟
0EE0	鐘	壌	織	紳	酔	瀬	誠	繊	漸	繕	塑	礎	阻	奏	族	惰
0F00	戴	諾	叩	旦	誕	恥	仲	宙	忠	抽	兆	懲	抵	敵	撤	党
0F20	盗	糖	陶	闘	督	馴	罰	媒	爆	縛	髪	閥	泌	匹	府	敷
0F40	仏	慕	縫	乏	霧	盟	勇	誘	踊	裸	雷	卵	里	隆	慮	虜
0F60	寮	塁	隷	霊	恋	浪	郎	功	坑	々	⇧					
0F80	`a	bc	de	fg	hi	jk	lm	no	pq	rs	tu	vw	xy	z{	¦}	~
0FA0	БГ	ДЖ	ЗИ	ЙК	ЛМ	ПУ	ФЦ	ЧШ	ЩЪ	ЫЬ	ЭЮ	Я	?			
0FC0	ÀÁ	ÃÂ	ÆÇ	ÈÉ	ÊË	ÌÍ	ÎÏ	ÒÓ	ÔÕ	ŒØ	ÙÚ	ÛŸ	ßà	áâ	ãä	åæ
0FE0	ÅÄ	ÖÜ	Ñ¿	çè	éê	ëì	íî	ïñ	òó	ôõ	öœ	øù	úû	üÿ	¡	

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
1000	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1020	Q	R	S	T	U	V	W	X	Y	Z	a	b	c	d	e	f
1040	g	h	i	j	k	l	m	n	o	p	q	r	s	t	u	v
1060	w	x	y	z	//				γ	ε	μ	π	φ	ω	Δ	Σ
1080	Ω								ｶﾞｷﾞ	ｸﾞｹﾞ	ｺﾞｻﾞ	ｼﾞｽﾞ	ｾﾞｿﾞ	ﾀﾞﾁﾞ	ﾂﾞﾃﾞ	ﾄﾞ
10A0	ﾊﾞﾋﾞ	ﾌﾞﾍﾞ	ﾎﾞﾊﾟ	ﾋﾟﾌﾟ	ﾍﾟﾎﾟ	ｳﾞヵ	ヶ		ァ	ア	ィ	イ	ゥ	ウ	ェ	エ
10C0	ォ	オ	カ	ガ	キ	ギ	ク	グ	ケ	ゲ	コ	ゴ	サ	ザ	シ	ジ
10E0	ス	ズ	セ	ゼ	ソ	ゾ	タ	ダ	チ	ヂ	ッ	ツ	ヅ	テ	デ	ト
1100	ド	ナ	ニ	ヌ	ネ	ノ	ハ	バ	パ	ヒ	ビ	ピ	フ	ブ	プ	ヘ
1120	ベ	ペ	ホ	ボ	ポ	マ	ミ	ム	メ	モ	ャ	ヤ	ュ	ユ	ョ	ヨ
1140	ラ	リ	ル	レ	ロ	ヮ	ワ	ヲ	ン	ヴ	ヵ	ヶ				
1160		、	。	，	．	・	：	；	？	！	゛	゜	´	｀	¨	＾
1180	￣	＿	ヽ	ヾ	ゝ	ゞ	〃	仝	々	〆	〇	ー	―	‐	／	＼
11A0	～	∥	｜	…	‥	‘	’	“	”	（	）	〔	〕	［	］	｛
11C0	｝	〈	〉	《	》	「	」	『	』	【	】	＋	－	±	×	÷
11E0	＝	≠	＜	＞	≦	≧	∞	∴	♂	♀	°	′	″	℃	￥	＄
1200	¢	£	％	＃	＆	＊	＠	§	☆	★	○	●	◎	◇	◆	□
1220	■	△	▲	▽	▼	※	〒	→	←	↑	↓	=	▢			
1240	1/1	2/2	3/3	4/4	5/5	6/6			mm	cm	km	cm²	m²	km²	cm³	m³
1260	mg	kg	cc	dℓ	ℓ	kℓ	ms	μs	ns	HP	ps	Hz	㈱	©		
1280	亜	芦	尉	壱	逸	芋	姻	韻	詠	疫	悦	謁	猿	殴	翁	虞
12A0	卸	嫁	禍	彦	蚊	餓	悔	塊	戒	嚇	岳	樫	喝	褐	轄	且
12C0	勘	堪	棺	款	憾	艦	頑	忌	紀	飢	棋	宜	儀	吉	虐	朽
12E0	凶	峡	恭	矯	暁	斤	桐	菌	謹	襟	吟	隈	勲	薫	恵	渓
1300	蛍	鶏	鯨	遣	賢	繭	顕	玄	孤	枯	鼓	呉	悟	碁	后	恒
1320	皇	慌	酵	拷	豪	獄	昆	恨	紺	魂	墾	懇	佐	唆	詐	宰
1340	斎	歳	搾	桟	蚕	惨	暫	氏	司	祉	肢	嗣	飼	雌	賜	諮
1360	児	侍	滋	慈	璽	疾	執	漆	舎	赦	遮	邪	蛇	勺	酌	爵
1380	寂	朱	珠	儒	囚	州	宗	拾	愁	酬	醜	汁	銃	獣	叔	淑
13A0	粛	塾	俊	旬	准	殉	循	潤	遵	庶	如	徐	升	召	匠	肖
13C0	償	抄	昌	晶	松	沼	宵	症	祥	硝	粧	詔	彰	礁	丈	冗
13E0	畳	嬢	譲	醸	殖	嘱	辱	臣	娠	慎	薪	仁	迅	甚	尋	炊

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
1400	帥	衰	睡	穂	錘	随	髄	枢	崇	菅	畝	姓	斉	牲	逝	婿
1420	脆	夕	斥	隻	惜	拙	窃	摂	仙	践	銭	遷	薦	禅	祖	租
1440	措	疎	壮	荘	桑	曹	喪	葬	僧	遭	槽	燥	藻	霜	憎	賊
1460	孫	駄	舵	胎	泰	逮	瀧	沢	但	丹	胆	鍛	壇	痴	稚	畜
1480	逐	窒	嫡	衷	著	弔	眺	潮	聴	勅	朕	陳	鎮	津	漬	坪
14A0	呈	弟	邸	亭	貞	帝	艇	逓	偵	提	迭	哲	徹	澱	斗	渡
14C0	奴	桃	悼	棟	痘	唐	塔	搭	謄	豆	騰	洞	童	匿	篤	屯
14E0	豚	尼	弐	妊	忍	寧	婆	俳	輩	梅	培	陪	賠	伯	迫	漠
1500	鉢	伐	帆	畔	煩	頒	藩	晩	蛮	妃	披	卑	碑	罷	姫	漂
1520	苗	猫	賓	頻	瓶	扶	赴	膚	賦	附	譜	侮	覆	雰	墳	陛
1540	塀	弊	遍	舗	暮	邦	奉	峰	抱	俸	砲	褒	坊	某	冒	貿
1560	謀	朴	牧	墨	堀	奔	凡	盆	麻	妹	埋	又	抹	慢	漫	岬
1580	眠	娘	銘	妄	猛	匁	厄	愉	癒	唯	幽	悠	猶	裕	雄	憂
15A0	羊	庸	窯	擁	謡	翌	羅	酪	濫	吏	痢	履	柳	竜	硫	涼
15C0	猟	陵	僚	糧	厘	倫	零	齢	麗	廉	錬	炉	露	朗	廊	楼
15E0	賄	國	揺	條	櫻	澤	濾	碌	緞	鐵	靱	靖	槻	浩	郁	

H ALARM LIST

Appendix H, "ALARM LIST", consists of the following items:

(1) Alarms on program and operation (PS alarm)
(2) Background edit alarms (BG alarm)
(3) Communication alarms (SR alarm)

Alarm numbers are common to all these alarm types.
Depending on the state, an alarm is displayed as in the following examples:
PS"alarm number" Example: PS0003
BG"alarm number" Example: BG0085
SR"alarm number" Example: SR0001

Number	Message	Description
0001	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
0002	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting TVC parameter No. 0000#0 to "0".
0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
0004	ADDRESS NOT FOUND	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0005	NO DATA AFTER ADDRESS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0006	ILLEGAL USE OF MINUS SIGN	A minus sign (–) was specified at an NC instruction word or system variable where no minus signal may be specified.
0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two decimal points were specified.
0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.
0010	IMPROPER G-CODE	An unusable G code is specified.

Number	Message	Description
0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead.
0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either divide the number of programmed move axes into two blocks.
0020	OVER TOLERANCE OF RADIUS	An arc was specified for which the difference in the radius at the start and end points exceeds the value set in parameter No. 3410. Check arc center codes I, J and K in the program. The tool path when parameter No. 3410 is set to a large value is spiral.
0021	ILLEGAL PLANE SELECT	The plane selection instructions G17 to G19 are in error. Reprogram so that same 3 basic parallel axes are not specified simultaneously. This alarm is also generated when an axis that should not be specified for plane machining is specified, for example, for circular interpolation. T In the 0*i* -TD, the helical interpolation option is needed to enable the specification of 3 or more axes for the G02/G03 block.
0022	R OR I,J,K COMMAND NOT FOUND	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.
0023	ILLEGAL RADIUS COMMAND	T A negative value is specified for the arc radius R command. In the T series, an arc at an angle larger than 180° cannot be specified with the R command. Modify the program.
0025	CIRCLE CUT IN RAPID (F0)	M F0 (rapid traverse in one-digit F code feed or inverse feed) was specified during circular interpolation (G02, G03).
0027	NO AXES COMMANDED IN G43/G44	M No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Multiple axes were specified for the same block when the tool length compensation type is C.
0028	ILLEGAL PLANE SELECT	The plane selection instructions G17 to G19 are in error. Reprogram so that same 3 basic parallel axes are not specified simultaneously. This alarm is also generated when an axis that should not be specified for plane machining is specified, for example, for circular interpolation. T In the 0*i* -TD, the helical interpolation option is needed to enable the specification of 3 or more axes for the G02/G03 block.
0029	ILLEGAL OFFSET VALUE	Illegal offset No.
0030	ILLEGAL OFFSET NUMBER	An illegal offset No. was specified.

Number	Message	Description
0031	ILLEGAL P COMMAND IN G10	Data input for the L No. of G10 or the corresponding function is not enabled. A data setting address such as P or R is not specified. An address command not concerned with data setting was specified. An address varies with the L No. The sign or decimal point of the specified address is in error, or the specified address is out of range.
0032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
0033	NO INTERSECTION AT G41/G42	The intersection cannot be obtained by the intersection calculation in tool radius/tool nose radius compensation. Modify the program.
0034	ONLY G00/G01 ALLOWED IN STUP/EXT BLK	In tool radius/tool nose radius compensation, a startup or cancellation is performed when the mode is not G00/G01. Modify the program.
0035	CAN NOT COMMANDED G31	1) G31 cannot be specified. This alarm is generated when a G code (such as for tool radius/tool nose radius compensation) of group 07 is not canceled. 2) A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like.
0037	CAN NOT CHANGE PLANE IN G41/G42	The compensation plane G17/G18/G19 was changed during cutter or tool-nose radius compensation. Modify the program.
0038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool radius/tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
0039	CHF/CNR NOT ALLOWED IN G41,G42	T Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in G41 and G42 commands (tool nose radius compensation). The program may cause overcutting to occur in chamfering or corner R. Modify the program.
0041	INTERFERENCE IN G41/G42	In tool radius/tool nose radius compensation, excessive cutting may occur. Modify the program.
0042	G45/G48 NOT ALLOWED IN CRC	M Tool offset (G45 to G48) is commanded in tool radius compensation mode. Modify the program.
0044	G27-G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 (G29 is only for the M series) is commanded in canned cycle mode. Modify the program.
0045	ADDRESS Q NOT FOUND (G73/G83)	In a high-speed peck drilling cycle or peck drilling cycle, the amount of each-time cutting is not specified by address Q, or Q0 is specified. Modify the program.
0046	ILLEGAL REFERENCE RETURN COMMAND	A command for a return to the second, third or fourth reference position is error. (The address P command is in error.)
0050	CHF/CNR NOT ALLOWED IN THRD BLK	The (optional) chamfering or corner R block is specified in a threading block. Modify the program.
0051	MISSING MOVE AFTER CNR/CHF	The travel or travel distance is incorrect in the block next to the (optional) chamfering or corner R. Modify the program.
0052	CODE IS NOT G01 AFTER CHF/CNR	T The block next to the chamfering or corner R block is not G01 (or vertical line). Modify the program.
0053	TOO MANY ADDRESS COMMANDS	T In the chamfering and corner R commands, two or more of I, J, K and R are specified.

Number	Message	Description
0054	NO TAPER ALLOWED AFTER CHF/CNR	T A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
0055	MISSING MOVE VALUE IN CHF/CNR	The travel distance specified in the (optional) chamfering or corner R block is smaller than the amount of the (optional) chamfering or corner R. Modify the program.
0056	NO END POINT & ANGLE IN CHF/CNR	T In direct dimension drawing programming, both an end point and an angle were specified in the block next to the block in which only an angle was specified (Aa). Modify the program.
0057	NO SOLUTION OF BLOCK END	T Block end point is not calculated correctly in direct dimension drawing programming. Modify the program.
0058	END POINT NOT FOUND	T Block end point is not found in direct dimension drawing programming. Modify the program.
0060	SEQUENCE NUMBER NOT FOUND	[External data input/output] The specified number could not be found for program number and sequence number searches. A request was issued for input/output of an offset amount for tool data, but a tool number has never been entered after power-up. The tool data corresponding to the entered tool number could not be found. [External workpiece number search] The program corresponding to the specified workpiece number could not be found. [Program restart] In the program restart sequence number specification, the specified sequence number could not be found.
0061	P OR Q COMMAND IS NOT IN THE MULTIPLE REPETIVE CYCLES BLOCK	T Address P or Q is not specified in multiple repetitive cycle (G70, G71, G72, or G73) command.
0062	THE CUTTING AMOUNT IS ILLEGAL IN THE ROUGH CUTTING CYCLE	T A zero or a negative value was specified in a multiple repetitive canned rough-cutting cycle (G71 or G72) as the depth of cut.
0063	THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND	T The sequence number specified by addresses P and Q in multiple repetitive cycle (G70, G71, G72, or G73) command cannot be searched.
0064	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)	T In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command for the first plane axis was not a monotonous increase or decrease.
0065	G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM	T In the first block of the shape program specified by P of the multiple repetitive canned cycle (G70, G71, G72, or G73), G00 or G01 was not specified.
0066	UNAVAILABLE COMMAND IS IN THE MULTIPLE REPETIVE CYCLES BLOCK	T An unavailable command was found in a multiple repetitive canned cycle (G70, G71, G72, or G73) command block.
0067	THE MULTIPLE REPETIVE CYCLES IS NOT IN THE PART PROGRAM STORAGE	T A multiple repetitive canned cycle (G70, G71, G72, or G73) command is not registered in a tape memory area.

Number	Message	Description
0069	LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND	T In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle.
0070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
0071	DATA NOT FOUND	1) The address to be searched was not found. 2) The program with specified program number was not found in program number search. 3) In the program restart block number specification, the specified block number could not be found. Check the data.
0072	DATA NOT FOUND	The number of programs to be stored exceeded 400 (1-path system) or 800 (2-path system of T series). Delete unnecessary programs and execute program registration again.
0073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
0074	PROGRAM NUMBER ALREADY IN USE	The program number is other than 1 to 9999. Modify the program number.
0075	PROTECT	An attempt was made to register a program whose number was protected. In program matching, the password for the encoded program was not correct. An attempt was made to select a program being edited in the background as the main program. An attempt was made to call a program being edited in the background as a subprogram.
0076	PROGRAM NOT FOUND	The specified program is not found in the subprogram call or macro call. The M, G, or T codes are called by a P instruction other than that in an M98, M198, G65, G66, or interrupt type custom macro, and a program is called by a specific address. This alarm is also generated when a program is not found by these calls.
0077	TOO MANY SUB,MACRO NESTING	The total number of subprogram and macro calls exceeds the permissible range. Another subprogram call was executed during an external memory subprogram call.
0078	SEQUENCE NUMBER NOT FOUND	The specified sequence No. was not found during sequence number search. The sequence No. specified as the jump destination in GOTO— and M99P— was not found.
0079	PROGRAM NOT MATCH	The program in memory does not match the program stored on tape. Multiple programs cannot be matched continuously when parameter No. 2200#3 is set to “1”. Set parameter No. 2200#3 to “0” before executing a match.

Number	Message	Description
0080	G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT	**M** When the tool length measurement function (G37) is performed, a measuring position reached signal goes 1 in front of the area determined by the ε value specified in parameter No.6254. Alternatively, the signal does not go 1. **T** When the automatic tool compensation function (G36, G37) is used, a measuring position reached signals (XAE1, XAE2) does not go 1 within the range determined by the ε value specified in parameters No.6254 and No.6255.
0081	G37 OFFSET NO. UNASSIGNED	**M** When the tool length measurement function is performed, the tool length measurement function (G37) is specified without specifying an H code. Correct the program. **T** When the automatic tool compensation function is used, the automatic tool compensation function (G36, G37) is specified without specifying an T code. Correct the program.
0082	G37 SPECIFIED WITH H CODE	**M** When the tool length measurement function is performed, the tool length measurement function (G37) is specified together with an H code in the same block. Correct the program. **T** When the automatic tool compensation function is used, the automatic tool compensation function (G36, G37) is specified together with an T code in the same block. Correct the program.
0083	G37 IMPROPER AXIS COMMAND	**M** An error has been found in axis specification of the tool length measurement function (G37). Alternatively, a move command is specified as an incremental command. Correct the program. **T** An error has been found in axis specification of the automatic tool compensation function (G36, G37). Alternatively, a command is specified as an incremental command. Correct the program.
0085	OVERRUN ERROR	The next character was received from the I/O device connected to reader/punch interface 1 before it could read a previously received character. An overrun, parity error, or framing error occurred during the reading by reader/puncher interface 1. The number of bits in the entered data, the baud rate setting, or the I/O unit specification number is incorrect.
0086	DR OFF	During I/O process by reader/punch interface 1, the data set ready input signal of the I/O device (DR) was OFF. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
0087	BUFFER OVERFLOW	During a read by reader/punch interface 1, although a read stop command was issued, more than 10 characters were input. The I/O device or printed circuit board was defective.

Number	Message	Description
0090	REFERENCE RETURN INCOMPLETE	1) The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2) An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
0091	MANUAL REFERENCE POSITION RETURN IS NOT PERFORMED IN FEED HOLD	Manual return to the reference position cannot be performed when automatic operation is halted. Perform the manual return to the reference position when automatic operation is stopped or reset.
0092	ZERO RETURN CHECK (G27) ERROR	The axis specified in G27 has not returned to reference position. Reprogram so that the axis returns to reference position.
0094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the Operator's Manual.
0095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece origin offset amount changed.) Perform the correct operation according to the Operator's Manual.
0096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece origin offset amount changed.) Perform the correct operation according to the Operator's Manual.
0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON or alarms 0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
0099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
0109	FORMAT ERROR IN G08	T A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
0110	OVERFLOW :INTEGER	An integer went out of range during arithmetic calculations.
0111	OVERFLOW :FLOATING	A decimal point (floating point number format data) went out of range during arithmetic calculations.
0112	ZERO DIVIDE	An attempt was made to divide by zero in a custom macro.
0113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
0114	ILLEGAL EXPRESSION FORMAT	The format used in an expression in a custom macro statement is in error. The parameter tape format is in error.
0115	VARIABLE NO. OUT OF RANGE	A number that cannot be used for a local variable, common variable, or system variable in a custom macro is specified.
0116	WRITE PROTECTED VARIABLE	An attempt was made in a custom macro to use on the left side of an expression a variable that can only be used on the right side of an expression.
0118	TOO MANY BRACKET NESTING	Too many brackets “[ ]” were nested in a custom macro. The nesting level including function brackets is 5.
0119	ARGUMENT VALUE OUT OF RANGE	The value of an argument in a custom macro function is out of range.

Number	Message	Description
0122	TOO MANY MACRO NESTING	Too many macro calls were nested in a custom macro.
0123	ILLEGAL MODE FOR GOTO/WHILE/DO	A GOTO statement or WHILE–DO statement was found in the main program in the MDI or DNC mode.
0124	MISSING END STATEMENT	The END instruction corresponding to the DO instruction was missing in a custom macro.
0125	MACRO STATEMENT FORMAT ERROR	The format used in a macro statement in a custom macro is in error.
0126	ILLEGAL LOOP NUMBER	DO and END Nos. in a custom macro are in error, or exceed the permissible range (valid range: 1 to 3).
0127	DUPLICATE NC,MACRO STATEMENT	An NC statement and macro statement were specified in the same block.
0128	ILLEGAL MACRO SEQUENCE NUMBER	The specified sequence No. could not be found for sequence number search. The sequence No. specified as the jump destination in GOTO-- and M99P-- could not be found.
0129	USE 'G' AS ARGUMENT	G is used as an argument in a custom macro call. G cannot be used as an argument.
0130	NC AND PMC AXIS ARE CONFLICTED	The NC command and the PMC axis control command were conflicted. Modify the program or ladder.
0136	SPOS AXIS - OTHER AXIS SAME TIME	T M code of the spindle positioning and the axis address which is not the spindle positioning axis are commanded at the same time. In the spindle positioning mode, the axis address of the spindle positioning axis and the axis address which is not the spindle positioning axis are commanded at the same time.
0137	M-CODE & MOVE CMD IN SAME BLK.	T M code of the spindle positioning and the axis address which is the spindle positioning axis are commanded at the same time.
0139	CANNOT CHANGE PMC CONTROL AXIS	The PMC axis was selected for the axis for which the PMC axis is being controlled.
0140	PROGRAM NUMBER ALREADY IN USE	In the background, an attempt was made to select or delete the program being selected in the foreground. Perform the correct operation for the background edition.
0142	ILLEGAL SCALE RATE	M The scaling rate is 0 times or 10000 times or more. Modify the setting of the scaling rate. (G51P_ … or G51I_J_K_ … or parameter (No. 5411 or 5421))
0143	COMMAND DATA OVERFLOW	An overflow occurred in the storage length of the CNC internal data. This alarm is also generated when the result of internal calculation of scaling (M series), coordinate rotation (M series), and cylindrical interpolation overflows the data storage. It also is generated during input of the manual intervention amount.
0144	ILLEGAL PLANE SELECTED	M The coordinate rotation plane and arc or cutter compensation plane must be the same. Modify the program.
0145	ILLEGAL USE OF G12.1/G13.1	T The axis No. of plane selection parameter No. 5460 (linear axis) and No. 5461(axis of rotation) in the polar coordinate interpolation mode is out of range (1 to number of controlled axes).

Number	Message	Description
0146	ILLEGAL USE OF G-CODE	T The G code must be G40 modal when the polar coordinate interpolation mode is set or canceled. An illegal G code was specified while in the polar coordinate interpolation mode. Only the following G codes can be specified in this mode. G01,G02,G03,G04,G40,G41,G42,G65,G66,G67, (G90 and G91 for the G code system B or C), G98,G99
0148	SETTING ERROR	M Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714).
0149	FORMAT ERROR IN G10L3	In registration (G10L3 to G11) of tool life management data, an address other than Q1, Q2, P1, and P2 or an unusable address was specified.
0150	ILLEGAL LIFE GROUP NUMBER	The tool group number exceeded the maximum allowable value. The tool group number (P after specification of G10 L3;) or the group number given by the tool life management T code in a machining program.
0151	GROUP NOT FOUND AT LIFE DATA	The tool group specified in a machining program is not set in tool life management data.
0152	OVER MAXIMUM TOOL NUMBER	The number of tools registered in one group exceeded the maximum allowable registration tool number.
0153	T-CODE NOT FOUND	In registration of tool life data, a block in which the T code needs to be specified does not include the T code. Alternatively, in tool exchange method D, M06 is specified solely. Modify the program.
0154	NOT USING TOOL IN LIFE GROUP	The H99 command, D99 command, or the H/D code set by parameters No. 13265 and No. 13266 was specified when no tool belonging to a group is used.
0155	ILLEGAL T-CODE COMMAND	In a machining program, the T code specified in the same block as M06 does not correspond to the group in current use. Modify the program.
0156	P/L COMMAND NOT FOUND	The P and L commands are not specified in the beginning of a program for setting a tool group. Modify the program.
0157	TOO MANY TOOL GROUPS	In registration of tool life management data, the group setting command block counts of P (group number) and L (tool life) exceeded the maximum group count.
0158	TOOL LIFE VALUE OUT OF RANGE	The life value that is being set is too large. Change the setting.
0159	ILLEGAL TOOL LIFE DATA	Tool life management data is corrupted for some reason. Register the tool data in the tool group or the tool data in the group again by G10L3; or MDI input.
0160	MISMATCH WAITING M-CODE	A waiting M-code is in error. Different waiting M codes are specified for paths 1 and 2.
0163	ILLEGAL COMMAND IN G68/G69	T G68 and G69 are not independently commanded in balance cut.
0169	ILLEGAL TOOL GEOMETRY DATA	T Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.

Number	Message	Description
0175	ILLEGAL G07.1 AXIS	An axis which cannot perform cylindrical interpolation was specified. More than one axis was specified in a G07.1 block. An attempt was made to cancel cylindrical interpolation for an axis that was not in the cylindrical interpolation mode. In cylindrical interpolation mode, to specify circular interpolation including a rotation axis (when bit 0 (ROT) of parameter No. 1006 is 1 and parameter No. 1260 is set), the value of rotation axis parameter No. 1022 must not be 0 but 5, 6, or 7 for parallel axis specification.
0176	ILLEGAL G-CODE USE(G07.1 MODE)	A G code was specified that cannot be specified in the cylindrical interpolation mode. This alarm also is generated when an 01 group G code was in the G00 modal or code G00 was instructed. Cancel the cylindrical interpolation mode before instructing code G00.
0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 5844.
0194	SPINDLE COMMAND IN SYNCHRO-MODE	T A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode. M A Cs contour control mode or rigid tapping mode was specified during the spindle synchronous control mode or simple spindle synchronous control mode.
0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
0199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
0200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value was out of range or was not specified. The parameter (Nos. 5241 to 5243) setting is an S value which can be specified for the rigid tap. Correct the parameters or modify the program.
0201	FEEDRATE NOT FOUND IN RIGID TAP	The command F code for a cutting feedrate is a zero. If the value of F command is much smaller than that of the S command, when a rigid tap command is specified, this alarm is generated. This is because cutting is not possible by the lead specified by the program.
0202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. (System error)
0203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
0204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (or G74) block. Modify the program.
0205	RIGID MODE DI SIGNAL OFF	Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (DGN G061.0) is not ON during execution of the G84 (or G74) block. Check the PMC ladder diagram to find the reason why the DI signal is not turned on.
0206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Modify the program.
0207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.

Number	Message	Description
0210	CAN NOT COMMAND M198/M99	1) The execution of an M198 or M99 command was attempted during scheduled operation. Alternatively, the execution of an M198 command was attempted during DNC operation. Modify the program. **T** 2) The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.
0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In feed axis control , the following errors occurred during the synchronous operation. 1) The program issued the move command to the slave axis. 2) The program issued the manual operation to the slave axis. 3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.
0214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool length compensation (M series) of the shift type is executed in the synchronous control. Correct the program.
0217	DUPLICATE G51.2(COMMANDS)	**T** G51.2 is further commanded in the G51.2 mode. Modify the program.
0218	NOT FOUND P/Q COMMAND	**T** P or Q is not commanded in the G51.2 block, or the command value is out of the range. Modify the program. For a polygon turning between spindles, more information as to why this alarm occurred is indicated in DGN No. 471.
0219	COMMAND G51.2/G50.2 INDEPENDENTLY	**T** G51.2 and 50.2 were specified in the same block for other commands. Modify the program in another block.
0220	ILLEGAL COMMAND IN SYNCHR-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. Modify the program or check the PMC ladder.
0221	ILLEGAL COMMAND IN SYNCHR-MODE	**T** Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.
0222	DNC OP. NOT ALLOWED IN BG-EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.
0224	ZERO RETURN NOT FINISHED	1) A reference position return has not been performed before the start of automatic operation. (Only when bit 0 (ZRNx) of parameter No. 1005 is 0) Perform a reference position return. **T** 2) The spindle positioning axis is commanded when the mode is not spindle positioning. Perform the spindle orientation.
0230	R CODE NOT FOUND	**M** Cut depth R is not specified in the block including G161. Alternatively, the value specified for R is negative. Modify the program.
0231	ILLEGAL FORMAT IN G10 L52	Errors occurred in the specified format at the programmable-parameter input.
0232	TOO MANY HELICAL AXIS COMMAND	Three or more axes were specified as helical axes in the helical interpolation mode.

Number	Message	Description
0233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS-232-C interface, other users were using it.
0245	T-CODE NOT ALLOWED IN THIS BLOCK	T One of the G codes, G04,G10,G28,G30,G50, and G53, which cannot be specified in the same block as a T code, was specified with a T code.
0247	THE MISTAKE IS FOUND IN THE OUTPUT CODE OF DATA.	When an encrypted program is output, EIA is set for the output code. Specify ISO.
0250	TOOL CHANGE ILLEGAL Z AXIS COMMAND	A Z-axis move command was performed in the same block for M06 command.
0251	TOOL CHANGE ILLEGAL T COMMAND	An unusable T code was specified in M06Txx.
0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program.
0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although parameter No. 1012#0 (IDGx) was set to 1 to inhibit the reference position from being set again for a return to the reference position without a dog, an attempt was made to perform a manual return to the reference position.
0302	SETTING THE REFERENCE POSITION WITHOUT DOG IS NOT PERFORMED	The reference position could not be set for a return to the reference position without a dog. Possible causes are: - The axis was not moved in the direction of a return to the reference position for jog feeding. - The axis was moved in the direction opposite to the direction of a manual return to the reference position.
0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
0305	INTERMEDIATE POSITION IS NOT ASSIGNED	M Although a G28 (automatic return to the reference position) or G30 (return to the second, third, or fourth reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.
0306	MISMATCH AXIS WITH CNR/CHF	T The correspondence between the moving axis and the I, J, or K command is incorrect in a block in which chamfering is specified.
0307	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	An attempt was made to set a butt-type reference position for an axis for which to use the function to set a reference position without a dog.
0310	FILE NOT FOUND	The specified file could not be found during a subprogram or macro call.
0311	CALLED BY FILE NAME FORMAT ERROR	An invalid format was specified to call a subprogram or macro using a file name.
0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	T Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. Two or more blocks not to be moved exist in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (parameter No. 3405#4 = 1) was specified for direct input of drawing dimensions, a comma was specified.

Number	Message	Description
0313	ILLEGAL LEAD COMMAND	T The variable-lead threading increment specified in address K exceeds the specified maximum value in variable-lead threading. Or, a negative lead value was specified.
0314	ILLEGAL SETTING OF POLYGONAL AXIS	T An axis was specified invalidly in polygon turning. For polygon turning: 1) A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning between spindles: 1) Valid spindles are not specified. (Parameter Nos. 7640 to 7643) 2) A spindle other than the serial spindle. 3) A spindle is not connected.
0315	ILLEGAL NOSE ANGLE COMMAND IS IN THE THREAD CUTTING CYCLE	T An invalid tool tip angle is specified in a multiple repetitive canned threading cycle (G76).
0316	ILLEGAL CUTTING AMOUNT IS IN THE THREAD CUTTING CYCLE	T An minimum depth of cut higher than the thread height is specified in a multiple repetitive canned threading cycle (G76).
0317	ILLEGAL THREAD COMMAND IS IN THE THREAD CUTTING CYCLE	T A zero or a negative value is specified in a multiple repetitive canned threading cycle (G76) as the thread height or the depth of cut.
0318	ILLEGAL RELIEF AMOUNT IS IN THE DRILLING CYCLE	T Although an escape directions is set in a multiple repetitive canned cutting-off cycle (G74 or G75), a negative value is specified for Δd.
0319	THE END POINT COMMAND IS ILLEGAL IN THE DRILLING CYCLE	T Although the Δi or Δk travel distance is set to 0 in a multiple repetitive canned cutting-off cycle (G74 or G75), a value other than 0 us specified for a U or W.
0320	ILLEGAL MOVEMENT AMOUNT/CUTTING AMOUNT IS IN THE DRILLING CYCLE	T A negative value is specified in a multiple repetitive canned cutting-off cycle (G74 or G75) as Δi or Δk (travel distance/the depth of cut).
0321	ILLEGAL REPEATED TIME IS IN THE PATTERN REPEATING CYCLE	T A zero or a negative value is specified in a multiple repetitive canned closed loop cycle (G73) as a repeated time.
0322	FINISHING SHAPE WHICH OVER OF STARTING POINT	T An invalid shape which is over the cycle starting point is specified in a shape program for a multiple repetitive canned rough-cutting cycle (G71 or G72).
0323	THE FIRST BLOCK OF SHAPE PROGRAM IS A COMMAND OF TYPE II	T Type II is specified in the first block of the shape program specified by P in a multiple repetitive canned rough-cutting cycle (G71 or G72). For G71, Z(W) is specified. For G72, X(U) is specified.
0324	THE INTERRUPTION TYPE MACRO WAS DONE IN THE MULTIPLE REPETIVE CYCLES	T An interruption type macro was issued during the multiple repetitive canned cycle (G70, G71, G72, or G73).
0325	UNAVAILABLE COMMAND IS IN SHAPE PROGRAM	T An usable command was issued in a shape program for a multiple repetitive canned cycle (G70, G71, G72, or G73).

Number	Message	Description
0326	LAST BLOCK OF SHAPE PROGRAM IS A DIRECT DRAWING DIMENSIONS	T In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for direct input of drawing dimensions in the last block is terminated in the middle.
0327	MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE	T A multiple repetitive canned cycle (G70, G71, G72, or G73) was commanded in a modal state in which a multiple repetitive canned cycle could not be commanded.
0328	ILLEGAL WORK POSITION IS IN THE TOOL NOSE RADIUS COMPENSATION	T The specification for the blank side for a tool nose radius compensation (G41 or G42) is incorrect in a multiple repetitive canned cycle (G71 or G72).
0329	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)	T In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command of the second plane axis was not a monotonous increase or decrease.
0330	ILLEGAL AXIS COMMAND IS IN THE TURNING CANNED CYCLE	T An axis other than the plane is specified n a canned cycle(G90, G92, or G94).
0334	OFFSET IS OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (malfunction prevention function)
0336	TOOL COMPENSATION COMMANDED MORE TWO AXES	M For a tool length compensation C, an attempt was made to command the offset to other axes without canceling the offset. Or, for a tool length compensation C, multiple axes are specified in G43 or G44 block.
0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (malfunction prevention function)
0338	ILLEGAL EXEC SEQ OF BLOCK	An incorrect value was detected in a check sum. (malfunction prevention function)
0345	TOOL CHANGE ILLEGAL Z AXIS POS	A tool change position on the Z-axis is incorrect.
0346	TOOL CHANGE ILLEGAL TOOL NUM	A tool number for tool change is incorrect.
0347	TOOL CHANGE ILLEGAL COMMAND IN SAME BLK.	Tool changing is commanded twice or more in the same block.
0348	TOOL CHANGE Z AXIS POS NOT ESTABLISHED	A tool change spindle on the Z-axis is not set.
0349	TOOL CHANGE SPINDLE NOT STOP	A tool change spindle stop is not stopped.
0350	PARAMETER OF THE INDEX OF THE SYNCHRONOUS CONTROL AXIS SET ERROR.	T An illegal synchronous control axis number (parameter No. 8180) is set.
0351	BECAUSE THE AXIS IS MOVING, THE SYNC CONTROL IS CAN'T BE USED.	T While the axis being subject to synchronous control was moving, an attempt was made to start or cancel the synchronous control by a synchronous control axis selection signal.

Number	Message	Description
0352	SYNCHRONOUS CONTROL AXIS COMPOSITION ERROR.	T This error occurred when: 1) An attempt was made to perform synchronous control for the axis during a synchronization, composite, or superimposed control. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation. 3) An attempt was made to operate synchronous control although a parent-child-grandchild relation was not set.
0353	THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.	T This error occurred when: 1) A move command was executed for an axis for which bit 7 (NUMx) of parameter No. 8163 was 1. 2) A move command was executed for a slave axis in synchronous control. 3) A move command was executed for an axis for which bit 7 (MUMx) of parameter No. 8162 was 1 in composite control.
0354	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE	T This error occurred when G28 was specified to the master axis being parking during synchronous control , but an axis reference position is not set for the slave axis.
0355	PARAMETER OF THE INDEX OF THE COMPOSITE CONTROL AXIS SET ERROR.	T An illegal composite control axis number (parameter No. 8183) is specified.
0356	BECAUSE THE AXIS IS MOVING, THE COMP CONTROL IS CAN'T BE USED.	T While the axis being subject to composite control was moving, an attempt was made to start or cancel the composite control by a composite control axis selection signal.
0357	COMPOSITE CONTROL AXIS COMPOSITION ERROR.	T This error occurred when an attempt was made to perform composite control for the axis during a synchronous, composite, or superimposed control.
0359	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE	T This error occurred when G28 was specified to the composite axis during composite control, but a reference position is not set to the other part of the composition.
0360	PARAMETER OF THE INDEX OF THE SUPERPOS CONTROL AXIS SET ERROR.	T An illegal superimposed control axis number (parameter No. 8186) is specified.
0361	BECAUSE THE AXIS IS MOVING, THE SUPERPOS CONTROL IS CAN'T BE USED.	T While the axis being subject to superimposed control was moving, an attempt was made to start or cancel the superimposed control by a superimposed control axis selection signal.
0362	SUPERPOSITION CONTROL AXIS COMPOSITION ERROR.	T This error occurred when: 1) An attempt was made to perform superimposed control for the axis during a synchronous, composite, or superimposed control. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation.
0363	THE G28 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	T This error occurred when G28 was specified to the superimposed control slave axis during superimposed control.
0364	THE G53 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	T This error occurred when G53 was specified to the slave axis being moved during superimposed control.

Number	Message	Description
0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The Max. total number of control axes, the Max. number of feed axes or the Max. number of control axes is exceeded. Check parameter No. 0981 and No. 0982. If this alarm is generated, the emergency stop state cannot be released.
0369	G31 FORMAT ERROR	1) No axis is specified or tow or more axes are specified in the torque limit switch instruction (G31P98/P99). 2) G31P90 cannot be specified.
0370	G31P/G04Q ERROR	1) The specified address P value for G31 is out of range. The address P range is 1 to 4 in a multistage skip function. 2) The specified address Q value for G04 is out of range. The address Q range is 1 to 4 in a multistage skip function. 3) P1-4 for G31, or Q1-4 for G04 was commanded without a multistage skip function option. T 4) In G72 or G74 in grinding canned cycles, the specified address P value is out of range. Address P ranges from 1 to 4 in the multistage skip function. P1-4 was specified in G72 or G74 even though the multistage skip function option is not present.
0372	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the orthogonal axis.
0373	ILLEGAL HIGH-SPEED SKIP SIGNAL	In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), the same high-speed signal is selected in different paths.
0375	CAN NOT ANGULAR CONTROL(SYNC:MIX:OVL)	Angular axis control is disabled for this axis configuration. 1) All related axes in angular axis control are not in synchronous control mode. Alternatively, settings must be made to provide synchronous control between angular axes, and also between orthogonal axes. 2) All related axes in angular axis control are not in composite control mode. Alternatively, settings must be made to provide composite control between angular axes, and also between orthogonal axes. 3) The related axes in angular axis control are in superimposed control mode.
0376	SERIAL DCL: ILLEGAL PARAMETER	1) When Parameter No.1815#1 is set to “1”, parameter No.2002#3 is set to “0” 2) The absolute-position detection function is enabled. (Parameter No.1815#5 is set to “1”.)
0412	ILLEGAL G CODE	An unusable G code was used.
0445	ILLEGAL AXIS OPERATION	The positioning command was issued in the speed control mode. Check the SV speed control mode signal (Fn521).
0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	G96.1, G96.2, G96.3, and G96.4 are specified in the block that includes other commands. Modify the program.
0447	ILLEGAL SETTING DATA	The spindle controlled with the servo motor is not set correctly. Check the parameters for the function of spindle control with the servo motor.

Number	Message	Description
0455	ILLEGAL COMMAND IN GRINDING	In grinding canned cycles: M 1) The signs of the I, J, and K commands do not match. 2) The amount of travel of the grinding axis is not specified.
0456	ILLEGAL PARAMETER IN GRINDING	Parameters related to grinding canned cycles are incorrectly set. Probable causes are given below. 1) The axis number of the grinding axis is incorrectly set (parameters No. 5176 to No. 5179). M 2) The axis number of the dressing axis is incorrectly set (parameters No. 5180 to No. 5183). 3) The axis numbers of the cut axis, grinding axis, and dressing axis (only for the M series) overlap.
0518	CODE OF CONTROL-IN IS NOT EXIST	The code of a control-in does not exist in comment of the program. It is necessary to end the comment by the code of a control-in ")", when the comment is started by the code of a control-out "(".
0601	ILLEGAL AXIS OPERATION FOR SERVO MOTOR SPINDLE	A move command is executer for the spindle controlled with the servo motor. Modify the program.
1001	AXIS CONTROL MODE ILLEGAL	The axis control mode is illegal.
1013	ILLEGAL POS. OF PROGRAM NO.	Address O or N is specified where it must not (After the macro statement etc.).
1014	ILLEGAL FORMAT OF PROGRAM NO.	Address O or N is not followed by a number.
1016	EOB NOT FOUND	EOB (End of Block) code is missing at the end of a program input in the MDI mode.
1077	PROGRAM IN USE	An attempt was made in the foreground to execute a program being edited in the background. The currently edited program cannot be executed, so end editing and restart program execution.
1079	PROGRAM FILE NOT FOUND	The program of the specified file No. is not registered in an external device. (external device subprogram call)
1080	DUPLICATE DEVICE SUB PROGRAM CALL	Another external device subprogram call was made from a subprogram after the subprogram called by the external device subprogram call.
1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	The external device subprogram call is not possible in this mode.
1091	DUPLICATE SUB-CALL WORD	More than one subprogram call instruction was specified in the same block.
1092	DUPLICATE MACRO-CALL WORD	More than one macro call instruction was specified in the same block.
1093	DUPLICATE NC-WORD & M99	An address other than O, N, P or L was specified in the same block as M99 during the macro modal call state.
1095	TOO MANY TYPE-2 ARGUMENT	More than ten sets of I, J and K arguments were specified in the type–II arguments (A, B, C, I, J, K, I, J, K, ...) for custom macros.
1096	ILLEGAL VARIABLE NAME	An illegal variable name was specified. A code that cannot be specified as a variable name was specified. The command of [#_OFSxx] does not match the type (A or C) of tool compensation memory in current use.
1097	TOO LONG VARIABLE NAME	The specified variable name is too long.
1098	NO VARIABLE NAME	The specified variable name cannot be used as it is not registered.

Number	Message	Description
1099	ILLLEGAL SUFFIX [ ]	A suffix was not specified to a variable name that required a suffix enclosed by [ ]. A suffix was specified to a variable name that did not require a suffix enclosed by [ ]. The value enclosed by the specified [ ] was out of range.
1100	CANCEL WITHOUT MODAL CALL	Call mode cancel (G67) was specified even though macro continuous–state call mode (G66) was not in effect.
1101	ILLEGAL CNC STATEMENT IRT.	An interrupt was made in a state where a custom macro interrupt containing a move instruction could not be executed.
1115	READ PROTECTED VARIABLE	An attempt was made in a custom macro to use on the right side of an expression a variable that can only be used on the left side of an expression.
1120	ILLEGAL ARGUMENT FORMAT	The specified argument in the argument function (ATAN, POW) is in error.
1124	MISSING DO STATEMENT	The DO instruction corresponding to the END instruction was missing in a custom macro.
1125	ILLEGAL EXPRESSION FORMAT	The description of the expression in a custom macro statement contains an error. A parameter program format error. The screen displayed to enter periodic maintenance data or item selection menu (machine) data does not match the data type.
1128	SEQUENCE NUMBER OUT OF RANGE	The jump destination sequence No. in a custom macro statement GOTO instruction was out of range (valid range: 1 to 99999).
1131	MISSING OPEN BRACKET	The number of left brackets ([) is less than the number of right brackets (]) in a custom macro statement.
1132	MISSING CLOSE BRACKET	The number of right brackets (]) is less than the number of left brackets ([) in a custom macro statement.
1133	MISSING '='	An equal sign (=) is missing in the arithmetic calculation instruction in a custom macro statement.
1134	MISSING ','	A delimiter (,) is missing in a custom macro statement.
1137	IF STATEMENT FORMAT ERROR	The format used in the IF statement in a custom macro is in error.
1138	WHILE STATEMENT FORMAT ERROR	The format used in the WHILE statement in a custom macro is in error.
1139	SETVN STATEMENT FORMAT ERROR	The format used in the SETVN statement in a custom macro is in error.
1141	ILLEGAL CHARACTER IN VAR. NAME	The SETVN statement in a custom macro contacts a character that cannot be used in a variable name.
1142	TOO LONG V-NAME (SETVN)	The variable name used in a SETVN statement in a custom macro exceeds 8 characters.
1143	BPRNT/DPRNT STATEMENT FORMAT ERROR	The format used in the BPRINT statement or DPRINT statement is in error.
1144	G10 FORMAT ERROR	Data input for the L No. of G10 or the corresponding function is not enabled. Data setting address P or R is not specified. An address not relating to the data setting is specified. Which address to specify varies according to the L No. The sign, decimal point or range of the specified address are in error.
1160	COMMAND DATA OVERFLOW	An overflow occurred in the position data within the CNC. This alarm is also generated if the target position of a command exceeds the maximum stroke as a result of calculation such as coordinate conversion, offset, or introduction of a manual intervention amount.

Number	Message	Description
1180	ALL PARALLEL AXES IN PARKING	T All of the axis specified for automatic operation are parked.
1196	ILLEGAL DRILLING AXIS SELECTED	An illegal axis was specified for drilling in a canned cycle for drilling. In the G code command block in a canned cycle, a Z point is not specified for the drilling axis.
1200	PULSCODER INVALID ZERO RETURN	The grid position could not be calculated during grid reference position return using the grid system as the one–revolution signal was not received before leaving the deceleration dog. This alarm is also generated when the tool does not reach a feedrate that exceeds the servo error amount preset to parameter No. 1836 before the deceleration limit switch is left (deceleration signal *DEC returns to “1”).
1202	NO F COMMAND AT G93	M F codes in the inverse time specification mode (G93) are not handled as modal, and must be specified in individual blocks.
1223	ILLEGAL SPINDLE SELECT	An attempt was made to execute an instruction that uses the spindle although the spindle to be controlled has not been set correctly.
1298	ILLEGAL INCH/METRIC CONVERSION	An error occurred during inch/metric switching.
1300	ILLEGAL ADDRESS	The axis No. address was specified even though the parameter is not an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Axis No. cannot be specified in pitch error compensation data.
1301	MISSING ADDRESS	The axis No. was not specified even though the parameter is an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Or, data No. address N, or setting data address P or R are not specified.
1302	ILLEGAL DATA NUMBER	A non–existent data No. was found while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. This alarm is also generated when illegal word values are found.
1303	ILLEGAL AXIS NUMBER	An axis No. address exceeding the maximum number of controlled axes was found while loading parameters from a tape or by entry of the G10 parameter.
1304	TOO MANY DIGIT	Data with too many digits was found while loading parameters or pitch error compensation data from a tape.
1305	DATA OUT OF RANGE	Out–of–range data was found while loading parameters or pitch error compensation data from a tape. The values of the data setting addresses corresponding to L Nos. during data input by G10 was out of range. This alarm is also generated when NC programming words contain out–of–range values.
1306	MISSING AXIS NUMBER	A parameter which requires an axis to be specified was found without an axis No. (address A) while loading parameters from a tape.
1307	ILLEGAL USE OF MINUS SIGN	Data with an illegal sign was found while loading parameters or pitch error compensation data from a tape, or by entry of the G10 parameter. A sign was specified to an address that does not support the use of signs.

Number	Message	Description
1308	MISSING DATA	An address not followed by a numeric value was found while loading parameters or pitch error compensation data from a tape.
1329	ILLEGAL MACHINE GROUP NUMBER	An machine group No. address exceeding the maximum number of controlled machine groups was found while loading parameters from a tape or by entry of the G10 parameter.
1330	ILLEGAL SPINDLE NUMBER	An spindle No. address exceeding the maximum number of controlled spindles was found while loading parameters from a tape or by entry of the G10 parameter.
1331	ILLEGAL PATH NUMBER	An path No. address exceeding the maximum number of controlled path was found while loading parameters from a tape or by entry of the G10 parameter.
1332	DATA WRITE LOCK ERROR	Could not load data while loading parameters, pitch error compensation data and work coordinate data from tape.
1333	DATA WRITE ERROR	Could not write data while loading data from tape.
1470	G40.1 –G42.1 PARAMETER MISS	M A parameter setting related to normal direction control is illegal. The axis number of a normal direction controlled axis is set in parameter No. 5480, but that axis number is in the range of the number of controlled axes. The axis set as a normal direction controlled axis is not set as a rotation axis (ROTx, bit 0 of parameter No. 1006) = 1 and No.1022=0). Set the feedrate at which to insert rotation about a normal direction controlled axis in parameter No. 5481, in the range of 1 to 15000 mm/min.
1508	DUPLICATE M-CODE (INDEX TABLE REVERSING)	M A function to which the same code as this M code is set exists. (index table indexing)
1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	T A function to which the same code as this M code is set exists. (spindle positioning, orientation)
1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	T A function to which the same code as this M code is set exists. (spindle positioning, positioning)
1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	T A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
1533	ADDRESS F UNDERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too slow in the feed per single rotation mode.
1534	ADDRESS F OVERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too fast in the feed per single rotation mode.
1537	ADDRESS F UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too slow.
1538	ADDRESS F OVERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too fast.
1541	S-CODE ZERO	“0” has been instructed as the S code.
1543	ILLEGAL GEAR SETTING	T The gear ratio between the spindle and position coder, or the set position coder number of pulses is illegal in the spindle positioning function.
1544	S-CODE OVER MAX	The S command exceeds the maximum spindle rotation number.
1548	ILLGAL AXIS MODE	The spindle positioning (T series) axis/Cs contour control axis was specified during switching of the controlled axis mode.

Number	Message	Description
1561	ILLEGAL INDEXING ANGLE	**M** The specified angle of rotation is not an integer multiple of the minimum indexing angle.
1564	INDEX TABLE AXIS – OTHER AXIS SAME TIME	**M** The index table indexing axis and another axis have been specified in the same block.
1567	INDEX TABLE AXIS DUPLICATE AXIS COMMAND	**M** Index table indexing was specified during axis movement or on an axis for which the index table indexing sequence was not completed.
1590	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
1591	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting TVC parameter No. 0000#0 to “0”.
1592	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read. For the program restart function, this alarm is generated if a specified block is not found.
1593	EGB PARAMETER SETTING ERROR	**M** Error in setting a parameter related to the EGB 1) The setting of SYN, bit 0 of parameter No. 2011, is not correct. 2) The slave axis specified with G81 is not set as a rotation axis. (ROT, bit 0 of parameter No. 1006) 3) Number of pulses per rotation (Parameter (No. 7772 or No. 7773) is not set.)
1594	EGB FORMAT ERROR	**M** Error in the format of the block of an EGB command 1) T (number of teeth) is not specified in the G81 block. 2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. 3) n the G81 block, only one of P and Q is specified.
1595	ILL-COMMAND IN EGB MODE	**M** During synchronization with the EGB, a command that must not be issued is issued. 1) Slave axis command using G27, G28, G29, G30, G33, G53, etc. 2) Inch/metric conversion command using G20, G21, etc.
1596	EGB OVERFLOW	**M** An overflow occurred in the calculation of the synchronization coefficient.
1805	ILLEGAL COMMAND	[I/O Device] An attempt was made to specify an illegal command during I/O processing on an I/O device. [G30 Reference Position Return] The address P numbers for specifying the 2nd, 3rd, and 4th reference position returns are not 2, 3, and 4. [Single Rotation Dwell] The specified spindle rotation is “0” when single rotation dwell is specified.

Number	Message	Description
1806	DEVICE TYPE MISS MATCH	An operation not possible on the I/O device that is currently selected in the setting was specified. This alarm is also generated when file rewind is instructed even though the I/O device is not a FANUC Cassette.
1807	PARAMETER SETTING ERROR	An illegal I/O interface is specified. The external I/O device and baud rate, stop bit and protocol selection settings are erroneous.
1808	DEVICE DOUBLE OPENED	An attempt was made to open a device that is being accessed.
1820	ILLEGAL DI SIGNAL STATE	1) An each axis workpiece coordinate system preset signal was turned "1" in the state in which all axes on the path including the axis on which to perform preset with the each axis workpiece coordinate system were not stopped or in which a command was in execution. 2) When the M code for performing preset with an each axis workpiece coordinate system preset signal is specified, the each axis workpiece coordinate system preset signal is not input. 3) The auxiliary function lock is enabled.
1823	FRAMING ERROR(1)	The stop bit of the character received from the I/O device connected to reader/punch interface 1 was not detected.
1830	DR OFF(2)	The data set ready input signal DR of the I/O device connected to reader/punch interface 2 turned OFF.
1832	OVERRUN ERROR(2)	The next character was received from the I/O device connected to reader/punch interface 2 before it could read a previously received character.
1833	FRAMING ERROR(2)	The stop bit of the character received from the I/O device connected to reader/punch interface 2 was not detected.
1834	BUFFER OVERFLOW(2)	The NC received more than 10 characters of data from the I/O device connected to reader/punch interface 2 even though the NC sent a stop code (DC3) during data reception.
1912	V-DEVICE DRIVER ERROR (OPEN)	An error occurred during device driver control.
1960	ACCESS ERROR (MEMORY CARD)	Illegal memory card accessing This alarm is also generated during reading when reading is executed up to the end of the file without detection of the EOR code.
1961	NOT READY (MEMORY CARD)	The memory card is not ready.
1962	CARD FULL (MEMORY CARD)	The memory card has run out of space.
1963	CARD PROTECTED (MEMORY CARD)	The memory card is write–protected.
1964	NOT MOUNTED (MEMORY CARD)	The memory card could not be mounted.
1965	DIRECTORY FULL (MEMORY CARD)	The file could not be generated in the root directory for the memory card.
1966	FILE NOT FOUND (MEMORY CARD)	The specified file could not be found on the memory card.
1967	FILE PROTECTED (MEMORY CARD)	The memory card is write–protected.
1968	ILLEGAL FILE NAME (MEMORY CARD)	Illegal memory card file name
1969	ILLEGAL FORMAT (MEMORY CARD)	Check the file name.
1970	ILLEGAL CARD (MEMORY CARD)	This memory card cannot be handled.
1971	ERASE ERROR (MEMORY CARD)	An error occurred during memory card erase.
1972	BATTERY LOW (MEMORY CARD)	The memory card battery is low.
1973	FILE ALREADY EXIST	A file having the same name already exists on the memory card.

Number	Message	Description
2032	EMBEDDED ETHERNET/DATA SERVER ERROR	An error was returned in the built-in Ethernet/data server function. For details, see the error message screen of the built-in Ethernet or data server.
2051	#200-#499ILLEGAL P-CODE MACRO COMMON INPUT(NO OPTION)	An attempt was made to enter a custom macro common variable not existing in the system.
2052	#500-#549P-CODE MACRO COMMON SELECT(CANNOT USE SETVN)	The variable name cannot be entered. The SETVN command cannot be used with the P-CODE macro common variables #500 to #549.
2053	P-CODE VARIABLE NUMBER IS OUTSIDE OF RANGE	An attempt was made to enter a P-CODE-only variable not existing in the system.
2054	EXTENDED P-CODE VARIABLE NUMBER IS OUTSIDE OF RANGE	An attempt was made to enter an extended P-CODE-only variable not existing in the system.
4010	ILLEGAL REAL VALUE OF OBUF :	The real value for a output buffer is in error.
5006	TOO MANY WORD IN ONE BLOCK	The number of words in a block exceeds the maximum. The maximum is 26 words. However, this figure varies according to NC options. Divide the instruction word into two blocks.
5007	TOO LARGE DISTANCE	Due to compensation, point of intersection calculation, interpolation or similar reasons, a movement distance that exceeds the maximum permissible distance was specified. Check the programmed coordinates or compensation amounts.
5009	PARAMETER ZERO (DRY RUN)	The dry run feedrate parameter No. 1410 or maximum cutting feedrate parameter No. 1430 for each axis has been set to 0.
5010	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read.
5011	PARAMETER ZERO (CUT MAX)	The maximum cutting feedrate parameter No. 1430 has been set to 0.
5014	TRACE DATA NOT FOUND	A transfer could not be made because of no trace data.
5016	ILLEGAL COMBINATION OF M CODES	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5018	POLYGON SPINDLE SPEED ERROR	T In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained. For polygon turning between spindles: More information as to why this alarm occurred is indicated in DGN No. 471.
5020	PARAMETER OF RESTART ERROR	An invalid value is set in parameter No. 7310, which specifies the axis order in which the tool is moved along axes to the machining restart position in dry run. A value ranging from 1 to the number of controlled axes may be set in this parameter.

Number	Message	Description
5046	ILLEGAL PARAMETER (S-COMP)	M The setting of a parameter related to simple straightness compensation contains an error. Possible causes include: 1) A non-existent axis number is set in a moving or compensation axis parameter. 2) The simple straightness compensation point numbers do not have correct magnitude relationships. 3) No simple straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. 4) The compensation per compensation point is either too large or too small.
5064	DIFFERRENT AXIS UNIT	Circular interpolation has been specified on a plane consisting of axes having different increment systems.
5065	DIFFERRENT AXIS UNIT(PMC AXIS)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5110	IMPROPER G-CODE (AICC MODE)	An unspecifiable G code was specified in the advanced preview control, AI advanced preview control, or AI contour control mode.
5130	NC AND PMC AXIS CONFLICT	The NC command and the PMC axis control command were conflicted. Modify the program or the ladder.
5131	NC COMMAND IS NOT COMPATIBLE	T The PMC axis control and polar coordinate interpolation were specified simultaneously.
5195	DIRECTION CAN NOT BE JUDGED	T Measurement is invalid in the tool compensation measurement value direct input B function. [For 1-contact input] 1) The recorded pulse direction is not constant. For example, the stopped state may be set during offset write mode, the servo off state may be entered, or the direction may vary. 2) The tool is moving along the two axes (X-axis and Y-axis). [For movement direction judgment in 4-contact input] 1) The recorded pulse direction is not constant. For example, the stopped state may be set during offset write mode, the servo off state may be entered, or the direction may vary. 2) The tool is moving along the two axes (X-axis and Z-axis). 3) The direction indicated by the tool compensation write signal does not match the movement direction of the axis.
5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting parameter (No.1819#2) is set to "1". Move the machine to reference position by manual operation and execute manual reference return.

Number	Message	Description
5257	G41/G42 NOT ALLOWED IN MDI MODE	Tool radius/tool nose radius compensation was specified in MDI mode. (Depending on the setting of the parameter MCR (No. 5008#4))
5303	TOUCH PANEL ERROR	The touch panel is not connected correctly, or the touch panel cannot be initialized when the power is turned on. Correct the cause then turn on the power again.
5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple spindle control, 1) Address P is not specified. 2) A P code for selecting a spindle is not set in parameter No. 3781. 3) An illegal G code which cannot be commanded with an S_P_; command is specified. 4) Multi-spindle control is not enabled because bit 1 (EMS) of parameter No. 3702 is 1. 5) The spindle amplifier number of each spindle is not set in parameter No. 3717. 6) A spindle command is executed from a path in which this command is prohibited (parameter No. 11090). 7) The setting of parameter No. 11090 is incorrect.
5329	M98 AND NC COMMAND IN SAME BLOCK	A subprogram call which is not a single block was commanded during canned cycle mode.
5306	MODE CHANGE ERROR	A mode switchover failed at the time of activation. An attempt to activate a one-touch macro was made while not in the reset state or during a reset or emergency stop.
5339	ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.	T 1. The value of P, Q, or L specified by G51.4/G50.4/G51.5/G50.5/G51.6/G50.6 is invalid. 2. A duplicate value is specified by parameter No. 12600.
5346	RETURN TO REFERENCE POINT	The coordinate establishment of the Cs contour control axis is not made. Perform a manual reference position return. 1) When Cs coordinate establishment is made for the Cs-axis for which the Cs-axis reference position status signal CSPENx is 0 2) When positional information is not sent from the spindle amplifier 3) When the servo off state is entered during the start of Cs-axis coordinate establishment 4) When the emergency stop state is entered during Cs-axis coordinate establishment T 5) When the Cs-axis is under synchronous or superimposed control 6) When an attempt is made to release composite control for the Cs-axis during Cs-axis coordinate establishment 7) When an attempt is made to start synchronous, composite, or superimposed control for the Cs-axis during Cs-axis coordinate establishment
5362	CONVERT INCH/MM AT REF-POS	An inch/metric conversion was performed at a position other than the reference position. Perform an inch/metric conversion after returning to the reference position.

Number	Message	Description
5391	CAN NOT USE G92	M Workpiece coordinate system setting G92 cannot be specified. 1) After tool length compensation was changed by tool length compensation shift type, G92 was specified when no absolute command is present. 2) G92 was specified in the block in which G49 is present.
5395	CS AXIS NUMBER OVER	The number of axes to be set for Cs-axis contour control exceeds the maximum number allowed in the system. Check parameter No. 1023. If this alarm is generated, the emergency stop state cannot be released.
5445	CAN NOT COMMAND MOTION IN G39	Corner circular interpolation (G39) of tool radius/tool nose radius compensation is not specified alone but is specified with a move command.
5446	NO AVOIDANCE AT G41/G42	Because there is no interference avoidance vector, the interference check avoidance function of tool radius/tool nose radius compensation does not work.
5447	DANGEROUS AVOIDANCE AT G41/G42	The interference check avoidance function of tool radius/tool nose radius compensation operation will lead to danger.
5448	INTERFERENCE TO AVD. AT G41/G42	In the interference check avoidance function of tool radius/tool nose radius compensation, a further interference occurs for an already created interference avoidance vector.

(4) Parameter writing alarm (SW alarm)

Number	Message	Description
SW0100	PARAMETER ENABLE SWITCH ON	The parameter setting is enabled (PWE, bit 0 of parameter No. 8900 is set to “1”). To set the parameter, turn this parameter ON. Otherwise, set to OFF.

(5) Servo alarms (SV alarm)

Number	Message	Description
SV0001	SYNC ALIGNMENT ERROR	In feed axis control , the amount of compensation for synchronization exceeded the parameter (No. 8325) setting value. This alarm occurs only for a slave axis.
SV0002	SYNC EXCESS ERROR ALARM 2	In feed axis control , the amount of synchronization error exceeded the parameter (No. 8332) setting value. When the synchronization is not completed after power-up, the determination is made by the parameter value (No. 8332) multiplied by the parameter (No. 8330) multiplier. This alarm occurs only for a slave axis only.
SV0003	SYNCHRONOUS/COMPOSITE/SUPER IMPOSED CONTROL MODE CAN'T BE CONTINUED	T Since as axis in synchronization, composite, or superimposed mode caused a servo alarm, the mode could not be continued, If one of the axes in a mode causes a servo alarm, all axes relating to the axis enter the servo-off state. This alarm is generated to enable the cause of the servo-off state to be checked.
SV0004	EXCESS ERROR (G31)	The amount of positional deviation during torque limit skip command operation exceeded the limit value of the parameter No.6287.

Number	Message	Description
SV0005	SYNC EXCESS ERROR (MCN)	In feed axis control , for synchronization, the difference value of the machine coordinate between a master and slave axes exceeded the parameter (No. 8314) setting value. This alarm occurs for a master or slave axis.
SV0006	ILLEGAL TANDEM AXIS	For the slave axis under tandem control, absolute position detection is set (parameter bit 5 (APC) of parameter No. 1815 = 1).
SV0007	SV ALM ANOTHER PATH(MULTI AMP.)	T When a multiaxis amplifier was used in a 2-path system across paths, a servo alarm occurred on an axis belonging to another path. When a 2-path system and multiple servo axes between paths are controlled by a multiaxis amplifier, if a servo alarm occurs on an axis belonging to another path of the same amplifier, the MCC of the amplifier goes down and SV0401 V-READY OFF occurs on an axis belonging to the local path in the same amplifier. Since SV0401 is caused by a servo alarm occurred on an axis in another path, SV0007 is caused together to clearly indicate the fact. The axis belonging to another path in the same amplifier resolves the cause of the servo alarm.
SV0301	APC ALARM: COMMUNICATION ERROR	Since the absolute-position detector caused a communication error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0302	APC ALARM: OVER TIME ERROR	Since the absolute-position detector caused an overtime error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0303	APC ALARM: FRAMING ERROR	Since the absolute-position detector caused a framing error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0304	APC ALARM: PARITY ERROR	Since the absolute-position detector caused a parity error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0305	APC ALARM: PULSE ERROR	Since the absolute-position detector caused a pulse error, the correct machine position could not be obtained. The absolute-position detector, or cable is thought to be defective.
SV0306	APC ALARM: OVER FLOW ERROR	Since the amount of positional deviation overflowed, the correct machine position could not be obtained. Check to see the parameter No. 2084 or No. 2085.
SV0307	APC ALARM: MOVEMENT EXCESS ERROR	Since the machine moved excessively, the correct machine position could not be obtained.
SV0360	ABNORMAL CHECKSUM(INT)	The checksum alarm occurred on the built–in Pulsecoder.
SV0361	ABNORMAL PHASE DATA(INT)	The phase data abnormal alarm occurred on the built–in Pulsecoder.
SV0362	ABNORMAL REV. DATA(INT)	The speed count abnormal alarm occurred on the built-in Pulsecoder.

Number	Message	Description
SV0363	ABNORMAL CLOCK(INT)	The clock alarm occurred on the built–in Pulsecoder.
SV0364	SOFT PHASE ALARM(INT)	A digital servo soft detected an abnormality on the built in Pulsecoder.
SV0365	BROKEN LED(INT)	The digital servo software detected abnormal data on the built–in Pulsecoder.
SV0366	PULSE MISS(INT)	A pulse error occurred on the built–in Pulsecoder.
SV0367	COUNT MISS(INT)	A count error occurred on the built–in Pulsecoder.
SV0368	SERIAL DATA ERROR(INT)	The communications data could not be received from the built–in Pulsecoder.
SV0369	DATA TRANS. ERROR(INT)	A CRC error or stop bit error occurred in the communications data from the built–in Pulsecoder.
SV0380	BROKEN LED(EXT)	Separate detector error
SV0381	ABNORMAL PHASE (EXT)	An abnormal alarm in the position data occurred on the separate detector.
SV0382	COUNT MISS(EXT)	A count error occurred on the separate detector.
SV0383	PULSE MISS(EXT)	A pulse error occurred on the separate detector.
SV0384	SOFT PHASE ALARM(EXT)	The digital servo software detected abnormal data on the separate detector.
SV0385	SERIAL DATA ERROR(EXT)	The communications data could not be received from the separate detector.
SV0386	DATA TRANS. ERROR(EXT)	A CRC error or stop bit error occurred in the communications data from the standalone detector.
SV0387	ABNORMAL ENCODER(EXT)	An abnormality occurred on a separate detector. For more information, contact the scale manufacturer.
SV0401	IMPROPER V_READY OFF	Although the ready signal (PRDY) of the position control was ON, the ready signal (VRDY) of the velocity control was OFF.
SV0403	CARD/SOFT MISMATCH	The combination of the axis control card and the servo software is incorrect. Probable causes are given below. 1) The correct axis control card is not attached. 2) The correct servo software is not installed in flash memory.
SV0404	IMPROPER V_READY ON	Although the ready signal (PRDY) of the position control was OFF, the ready signal (VRDY) of the velocity control was ON.
SV0407	EXCESS ERROR	T The difference value of the amount of positional deviation for the synchronization axis exceeded the setting value. (during synchronous control only)
SV0409	DETECT ABNORMAL TORQUE	An abnormal load was detected on the servo motor, during Cs axis, or spindle positioning (T series) axis. The alarm can be canceled by RESET.
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter (No. 1829) setting value.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter (No.1828) setting value.
SV0413	LSI OVERFLOW	The counter for the amount of positional deviation overflowed
SV0415	MOTION VALUE OVERFLOW	The velocity exceeding the travel velocity limit was commanded.

Number	Message	Description
SV0417	ILL DGTL SERVO PARAMETER	A digital serve parameter setting is incorrect. [When bit 4 of diagnosis information No. 203 is 1.] An illegal parameter was detected by the servo software. Identify the cause with reference to diagnosis information No. 352. [When bit 4 of diagnosis information No. 203 is 0.] The CNC software detected an illegal parameter. Probable causes are given below (see diagnosis information No. 280). 1) The value specified in parameter No. 2020 as the motor model falls outside the specified range. 2) The motor rotation direction in parameter No. 2022 is not set to a correct value (111 or -111). 3) The speed feedback pulse count per motor rotation in parameter No. 2023 is set to a negative or other incorrect value. 4) The position feedback pulse count per motor rotation in parameter No. 2024 is set to a negative or other incorrect value.
SV0420	SYNC TORQUE EXCESS	In feed axis control , for synchronization, the difference value of torque between a master and slave axes exceeded the parameter (No. 2031) setting value. This alarm occurs for a master axis.
SV0421	EXCESS ERROR(SEMI-FULL)	The difference between the feedback from the semi and full sides exceeded the setting of parameter No.1729.
SV0422	EXCESS VELOCITY IN TORQUE	In torque control, the commanded permissible velocity was exceeded.
SV0423	EXCESS ERROR IN TORQUE	In torque control, the total permissible move value specified as a parameter was exceeded.
SV0430	SV MOTOR OVERHEAT	The servo motor has overheated.
SV0431	CNV. OVERLOAD	Power Supply (PS) : Overheat Servo Amplifier : Overheat
SV0432	CNV. LOW VOLT CONTROL	Power Supply (PS) : The control power supply voltage has dropped. Servo Amplifier : The control power supply voltage has dropped.
SV0433	CNV. LOW VOLT DC LINK	Power Supply (PS) : Low DC link voltage Servo Amplifier : Low DC link voltage
SV0434	INV. LOW VOLT CONTROL	Servo Amplifier : Low control power voltage
SV0435	INV. LOW VOLT DC LINK	Servo Amplifier : Low DC link voltage
SV0436	SOFTTHERMAL(OVC)	The digital servo software detected a software thermal (OVC).
SV0437	CNV. OVERCURRENT POWER	Power Supply (PS) : Overcurrent on input circuit section.
SV0438	INV. ABNORMAL CURRENT	Servo Amplifier : Motor overcurrent
SV0439	CNV. OVER VOLT DC LINK	Power Supply (PS) : The DC link voltage is too high. Servo Amplifier : The DC link voltage is too high.
SV0440	CNV. EX DECELERATION POW.	Power Supply (PS) : Excessive generative discharge Servo Amplifier : Excessive generative discharge, or abnormal error in generative power circuit
SV0441	ABNORMAL CURRENT OFFSET	The digital servo software detected an abnormality in the motor current detection circuit.
SV0442	CNV. CHARGE FAILURE	Power Supply (PS) : The spare charge circuit for the DC link is abnormal.
SV0443	CNV. COOLING FAN FAILURE	Power Supply (PS) : Internal cooling fan failure. Servo Amplifier : Internal cooling fan failure.

Number	Message	Description
SV0444	INV. COOLING FAN FAILURE	Servo Amplifier : Internal cooling fan failure.
SV0445	SOFT DISCONNECT ALARM	The digital servo software detected a disconnected Pulsecoder.
SV0446	HARD DISCONNECT ALARM	The hardware detected a disconnected built–in Pulsecoder.
SV0447	HARD DISCONNECT(EXT)	The hardware detected a disconnected separate detector.
SV0448	UNMATCHED FEEDBACK ALARM	The sign of the feedback signal from the standalone detector is opposite to that from the feedback signal from the built–on Pulsecoder.
SV0449	INV. IPM ALARM	Servo Amplifier : The IPM (Intelligent Power Module) detected an alarm.
SV0453	SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the α Pulsecoder. Turn off the power to the CNC, then remove and insert the Pulsecoder cable. If this alarm is issued again, replace the Pulsecoder.
SV0454	ILLEGAL ROTOR POS DETECT	The magnetic pole detection function terminated abnormally. The magnetic pole could not be detected because the motor did not run.
SV0456	ILLEGAL CURRENT LOOP	An attempt was made to set the current loop that could not be set. The amplifier pulse module in use does not comply with HIGH SPEED HRV. Or, requirements to control are not satisfied in the system.
SV0458	CURRENT LOOP ERROR	The specified current loop differs from the actual current loop.
SV0459	HI HRV SETTING ERROR	For two axes whose servo axis numbers (parameter No. 1023) are consecutively even and odd numbers, HIGH SPEED HRV control is possible for one axis and impossible for the other.
SV0460	FSSB DISCONNECT	The FSSB connection was discontinued. Probable causes are: 1) The FSSB connection cable was disconnected or broken. 2) The amplifier was turned off . 3) In the amplifier, the low-voltage alarm occurred.
SV0462	SEND CNC DATA FAILED	The correct data could not be received on a slave side because of the FSSB communication error.
SV0463	SEND SLAVE DATA FAILED	The correct data could not be received in the servo software side because of the FSSB communication error.
SV0465	READ ID DATA FAILED	A read of the ID information for the amplifier has failed at power-on.
SV0466	MOTOR/AMP. COMBINATION	The maximum current of an amplifier is different to that of a motor. Probable causes are: 1) The connection command for an amplifier is incorrect. 2) The parameter (No.2165) setting is incorrect
SV0468	HI HRV SETTING ERROR(AMP)	An attempt was made to set up HIGH SPEED HRV control for use when the controlled axis of an amplifier for which HIGH SPEED HRV control could not be used.
SV0600	INV. DC LINK OVER CURRENT	DC link overcurrent.
SV0601	INV. RADIATOR FAN FAILURE	External radiator cooling fan failure.
SV0602	INV. OVERHEAT	The servo motor has overheated.
SV0603	INV. IPM ALARM(OH)	The IPM (Intelligent Power Module) detected an overheat alarm.
SV0604	AMP. COMMUNICATION ERROR	The communication between Servo amplifier and Power Supply (PS) is in error.

Number	Message	Description
SV0605	CNV. EX. DISCHARGE POW.	Power Supply (PS) : The motor regenerative power is too much.
SV0606	CNV. RADIATOR FAN FAILURE	Power Supply (PS) : External radiator cooling fan failure.
SV0607	CNV. SINGLE PHASE FAILURE	Power Supply (PS) : The input power supply has a missing phase.
SV0646	ABNORMAL ANALOG SIGNAL(EXT)	An error occurred in the analog 1Vp-p output of the separate detector. The separate detector, cable, or separate detector interface unit may be failed.
SV1025	V_READY ON (INITIALIZING)	The ready signal (VRDY) of the velocity control which should be OFF is ON while the servo control is ON.
SV1026	ILLEGAL AXIS ARRANGE	The parameter for servo axis arrange is not set correctly. A negative value, duplicate value, or greater value than the number of control axes was set to the parameter No. 1023 "The servo axis number of each axis."
SV1055	ILLEGAL TANDEM AXIS	In tandem control, the setting of the parameter No. 1023 is incorrect.
SV1056	ILLEGAL TANDEM PAIR	In tandem control, the setting of the parameter TDM (No.1817#6) is incorrect.
SV1067	FSSB:CONFIGURATION ERROR(SOFT)	An FSSB configuration error occurred (detected by software). The connected amplifier type is incompatible with the FSSB setting value.
SV1100	S-COMP. VALUE OVERFLOW	M The amount of compensation for the simple straightness exceeded a maximum value of 32767.
SV5134	FSSB:OPEN READY TIME OUT	In the initialization, the FSSB could not be in an open ready sate. The axis card is thought to be defective.
SV5136	FSSB:NUMBER OF AMP. IS INSUFFICIENT	The number of amplifier identified by the FSSB is insufficient than the number of control axes. Or, the setting of the number of axes or the amplifier connection is in error.
SV5137	FSSB:CONFIGURATION ERROR	An FSSB configuration error occurred. The connecting amplifier type is incompatible with the FSSB setting value.
SV5139	FSSB:ERROR	Servo initialization has not completed successfully. It is probable that an optical cable failed or a connection between the amplifier and another module failed.
SV5197	FSSB:OPEN TIME OUT	The initialization of the FSSB was completed, but it could not be opened. Or, the connection between the CNC and the amplifier in is incorrect.

(6) Overtravel alarms (OT alarm)

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	Exceeded the positive side stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	Exceeded the negative side stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	Exceeded the positive side stored stroke check 2. T Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	Exceeded the negative side stored stroke check 2. T Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the negative direction.
OT0504	+ OVERTRAVEL (SOFT 3)	Exceeded the positive side stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	Exceeded the - side stored stroke check 3.

Number	Message	Description
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0508	INTERFERENCE:+	T A tool moving in the positive direction along the n axis has fouled another tool post.
OT0509	INTERFERENCE:-	T A tool moving in the negative direction along the n axis has fouled another tool post.
OT0510	+ OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the negative direction during the stroke check before movement.
OT0511	- OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the positive direction during the stroke check before movement.

(7) Memory file alarms (IO alarm)

Number	Message	Description
IO1001	FILE ACCESS ERROR	The resident–type file system could not be accessed as an error occurred in the resident–type file system.
IO1002	FILE SYSTEM ERROR	The file could not be accessed as an error occurred in the CNC file system.
IO1030	CHECK SUM ERROR	The checksum of the CNC part program storage memory is incorrect.
IO1032	MEMORY ACCESS OVER RANGE	Accessing of data occurred outside the CNC part program storage memory range.
IO1104	OVER MAXIMUM TOOL LIFE PAIRS	The maximum number of tool life management pairs is exceeded. Modify the setting of the maximum number of tool life management pairs in parameter No. 6813.

(8) Alarms requiring power to be turned off (PW alarm)

Number	Message	Description
PW0000	POWER MUST BE OFF	A parameter was set for which the power must be turned OFF then ON again.
PW0001	X-ADDRESS(*DEC) IS NOT ASSIGNED.	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: 1) During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position.
PW0002	PMC address is not correct(AXIS).	The address to assign the axis signal is incorrect. This alarm may occur in the following case: 1) The parameter No.3021 setting is incorrect.

Number	Message	Description
PW0003	PMC address is not correct(SPINDLE).	The address to assign the spindle signal is incorrect. This alarm may occur in the following case: 1) The parameter No.3022 setting is incorrect.
PW0006	POWER MUST BE OFF (ILL-EXEC-CHK)	The malfunction prevention function detected an alarm to require the power off.
PW0007	X-ADDRESS(SKIP) IS NOT ASSIGNED	The X address of PMC could not be assigned correctly. Possible causes are: 1) During the set of parameter No. 3012, the skip signal of the X address was not assigned correctly. 2) During the set of parameter No. 3019, the address other than the skip signal of the X address was not assigned correctly.
PW1102	ILLEGAL PARAMETER (I-COMP.)	The parameter for setting slope compensation is incorrect. This alarm occurs in the following cases: 1) When the size relationship between the slope compensation point Nos. is incorrect 2) When the slope compensation point is not located between the most negative side and most positive side of pitch error compensation 3) When the compensation per compensation point is too small or too great.
PW1110	ILLEGAL PARAMETER (SERVO MOTOR SPINDLE)	The parameter for the spindle controlled with the servo motor is not set correctly.
PW1111	ILLEGAL SPINDLE NUMBER (SERVO MOTOR SPINDLE)	The spindle number (parameter No. 11010) or spindle amplifier number (parameter No. 3717) for the spindle controlled with the servo motor is not set correctly.
PW5046	ILLEGAL PARAMETER (S-COMP.)	M The parameter for setting simple straightness compensation is incorrect.

(9) Spindle alarms (SP alarm)

Number	Message	Description
SP0740	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the stopped spindle has exceeded the set value during rigid tapping.
SP0741	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the moving spindle has exceeded the set value during rigid tapping.
SP0742	RIGID TAP ALARM : LSI OVERFLOW	An LSI overflow has occurred for the spindle during rigid tapping.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning (T series), rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
SP0754	ABNORMAL TORQUE	An abnormal load was detected in a spindle motor. The alarm can be canceled by RESET.
SP1202	SPINDLE SELECT ERROR	In a multi spindle control, the spindle number other than the valid spindle number was selected by a position coder select signal. An attempt was made to select the spindle number of the system having no valid spindle.
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
SP1221	ILLEGAL MOTOR NUMBER	The spindle No. and the motor No. are incorrectly matched.
SP1224	ILLEGAL SPINDLE-POSITION CODER GEAR RATIO	The spindle–position coder gear ratio was incorrect.

Number	Message	Description
SP1225	CRC ERROR (SERIAL SPINDLE)	A CRC error (communications error) occurred in communications between the CNC and the serial spindle amplifier.
SP1226	FRAMING ERROR (SERIAL SPINDLE)	A framing error occurred in communications between the CNC and the serial spindle amplifier.
SP1227	RECEIVING ERROR (SERIAL SPINDLE)	A receive error occurred in communications between the CNC and the serial spindle amplifier.
SP1228	COMMUNICATION ERROR (SERIAL SPINDLE)	A communications error occurred between the CNC and the serial spindle amplifier.
SP1229	COMMUNICATION ERROR SERIAL SPINDLE AMP.	A communications error occurred between serial spindle amplifiers (motor Nos. 1 and 2, or motor Nos. 3–4).
SP1231	SPINDLE EXCESS ERROR (MOVING)	The position deviation during spindle rotation was greater than the value set in parameters.
SP1232	SPINDLE EXCESS ERROR (STOP)	The position deviation during spindle stop was greater than the value set in parameters.
SP1233	POSITION CODER OVERFLOW	The error counter/speed instruction value of the position coder overflowed.
SP1234	GRID SHIFT OVERFLOW	Grid shift overflowed.
SP1240	DISCONNECT POSITION CODER	The analog spindle position coder is broken.
SP1241	D/A CONVERTER ERROR	The D/A converter for controlling analog spindles is erroneous.
SP1243	ILLEGAL SPINDLE PARAMETER SETTING(GAIN)	The setting for the spindle position gain is incorrect.
SP1244	MOTION VALUE OVERFLOW	The amount of distribution to a spindle is too much
SP1245	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1246	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1247	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1969	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1970	SPINDLE CONTROL ERROR	Initialization of spindle control ended in error.
SP1971	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1972	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1974	ANALOG SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1975	ANALOG SPINDLE CONTROL ERROR	An position coder error was detected on the analog spindle.
SP1976	SERIAL SPINDLE COMMUNICATION ERROR	The amplifier No. could not be set to the serial spindle amplifier.
SP1977	SERIAL SPINDLE COMMUNICATION ERROR	An error occurred in the spindle control software.
SP1978	SERIAL SPINDLE COMMUNICATION ERROR	A time–out was detected during communications with the serial spindle amplifier.
SP1979	SERIAL SPINDLE COMMUNICATION ERROR	The communications sequence was no longer correct during communications with the serial spindle amplifier.
SP1980	SERIAL SPINDLE AMP. ERROR	Defective SIC–LSI on serial spindle amplifier
SP1981	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the analog spindle amplifier side.
SP1982	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the serial spindle amplifier side.
SP1983	SERIAL SPINDLE AMP. ERROR	Could not clear on the spindle amplifier side.
SP1984	SERIAL SPINDLE AMP. ERROR	An error occurred during re–initialization of the spindle amplifier.
SP1985	SERIAL SPINDLE CONTROL ERROR	Failed to automatically set parameters
SP1986	SERIAL SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1987	SERIAL SPINDLE CONTROL ERROR	Defective SIC–LSI on the CNC
SP1988	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1989	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.

Number	Message	Description
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Alternatively, the number of spindles exceeded the maximum number allowed in the system. Check to see the following parameter. (No.3701#1,#4,#5, 3716, 3717)
SP1998	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1999	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.

(10) Alarm list (serial spindle)

When a serial spindle alarm occurs, the following number is displayed on the CNC.

NOTE

*1 Note that the meanings of the Spindle Amplifier indications differ depending on which LED, the red or yellow LED, is on. When the red LED is on, the Spindle Amplifier indicates a 2-digit alarm number. When the yellow LED is on, the Spindle Amplifier indicates an error number that designates a sequence problem (for example, when a rotation command is entered with the emergency stop state not released).
See "Error Codes (Serial Spindle)."

*2 For serial spindle alarms with a number not listed, refer to the following documents depending on the spindle motor to which a connection is actually made.
- FANUC AC SPINDLE MOTOR αi series Maintenance Manual (B-65285EN)
- Technical Report etc.

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9001	SSPA:01 MOTOR OVERHEAT	01	1 Check and correct the peripheral temperature and load status. 2 If the cooling fan stops, replace it.	The internal temperature of the motor exceeds the specified level. The motor is used in excess of the continuous rating, or the cooling component is abnormal.
SP9002	SSPA:02 EX DEVIATION SPEED	02	1 Check and correct the cutting conditions to decrease the load. 2 Correct parameter No. 4082.	The motor speed cannot follow a specified speed. An excessive motor load torque is detected. The acceleration/deceleration time in parameter No. 4082 is insufficient.
SP9003	SSPA:03 DC-LINK FUSE IS BROKEN	03	1 Replace the Spindle Amplifier. 2 Check the motor insulation status.	The Power Supply (PS) becomes ready (“00” is indicated), but the DC link voltage is too low in the Spindle Amplifier. The fuse in the DC link section in the Spindle Amplifier is blown. (The power device is damaged or the motor is ground-fault.)
SP9004	SSPA:04 POWER SUPPLY ERROR	04	Check the voltage of the power input to the Power Supply (PS) and the connection status.	The Power Supply (PS) found a missing power supply phase. (Power Supply (PS) alarm 5)

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9006	THERMAL SENSOR DISCONNECT	06	1 Check and correct the parameter. 2 Replace the feedback cable.	The temperature sensor of the motor is disconnected.
SP9007	SSPA:07 OVER SPEED	07	Check for a sequence error. (For example, check whether spindle synchronization was specified when the spindle could not be turned.)	The motor speed has exceeded 115% of its rated speed. When the spindle axis was in position control mode, positional deviations were accumulated excessively (SFR and SRV were turned off during spindle synchronization.)
SP9009	SSPA:09 OVERHEAT MAIN CIRCUIT	09	1 Improve the heat sink cooling status. 2 If the external radiator cooling fan stops, replace the Spindle Amplifier.	The temperature in the power semiconductor cooling radiator is abnormally high.
SP9010	SSPA:10 LOW VOLT INPUT POWER	09	1 The input power voltage in the Power Supply (PS) is too low. 2 The power cable between amplifiers is abnormal. 3 The spindle amplifier is abnormal.	A drop in the input power voltage in the spindle amplifier is detected.
SP9011	SSPA:11 OVERVOLT POWER CIRCUIT	11	1 Check the selected Power Supply (PS). 2 Check the input power voltage and change in power during motor deceleration. If the voltage exceeds 253 VAC (for the 200-V system) or 530 VAC (for the 400-V system), improve the power supply impedance.	Overvoltage of the DC link section of the Power Supply (PS) was detected. (Power Supply (PS) alarm indication: 7) Power Supply (PS) selection error. (The maximum output specification of the Power Supply (PS) is exceeded.)
SP9012	SSPA:12 OVERCURRENT POWER CIRCUIT	12	1 Check the motor insulation status. 2 Check the spindle parameters. 3 Replace the Spindle Amplifier.	The motor current is abnormally high. A motor-specific parameter does not match the motor model. Poor motor insulation
SP9013	SSPA:13 CPU DATA MEMORY FAULT	13	Replace the Spindle Amplifier control printed circuit board.	Abnormality in an Spindle Amplifier control circuit component is detected. (Internal RAM is abnormal.)
SP9015	SSPA:15 SPINDLE SWITCHING FAULT	15	1 Check and correct the ladder sequence. 2 Replace the switching MC.	The switch sequence in spindle switch/output switch operation is abnormal. The switching MC contact status check signal and command do not match.

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9018	SSPA:18 SUMCHECK ERROR PROGRAM ROM	18	Replace the Spindle Amplifier control printed circuit board.	Abnormality in an Spindle Amplifier control circuit component is detected. (Program ROM data is abnormal.)
SP9019	SSPA:19 EXCESS OFFSET CURRENT U	19	Replace the Spindle Amplifier.	Abnormality in an Spindle Amplifier component is detected. (The initial value for the U phase current detection circuit is abnormal.)
SP9020	SSPA:20 EXCESS OFFSET CURRENT V	20	Replace the Spindle Amplifier.	Abnormality in an Spindle Amplifier component is detected. (The initial value of the V phase current detection circuit is abnormal.)
SP9021	POS SENSOR POLARITY ERROR	21	Check and correct the parameters. (Parameter No. 4000#0, 4001#4)	The polarity parameter setting of the position sensor is wrong.
SP9024	SSPA:24 SERIAL TRANSFER ERROR	24	1 Place the CNC-to-spindle cable away from the power cable. 2 Replace the cable.	The CNC power is turned off (normal power-off or broken cable). An error is detected in communication data transferred to the CNC.
SP9027	SSPA:27 DISCONNECT POSITION CODER	27	Replace the cable.	The spindle position coder (connector JYA3) signal is abnormal.
SP9029	SSPA:29 OVERLOAD	29	Check and correct the load status.	Excessive load has been applied continuously for a certain period of time. (This alarm is issued also when the motor shaft has been locked in the excitation state.)
SP9030	SSPA:30 OVERCURRENT INPUT CIRCUIT	30	Check and correct the power supply voltage.	Overcurrent is detected in Power Supply (PS) main circuit input. (Power Supply (PS) alarm indication: 1) Unbalanced power supply. Power Supply (PS) selection error (The maximum Power Supply (PS) output specification is exceeded.)
SP9031	SSPA:31 MOTOR LOCK OR DISCONNECT DETECTOR	31	1 Check and correct the load status. 2 Replace the motor sensor cable (JYA2).	The motor cannot rotate at a specified speed. (A level not exceeding the SST level for the rotation command has existed continuously.)
SP9032	SSPA:32 SIC-LSI RAM FAULT	32	Replace the Spindle Amplifier control printed circuit board.	Abnormality in an Spindle Amplifier control circuit component is detected. (The LSI device for serial transfer is abnormal.)
SP9033	SSPA:33 SHORTAGE POWER CHARGE	33	1 Check and correct the power supply voltage. 2 Replace the Power Supply (PS).	The DC power voltage in the power circuit is insufficient when the magnetic contactor is turned on. (Power Supply (PS) alarm indication: 5) (Such as open phase and defective charging resistor).

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9034	SSPA:34 ILLEGAL PARAMETER	34	Correct a parameter value according to the FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN). If the parameter number is unknown, connect the spindle check board, and check the indicated parameter.	Parameter data exceeding the allowable limit is set.
SP9036	SSPA:36 OVERFLOW ERROR COUNTER	36	Check whether the position gain value is too large, and correct the value.	An error counter overflow occurred.
SP9037	SSPA:37 ILLEGAL SETTING VELOCITY DETECTOR	37	Correct the value according to the FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN).	The setting of the parameter for the number of pulses in the speed detector is incorrect.
SP9041	SSPA:41 ILLEGAL 1REV SIGN OF POSITION CODER	41	1 Check and correct the parameter. 2 Replace the cable.	1 The 1-rotation signal of the spindle position coder (connector JYA3) is abnormal. 2 Parameter setting error
SP9042	SSPA:42 NO 1REV SIGN OF POSITION CODER	42	Replace the cable.	The 1-rotation signal of the spindle position coder (connector JY4) is disconnected.
SP9043	SSPA:43 DISCONNECT POSITION CODER DEF. SPEED	43	Replace the cable.	The differential speed position coder signal (connector JYA3S) in the submodule SW is abnormal.
SP9047	SSPA:47 ILLEGAL SIGNAL OF POSITION CODER	47	1 Replace the cable. 2 Correct the cable layout (vicinity of the power line).	The A/B phase signal of the spindle position coder (connector JYA3) is abnormal. The relationship between the A/B phase and 1-rotation signal is incorrect (Pulse interval mismatch).
SP9049	SSPA:49 DEF. SPEED IS OVER VALUE	49	Check whether the calculated differential speed value exceeds the maximum motor speed.	In differential speed mode, the speed of the other spindle converted to the speed of the local spindle has exceeded the allowable limit (the differential speed is calculated by multiplying the speed of the other spindle by the gear ratio).
SP9050	SSPA:50 SYNCRONOUS VALUE IS OVER SPEED	50	Check whether the calculated value exceeds the maximum motor speed.	In spindle synchronization, the speed command calculation value exceeded the allowable limit (the motor speed is calculated by multiplying the specified spindle speed by the gear ratio).
SP9051	SSPA:51 LOW VOLT POWER CIRCUIT	51	1 Check and correct the power supply voltage. 2 Replace the MC.	Input voltage drop was detected. (Power Supply (PS) alarm indication: 4) (Momentary power failure or poor MC contact)

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9052	SSPA:52 ITP FAULT 1	52	1 Replace the Spindle Amplifier control printed circuit board. 2 Replace the main board or sub CPU board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier (the ITP signal stopped).
SP9053	SSPA:53 ITP FAULT 2	53	1 Replace the Spindle Amplifier control printed circuit board. 2 Replace the main board or sub CPU board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier the ITP signal stopped).
SP9054	SSPA:54 OVERCURRENT	54	Review the load state.	An overload current was detected.
SP9055	SSPA:55 ILLEGAL POWER LINE	55	1 Replace the magnetic contactor. 2 Check and correct the sequence.	The power line state signal of the magnetic contactor for selecting a spindle or output is abnormal.
SP9056	COOLING FAN FAILURE	56	Replace the internal cooling fan.	The internal cooling fan stopped.
SP9057	CONV. EX. DECELERATION POW.	57	1 Decrease the acceleration/deceleration duty. 2 Check the cooling condition (peripheral temperature). 3 If the cooling fan stops, replace the resistor. 4 If the resistance is abnormal, replace the resistor.	An overload was detected in the regenerative resistance. (Power Supply (PS) alarm indication: H) Thermostat operation or short-time overload was detected. The regenerative resistor was disconnected, or an abnormal resistance was detected.
SP9058	CNV. OVERLOAD	58	1 Check the Power Supply (PS) cooling status. 2 Replace the Power Supply (PS).	The temperature of the radiator of the Power Supply (PS) has increased abnormally. (Power Supply (PS) alarm indication: 3)
SP9059	CNV. COOLING FAN FAILURE	59	Replace the Power Supply (PS).	The internal cooling fan for the Power Supply (PS) stopped. (Power Supply (PS) alarm indication: 2)
SP9061	SERIAL SPINDLE ALARM	61	Check parameter settings.	The error between the semi-closed and full-closed sides when the dual position feedback function is used is too large.
SP9065	SERIAL SPINDLE ALARM	65	1 Check parameter settings. 2 Check sensor connections and signals. 3 Check power line connections.	The move distance is too long when the magnetic pole is confirmed (synchronization spindle)
SP9066	COM. ERROR BETWEEN SP AMPS	66	1 Replace the cable. 2 Check and correct the connection.	An error was found in communication between spindle amplifiers (connector JX4).

Number	Message	Amplifier indicatio n (*1)	Faulty location and remedy	Description
SP9073	MOTOR SENSOR DISCONNECTED	73	1 Replace the feedback cable. 2 Check the shield processing. 3 Check and correct the connection. 4 Adjust the sensor.	The motor sensor feedback signal is not present (connector JYA2).
SP9080	ALARM AT THE OTHER SP AMP.	80	Remove the cause of the alarm of the remote Spindle Amplifier.	During inter-Spindle Amplifier communication, an alarm was generated on the remote Spindle Amplifier.
SP9081	1-ROT MOTOR SENSOR ERROR	81	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the motor sensor cannot be correctly detected(connector JYA2).
SP9082	NO 1-ROT MOTOR SENSOR	82	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the motor sensor is not generated(connector JYA2).
SP9083	MOTOR SENSOR SIGNAL ERROR	83	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a motor sensor feedback signal(connector JYA2).
SP9084	SPNDL SENSOR DISCONNECTED	84	1 Replace the feedback cable. 2 Check the shield processing. 3 Check and correct the connection. 4 Check and correct the parameter. 5 Adjust the sensor.	The spindle sensor feedback signal is not present(connector JYA4).
SP9085	1-ROT SPNDL SENSOR ERROR	85	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the spindle sensor cannot be correctly detected(connector JYA4).
SP9086	NO 1-ROT SPNDL SENSOR	86	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the spindle sensor is not generated(connector JYA4).
SP9087	SPNDL SENSOR SIGNAL ERROR	87	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a spindle sensor feedback signal(connector JYA4).
SP9088	COOLING RADI FAN FAILURE	88	Replace the Spindle Amplifier external radiator cooling fan.	The external radiator cooling fan stopped.
SP9089	COOLING RADI FAN FAILURE	89	1 Check the connection between the Spindle Amplifier and the submodule SM (SSM). 2 Replace the submodule SM(SSM). 3 Replace the Spindle Amplifier control printed-circuit board.	Submodule SM (SSM) error (synchronous spindle)

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9110	AMP COMMUNICATION ERROR	b0	1 Replace the communication cable between Spindle Amplifier and Power Supply (PS). 2 Replace the Spindle Amplifier or Power Supply (PS) control printed circuit board.	Communication error between Spindle Amplifier and Power Supply (PS).
SP9111	CONV. LOW VOLT CONTROL	b1	Replace the Power Supply (PS) control printed circuit board.	Low Power Supply (PS) control power supply voltage (Power Supply (PS) indication = 6)
SP9112	CONV. EX. DISCHARGE POW.	b2	1 Check the regenerative resistance. 2 Check the motor selection. 3 Replace the Power Supply (PS)	Excessive Power Supply (PS) regenerative power (Power Supply (PS) indication = 8)
SP9113	CONV. COOLING FAN FAILURE	b3	Replace the external radiator cooling fan for Power Supply (PS).	Stopped the external radiator cooling fan for Power Supply (PS) (Power Supply (PS) indication = A)
SP9120	COMMUNICATION DATA ERROR	C0	1 Replace the communication cable between CNC and Spindle Amplifier. 2 Replace the Spindle Amplifier control printed circuit board. 3 Replace the CNC side main board or sub CPU board.	Communication data alarm
SP9121	COMMUNICATION DATA ERROR	C1	1 Replace the communication cable between CNC and Spindle Amplifier. 2 Replace the Spindle Amplifier control printed circuit board. 3 Replace the CNC side main board or sub CPU board.	Communication data alarm
SP9122	COMMUNICATION DATA ERROR	C2	1 Replace the communication cable between CNC and Spindle Amplifier. 2 Replace the Spindle Amplifier control printed circuit board. 3 Replace the CNC side main board or sub CPU board.	Communication data alarm
SP9123	SERIAL SPINDLE ALARM	C3	Replace the submodule SW(SSW).	Submodule SW (SSW) error (spindle switching)
SP9135	SAFETY SPEED ZERO ERROR(SP)	d5	Perform operation within the safety speed zero range.	The motor position exceeded the safety speed zero monitoring width.
SP9136	MISMATCH RESULT OF SAFETY SPEED ZERO CHECK(SP)	d6	Replace the spindle amplifier control printed circuit board.	The spindle amplifier speed zero determination result did not match the CNC speed zero determination result.

Error codes (serial spindle)

NOTE

*1 Note that the meanings of the Spindle Amplifier indications differ depending on which LED, the red or yellow LED, is on. When the yellow LED is on, an error code is indicated with a 2-digit number. When the red LED is on, the Spindle Amplifier indicates the number of an alarm generated in the serial spindle.
→ See "(10) Serial spindle alarms (SP alarm)."

Diagnosis indication(*1)	Faulty location and remedy	Description
01	Although neither *ESP (emergency stop signal; there are two types of signals including the input signal and contact signal of Power Supply (PS)) nor MRDY (machine ready signal) is input, SFR (forward rotation signal)/SRF (reverse rotation signal)/ORCM (orientation command) is input.	Check the *ESP and MRDY sequence. For MRDY, pay attention to the parameter setting regarding the use of the MRDY signal (parameter No. 4001#0).
03	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No. 4002 = 0, 0, 0, 0), but a Cs contour control command is input. In this case, the motor is not excited.	Check the parameter settings.
04	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No.4002 = 0, 0, 0,), but a servo mode (rigid tapping, spindle positioning, etc.) or spindle synchronization command is input. In this case, the motor is not excited.	Check the parameter settings.
05	The orientation function option parameter is not specified, but ORCM (orientation command) is input.	Check the orientation function parameter settings.
06	The output switching control function option parameter is not specified, but low-speed characteristic winding is selected (RCH = 1).	Check the output switching control function parameter settings and the power line state check signal (RCH).
07	A Cs contour control command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
08	A servo mode (rigid tapping, spindle positioning, etc.) control command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
09	A spindle synchronization command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
10	A Cs contour control command is input, but another mode (servo mode, spindle synchronization, or orientation) is specified.	Do not switch to another mode during a Cs contour control command. Before moving to another mode, cancel the Cs contour control command.
11	A servo mode (rigid tapping, spindle positioning, etc.) command is input, but another mode (Cs contour control, spindle synchronization, or orientation) is specified.	Do not switch to another mode during a servo mode command. Before moving to another mode, cancel the servo mode command.

Diagnosis indication(*1)	Faulty location and remedy	Description
12	A spindle synchronization command is input, but another mode (Cs contour control, servo mode, or orientation) is specified.	Do not switch to another mode during a spindle synchronization command. Before moving to another mode, cancel the spindle synchronization command.
14	Both SFR (clockwise rotation command) and SRV (counterclockwise rotation command) are input at the same time.	Issue either of them.
17	The speed detector parameter settings (bits 2, 1, and 0 of parameter No. 4011) are not valid. There is no corresponding speed detector.	Check the parameter settings.
18	The parameter settings are such that a position sensor is not used (position control not performed (bits 3, 2, 1, and 0 of parameter No. 4002), but position coder system orientation is issued.	Check the parameter settings and the input signal.
24	If index is performed continuously in position coder system orientation, an incremental operation is performed first (INCMD = 1), then an absolute position command (INCMD = 0) is input.	Check INCMD (incremental command). If an absolute position command is to follow, be sure to perform absolute position command orientation first.
29	The parameter settings are such that the shortest-time orientation function is used (bit 6 of parameter No. 4018 = 0, Nos. 4320 to 4323 ≠0).	In the α*i* series spindle amplifier, the shortest-time orientation function cannot be used. Use other system orientation.
36	The submodule SM (SSM) is faulty or the connection between Spindle Amplifier and SSM is in error.	Submodule SM (SSM) error (synchronous spindle)

(11) Overheat alarms (OH alarm)

Number	Message	Description
OH0700	LOCKER OVERHEAT	CNC cabinet overheat
OH0701	FAN MOTOR STOP	PCB cooling fan motor abnormality
OH0704	OVERHEAT	T Spindle overheat due to detection of changes in the spindle speed 1) When the cutting load is large, offload the cutting conditions. 2) Check if the cutting tool became dull. 3) Check if the spindle amplifier malfunctions.

(12) Other alarms (DS alarm)

Number	Message	Description
DS0001	SYNC EXCESS ERROR (POS DEV)	In feed axis control , the difference in the amount of positional deviation between the master and slave axes exceeded the parameter (No. 8323) setting value. This alarm occurs only for the slave axis.
DS0003	SYNCHRONIZE ADJUST MODE	The system is in the synchronize adjust mode.
DS0004	EXCESS MAXIMUM FEEDRATE	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
DS0005	EXCESS MAXIMUM ACCELERATION	The malfunction prevention function detected the command in which a value exceeding the maximum acceleration was specified.

Number	Message	Description
DS0006	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0007	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0008	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0009	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0010	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0011	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0012	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0013	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0014	TOOL CHANGE DETECT MACHINE LOCK	A machine lock is turned on for the Z axis for which the tool is being changed.
DS0015	TOOL CHANGE DETECT MIRROR IMAGE	A mirror image is turned on for the Z axis for which the tool is being changed.
DS0016	SERIAL DCL:FOLLOW-UP ERROR	1) The settings of parameters No.1883 and No.1884 fall outside the range. 2) The current position at establishment of the reference position subtracted by the distance between the reference positions (detection unit) exceeded ±2147483647. Change the current position or reference position to prevent this situation.
DS0017	SERIAL DCL:REF-POS ESTABLISH ERR	The travel amount at the FL speed at establishment of the reference position exceeded the setting of parameter No. 14010.
DS0018	SERIAL DCL:MISMATCH(SSYNC CTRL)	Of the master and slave axes for feed axis control, one axis is a linear scale with the origin and the other is not a linear scale with the reference position. In such a configuration, the feed axis control selection signal (SYNC<Gn138> or SYNCJ <Gn140>) needs to be set to 0 to establish the origin.
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the perpendicular axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the perpendicular axis.
DS0021	START ERROR(ONE TOUCH MACRO)	A macro program start operation cannot be accepted. 1) The feed hold signal *SP is 0. 2) An alarm is generated. 3) The SRN signal is 1.
DS0023	ILLEGAL PARAMETER (I-COMP VAL)	The setting of the inclination compensation parameter is incorrect. The compensation per compensation point is too large or too small.
DS0024	UINT SIGNAL WAS ILLEGALLY INPUT	An interruption custom macro was started during movement to the machining restart position at the dry run speed.

Number	Message	Description
DS0025	G60 CANNOT BE EXECUTED	M The state of a mirror image is different between the time when look-ahead of a block for unidirectional positioning was performed and the time when execution of the block was started, so unidirectional positioning cannot be performed. Modify the program.
DS0026	MISMATCH OF ANGULAR AXIS(D.C.S)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.
DS0027	MISMATCH OF SYNCHRONOUS AXIS(D.C.S)	Master/slave axes of axis synchronous control, one of them is the linear scale with distance-coded reference marks, and the other of them is not the linear scale with distance-coded reference marks. Please establish reference position with the input signal SYNCn<G138>, SYNCJn<G140> or parameter setting to 0.
DS0059	SPECIFIED NUMBER NOT FOUND	[External data I/O] The No. specified for a program No. or sequence No. search could not be found. There was an I/O request issued for offset (tool data), but either no tool numbers have been input since power ON or there is no data for the entered tool No. [External workpiece No. search] The program corresponding to the specified workpiece No. could not be found.
DS0131	TOO MANY MESSAGE	An attempt was made to display an external operator message or external alarm message, but five or more displays were required simultaneously.
DS0132	MESSAGE NUMBER NOT FOUND	An attempt to cancel an external operator message or external alarm message failed because the specified message number was not found.
DS0133	TOO LARGE NUMBER	A value other than 0 to 4095 was specified as the external operator message or the external alarm message number.
DS0300	APC ALARM: NEED REF RETURN	A setting to zero position for the absolute position detector (association with reference position and the counter value of the absolute position detector) is required. Perform the return to the reference position. This alarm may occur with other alarms simultaneously. In this case, other alarms must be handled first.
DS0306	APC ALARM: BATTERY VOLTAGE 0	The battery voltage of the absolute position detector has dropped to a level at which data can no longer be held. Or, the power was supplied to the Pulsecoder for the first time. If this problem recurs after the power is turned off and then back on, the battery or cable may be defective. Replace the battery with the machine turned on.
DS0307	APC ALARM: BATTERY LOW 1	The battery voltage of the absolute position detector has dropped to a level at which a replacement is required. Replace the battery with the machine turned on.
DS0308	APC ALARM: BATTERY LOW 2	The battery voltage of the absolute position detector dropped to a level at which a replacement was required in the past. (including during power off) Replace the battery with the machine turned on.

Number	Message	Description
DS0309	APC ALARM: REF RETURN IMPOSSIBLE	An attempt was made to set the zero point for the absolute position detector by MDI operation when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
DS0405	ZERO RETURN END NOT ON REF	The axis specified in automatic zero return was not at the correct zero point when positioning was completed. Perform zero return from a point whose distance from the zero return start position to the zero point is 2 or more revolutions of the motor. Other probable causes are: - The positional deviation after triggering the deceleration dog is less than 128. - Insufficient voltage or malfunctioning Pulsecoder.
DS1120	UNASSIGNED ADDRESS (HIGH)	The upper 4 bits (EIA4 to EIA7) of an external data I/O interface address signal are set to an undefined address (high bits).
DS1121	UNASSIGNED ADDRESS (LOW)	The lower 4 bits (EIA0 to EIA3) of an external data I/O interface address signal are set to an undefined address (low bits).
DS1124	OUTPUT REQUEST ERROR	OUTPUT REQUEST ERROR An output request was issued during external data output, or an output request was issued for an address that has no output data.
DS1128	DI.EIDLL OUT OF RANGE	The numerical value input by external data input signals ED0 to ED31 has exceeded the permissible range.
DS1130	SEARCH REQUEST NOT ACCEPTED	No requests can be accepted for a program No. or a sequence No. search as the system is not in the memory mode or the reset state.
DS1131	EXT-DATA ERROR (OTHER)	[External Data I/O] An attempt was made to input tool data for tool offset by a tool No. during loading by the G10 code.
DS1150	A/D CONVERT ALARM	A/D converter malfunction
DS1184	PARAMETER ERROR IN TORQUE	An invalid parameter was set for torque control. The torque constant parameter is set to “0”.
DS1448	ILLEGAL PARAMETER (D.C.S.)	The setting value of parameter for reference marks is satisfied the following any conditions. 1) A setting is made to use the absolute position detector (bit 5 (APC) of parameter No. 1815 is 1). 2) Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0. 3) Parameters 1821 and 1882 have identical settings. 4) The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting. 5) The setting value of parameters 1883 and 1884 are over the valid data range.
DS1449	REFERENCE MARK ARE DIFFERENT FROM PARAMETER	In case of distance coded linear scale I/F, the actual interval of reference marks is different from parameter (No.1821,1882) setting value.
DS1450	ZERO RETURN NOT FINISHED	When bit 0 (ZRN) of parameter No. 1005 is 0, if a manual reference position return has never been performed after power-up, the 1st reference position return (07h) is specified.
DS1451	IMPROPER PMC AXIS COMMAND	The PMC axes cannot be controlled in this state.

Number	Message	Description
DS1512	EXCESS VELOCITY	T The feedrate of the linear axis during polar coordinate interpolation exceeded the maximum cutting feedrate.
DS1933	NEED REF RETURN(SYNC:MIX:OVL)	T The relation between a machine coordinate of an axis in synchronization, composite, or superimposed control, and the absolute, or relative coordinate was displaced. Perform the manual return to the reference position.
DS2003	ILLEGAL PARAMETER SETTING FOR LIVE TOOL AXIS (PMC AX-CTRL)	The spindle controlled with the servo motor is set as a PMC-controlled axis.
DS2005	NOW GAIN TUNING	Automatic operation cannot be started during automatic speed gain adjustment. Start automatic operation after confirming the completion of automatic adjustment.
DS5340	PARAMETER CHECK SUM ERROR	The parameter check sum does not match the reference check sum because of a change in the parameters. Restore the parameters or set the reference check sum again.

(13) Malfunction prevention function alarms (IE alarm)

Number	Message	Description
IE0001	+ OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the positive side was exceeded.
IE0002	- OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the negative side was exceeded.
IE0003	+ OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the positive side was exceeded.
IE0004	- OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the negative side was exceeded.
IE0005	+ OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the positive side was exceeded.
IE0006	- OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the negative side was exceeded.
IE0007	EXCESS MAXIMUM REV. DATA	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
IE0008	ILLEGAL ACC/DEC	The malfunction prevention function detected the acceleration/deceleration error.
IE0009	ILLEGAL MCN COODINATE	The malfunction prevention function detected the displacement of a machine coordinate in the check point.

I PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

Appendix I, "PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING", consists of the following sections:

I.1 PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

Overview

By using this PC tool, you can make the memory card program file ("FANUCPRG.BIN") which is needed for the function "Memory Card Program Operation/Editing".
The maximum size of the memory card program file is 2048 Mbytes (2 Gbytes). The "Memory Card Program Operation/Editing" function needs the memory card which has the memory card program file on the FAT formatted memory card.
This PC tool can be operated on a PC in the marketplace and with following OS :

- Windows(R)NT4.0 Workstation (SP5 or later)
- Windows(R)2000 Professional (SP4 or later)
- Windows(R)XP Professional (SP2 or later)
- Windows(R)Vista Ultimate

And acceptable specification is as followings :

- Memory : 32Mbytes or more
- Hard disk : 10Mbytes or more free space and additional space for the memory card program file

I.1.1 Usage Notes

Before using this PC tool, please make sure there is no [temp] folder on the same place of this PC tool.
[temp] folder is created and used by this PC tool as work folder.
If [temp] folder is created, please do not access this folder.
[temp] folder and the files located in this folder will be deleted by this PC tool.

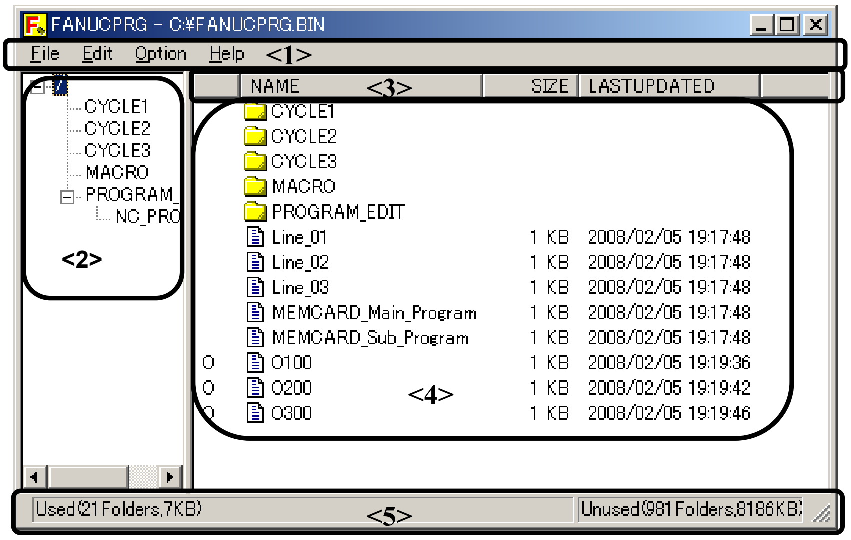
I.1.2 List of Functions of PC Tool

- Browsing the folders of the memory card program file
- Adding a program file
  Drag a program file from the Explorer or any other similar tool to this PC tool.
  (This operation is referred to below as "drag-in".)
- Saving, in text format for the Windows file system, a program file in the memory card program file
  Drag a program file from this PC tool to the Explorer or any other similar tool.
  (This operation is referred to below as "drag-out".)
- Renaming a program in the memory card program file
- Deleting a program in the memory card program file
- Creating a new folder into the memory card program file

- Renaming a folder in the memory card program file
- Deleting a folder in the memory card program file
- Display of free space on the memory card program file
- Sorting list view of the memory card program file

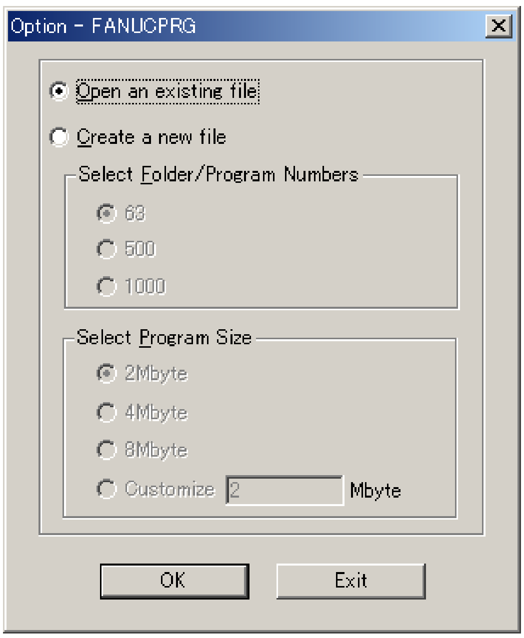
I.1.3 Explanation of Operations

- Outline of screen



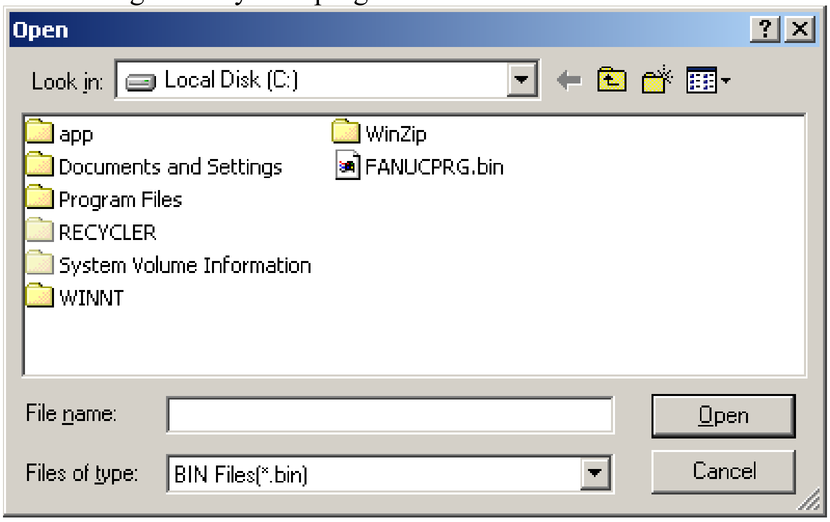
<1> Menu bar : The menu of this PC tool is displayed.
<2> Tree view : Browsing the folders of the memory card program file.
<3> Column : Attributes of each file or folder in the memory card program file.
<4> List view : Contents of selected folder are displayed.
<5> Status bar : Used and Unused space in the memory card program file are displayed.

- Initial Option dialogue window

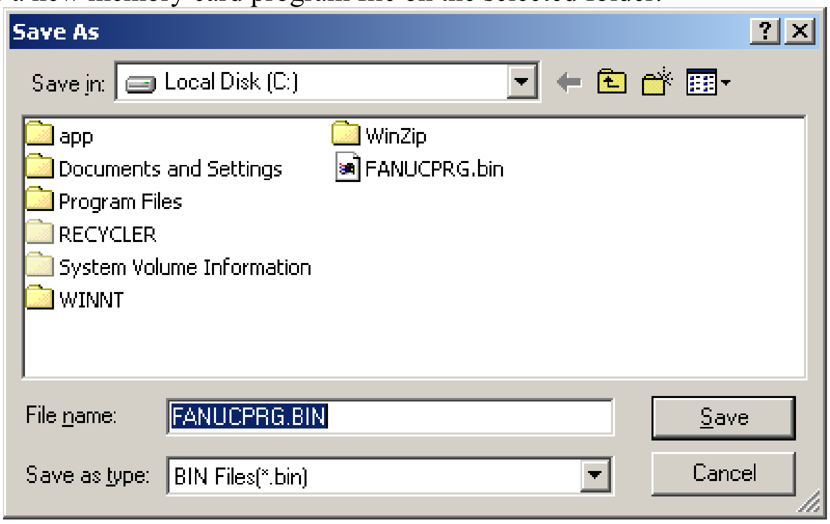


When this PC tool starts up, the Option dialogue window is displayed. Please select "Open an existing file" or "Create a new file".

- **When "Open an existing file" is selected**
  After OK button pushed, "Open" dialogue window is displayed.
  Please select the existing memory card program file.



- **When "Create a new file" is selected**
  After OK button pushed, "Save As" dialogue window is displayed.
  Please create a new memory card program file on the selected folder.



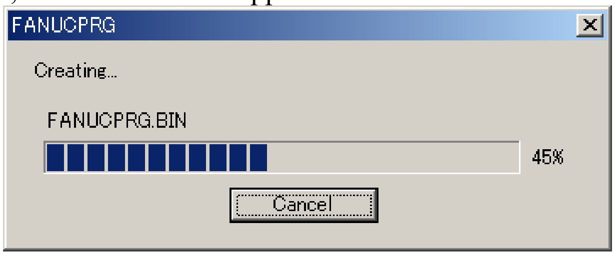
When the new the memory card program file is created, the following items need to be selected:

- Folder/Program Numbers
- Program Size

"Folder/Program Numbers" can be selected among 63 / 500 / 1000. The default value is 63.
To create the file for the 0*i*-D/0*i* Mate-D, select 63.
"Program Size" can be selected among 2Mbyte, 4Mbyte, 8Mbyte, and Customize. The default value is 2Mbyte.

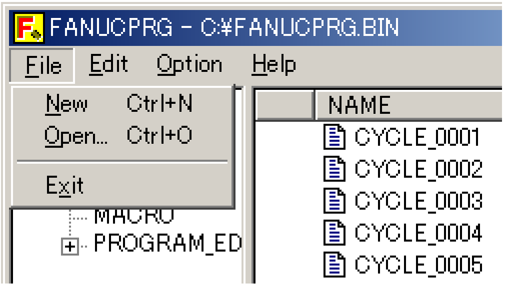
NOTE
1 When "Customize" size is selected, it is available to cover the range from 2Mbyte to 2048Mbyte.
2 Though the maximum size is 2048Mbyte, a slight loss exists by system use.
3 The number of "Folders" in the status bar includes number of program file and folder.

During creating of the memory card program file, the progress bar is being displayed. The progress bar is also displayed during drag-in or drag-out.
If you push [Cancel] button, the execution is stopped.



- Menu

File menu



[New]

Create a new memory card program file.

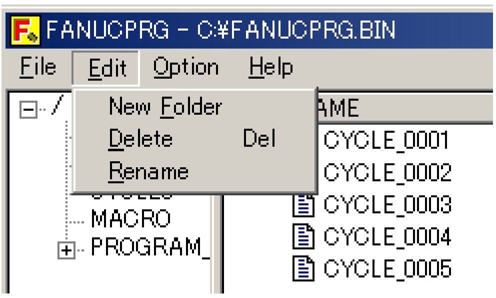
[Open...]

Open the existing memory card program file.

[Exit]

Terminate this PC tool.

Edit menu



[New Folder]

Create new folder. It is available during Tree view selected.
Up to seven hierarchical levels starting from the user root folder (/USER) are permitted.

/USER/PATH1/Aaa/Bbb/Ccc/Ddd/O123
1 2 3 4 5 6 7(not folder)

[Delete]

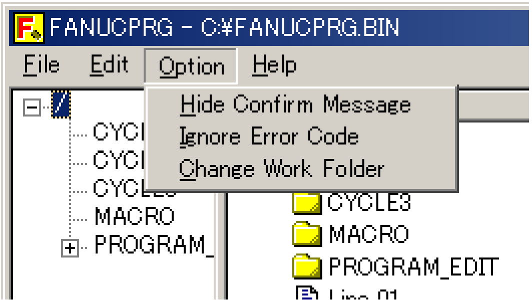
Delete program files or folders.
If you delete a folder, all folders and program files in the folder will be deleted.

[Rename]

Rename a folder or file.

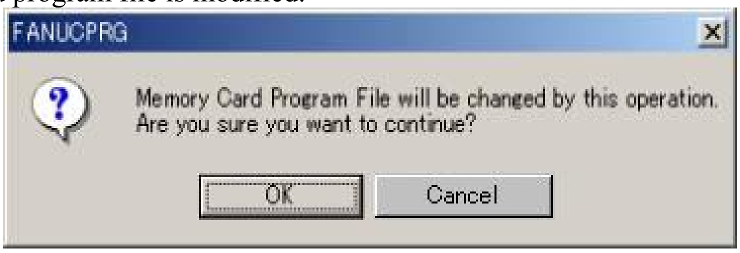
> **NOTE**
> For naming folder and program file, characters which can be used are limited. See "NAMING RULES" below.

Option menu



[Hide Confirm Message]

When the following operations are executed, the following Confirm Message is popped up before the memory card program file is modified.



1. Delete folder or program file
2. Rename folder or program file
3. Drag in program file
4. Add folder

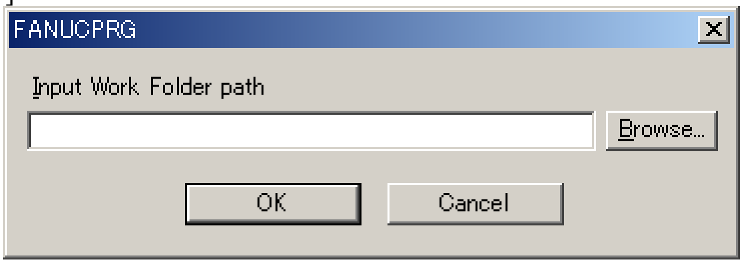
If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

When the [Hide Confirm Message] in menu bar is checked, the Confirm Message is not displayed and the operation advances at once. The default setting is the Confirm Message displayed.

[Ignore Error Code]

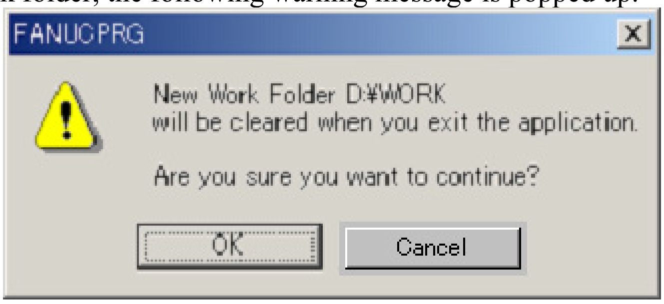
When you check [Ignore Error Code] in the menu bar and perform drag-in, unusable characters are skipped, without causing an error. The unusable characters in the program file will be ignored and will not be written into the memory card program file.

[Change Work Folder]



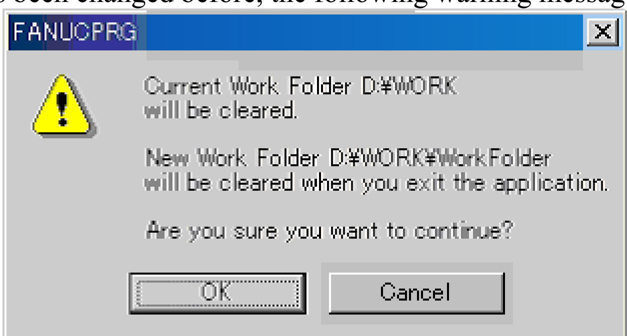
When you select this option, you can set a work folder. By default, the [temp] folder is created in the folder that contains the executable file (FANUCPRG.exe), and [temp] is used as the work folder.
During drag-out, a program file is moved from the memory card program file to the specified destination via the work folder. If there is not enough free space on the hard disk that contains the work folder, drag-out cannot be performed successfully.
To work around this problem, you can specify another work folder.

When you start this PC tool, and then try to specify a new work folder for the first time, the work folder provided by the system has been set. The text box in the dialogue is therefore blank. When you change the work folder, the following warning message is popped up.



If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

If the work folder has been changed before, the following warning message is popped up.

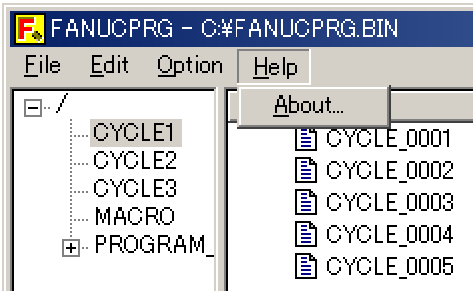


If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

⚠ **WARNING**

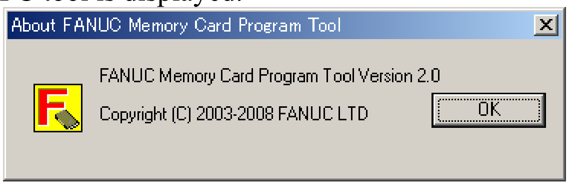
1 At this PC tool terminated, all files in the work folder are deleted.
2 During this PC tool executing, do not access the files in the work folder. If the access is done, there is no assurance of normal operation.

Help menu



[About...]

Version number of this PC tool is displayed.



- Mouse Operation

[Drop-in and Drop-out]

- Drag-in from the Explorer
  You can add NC programs by dragging in files.

  The NC program names and update time are generated based on the files that have been dragged in.

  If "Oxxxxxxxx" or "<xxxx>" exists at the top of files, These "Oxxxxxxxx" and "<xxxx>" become NC program name. If not exist, each file name becomes NC program name.

Relationships between external files and internal files after drag-in			
Name of external file	**Top of external file**	**Name of internal file**	**Program number**
O1234	N10G00	O1234	1234
O123N10G00	N10G00	O123N10G00	Not an O number program
test.txt	O1234N10G00	O1234	1234
test.txt	<O1234>	O1234	1234
test.txt	<O1234N10>	O1234N10	Not an O number program
O1234	<O1234N10>	O1234N10	Not an O number program
O001234	N10G00	O1234	1234
O001234N10G00	N10G00	O001234N10G00	Not an O number program
test.txt	O001234	O1234	1234
test.txt	<O001234>	O1234	1234
test.txt	<O001234N10G00>	O001234N10G00	Not an O number program

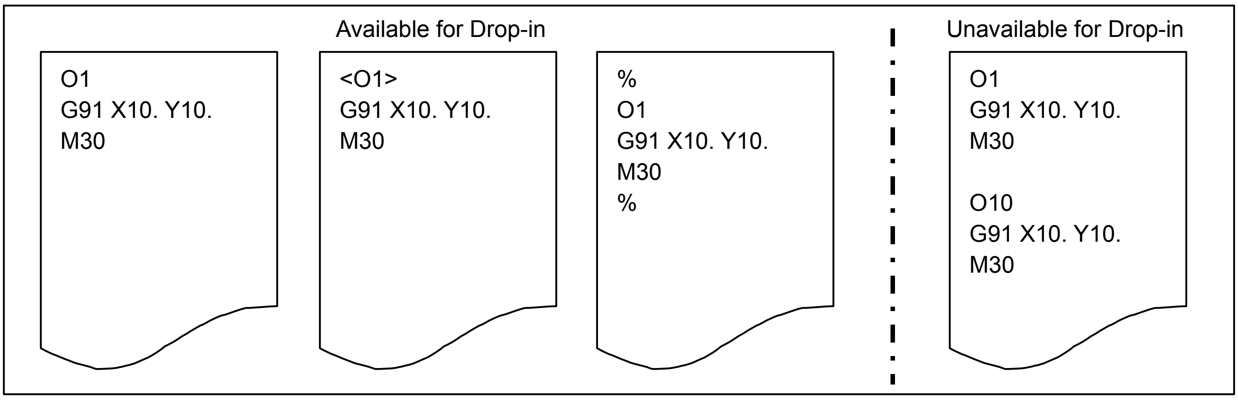
NOTE

1. For an explanation of program file names, see "Naming Rules of Program File " below.
2. For an explanation of characters that can be used in program files, see "RULES OF CHARACTERS IN PROGRAM FILE" below.
3. The updating time of program file is available from 1997 to 2037.
4. The program numbers allowed in the 0*i*-D/0*i* Mate-D are 1 to 9999. The 0*i*-D/0*i* Mate-D cannot therefore handle a file having a program number that is out of the range of 1 to 9999.

Based on the description in "RULES OF CHARACTERS IN PROGRAM FILE" below, this PC tool checks the characters in a file that has been dragged in. However, this PC tool does not check grammar of NC program.

The program file can only have one NC program. Therefore, the NC program output from CNC cannot be dragged in directly.

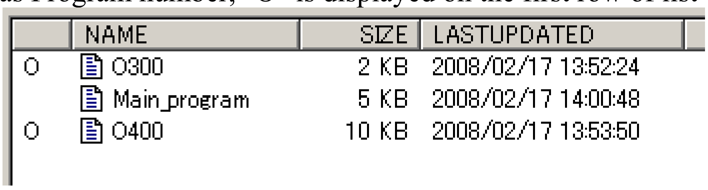
Example of Program



NOTE

1. If another program having the same name already exists, drag-in cannot be performed.
2. If the memory card program file does not have enough free space, drag-in cannot be performed.
3. If an NC program name does not conform to the rules described in "Naming Rules of Program File" below, drag-in cannot be performed.

If the File name acts as Program number, "O" is displayed on the first row of list view.



	NAME	SIZE	LASTUPDATED
O	O300	2 KB	2008/02/17 13:52:24
	Main_program	5 KB	2008/02/17 14:00:48
O	O400	10 KB	2008/02/17 13:53:50

- Drag-out from this PC tool
  You can drag out program files from this PC tool to the Explorer or other folders.

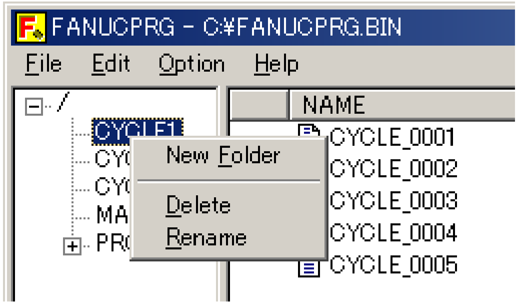
⚠ **CAUTION**

Do not drag out a file from the work folder. If you do this, there is no guarantee of proper operation.

- Pop-up menu

Pop-up menu is displayed by clicking the right mouse button.

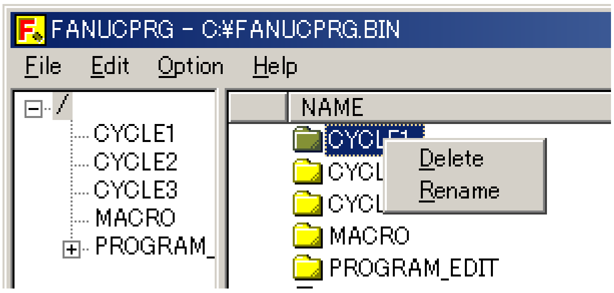
- Focus on Tree view



Clicking "New Folder", a new folder is created on selected folder.
Clicking "Delete", the selected folder is deleted.
Clicking "Rename", the selected folder is renamed.

If clicking on root folder, "Delete" and "Rename" are not activated.

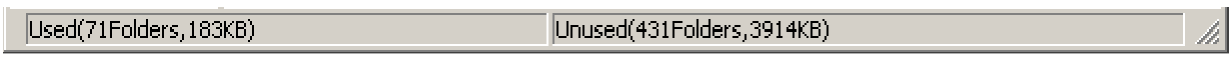
- Focus on List view



Clicking "Delete", the selected folder or program file is deleted.
Clicking "Rename", the selected folder or program file is renamed.

- Display of free space on the memory card program file ("FANUCPRG.BIN")

Number of used folder, size of used space, number of unused folder, and size of free space are displayed on status bar in the lower portion of the screen.

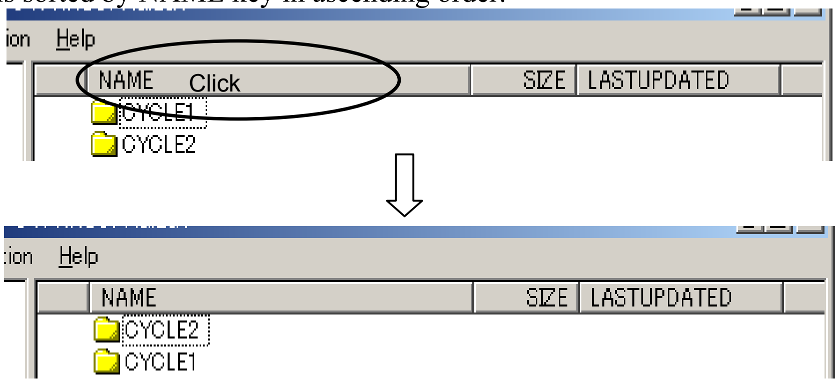


When a new memory card program file is created, two reserved folder are also created. Therefore, the number of used folder is two. However it does not show that number of unused folder is reduced.

The status bar display is updated as you create a new folder, delete a folder, drag in a program file from the Explorer or any other similar tool to this PC tool, or delete a program file.

- Sorting list view of the memory card program file

When a column is being clicked, the list view of the memory card program file is being sorted by the column key in ascending or descending order.
The initial display is sorted by NAME key in ascending order.



I.2 NAMING RULES

Overview

Naming rules of folder and program file are described as follows.

I.2.1 Naming Rules of Program File

Here are Naming rules of Program file:

- Program file name can have a maximum of 32 characters.
- Program file name can have following characters.
  Alphabet(Upper and lower case letter), numeric character,
  "-"(minus), "+"(plus), "_"(under bar), "."(dot)
  "." and ".." can not be used since these are reserved for system use.

- The File name acts as Program number

When a program has a file name consisting of "O" and 1 to 9999, the file name can also be used as the program number to handle the program.
To use a file name including "O" and numeric characters, you must ensure that the following conditions are satisfied.

1. Only numeric characters are used after "O".
2. Up to 4 digits are used after "O".

Example)

File name that can be used as a program number

"O123" Program number 123
"O1" Program number 1
"O3000" Program number 3000
"O0123" Program number 123

File name that does not prevent drag-in but is invalid

"O99999999" Program number 9999999
(The program numbers allowed in the 0i-D/0i Mate-D are 1 to 9999.)

File name that does not prevent drag-in but cannot be used as a program number

"ABC" Top of character is not upper case letter of "O".
"o123" Top of character is not upper case letter of "O".
"O0123XY" A nonnumeric character is included after "O".

File name that prevents drag-in
"O123456789"Numeric characters exceed 8 digits.

NOTE
1 Program file name cannot be repeated in a Folder.
2 If program file name starts with "O" and the next eight characters are all numeric characters, the "0" (zero) after the "O" will be deleted.

I.2.2 Naming Rules of Folder

Here are Naming rules of Folder:
- Folder name can have a maximum of 32 characters.
- Folder name can have following characters.
  Alphabet(Upper and lower case letter), numeric character,
  "-"(minus), "+"(plus), "_"(under bar), "." (dot)
  "." and ".." can not be used since these are reserved for system use.

NOTE
Folder name cannot be repeated in a Folder.

I.3 RULES OF CHARACTERS IN PROGRAM FILE

Overview

Words in parentheses "( )" in Program file are treated as comments.
The mark of comment start "(" is named "Control-out".
The mark of comment end ")" is named "Control-in".
"Control-out" and "Control-in" must make a pair. The order is 1st - "Control-out" and 2nd -"Control-in".
And nested parentheses is not available.

NOTE
1 When a program file is dragged in, space code (0x20 SPC) , tab code (0x09 HT) , carriage return code(0x0d CR) and percent code(0x25 %) is deleted. If "%" is found in Control-in, characters between "%" and next "LF" (0x0a) is deleted.
2 The front of program number ":" is changed to "O"(O as in Oscar) while the program file dragged in.
3 The program file can only have one NC program.

I.3.1 Usable Characters in Program File

- Usable characters in Control-in

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3f	?	58	X	74	t
23	#	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
28	(	42	B	5b	[	77	w
29	)	43	C	5d	]	78	x
2a	*	44	D	5f	_	79	y
2b	+	45	E	61	a	7a	z

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
2c	,	46	F	62	b		
2d	-	47	G	63	c		
2e	.	48	H	64	d		
2f	/	49	I	65	e		
30	0	4a	J	66	f		
31	1	4b	K	67	g		
32	2	4c	L	68	h		
33	3	4d	M	69	i		
34	4	4e	N	6a	j		
35	5	4f	O	6b	k		
36	6	50	P	6c	l		
37	7	51	Q	6d	m		
38	8	52	R	6e	n		
39	9	53	S	6f	o		
3a	:	54	T	70	p		
3c	<	55	U	71	q		
3d	=	56	V	72	r		
3e	>	57	W	73	s		

NOTE

In the Control-in, "O", ":", and "<" can not be used at top of the line except for the 1st line.

- Usable characters in Control-out(characters in parentheses)

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3c	<	55	U	71	q
20	SPC	3d	=	56	V	72	r
22	“	3e	>	57	W	73	s
23	#	3f	?	58	X	74	t
24	$	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
27	‘	42	B	5b	[	77	w
2a	*	43	C	5d	]	78	x
2b	+	44	D	5f	_	79	y
2c	,	45	E	61	a	7a	z
2d	-	46	F	62	b		
2e	.	47	G	63	c		
2f	/	48	H	64	d		
30	0	49	I	65	e		
31	1	4a	J	66	f		
32	2	4b	K	67	g		
33	3	4c	L	68	h		
34	4	4d	M	69	i		
35	5	4e	N	6a	j		
36	6	4f	O	6b	k		
37	7	50	P	6c	l		
38	8	51	Q	6d	m		
39	9	52	R	6e	n		
3a	:	53	S	6f	o		
3b	;	54	T	70	p		

I.4 ERROR MESSAGE AND NOTE

Error may occur when using this application, hereafter explains the error messages and gives relative instructions.

I.4.1 List of Error Message

When an error occurred, the error message box is displayed as follows.

Message	Remarks
Failed to open the file you specified.	The memory card program file cannot be opened. When you retry and the failure recurs, the file may be corrupt.
Failed to read or write to the specified file.	An attempt to read from or write to the memory card program file has failed.
There is insufficient disk space.	There is not enough disk space for the work folder to create a memory card program file or drag out a program file. Allocate sufficient disk space, and then create a file. See an explanation of [Change Work Folder] in "Explanation of Operations".
File name is not correct.	The name cannot be used as a program file name. See "NAMING RULES".
Input name is already exists.	The entered name is already used. Enter another name.
File name is already exists.	The added program file name already exists. Change the name.
Input name is not correct.	The entered name cannot be used. See "NAMING RULES".
Please input an integer between 2 and 2048	A memory card program file cannot be created with the specified size. Enter a numeric value in the range of 2 to 2048.
An illegal character is included in the specified file.	An invalid character is included in the program file. See "RULES OF CHARACTERS IN PROGRAM FILE".
Last update time of the specified file is unsupported.	The update time of the program file is beyond the range of years (1997 to 2037) that can be supported by this PC tool. Create a new file or change the update time.
The memory card program file you specified cannot be identified.	The format of the memory card program file is not the format supported by this PC tool. The file may be rewritten with some other tool.
There is insufficient free folder.	A folder cannot be added. Delete unnecessary folders and program files.
There is insufficient free program space.	A program file cannot be added. Delete unnecessary program files.
Folder deeper than 7 cannot be created.	An attempt was made to create a folder deeper than the 7th level. Folders cannot be created deeper than the 7th level.
Root folder cannot be deleted.	An attempt was made to delete the root folder.
Root folder cannot be renamed.	An attempt was made to rename the root folder.
Failed to create work folder.	An attempt to create [temp] to be used as the work folder has failed. Drag-out cannot be performed. Check whether the [temp] folder can be created in the folder that contains the executable file (FANUCPRG.exe).
Process has been cancelled.	Processing has been discontinued.
The specified work folder is not found.	The folder specified in the path is not found. Drag-out cannot be performed. Exit this PC tool, and set the work folder again.
Only one instance of this application can be executed.	More than one instance of this PC tool cannot be run.

I.4.2 Note

- Folder and Program Numbers

This PC tool allows you to select the number of folders or programs that can be stored in a memory card program file; you can select 63, 500, or 1000. The number of folders or programs that can be used with the 0*i*-D/0*i* Mate-D is 63, however.
If you select 500 or 1000 as the number of folders or programs when you create a memory card program file, the 0*i*-D/0*i* Mate-D cannot use that memory card program file.

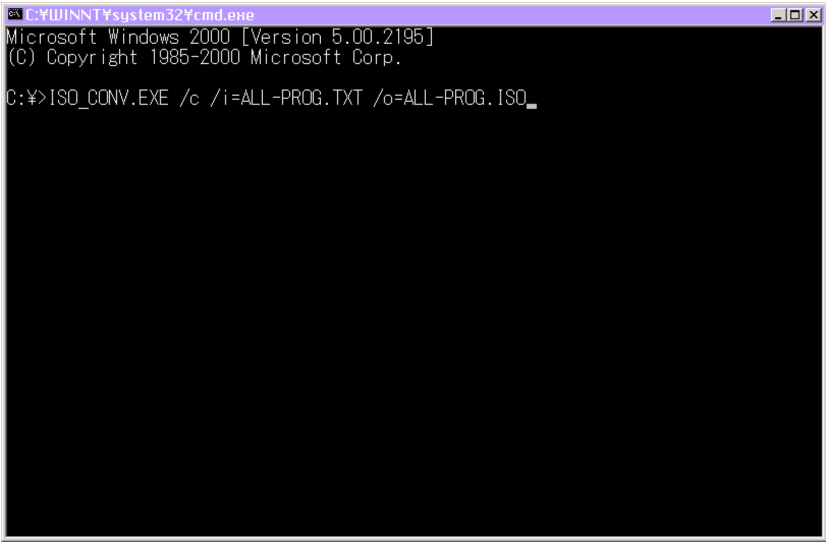
J ISO/ASCII CODE CONVERSION TOOL

Overview

FANUC ISO Converter is a tool that converts a file created or externally output with ASCII code to ISO code format.
This tool runs on Windows 2000, Windows XP, and Windows Vista.
This tool can be used in two modes, CUI and GUI.

CUI

Start the executable file from the command prompt.



Specify the following arguments when starting the tool.

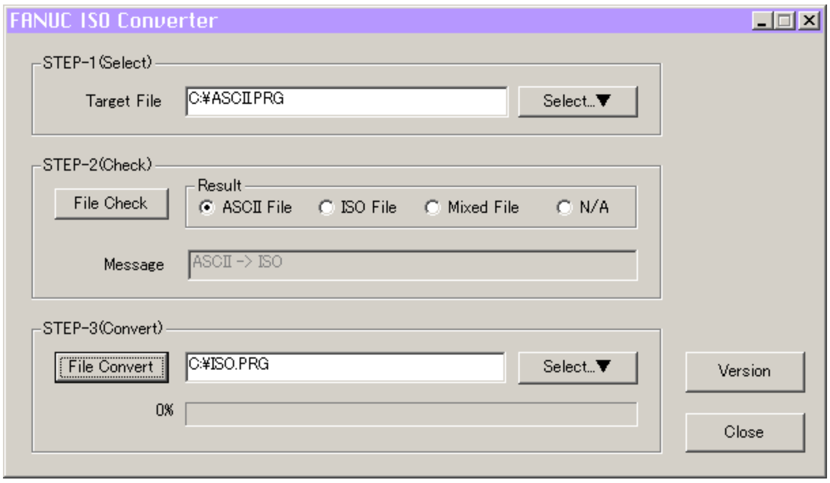
/c : Specify that the tool runs in command line mode.
When /c is omitted, the tool runs in GUI mode.
/i= : Specify the name of an input file to be converted.
/o= :Specify the name of the converted file.

Example
C:¥>ISO_CONV.EXE /c /i=ALL-PROG.TXT /o=ALL-PROG.ISO

Multiple files can be converted at a time by listing them in a batch file.

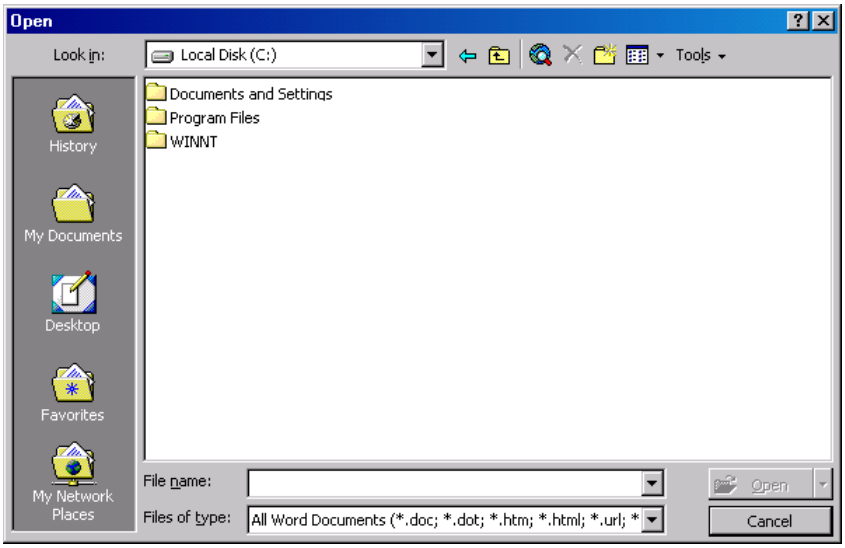
GUI

When you double-click the icon, the following screen appears, allowing you to select and convert a file.



Conversion procedure

1. Step 1
   In [Target File], specify a file you want to convert.
   When you click the [Select...▼] button, a file selection dialogue appears, allowing you to select a file.

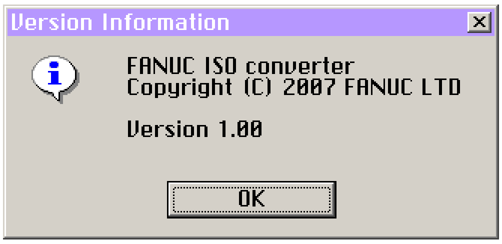


2. Step 2
   After you specify the file name in step 1, you can determine the code of the specified file by clicking the [File Check] button. When you select the file by clicking the [Select...▼] button, the code is automatically determined.
   The file is determined as an ASCII file, ISO file, or ASCII/ISO-mixed file, or is indeterminable. In [Message], an additional message appears depending on the determination result.
   If the file is determined as an ASCII/ISO-mixed file, it cannot be converted, so you cannot proceed to the next step. The [File Convert] button is disabled and you cannot click it. If file contains only characters represented by the code values common to ASCII and ISO, the code of the file is indeterminable. This file need not be converted, so the [File Convert] button is disabled and you cannot click it.

3. Step 3
   When you specify the name of a converted file and click the [File Convert] button, the converted file is created. When the original file is an ASCII file, an ISO file is created; when the original is an ISO file, an ASCII file is created.
   You can also click the [Select...▼] button and specify the name of the converted file from a file selection dialogue.

Other

When you click the [Version] button, the software version information is displayed as shown below.



K DIFFERENCES FROM SERIES 0*i*-C

Appendix K, "Differences from Series 0*i*-C", consists of the following sections:

K.1 SETTING UNIT

K.1.1 Differences in Specifications

Function	Explanation
Diameter/radius specification in the move command for each axis	- Make a selection using bit 3 (DIAx) of parameter No. 1006. **<u>Bit 3 (DIAx) of parameter No. 1006</u>** The move command for each axis specifies: 0: Radius. 1: Diameter. With Series 0*i*-C, in order for an axis whose diameter is specified to travel the specified distance, it is necessary not only to set 1 in bit 3 (DIAx) of parameter No. 1006 but also to make either of the following two changes: - Reduce the command multiplier (CMR) to half. (The detection unit does not need to be changed.) - Reduce the detection unit to half, and double the flexible feed gear (DMR). With Series 0*i*-D, by contrast, just setting 1 in bit 3 (DIAx) of parameter No. 1006 causes the CNC to reduce the command pulses to half, eliminating the need to make the changes described above (when the detection unit is not changed). Note that, when the detection unit is reduced to half, both the CMR and DMR need to be doubled.

K.1.2 Differences in Diagnosis Display

None.

K.2 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES)/AUTOMATIC TOOL OFFSET (T SERIES)

M

K.2.1 Automatic Tool Length Measurement (M Series)

K.2.1.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Operation of the current offset for the measurement result	- Added to the current offset.	- Select whether to add or subtract, by using bit 6 (MDC) of parameter No. 6210. **Bit 6 (MDC) of parameter No. 6210** The measurement result of automatic tool length measurement (system M) or automatic tool compensation (system T) is: 0: Added to the current offset. 1: Subtracted from the current offset.
Setting of the feedrate for measurement	- Set the value in parameter No. 6241. This is a parameter common to the measuring position reached signals (XAE, YAE, and ZAE).	- **Parameter No. 6241** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6242** This is a parameter for the measuring position reached signals (XAE2 and GAE2). - **Parameter No. 6243** This is a parameter for the measuring position reached signals (XAE3 and GAE3). **NOTE** When 0 is set in parameter Nos. 6242 and 6243, the value in parameter No. 6241 becomes valid.
Setting of the γ value	- Set the value in parameter No. 6251. This is a parameter common to the measuring position reached signals (XAE, YAE, and ZAE).	- **Parameter No. 6251** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6252** This is a parameter for the measuring position reached signals (XAE2 and GAE2). - **Parameter No. 6253** This is a parameter for the measuring position reached signals (XAE3 and GAE3). **NOTE** When 0 is set in parameter Nos. 6252 and 6253, the value in parameter No. 6251 becomes valid.

Function	Series 0*i*-C	Series 0*i*-D
Setting of the ε value	- Set the value in parameter No. 6254. This is a parameter common to the measuring position reached signals (XAE, YAE, and ZAE).	- **Parameter No. 6254** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6255** This is a parameter for the measuring position reached signals (XAE2 and GAE2). - **Parameter No. 6256** This is a parameter for the measuring position reached signals (XAE3 and GAE3). **NOTE** When 0 is set in parameter Nos. 6255 and 6256, the value in parameter No. 6254 becomes valid.

K.2.1.2 Differences in Diagnosis Display

None.

T

K.2.2 Automatic Tool Offset (T Series)

K.2.2.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Operation of the current offset for the measurement result	- Added to the current offset.	- Select whether to add or subtract, by using bit 6 (MDC) of parameter No. 6210. **Bit 6 (MDC) of parameter No. 6210** The measurement result of automatic tool length measurement (system M) or automatic tool compensation (system T) is: 0: Added to the current offset. 1: Subtracted from the current offset.
Setting of the feedrate for measurement	- Set the value in parameter No. 6241. This is a parameter common to the measuring position reached signals (XAE and ZAE).	- **Parameter No. 6241** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6242** This is a parameter for the measuring position reached signals (XAE2 and GAE2). **NOTE** When 0 is set in parameter No. 6242, the value in parameter No. 6241 becomes valid.

Function	Series 0*i*-C	Series 0*i*-D
Setting of the γ value for the X axis	- Set the value in parameter No. 6251. This is a parameter common to the measuring position reached signals (XAE and ZAE).	- **Parameter No. 6251** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6252** This is a parameter for the measuring position reached signals (XAE2 and GAE2). **NOTE** When 0 is set in parameter No. 6252, the value in parameter No. 6251 becomes valid.
Setting of the ε value for the X axis	- Set the value in parameter No. 6254. This is a parameter common to the measuring position reached signals (XAE and ZAE).	- **Parameter No. 6254** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6255** This is a parameter for the measuring position reached signals (XAE2 and GAE2). **NOTE** When 0 is set in parameter No. 6255, the value in parameter No. 6254 becomes valid.

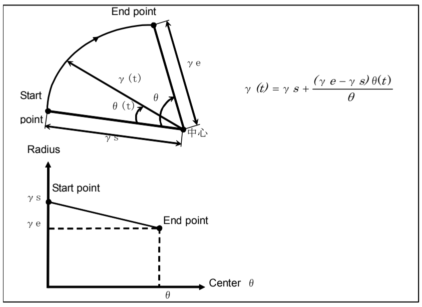
K.2.2.2 Differences in Diagnosis Display

None.

K.3 CIRCULAR INTERPOLATION

K.3.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Interpolation method when the arc end point is not on the arc</td><td colspan="2">If the difference between the radius values of the start point and end point of an arc is greater than the value set in parameter No. 3410, alarm PS0020 is issued. If the difference is smaller (the end point is not on the arc), circular interpolation is performed as follows.</td></tr>
<tr><td>- Circular interpolation is performed using the radius value of the start point and, when an axis reaches the end point, it is moved linearly.

<u>Parameter No. 3410</u>
In a circular interpolation command, set the limit allowed for the difference between the radius values of the start point and end point.</td><td>- Helical interpolation is performed as shown in the figure below.

In other words, the radius of the arc moves linearly according to the center angle θ(t). Specifying an arc where the arc radius of the start point differs from that of the end point enables helical interpolation. When performing helical interpolation, set a large value in parameter No. 3410 that specifies the limit for the arc radius difference.</td></tr>
</table>

K.3.2 Differences in Diagnosis Display

None.

K.4 HELICAL INTERPOLATION

K.4.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Specification of the feedrate	- Specify the feedrate along a circular arc. Therefore, the feedrate of the linear axis is as follows: $F \times \dfrac{\text{Length of linear axis}}{\text{Length of circular arc}}$	- Make a selection using bit 5 (HTG) of parameter No. 1403. 0: Same as left. 1: Specify a feedrate along the tool path including the linear axis. Therefore, the tangential velocity of the arc is expressed as follows: $F \times \dfrac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$ The velocity along the linear axis is expressed as follows: $F \times \dfrac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$ For details, refer to "HELICAL INTERPOLATION" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).
Helical cutting feedrate clamp	- Make a selection using bit 0 (HFC) of parameter No. 1404. 0: The feedrate of the arc and linear axes is clamped by parameter No. 1422 or No.1430. 1: The combined feedrate along the tool path including the linear axis is clamped by parameter No. 1422.	- Bit 0 (HFC) of parameter No. 1404 is not available. The feedrate of the arc and linear axes is clamped by parameter No. 1430.

K.4.2 Differences in Diagnosis Display

None.

K.5 SKIP FUNCTION

K.5.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Setting to enable the high-speed skip signal for normal skip (G31) when the multi-stage skip function is enabled</td><td>- Set 1 in bit 5 (SLS) of parameter No. 6200.</td><td>- Set 1 in bit 4 (HSS) of parameter No. 6200.</td></tr>
<tr><td colspan="2"><table>
<tr><th rowspan="2">Multi-stage skip function</th><th rowspan="2">Command</th><th colspan="2">Parameter to decide on use of the high-speed skip signal</th></tr>
<tr><th>FS0i-C</th><th>FS0i-D</th></tr>
<tr><td>Disabled</td><td>G31 (normal skip)</td><td>HSS</td><td>HSS</td></tr>
<tr><td rowspan="2">Enabled</td><td>G31 (normal skip)</td><td>SLS</td><td>HSS</td></tr>
<tr><td>G31P1 to G31P4 (multi-stage skip)</td><td>SLS</td><td>SLS</td></tr>
</table></td></tr>
<tr><td>Target of acceleration/deceleration and servo system delay compensation</td><td>- Compensation is performed for the skip coordinates obtained when the high-speed skip signal is set to "1".</td><td>- Compensation is performed for the skip coordinates obtained when the skip or high-speed skip signal is set to "1".</td></tr>
<tr><td>Method of acceleration/deceleration and servo system delay compensation</td><td>- There are two ways to perform compensation, as follows.
[Compensating the value calculated from the cutting constant and servo constant]
Set 1 in bit 0 (SEA) of parameter No. 6201.
[Compensating the accumulated pulses and positional deviation due to acceleration/deceleration]
Set 1 in bit 1 (SEB) of parameter No. 6201.</td><td>- Bit 0 (SEA) of parameter No. 6201 is not available.
There is only one way to perform compensation, as follows.
[Compensating the accumulated pulses and positional deviation due to acceleration/deceleration]
Set 1 in bit 1 (SEB) of parameter No. 6201.</td></tr>
<tr><td>Skip cutting feedrate (normal skip)</td><td>- Feedrate specified by the F code in the program</td><td>- Depends on bit 1 (SFP) of parameter No. 6207. When 0 is set, the processing is the same as Series 0i-C.

<u>Bit 1 (SFP) of parameter No. 6207</u>
The feedrate during the skip function (G31) is:
0: Feedrate specified by the F code in the program.
1: Feedrate specified in parameter No. 6281.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Skip cutting feedrate (skip using the high-speed skip signal or multi-step skip)	- Feedrate specified by the F code in the program	- Depends on bit 2 (SFN) of parameter No. 6207. When 0 is set, the processing is the same as Series 0*i*-C. **<u>Bit 2 (SFP) of parameter No. 6207</u>** When the skip function using the high-speed skip signal (1 is set in bit 4 (HSS) of parameter No. 6200) or the multi-step skip function is executed, the feedrate is: 0: Feedrate specified by the F code in the program. 1: Feedrate specified in parameter Nos. 6282 to 6285.
Axis to monitor to check whether the torque limit has been reached (torque limit skip)	- Depends on bit 3 (TSA) of parameter No. 6201. **<u>Bit 3 (TSA) of parameter No. 6201</u>** To check whether the torque limit has been reached, the torque limit skip function (G31 P99/98) monitors: 0: All axes. 1: Only the axis specified in the same block as G31 P99/98.	- Bit 3 (TSA) of parameter No. 6201 is not available. Only the axis specified in the same block as G31 P99/98 is monitored.
High-speed skip signal input for the G31 P99 command (torque limit skip)	As the skip signal for the G31 P99 command, the high-speed skip signal:	
	- Cannot be input.	- Can be input.
Setting of a positional deviation limit in the torque limit skip command (torque limit skip)	- No parameter is available dedicated to setting a positional deviation limit for the torque limit skip function.	- The value can be set in parameter No. 6287. **<u>Parameter No. 6287</u>** Set a positional deviation limit in the torque limit skip command for each axis.
When G31 P99/98 is specified without a torque limit being specified in advance (torque limit skip)	- The G31 P99/98 command is executed as is. (No alarm is issued.)	- Alarm PS0035 is issued.

K.5.2 Differences in Diagnosis Display

None.

K.6 MANUAL REFERENCE POSITION RETURN

K.6.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Conditions for performing manual reference position return during feed hold</td><td colspan="2">Manual reference position return is performed when automatic operation is halted (feed hold) and when any of the following conditions is met:
<Conditions>
(1) Travel distance is remaining.
(2) An auxiliary function (M, S, T, or B function) is being executed.
(3) A dwell, canned cycle, or other cycle is in progress.</td></tr>
<tr><td>- Depends on bit 2 (OZR) of parameter No. 1800.
[When OZR = 0]
Alarm PS0091 occurs, and manual reference position return is not performed.
[When OZR = 1]
Manual reference position return is performed without issuing an alarm.</td><td>- Bit 2 (OZR) of parameter No. 1800 is not available.
Alarm PS0091 occurs, and manual reference position return is not performed.</td></tr>
<tr><td>When inch/metric switch is done</td><td>- The reference position is lost.
(The reference position is not established.)</td><td>- The reference position is not lost.
(The reference position remains established.)</td></tr>
<tr><td>Reference position setting without dogs for all axes</td><td>- Set 1 in bit 1 (DLZ) of parameter No. 1002.</td><td>- Bit 1 (DLZ) of parameter No. 1002 is not available.
Reference position setting without dogs (bit 1 (DLZx) of parameter No. 1005) is set for all axes.</td></tr>
<tr><td>Function that performs reference position setting without dogs two or more times when the reference position is not established in absolute position detection</td><td>- Not available.</td><td>- Depends on bit 4 (GRD) of parameter No. 1007.

<u>Bit 4 (GRD) of parameter No. 1007</u>
For the axis on which absolute values are detected, when correspondence between the machine position and the position by the absolute position detector is not completed, the reference position setting without dogs is:
0: Not performed two or more times.
1: Performed two or more times.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Behavior when manual reference position return is started on a rotation axis with the deceleration dog pressed before a reference position is established	T - [When bit 0 (RTLx) of parameter No. 1007 = 0] Movement is made at the rapid traverse feedrate until the grid is established. If the deceleration dog is released before the grid is established, one revolution is made at the rapid traverse feedrate, thus establishing the grid. Pressing the deceleration dog again establishes the reference position. [When bit 0 (RTLx) of parameter No. 1007 = 1] Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090. M - Does not depend on bit 0 (RTLx) of parameter No. 1007. Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090.	- [Rotation axis type = A and bit 0 (RTLx) of parameter No. 1007 = 0] Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090. [Rotation axis type = A and bit 0 (RTLx) of parameter No. 1007 = 1] Movement is made at the rapid traverse feedrate until the grid is established. If the deceleration dog is released before the grid is established, one revolution is made at the rapid traverse feedrate, thus establishing the grid. Pressing the deceleration dog again establishes the reference position. [Rotation axis type = B] Does not depend on bit 0 (RTLx) of parameter No. 1007. Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090.
Reference position shift function	- Available only for the M series in Series 0*i*-C and earlier.	- Available for all series in Series 0*i*-D.
Reference position shift function setting	- The function is enabled for all axes by setting 1 in bit 2 (SFD) of parameter No. 1002.	- Bit 2 (SFD) of parameter No. 1002 is not available. Set bit 4 (SFDx) of parameter No. 1008 for each axis.
Setting of whether to preset the coordinate system upon high-speed manual reference position return	- Not available. The coordinate system is not preset.	- Depends on bit 1 (HZP) of parameter No. 1206. **Bit 1 (HZP) of parameter No.1206** Upon high-speed manual reference position return, the coordinate system is: 0: Preset. 1: Not preset (FS0*i*-C compatible specification).

M

Function	Series 0*i*-C	Series 0*i*-D
G28/G30 command in the coordinate system rotation, scaling, or programmable mirror image mode	- Not available. Cancel the mode before executing the command.	- The command can be executed only when all of the conditions described below are met. Otherwise, alarm PS0412 occurs. <Conditions> [Conditions required before specifying the command] (1) An absolute command is specified for the target axis of coordinate system rotation, scaling, or programmable mirror image. (2) Tool length compensation has not been performed for the target axis of coordinate system rotation, scaling, or programmable mirror image when it is moved by reference position return. (3) Tool length compensation has been canceled. [Conditions required when specifying the command] (4) In an incremental command, the travel distance of the middle point is 0. [Conditions required after specifying the command] (5) The first move command specified for the target axis of coordinate system rotation, scaling, or programmable mirror image is an absolute command.

K.6.2 Differences in Diagnosis Display

None.

K.7 WORKPIECE COORDINATE SYSTEM

K.7.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Change in absolute position display when the workpiece zero point offset value is changed	- Make a selection using bit 5 (AWK) of parameter No. 1201. **Bit 5 (AWK) of parameter No. 1201** When the workpiece zero point offset value is changed: 0: Changes the absolute position display when the program executes the block that is buffered next. 1: Changes the absolute position display immediately. In either case, the changed value does not take effect until the block that is buffered next.	- Bit 5 (AWK) of parameter No. 1201 is not available. The tool always behaves as when AWK is set to 1.

K.7.2 Differences in Diagnosis Display

None.

K.8 LOCAL COORDINATE SYSTEM

K.8.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Clearing of the local coordinate system after servo alarm cancellation	- The processing is determined by the settings of bit 5 (SNC) and bit 3 (RLC) of parameter No. 1202. **Bit 3 (RLC) of parameter No. 1202** Upon reset, the local coordinate system is: 0: Not canceled. 1: Canceled. **Bit 5 (SNC) of parameter No. 1202** After servo alarm cancellation, the local coordinate system is: 0: Cleared. 1: Not cleared. **NOTE** When the RLC bit of the parameter is set to 1, the local coordinate system is cleared, even if the SNC bit of the parameter is set to 1.	- The processing is determined by the settings of bit 7 (WZR) of parameter No. 1201, bit 3 (RLC) of parameter No. 1202, bit 6 (CLR) of parameter No. 3402, and bit 6 (C14) of parameter No. 3407. Bit 5 (SNC) of parameter No. 1202 is not available. **Bit 7 (WZR) of parameter No. 1201** If the CNC is reset by the reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 0, the G code of group number 14 (workpiece coordinate system) is: 0: Placed in the reset state. 1: Not placed in the reset state. **NOTE** When bit 6 (CLR) of parameter No. 3402 is set to 1, the processing depends on the setting of bit 6 (C14) of parameter No. 3407. **Bit 3 (RLC) of parameter No. 1202** Upon reset, the local coordinate system is: 0: Not canceled. 1: Canceled. **NOTE** - When bit 6 (CLR) of parameter No. 3402 is set to 0 and bit 7 (WZR) of parameter No. 1201 is set to 1, the local coordinate system is canceled, regardless of the setting of this parameter. - When bit 6 (CLR) of parameter No. 3402 is set to 1 and bit 6 (C14) of parameter No. 3407 is set to 0, the local coordinate system is canceled, regardless of the setting of this parameter. **Bit 6 (CLR) of parameter No. 3402** The reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal places the local coordinate system in: 0: Reset state. 1: Clear state. **Bit 6 (C14) of parameter No. 3407** If the CNC is reset by the reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 1, the G code of group number 14 (workpiece coordinate system) is: 0: Placed in the clear state. 1: Not placed in the clear state.

M

Function	Series 0*i*-C	Series 0*i*-D
Operation with the local coordinate system setting (G52)	- Make a selection using bit 4 (G52) of parameter No. 1202. **Bit 4 (G52) of parameter No. 1202** 1) If there are two or more blocks that are not moved before G52 is specified during cutter compensation, or if G52 is specified after the cutter compensation mode is turned off, with the offset vector maintained, the local coordinate system setting is performed: 0: Without considering the cutter compensation vector. 1: Considering the cutter compensation vector. 2) When G52 is specified, the local coordinate system setting is performed for: 0: All axes. 1: Only those axes whose command addresses are found in the G52-specified block.	- Bit 4 (G52) of parameter No. 1202 is not available. The tool always behaves as when G52 is set to 1.

K.8.2 Differences in Diagnosis Display

None.

K.9 Cs CONTOUR CONTROL

K.9.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
In-position check when the Cs contour control mode is off	- The in-position check is not made.	- Make a selection using bit 2 (CSNs) of parameter No. 3729. **Bit 2 (CSNs) of parameter No. 3729** When the Cs contour control mode is off, the in-position check is: 0: Made. 1: Not made. When 1 is set in this parameter, the processing is the same as Series 0*i*-C.

K.9.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Position error display for Cs contour control	For the first spindle, diagnosis display No. 418 is used. For the second spindle, diagnosis display No. 420 is used.	For both the first and second spindles, diagnosis display No. 418 (spindle) is used.

K.10 MULTI-SPINDLE CONTROL

T

K.10.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Number of gear stages for each spindle</td><td>- The first spindle has four stages. Set the maximum spindle speeds for the individual gears in parameter Nos. 3741 to 3744, respectively.
- The second spindle has two stages. Set the maximum spindle speeds for the individual gears in parameter No. 3811 and 3812.</td><td>- Both the first and second spindles each have four stages. Set the maximum spindle speeds for the individual gears in parameter Nos. 3741 to 3744, respectively.
(The data type of parameter Nos. 3741 to 3744 is spindle.)</td></tr>
<tr><td rowspan="2">Spindle override when the override function is used for each axis in multi-spindle control type C</td><td colspan="2">When the override function is used for each axis in multi-spindle control type C, the following spindle override specifications apply during the tapping cycle mode (G84 or G88) or threading mode (G32, G92, or G76).</td></tr>
<tr><td>- No function is available to clamp spindle override to 100%. (It does not depend on bit 6 (TSO) of parameter No. 3708.)
Modify the ladder code as necessary.</td><td>- Depends on bit 6 (TSO) of parameter No. 3708.

<u>Bit 6 (TSO) of parameter No. 3708</u>
During the threading or tapping cycle, spindle override is:
0: Disabled (clamped to 100%).
1: Enabled.</td></tr>
</table>

K.10.2 Differences in Diagnosis Display

None.

K.11 SERIAL/ANALOG SPINDLE CONTROL

K.11.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Spindle number of the analog spindle</td><td colspan="2">- When one serial spindle and one analog spindle are simultaneously controlled in one path (serial/analog spindle control), the spindle number of the analog spindle is as follows.</td></tr>
<tr><td>Third spindle</td><td>Second spindle
For details about the parameters and other settings, refer to "SERIAL/ANALOG SPINDLE CNOTROL" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).</td></tr>
</table>

K.11.2 Differences in Diagnosis Display

None.

K.12 CONSTANT SURFACE SPEED CONTROL

K.12.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Constant surface speed control with no position coder	- This is an optional function for the T series. It is not available with the M series.	- This is a basic function for both M series and T series. It can be used by enabling constant surface speed control (setting 1 in bit 0 (SSC) of parameter No. 8133) and setting 1 in bit 2 (PCL) of parameter No. 1405.
	- Using bit 0 (PSSCL) of parameter No. 1407, select whether to enable or disable the axis feedrate clamp in feed per revolution when the spindle speed is clamped by the maximum spindle speed set in parameter No. 3772. **<u>Bit 0 (PSSCL) of parameter No. 1407</u>** In constant surface speed control with no position coder, when the spindle speed is clamped by the maximum spindle speed parameter, the axis feedrate in feed per revolution is: 0: Not clamped. 1: Clamped. When 1 is set in this parameter, select the spindle to be used for feed per revolution by using the position coder selection signal. (To use the position coder selection signal requires enabling multi-spindle control.)	- Bit 0 (PSSCL) of parameter No. 1407 is not available. The axis feedrate is always clamped. Using the position coder selection signal, select the spindle to be used for feed per revolution. (To use the position coder selection signal requires enabling multi-spindle control.)

K.12.2 Differences in Diagnosis Display

None.

K.13 SPINDLE POSITIONING (T SERIES)

T

K.13.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Display unit of machine coordinates on the spindle positioning axis	- Pulses	- Make a selection using bit 0 (DMD) of parameter No. 4959. **<u>Bit 0 (DMD) of parameter No. 4959</u>** A machine coordinate on the spindle positioning axis is displayed in: 0: Degrees. 1: Pulses.

Function	Series 0*i*-C	Series 0*i*-D
Spindle positioning using the second spindle	- Not available.	- Spindle positioning using the second spindle is possible when multi-spindle control is enabled.
Number of M codes for specifying the spindle positioning angle	- Make a selection using bit 6 (ESI) of parameter No. 4950. **Bit 6 (ESI) of parameter No. 4950** Select the specification of spindle positioning. (Bit) 0: Standard specification. 1: Extended specification. When the extended specification is selected, the number of M codes for specifying the spindle positioning angle can be changed from 6 to any number in the range of 1 to 255, depending on the setting of parameter No. 4964.	- Regardless of the setting of bit 6 (ESI) of parameter No. 4950, the setting of parameter No. 4964 takes effect.
Rapid traverse rate unit for spindle positioning	- Selecting the extended specification by setting 1 in bit 6 (ESI) of parameter No. 4950 extends the upper limit of the rapid traverse rate for spindle positioning from 240000 to 269000 (unit: 10 degrees/min).	- Make a selection using bit 6 (ESI) of parameter No. 4950. **Bit 6 (ESI) of parameter No. 4950** Select the rapid traverse rate unit for spindle positioning (bit spindle). 0: Not increased by a factor of 10. (Unit: degrees/min) 1: Increased by a factor of 10. (Unit: 10 degrees/min)
Rapid traverse rate for spindle orientation in the case of an analog spindle	- The feedrate set in parameter No. 1420 takes effect.	- The feedrate set in parameter No. 1428 takes effect. When 0 is set in parameter No. 1428, the value set in parameter No. 1420 takes effect.

K.13.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Diagnosis data indicating the spindle positioning sequence status (spindle)	- None.	- Diagnosis No.1544
Diagnosis data indicating the clamp/unclamp sequence status (servo)	- None.	- Diagnosis No.5207

K.14 TOOL FUNCTIONS

K.14.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Specification of a G code of the 00 group other than G50 (T series) and a T code in the same block	- Not allowed.	- Not allowed. Specifying a G code in this way causes alarm PS0245.

T

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Number of digits of an offset number in a T code command</td><td>- Set the value in bit 0 (LD1) of parameter No. 5002.</td><td>- Bit 0 (LD1) of parameter No. 5002 is not available.
Use parameter No. 5028.</td></tr>
<tr><td rowspan="2">Method of wear compensation</td><td colspan="2">- When 1 is set in bit 2 (LWT) and bit 4 (LGT) of parameter No. 5002, the method of wear compensation is as follows.</td></tr>
<tr><td>Compensation with tool movement</td><td>Compensation with coordinate shift</td></tr>
<tr><td>Offset cancellation by reset</td><td colspan="2">- Select the cancellation operation using bit 3 (LVC) of parameter No. 5006 and bit 7 (TGC) of parameter No. 5003.
<table>
<tr><td colspan="2" rowspan="2">Compensation method</td><td colspan="4">Parameter</td></tr>
<tr><td>LVC="0"
TGC="0"</td><td>LVC="1"
TGC="0"</td><td>LVC="0"
TGC="1"</td><td>LVC="1"
TGC="1"</td></tr>
<tr><td rowspan="2">Tool movement</td><td>Wear compensation</td><td rowspan="2">×</td><td rowspan="2">○
(When axis is moved)</td><td rowspan="2">×</td><td rowspan="2">○
(When axis is moved)</td></tr>
<tr><td>Geometry compensation</td></tr>
<tr><td rowspan="2">Coordinate shift</td><td>Wear compensation</td><td>×</td><td>○</td><td>×</td><td>○</td></tr>
<tr><td>Geometry compensation</td><td>×</td><td>×</td><td>*</td><td>○</td></tr>
</table>
○: Canceled ×: Not canceled

The operation marked by “*” differs between Series 0i-C and Series 0i-D.
Series 0i-C: × (Not canceled)
Series 0i-D: ○ (Canceled)</td></tr>
</table>

M

Function	Series 0*i*-C	Series 0*i*-D
Behavior when G49 and G40 are specified in the same block	- Make a selection using bit 6 (GCS) of parameter No. 5008. **<u>Bit 6 (GCS) of parameter No. 5008</u>** When G49 (tool length compensation cancellation) and G40 (cutter compensation cancellation) are specified in the same block: 0: Tool length compensation is canceled in the next block. 1: Tool length compensation is canceled in the block in which the command is specified.	- Bit 6 (GCS) of parameter No. 5008 is not available. The tool always behaves as when 1 is set in bit 6 (GCS) of parameter No. 5008.

Function	Series 0*i*-C	Series 0*i*-D
Specification of the tool length compensation amount (Select the compensation amount number with H code.)	- Depends on whether the order of compensation amount numbers specified by the H code is that of tool length compensation types A, B, and C, whether the cutter compensation mode is on or off, and the setting of bit 2 (OFH) of parameter No. 5001. For details, refer to Section 14.1, "TOOL LENGTH COMPENSATION", in "OPERATOR'S MANUAL" (B-64124EN).	- Not dependent on the conditions described at left. In Series 0*i*-D, the H code is used to specify the compensation amount number (select the compensation amount), and G43, G44, and G49 are used to select whether to enable or disable tool length compensation. For details, refer to Section 6.1, "TOOL LENGTH COMPENSATION", in "OPERATOR'S MANUAL (MACHINING CENTER)" (B-64304EN-2).
Restoration of the tool length compensation vector canceled by specifying G53, G28, or G30 during tool length compensation	- The restoration conditions differ depending on the setting of bit 2 (OFH) of parameter No. 5001, as well as on whether the cutter compensation mode is on or off. For details, refer to Section 14.1, "TOOL LENGTH COMPENSATION", in "OPERATOR'S MANUAL" (B-64124EN).	- Not dependent on the setting of bit 2 (OFH) of parameter No. 5001 or the cutter compensation mode. Depends only on the setting of bit 6 (EVO) of parameter No. 5001. **Bit 6 (EVO) of parameter No. 5001** For tool length compensation type A or B, if the tool compensation amount is changed during the offset mode (G43 or G44), the vector is restored in: 0: Subsequent block containing a G43 or G44 command or a H code. 1: Block buffered next.

K.14.2 Differences in Diagnosis Display

None.

K.15 TOOL COMPENSATION MEMORY

K.15.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Unit and range of tool compensation values</td><td>- The unit and range of tool compensation values are determined by the setting unit.</td><td>- Set the unit and range using bit 0 (OFA) and bit 1 (OFC) of parameter No. 5042.

<u>Bit 0 (OFA) and bit 1 (OFC) of parameter No. 5042</u>
Select the setting unit and range of tool offset values.

Metric input
<table>
<tr><th>OFC</th><th>OFA</th><th>Unit</th><th>Range</th></tr>
<tr><td>0</td><td>1</td><td>0.01mm</td><td>±9999.99mm</td></tr>
<tr><td>0</td><td>0</td><td>0.001mm</td><td>±9999.999mm</td></tr>
<tr><td>1</td><td>0</td><td>0.0001mm</td><td>±9999.9999mm</td></tr>
</table>
Inch input
<table>
<tr><th>OFC</th><th>OFA</th><th>Unit</th><th>Range</th></tr>
<tr><td>0</td><td>1</td><td>0.001inch</td><td>±999.999inch</td></tr>
<tr><td>0</td><td>0</td><td>0.0001inch</td><td>±999.9999inch</td></tr>
<tr><td>1</td><td>0</td><td>0.00001inch</td><td>±999.99999inch</td></tr>
</table></td></tr>
<tr><td>Automatic conversion of tool compensation values upon inch/metric switch</td><td>- Make a selection using bit 0 (OIM) of parameter No. 5006.

<u>Bit 0 (OIM) of parameter No. 5006</u>
Upon inch/metric switch, automatic conversion of tool compensation values is:
0: Not performed.
1: Performed.
If the setting of this parameter is changed, set the tool compensation data again.</td><td>- Bit 0 (OIM) of parameter No. 5006 is not available.
Tool compensation values are always converted automatically.</td></tr>
</table>

T

Function	Series 0*i*-TTC	Series 0*i*-D
Number of tool compensation values for each axis during 2-path control	- Up to 64 tool compensation values can be used per path.	- Up to 128 tool compensation values can be used per system. Using parameter No. 5024 whose data type is path, set the number of tool compensation values to be assigned to each path. **NOTE** It is possible to increase to 200 tool compensation values by the option.

Function	Series 0*i*-TTC	Series 0*i*-D
Tool compensation memory sharing during 2-path control	- Set this item using bit 5 (COF) of parameter No. 8100. All tool compensation memories can be shared by the paths. Note that it is not allowed to share only part of the memories. **Bit 5 (COF) of parameter No. 8100** Paths 1 and 2: 0: Do not share tool compensation memories. 1: Share tool compensation memories.	- Set this item using parameter No. 5029. The number of tool compensation memories to be shared can be set arbitrarily.

K.15.2 Differences in Diagnosis Display

None.

K.16 INPUT OF TOOL OFFSET VALUE MEASURED B (T SERIES)

T

K.16.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Setting of the X and Z axes	- It is necessary to set the X axis as the first axis and the Z axis as the second axis.	- It is necessary to set the X axis as the X axis of the basic three axes (set 1 in parameter No. 1022) and the Z axis as the Z axis of the basic three axes (set 3 in parameter No. 1022).
Relationship with arbitrary angular axis control	- By setting 1 in bit 3 (QSA) of parameter No. 5009, the function can be used together with arbitrary angular axis control.	- Cannot be used together with arbitrary angular axis control. The correct value cannot be set for an angular axis under arbitrary angular axis control.
Relationship with composite control	- By setting bit 0 (MXC), bit 1 (XSI), and bit 2 (ZSI) of parameter No. 8160 as appropriate for the machine configuration, the function can be used together with composite control.	- Cannot be used together with composite control. The correct value cannot be set for a composite axis under composite control.

K.16.2 Differences in Diagnosis Display

None.

K.17 CUSTOM MACRO

K.17.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Keep-type common variable
(#500 to #999)</td><td>- The default value is <null>.</td><td>- The default value is 0.</td></tr>
<tr><td>- The Series 0i-D function (described at right) is not available.</td><td>- The range specified by parameter Nos. 6031 and 6032 can be made write-protected (read-only).</td></tr>
<tr><td>System variables to read and write the workpiece coordinate system shift amount
#2501,#2601</td><td>T
- The workpiece coordinate system shift amount of the first axis is read and written by using #2501.
- The workpiece coordinate system shift amount of the second axis is read and written by using #2601.</td><td>T
- The workpiece coordinate system shift amount of the axis of parameter (No.1022)=1(X axis of the basic three axes) is read and written by using #2501.
- The workpiece coordinate system shift amount of the axis of parameter (No.1022)=3(Z axis of the basic three axes) is read and written by using #2601.</td></tr>
<tr><td>System variable to read machine coordinates
#5021 to #5025</td><td>- Machine coordinates are always read in machine units (output units).</td><td>- Machine coordinates are always read in input units.
Example) When the setting unit is IS-B, the input unit is the inch, the machine unit is the millimeter, and the coordinate value of the X axis (first axis) is as follows:
Machine coordinate = 30.000 (mm)
Since the value of #5021 is read in input units (inches), #5021 is 1.1811.</td></tr>
<tr><td>Logical operations in an if statement</td><td>- Logical operations can be used by setting 1 in bit 0 (MLG) of parameter No. 6006.

<u>Bit 0 (MLG) of parameter No. 6006</u>
In an if statement in a custom macro, logical operations:
0: Cannot be used. (P/S alarm No. 114 is issued.)
1: Can be used.</td><td>- Bit 0 (MLG) of parameter No. 6006 is not available.
Logical operations can always be used.</td></tr>
<tr><td rowspan="2">Behavior of the GOTO statement when a sequence number is not found at the start of the block</td><td>- The command after the sequence number of the block (to the right of the sequence number) is executed.</td><td>- If a move command is specified before the sequence number (left side), alarm PS0128 is issued.
If no move command is specified before the sequence number (left side), a block containing a sequence number is executed from the beginning.</td></tr>
<tr><td colspan="2">* Use a sequence number at the start of a block.</td></tr>
<tr><td rowspan="2">Behavior of "GOTO 0" when there is a sequence number</td><td>- The program jumps to the block containing the sequence number.</td><td>- No jump occurs.
Alarm PS1128 is issued.</td></tr>
<tr><td colspan="2">* Do not use a sequence number.</td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>When another NC command is found in a G65 block or in an M code block where a macro is called by an M code
Example) G01 X100. G65 P9001 ;</td><td>- In a program like the one shown in the example, G01 changes the G code group to 01, while the move command X100. is not executed. X100. is regarded as an argument of G65.</td><td>- A program like the one shown in the example cannot be executed. Alarm PS0127 is issued.
A G65 code or an M code that calls a macro must be specified at the beginning of a block (before all other arguments).</td></tr>
<tr><td rowspan="2">Behavior when subprogram call using an M code and subprogram call using an T code are done</td><td colspan="2">- When the machine is run under the conditions and program described below:
[Conditions]
・ Subprogram call by T code is enabled (bit 5 (TCS) of parameter No. 6001 is set to 1).
・ The M code that calls subprogram No. 9001 is M06 (parameter No. 6071 is set to 6).
[Program]
O0001;
T100; (1)
M06 T200; (2)
T300 M06; (3)
M30;
%</td></tr>
<tr><td>In FS0i-C, blocks (1) to (3) of the program causes the machine to behave as follows:
1) Calls and executes O9000.
2) Outputs T200 and waits for FIN. Upon receipt of the FIN signal, the machine calls and executes O9001.
3) Outputs T300 and waits for FIN. Upon receipt of the FIN signal, the machine calls and executes O9001.</td><td>In FS0i-D, blocks (1) to (3) of the program causes the machine to behave as follows:
1) Calls and executes O9000.
2) Issues alarm PS1091.
3) Issues alarm PS1091 (when the program is run with block (2) deleted).</td></tr>
<tr><td rowspan="2">Block containing "M98 Pxxxx" or "M99" without any addresses other than O, N, P, and L</td><td>T
- Using bit 4 (NPS) of parameter No. 3450, it is possible to select whether the block is treated as an NC statement or a macro statement.

<u>Bit 4 (NPS) of parameter No. 3450</u>
0: Treated as a single-block NC statement without movement. (Single block stop is performed.)
1: Treated as a macro statement. (Single block stop is not performed.)

M
- Bit 4 (NPS) of parameter No. 3450 is not available. The block is always treated as a macro statement. (Single block stop is not performed.)</td><td>- Bit 4 (NPS) of parameter No. 3450 is not available. The block is always treated as a macro statement. (Single block stop is not performed.)</td></tr>
<tr><td colspan="2">* For details about macro and NC statements, refer to Section 14.5, "MACRO STATEMENTS AND NC STATEMENTS", in "OPERATOR'S MANUAL" (B-64304EN).</td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Subprogram and macro calls</td><td colspan="2">- The call nesting level differs as follows.
<table>
<tr><td rowspan="2">Model / Call method</td><td colspan="2">Series 0i-C</td><td colspan="2">Series 0i-D</td></tr>
<tr><td>Independent nesting level</td><td>Total</td><td>Independent nesting level</td><td>Total</td></tr>
<tr><td>Macro call (G65/G66)</td><td>4 in all</td><td rowspan="2">(G65/G66/M98) 8 in all</td><td>5 in all</td><td rowspan="2">(G65/G66/M98) 15 in all</td></tr>
<tr><td>Subprogram call (M98)</td><td>4</td><td>10</td></tr>
</table></td></tr>
<tr><td>Local variable clear operation by reset</td><td>- Make a selection using bit 7 (CLV) of parameter No. 6001.

<u>Bit 7 (CLV) of parameter No. 6001</u>
When reset, the local variables in the custom macro are:
0: Cleared to <null>.
1: Not cleared.</td><td>- Bit 7 (CLV) of parameter No. 6001 is not available.
Local variables are always cleared to <null> when reset.</td></tr>
</table>

K.17.2 Differences in Diagnosis Display

None.

K.17.3 Miscellaneous

Series 0*i*-D allows you to customize the specifications related to the maximum and minimum variable values and accuracy by using bit 0 (F0C) of parameter No. 6008. When 1 is set in bit 0 (F0C) of parameter No. 6008, the specifications are the same as Series 0*i*-C. For details, refer to Section II-14, "CUSTOM MACRO", in "OPERATOR'S MANUAL" (B-64304EN).

K.18 INTERRUPTION TYPE CUSTOM MACRO

K.18.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Interruption type custom macro in DNC operation</td><td>- Not available.</td><td>- Available.</td></tr>
<tr><td rowspan="2">Program restart</td><td colspan="2">- When an interruption type custom macro is executed during return operation in dry run after search operation invoked by program restart:</td></tr>
<tr><td>The interruption type custom macro is executed after all axes have restarted.</td><td>Alarm DS0024 is issued.</td></tr>
</table>

K.18.2 Differences in Diagnosis Display

None.

K.19 PROGRAMMABLE PARAMETER INPUT (G10)

K.19.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Parameter input mode setting	- Specify G10 L50.	- Specify G10 L52.

K.19.2 Differences in Diagnosis Display

None.

K.20 ADVANCED PREVIEW CONTROL (T SERIES)/AI ADVANCED PREVIEW CONTROL (M SERIES)/AI CONTOUR CONTROL (M SERIES)

K.20.1 Differences in Specifications

Differences common to advanced preview control, AI advanced preview control, and AI contour control

Function	Series 0*i*-C	Series 0*i*-D
Function name	Some function names have been changed as follows.	
	- Automatic corner deceleration	- Speed control based on the feedrate difference on each axis
	- Arc radius-based feedrate clamp	- Speed control with acceleration in circular interpolation
Setting to enable bell-shaped acceleration/deceleration in rapid traverse	- Setting 1 in bit 6 (RBL) of parameter No. 1603 enables bell-shaped acceleration/deceleration in rapid traverse.	- Bit 6 (RBL) of parameter No. 1603 is not available. Bell-shaped acceleration/deceleration in rapid traverse is enabled by setting the time constant of bell-shaped acceleration/deceleration after interpolation in rapid traverse in parameter No. 1621 or the acceleration change time of bell-shaped acceleration/deceleration before interpolation in rapid traverse in parameter No. 1672.
Selection of acceleration/deceleration before interpolation in rapid traverse or acceleration/deceleration after interpolation in rapid traverse	- The combination of bit 1 (AIR) of parameter No. 7054 and bit 1 (LRP) of parameter No. 1401 determines acceleration/deceleration before interpolation or acceleration/deceleration after interpolation.	- Bit 1 (AIR) of parameter No. 7054 is not available. The combination of bit 5 (FRP) of parameter No. 19501 and bit 1 (LRP) of parameter No. 1401 determines acceleration/deceleration before interpolation or acceleration/deceleration after interpolation. For details, refer to "PARAMETER MANUAL" (B-64310EN).

Function	Series 0*i*-C	Series 0*i*-D
Setting of acceleration for look-ahead linear acceleration/deceleration before interpolation	- Set acceleration by specifying the maximum cutting feedrate for linear acceleration/deceleration before interpolation in parameter No. 1770 and the time to elapse before reaching the maximum cutting feedrate for linear acceleration/deceleration before interpolation in parameter No. 1771.	- Parameter Nos. 1770 and 1771 are not available. In parameter No. 1660, set the maximum permissible cutting feedrate for acceleration/deceleration before interpolation for each axis.
Time constant setting of linear/bell-shaped acceleration/deceleration after interpolation in cutting feed common to all axes	- Set the value in parameter No. 1768.	- Parameter No. 1768 is not available. Set the time constant for each axis in parameter No. 1769.
Time constant setting of exponential acceleration/deceleration after interpolation in cutting feed for each axis	- Set the value in parameter No. 1762. (To set the value for linear or bell-shaped acceleration/deceleration, use parameter No. 1769.)	- Parameter No. 1762 is not available. Set the value in parameter No. 1769. (Use parameter No. 1769 for any acceleration/deceleration type - linear, bell-shaped, or exponential.)
Automatic corner deceleration based on angle difference	- Setting 0 in bit 4 (CSD) of parameter No. 1602 enables the function. Set the lower limit speed in parameter No. 1777 and the critical angle between the two blocks in parameter No. 1779.	- Automatic corner deceleration based on angle difference is not available. Therefore, bit 4 (CSD) of parameter No. 1602 and parameter Nos. 1777 and 1779 are not available.
Permissible speed difference common to all axes for automatic corner deceleration based on angle difference (speed control based on the feedrate difference on each axis)	- Set the value in parameter No. 1780.	- Parameter No. 1780 is not available. Set the permissible speed difference for each axis in parameter No. 1783.
Setting of arc radius-based feedrate clamp (speed control with acceleration in circular interpolation)	- Set the upper limit of the feedrate and the corresponding arc radius value in parameter Nos. 1730 and 1731, respectively.	- Parameter Nos. 1730 and 1731 are not available. Set the permissible acceleration for each axis in parameter No. 1735.
Setting of the maximum cutting feedrate common to all axes	- Set the value in parameter No. 1431.	- Parameter No. 1431 is not available. Set the maximum cutting feedrate for each axis in parameter No. 1432.
Rapid traverse block overlap	- Disabled in the advanced preview control (T series), AI advanced preview control (M series), or AI contour control (M series) mode.	- Enabled only when acceleration/deceleration after interpolation is used in the advanced preview control (T series), AI advanced preview control (M series), or AI contour control (M series) mode.

M

Differences regarding AI advanced preview control and AI contour control

Function	Series 0*i*-C	Series 0*i*-D
Function name	Some function names have been changed as follows.	
	- Acceleration-based feedrate clamp	- Speed control with the acceleration on each axis

Function	Series 0*i*-C	Series 0*i*-D
Setting of acceleration-based feedrate clamp (speed control with the acceleration on each axis)	- Set the permissible acceleration by specifying the time to elapse before reaching the maximum cutting feedrate in parameter No. 1785. The maximum cutting feedrate set in parameter No. 1432 is used.	- Parameter No. 1785 is not available. Set the permissible acceleration for each axis in parameter No. 1737.

Differences regarding AI contour control

Function	Series 0*i*-C	Series 0*i*-D
Time constant of acceleration/deceleration in rapid traverse in the AI contour control mode	- Set parameter Nos. 1773 and 1774. If these parameters are not set, parameter Nos. 1620 and 1621 are used.	- Parameter Nos. 1773 and 1774 are not available. In the case of acceleration/deceleration before interpolation in rapid traverse, set parameter Nos. 1660 and 1672. In the case of acceleration/deceleration after interpolation in rapid traverse, set parameter Nos. 1620 and 1621.
Setting to enable look-ahead bell-shaped acceleration/deceleration before interpolation	- Setting 1 in bit 7 (BEL) of parameter No. 1603 enables bell-shaped acceleration/deceleration before interpolation.	- Bit 7 (BEL) of parameter No. 1603 is not available. Setting the acceleration change time of bell-shaped acceleration/deceleration before interpolation in parameter No. 1772 enables bell-shaped acceleration/deceleration before interpolation.

K.20.2 Differences in Diagnosis Display

None.

K.21 MACHINING CONDITION SELECTION FUNCTION

K.21.1 Differences in Specifications

Differences common to advanced preview control, AI advanced preview control, and AI contour control

Function	Series 0*i*-C	Series 0*i*-D
Parameters set by "acceleration/deceleration before interpolation" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1770] Maximum cutting feedrate in linear acceleration/deceleration before interpolation [Parameter No. 1771] Time before the maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1770) is reached	- The following parameters are set according to the precision level: [Parameter No. 1660] Maximum permissible cutting feedrate in acceleration/deceleration before interpolation on each axis (Series 0*i*-D does not have parameter Nos. 1770 and 1771.)

Function	Series 0*i*-C	Series 0*i*-D
Parameter 1 set by "permissible acceleration" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1730] Upper limit of the feedrate by arc radius-based feedrate clamp [Parameter No. 1731] Arc radius corresponding to the upper limit of the feedrate by arc radius-based feedrate clamp (parameter No. 1730)	- The following parameters are set according to the precision level: [Parameter No. 1735] Permissible acceleration in speed control with acceleration in circular interpolation (Series 0*i*-D does not have parameter Nos. 1730 and 1731. Also, "arc radius-based feedrate clamp" has been renamed "speed control with acceleration in circular interpolation".)

M

Differences regarding AI advanced preview control and AI contour control

Function	Series 0*i*-C	Series 0*i*-D
Parameter 2 set by "permissible acceleration" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1432] Maximum cutting feedrate [Parameter No. 1785] Time before the maximum cutting feedrate (parameter No. 1432) is reached (Set this to determine the permissible acceleration for acceleration-based feedrate clamp.)	- The following parameters are set according to the precision level: [Parameter No. 1737] Permissible acceleration for speed control with the acceleration on each axis (Series 0*i*-D does not have parameter No. 1785. Also, "acceleration-based feedrate clamp" has been renamed "speed control with the acceleration on each axis".)

K.21.2 Differences in Diagnosis Display

None.

K.22 AXIS SYNCHRONOUS CONTROL

K.22.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Function name	- Quick synchronous control	- Axis synchronous control

Function	Series 0*i*-C	Series 0*i*-D
Setting to perform synchronous operation all the time	- Not available.	- Depends on bit 5 (SCA) of parameter No. 8304 for the slave axis. When 0 is set, the processing is the same as Series 0*i*-C. **<u>Bit 5 (SCA) of parameter No. 8304</u>** In axis synchronous control: 0: Synchronous operation is performed when the axis synchronous control selection signal SYNCx or axis synchronous control manual feed selection signal Synch for the slave axis is set to "1". 1: Synchronous operation is performed all the time. Synchronous operation is performed regardless of the setting of the SYNCx or SYNCJx signal.
Setting to move multiple slave axes in synchronism with the master axis	- Not available.	- Available. This is possible by setting the same master axis number in parameter No. 8311 for the multiple slave axes.
Setting of the same name for the master and slave axes	- The same name cannot be set for the master and slave axes.	- The same name can be set for the master and slave axes. In that case, however, automatic operation cannot be performed in normal operation; only manual operation is allowed. (No alarm is caused even if an attempt to perform automatic operation is made.)
Setting of axes for which to perform simple synchronous control (axis synchronous control)	**T** - The setting method of parameter No. 8311 is different from that used for the M series. See Series 0*i*-C Connection Manual (Function) for details. **M** - The master axis number set in parameter No. 8311 must be smaller than the slave axis number.	- The master axis number set in parameter No. 8311 may or may not be smaller than the slave axis number. - The setting method of parameter No. 8311 for the M series of Series 0*i*-C is always used.

Function	Series 0*i*-C	Series 0*i*-D
Synchronization error check based on positional difference	**T** - Not available. **M** - The servo positional difference between the master and slave axes is monitored, and alarm PS0213 is issued if the difference exceeds the limit value set in parameter No. 8313 when the number of synchronized axis pairs is one or the limit value set in parameter No. 8323 for the master axis when the number of synchronized axis pairs is two. - The data range of parameter No. 8323 is as follows: [Data range] 0 to 32767	- The servo positional difference between the master and slave axes is monitored, and alarm DS0001 is issued if the difference exceeds the limit value set in parameter No. 8323 for the slave axis. At the same time, the signal for indicating a positional difference error alarm for axis synchronous control SYNER<F403.0> is output. Parameter No. 8313 is not available. Regardless of the number of pairs, set the limit value in parameter No. 8323. - The data range of parameter No. 8323 is as follows: [Data range] 0 to 999999999
Synchronization error check based on machine coordinates	**T** - Not available. **M** - The machine coordinates of the master and slave axes are compared and, if the difference is greater than the value set in parameter No. 8314 for the master axis, alarm SV0407 is issued and the motor is stopped immediately. - The data range of parameter No. 8314 is as follows: [Data range] 0 to 32767	- The machine coordinates of the master and slave axes are compared and, if the difference is greater than the value set in parameter No. 8314 for the slave axis, alarm SV0005 is issued and the motor is stopped immediately. - The data range of parameter No. 8314 is as follows: [Data range] 0 or positive 9 digits of the minimum unit of data. (For IS-B, 0.0 to +999999.999)
Setting of synchronization establishment	**T** - Synchronization establishment is not available. **M** - Synchronization establishment is enabled by setting 1 in bit 7 (SOF) of parameter No. 8301 when the number of synchronized axis pairs is one or by setting 1 in bit 7 (SOF) of parameter No. 8303 for the master axis when the number of synchronized axis pairs is two.	- Synchronization establishment is enabled by setting 1 in bit 7 (SOF) of parameter No. 8303 for the slave axis. (Bit 7 (SOF) of parameter No. 8301 is not available. Regardless of the number of pairs, set 1 in bit 7 (SOF) of parameter No. 8303.)
Timing of synchronization establishment	**T** - Synchronization establishment is not available. **M** - Synchronization establishment is performed when: 1. Power is turned on when the absolute position detector is used. 2. Emergency stop is canceled.	- Synchronization establishment is performed when: 1. Power is turned on when the absolute position detector is used. 2. Manual reference position return operation is performed. 3. The state of servo position control is changed from off to on. (This occurs when emergency stop, servo alarm, servo off, etc. is canceled. Note, however, that synchronization establishment is not performed at the time of axis removal cancellation.)

Function	Series 0*i*-C	Series 0*i*-D
Maximum compensation for synchronization	**T** - Synchronization establishment is not available. **M** - Set the value in parameter No. 8315 when the number of synchronized axis pairs is one or in parameter No. 8325 for the master axis when the number of synchronized axis pairs is two. If the compensation amount exceeds the values set in the relevant parameter, alarm SV0410 occurs. - The data unit and data range of parameter Nos. 8315 and 8325 are as follows: [Data unit] Detection unit [Data range] 0 to 32767	- Set the value in parameter No. 8325 for the slave axis. If the compensation amount exceeds the values set in this parameter, alarm SV0001 occurs. (Parameter No. 8315 is not available. Regardless of the number of pairs, set the value in parameter No. 8325.) - The data unit and data range of parameter No. 8325 are as follows: [Data unit] Machine unit [Data range] 0 or positive 9 digits of the minimum unit of data. (For IS-B, 0.0 to +999999.999)
Automatic setting for grid position matching	**T** - Automatic setting for grid position matching is not available. **M** - Enable automatic setting for grid position matching by setting 1 in bit 0 (ATE) of parameter No. 8302 when the number of synchronized axis pairs is one or in bit 0 (ATE) of parameter No. 8303 when the number of synchronized axis pairs is two. - Start automatic setting for grid position matching by setting 1 in bit 1 (ATS) of parameter No. 8302 when the number of synchronized axis pairs is one or in bit 1 (ATS) of parameter No. 8303 when the number of synchronized axis pairs is two.	- Set 1 in bit 0 (ATE) of parameter No. 8303 for the slave axis to enable automatic setting for grid position matching. (Bit 0 (ATE) of parameter No. 8302 is not available. Regardless of the number of pairs, set the value in bit 0 (ATE) of parameter No. 8303.) - Set 1 in bit 1 (ATS) of parameter No. 8303 for the slave axis to start automatic setting for grid position matching. (Bit 1 (ATS) of parameter No. 8302 is not available. Regardless of the number of pairs, set the value in bit 1 (ATS) of parameter No. 8303.)
Difference between the master axis reference counter and slave axis reference counter obtained through automatic setting for grid positioning	**T** - Automatic setting for grid position matching is not available. **M** - Set the value in parameter No. 8316 when the number of synchronized axis pairs is one or in parameter No. 8326 for the master axis.	- Set the value in parameter No. 8326 for the slave axis. (Parameter No. 8316 is not available. Regardless of the number of pairs, set the value in parameter No. 8326.)
Time from the servo preparation completion signal SA <F000.6> being set to 1 until torque difference alarm detection is started	**T** - Torque difference alarm detection is not available. **M** - Set the value in parameter No. 8317 when the number of synchronized axis pairs is one or in parameter No. 8327 for the master axis when the number of synchronized axis pairs is two.	- Set the value in parameter No. 8327 for the slave axis. (Parameter No. 8317 is not available. Regardless of the number of pairs, set the value in parameter No. 8327.)

Function	Series 0*i*-C	Series 0*i*-D
Setting to use the external machine coordinate system shift function for the slave axis	**T** - Not available. **M** - When 1 is set in bit 3 (SSE) of parameter No. 8302, setting an external machine coordinate system shift for the master axis causes the slave axis to shift as well. This parameter is used for all the pairs.	- Bit 3 (SSE) of parameter No. 8302 is not available. By setting 1 in bit 7 (SYE) of parameter No. 8304 for the slave axis, the slave axis is shifted as well when an external machine coordinate system shift is set for the corresponding master axis. This parameter is used individually for each slave axis.
Setting to prevent slave axis movement from being added to the actual feedrate display	**T** - Not available. Slave axis movement is always added to the actual feedrate display. **M** - Setting 1 in bit 7 (SMF) of parameter No. 3105 prevents slave axis movement from being added to the actual feedrate display. This parameter is used for all the pairs.	- Bit 7 (SMF) of parameter No. 3105 is not available. Setting 0 in bit 2 (SAF) of parameter No. 8303 prevents slave axis movement from being added to the actual feedrate display. (Note that the meaning of the value is the opposite from bit 7 (SMF) of parameter No. 3105.) This parameter is used individually for each slave axis.
Change of the synchronization state during a program command	- Specify an M code that is not to be buffered. Using this M code, change the input signal - SYNCx<G138> or SYNCJx<G140> - from the PMC side.	- Specify an M code that changes the synchronization state (parameter No. 8337 or 8338). By changing the input signal - SYNCx<G138> or SYNCJx<G140> - from the PMC side using this M code, it is possible to change the synchronization state during a program command. **Parameter No. 8337** Specify an M code that changes synchronous operation to normal operation. **Parameter No. 8338** Specify an M code that changes normal operation to synchronous operation.
Automatic slave axis parameter setting	**T** - This function is enabled by setting 1 in bit 4 (TRP) of parameter No. 12762 for the master axis. **M** - This function is enabled by setting 1 in bit 4 (SYP) of parameter No. 8303 for the master axis.	- Bit 4 (TRP) of parameter No. 12762 is not available. This function is enabled by setting 1 in bit 4 (SYP) of parameter No. 8303 for the master and slave axes.

T

Function	Series 0*i*-C	Series 0*i*-D
Number of pairs for synchronous operation	- One pair (two pairs for the M series)	- Two pairs (also two pairs for the M series)
Synchronous operation during manual operation	- Synchronous operation is not available in jog, handle, or manual incremental feed.	- Setting axis synchronous control manual feed selection signal SYNCJx to 1 enables synchronous operation even in jog, handle, or manual incremental feed.

M

Function	Series 0*i*-C	Series 0*i*-D
Mirror image for the slave axis	- A mirror image cannot be applied to a slave axis during simple synchronous control. It can be applied only to the T series.	- By setting parameter No. 8312 for the slave axis, a mirror image can be applied to a slave axis during simple synchronous control. **<u>Parameter No. 8312</u>** This parameter sets mirror image for the slave axis. When 100 or a more value is set with this parameter, the mirror image function is applied to synchronous control.
Setting to cancel the check of positional difference between the master and slave axes during synchronization establishment	- Depends on bit 5 (SYE) of parameter No. 8301. **<u>Bit 5 (SYE) of parameter No. 8301</u>** During synchronization establishment, the positional difference limit is: 0: Checked. 1: Not checked.	- Not available. Therefore, bit 5 (SYE) of parameter No. 8301 is not available. Since the positional difference is always checked, parameter No. 8318 is not available, either. **<u>Parameter No. 8318</u>** Set the time from the synchronization establishment function outputting a compensation pulse to the slave axis until the check of the positional difference limit between the master and slave axes starts.

K.22.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Positional difference between the master and slave axes	- This item is displayed in diagnosis No. 540 for the master axis when the number of synchronized axis pairs is one or in diagnosis No. 541 for the master axis when the number of synchronized axis pairs is two.	- This item is displayed in diagnosis No. 3500 for the slave axis. (Regardless of the number of pairs, the item is displayed in diagnosis No. 3500.)

K.23 ARBITRARY ANGULAR AXIS CONTROL

K.23.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Angular and perpendicular axes when an invalid value is set in parameter No. 8211 or 8212</td><td colspan="2">
<table>
<tr><th rowspan="2"></th><th colspan="2">Series 0i-C</th><th colspan="2">Series 0i-D</th></tr>
<tr><th>Angular axis</th><th>Perpendicular axis</th><th>Angular axis</th><th>Perpendicular axis</th></tr>
<tr><td>M series</td><td>Y axis (2nd axis)</td><td>Z axis (3rd axis)</td><td>Y-axis of the basic three axes (axis with 2 set in parameter No. 1022)</td><td>Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)</td></tr>
<tr><td>T series</td><td>X axis (1st axis)</td><td>Z axis (2nd axis)</td><td>X-axis of the basic three axes (axis with 1 set in parameter No. 1022)</td><td>Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)</td></tr>
</table>
</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Reference position return completion signal ZP for the perpendicular axis moved with the angular axis <Fn094, Fn096, Fn098, Fn100>	- Select the signal using bit 3 (AZP) of parameter No. 8200. When the bit is set to 0, ZP is not set to "0". (The signal is not cleared.) When the bit is set to 1, ZP is set to "0". (The signal is cleared.)	- Bit 3 (AZP) of parameter No. 8200 is not available. ZP is always set to "0". (The signal is cleared.)
When an angular axis is specified individually in machine coordinate system selection (G53) during arbitrary angular axis control	- Select the perpendicular axis operation using bit 6 (A53) of parameter No. 8201. When the bit is set to 0, the perpendicular axis is also moved. When the bit is set to 1, only the angular axis is moved.	- Bit 6 (A53) of parameter No. 8201 is not available. Only the angular axis is always moved.
G30 command during arbitrary angular axis control	- Select the operation using bit 0 (A30) of parameter No. 8202. When the bit is set to 0, the operation is for the perpendicular coordinate system. When the bit is set to 1, the operation is for the angular coordinate system.	- Bit 0 (A30) of parameter No. 8202 is not available. The operation is always for the angular coordinate system.

K.23.2 Differences in Diagnosis Display

None.

K.24 RUN HOUR AND PARTS COUNT DISPLAY

K.24.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Data range of the M code that counts the number of machined parts	**<u>Parameter No. 6710</u>** The data range of the M code that counts the number of machined parts is as follows.	
	- 0 to 255	- 0 to 99999999 (8 digits)
Data range of the number of parts required	**<u>Parameter No. 6713</u>** The data range of the number of parts required is as follows.	
	- 0 to 9999	- 0 to 999999999 (9 digits)
Data range of the number and total number of parts machined	**<u>Parameter No. 6711</u>** Number of parts machined The data range is as follows.	**<u>Parameter No. 6712</u>** Total number of parts machined
	- 0 to 99999999 (8 digits)	- 0 to 999999999 (9 digits)
Data range of the power-on period, time during automatic operation, cutting time, input signal TMRON on time, and one automatic operation time	**<u>Parameter No. 6750</u>** Integrated value of power-on period **<u>Parameter No. 6752</u>** Integrated value of time during automatic operation **<u>Parameter No. 6756</u>** Integrated value of time when input signal TMRON (G053.0) is on The data range is as follows.	**<u>Parameter No. 6754</u>** Integrated value of cutting time **<u>Parameter No. 6758</u>** Integrated value of one automatic operation time
	- 0 to 99999999 (8 digits)	- 0 to 999999999 (9 digits)

K.24.2 Differences in Diagnosis Display

None.

K.25 MANUAL HANDLE FEED

K.25.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Handle pulses exceeding the rapid traverse rate</td><td colspan="2">If manual handle feed exceeding the rapid traverse rate is specified, whether to ignore or accumulate handle pulses exceeding the rapid traverse feedrate can be set as follows.</td></tr>
<tr><td>- Depends on bit 4 (HPF) of parameter No. 7100. The amount of pulses to be accumulated is set in parameter No. 7117.</td><td>- Bit 4 (HPF) of parameter No. 7100 is not available. Whether to ignore or accumulate excess handle pulses is determined by the amount to be accumulated that is set in parameter No. 7117.
[When parameter No. 7117 = 0]
Ignored.
[When parameter No. 7117 > 0]
Accumulated in the CNC without being ignored.</td></tr>
<tr><td>Permissible amount of pulses for manual handle feed</td><td>- The value range of parameter No. 7117 is 0 to 99999999 (8 digits).</td><td>- The value range of parameter No. 7117 is 0 to 999999999 (9 digits).</td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="5">Value range of the magnification parameter for manual handle feed</td><td>- For parameter Nos. 7113, 7131, 7133, and 12350, magnification ranges from 1 to 127.
For parameter Nos. 7114, 7132, 7134, and 12351, magnification ranges from 1 to 1000.</td><td>- For parameter No. 7113, 7114, 7131, 7132, 7133, 7134, 12350, and 12351, magnification ranges from 1 to 2000.</td></tr>
<tr><td colspan="2"><u>Parameter No. 7113</u>
Magnification when manual handle feed amount selection signals MP1 = 0 and MP2 = 1
<u>Parameter No. 7114</u>
Magnification when manual handle feed amount selection signals MP1 = 1 and MP2 = 1
[When bit 5 (MPX) of parameter No. 7100 = 0]
Magnification common to all the generators in the path
[When bit 5 (MPX) of parameter No. 7100 = 1]
Magnification used by the first generator in the path</td></tr>
<tr><td colspan="2"><u>Parameter No. 7131</u>
Magnification when manual handle feed amount selection signals MP21 = 0 and MP22 = 1
<u>Parameter No. 7132</u>
Magnification when manual handle feed amount selection signals MP21 = 1 and MP22 = 1
When bit 5 (MPX) of parameter No. 7100 is set to 1, the magnification used by the second generator in the path applies.</td></tr>
<tr><td colspan="2"><u>Parameter No. 7133</u>
Magnification when manual handle feed amount selection signals MP31 = 0 and MP32 = 1
<u>Parameter No. 7134</u>
Magnification when manual handle feed amount selection signals MP31 = 1 and MP32 = 1
When bit 5 (MPX) of parameter No. 7100 is set to 1, the magnification used by the third generator in the path applies.</td></tr>
<tr><td colspan="2"><u>Parameter No. 12350</u>
Magnification when per-axis manual handle feed amount selection signals MP1 = 0 and MP2 = 1
<u>Parameter No. 12351</u>
Magnification when per-axis manual handle feed amount selection signals MP1 = 1 and MP2 = 1</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Number of manual pulse generators used	- Set the value in parameter No. 7110.	- Parameter No. 7110 is not available. Up to two generators or three generators (an optional function for 0i-TD) can be used without setting the parameter.

K.25.2 Differences in Diagnosis Display

None.

K.26 PMC AXIS CONTROL

K.26.1 Differences in Specifications

Differences common to 1-path control and 2-path control

Function	Series 0*i*-C	Series 0*i*-D
Relationship with synchronous control (synchronous control of synchronous/composite control)	- PMC axis control can be applied for any axis other than a synchronous slave axis.	- PMC axis control cannot be applied for any axis under synchronous control.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Relationship with the feed-forward and advanced preview feed-forward functions</td><td>- Enable or disable the functions by using bit 7 (NAH) of parameter No. 1819, bit 3 (G8C) of parameter No. 8004, and bit 4 (G8R) of parameter No. 8004 in combination.</td><td>- Neither the feed-forward nor advanced preview feed-forward function is available for an axis under PMC axis control.
Bit 3 (G8C) and bit 4 (G8R) of parameter No. 8004 are not available.</td></tr>
<tr><td>Data range of rapid traverse rate for rapid traverse (00h), 1st to 4th reference position return (07h to 0Ah), and machine coordinate system selection (20h)</td><td>- The data range is as follows.
<table>
<tr><th colspan="2"></th><th colspan="2">Valid data range</th><th rowspan="2">Unit of data</th></tr>
<tr><th colspan="2"></th><th>IS-A, IS-B</th><th>IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Millimeter machine</td><td>30 to 15000</td><td>30 to 12000</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>30 to 6000</td><td>30 to 4800</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>30 to 15000</td><td>30 to 12000</td><td>deg/min</td></tr>
</table></td><td>- 1 to 65535
The data unit is as follows.
<table>
<tr><th colspan="2" rowspan="2"></th><th>Data unit</th><th rowspan="2">Unit</th></tr>
<tr><th>IS-A to IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Metric machine</td><td>1</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>0.1</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>1</td><td>deg/min</td></tr>
</table></td></tr>
<tr><td>Data range of total moving distance for rapid traverse (00h), cutting feed - feed per minute (01h), cutting feed - feed per revolution (02h), and skip - feed per minute (03h)</td><td>- The data range is as follows.
<table>
<tr><th>Input increment</th><th>IS-B</th><th>IS-C</th><th>Unit</th></tr>
<tr><td>mm unit input
deg unit input</td><td>±99999.999</td><td>±9999.9999</td><td>mm
deg</td></tr>
<tr><td>inch unit input</td><td>±9999.9999</td><td>±999.99999</td><td>inch</td></tr>
</table></td><td>- The data range is as follows.
<table>
<tr><th>IS-A</th><th>IS-B,IS-C</th></tr>
<tr><td>-99999999 to 99999999 (8 digits)</td><td>-999999999 to 999999999 (9 digits)</td></tr>
</table>
The data unit is the minimum setting unit for the corresponding axis. (See the table below.)
<table>
<tr><th>Setting unit</th><th>Minimum data unit</th></tr>
<tr><td>IS-A</td><td>0.01</td></tr>
<tr><td>IS-B</td><td>0.001</td></tr>
<tr><td>IS-C</td><td>0.0001</td></tr>
</table></td></tr>
<tr><td>Data range of cutting feedrate for rapid traverse (01h) and skip - feed per minute (03h)</td><td>- 1 to 65535
The specified feedrate must be within the range shown in the table below.
<table>
<tr><th colspan="2"></th><th colspan="2">Valid data range</th><th rowspan="2">Unit of data</th></tr>
<tr><th colspan="2"></th><th>IS-B</th><th>IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Millimeter machine</td><td>1 to 100000</td><td>0.1 to 12000.0</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>0.01 to 4000.00</td><td>0.01 to 480.000</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>1 to 100000</td><td>0.1 to 12000.0</td><td>deg/min</td></tr>
</table></td><td>- 1 to 65535</td></tr>
<tr><td>Function to increase the specification unit by a factor of 200 for continuous feed (06h)</td><td>- Not available.</td><td>- By setting 1 in bit 2 (JFM) of parameter No. 8004, it is possible to increase the specification unit by a factor of 200.

<u>Bit 2 (JFM) of parameter No. 8004</u>
Set the specification unit of feedrate data for specifying the continuous feed command for PMC axis control.
<table>
<tr><th>Increment system</th><th>Bit 2 (JFM) of No. 8004</th><th>Millimeter input (mm/min)</th><th>Inch input (inch/min)</th><th>Rotation axis (min^{-1})</th></tr>
<tr><td rowspan="2">IS-B</td><td>0</td><td>1</td><td>0.01</td><td>0.00023</td></tr>
<tr><td>1</td><td>200</td><td>2.00</td><td>0.046</td></tr>
<tr><td rowspan="2">IS-C</td><td>0</td><td>0.1</td><td>0.001</td><td>0.000023</td></tr>
<tr><td>1</td><td>20</td><td>0.200</td><td>0.0046</td></tr>
</table></td></tr>
<tr><td>Maximum feedrate for continuous feed (06h)</td><td>- When an override of 254% is applied
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr>
<tr><th>1 time</th><td>166458 mm/min</td><td>1664.58 inch/min</td><td>16645 mm/min</td><td>166.45 inch/min</td></tr>
<tr><th>10 times</th><td>1664589 mm/min</td><td>16645.89 inch/min</td><td>166458 0mm/min</td><td>1664.58 inch/min</td></tr>
</table>
- When override is canceled
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr>
<tr><th>1 time</th><td>65535 mm/min</td><td>655.35 inch/min</td><td>6553 mm/min</td><td>65.53 inch/min</td></tr>
<tr><th>10 times</th><td>655350 mm/min</td><td>6553.50 inch/min</td><td>65535 mm/min</td><td>655.35 inch/min</td></tr>
</table></td><td>- When an override of 254% is applied
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th></tr>
<tr><th>1 time</th><td>166458</td><td>1664.58</td><td>16645</td><td>166.46</td></tr>
<tr><th>10 times</th><td>999000</td><td>16645.89</td><td>99900</td><td>1664.58</td></tr>
<tr><th>200 times</th><td>999000</td><td>39330.0</td><td>99900</td><td>3933.0</td></tr>
</table>
- When override is canceled
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th></tr>
<tr><th>1 time</th><td>65535</td><td>655.35</td><td>6553</td><td>65.53</td></tr>
<tr><th>10 times</th><td>655350</td><td>6553.5</td><td>65535</td><td>655.35</td></tr>
<tr><th>200 times</th><td>999000</td><td>39330.0</td><td>999000</td><td>3933.0</td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Minimum unit of feedrate for the speed command (10h)</td><td colspan="2">The minimum unit of feedrate is given by the expressions shown below. The value must be specified as an integer. No finer value may be specified.
A calculation is made according to IS-B.
Fmin: Minimum feedrate unit
P: Number of pulses per revolution of a detector for speed feedback</td></tr>
<tr><td>- Fmin = P ÷ 7500 (mm/min)</td><td>- Fmin = P ÷ 1000 (mm/min)</td></tr>
<tr><td rowspan="2">Speed specification in the speed command (10h)</td><td colspan="2">A speed is specified according to the expressions shown below.
A calculation is made according to IS-B.
F: Speed command (integer)
N: Servo motor speed (min^{-1})
P: Number of pulses per revolution of a detector for speed feedback</td></tr>
<tr><td>- F = N × P ÷ 7500 (mm/min)</td><td>- F = N × P ÷ 1000 (mm/min)</td></tr>
<tr><td>Setting range of torque data for torque control (11h)</td><td>- The setting range is as follows.<table><tr><th>Valid data range</th><th>Unit</th></tr><tr><td>-99999999 to +99999999</td><td>0.0000 1 Nm</td></tr></table></td><td>- The setting range is as follows.<table><tr><th>Valid data range</th><th>Unit</th></tr><tr><td>-999999999 to +999999999 (9 digits)</td><td>0.0000 1 Nm</td></tr></table></td></tr>
<tr><td>Note on executing an absolute command from the program for an axis subject to PMC axis control during automatic operation</td><td colspan="2">- [For Series 0i-D]
When you switch to PMC axis control to execute a move command during automatic operation and then switch back to NC axis control to execute an absolute command from the program for the moved axis, that PMC command needs to be executed using a non-buffering M code.

For example, when an absolute command is executed in a N40 block after PMC control is applied to Y axis, as in the example below, PMC axis control needs to be executed in a non-buffering M code (N20 block).

O0001 ;
N10 G94 G90 G01 X20. Y30. F3000 ;
N20 M55 ; → Executes PMC axis control for the Y axis.
N30 X70. ;
N40 Y50. ;
N50 M30 ;

Execute PMC axis control as follows.
1. After the output of the auxiliary function strobe signal MF for M55, start PMC axis control.
2. Upon completion of PMC axis control, input the completion signal FIN for M55.
- [For Series 0i-C]
Control does not need to be executed using a non-buffering M code.</td></tr>
<tr><td>Acceleration/deceleration control for an axis synchronized with external pulses using external pulse synchronization (0Bh, 0Dh to 0Fh)</td><td>- Depends on bit 2 (SUE) of parameter No. 8002.

<u>Bit 2 (SUE) of parameter No. 8002</u>
With the external pulse synchronization command for PMC axis control, the acceleration/deceleration of the axis synchronized with external pulses is:
0: Controlled (exponential acceleration/deceleration).
1: Not controlled.</td><td>- Bit 2 (SUE) of parameter No. 8002 is not available.
The acceleration/deceleration of the axis synchronized with external pulses is controlled (exponential acceleration/deceleration).</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Inch/metric conversion for a linear axis controlled only by PMC axis control	- Depends on bit 0 (PIM) of parameter No. 8003. **Bit 0 (PIM) of parameter No. 8003** When the axis controlled only by PMC axis control (see parameter No. 1010) is a linear axis, inch/metric input: 0: Influences the axis. 1: Does not influence the axis.	- Bit 0 (PIM) of parameter No. 8003 is not available. Parameter No. 1010 is not available, either. For a linear axis controlled only by PMC axis control, set rotation axis type B (set 1 in both bit 1 and bit 0 of parameter No. 1006) to avoid the influence of inch/metric input.
Setting to change all axes to CNC axes or PMC axes	- Depends on bit 1 (PAX) of parameter No. 8003. **Bit 1 (PAX) of parameter No. 8003** When 0 is set as the number of CNC control axes (parameter No. 1010), all axes are changed to: 0: CNC axes. 1: PMC axes.	- Bit 1 (PAX) of parameter No. 8003 is not available. Parameter No. 1010 is not available, either. There is no parameter to change all axes to PMC axes.
If the PMC issues an axis control command for an axis when the tool is waiting for the auxiliary function completion signal after moving that axis according to a move command and an auxiliary function specified from the CNC side	- Depends on bit 0 (CMV) of parameter No. 8004. **Bit 0 (CMV) of parameter No. 8004** If the PMC issues an axis control command for an axis when the tool is waiting for the auxiliary function completion signal after moving that axis according to a move command and an auxiliary function specified from the CNC side: 0: Alarm PS0130 is issued. 1: The axis control command from the PMC side is executed.	- Bit 0 (CMV) of parameter No. 8004 is not available. The axis control command from the PMC side is executed.
If the CNC issues a command for an axis when that axis is being moved by the axis control command from the PMC side	- Depends on bit 1 (NMT) of parameter No. 8004. **Bit 1 (NMT) of parameter No. 8004** If the CNC issues a command for an axis when that axis is being moved by the axis control command from the PMC side: 0: Alarm PS0130 is issued. 1: A command that does not involve moving the axis is executed without an alarm.	- Bit 1 (NMT) of parameter No. 8004 is not available. A command that does not involve moving the axis is executed without an alarm. (If the command involves moving the axis, alarm PS0130 is issued.)
Setting of diameter/radius specification for the amount of travel and feedrate when diameter programming is specified for a PMC-controlled axis	- This item is determined by using bit 7 (NDI) of parameter No. 8004 and bit 1 (CDI) of parameter No. 8005 in combination.	- Bit 7 (NDI) of parameter No. 8004 is not available. The item is determined by bit 1 (CDI) of parameter No. 8005. **Bit 1 (CDI) of parameter No. 8005** In PMC axis control, when diameter programming is specified for a PMC-controlled axis: 0: The amount of travel and feedrate are each specified with a radius. 1: The amount of travel is specified with a diameter while the feedrate is specified with a radius.

Function	Series 0*i*-C	Series 0*i*-D
Individual output of the auxiliary function	- Depends on bit 7 (MFD) of parameter No. 8005. **Bit 7 (MFD) of parameter No. 8005** The individual output of the auxiliary function for PMC axis control function is: 0: Disabled. 1: Enabled.	- Bit 7 (MFD) of parameter No. 8005 is not available. The individual output of the auxiliary function for PMC axis control function is enabled.
Function to exert position control for the speed command (10h)	- Depends on bit 4 (EVP) of parameter No. 8005. **Bit 4 (EVP) of parameter No. 8005** The speed of PMC axis control is specified by: 0: Speed command. 1: Position command.	- Depends on bit 4 (EVP) of parameter No. 8005. Note that, for the EVP=1 setting to take effect, 1 must be set in bit 2 (VCP) of parameter No. 8007. **Bit 2 (VCP) of parameter No. 8007** The speed command in PMC axis control is: 0: FS10/11 type. 1: FS0 type.
In-position check for an axis controlled only by PMC axis control	- Depends on bit 2 (IPA) of parameter No. 8006. **Bit 2 (IPA) of parameter No. 8006** In the case of an axis controlled only by PMC axis control (see parameter No. 1010), in-position check is: 0: Performed when no move command is specified for the PMC axis. 1: Always not performed.	- Bit 2 (IPA) of parameter No. 8006 is not available. Parameter No. 1010 is not available, either. The check is performed when no move command is specified for the PMC axis. Otherwise, the processing is determined by bit 6 (NCI) of parameter No. 8004. **Bit 6 (NCI) of parameter No. 8004** When the PMC-controlled axis is decelerated, in-position check is: 0: Performed. 1: Not performed.
No in-position check signal for a PMC-controlled axis and no in-position check signals for individual axes	- Depends on bit 0 (NIS) of parameter No. 8007. **Bit 0 (NIS) of parameter No. 8007** For in-position check for a PMC axis, the no in-position check signal NOINPS<G023.5> and no in-position check signals for individual axes NOINP1<G359> to NOINP5<G359> are: 0: Disabled. 1: Enabled.	- Bit 0 (NIS) of parameter No. 8007 is not available. The no in-position check signal NOINPS<G023.5> and no in-position check signals for individual axes NOINP1<G359> to NOINP5<G359> are disabled for in-position check for a PMC axis.
Minimum speed for rapid traverse override in PMC axis control	- Set the value in parameter No. 8021.	- Parameter No. 8021 is not available. The minimum speed for rapid traverse override cannot be set.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Operation when instructing in machine coordinate system selection (20h) to the axis to which roll-over is effective</td><td>- Depends on bit 1 (RAB) of parameter No. 1008.

<u>Bit 1 (RAB) of parameter No. 1008</u>
In the absolute commands, the axis rotates in the direction:
0: In which the distance to the target is shorter.
(Specified by the shortest path)
1: Specified by the sign of command value.</td><td>- Depends on bit 1 (RAB) of parameter No. 1008 and bit 4 (R20) of parameter No. 8013.
<table>
<tr><td></td><td></td><td colspan="2">Bit 4 (R20) of parameter No.8013</td></tr>
<tr><td></td><td></td><td>0</td><td>1</td></tr>
<tr><td rowspan="2">Bit 1 (RAB) of parameter No.1008</td><td>0</td><td>Direction of the shortest path</td><td>Direction of the shortest path</td></tr>
<tr><td>1</td><td>Direction of sign of the amount of the movement to be made</td><td>Direction of sign of command value</td></tr>
</table></td></tr>
</table>

T

Differences regarding 2-path control

Function	Series 0*i*-C	Series 0*i*-D
Relationship with composite control	- PMC axis control can also be applied to axes subject to composite control.	- PMC axis control cannot be applied to axes subject to composite control.
Setting when groups A to D in the path 2 is used	- 1 (group A) to 4 (group D) are set in parameter No. 8010 for the path 2.	- 5 (group A for the path 2) to 8 (group D for the path 2) are set in the axis parameter No. 8010 controlled in the path 2. **<u>Parameter No. 8010</u>** Specify the DI/DO group to be used to specify a command for each PMC-controlled axis.

K.26.2 Differences in Diagnosis Display

None.

K.27 EXTERNAL SUBPROGRAM CALL (M198)

K.27.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Address P format when calling a subprogram on the memory card (file number specification/program number specification)</td><td>- Depends on bit 2 (SBP) of parameter No. 3404.

<u>Bit 2 (SBP) of parameter No. 3404</u>
In the external device subprogram call M198, address P is specified using:
0: File number.
1: Program number.</td><td>- To call a subprogram, the program number must always be specified in address P.
When calling a subprogram on the memory card, the processing is not dependent on the setting of bit 2 (SBP) of parameter No. 3404.</td></tr>
<tr><td rowspan="2">Multiple call alarm</td><td colspan="2">If a subprogram called by an external subprogram call specifies a further external subprogram call, the following alarms are issued, respectively:</td></tr>
<tr><td>- Alarm PS0210</td><td>- Alarm PS1080</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
External subprogram call in MDI mode	- Enabled.	- Depends on bit 1 (MDE) of parameter No. 11630. **Bit 1 (MDE) of parameter No. 11630** In MDI mode, an external device subprogram call (M198 command) is: 0: Disabled. (Alarm PS1081 is issued.) 1: Enabled.

K.27.2 Differences in Diagnosis Display

None.

K.28 SEQUENCE NUMBER SEARCH

K.28.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Return from a subprogram to the calling program's block that has a specified sequence number
Sequence number search when (M99 Pxxxxx) is executed</td><td>- The calling program is searched from the beginning, and control is returned to the first block found to have sequence number Nxxxxx.</td><td>- The calling program is searched in a forward direction from the block that called the subprogram, and control is returned to the first block found to have sequence number Nxxxxx.
If the specified sequence number is not found, the calling program is searched from the beginning, and control is returned to the first block found to have sequence number Nxxxxx.</td></tr>
<tr><td colspan="2">Example) Main program
O0001 ;
N100 ; (1)
N100 ; (2)
M98 P9001 ;
N100 ; (3)
N100 ; (4)
M30 ;
Sub program
O9001 ;
M99 P100 ;
- [For Series 0i-C]
Control is returned to block (1).
- [For Series 0i-D]
Control is returned to block (3).

WARNING
Be sure to avoid writing two or more identical sequence numbers in a program. Doing so may cause the search to find unintended blocks.</td></tr>
</table>

K.28.2 Differences in Diagnosis Display

None.

K.29 STORED STROKE CHECK

K.29.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Stored stroke check immediately following powering on</td><td>- This function is always enabled for all axes.</td><td>- It is possible to select whether to enable or disable the function on an axis-by-axis basis using bit 0 (DOT) of parameter No. 1311.

<u>Bit 0 (DOT) of parameter No. 1311</u>
The stored stroke limit check immediately following powering on is:
0: Disabled.
1: Enabled.
NOTE
This function stores machine coordinates using software and therefore imposes a burden on the system. Disable the function for those axes that do not require it. Movements made while the power is off are not reflected on the machine coordinate system immediately after powering on.</td></tr>
<tr><td>- Machine coordinates are set upon powering on.
Absolute and relative coordinates are not set.
(They are set when the absolute position detector is provided.)</td><td>- Machine coordinates are set upon powering on.
Absolute and relative coordinates are set based on these machine coordinates.</td></tr>
<tr><td>Y and J address specification using G22</td><td>T
Not available.
M
Not available.</td><td>- Available for both the T series and M series.</td></tr>
<tr><td>Overtravel alarm</td><td>- Stored stoke check 2 does not support bit 7 (BFA) of parameter No. 1300. Therefore, if an interference alarm occurs, the tool stops after entering the prohibited area.
This makes it necessary to make the prohibited area slightly larger than actually necessary.</td><td>- Stored stoke check 2 also supports bit 7 (BFA) of parameter No. 1300. Setting 1 in BFA allows the tool to stop before entering the prohibited area, thus eliminating the need to make the prohibited area slightly larger than actually necessary.

<u>Bit 7 (BFA) of parameter No. 1300</u>
If a stored stoke check 1, 2, or 3 alarm occurs, if an interference alarm occurs with the inter-path interference check function (T series), or if an alarm occurs with chuck/tail stock barrier (T series), the tool stops:
0: After entering the prohibited area.
1: Before entering the prohibited area.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Operation continuation after automatic alarm cancellation when a soft OT1 alarm is issued during the execution of an absolute command in automatic operation	- When the operation is resumed, the tool moves the remaining travel distance of the block that caused the soft OT. Therefore, the program can be continued if the tool is moved through manual intervention beyond the remaining travel distance.	- When the operation is resumed, the tool moves toward the end point of the block that caused the soft OT, causing another soft OT and making it impossible to continue the program. For details, refer to "STORED STROKE CHECK 1" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).

M

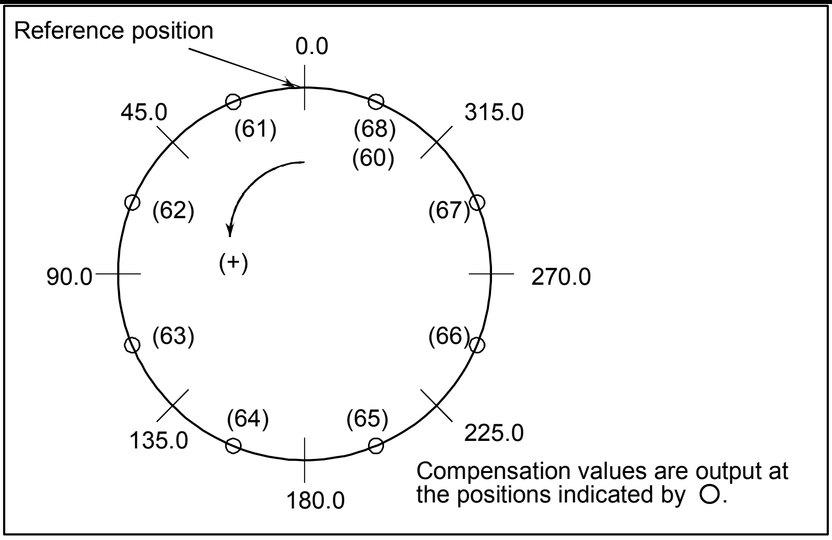
Function	Series 0*i*-C	Series 0*i*-D
Block that judges the distance to the stored stroke limit in AI advanced preview control or AI contour control mode	- A selection can be made using bit 5 (ODA) of parameter No. 7055. **Bit 5 (ODA) of parameter No. 7055** The distance to the stored stroke limit in AI advanced preview control or AI contour control mode is judged with respect to: 0: Axes specified in the current and next blocks. 1: Axes specified in the current block.	- Bit 5 (ODA) of parameter No. 7055 is not available. The distance is always judged with respect to the axes specified in the current block.

K.29.2 Differences in Diagnosis Display

None.

K.30 STORED PITCH ERROR COMPENSATION

K.30.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Explanation</th></tr>
<tr><td>Value of parameter No. 3621 for the setting of a rotary axis (type A)</td><td>


- Amount of movement per rotation: 360°

- Interval between pitch error compensation positions: 45°

- Number of the compensation position of the reference position: 60

In the above case, the values of the parameters are as follows.
<table>
<tr><th>Parameter</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>No. 3620: Number of the compensation position of the reference position</td><td>60</td><td>60</td></tr>
<tr><td>No. 3621: Smallest compensation position number</td><td>60</td><td>61</td></tr>
<tr><td>No. 3622: Largest compensation position number</td><td>68</td><td>68</td></tr>
<tr><td>No. 3623: Compensation magnification</td><td>1</td><td>1</td></tr>
<tr><td>No. 3624: Interval between compensation positions</td><td>45000</td><td>45000</td></tr>
<tr><td>No. 3625: Amount of movement per rotation</td><td>360000</td><td>360000</td></tr>
</table>
The value of parameter No. 3621 is as follows.

Series 0i-C

= Number of the compensation position of the reference position (parameter No. 3620)

Series 0i-D

= Number of the compensation position of the reference position (parameter No. 3620) + 1
</td></tr>
</table>

K.30.2 Differences in Diagnosis Display

None.

K.31 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION

K.31.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Behavior of the manual screen erasure function ("<CAN> + function key") when an alarm is issued	- When an alarm is issued (including one associated with the other path), the manual screen erasure function is enabled. ("<CAN> + function key" erases the screen.)	- When an alarm is issued (including one associated with the other path), the manual screen erasure function is disabled. ("<CAN> + function key" does not erase the screen.)
Redisplay of the screen upon mode switching	- When the operation mode is switched while the screen is erased:	
	The screen is not redisplayed. (The screen remains erased.) Please set "1" to screen clear invalidation signal *CRTOF<G0062.1> to redisplay the screen when operation mode is switched.	The screen is redisplayed.
Function key input when the screen is erased or displayed	- Select the behavior using bit 2 (NFU) of parameter No. 3209. **Bit 2 (NFU) of parameter No. 3209** When a function key is pressed to erase or display the screen for the screen erasure or automatic screen erasure function, the screen change using a function key is: 0: Performed. 1: Not performed.	- Bit 2 (NFU) of parameter No. 3209 is not available. The tool always behaves as when 1 is set in bit 2 (NFU) of parameter No. 3209.
Time before the automatic screen erasure function starts	- Set the value in parameter No.3123.	
	The value range is 1 to 255 (minutes).	The value range is 1 to 127 (minutes).
Redisplay of the screen upon external message	- When the external message is input while the screen is erased:	
	The screen is redisplayed.	The screen is not redisplayed. (The screen remains erased.) Please set "1" to screen clear invalidation signal *CRTOF<G0062.1> to redisplay the screen when external message is input.

K.31.2 Differences in Diagnosis Display

None.

K.32 RESET AND REWIND

K.32.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Modal data when reset during the execution of a block</td><td colspan="2">- If reset occurs during the execution of a block, the states of the modal G codes and modal addresses (N, F, S, T, M, etc.) specified in that block are handled as follows.</td></tr>
<tr><td>Maintained.</td><td>Not maintained. The states return to those of the modal data specified in the preceding blocks.
(The modal data is updated after the specified block is fully executed.)

Example) If reset occurs before positioning is completed in the N2 block in the program shown below, the T code and offset return to the data of the preceding tool (T0101) data.

N1 G00 X120. Z0. T0101 ;
;
N2 G00 X180. Z20. T0202 ;
;</td></tr>
<tr><td>Information in a block that is pre-read when a reset is made during an automatic operation (contents of the buffer)</td><td>- The information in the block may or may not be held depending on whether MDI mode is in progress.
<u>In MDI mode</u>
The information in the block is held.
<u>In modes other than MDI mode</u>
The information in the block is not held.</td><td>- The information in the block is not held regardless whether MDI mode is in progress.</td></tr>
</table>

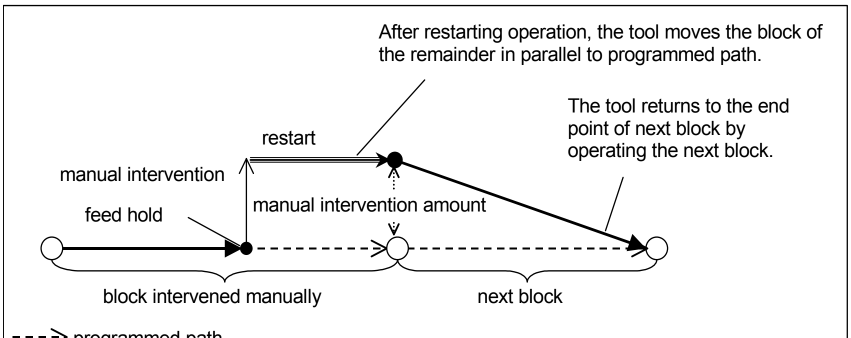
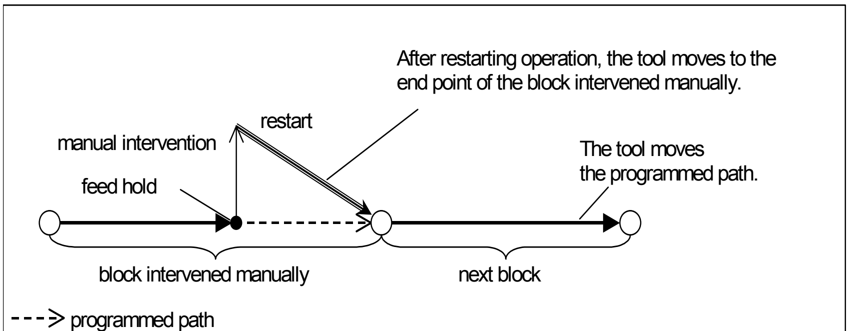
K.32.2 Differences in Diagnosis Display

None.

K.33 MANUAL ABSOLUTE ON AND OFF

K.33.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Absolute coordinates during automatic tool compensation change</td><td colspan="2">- If tool compensation is automatically changed when the manual absolute signal *ABSM(Gn006.2) is set to 1, absolute coordinates are handled as follows.</td></tr>
<tr><td>Absolute coordinates are not changed.</td><td>Absolute coordinates are changed according to the amount of tool compensation resulting from the coordinate shift.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Operation at manual absolute on	- When the block intervened manually ends, the tool is at the position which shifts by manual intervention. (Fig.1) (Even incremental command and absolute command, the result is the same)	- In case of incremental command and bit 1 (ABS) of parameter No. 7001 is set to 0, when the block intervened manually ends, the tool is at the position which shifts by manual intervention. (Fig.1) - In case of absolute command or bit 1 (ABS) of parameter No. 7001 is set to 1, when the block intervened manually ends, the tool is at the programmed position. (Fig.2)
	 Fig. 1	
	 Fig. 2	

K.33.2 Differences in Diagnosis Display

None.

K.34 MEMORY PROTECTION SIGNAL FOR CNC PARAMETER

T

K.34.1 Differences in Specifications

Function	Series 0*i*-TTC	Series 0*i*-D
Memory protection signal for CNC parameter KEYP, KEY1 to KEY4 <G046.0, G046.3 to G046.6>	- The signal is different for each path.	- The signal is common to all paths.

Function	Series 0*i*-TTC	Series 0*i*-D
Parameter to enable the KEYP signal	- Enable or disable the signal using bit 7 (PK5) of parameter No. 3292. This is a bit path parameter.	- Enable or disable the signal using bit 0 (PKY) of parameter No. 3299. This is a bit system common parameter.

K.34.2 Differences in Diagnosis Display

None.

K.35 EXTERNAL DATA INPUT

K.35.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Number of external alarm messages and message length	- [Number of messages that can be set at a time] Up to 4 messages [Length of a message] Up to 32 characters	- [Number of messages that can be set at a time] Depends on bit 1 (M16) of parameter No. 11931. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 1 (M16) of parameter No. 11931** The maximum number of external alarm messages or external operator messages that can be displayed in connection with external data input or external messages is: 0: 4. 1: 16. [Length of a message] Up to 32 characters
Display format of external alarm messages	- [Alarm numbers that can be sent] 0 to 999 [How to distinguish these numbers from general alarm numbers] Add 1000 to the number sent	- Depends on bit 0 (EXA) of parameter No. 6301. **Bit 0 (EXA) of parameter No. 6301** Select the external alarm message specification. 0: The alarm numbers that can be sent range from 0 to 999. The CNC displays an alarm number, with 1000 added to the number following the character string "EX". 1: The alarm numbers that can be sent range from 0 to 4095. The CNC displays an alarm number, with the character string "EX" added in front of it.

Function	Series 0*i*-C	Series 0*i*-D
Number of external operator messages and message length	- Depends on bit 0 (OM4) of parameter No. 3207. **<u>Bit 0 (OM4) of parameter No. 3207</u>** The external operator message screen can display: 0: Up to 256 characters in up to 1 message. 1: Up to 64 characters in up to 4 messages.	- Bit 0 (OM4) of parameter No. 3207 is not available. [Number of messages that can be set at a time] Depends on bit 1 (M16) of parameter No. 11931. Select either up to 4 or 16 messages. [Length of a message] 256 characters or less
Display format of external operator messages	- [Message numbers that can be sent] 0 to 999 [How to distinguish these numbers from alarm and other numbers] **<u>Messages from 0 to 99</u>** The message is displayed on the screen along with the number. The CNC adds 2000 to this number for distinction. **<u>Messages from 100 to 999</u>** Only the message is displayed on the screen without the number.	- Depends on bit 1 (EXM) of parameter No. 6301. When 0 is set, the processing is the same as Series 0*i*-C. **<u>Bit 1 (EXM) of parameter No. 6301</u>** Select the external operator message specification. 0: The message numbers that can be sent range from 0 to 999. A message from 0 to 99 is displayed on the screen along with the number. The CNC adds 2000 to this number for distinction. As for the messages from 100 to 999, only the message is displayed on the screen without the number. 1: The message numbers that can be sent range from 0 to 4095. A message from 0 to 99 is displayed on the screen along with the number. The CNC adds the character string "EX" in front of the number. As for the messages from 100 to 4095, only the message is displayed on the screen without the number.
Data range of external operator message numbers	**<u>Parameter No. 6310</u>** The data range of external operator message numbers is as follows.	
	- 0 to 1000	- 0 to 4096
When an external program number search is done with 0 set as the program number	- An alarm is not issued; the search is not done, either.	- Alarm DS0059 is issued.
Input of an external tool offset for an invalid function compensation value	- The input is ignored without issuing an alarm.	- Alarm DS1121 is issued.

K.35.2 Differences in Diagnosis Display

None.

K.36 DATA SERVER FUNCTION

K.36.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Memory operation mode	- The memory operation mode is not supported.	- In the memory operation mode, the following operations can be performed for a program registered with the data server: 1. Select the program on the data server as the main program and run it in the memory mode. 2. Call a subprogram or custom macro in the same directory as the main program on the data server. 3. Edit the program, including inserting, deleting, and replacing words.

T

Function	Series 0*i*-C	Series 0*i*-D
Simultaneous call from two paths	In a 2-path system, a simultaneous external subprogram call (M198) of a data server program from both paths is:	
	- Allowed under the following conditions. [Storage mode] Both paths must use the same work directory. [FTP mode] Both paths must use the same connection host.	- Not allowed. Use the subprogram/custom macro call for the memory operation mode instead.

K.36.2 Differences in Diagnosis Display

None.

K.37 POWER MATE CNC MANAGER

K.37.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
4-slave display function	- By setting 1 in bit 0 (SLV) of parameter No. 0960, it is possible to split the screen into four windows, enabling up to four slaves to be displayed. **Bit 0 (SLV) of parameter No. 0960** When Power Mate CNC Manager is selected, the screen: 0: Displays one slave. 1: Is split into four windows, enabling up to four slaves to be displayed.	- Bit 0 (SLV) of parameter No. 0960 is not available. One slave is always displayed. When there is more than one slave, you switch the active slave by using the relevant soft key.

K.37.2 Differences in Diagnosis Display

None.

K.38 CHUCK/TAIL STOCK BARRIER (T SERIES)

T

K.38.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Overtravel alarm	- Bit 7 (BFA) of parameter No. 1300 is not supported. Therefore, if an interference alarm occurs, the tool stops after entering the prohibited area. This makes it necessary to make the prohibited area slightly larger than actually necessary.	- Bit 7 (BFA) of parameter No. 1300 is supported. Setting 1 in BFA allows the tool to stop before entering the prohibited area, thus eliminating the need to make the prohibited area slightly larger than actually necessary. **Bit 7 (BFA) of parameter No. 1300** If a stored stoke check 1, 2, or 3 occurs, if an interference alarm occurs with the inter-path interference check function (T series), or if an alarm occurs with chuck/tail stock barrier (T series), the tool stops: 0: After entering the prohibited area. 1: Before entering the prohibited area.

K.38.2 Differences in Diagnosis Display

None.

K.39 THREADING CYCLE RETRACT (CANNED CUTTING CYCLE/MULTIPLE REPETITIVE CANNED CUTTING CYCLE) (T SERIES)

T

K.39.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Return position after chamfering in multiple repetitive threading cycle (G76)	- The tool returns to the start point of the current cycle. For example, if it is the nth cycle, the tool returns to the position where the nth cut has been made.	- The tool returns to the start point of the threading cycle. This means that the tool returns to the position where it was before cutting, no matter how many cycles it has undergone.

Function	Series 0*i*-C	Series 0*i*-D
Retraction after chamfering	- The specifications are as follows. [Acceleration/deceleration type] Acceleration/deceleration after interpolation for threading is used. [Time constant] The time constant for threading (parameter No. 1626) is used. [Feedrate] The feedrate set in parameter No. 1466 is used.	- Depends on bit 0 (CFR) of parameter No. 1611. When 0 is set, the processing is the same as Series 0*i*-C. **<u>Bit 0 (CFR) of parameter No. 1611</u>** In threading cycle G92 or G76, retraction after threading uses: 0: Type of acceleration/deceleration after interpolation for threading, together with the threading time constant (parameter No.1626) and the feedrate set in parameter No.1466. 1: Type of acceleration/deceleration after interpolation for rapid traverse, together with the rapid traverse time constant and the rapid traverse rate.

K.39.2 Differences in Diagnosis Display

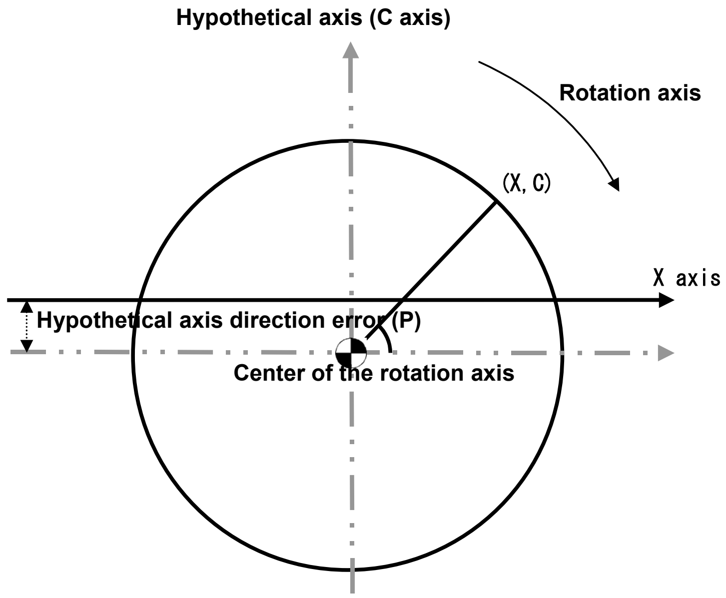
None.

K.40 POLAR COORDINATE INTERPOLATION (T SERIES)

T

K.40.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Coordinate system shift during polar coordinate interpolation (polar coordinate interpolation shift function)	- Not available.	- Enable or disable the function using bit 2 (PLS) of parameter No. 5450. **<u>Bit 2 (PLS) of parameter No. 5450</u>** The polar coordinate interpolation shift function is: 0: Not used. 1: Used. This enables machining using the workpiece coordinate system with a desired point which is not the center of the rotation axis set as the origin of the coordinate system in polar coordinate interpolation. For details, refer to "POLAR COORDINATE INTERPOLATION" in "OPERATOR'S MANUAL (LATHE SYSTEM)" (B-64304EN-1).

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Hypothetical axis direction compensation during polar coordinate interpolation</td><td colspan="2">- If the first axis of the plane is in a hypothetical axis direction relative to the center of the rotation axis, i.e. the center of the rotation axis is not on the X axis, the hypothetical axis direction compensation function in polar coordinate interpolation mode performs polar coordinate interpolation while taking the error into consideration. Set the error value in parameter No. 5464.


(X,C) X-C plane point (The center of the rotation axis is the origin o the X-C plane.)
X X axis coordinate value in the X-C plane
C Hypothetical axis coordinate value in the X-C plane
P Hypothetical axis direction error (Set the value in parameter No. 5464.)</td></tr>
<tr><td>- This function is not available.</td><td>- This function is available.</td></tr>
<tr><td>Maximum cutting feedrate and feedrate clamp during polar coordinate interpolation</td><td>- Set the value in parameter No. 5462. When the value is 0, the feedrate is clamped by parameter No. 1422.</td><td>- Parameter No. 5462 is not available. Set the value in parameter No. 1430.</td></tr>
<tr><td>Automatic override and automatic feedrate clamp during polar coordinate interpolation</td><td>- Enable or disable the function using bit 1 (AFC) of parameter No. 5450.

<u>Bit 1 (AFC) of parameter No. 5450</u>
In the polar coordinate interpolation mode, automatic override and automatic feedrate clamp are:
0: Not performed.
1: Performed.</td><td>- Bit 1 (AFC) of parameter No. 5450 is not available. Automatic override and automatic feedrate clamp are always performed.</td></tr>
</table>

K.40.2 Differences in Diagnosis Display

None.

K.41 PATH INTERFERENCE CHECK (T SERIES (2-PATH CONTROL))

T

K.41.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Interference alarm	- Bit 7 (BFA) of parameter No. 1300 is not supported. Therefore, if an interference alarm occurs, the tool stops after entering the prohibited area. This makes it necessary to make the prohibited area slightly larger than actually necessary.	- Bit 7 (BFA) of parameter No. 1300 is supported. Setting 1 in BFA allows the tool to stop before entering the prohibited area, thus eliminating the need to make the prohibited area slightly larger than actually necessary. **Bit 7 (BFA) of parameter No. 1300** If a stored stoke check 1, 2, or 3 alarm occurs, if an interference alarm occurs with the inter-path interference check function (T series), or if an alarm occurs with chuck/tail stock barrier (T series), the tool stops: 0: After entering the prohibited area. 1: Before entering the prohibited area.

K.41.2 Differences in Diagnosis Display

None.

K.42 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (T SERIES (2-PATH CONTROL))

T

K.42.1 Differences in Specifications

Function	Series 0*i*-TTC	Series 0*i*-D
Axis synchronous control (Series 0*i*-C: Quick synchronous control)	- Adding synchronous or composite control disables simple synchronous control.	- Adding synchronous or composite control does not disable simple synchronous control. - The master and slave axes used for axis synchronous control cannot be used for synchronous control. - Composite control is available for the master axis used for axis synchronous control, while it is not available for the slave axis.

Function	Series 0*i*-TTC	Series 0*i*-D
Feed forward function and cutting/rapid traverse change function for synchronous and composite axes of another path	- Make a selection using bit 1 (SVF) of parameter No. 8165. **Bit 1 (SVF) of parameter No. 8165** In synchronous or composite control, the feed forward function and cutting/rapid traverse change function for synchronous and composite axes of another path are: 0: Disabled. 1: Enabled.	- Bit 1 (SVF) of parameter No. 8165 is not available. The tool always behaves as when SVF is set to 1. (The feed forward function and cutting/rapid traverse change function are enabled for synchronous and composite axes of another path.)
Move command when neither synchronous nor composite control is in effect	- Not prohibited.	- Make a selection using bit 7 (NUMx) of parameter No. 8163. **Bit 7 (NUMx) of parameter No. 8163** When neither synchronous nor composite control is in effect, specifying the move command for an axis that is set with this parameter is: 0: Not prohibited. 1: Prohibited. (Alarm PS0353 is issued.)
Behavior when an alarm is issued in relation to synchronous or composite control	- Both paths are placed in the feed hold state.	- Make a selection using bit 0 (MPA) of parameter No. 8168. **Bit 0 (MPA) of parameter No. 8168** If an alarm is issued in relation to synchronous, composite, or superposition control: 0: Both paths are placed in the feed hold state. 1: Only the path including axes related to synchronous, composite, or superposition control is placed in the feed hold state. For example, when synchronous control is exerted in one path, only the path that caused the alarm is placed in the feed hold state. The handling of the other path depends on the setting of bit 1 (IAL) of parameter No. 8100.
Behavior when overtravel occurs for an axis under synchronous or composite control	- The synchronous or composite control mode is canceled.	- Make a selection using bit 5 (NCS) of parameter No. 8160. **Bit 5 (NCSx) of parameter No. 8160** If overtravel occurs for an axis under synchronous, composite, or superposition control, the synchronous, composite, or superposition control mode is: 0: Canceled. 1: Not canceled.
Switch between synchronous control axis selection signal and composite control axis selection signal during automatic operation	- The signals can be switched at any time.	- Use an M code command. Specify a waiting M code (M code without buffering) before and after the M code. When synchronous or composite control is exerted in one path, specify an M or other code without buffering before and after the M code that starts or cancels the control so as to prohibit the look-ahead operation.

Synchronous control

Item	Series 0*i*-TTC	Series 0*i*-D
G28 when the master axis is parking	- When the reference position of the slave axis is not established, the machine coordinates are moved to the coordinates set in parameter No. 1240, completing the reference position return.	- When the reference position of the slave axis is not established, alarm PS0354 occurs.
Update of the workpiece coordinates and relative coordinates of the slave axis under synchronous control	- Make a selection using bit 4 (SPN) of parameter No. 8164. **Bit 4 (SPN) of parameter No. 8164** The workpiece coordinates and relative coordinates of the slave axis under synchronous control are: 0: Updated. 1: Not updated.	- Bit 4 (SPN) of parameter No. 8164 is not available. The tool always behaves as when SPNx is set to 0 (coordinates are updated).
Out-of-synchronization detection when synchronous control is exerted in one path (1 is set in bit 1 (SER) of parameter No. 8162)	- Out-of-synchronization detection is not performed.	- Out-of-synchronization detection is performed.
Manual handle interruption amount or mirror image mode for the master axis	- Always reflected on the slave axis.	- Select whether to reflect the amount or mode on the slave axis, using bit 5 (SMIx) of parameter No. 8163. **Bit 5 (SMIx) of parameter No. 8163** During synchronous control, the manual handle interruption amount or mirror image mode for the master axis is: 0: Reflected on the slave axis. 1: Not reflected on the slave axis.
Automatic setting of a workpiece coordinate system for the slave axis at the end of synchronous control	- A workpiece coordinate system is not automatically set for the slave axis.	- Make a selection using bit 6 (SPVx) of parameter No. 8167. **Bit 6 (SPVx) of parameter No. 8167** At the end of synchronous control, a workpiece coordinate system for the slave axis is: 0: Not automatically set. 1: Automatically set. The workpiece coordinate system to be set is determined by the machine coordinate values and the workpiece coordinate values of the reference points of the individual axes defined by parameter No. 1250.

Composite control

Item	Series 0*i*-TTC	Series 0*i*-D
G28 during composite control	- When the reference position of the composite axis of the other path is not established, the machine coordinates are moved to the coordinates set in parameter No. 1240, completing the reference position return.	- When the reference position of the composite axis of the other path is not established, alarm PS0359 occurs.

Item	Series 0*i*-TTC	Series 0*i*-D
Composite control for the Cs contour axis reference position return command when composite control is exerted for Cs contour axes	- Select whether to use the composite function of the Cs contour axis reference position return command, by using bit 1 (CZMx) of parameter No. 8161. **Bit 1 (CZMx) of parameter No. 8161** When composite control is exerted for Cs contour axes, the composite control function for the Cs contour axis reference position return command is: 0: Not used. 1: Used.	- Bit 1 (CZMx) of parameter No. 8161 is not available. The tool always behaves as when CZMx is set to 1 (composite control is used).
Manual handle interruption for composite axes	- Disabled.	- Enable or disable the interruption using bit 6 (MMIx) of parameter No. 8163. **Bit 6 (MMIx) of parameter No. 8163** During composite control, manual handle interruption for composite axes is: 0: Enabled. 1: Disabled.
Current position display during composite control (absolute/relative coordinates)	- Make a selection using bit 0 (MDXx) of parameter No. 8163. **Bit 0 (MDXx) of parameter No. 8163.** During composite control, the current position display (absolute/relative coordinates) shows: 0: Coordinate values of the local path. 1: Coordinate values of the mate path.	- Bit 0 (MDXx) of parameter No. 8163 is not available. The coordinate values of the local path are always displayed.
G53 during composite control	- Make a selection using bit 2 (CPMx) of parameter No. 8165. **Bit 2 (CPMx) of parameter No. 8165.** During composite control, machine coordinate system selection (G53) is: 0: Disabled. 1: Enabled. (The travel distance is calculated so that the machine moves according to the machine coordinate system selection signal of the mate path.)	- Bit 2 (CPMx) of parameter No. 8165 is not available. The tool always behaves as when CPMx is set to 1. (G53 is enabled.)
Constant acceleration/deceleration of acceleration time for acceleration/deceleration in rapid traverse for an axis subject to composite control (bit 4 (RPT) of parameter No. 1603)	- Make a selection using bit 0 (NLSx) of parameter No. 8167. **Bit 0 (NLSx) of parameter No. 8167** Constant acceleration/deceleration of acceleration time for acceleration/deceleration in rapid traverse for an axis subject to composite control (bit 4 (RPT) of parameter No. 1603) is: 0: Enabled. 1: Disabled.	- Bit 0 (NLSx) of parameter No. 8167 is not available. The tool always behaves as when NLSx is set to 1. (Constant acceleration/deceleration of acceleration time is enabled.)

Item	Series 0*i*-TTC	Series 0*i*-D
Machine coordinates during composite control	- The coordinate values of the local path are displayed.	- Make a selection using bit 0 (MDMx) of parameter No. 8169. **Bit 0 (MDMx) of parameter No. 8169** The machine coordinates displayed during composite control are: 0: Coordinate values of the local path. 1: Machine coordinate values of the mate path.
Reading of machine coordinates (#5021 and later) during composite control	- The coordinate values of the local path are read.	- Make a selection using bit 1 (MVMx) of parameter No. 8169. **Bit 1 (MVMx) of parameter No. 8169** The machine coordinates (#5021 and later) that are read during composite control are: 0: Machine coordinate values of the local path. 1: Machine coordinate values of the mate path.
Rapid traverse feedrate during composite control	- The rapid traverse feedrate of the specified axis is used.	- Make a selection using bit 2 (MRFx) of parameter No. 8169. **Bit 2 (MRFx) of parameter No. 8169** The rapid traverse feedrate used during composite control is: 0: Rapid traverse feedrate of the specified axis. 1: Rapid traverse feedrate of the moving axis.

K.42.2 Differences in Diagnosis Display

Item	Series 0*i*-TTC	Series 0*i*-D
Synchronization error value display for each axis	- Displayed in parameter No. 8182.	- Displayed in diagnosis No. 3502.

K.43 SUPERIMPOSED CONTROL (T SERIES (2-PATH CONTROL))

T

K.43.1 Differences in Specifications

Function	Series 0*i*-TTC	Series 0*i*-D
Axis synchronous control (Series 0*i*: Quick synchronous control)	- Adding superimposed control disables simple synchronous control.	- Adding superimposed control does not disable simple synchronous control. - The same axis can be used as the master axis for axis synchronous control and the master axis for superimposed control.

Function	Series 0*i*-TTC	Series 0*i*-D
Feed hold when an alarm occurs with respect to superimposed control	- Both paths are placed in the feed hold state.	- Make a selection using bit 0 (MPA) of parameter No. 8168. **Bit 0 (MPA) of parameter No. 8168** The axis movement in-progress signal <Fn102> or axis movement direction signal <Fn106> for the slave axis during superimposed control: 0: Places both paths in the feed hold state. 1: Places only the path including axes related to superposition control in the feed hold state. (For example, when superposition control is exerted in one path, only the path that caused the alarm is placed in the feed hold state.)
Reference position return of the slave axis during superimposed control	- Not available.	- Not available. Alarm PS0363 occurs.
Multiple slave axes	- Superimposed control cannot be exerted when there are multiple slave axes and one master axis.	- Superimposed control can be exerted when there are multiple slave axes and one master axis.
Axis movement in-progress signal and axis movement direction signal for the slave axis during superimposed control	- State output is performed according to the result of adding superimposed move pulses.	- Make a selection using bit 4 (AXS) of parameter No. 8160. **Bit 4 (AXS) of parameter No. 8160** The axis movement in-progress signal <Fn102> or axis movement direction signal <Fn106> for the slave axis during superimposed control: 0: Performs state output according to the result of adding superimposed move pulses. 1: Performs state output according to the result of moving the individual axes, regardless of superimposed move pulses.
Axis overtravel during superimposed control	- The superimposed control mode is canceled.	- Make a selection using bit 5 (NCS) of parameter No. 8160. **Bit 5 (NCS) of parameter No. 8160** If overtravel occurs for an axis under synchronous, composite, or superposition control, the synchronous, composite, or superposition control mode is: 0: Canceled. 1: Not canceled.
Switch between superimposed control axis selection signals during automatic operation	- The signals can be switched at any time. Note that both the master and slave axes must be stopped.	- Use an M code command. Specify a waiting M code (M code without buffering) before and after the M code. When superimposed control is exerted in one path, specify an M or other code without buffering before and after the M code that starts or cancels the control so as to prohibit the look-ahead operation.

K.43.2 Differences in Diagnosis Display

None.

K.44 Y AXIS OFFSET (T SERIES)

T

K.44.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Number of the axis for which the Y axis offset is used	- Make a selection using bit 7 (Y03) of parameter No. 5004. **Bit 7 (Y03) of parameter No. 5004** The Y axis offset is used for: 0: 4th axis. 1: 3rd axis.	- Make a selection using parameter No. 5043. When 0 or a value outside the data range is set, the Y axis offset is used for the Y axes of the basic three axes (X, Y, and Z).

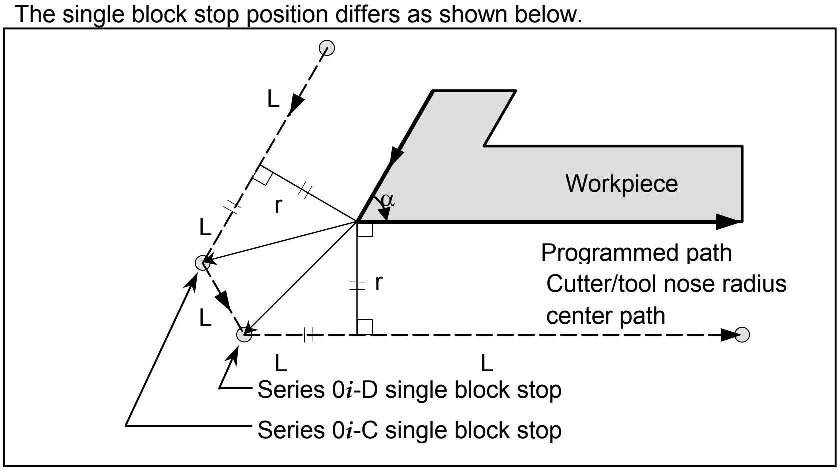
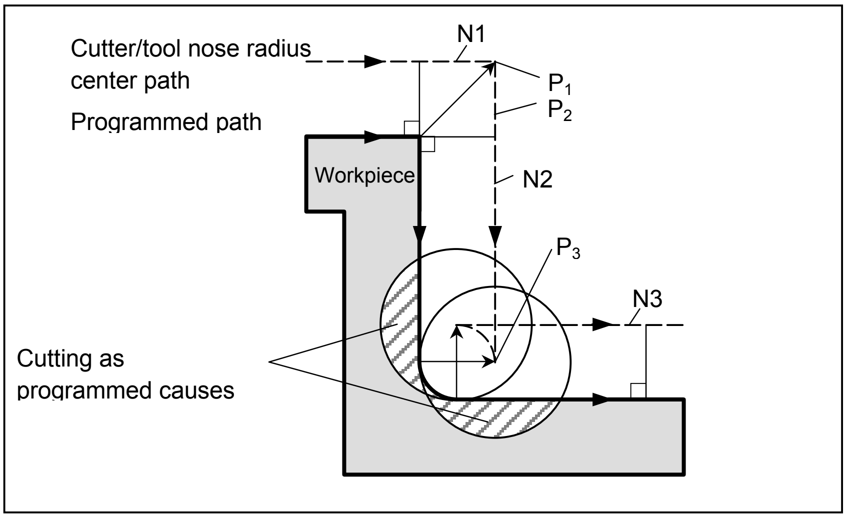
K.44.2 Differences in Diagnosis Display

None.

K.45 CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION

K.45.1 Differences in Specifications

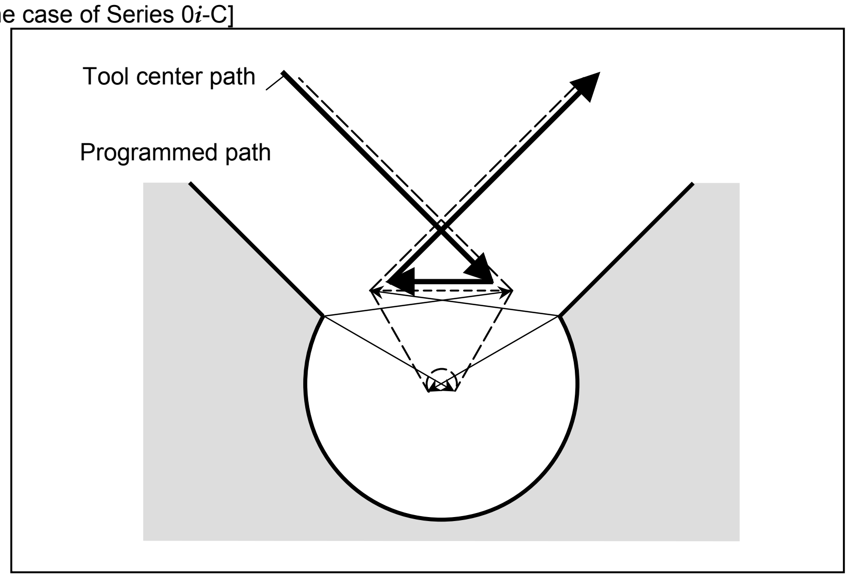
Function	Series 0*i*-C	Series 0*i*-D
Cutter compensation/tool nose radius compensation	- In Series 0*i*-D, the cutter compensation C (M series) and tool-nose radius compensation (T series) functions of Series 0*i*-C are collectively referred to as cutter compensation/tool nose radius compensation.	
Corner circular interpolation (G39)	**M** - Enabled by setting 1 in bit 2 (G39) of parameter No. 5008. **T** - Not available.	- Available. It is included in cutter compensation/tool nose radius compensation. Since corner circular interpolation (G39) is always enabled, bit 2 (G39) of parameter No. 5008 is not available.
Cutter compensation/tool nose radius compensation in MDI operation	- Neither cutter compensation C nor tool nose radius compensation is available in MDI operation.	- Cutter compensation/tool nose radius compensation is also available in MDI operation.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Single block stop position during the cutter compensation/tool nose radius compensation mode</td><td colspan="2">- The single block stop position differs as shown below.

</td></tr>
<tr><td>Function to change the compensation direction intentionally (IJ type vector, KI type vector, and JK type vector)</td><td>- Not available.</td><td>- At the start of or during the cutter compensation/tool nose radius compensation mode, specify I, J, or K in a G00 or G01 block. This makes the compensation vector at the end point of the block perpendicular to the direction specified by I, J, or K. This way, you can change the compensation direction intentionally.</td></tr>
<tr><td>Stop position upon an overcutting alarm</td><td colspan="2">- If the specified radius value for circular interpolation is smaller than that for cutter compensation/tool nose radius compensation, as in the example below, performing compensation inwardly through cutter compensation/tool nose radius compensation causes overcutting, generating an alarm and stopping the tool. The stop position differs.

</td></tr>
<tr><td></td><td colspan="2">[When single block stop occurs in the preceding block in Series 0i-C]
Since the tool moves until it reaches the end point of the block (P_3 in the figure), overcutting may result.
[When single block stop does not occur in the preceding block in Series 0i-C]
The tool stops immediately after executing the block (P_2 in the figure).
[In the case of Series 0i-D]
Since the tool stops at the start point of the block (P_1 in the figure), regardless of the single block state, overcutting can be prevented.</td></tr>
</table>

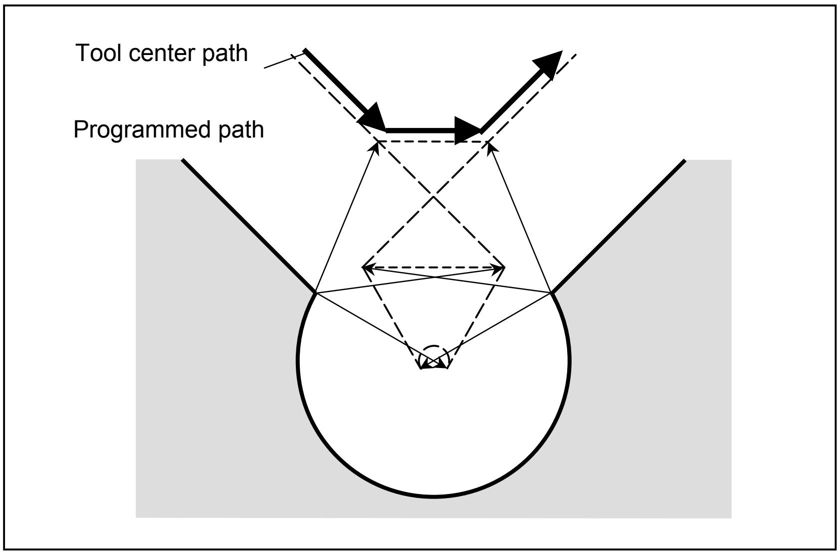
Function	Series 0*i*-C	Series 0*i*-D
Single block stop in a block created internally for cutter compensation/tool nose radius compensation	- Not available.	- Depends on bit 0 (SBK) of parameter No. 5000. **Bit 0 (SBK) of parameter No. 5000** In a block created internally for cutter compensation/tool nose radius compensation, single block stop is: 0: Not performed. 1: Performed. This parameter is used to check a program including cutter compensation/tool nose radius compensation.

Function	Series 0*i*-C	Series 0*i*-D
Setting to disable interference checking and to delete interfering vectors	- Set 1 in bit 0 (CNI) of parameter No. 5008. In the example below, an interference check is made on the vectors inside V_1 and V_4, and the interfering vectors are deleted. As a result, the tool center path is from V_1 to V_4.	- Not available. (Bit 0 (CNI) of parameter No. 5008 is not available.) To prevent overcutting, the interference check avoidance function (bit 5 (CAV) of parameter No. 19607) is used. In the example below, interference occurs between V_1 and V_4 and between V_2 and V_3. Therefore, vectors V_A and V_B are created. The tool center path is from V_A to V_B.

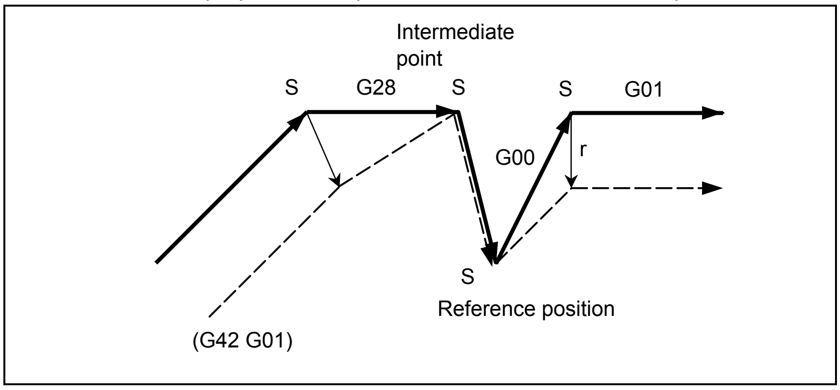
[In the case of Series 0*i*-C]

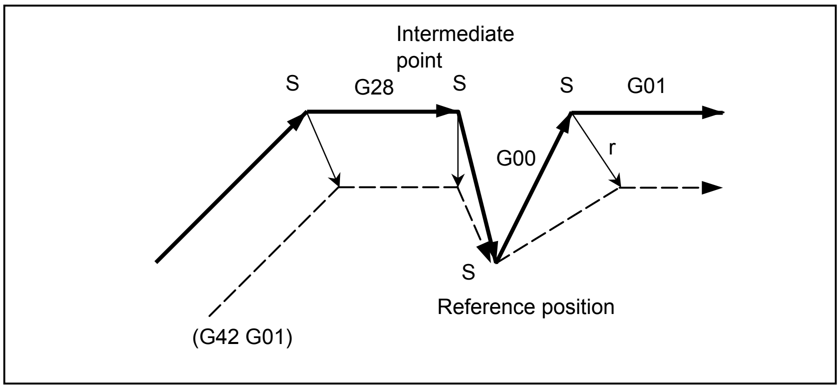


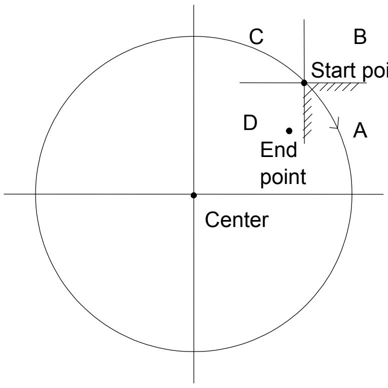
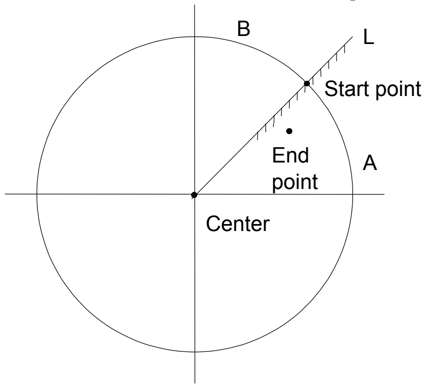
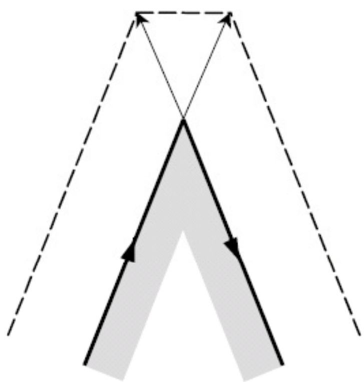
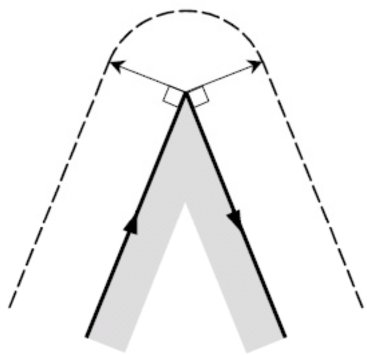
[In the case of Series 0*i*-D]



Function	Series 0*i*-C	Series 0*i*-D
Number of blocks to be read in the cutter compensation/tool nose radius compensation mode	- Always 3 blocks	- The number can be set in parameter No. 19625. The specifiable range is 3 to 8 blocks. If the parameter is not set (0 is set), the same number as Series 0*i*-C (3 blocks) is assumed.

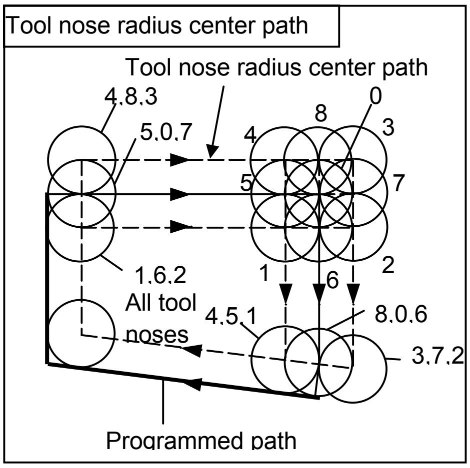
<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>When circular interpolation is specified that causes the center to coincide with the start or end point during the cutter compensation/tool nose radius compensation mode</td><td>- Alarm PS0038 is issued, and the tool stops at the end point of the block preceding the circular interpolation block.</td><td>- Alarm PS0041 is issued, and the tool stops at the start point of the block preceding the circular interpolation block.</td></tr>
<tr><td rowspan="2">Behavior when automatic reference position return is specified during the cutter compensation/tool nose radius compensation mode</td><td>- Depends on bit 2 (CCN) of parameter No. 5003.</td><td>- Bit 2 (CCN) of parameter No. 5003 is not available. The tool always behaves as when CCN is set to 1.</td></tr>
<tr><td colspan="2">[When CCN = 0]
The offset vector is canceled when the tool moves to the middle point.
Also, the start-up operation is performed from the reference position.


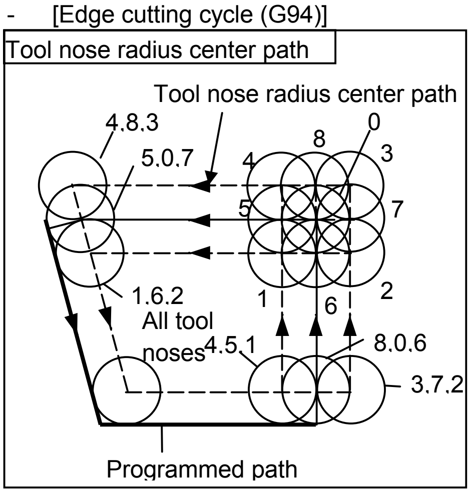
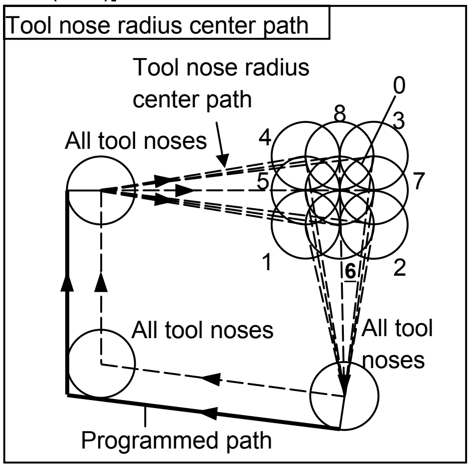
[When CCN = 1 or for Series 0i-D]
The offset vector is not canceled when the tool moves to the middle point; it is canceled when the tool moves to the reference position.
Also, the tool moves from the reference position to the next intersection point.

</td></tr>
</table>

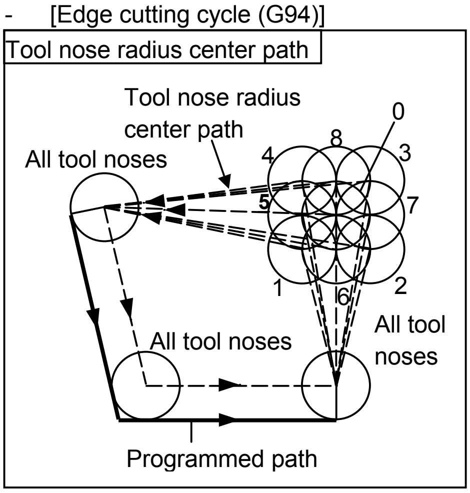
Function	Series 0*i*-C	Series 0*i*-D
Travel distance judgment method for circular interpolation in cutter compensation/tool nose radius compensation	- Depends on bit 5 (QCR) of parameter No. 5008. [When QCR = 0]  If the end point is on side A when viewed from the start point, the travel distance is small. If it is on side B, C, or D, the tool has traveled almost one round.	- Bit 5 (QCR) of parameter No. 5008 is not available. The tool always behaves as when QCR is set to 1. [When QCR = 1 or for Series 0*i*-D]  If the end point is on side A of line L connecting the start point and center, the travel distance is small. If it is on side B, the tool has traveled almost one round.
Compensation vector connection method when the tool travels around an external corner during the cutter compensation/tool nose radius compensation mode	- Connected by linear interpolation. [When CCC = 0 or for Series 0*i*-C] Connect vectors by linear interpolation 	- Depends on bit 2 (CCC) of parameter No. 19607. [When CCC = 1] Connect vectors by circular interpolation 

T

Function	Series 0*i*-C	Series 0*i*-D
Virtual tool tip direction and plane selection	- Virtual tool tip directions 1 to 8 can be used only for the G18 (Z-X) plane. When the virtual tool tip direction is 0 or 9, compensation can be performed for the G17 and G19 planes as well.	- All virtual tool tip directions can be used for the G17, G18, and G19 planes.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Tool nose radius center path for tool nose radius compensation in a canned cycle (G90 or G94)</td><td>- [Outer surface turning/boring cycle (G90)]


- [Edge cutting cycle (G94)]

</td><td>- [Outer surface turning/boring cycle (G90)]


- [Edge cutting cycle (G94)]

</td></tr>
<tr><td></td><td colspan="2">* Numbers 0 to 8 in the figure are virtual tool tip numbers.</td></tr>
<tr><td>Start-up/cancellation type of tool nose radius compensation</td><td>- The start-up/cancellation type cannot be set.</td><td>- Depends on bit 0 (SUP) and bit 1 (SUV) of parameter No. 5003.
When SUV and SUP are respectively set to 0 and 1 (type B), the processing is the same as Series 0i-C.</td></tr>
</table>

K.45.2 Differences in Diagnosis Display

None.

K.46 CANNED CYCLE FOR DRILLING

K.46.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
M05 output in a tapping cycle	- Make a selection using bit 6 (M5T) of parameter No. 5101. **<u>Bit 6 (M5T) of parameter No. 5101</u>** When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G74 with the M series, or G84/G88 with the T series): **T** 0: M05 is not output before output of M04 or M03. 1: M05 is output before output of M04 or M03. **M** 0: M05 is output before output of M04 or M03. 1: M05 is not output before output of M04 or M03.	- Make a selection using bit 3 (M5T) of parameter No. 5105. **<u>Bit 3 (M5T) of parameter No. 5105</u>** When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G74 with the M series, or G84/G88 with the T series): 0: M05 is output before output of M04 or M03. 1: M05 is not output before output of M04 or M03. **NOTE** This parameter corresponds to bit 6 (M5T) of parameter No. 5101 of Series 0*i*-C. With the T series, the logic of the values 0 and 1 is opposite from that of Series 0*i*-C.
Behavior when K0 is specified for the number of repetitions K	**T** - Make a selection using bit 5 (K0E) of parameter No. 5102. **<u>Bit 5 (K0E) of parameter No. 5102</u>** When K0 is specified in a drilling canned cycle (G80 to G89): 0: One drilling operation is performed. 1: Drilling operation is not performed, and only drilling data is stored. **M** - Drilling operation is not performed, and only drilling data is stored.	- Make a selection using bit 4 (K0D) of parameter No. 5105 for both T series and M series. **<u>Bit 4 (K0D) of parameter No. 5105</u>** When K0 is specified in a drilling canned cycle (G80 to G89): 0: Drilling operation is not performed, and only drilling data is stored. 1: One drilling operation is performed. **NOTE** With the T series, the logic of the values 0 and 1 is opposite from that of bit 5 (K0E) of parameter No. 5102 of Series 0*i*-C.
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **<u>Bit 1 (NRF) of parameter No. 3700</u>** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.

T

Function	Series 0*i*-C	Series 0*i*-D
Retraction in a boring cycle (G85, G89)	- Select the retraction operation using bit 1 (BCR) of parameter No. 5104. **<u>Bit 1 (BCR) of parameter No. 5104</u>** The retraction operation in a boring cycle is performed: at 0: Cutting feedrate In this case, the cutting feedrate of the retraction operation can be multiplied by the override value set in parameter No. 5121. The override value range is 100% to 2000%. 1: Rapid traverse rate In this case, rapid traverse override is also enabled.	- Bit 1 (BCR) of parameter No. 5104 is not available. The retraction operation is always performed at the cutting feedrate. In this case, the cutting feedrate of the retraction operation can be multiplied by the override value set in parameter No. 5149. The override value range is 1% to 2000%.
Clearance value in a peck drilling cycle	- Set the value in parameter No. 5114.	- Set the value in parameter No. 5115.
Drilling axis in the Series 10/11 format	- Y axis cannot be used as a drilling axis. P/S alarm No. 028 is issued.	- Y axis can be used as a drilling axis.

M

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Forward/retraction feedrate for the small-hole peck drilling cycle (G83)</td><td colspan="2">- When the I command (forward/retraction feedrate) is omitted and 0 is set in parameter Nos. 5172 and 5173, the forward/retraction feedrate is as follows.</td></tr>
<tr><td>0</td><td>Same feedrate as that specified by the F command</td></tr>
<tr><td>Tool retraction direction in a fine boring cycle (G76) or back boring cycle (G87)</td><td>- Set the direction using bit 5 (RD2) and bit 4 (RD1) of parameter No. 5101 in combination.</td><td>- Bit 5 (RD2) and bit 4 (RD1) of parameter No. 5101 is not available.
Set the direction in axis-type parameter No. 5148.</td></tr>
<tr><td rowspan="2">Address Q command in a high-speed peck drilling cycle (G73), peck drilling cycle (G83), or small-hole peck drilling cycle (G83)</td><td colspan="2">- In a high-speed peck drilling cycle (G73), peck drilling cycle (G83), and small-hole peck drilling cycle (G83), when the address Q (amount of each-time cutting) command is not specified or Q0 is specified:</td></tr>
<tr><td>Select the operation using bit 1 (QZA) of parameter No. 5103.

<u>Bit 1 (QZA) of parameter No. 5103</u>
0: The tool repeats the upward and downward movement at the same position without cutting.
1: P/S alarm No. 045 is issued.</td><td>Bit 1 (QZA) of parameter No. 5103 is not available.
The tool always behaves as when 1 is set in bit 1 (QZA) of parameter No. 5103. (Alarm PS0045 is issued.)</td></tr>
<tr><td>Tool length compensation (G43 or G44) in a canned cycle when tool length compensation type C is selected (1 is set in bit 0 (TLC) of parameter No. 5001)</td><td>- Select the axis for which to enable tool length compensation, by using bit 4 (TCE) of parameter No. 5006.

<u>Bit 4 (TCE) of parameter No. 5006</u>
When tool length compensation (G43 or G44) is specified in a canned cycle, tool length compensation is enabled for:
0: Axis selected according to tool length compensation type C.
1: Drilling axis.</td><td>- Bit 4 (TCE) of parameter No. 5006 is not available.
The tool always behaves as when 1 is set in bit 4 (TCE) of parameter No. 5006.</td></tr>
</table>

K.46.2 Differences in Diagnosis Display

None.

K.47 CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (T SERIES)

T

K.47.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Machining plane</td><td>- The plane on which the canned cycle is performed is always the ZX plane.</td><td>- The plane on which the canned cycle ca be selected arbitrarily (including a parallel axis).
Note that, with G code system A, an axis whose name is U, V, or W cannot be set as a parallel axis.</td></tr>
<tr><td>Address R setting unit (Address I, J, or K for the Series 10/11 format)</td><td>- The setting unit common to all axes is used.</td><td>- The setting unit applies to a different axis depending on the machining plane and the command.
Second axis of the axes comprising the machining plane for G90 and G92
First axis of the axes comprising the machining plane for G94</td></tr>
<tr><td>Application of tool nose radius compensation</td><td colspan="2">- Refer to Section 4.1.5, "CANNED CYCLE AND TOOL NOSE RADIUS COMPENSATION" in "OPERATOR'S MANUAL (T SERIES)" (B-64304EN-1). The differences in specifications are detailed.</td></tr>
<tr><td>Inch threading by address E (Series 10/11 format)</td><td>- Threading is performed as the lead threading command of address F.</td><td>- Inch threading is performed.</td></tr>
<tr><td>Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle</td><td>- The behavior can be selected using bit 1 (NRF) of parameter No. 3700.

<u>Bit 1 (NRF) of parameter No. 3700</u>
After a serial spindle is changed to a Cs contour control axis, the first move command:
0: Performs the normal positioning operation after executing the reference position return operation.
1: Performs the normal positioning operation.</td><td>- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.</td></tr>
</table>

K.47.2 Differences in Diagnosis Display

None.

K.48 CANNED GRINDING CYCLE

K.48.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Grinding axis specification	**T** - The grinding axis is always the Z axis. **M** - The grinding axis is the X or Z axis.	- Set the grinding axes for the individual canned grinding cycles in parameter Nos. 5176 to 5179. If the same axis number as the cutting axis is specified in any of these parameters, or if a canned grinding cycle is executed when 0 is set, alarm PS0456 is issued.
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **Bit 1 (NRF) of parameter No. 3700** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.

T

Function	Series 0*i*-C	Series 0*i*-D
Exclusive control against the multiple respective canned cycle (standard function)	- When the grinding canned cycle option is specified, the multiple respective canned cycle (standard function) cannot be used.	- When the grinding canned cycle option is specified, select whether to use the multiple respective canned cycle (standard function) or grinding canned cycle, by using bit 0 (GFX) of parameter No. 5106. **Bit 0 (GFX) of parameter No. 5106** When the grinding canned cycle option is specified, the G71, G72, G73, and G74 commands are intended for: 0: Multiple respective canned cycle. 1: Grinding canned cycle.

M

Function	Series 0*i*-C	Series 0*i*-D
Dressing axis specification	- The dressing axis is always the fourth axis.	- Set the dressing axes for the individual canned grinding cycles in parameter Nos. 5180 to 5183. If the same axis number as the cutting axis or grinding axis is specified in any of these parameters, or if a canned grinding cycle is executed when 0 is set, alarm PS0456 is issued.

K.48.2 Differences in Diagnosis Display

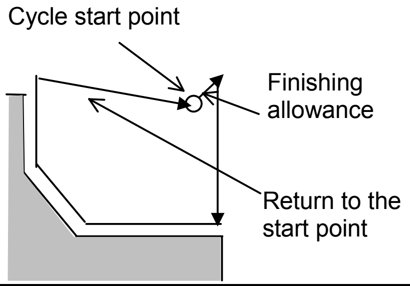
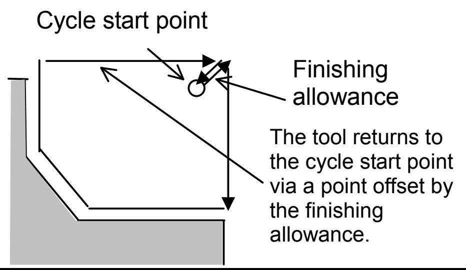
None.

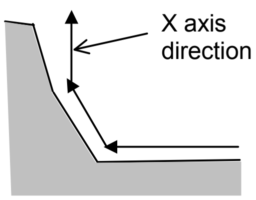
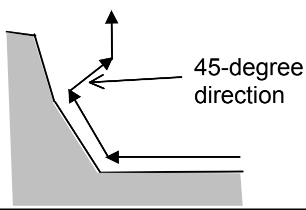
K.49 MULTIPLE RESPECTIVE CANNED CYCLE FOR TURNING (T SERIES)

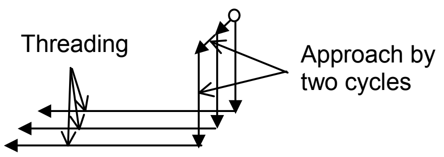
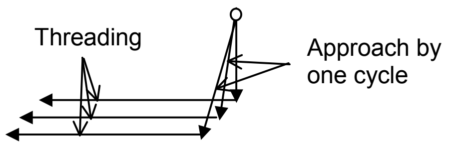
T

K.49.1 Differences in Specifications

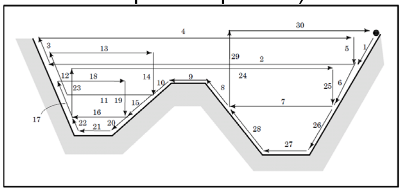
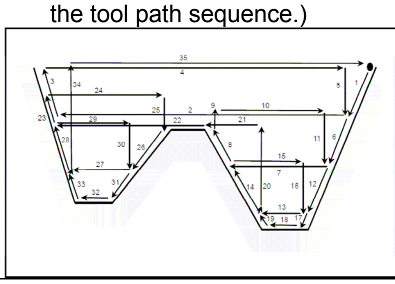
Differences common to the Series 0 standard format and Series 10/11 format

Function	Series 0*i*-C	Series 0*i*-D
Specifiable plane	- The cycle can be specified for a Z-X plane, with the X axis set as the first axis and the Z axis set as the second axis.	- The cycle can be specified for an arbitrary plane selected with the basic three axes and their parallel axes.
Specification for a plane including a parallel axis	- Not allowed.	- For G code system A, the cycle can be specified when the name of the parallel axis is other than U, V, or W. (To use U, V, or W as an axis name is not allowed for G code system A.)
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **<u>Bit 1 (NRF) of parameter No. 3700</u>** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.
Cycle start point return path when the finishing allowance is specified in G71 or G72	- The tool returns directly to the cycle start point. 	- The tool returns to the cycle start point via a point offset by the finishing allowance. 

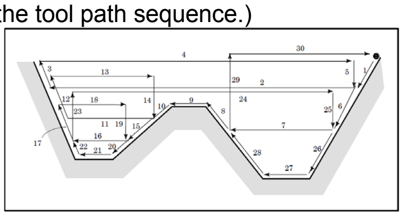
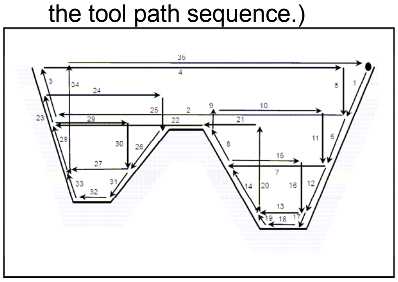
Function	Series 0*i*-C	Series 0*i*-D
Monotonous increase/decrease check in G71/G72 type I (multiple respective canned cycle for turning)	- Depends on bit 1 (MRC) of parameter No. 5102. **<u>Bit 1 (MRC) of parameter No. 5102</u>** When any target figure other than monotonous increase or decrease is specified in a multiple respective canned cycle for turning (G71 or G72): 0: An alarm is not issued. 1: Alarm PS0064 is issued.	- Bit 1 (MRC) of parameter No. 5102 is not available. If monotonous increase or decrease is not specified for the first axis direction of the plane, alarm PS0064 is issued. If monotonous increase or decrease is not specified for the second axis direction of the plane, alarm PS0329 is issued. Note that, by setting a permissible amount in parameter Nos. 5145 and 5146, it is possible to prevent the alarm from occurring, even if the monotonous increase/decrease condition is not met, as long as the permissible amount is not exceeded.
Monotonous increase/decrease check in G71/G72 type II (multiple respective canned cycle for turning II)	- Not checked. Bit 1 (MRC) of parameter No. 5102 does not take effect for multiple respective canned cycle for turning II (type II).	- Always checked. If monotonous increase or decrease is not specified for the first axis direction of the plane, alarm PS0064 is issued. Note that, by setting a permissible amount in parameter No. 5145, it is possible to prevent the alarm from occurring, even if the monotonous increase/decrease condition is not met, as long as the permissible amount is not exceeded.
Roughing after start point return by G71 or G72	- Not performed.	- [Multiple respective canned cycle for turning I (type I)] Depends on bit 1 (RF1) of parameter No. 5105. [Multiple respective canned cycle for turning II (type II)] Depends on bit 2 (RF2) of parameter No. 5105.
	<u>Bit 1 (RF1) of parameter No. 5105</u> In the multiple repetitive canned cycle (T series) (G71/G72) of type I, roughing is: 0: Performed. 1: Not performed.	**<u>Bit 2 (RF2) of parameter No. 5105</u>** In the multiple repetitive canned cycle (T series) (G71/G72) of type II, roughing is: 0: Performed. 1: Not performed.
Retraction operation at the bottom of a hole in G71/G72 type II (multiple respective canned cycle for turning II)	- The tool retracts in the X axis direction after chamfering. 	- After chamfering, the tool first retracts in the 45-degree direction and then in the second axis direction of the plane. 

Function	Series 0*i*-C	Series 0*i*-D
G70 to G76 commands during the tool nose radius compensation mode	- [G70 command] Tool nose radius compensation is performed. [G71 to G73 commands] While tool nose radius compensation is not performed, it is possible to apply tool nose radius compensation partially by setting bit 4 (RFC) of parameter No. 5102. **Bit 4 (RFC) of parameter No. 5102** For a G71 or G72 semi-finished shape or a G73 cutting pattern, tool nose radius compensation is: 0: Not performed. 1: Performed. [G74 to G76 commands] Tool nose radius compensation is not performed.	- Bit 4 (RFC) of parameter No. 5102 is not available. [G70 to G73 commands] Tool nose radius compensation is performed. [G74 to G76 commands] Tool nose radius compensation is not performed.
Positioning in G70 to G76 cycle operations	- Non-linear type positioning is always used, regardless of the setting of bit 1 (LRP) of parameter No. 1401.	- [Start point return by G70] Non-linear type positioning is always used. [Other positioning operations] Depends on bit 1 (LRP) of parameter No. 1401.
T code specified in the same block as G74 or G75	- Invalid	- Valid
Chamfering and corner R commands and direct drawing dimension programming command for a target figure program	- Cannot be specified.	- Can be specified. Note that the last block of the target figure program must not be in the middle of the chamfering, corner R, or direct drawing dimension programming command.
Approach to the threading start point in G76	- Approach by two cycles 	- Approach by one cycle 

Differences regarding the Series 0 standard format

Function	Series 0*i*-C	Series 0*i*-D
Pocketing path in G71/G72 type II (multiple respective canned cycle for turning II)	- The tool moves from one pocket to another for each cut. (The numbers in the figure represent the tool path sequence.) 	- The tool completes one pocketing process before proceeding to cut the next pocket. (The numbers in the figure represent the tool path sequence.) 
Limitation on the number of pockets in G71/G72 type II (multiple respective canned cycle for turning II)	- Up to 10 pockets can be specified. Specifying 11 or more pockets causes alarm PS0068.	- Not limited.
Number of divisions in G73	- The number of divisions is also 2 for the R1 command. For R2 and subsequent commands, the number of divisions specified by R applies.	- The number of divisions specified by R applies.

Differences regarding the Series 10/11 format

Function	Series 0*i*-C	Series 0*i*-D
Pocketing path in G71/G72 type II (multiple respective canned cycle for turning II)	- Depends on bit 2 (P15) of parameter No. 5103. [When P15 = 0] The tool moves from one pocket to another for each cut. (The numbers in the figure represent the tool path sequence.)  [When P15 = 1] The tool completes one pocketing process before proceeding to cut the next pocket. (See the figure at right.)	- Bit 2 (P15) of parameter No. 5103 is not available. The tool completes one pocketing process before proceeding to cut the next pocket. (The numbers in the figure represent the tool path sequence.) 
Limitation on the number of pockets in G71/G72 type II (multiple respective canned cycle for turning II)	- Depends on bit 2 (P15) of parameter No. 5103. [When P15 = 0] Up to 10 pockets can be specified. Specifying 11 or more pockets causes alarm PS0068. [When P15 = 1] Not limited.	- Bit 2 (P15) of parameter No. 5103 is not available. Not limited.
Specification of finishing allowance in G71/G72	- Not allowed. The finishing allowance is ignored if specified.	- Allowed.

Function	Series 0*i*-C	Series 0*i*-D
Number of divisions in G73	- The number of divisions is also 2 for the D1 command. For D2 and subsequent commands, the number of divisions specified by D applies.	- The number of divisions specified by D applies.
Address E command in G76	- Threading is performed as the lead threading command of address F.	- Inch threading is performed.

K.49.2 Differences in Diagnosis Display

None.

K.50 CHAMFERING AND CORNER ROUNDING (T SERIES)

T

K.50.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Chamfering and corner rounding commands for a plane other than the Z-X plane	- Not available. Alarm PS0212 is issued.	- Available. The commands can be specified for any plane, even one that includes a parallel axis.
Single block operation	- [Chamfering] Single block stop is not performed at the start point of an inserted chamfering block. [Corner rounding] Single block stop is performed at the start point of an inserted corner rounding block.	- [Common to chamfering and corner rounding] Whether to perform single block stop at the start point of an inserted block depends on bit 0 (SBC) of parameter No. 5105. **Bit 0 (SBC) of parameter No. 5105** In a drilling canned cycle, chamfer cycle/corner rounding (T series) or optional angle chamfering/corner rounding cycle (M series): 0: Single block stop is not performed. 1: Single block stop is performed.

K.50.2 Differences in Diagnosis Display

None.

K.51 DIRECT DRAWING DIMENSIONS PROGRAMMING (T SERIES)

T

K.51.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Specification of the direct drawing dimension programming command for a plane other than the Z-X plane	- P/S alarm No. 212 is issued.	- No alarm is issued. The command can be specified for a plane other than the Z-X plane.
When two or more blocks not to be moved exist between consecutive commands that specify direct input of drawing dimensions	- No alarm is issued.	- Alarm PS0312 is issued.

K.51.2 Differences in Diagnosis Display

None.

K.52 SINGLE DIRECTION POSITIONING (M SERIES)

M

K.52.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Behavior when linear interpolation type positioning is used with mirror image	If positioning of linear interpolation type is used (1 is set in bit 1 (LRP) of parameter No. 1401), and if the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, the following alarms are issued, respectively.	
	- Alarm PS5254	- Alarm DS0025

K.52.2 Differences in Diagnosis Display

None.

K.53 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING (M SERIES)

M

K.53.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Optional angle chamfering and corner rounding commands for a plane including a parallel axis	- Not available. Alarm PS0212 is issued.	- Available.
Single block operation	- Single block stop is not performed at the start point of an inserted optional angle chamfering or corner rounding block.	- Whether to perform single block stop at the start point of an inserted block depends on bit 0 (SBC) of parameter No. 5105. **Bit 0 (SBC) of parameter No. 5105** In a drilling canned cycle, chamfer cycle/corner rounding (T series) or optional angle chamfering/corner rounding cycle (M series): 0: Single block stop is not performed. 1: Single block stop is performed.
Negative value specified in a ,C_ or ,R_ command	- The value is regarded as positive.	- Alarm PS0006 is issued.
Number of dwells to be inserted between two blocks for which to perform optional angle chamfering or corner rounding	- Not limited.	- Only one block can be inserted. Inserting more than one block causes alarm PS0051.
DNC operation	- Optional angle chamfering and corner rounding are not available in DNC operation.	- Optional angle chamfering and corner rounding are also available in DNC operation.

K.53.2 Differences in Diagnosis Display

None.

INDEX

Revision Record

FANUC Series 0*i*-MODEL D/Series 0*i* Mate-MODEL D OPERATOR'S MANUAL
(Common to Lathe System/Machining Center System) (B-64304EN)

Edition	Date	Contents	Edition	Date	Contents
02	Aug., 2010	Total revision			
01	Jun., 2008	———			