

FANUC Series *0i*-PD

OPERATOR'S MANUAL

B-64554EN/01

- No part of this manual may be reproduced in any form.
- All specifications and designs are subject to change without notice.

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.

The products in this manual are manufactured under strict quality control. However, when using any of the products in a facility in which a serious accident or loss is predicted due to a failure of the product, install a safety device.

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".

This manual contains the program names or device names of other companies, some of which are registered trademarks of respective owners. However, these names are not followed by ® or ™ in the main body.

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	s-1
GENERAL WARNINGS AND CAUTIONS	s-2
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING	s-3
WARNINGS AND CAUTIONS RELATED TO HANDLING	s-4
WARNINGS RELATED TO DAILY MAINTENANCE	s-6

DEFINITION OF WARNING, CAUTION, AND NOTE

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



WARNING

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



CAUTION

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

NOTE

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

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

GENERAL WARNINGS AND CAUTIONS

WARNING

- 1 Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 2 Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 3 Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 4 When using a tool compensation function, thoroughly check the direction and amount of compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

CAUTION

- 1 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.
- 2 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.

 CAUTION

- 3 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.
- 4 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 normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4 Inch/metric conversion

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

 WARNING**5 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.

 CAUTION**1 Absolute/incremental mode**

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

2 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.

3 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.

4 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.

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.

 WARNING**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), 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 and menu switches

Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.

Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

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.

 CAUTION**1 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 parameters, and absolute/incremental command mode.

2 Feed hold, override, and single block

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

 CAUTION**3 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.

4 Cutter radius compensation in MDI mode

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

5 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 "IV. MAINTENANCE - Method of replacing battery" in the OPERATOR'S MANUAL for details of the battery replacement procedure.

 WARNING**2 Absolute pulse coder battery replacement**

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

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

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

NOTE

The absolute pulse coder uses batteries to preserve its absolute position. If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost. Refer to the Section "IV. MAINTENANCE - Method of replacing battery" in the OPERATOR'S MANUAL 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

SAFETY PRECAUTIONS	S-1
DEFINITION OF WARNING, CAUTION, AND NOTE	S-1
GENERAL WARNINGS AND CAUTIONS.....	S-2
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING	S-3
WARNINGS AND CAUTIONS RELATED TO HANDLING	S-4
WARNINGS RELATED TO DAILY MAINTENANCE	S-6
 I. GENERAL	
1 GENERAL	3
1.1 NOTES ON READING THIS MANUAL.....	5
1.2 NOTES ON VARIOUS KINDS OF DATA	5
 II. PROGRAMMING	
1 GENERAL	9
1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS	
FIGURE-INTERPOLATION	10
1.2 FEED-FEED FUNCTION	12
1.3 PART DRAWING AND TOOL MOVEMENT	12
1.3.1 Reference Position (Machine-specific Position)	12
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC -	
Coordinate System	13
1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute and	
Incremental Programming).....	15
1.4 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL	
FUNCTION	16
1.5 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION	16
1.6 PROGRAM CONFIGURATION	17
1.7 TOOL MOVEMENT RANGE - STROKE.....	19
2 CONTROLLED AXES	20
2.1 NUMBER OF CONTROLLED AXES	20
2.2 NAMES OF AXES	20
2.3 INCREMENT SYSTEM.....	20
2.4 MAXIMUM STROKE.....	21
3 PREPARATORY FUNCTION (G FUNCTION)	22
3.1 G CODE LIST	23
4 INTERPOLATION FUNCTIONS.....	25
4.1 POSITIONING (G00).....	25
4.2 LINEAR INTERPOLATION (G01).....	26
4.3 CIRCULAR INTERPOLATION (G02, G03).....	28
4.4 HELICAL INTERPOLATION (G02, G03)	32
4.5 SKIP FUNCTION (G33).....	33

4.6	MULTI-STEP SKIP (G33)	35
4.7	HIGH-SPEED SKIP SIGNAL (G33)	36
4.8	SKIP POSITION MACRO VARIABLE IMPROVEMENT	36
4.9	TORQUE LIMIT SKIP	36
5	FEED FUNCTIONS	40
5.1	OVERVIEW	40
5.2	RAPID TRAVERSE	41
5.2.1	Rapid Traverse Rate by F Command	41
5.2.2	F1-digit (Programmable Rapid Traverse Override)	42
5.3	CUTTING FEED	42
5.4	CUTTING FEEDRATE CONTROL	43
5.4.1	Exact Stop (G09, G61), Cutting Mode (G64)	44
5.4.2	Automatic Corner Override	45
5.4.2.1	Automatic override for inner corners (G62)	45
5.4.2.2	Internal circular cutting feedrate change	47
5.5	FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS	47
5.6	DWELL	50
6	REFERENCE POSITION.....	52
6.1	REFERENCE POSITION RETURN.....	52
7	COORDINATE SYSTEM.....	56
7.1	MACHINE COORDINATE SYSTEM.....	56
7.2	WORKPIECE COORDINATE SYSTEM	58
7.2.1	Setting a Workpiece Coordinate System.....	58
7.2.2	Selecting a Workpiece Coordinate System	59
7.2.3	Changing Workpiece Coordinate System	60
7.2.4	Workpiece Coordinate System Preset (G92.1).....	61
7.2.5	Automatic Coordinate System Setting	63
7.3	LOCAL COORDINATE SYSTEM	63
7.4	PLANE SELECTION.....	65
8	COORDINATE VALUE AND DIMENSION	66
8.1	ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91).....	66
8.2	INCH/METRIC CONVERSION (G20, G21)	66
8.3	DECIMAL POINT PROGRAMMING	69
9	PRESS FUNCTION	71
9.1	PUNCH FUNCTION (1-CYCLE PRESSING).....	71
9.1.1	Block in which Punching is Made.....	71
9.2	POSITIONING & PRESSING OFF (G70)	73
9.3	NIBBLING FUNCTION	73
9.3.1	Circular Nibbling (G68)	75
9.3.2	Linear Nibbling (G69).....	79
9.3.3	Notes on Circular Nibbling (G68) and Linear Nibbling (G69)	80
9.4	NIBBLING BY M FUNCTION.....	82
9.4.1	G00 Command in Nibbling Mode.....	82
9.4.2	G01, G02, and G03 Commands in Nibbling Mode.....	83
9.4.3	Notes on Nibbling by M Function.....	86

9.5	EXTERNAL MOTION FUNCTION	86
10	S FUNCTION.....	87
10.1	SPECIFYING THE S CODE WITH A BINARY CODE	87
11	TOOL FUNCTION (T FUNCTION)	88
11.1	TOOL SELECTION FUNCTION	88
11.2	T COMMAND NEGLECT	89
11.3	TOOL OFFSET	90
11.4	CONTROLLING THE TURRET-AXIS (T-AXIS)	90
11.5	MULTIPLE TOOL CONTROL	90
11.5.1	Tool Number	91
11.5.2	Relationship between the Multiple-Tool System and the C Axis	92
11.5.3	Tool Compensation	92
11.5.4	Operational Notes	93
11.6	TOOL LIFE MANAGEMENT FUNCTION	93
11.6.1	Tool Life Management Data	93
11.6.2	Register and Change of Tool Life Management Data	93
11.6.3	Tool Life	94
12	AUXILIARY FUNCTION.....	95
12.1	AUXILIARY FUNCTION (M FUNCTION).....	95
12.2	MULTIPLE M COMMANDS IN A SINGLE BLOCK.....	97
12.3	SECOND AUXILIARY FUNCTIONS (B CODES)	98
12.4	M COMMAND FOR SWITCHING THE PUNCH AND LASER MODE	100
13	PROGRAM MANAGEMENT	101
13.1	PROGRAM ATTRIBUTES	101
13.2	RELATED PARAMETERS.....	101
13.3	PART PROGRAM STORAGE SIZE / NUMBER OF REGISTERABLE PROGRAMS.....	102
14	PROGRAM CONFIGURATION.....	103
14.1	PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS.....	104
14.2	PROGRAM SECTION CONFIGURATION	106
14.3	SUBPROGRAM (M98, M99)	112
15	FUNCTIONS TO SIMPLIFY PROGRAMMING	116
15.1	PATTERN FUNCTION	116
15.1.1	Base Point Command (G72).....	117
15.1.2	Bolt Hole Circle (G26)	118
15.1.3	Line at Angle (G76)	120
15.1.4	Arc (G77)	121
15.1.5	Grid (G78, G79)	122
15.1.6	Share Proofs (G86).....	125
15.1.7	Square (G87)	127
15.1.8	Radius (G88)	129
15.1.9	Cut at Angle (G89).....	130
15.1.10	Incremental Command Just After Pattern Function	130
15.1.11	Notes on Pattern Functions.....	133
15.2	MEMORY AND CALL BY A/B MACRO	134

15.3	AUTOMATIC REPOSITIONING (G75)	135
15.4	U/V/W MACRO FUNCTION	141
15.4.1	Storage of Macros	141
15.4.2	Macro Call.....	142
15.4.3	Nesting Call of Macros.....	143
15.4.4	Macro Storage Capacity	144
15.4.5	Storage and Call of Multiple Macros (Macro Numbers 90 to 99).....	144
15.4.6	Deletion of Stored Macros	145
15.5	MULTI-PIECE MACHINING FUNCTION	145
15.5.1	Base Point Command of Multi-Piece Machining (G98)	145
15.5.2	Multi-Piece Machining Commands (G73, G74).....	148
15.5.3	Setting of Machining Method for Multi-Piece Machining	149
15.5.4	Command for Restarting Machining Multiple Products	150
15.6	BENDING COMPENSATION (G38, G39)	152
15.7	LINEAR AND CIRCULAR PUNCH COMMAND	154
15.7.1	Linear Punch Command (G45)	154
15.7.2	Circular Punch Commands (G46 and G47).....	157
15.7.3	Controlling the C-axis.....	158
15.8	Y-AXIS CRACK CANCEL	159
16	COMPENSATION FUNCTION	160
16.1	SCALING (G50, G51)	160
16.2	NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1, or G150,G151, G152).....	166
16.3	OVERVIEW OF TOOL COMPENSATION (G40-G42).....	170
16.4	DETAILS OF TOOL RADIUS COMPENSATION.....	175
16.4.1	Overview	175
16.4.2	Tool Movement in Start-up	179
16.4.3	Tool Movement in Offset Mode.....	184
16.4.4	Tool Movement in Offset Mode Cancel.....	202
16.4.5	Prevention of Overcutting Due to Tool Radius Compensation	208
16.4.6	Interference Check	211
16.4.6.1	Operation to be performed if an interference is judged to occur	214
16.4.6.2	Interference check alarm function	215
16.4.6.3	Interference check avoidance function	216
16.4.7	Tool Radius Compensation for Input from MDI.....	222
16.5	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	223
16.6	COORDINATE SYSTEM ROTATION (G84, G85).....	224
17	CUSTOM MACRO.....	231
17.1	VARIABLES.....	231
17.2	SYSTEM VARIABLES	236
17.3	ARITHMETIC AND LOGIC OPERATION	256
17.4	READING PARAMETERS.....	262
17.5	MACRO STATEMENTS AND NC STATEMENTS.....	263
17.6	BRANCH AND REPETITION.....	264
17.6.1	Unconditional Branch (GOTO Statement)	264
17.6.2	GOTO Statement Using Stored Sequence Numbers	264
17.6.3	Conditional Branch (IF Statement)	265
17.6.4	Repetition (WHILE Statement)	266

17.7	MACRO CALL	268
17.7.1	Simple Call (G65)	269
17.7.2	Modal Call: Call After the Move Command (G66)	272
17.7.3	Macro Call Using a G Code	273
17.7.4	Macro Call Using a G Code (Specification of Multiple Definitions).....	275
17.7.5	Macro Call Using an M Code.....	275
17.7.6	Macro Call Using an M Code (Specification of Multiple Definitions).....	276
17.7.7	Subprogram Call Using an M Code	277
17.7.8	Subprogram Call Using an M Code (Specification of Multiple Definitions).....	278
17.7.9	Subprogram Calls Using a T Code	278
17.7.10	Subprogram Call Using a Specific Address	279
17.8	PROCESSING MACRO STATEMENTS	281
17.9	REGISTERING CUSTOM MACRO PROGRAMS	283
17.10	CODES AND RESERVED WORDS USED IN CUSTOM MACROS	283
17.11	EXTERNAL OUTPUT COMMANDS.....	284
17.12	RESTRICTIONS	287
17.13	INTERRUPTION TYPE CUSTOM MACRO	288
17.13.1	Specification Method	289
17.13.2	Details of Functions.....	290
18	PROGRAMMABLE PARAMETER INPUT (G10).....	296
18.1	PROGRAMMABLE PARAMETER INPUT	296
18.2	TOOL DATA INPUT.....	298
19	HIGH-SPEED CUTTING FUNCTIONS.....	300
19.1	AI ADVANCED PREVIEW CONTROL (G05.1)	300
20	AXIS CONTROL FUNCTIONS.....	311
20.1	AXIS SYNCHRONOUS CONTROL	311
20.1.1	Axis Configuration for Axis Synchronous Control.....	311
20.1.2	Synchronous Establishment	314
20.1.3	Automatic Setting for Grid Position Matching	315
20.1.4	Synchronous Error Check	316
20.1.5	Methods of Alarm Recovery by Synchronous Error Check.....	317
20.1.6	Axis Synchronous Control Torque Difference Alarm.....	318
20.2	ROTARY AXIS ROLL-OVER	320
20.2.1	Rotary Axis Roll-over	320
20.3	TANDEM CONTROL	321
20.4	C AXIS CONTROL (DIE ANGLE INDEXING).....	321
20.4.1	Simultaneously Controlled Axes.....	322
20.4.2	Increment System	322
20.4.3	Maximum Programmable Dimension	322
20.4.4	Automatic Acceleration/Deceleration	322
20.4.5	Manual Continuous Feed, Incremental Feed, Manual Reference Point Return ...	322
20.4.6	Relationship with Absolute/Incremental Command (G90/G91)	323
20.4.7	Positioning in Smaller Angle Rotating Direction.....	323
20.4.8	Blocks Where C-axis Command is Possible.....	323
20.4.9	C-axis Command and its Operation.....	323
20.4.10	Pattern Function, Nibbling Function and C-axis Command.....	324
20.4.11	C-axis Command in Nibbling Mode.....	325
20.4.12	T-axis Command Ignore Signal TNG and C-axis Command	325
20.4.13	Compensating the Position of the C-axis.....	326
20.4.14	Compensating Backlash Along the C-axis for Each Tool Group	326

20.5	RAM-AXIS CONTROL	326
20.5.1	Setting RAM-axis Motion Pattern.....	326
20.5.2	Press Motion by RAM-axis Control.....	327
20.5.3	RAM-axis Up Motion	327

III. OPERATION

1	GENERAL	331
1.1	MANUAL OPERATION.....	331
1.2	TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION	332
1.3	AUTOMATIC OPERATION	333
1.4	TESTING A PROGRAM	334
1.4.1	Check by Running the Machine	334
1.4.2	How to View the Position Display Change without Running the Machine	336
1.5	EDITING A PROGRAM	336
1.6	DISPLAYING AND SETTING DATA.....	336
1.7	DISPLAY	339
1.7.1	Program Display	339
1.7.2	Current Position Display	340
1.7.3	Alarm Display	341
1.7.4	Parts Count Display, Run Time Display	341
2	OPERATIONAL DEVICES	343
2.1	POWER ON/OFF.....	343
2.1.1	Turning on the Power	343
2.1.2	Power Disconnection.....	344
2.2	SETTING AND DISPLAY UNITS.....	344
2.2.1	8.4" LCD/MDI	345
2.2.2	10.4" LCD	346
2.2.3	Standard MDI Unit (ONG Key)	347
2.2.4	Small MDI Unit (ONG Key).....	347
2.3	EXPLANATION OF THE MDI UNIT	349
2.4	FUNCTION KEYS AND SOFT KEYS	350
2.4.1	General Screen Operations.....	350
2.4.2	Function Keys	351
2.4.3	Soft Keys	352
2.4.4	Key Input and Input Buffer	359
2.4.5	Warning Messages	360
2.5	EXTERNAL I/O DEVICES	360
3	MANUAL OPERATION	362
3.1	MANUAL REFERENCE POSITION RETURN.....	362
3.2	JOG FEED (JOG)	363
3.3	INCREMENTAL FEED	364
3.4	MANUAL HANDLE FEED.....	365
3.5	MANUAL ABSOLUTE ON	368
3.6	DISTANCE CODED LINEAR SCALE INTERFACE.....	372
3.6.1	Procedure for Reference Position Establishment	372
3.6.2	Reference Position Return.....	373
3.6.3	Distance Coded Rotary Encoder	374
3.6.4	Axis Synchronization Control	374

3.6.5	Axis Control by PMC.....	375
3.6.6	Note	375
3.7	LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)	376
4	AUTOMATIC OPERATION	380
4.1	MEMORY OPERATION	380
4.2	MDI OPERATION	382
4.3	DNC OPERATION	385
4.4	SCHEDULE OPERATION	387
4.5	EXTERNAL SUBPROGRAM CALL (M198).....	391
4.6	MANUAL HANDLE INTERRUPTION	393
4.7	MIRROR IMAGE.....	399
5	TEST OPERATION	401
5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK	401
5.2	FEEDRATE OVERRIDE	402
5.3	RAPID TRAVERSE OVERRIDE	403
5.4	DRY RUN	404
5.5	SINGLE BLOCK	405
5.6	PUNCH	406
5.7	MANUAL PUNCH	407
6	SAFETY FUNCTIONS.....	408
6.1	EMERGENCY STOP	408
6.2	OVERTRAVEL.....	408
6.3	STORED STROKE CHECK.....	409
6.4	STROKE LIMIT CHECK BEFORE MOVE	412
6.5	WRONG OPERATION PREVENTION FUNCTIONS	414
6.5.1	Functions that are Used When Data is Set	415
6.5.1.1	Input data range check.....	415
6.5.1.2	Confirmation of incremental input	416
6.5.1.3	Prohibition of the absolute input by the soft key	417
6.5.1.4	Confirmation of the deletion of the program.....	417
6.5.1.5	Confirmation of the deletion of all data.....	418
6.5.1.6	Confirmation of a data update during the data setting process	418
6.5.2	Functions that are Used when the Program is Executed	418
6.5.2.1	Display of updated modal information	419
6.5.2.2	Start check signal.....	419
6.5.2.3	Axis status display	420
6.5.2.4	Confirmation of the start from a middle block	420
6.5.2.5	Data range check	421
6.5.2.6	Maximum incremental value check.....	421
6.5.2.7	Warning display during a reset in program operation	422
6.5.3	Setting Screen.....	422
6.5.3.1	Operation confirmation function setting screen.....	423
6.5.3.2	Tool offset range setting screen	424
6.5.3.3	Workpiece origin offset range setting screen	426
6.6	SAFETY ZONE CHECK	427
6.6.1	Punch Forbidden Area and Approach Forbidden Area (Type A)	428
6.6.2	Punch Forbidden Area and Approach Forbidden Area (Type B).....	429
6.6.3	Setting the Safety Zone	430

6.6.4	Setting the Tool Shape Area.....	430
6.6.5	Automatic Setting of the Safety Zone	431
6.6.6	Displaying the Safety Zones and Tool Zone	432
7	ALARM AND SELF-DIAGNOSIS FUNCTIONS.....	434
7.1	ALARM DISPLAY	434
7.2	ALARM HISTORY DISPLAY	436
7.3	CHECKING BY DIAGNOSTIC DISPLAY	437
7.4	RETURN FROM THE ALARM SCREEN	438
8	DATA INPUT/OUTPUT	440
8.1	OVERWRITING FILES ON A MEMORY CARD	441
8.2	INPUT/OUTPUT ON EACH SCREEN	443
8.2.1	Inputting and Outputting a Program.....	443
8.2.1.1	Inputting a program	443
8.2.1.2	Outputting a program.....	444
8.2.2	Inputting and Outputting Parameters.....	445
8.2.2.1	Inputting parameters	445
8.2.2.2	Outputting parameters	445
8.2.3	Inputting and Outputting Offset Data.....	446
8.2.3.1	Inputting offset data.....	446
8.2.3.2	Outputting offset data	447
8.2.4	Inputting and Outputting Pitch Error Compensation Data	447
8.2.4.1	Inputting pitch error compensation data	447
8.2.4.2	Outputting pitch error compensation data	448
8.2.4.3	Input/output format of pitch error compensation data	448
8.2.5	Inputting and Outputting Custom Macro Common Variables	449
8.2.5.1	Inputting custom macro common variables	449
8.2.5.2	Outputting custom macro common variables	450
8.2.6	Inputting and Outputting Workpiece Coordinates System Data	451
8.2.6.1	Inputting workpiece coordinate system data.....	451
8.2.6.2	Outputting workpiece coordinate system data.....	452
8.2.7	Inputting and Outputting Operation History Data.....	452
8.2.7.1	Outputting operation history data	452
8.2.8	Inputting Outputting Tool Data.....	453
8.2.8.1	Inputting tool data.....	453
8.2.8.2	Outputting tool data	453
8.2.8.3	Output format of tool data	454
8.3	INPUT/OUTPUT ON THE ALL IO SCREEN.....	454
8.3.1	Inputting/Outputting a Program	455
8.3.2	Inputting and Outputting Parameters.....	456
8.3.3	Inputting and Outputting Offset Data.....	457
8.3.4	Inputting/Outputting Pitch Error Compensation Data.....	458
8.3.5	Inputting/Outputting Custom Macro Common Variables	459
8.3.6	Inputting and Outputting Workpiece Coordinates System Data	460
8.3.7	Inputting and Outputting Operation History Data.....	460
8.3.8	File Format	461
8.4	MEMORY CARD SCREEN.....	461
8.4.1	Displaying the Memory Card Screen	461
8.4.2	Displaying and Operating the File List	462
8.4.3	Inputting/Outputting a File.....	463
8.5	EMBEDDED ETHERNET OPERATIONS.....	465
8.5.1	FTP File Transfer Function	465
8.6	FLOPPY CASSETTE SCREEN.....	469

8.6.1	Displaying the Floppy Cassette Screen	469
8.6.2	Displaying and Operating the File List	469
8.6.3	Inputting/Outputting a File	470
8.7	SCREEN HARD COPY FUNCTION	471
9	CREATING PROGRAMS	473
9.1	CREATING PROGRAMS USING THE MDI PANEL	473
9.2	AUTOMATIC INSERTION OF SEQUENCE NUMBERS	474
9.3	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION	475
10	EDITING PROGRAMS	480
10.1	EDIT DISABLE ATTRIBUTE	480
10.2	INSERTING, ALTERING AND DELETING A WORD	481
10.2.1	Word Search	482
10.2.2	Heading a Program	483
10.2.3	Inserting a Word	484
10.2.4	Altering a Word	485
10.2.5	Deleting a Word	485
10.3	DELETING BLOCKS	486
10.3.1	Deleting a Block	486
10.3.2	Deleting Multiple Blocks	486
10.4	PROGRAM SEARCH	487
10.5	SEQUENCE NUMBER SEARCH	488
10.6	DELETING PROGRAMS	490
10.6.1	Deleting One Program	490
10.6.2	Deleting All Programs	490
10.7	COPYING/MOVING PROGRAMS	491
10.7.1	Copying a Part of a Program	491
10.7.2	Moving a Part of a Program	494
10.7.3	Copying an Entire Program	497
10.7.4	Moving an Entire Program	499
10.7.5	Copy Specifying a Program Number	501
10.7.6	Copying/Moving to the Key-in Buffer	502
10.8	REPLACING	503
10.9	EDITING OF CUSTOM MACROS	503
10.10	PASSWORD FUNCTION	504
10.11	COMPACT-TYPE MDI KEY INPUT	505
11	PROGRAM MANAGEMENT	508
11.1	SELECTING A DEVICE	508
11.1.1	Selecting a Memory Card Program as a Device	509
11.1.2	Selecting a From Cassette as a Device	512
11.2	DELETING A PROGRAM	513
11.3	CHANGING PROGRAM ATTRIBUTES	513
11.4	SELECTING A MAIN PROGRAM	514
11.5	MAKING A PROGRAM COMPACT	514
12	SETTING AND DISPLAYING DATA	516
12.1	SCREENS DISPLAYED BY FUNCTION KEY 	526
12.1.1	Position Display in the Workpiece Coordinate System	526
12.1.2	Position Display in the Relative Coordinate System	527

12.1.3	Overall Position Display	529
12.1.4	Workpiece Coordinate System Preset	530
12.1.5	Actual Feedrate Display	531
12.1.6	Display of Run Time and Parts Count.....	533
12.1.7	Operating Monitor Display	534
12.2	SCREENS DISPLAYED BY FUNCTION KEY 	535
12.2.1	Program Contents Display.....	535
12.2.2	Editing a Program.....	536
12.2.3	Program Screen for MDI Operation	538
12.2.4	Program List Screen	538
12.2.5	Next Block Display Screen	539
12.2.6	Program Check Screen	540
12.2.7	Current Block Display Screen (Only for the 8.4-Inch Display Unit).....	541
12.2.8	Graphical Conversational Programming Screen	541
12.2.9	Background Editing.....	543
12.3	SCREENS DISPLAYED BY FUNCTION KEY 	547
12.3.1	Setting and Displaying the Tool Compensation Value	547
12.3.2	Displaying and Entering Setting Data	549
12.3.3	Sequence Number Comparison and Stop	551
12.3.4	Displaying and Setting Run Time, Parts Count, and Time	553
12.3.5	Displaying and Setting the Workpiece Origin Offset Value	555
12.3.6	Direct Input of Workpiece Origin Offset Value Measured	555
12.3.7	Displaying and Setting Custom Macro Common Variables.....	557
12.3.8	Displaying and Setting the Software Operator's Panel	558
12.3.9	Displaying and Switching the Display Language	561
12.3.10	Protection of Data at Eight Levels.....	562
	12.3.10.1 Operation level setting.....	562
	12.3.10.2 Password modification.....	564
	12.3.10.3 Protection level setting	565
12.3.11	Displaying and Setting Items on the Tool Registration Screens	567
	12.3.11.1 Displaying and setting items on the initial tool registration screen	567
	12.3.11.2 Displaying and setting items on the tool number registration screen	569
	12.3.11.3 Displaying and setting items on the screen for the number of press operations.....	570
	12.3.11.4 Displaying and setting items on the tool figure registration screen (for drawing figures).....	572
12.3.12	Displaying and Setting Items on the Tool Registration Screen for Multiple Tools.....	574
	12.3.12.1 Displaying and setting items on the tool number registration screen for multiple tools	574
	12.3.12.2 Displaying and setting items on the tool figure registration screen for multiple tools (for drawing figures).....	575
12.3.13	Displaying and Setting Items on the Safety Zone Setting Screen	578
12.4	SCREENS DISPLAYED BY FUNCTION KEY 	582
12.4.1	Displaying and Setting Parameters.....	582
12.4.2	Displaying and Setting Pitch Error Compensation Data	584
12.4.3	Servo Setting	587
12.4.4	Servo Tuning	590
12.4.5	Color Setting Screen.....	591
12.4.6	Parameter Setting Support Screen.....	593
	12.4.6.1 Displaying the menu screen and selecting a menu item	593
	12.4.6.2 Displaying and setting the axis setting screen	596
	12.4.6.3 Displaying and setting the FSSB amplifier setting screen.....	597

12.4.6.4	Displaying and setting the FSSB axis setting screen	598
12.4.6.5	Displaying and setting the servo setting screen	599
12.4.6.6	Displaying and setting the servo setting screen	599
12.4.6.7	Displaying and setting the servo gain tuning screen.....	601
12.4.6.8	Displaying and setting the high-precision setting screen.....	611
12.4.6.9	Displaying and setting the miscellaneous setting screen	613
12.4.6.10	Displaying and setting the servo tuning screen	613
12.4.7	Periodic Maintenance Screen	615
12.4.8	System Configuration Screen	622
12.4.9	Overview of the History Function.....	624
12.4.9.1	Operation history	624
12.4.9.2	Selecting operation history signals	629
12.4.9.3	Outputting all history data	630
12.5	SCREENS DISPLAYED BY FUNCTION KEY 	633
12.6	DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION	633
12.6.1	Displaying the Program Number, and Sequence Number.....	633
12.6.2	Displaying the Status and Warning for Data Setting or Input/Output Operation.....	634
12.7	SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION	636
13	GRAPHIC FUNCTION.....	638
13.1	OPERATION.....	639
13.2	REGISTERING THE TOOL FIGURE.....	639
13.3	SPECIFYING DRAWING PARAMETERS	640
13.4	GRAPHIC DISPLAY SCREEN AND DRAWING.....	644
13.5	EXAMPLE.....	647
14	VIRTUAL MDI KEY FUNCTION.....	650
14.1	VIRTUAL MDI KEY	650
14.1.1	Limitations.....	653
IV. MAINTENANCE		
1	ROUTINE MAINTENANCE	657
1.1	ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED	658
1.2	BACKING UP VARIOUS DATA ITEMS	659
1.3	METHOD OF REPLACING BATTERY	660
1.3.1	Replacing Battery for CNC Control Unit.....	660
1.3.2	Battery in the PANEL <i>i</i> (3 VDC)	664
1.3.3	Replacing Battery for Absolute Pulsecoders	665
1.3.3.1	Overview	665
1.3.3.2	Replacing batteries.....	666
1.3.3.3	Replacing the batteries in a separate battery case.....	666
1.3.3.4	Replacing the battery built into the servo amplifier.....	667
APPENDIX		
A	PARAMETERS.....	671
A.1	DESCRIPTION OF PARAMETERS.....	671
A.2	DATA TYPE.....	781

A.3	STANDARD PARAMETER SETTING TABLES.....	783
B	PROGRAM CODE LIST	784
C	LIST OF FUNCTIONS AND PROGRAM FORMAT	786
D	RANGE OF COMMAND VALUE.....	791
E	NOMOGRAPHS	793
E.1	TOOL PATH AT CORNER	793
E.2	RADIUS DIRECTION ERROR AT CIRCLE CUTTING	796
F	SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE.....	797
G	CHARACTER-TO-CODES CORRESPONDENCE TABLE	799
G.1	CHARACTER-TO-CODES CORRESPONDENCE TABLE.....	799
G.2	FANUC DOUBLE-BYTE CHARACTER CODE TABLE	800
H	ALARM LIST	806
I	PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING	831
I.1	PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING	831
I.1.1	Usage Notes.....	831
I.1.2	List of Functions of PC Tool.....	831
I.1.3	Explanation of Operations.....	832
I.2	NAMING RULES	840
I.2.1	Naming Rules of Program File.....	840
I.2.2	Naming Rules of Folder	841
I.3	RULES OF CHARACTERS IN PROGRAM FILE.....	841
I.3.1	Usable Characters in Program File.....	841
I.4	ERROR MESSAGE AND NOTE.....	843
I.4.1	List of Error Message	843
I.4.2	Note	844
J	ISO/ASCII CODE CONVERSION TOOL.....	845
K	DIFFERENCES FROM SERIES 0i-PC	848
K.1	CIRCULAR INTERPOLATION.....	849
K.1.1	Differences in Specifications.....	849
K.1.2	Differences in Diagnosis Display	849
K.2	HELICAL INTERPOLATION	850
K.2.1	Differences in Specifications.....	850
K.2.2	Differences in Diagnosis Display	850
K.3	SKIP FUNCTION	851
K.3.1	Differences in Specifications.....	851
K.3.2	Differences in Diagnosis Display	852
K.4	MANUAL REFERENCE POSITION RETURN.....	853
K.4.1	Differences in Specifications.....	853
K.4.2	Differences in Diagnosis Display	855
K.5	WORKPIECE COORDINATE SYSTEM	855
K.5.1	Differences in Specifications.....	855

	K.5.2	Differences in Diagnosis Display	855
K.6		LOCAL COORDINATE SYSTEM	856
	K.6.1	Differences in Specifications.....	856
	K.6.2	Differences in Diagnosis Display	857
K.7		TOOL COMPENSATION MEMORY	857
	K.7.1	Differences in Specifications.....	857
	K.7.2	Differences in Diagnosis Display	858
K.8		CUSTOM MACRO	858
	K.8.1	Differences in Specifications.....	858
	K.8.2	Differences in Diagnosis Display	860
	K.8.3	Miscellaneous	860
K.9		INTERRUPTION TYPE CUSTOM MACRO	860
	K.9.1	Differences in Specifications.....	860
	K.9.2	Differences in Diagnosis Display	860
K.10		PROGRAMMABLE PARAMETER INPUT (G10)	860
	K.10.1	Differences in Specifications.....	860
	K.10.2	Differences in Diagnosis Display	861
K.11		AI ADVANCED PREVIEW CONTROL	861
	K.11.1	Differences in Specifications.....	861
	K.11.2	Differences in Diagnosis Display	862
K.12		AXIS SYNCHRONOUS CONTROL	862
	K.12.1	Differences in Specifications.....	862
	K.12.2	Differences in Diagnosis Display	866
K.13		RUN HOUR AND PARTS COUNT DISPLAY	867
	K.13.1	Differences in Specifications.....	867
	K.13.2	Differences in Diagnosis Display	867
K.14		MANUAL HANDLE FEED	867
	K.14.1	Differences in Specifications.....	867
	K.14.2	Differences in Diagnosis Display	868
K.15		PMC AXIS CONTROL	868
	K.15.1	Differences in Specifications.....	868
	K.15.2	Differences in Diagnosis Display	872
K.16		EXTERNAL SUBPROGRAM CALL (M198).....	872
	K.16.1	Differences in Specifications.....	872
	K.16.2	Differences in Diagnosis Display	873
K.17		SEQUENCE NUMBER SEARCH	873
	K.17.1	Differences in Specifications.....	873
	K.17.2	Differences in Diagnosis Display	873
K.18		STORED STROKE CHECK.....	874
	K.18.1	Differences in Specifications.....	874
	K.18.2	Differences in Diagnosis Display	875
K.19		STORED PITCH ERROR COMPENSATION	876
	K.19.1	Differences in Specifications.....	876
	K.19.2	Differences in Diagnosis Display	876
K.20		SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION	877
	K.20.1	Differences in Specifications.....	877
	K.20.2	Differences in Diagnosis Display	877
K.21		RESET AND REWIND	878
	K.21.1	Differences in Specifications.....	878
	K.21.2	Differences in Diagnosis Display	878

K.22	MANUAL ABSOLUTE ON	878
	K.22.1 Differences in Specifications.....	878
	K.22.2 Differences in Diagnosis Display	879
K.23	EXTERNAL DATA INPUT	879
	K.23.1 Differences in Specifications.....	879
	K.23.2 Differences in Diagnosis Display	881
K.24	DATA SERVER FUNCTION	881
	K.24.1 Differences in Specifications.....	881
	K.24.2 Differences in Diagnosis Display	881
K.25	POWER MATE CNC MANAGER	882
	K.25.1 Differences in Specifications.....	882
	K.25.2 Differences in Diagnosis Display	882
K.26	CUTTER COMPENSATION	882
	K.26.1 Differences in Specifications.....	882
	K.26.2 Differences in Diagnosis Display	886
K.27	RAPID TRAVERSE OVERRIDE	887
	K.27.1 Differences in Specifications.....	887
	K.27.2 Differences in Diagnosis Display	887
K.28	POSITIONING BY OPTIMUM ACCELERATIONS	887
	K.28.1 Differences in Specifications.....	887
	K.28.2 Differences in Diagnosis Display	888
K.29	CONSTANT CONTROL OF POSITIONING TIME.....	888
	K.29.1 Differences in Specifications.....	888
	K.29.2 Differences in Diagnosis Display	889

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 does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-64310EN, B-64560EN). 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.
- 2 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 model indicated in the table below.
In the text, the abbreviations indicated below may be used.

Model name	Abbreviation	
FANUC Series 0i -PD	0i -PD	Series 0i -PD

Special symbols

This manual uses the following symbols:

- 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 0i-PD

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

Table 1 Related manuals

Manual name	Specification number	
FANUC Series 0i-D / 0i Mate-D DESCRIPTIONS	B-64302EN	
FANUC Series 0i-D / 0i Mate-D CONNECTION MANUAL (HARDWARE)	B-64303EN	
FANUC Series 0i-D / 0i Mate-D CONNECTION MANUAL (FUNCTION)	B-64303EN-1	
FANUC Series 0i-PD CONNECTION MANUAL (FUNCTION)	B-64553EN	
FANUC Series 0i-PD OPERATOR'S MANUAL	B-64554EN	*
FANUC Series 0i-D / 0i Mate-D MAINTENANCE MANUAL	B-64305EN	
FANUC Series 0i-D / 0i Mate-D PARAMETER MANUAL	B-64310EN	
FANUC Series 0i-PD PARAMETER MANUAL	B-64560EN	
Programming		
Macro Executor PROGRAMMING MANUAL	B-64303EN-2	
Macro Compiler PROGRAMMING MANUAL	B-64303EN-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	

Related manuals of SERVO MOTOR $\alpha i/\beta i$ series

The following table lists the manuals related to SERVO MOTOR $\alpha i/\beta i$ series

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR αi series DESCRIPTIONS	B-65262EN
FANUC AC SERVO MOTOR βi series DESCRIPTIONS	B-65302EN
FANUC SERVO AMPLIFIER αi series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER βi series DESCRIPTIONS	B-65322EN
FANUC AC SERVO MOTOR αis series FANUC AC SERVO MOTOR αi series FANUC AC SPINDLE MOTOR αi series FANUC SERVO AMPLIFIER αi series MAINTENANCE MANUAL	B-65285EN
FANUC AC SERVO MOTOR βis series FANUC AC SPINDLE MOTOR βi series FANUC SERVO AMPLIFIER βi series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR αi series FANUC AC SERVO MOTOR βi series FANUC LINEAR MOTOR LiS series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR DiS series PARAMETER MANUAL	B-65270EN

CNCs that are described in this manual can be connected to above servo motors.

This manual mainly assumes that the FANUC SERVO MOTOR αi series of servo motor is used. For servo motor information, refer to the manuals for the servo motor 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.

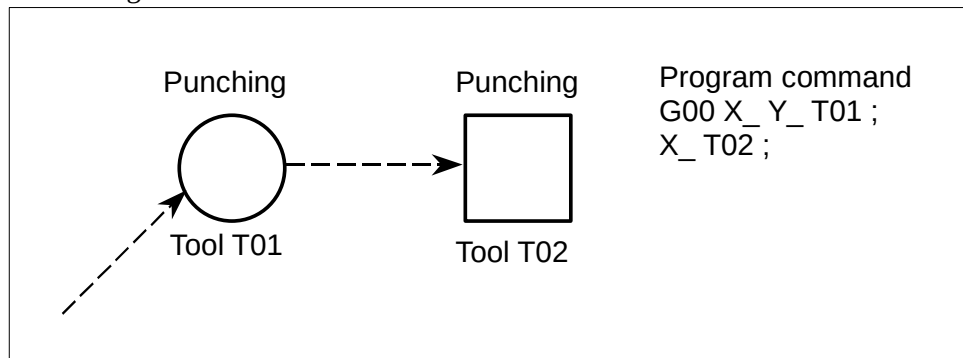
II. PROGRAMMING

1 GENERAL

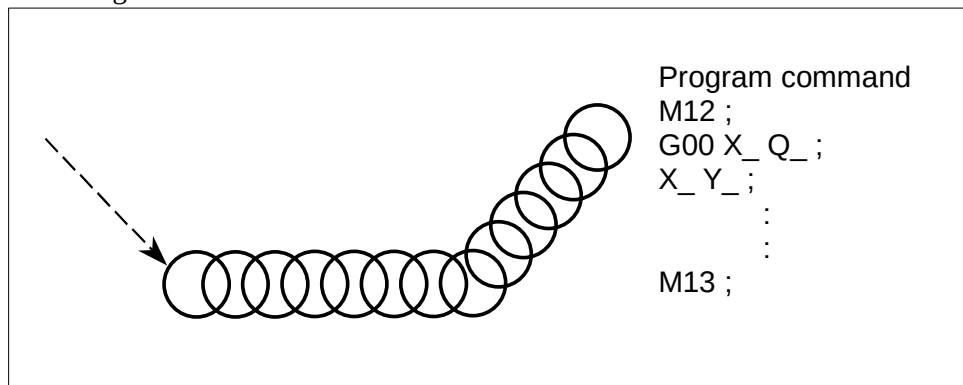
Chapter 1, "GENERAL", consists of the following sections:

1.1	TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION	10
1.2	FEED-FEED FUNCTION	12
1.3	PART DRAWING AND TOOL MOVEMENT	12
1.4	SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION	16
1.5	COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION	16
1.6	PROGRAM CONFIGURATION	17
1.7	TOOL MOVEMENT RANGE - STROKE	19

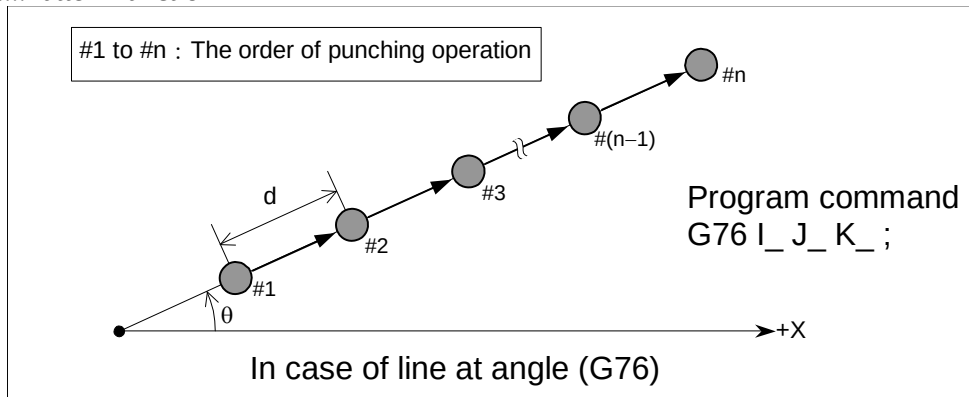
- (1) Punching is performed after positioning
Punching function



- (2) Continuous, repetitive punching can be performed without halting the pressing process after positioning
Nibbling function



- (3) By giving commands for block, it is possible to perform at multiple positions in a given profile.
Pattern function



This CNC supports the eight different patterns that will be used most frequently.

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION

The tool moves along straight lines and arcs constituting the workpiece parts figure (See Chapter, "INTERPOLATION FUNCTIONS").

Explanation

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

- Tool movement along a straight line

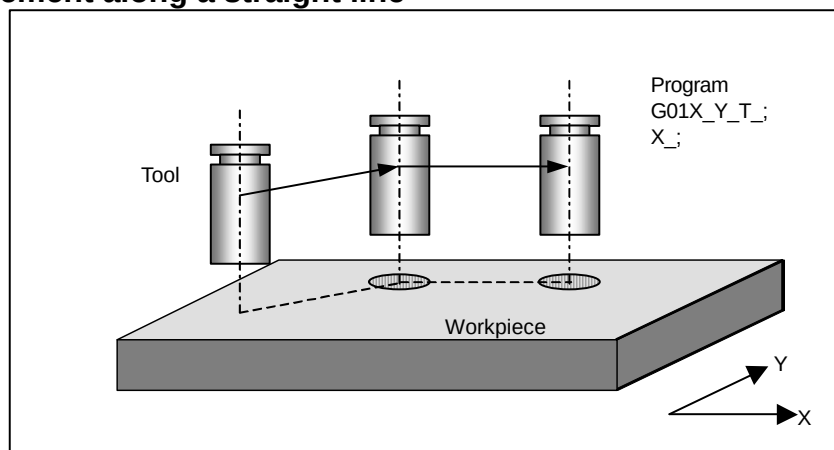


Fig. 1.1 (a) Tool movement along a straight line

- Tool movement along an arc

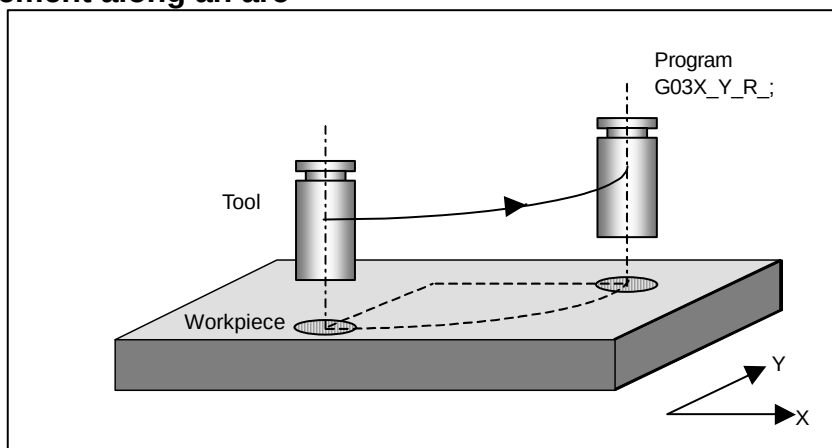


Fig. 1.1 (b) Tool movement along an arc

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.

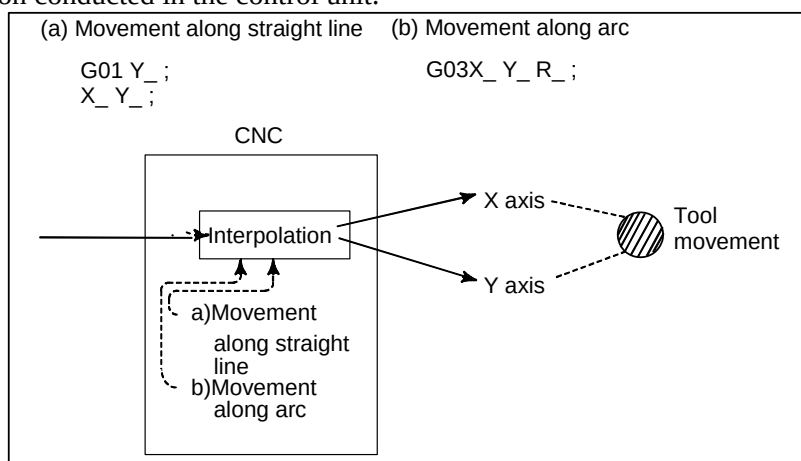


Fig. 1.1 (c) Interpolation function

NOTE

Some machines move tables 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 punching a workpiece is called the feed.

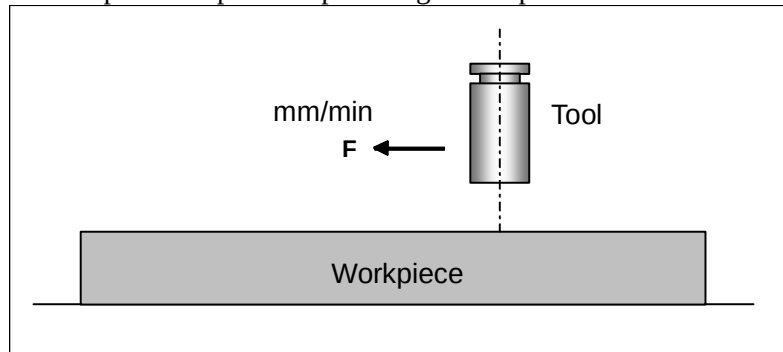


Fig. 1.2 (a) Feed function

For example, to feed the tool at a rate of 150 mm/min (feed per minute), specify the following in the program:

F150.0

The function of deciding the feed rate is called the feed function (See Chapter, “FEED FUNCTIONS”).

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.

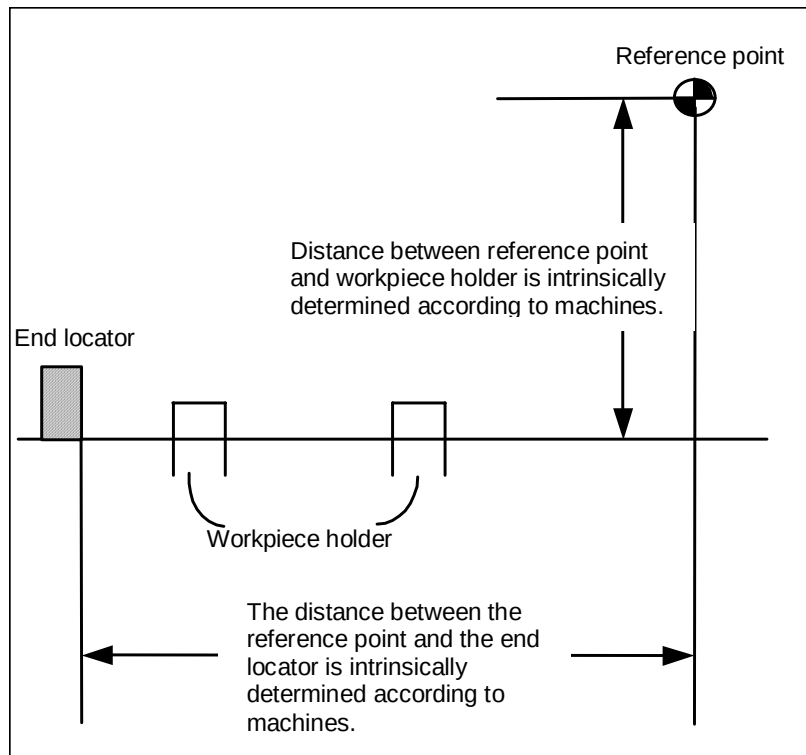


Fig. 1.3.1 (a) Reference position

Explanation

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

1. Manual reference position return (See Section, "MANUAL REFERENCE POSITION RETURN")
Reference position return is performed by manual button operation.
2. Automatic reference position return (See Section, "REFERENCE POSITION RETURN")

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

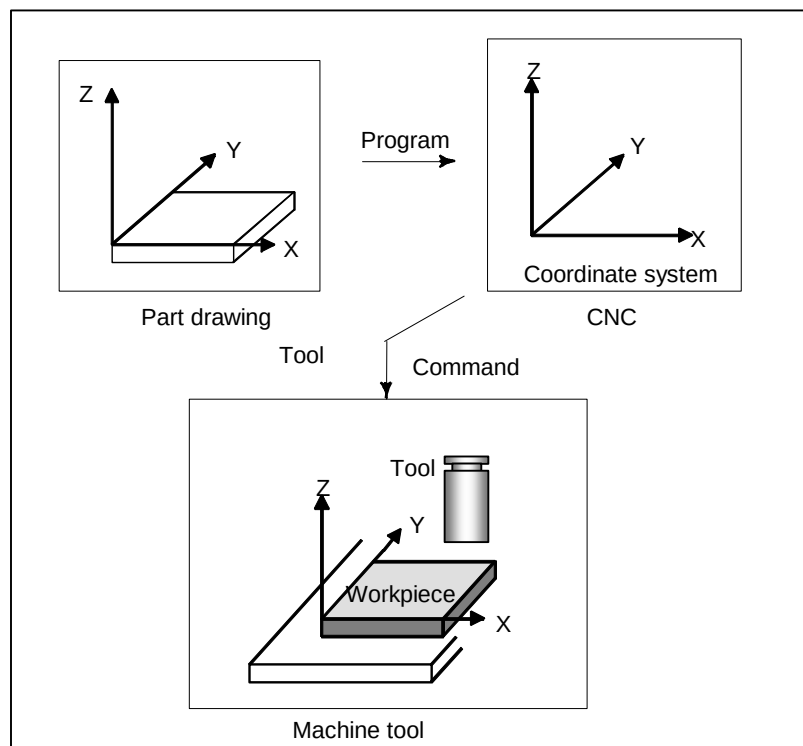


Fig. 1.3.2 (a) Coordinate system

Explanation

- Coordinate system

The following two coordinate systems are specified at different locations:
(See Chapter, "COORDINATE SYSTEM")

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.

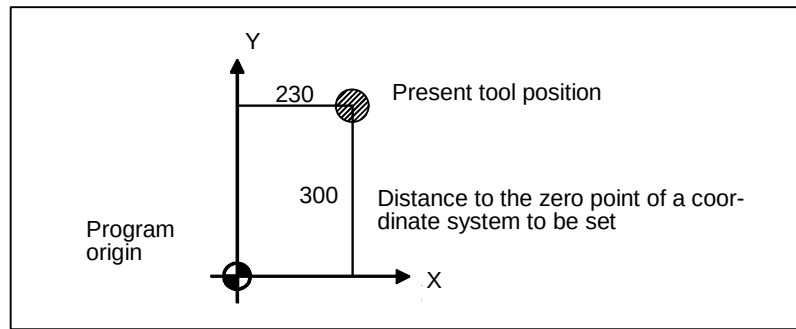


Fig. 1.3.2 (b) Coordinate system specified by the CNC

Concrete programming methods for setting coordinate systems specified by the CNC are explained in Chapter, “COORDINATE SYSTEM”

The positional relation between these two coordinate systems is determined when a workpiece is set on the table.

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

When a workpiece is set on the table, these two coordinate systems lay as follows:

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 cut 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.

To set the two coordinate systems at the same position, when setting a workpiece to be machined to general turret punch press, the workpiece is held by the workpiece holders after positioning it by applying the end face of the workpiece to the end locator and workpiece holders mounted on the machine as illustrated below.

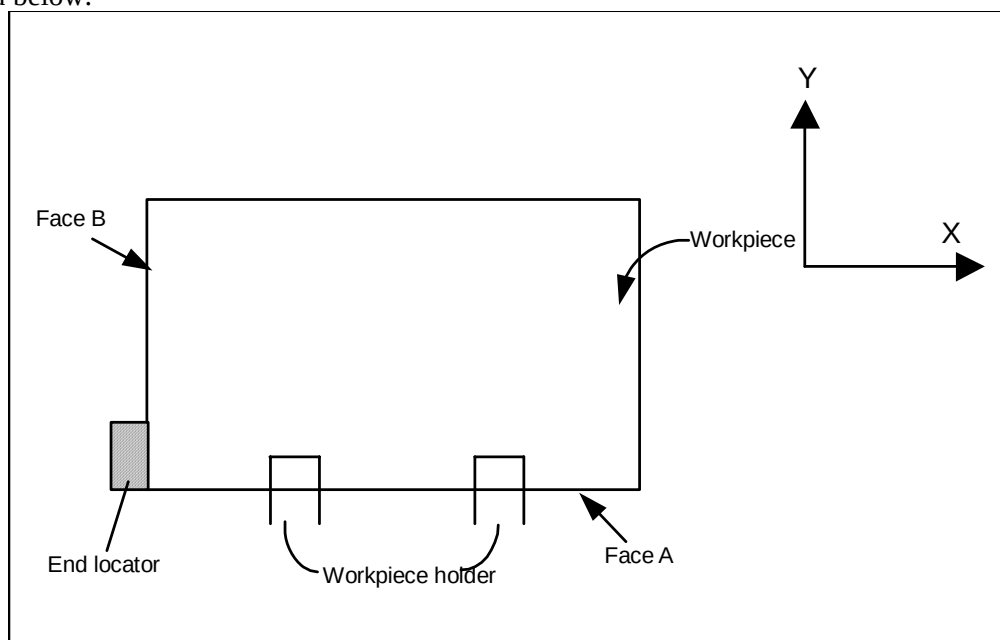


Fig. 1.3.2 (c)

Generally, the distance between the reference point and the locator as well as the distance between the reference point and the workpiece holders are intrinsically determined according to machines, and they are separated from each other by a fixed distance.

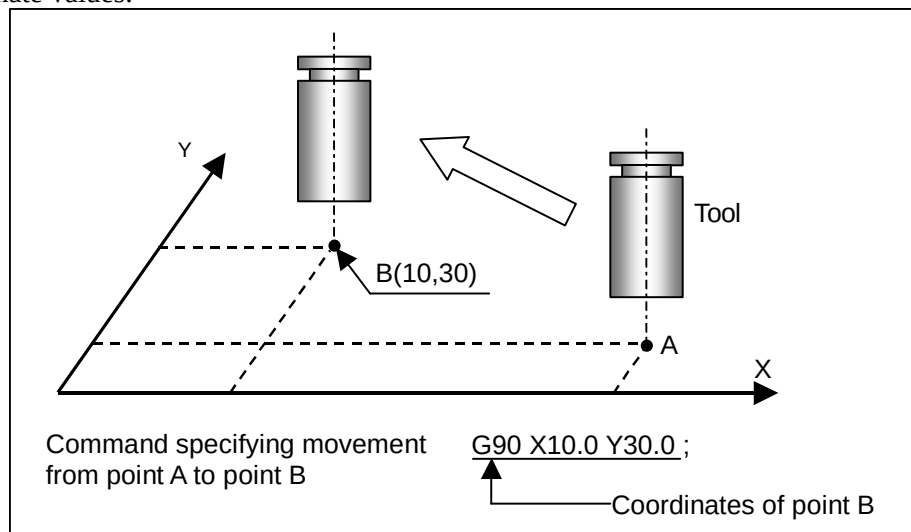
1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute and Incremental Programming)

Explanation

Command for moving the tool can be indicated by absolute command or incremental command (See Section, "ABSOLUTE AND INCREMENTAL PROGRAMMING").

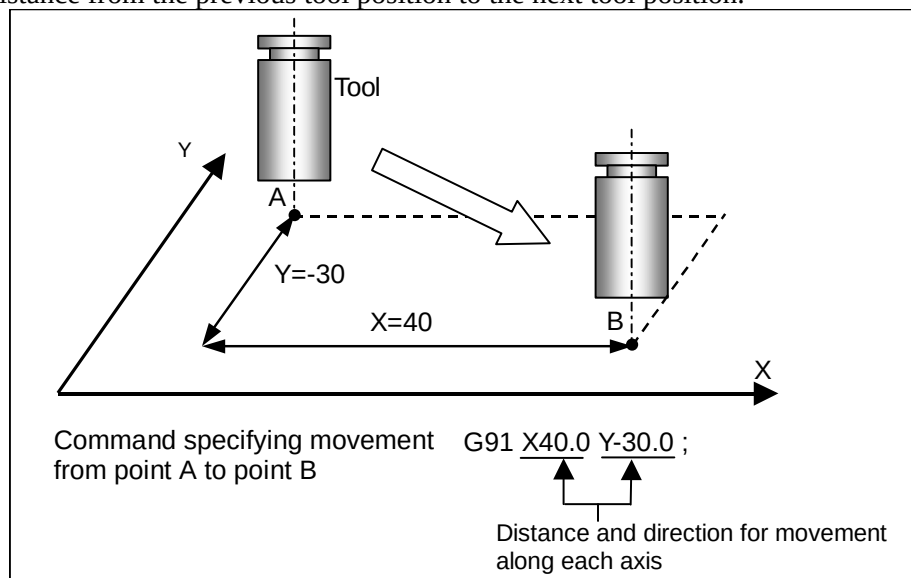
- Absolute command

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.



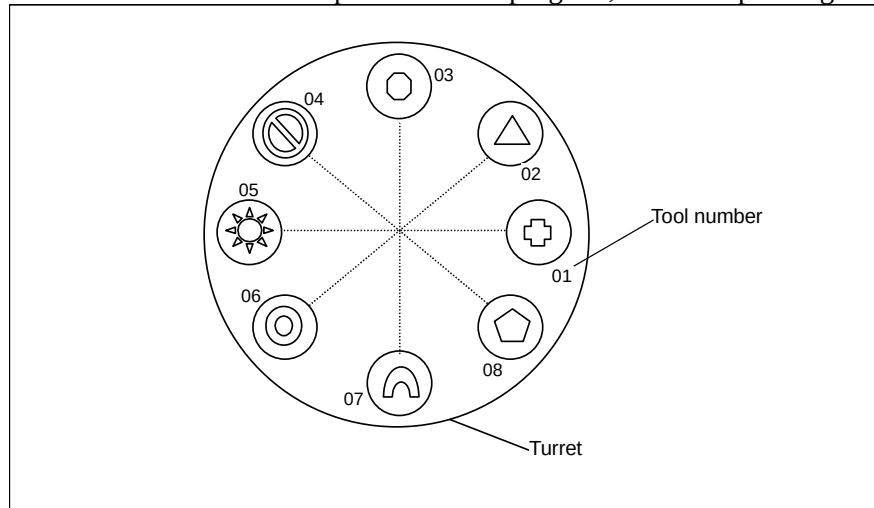
- Incremental command

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



1.4 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION

When drilling, tapping, or the like, is performed, it is necessary to select a suitable tool. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.



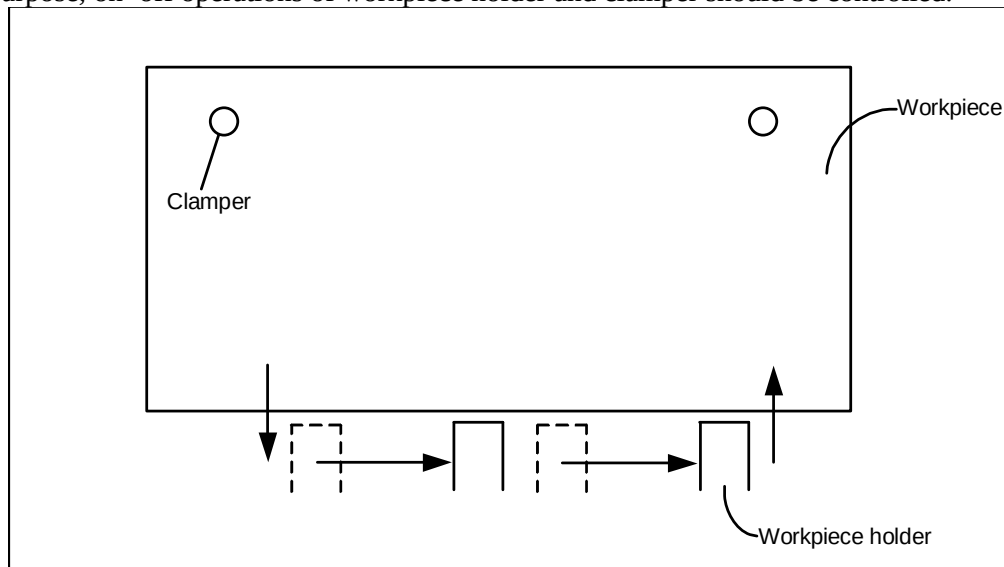
<When No.01 is assigned to a punching tool>

When the tool is stored at location 01 in the turret, the tool can be selected by specifying T01. This is called the tool function (See Chapter, "TOOL FUNCTION (T FUNCTION)").

1.5 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION

During machining, on-off operation of workpiece holder and clamber is performed.

For this purpose, on-off operations of workpiece holder and clamber should be controlled.



The function of specifying the on-off operations of the components of the machine is called the miscellaneous function. In general, the function is specified by and M code.

1.6 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.

In the program, specify the commands in the sequence of actual tool movements.

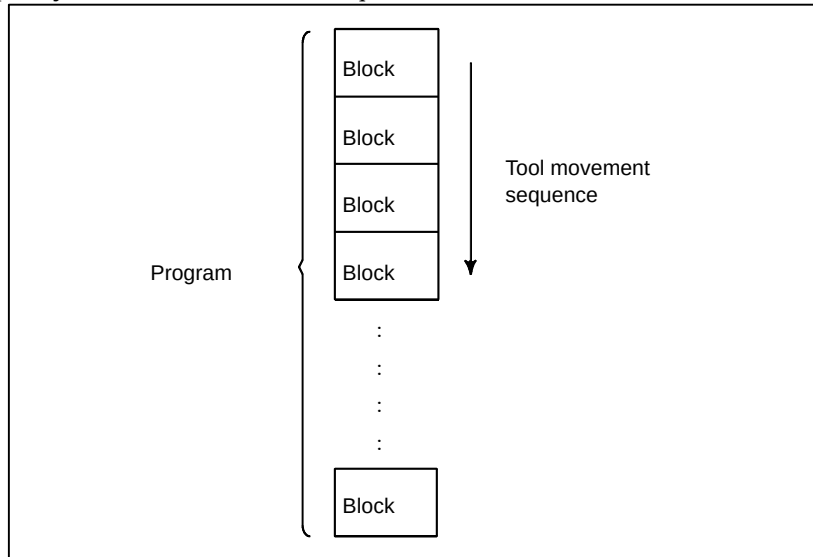


Fig. 1.6 (a) Program configuration

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 Chapter, "PROGRAM CONFIGURATION").

Explanation

The block and the program have the following configurations.

- Block

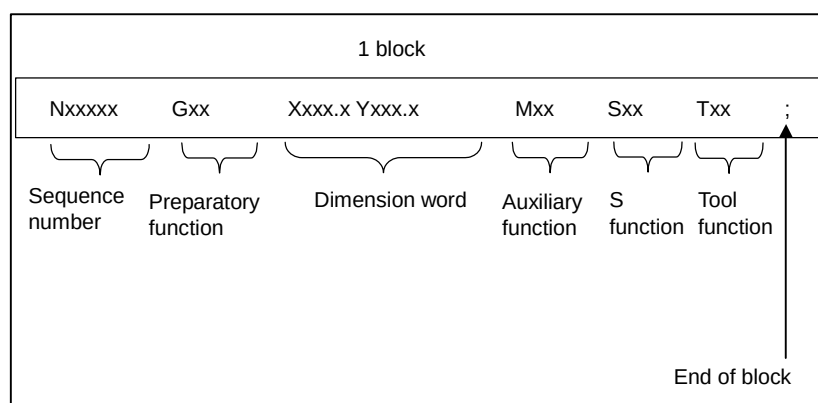


Fig. 1.6 (b) Block configuration

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

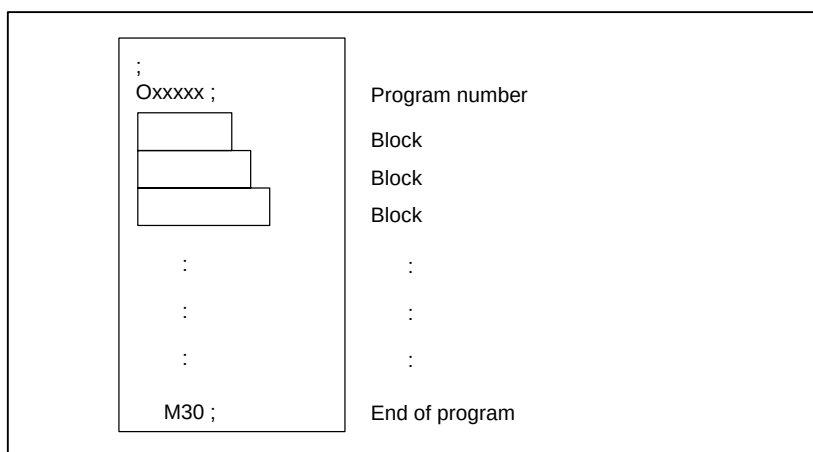


Fig. 1.6 (c) Program configuration

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.

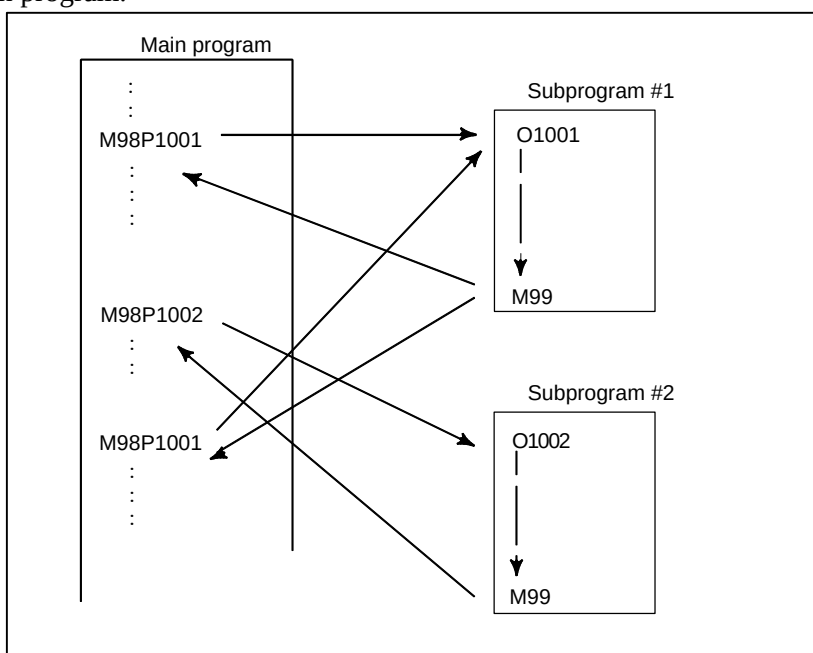
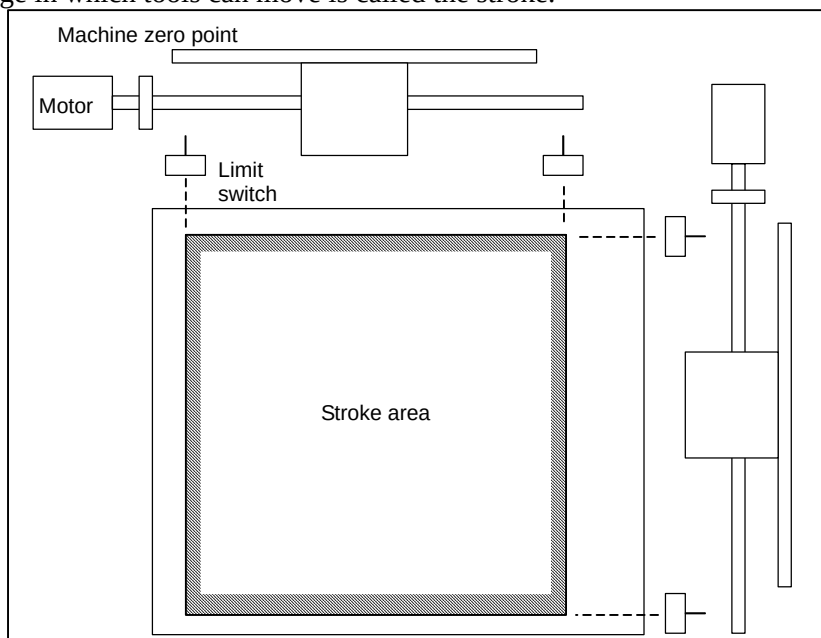


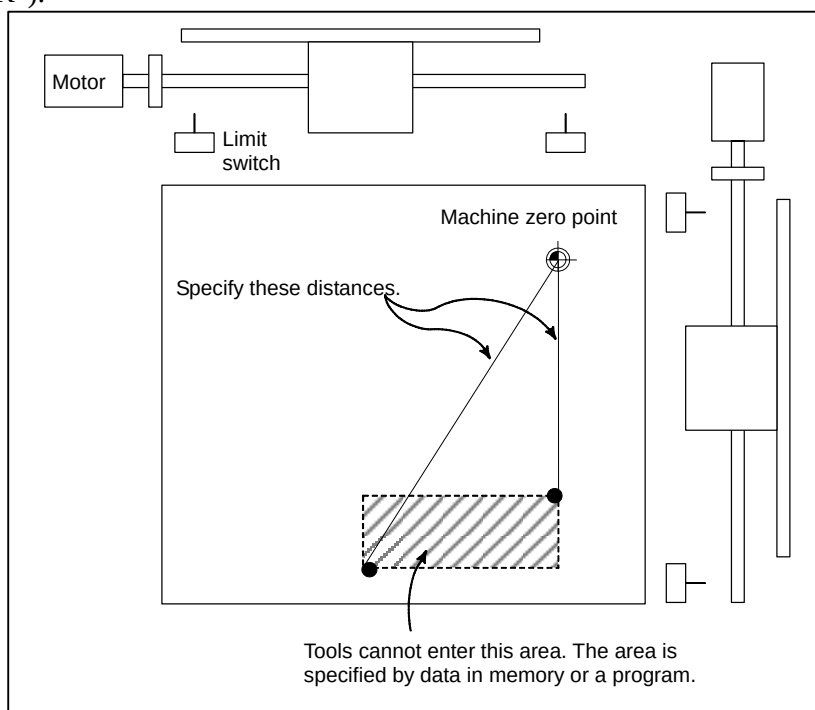
Fig. 1.6 (d) Subprogram execution

1.7 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 Section, "STORED STROKE CHECK").



2 CONTROLLED AXES

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

2.1	NUMBER OF CONTROLLED AXES.....	20
2.2	NAMES OF AXES	20
2.3	INCREMENT SYSTEM.....	20
2.4	MAXIMUM STROKE.....	21

2.1 NUMBER OF CONTROLLED AXES

Explanation

The number of controlled axes used with this NC system is as indicated below.

	Series 0i-PD
Controlled paths	1 path
Total number of controlled axes	Max. 7 axes
Feed axes	Max. 7 axes
Simultaneously controlled axes	Max. 4 axes
Axis control by PMC	Max. 4 axes at a time

NOTE

- 1 The maximum number of controlled axes that can be used is limited depending on 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.

Two 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 (ISA) 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

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the bit 0 (INM) of parameter No. 0100.

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, etc.). For the increment system, see the machine tool builder's manual.

2.4 MAXIMUM STROKE

Explanation

The maximum stroke controlled by this CNC is shown in the table below:

Maximum stroke = Least command increment × 999999999 (999999999 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

NOTE

The actual stroke depends on the machine tool.

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_ ;
    Y_ ;
    X_ ;
G00 Y_ ;
:
```

} G01 is effective in this range.

Explanation

- 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.
 - The modal G codes are placed in the states marked with  as indicated in Table3.1 (a).
 - G20 and G21 remain unchanged when the clear state is set at power-up or reset.
 - 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.
 - The user can select G00 or G01 by setting bit 0 (G01) of parameter No. 3402.
 - The user can select G90 or G91 by setting bit 3 (G91) of parameter No. 3402.
 - In the machining center system, the user can select G17, G18, or G19 by setting bits 1 (G18) and 2 (G19) of parameter No. 3401.
- G codes other than G10 and G11 are one-shot G codes.
- 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.
- 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.
- G codes are indicated by group.
- There are two G code systems in the FANUC Series 0i-PD (Table3.1 (a)). Select a G code system using bit 6 (GSB) of parameter No. 3401.

3.1 G CODE LIST

Table 3.1 (a) G code list

G code system		Group	Function
A	B		
G00	G00	01	Positioning (rapid traverse)
G01	G01		Linear interpolation (cutting feed)
G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03		Circular interpolation CCW or helical interpolation CCW
G04	G04		Dwell
G05.1	G05.1	00	AI advanced preview control (*1)
G05.4	G05.4		High-speed HRV current control on/off
G09	G09		Exact stop
G10	G10		Programmable data input
G11	G11		Programmable data input mode cancel
G17	G17	02	XpYp plane selection
G18	G18		ZpXp plane selection
G19	G19		YpZp plane selection
G20	G20	06	Input in inch
G21	G21		Input in mm
G22	G22	04	Stored stroke check function on
G23	G23		Stored stroke check function off
G26	G26	00	Bolt hole circle
G28	G50		Automatic reference point return
G32	G32		Automatic safety zone setting
G33	G33		Skip function
G38	G38		Bending compensation X
G39	G39	07	Bending compensation X
G40	G40		Tool radius compensation : cancel
G41	G41		Tool radius compensation : left
G42	G42	18	Tool radius compensation : right
G40.1(G150)	G40.1(G150)		Normal direction control cancel mode
G41.1(G151)	G41.1(G151)		Normal direction control on : left
G42.1(G152)	G42.1(G152)	00	Normal direction control on : right
G45	G45		Linear punching
G46	G46		Circular punching (CW)
G47	G47	11	Circular punching (CCW)
G50	G34		Scaling cancel
G51	G35	00	Scaling
G52	G93		Local coordinate system setting
G53	G53	14	Machine coordinate system setting
G54	G54		Workpiece coordinate system 1 selection
G55	G55		Workpiece coordinate system 2 selection
G56	G56		Workpiece coordinate system 3 selection
G57	G57		Workpiece coordinate system 4 selection
G58	G58		Workpiece coordinate system 5 selection
G59	G59		Workpiece coordinate system 6 selection
G61	G61	15	Exact stop mode
G62	G62		Automatic corner override
G64	G64	00	Cutting mode
G65	G95		Macro call
G66	G96	12	Macro modal call
G67	G97		Macro modal call cancel

3. PREPARATORY FUNCTION (G FUNCTION)

PROGRAMMING

B-64554EN/01

G code system		Group	Function
A	B		
G68	G68	00	Circular nibbling
G69	G69		Linear nibbling
G70	G70		Positioning & press off
G72	G72		Standard point command
G73	G75		Multi-piece machining command X
G74	G76		Multi-piece machining command Y
G75	G27		Automatic repositioning
G76	G28		Line at angle
G77	G29		Arc
G78	G36		Grid I
G79	G37		Grid II
G84	G84	16	Coordinate system rotation start
G85	G85		Coordinate system rotation cancel
G86	G66	00	Share proof
G87	G67		Square
G88	G78		Radius
G89	G79		Cut at angle
G90	G90	03	Absolute programming
G91	G91		Incremental programming
G91.1	G91.1	00	Checking the maximum incremental amount specified
G92	G92		Setting for workpiece coordinate system
G92.1	G92.1		Workpiece coordinate system preset
G98	G98		Coordinate system setting (Multi-piece machining)

*1: In punching mode, AI Advanced Preview Control I function cannot be used.

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).....	25
4.2	LINEAR INTERPOLATION (G01).....	26
4.3	CIRCULAR INTERPOLATION (G02, G03).....	28
4.4	HELICAL INTERPOLATION (G02, G03).....	32
4.5	SKIP FUNCTION (G33)	33
4.6	MULTI-STEP SKIP (G33)	35
4.7	HIGH-SPEED SKIP SIGNAL (G33)	36
4.8	SKIP POSITION MACRO VARIABLE IMPROVEMENT	36
4.9	TORQUE LIMIT SKIP	36

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

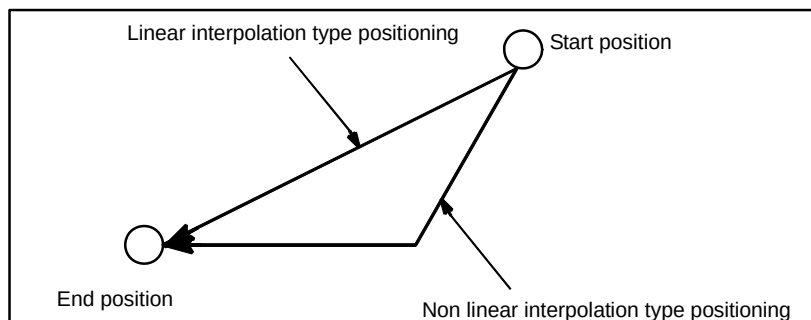
G00 IP_ ;

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.



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.

When G00X_Y_T ; is specified in a machine having a turret axis (T-axis), the X and Y axes move to the specified positions at rapid traverse rate and also the T-axis moves at the predetermined rapid traverse rate in such a way as to select a specified tool number.

In a machine provided with a die angle index (C-axis), if "G00X_Y_ ;" is specified, the X, Y, and C axes move simultaneously at the predetermined rapid traverse rate.

Refer to "C axis control" for the details.

Since this control system treats the turret punch press as a controlled system, the tool moves to the commanded position as fast as possible for punching as the basic principle.

Accordingly, the tool is positioned at rapid traverse, punching is done after axis movement in the G00 mode, in principle.

Refer to "Punch function" for details.

**CAUTION**

For T- or C-axis command blocks, nonlinear interpolation positioning is performed, even if linear interpolation positioning is specified.

And, for block including G28 or G53 command, nonlinear interpolation positioning is performed.

Please pay attention for the interference between tool and workpiece.

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 $\alpha\beta\gamma\zeta$ Ff ;

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 the straight line axis α (such as X, Y, or Z) and the rotating axis β (such as A, B, or C) are linearly interpolated, the feedrate is that in which the tangential feedrate in the α and β Cartesian coordinate system is commanded by F (mm/min).

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:

$$\frac{\sqrt{20^2 + 40^2}}{300} = 0.14907(mm)$$

The feedrate for the C axis is

$$\frac{40}{0.14907} = 268.3 \text{ deg/min}$$

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

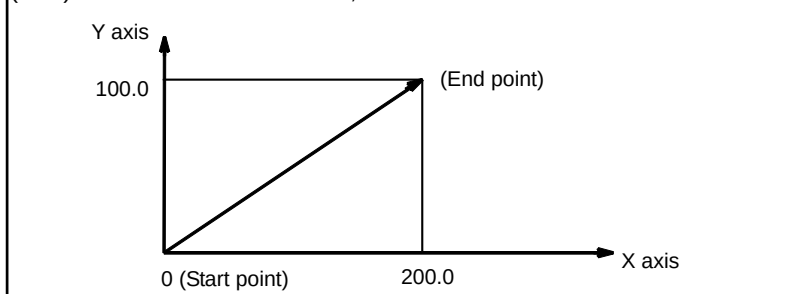
Limitations

- Punching (1-cycle pressing) is not performed in G01 mode. However, when bit 7 (CPF) of parameter No.16001 is set to 1, punching can be performed on the end point in G01 mode.
- T code can't be specified in G01 mode. If specified, an alarm (PS 4600) occurs. However, when T code is specified independently and bit 0 (NMG) of parameter No.16181 is set, an alarm does not occur.

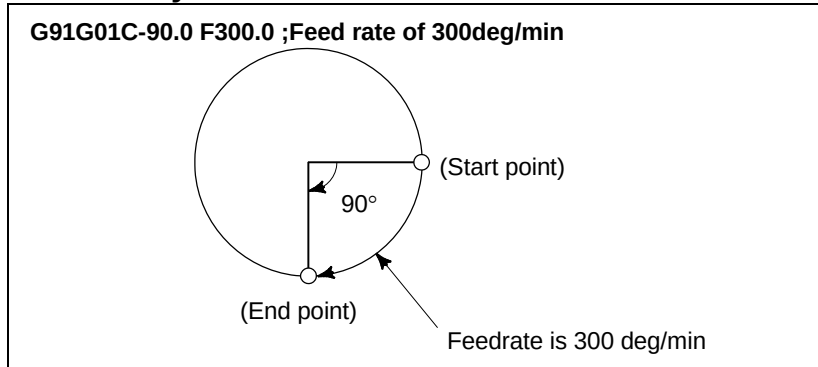
Example

- Linear interpolation

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



- **Feedrate for the rotary axis**



4.3 CIRCULAR INTERPOLATION (G02, G03)

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

Format

Arc in the XpYp plane

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

Arc in the ZpXp plane

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

Arc in the YpZp plane

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

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

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 (Fig. 4.3 (a)).

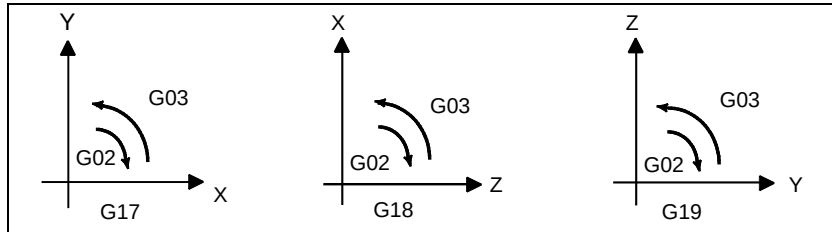


Fig. 4.3 (a)

- Distance moved on an arc

The end point of an arc is specified by address X_p , Y_p or Z_p , 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 X_p , Y_p , and Z_p 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 (Fig. 4.3 (b)).

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

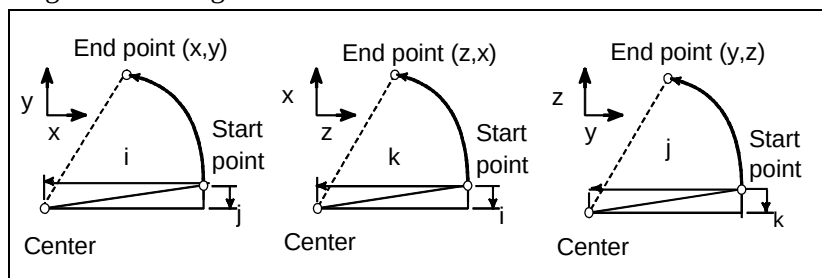


Fig. 4.3 (b)

I0, J0, and K0 can be omitted.

When X_p , Y_p , and Z_p 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.

G02I; Command for a circle

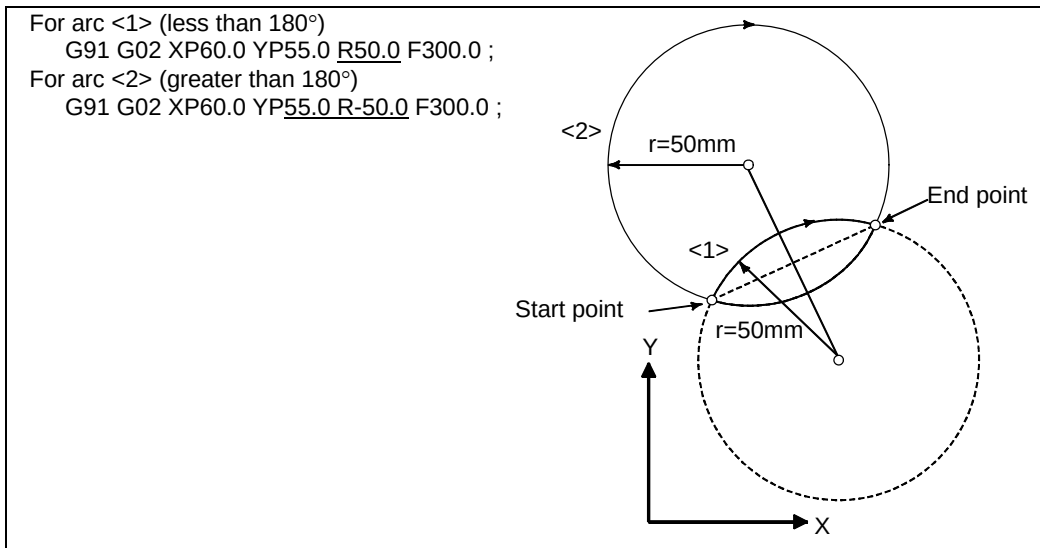
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (PS0020) occurs.

- 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. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If X_p , Y_p , and Z_p 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.)



- 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 error between the specified feedrate and the actual tool feedrate is $\pm 2\%$ or less. 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 PS0021 occurs.

For example, if the X-axis and a U-axis parallel to the X-axis are specified when the XY plane is specified

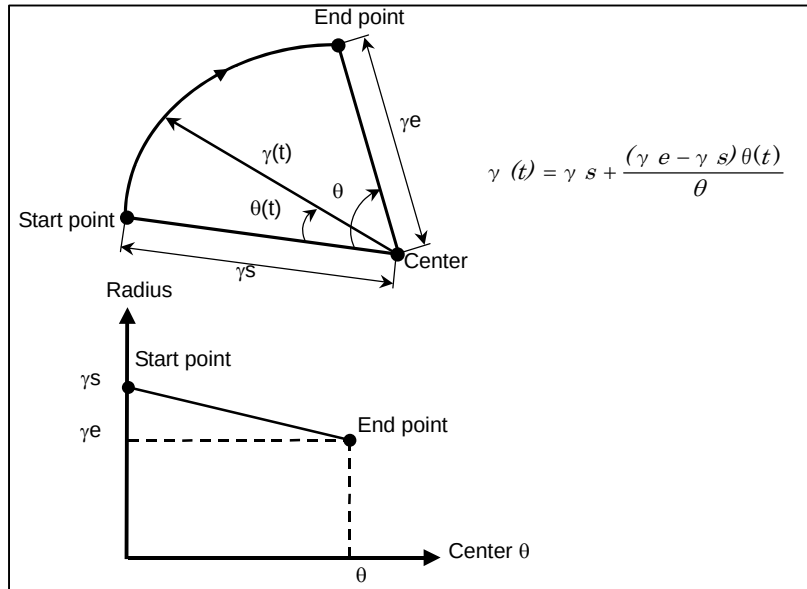
- Specifying a semicircle with R

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.



The arc radius changes linearly with the center angle $\theta(t)$. 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.

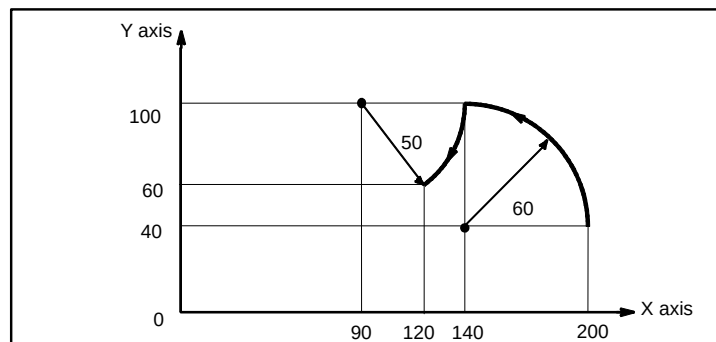
- Punching operation

Punching (1-cycle pressing) is not performed in G02 and G03 mode. However, when bit 7 (CPF) of parameter No.16001 is set to 1, punching can be performed on the end point in G02 or G03 mode.

- T code

If T command is specified in G02 and G03 mode, however, when T code is specified independently and bit 0 (NMG) of parameter No. 16181 is set, an alarm (PS 4600) doesn't occur.

Example



The above tool path can be programmed as follows;

- (1) In absolute programming
 G92X200.0 Y40.0 Z0 ;
 G90 G03 X140.0 Y100.0 R60.0 F300. ;
 G02 X120.0 Y60.0 R50.0 ;
 or
 G92X200.0 Y40.0 Z0 ;
 G90 G03 X140.0 Y100.0I-60.0 F300. ;
 G02 X120.0 Y60.0I-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 ;

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

G17 { G02 } Xp_ Yp_ { I_ J_ } $\alpha_$ ($\beta_$) F_ ;
 { G03 } { R_ }

Arc in the ZpXp plane

G18 { G02 } Zp_ Xp_ { K_ I_ } $\alpha_$ ($\beta_$) F_ ;
 { G03 } { R_ }

Arc in the YpZp plane

G19 { G02 } Yp_ Zp_ { J_ K_ } $\alpha_$ ($\beta_$) F_ ;
 { G03 } { R_ }

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

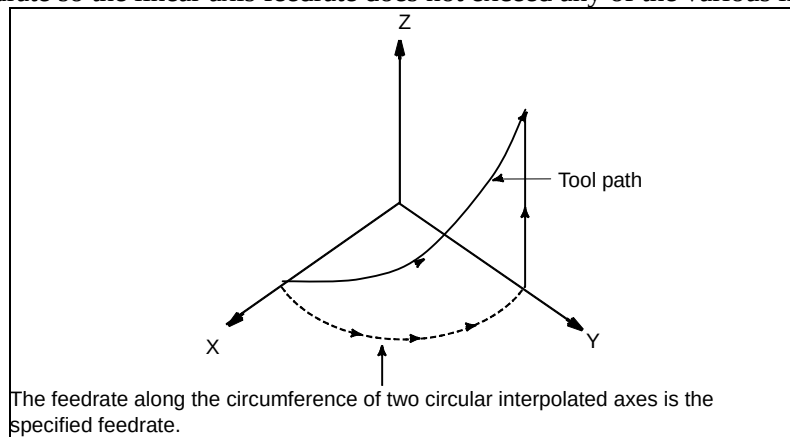
Explanation

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.

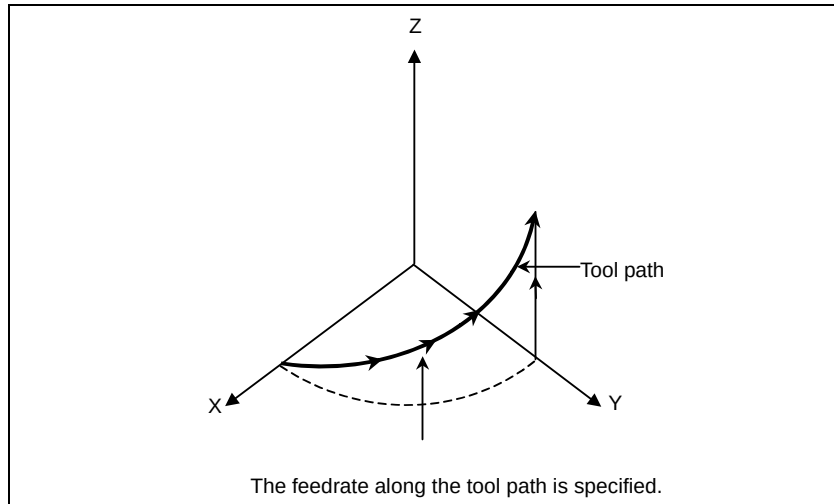


If HTG is set to 1, specify a feedrate along the tool path about the linear axis. Therefore, the tangential velocity of the arc is expressed as follows:

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

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

- Tool radius compensation is applied only for a circular arc.
- T-axis command cannot be specified in a block in which a helical interpolation is commanded.

4.5 SKIP FUNCTION (G33)

Linear interpolation can be commanded by specifying axial move following the G33 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. It is used also for measuring the dimensions of a workpiece.

Format

G33 IP ;

G33 : 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 #100151 to #100157, as follows. FS0i-PC compatible system variables (#5061 to #5067) may also be used.

#100151 (#5061) : First axis coordinate value
 #100152 (#5062) : Second axis coordinate value
 :
 #100157 (#5067) : 7th 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.), allowing for an error in the position of the tool when a skip signal is input.

NOTE

If G33 command is issued while tool radius compensation is applied, an alarm PS0035 is displayed. Cancel the tool radius compensation with the G40 command before the G33 command is specified.

Example

- The next block to G33 is an incremental programming

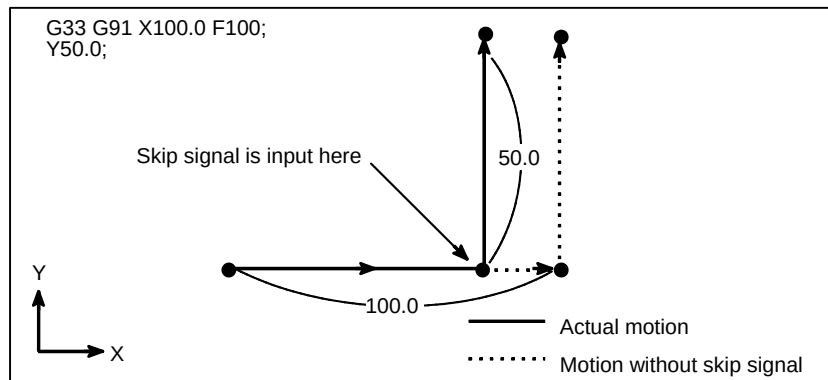


Fig. 4.5 (a) The next block is an incremental programming

- The next block to G33 is an absolute programming for 1 axis

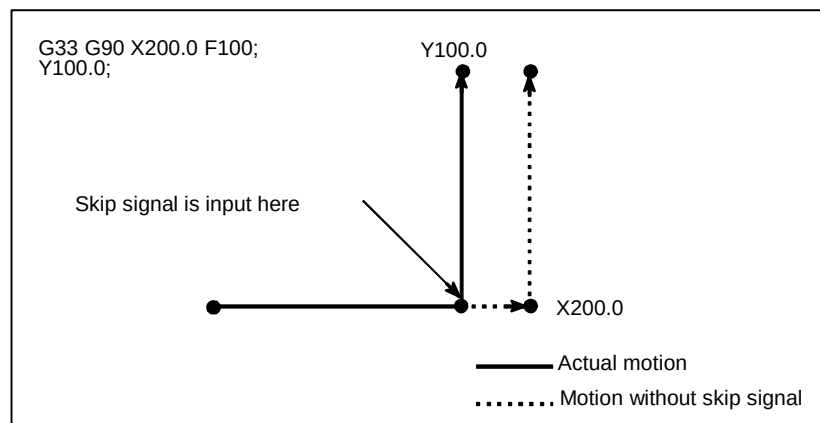


Fig. 4.5 (b) The next block is an absolute programming for 1 axis

- The next block to G33 is an absolute programming for 2 axes

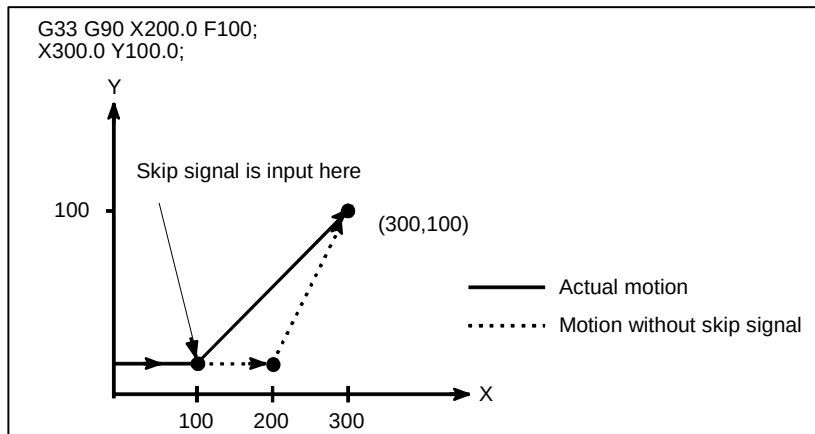


Fig. 4.5 (c) The next block is an absolute programming for 2 axes

4.6 MULTI-STEP SKIP (G33)

In a block specifying P1 to P4 after G33, the multi-step skip function stores coordinates in a custom macro variable when a skip signal (4-point or 8-point ; 8-point when a high-speed skip signal is used) is turned on. In the block where Q1 to Q4 are specified after G04, dwell can be skipped when skip signals (four or eight signals, or eight 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.

For the usage of this function, refer to the manual supplied by the machine tool builder.

Format

Move command

G33 IP_ F_ P_ ;

IP_ : End point

F_ : Feedrate

P_ : P1 to P4

Dwell

G04X(P)_ (Q_);

X(P)_ : Dwell time

Q_ : Q1 to Q4

Explanation

Multi-step skip is caused by specifying P1, P2, P3, or P4 in a G33 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 G33. 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 whether the 4-point or 8-point skip signal is used (when a high-speed skip signal is used). Specification is not limited to one-to-one correspondence. It is possible to specify that one skip signal correspond to two or more Pn'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.

4.7 HIGH-SPEED SKIP SIGNAL (G33)

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

G33 IP ;

G33; One-shot G code (If is effective only in the block in which it is specified)

4.8 SKIP POSITION MACRO VARIABLE IMPROVEMENT

Overview

The macro variables #100151 to #100157 (#5061 to #5067) for reading the skip position when executing the skip command (G33) is improved.

By setting bit 7 (SKM) of parameter No. 6007 to 1, it is possible to read the skip position in the workpiece coordinate system at the time a skip is made even if the workpiece coordinate system setting/selection command is executed after the skip command is executed.

By setting bit 7 (SKM) of parameter No. 6007 to 0, it is possible to read the skip position that reflects the workpiece coordinate system at the time it is read, as usual, if the workpiece coordinate system setting/selection command is executed after the skip command is executed.

Macro variable

The following are the macro variables that can read the position at the time a skip is made even if the workpiece coordinate system setting/selection command is executed after the skip command is executed, by setting bit 7 (SKM) of parameter No. 6007 to 1.

System variable number	System variable name	Attribute	Description
#5061 to #5067	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 7).
#100151 to #100157			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).
#5421 to #5427	[#_SKPDTC[n]]	R	Skip position (workpiece coordinate system, detection unit) Note) Subscript n represents an axis number (1 to 7).
#100701 to #100707			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).

4.9 TORQUE LIMIT SKIP

Overview

Executing the move command following G33P99 (or G33P98) 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:

- 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

G33 P98 α F

G33 P99 α F

G33 : 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

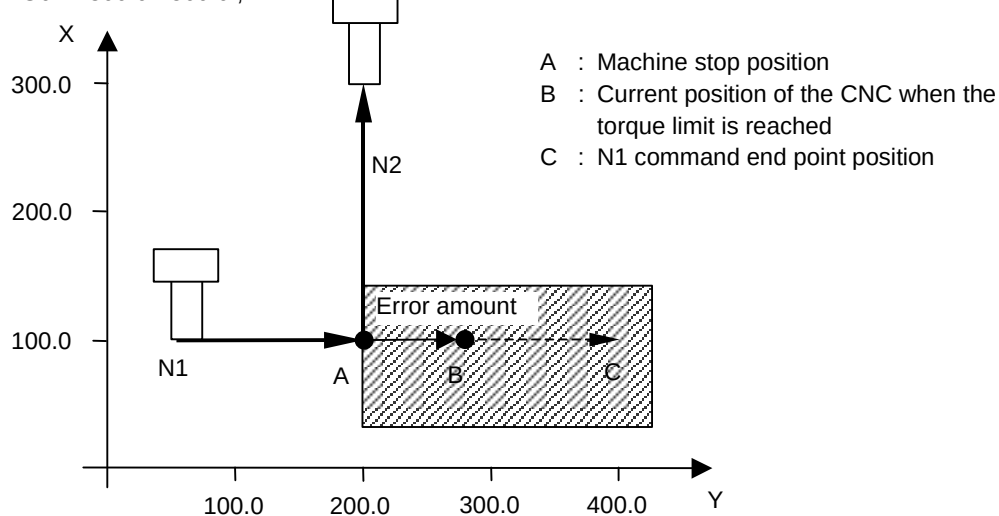
Condition	Command	
	G33P98	G33P99
The torque limit value is reached.	A skip operation is performed.	A skip operation is performed.
A skip signal is input.	No skip operation is performed.	A skip operation is performed.

- Operation during a torque limit skip

(Example)

N1 G33 P99 Y400.0 F100.0 ;

N2 G01 X300.0 F500.0 ;



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

If, with the torque limit skip command, no torque limit command is issued from the PMC window, alarm PS0035 is issued.

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

A torque limit is to be specified as in the program example below.

(Program example)

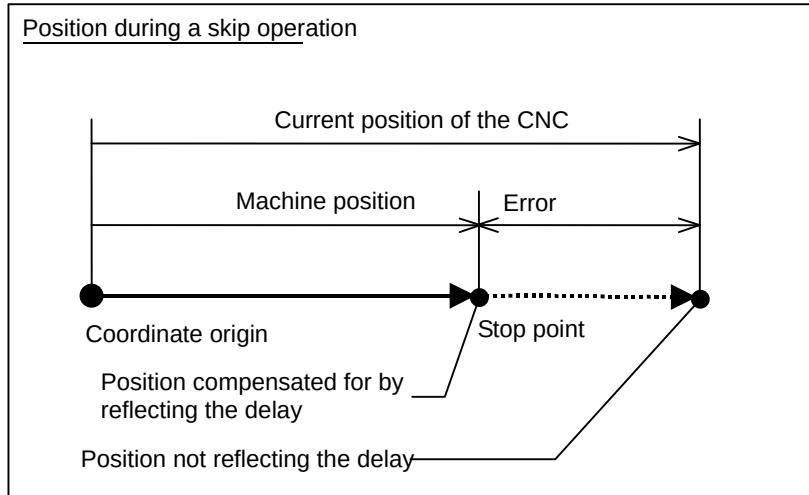
```
O0012
:
Mxx (Specify a torque limit from the PMC via the window)
:
G33 P99 X200. F100.(Torque limit skip command)
:
G01 X100. F500.(Move command with the torque limit being still effective)
:
Myy (Cancel the torque limit from the PMC)
:
M30
```

- 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 #5067 or #100151 to #100157 (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 synchronous control.
- 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	40
5.2 RAPID TRAVERSE	41
5.3 CUTTING FEED	42
5.4 CUTTING FEEDRATE CONTROL	43
5.5 FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS	47
5.6 DWELL	50

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)).

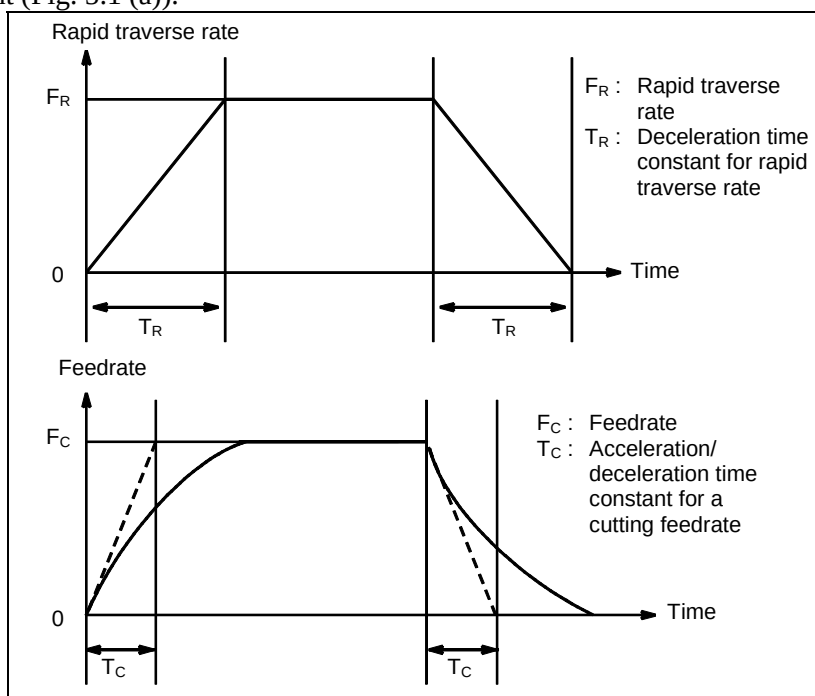


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- 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)).

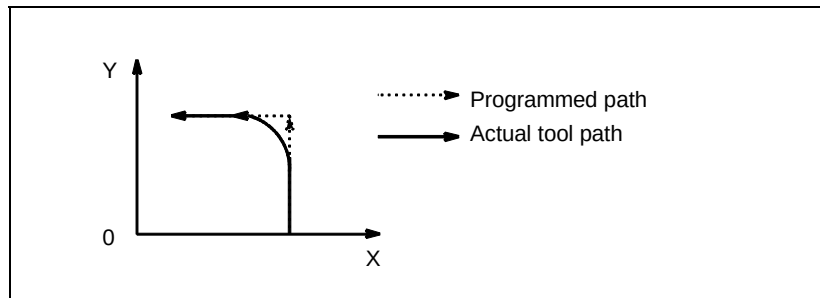


Fig. 5.1 (b) Example of tool path between two blocks

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

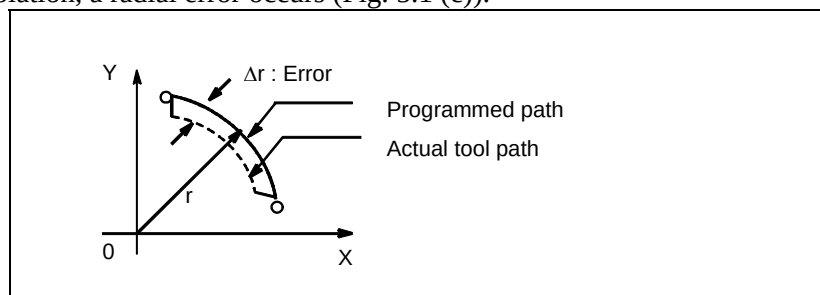


Fig. 5.1 (c) Example of radial error in circular interpolation

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 and punching is performed. 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: 25, 50, 75, 100 %

5.2.1 Rapid Traverse Rate by F Command

Each axis rapid traverse rate of rapid traverse command (G00) are set independently to parameter by machine tool builders.

Whereas, by setting bit 0 (G0F) of parameter No. 16050 to 1, the rapid traverse rate of X and Y axes to rapid traverse command (G00) can be designated by F code. Refer to the manual issued by a machine tool builder for this function.

There are following specifications notices for this function.

- (1) The feedrate specified by F code is the each axis rapid traverse rate of X and Y axes.
- (2) 4-step rapid traverse override can be applied to the rapid traverse rate designated by F code, using signals (ROV2, ROV1) from the machine side.
- (3) When axial move of rapid traverse (G00) is specified in the DNC, memory and MDI modes, the rapid traverse rate may not specified by F code or when the speed command is 0, an alarm (PS 0011) occurs.
- (4) In circular nibbling (G68), linear nibbling (G69) and nibbling by M function, the speed to nibbling pitch after the first punch point corresponds to the rapid traverse rate preset by the parameter (No. 1420).
- (5) F1-digit function for programmable rapid traverse override is ineffective.
- (6) When the rapid traverse rate designated by F code exceeds the speed preset by a parameter (set by a machine tool builder), it is clamped to the speed preset by the parameter (No. 1420).

5.2.2 F1-digit (Programmable Rapid Traverse Override)

By specifying one-digit number from 1 to 4 following address F, and override can be applied to the rapid traverse rate in automatic operation.

F1-digit command	Rapid traverse override
F1	100%
F2	75%
F3	50%
F4	25%

An override can be applied to the rapid traverse rate by the switch on the machine operator's panel as well as by F1-digit command in automatic operation.

Either rapid traverse override being set by the switch or the F1-digit command, whichever lower, becomes effective.

NOTE

F1-digit (Programmable rapid traverse override) cannot be used together with the Rapid traverse rate by F command.

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.

The feed of FANUC Series 0i-PD is performed at feed per minute always.

Format

Feed per minute

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

Explanation

- Direction of the cutting feedrate

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

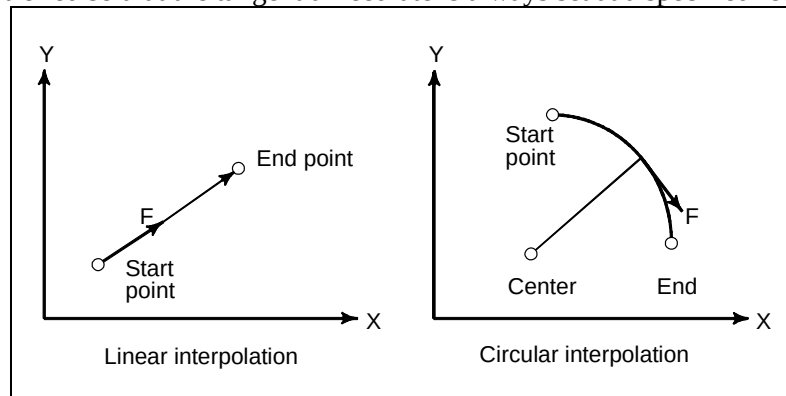


Fig. 5.3 (a) Tangential feedrate (F)

- Feed per minute

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.

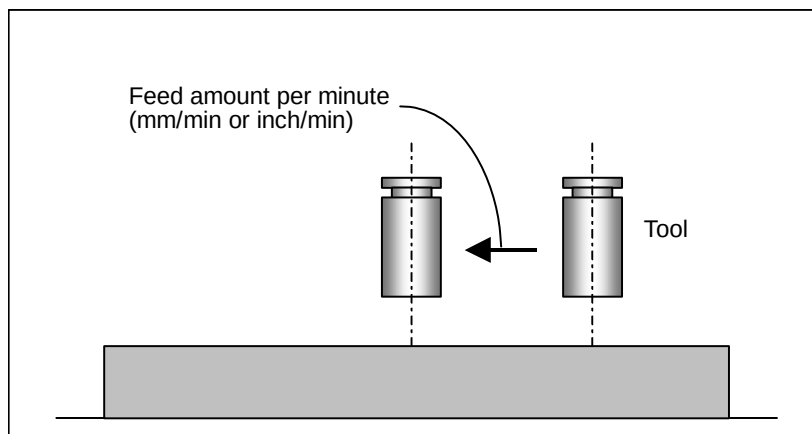


Fig. 5.3 (b) Feed per minute



CAUTION

Cutting feed is invalid for the turret axis (T-axis) and C-axis. T-axis and C-axis commands, therefore, cannot be specified in linear interpolation (G01) mode and circular interpolation (G02, G03) mode. However, when bit 5 (CIP) of parameter No.16360 is set to 1, C-axis can be specified.

Reference

See Appendix D for range of feedrate command value.

5.4 CUTTING FEEDRATE CONTROL

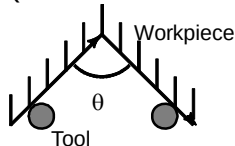
Cutting feedrate can be controlled, as indicated in Table 5.4 (a).

Table 5.4 (a) Cutting Feedrate Control

Function name		G code	Validity of G code	Description
Exact stop		G09	This function is valid for specified blocks only.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Exact stop mode		G61	Once specified, this function is valid until G62, G63, or G64 is specified.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Cutting mode		G64	Once specified, this function is valid until G61, G62, or G63 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed.
Automatic corner override	Automatic override for inner corners	G62	Once specified, this function is valid until G61, G63, or G64 is specified.	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.
	Internal circular cutting feedrate change	-	This function is valid in the tool radius compensation mode, regardless of the G code.	The internal circular cutting feedrate is changed.

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 ;
Automatic corner override	G62 ;

5.4.1 Exact Stop (G09, G61), Cutting Mode (G64)**Explanation**

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

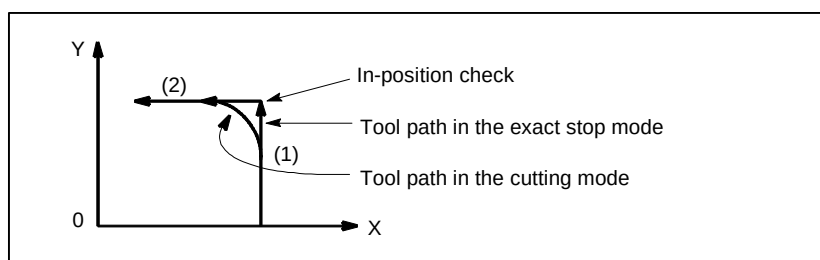


Fig. 5.4.1 (a) Example of tool paths from block (1) to block (2)

**CAUTION**

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

5.4.2 Automatic Corner Override

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)

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)).

$2^\circ \leq \theta \leq \theta_p \leq 178^\circ$ in Fig. 5.4.2 (a) is a value set with parameter No. 1711. When θ is approximately equal to θ_p , the inner corner is determined with an error of 0.001° or less.

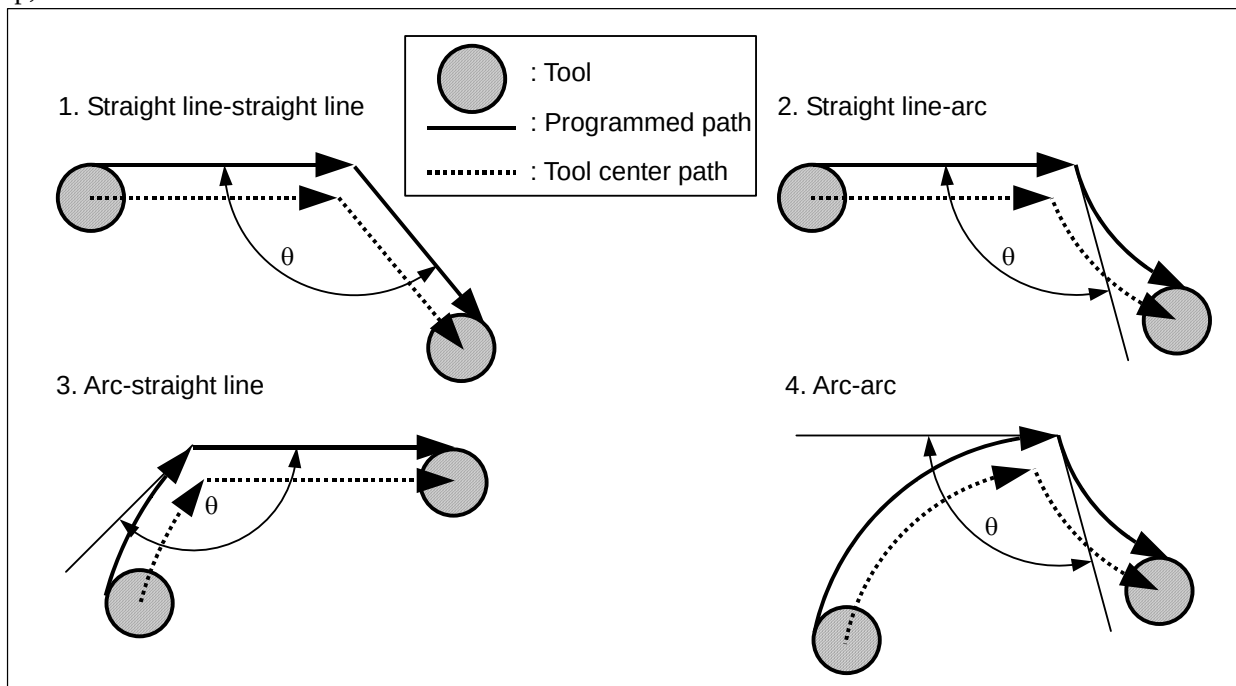


Fig. 5.4.2 (a) Inner corner

- Override range

When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances L_s and L_e , 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)). L_s and L_e are set with parameters Nos. 1713 and 1714.

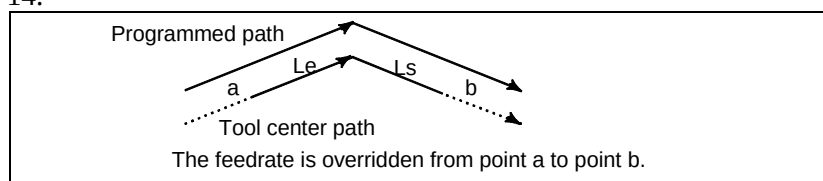


Fig. 5.4.2 (b) Override Range (Straight Line to Straight Line)

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)).

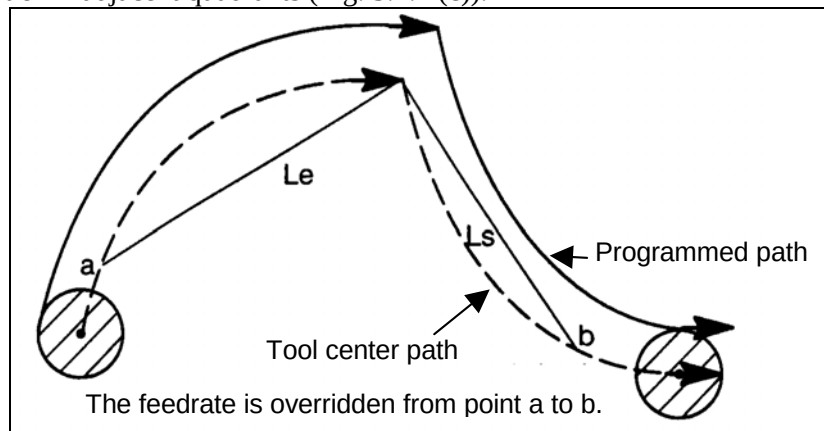


Fig. 5.4.2 (c) Override Range (Arc to Arc)

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)).

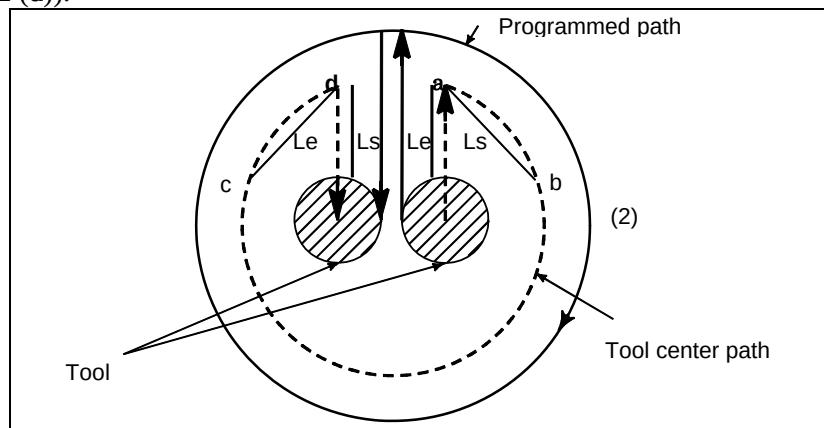


Fig. 5.4.2 (d) Override Range (Straight Line to Arc, Arc to Straight Line)

- Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run.

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

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.

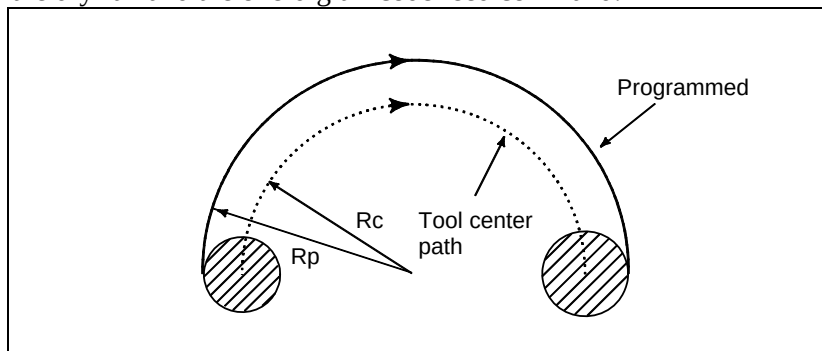


Fig. 5.4.2 (e) Internal circular cutting feedrate change

If Rc is much smaller than Rp, $Rc/Rp \approx 0$; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When $Rc/Rp \leq MDR$, the feedrate of the tool is $(F \times MDR)$. If parameter No. 1710 is 0, the minimum deceleration ratio (MDR) is at 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

For the feedrate instruction on imaginary circle for a rotary axis, a circle with the radius specified with parameter No. 1465 (which is referred to as an imaginary circle) is considered, and the feedrate of the rotary axis is set to the feedrate on its circumference.

Explanation

- Cutting feedrate

Conventional method

Usually, the feedrate of a rotary axis is set to the rotation feedrate (deg/mm).

Feedrate of liner axis(X axis)
$$F_x = F \times \frac{\Delta X}{L} \text{ (mm/min)}$$

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

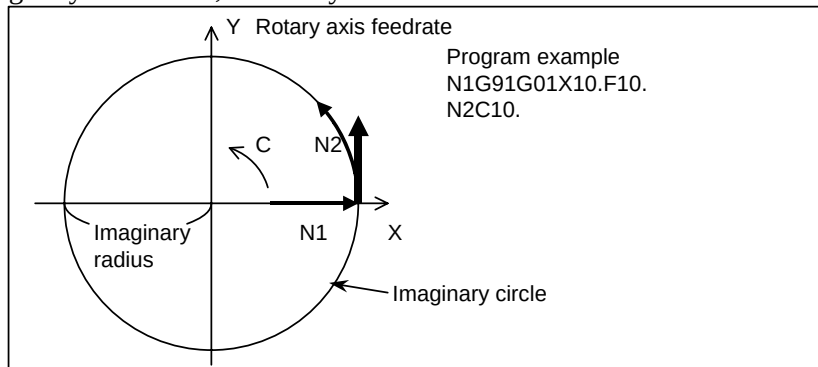
$$\text{Synthetic movement distance} \quad L = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2 + \Delta C^2} \text{ (mm)}$$

$$\text{Movement time} \quad T = \frac{L}{F} \text{ (min)}$$

Feedrate instruction on imaginary circle of a rotary axis

For the feedrate instruction on imaginary circle for a rotary axis, the feedrate of the rotary axis is set to the movement feedrate on an imaginary circle with the radius specified with parameter No. 1465.

By setting the imaginary radius to 0, the rotary axis can be excluded from feedrate calculation.



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

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

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

$$\text{Movement time} \quad T' = \frac{L'}{F} \text{ (min)}$$

l_C : imaginary radius(parameter No.1465)

For the feedrate instruction on imaginary circle for a rotary axis, due to the difference in how to determine the movement distance, if a particularly small value is set for the imaginary radius, the movement on the axis will be fast. Pay careful attention to the input to the parameter.

The cutting feedrate is clamped on the basis of the maximum cutting feedrate parameter for each axis No.1430 and the feedrate of the actual axis (data before this function is converted). It is, therefore, possible to specify a command with a feedrate greater than the setting of the maximum cutting feedrate by setting a large value as the imaginary radius parameter No. 1465. By setting a small value as an imaginary radius, the cutting feedrate is clamped at the feedrate less than the setting of the maximum cutting feedrate.

The feedrate instruction on imaginary circle for a rotary axis is also effective to a dry run.

- AI Advanced Preview Control

AI advanced preview control is performed on the movement feedrate on an imaginary circle. The movement feedrate may, therefore, not be the feedrate for the calculation in the feedrate instruction on imaginary circle for a rotary axis if the feedrate is changed with the feedrate control of AI advanced preview control. The feedrate during AI advanced preview control is clamped with parameter No.1432. If parameter No. 8465 is not 0, it is also clamped with parameter No. 8465.

- 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.

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. ;

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

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

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

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

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).

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

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

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

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

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).

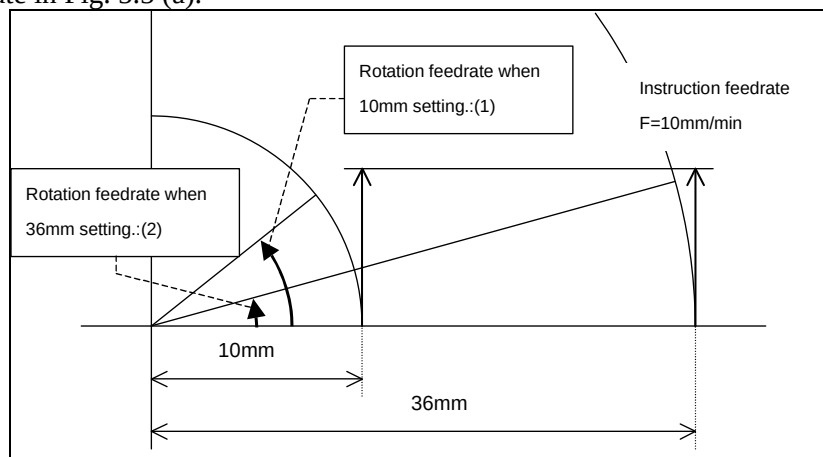


Fig. 5.5 (a)

Limitation

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

- Normal direction control
- Axis control by PMC

NOTE

- 1 When the bit 0 (ROT_x) of parameter No. 1006 and the bit 0 (RFD_x) of parameter No. 1408 are 1, this function becomes effective.
- 2 The bit 0 (RFD_x) of parameter No. 1408 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 bit 0 (RFD_x) of parameter No. 1408 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

G04 X_ ; or G04 P_ ;

X_ : Specify a time (decimal point permitted)

P_ : Specify a time (decimal point not permitted)

Explanation

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

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

Increment system	Command value range	Dwell time unit
IS-A	0.01 to 999999.99	s
IS-B	0.001 to 99999.999	

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 s
IS-B	1 to 99999999	0.001 s

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, or P is specified without a decimal point, the specification unit depends on the increment system of the X-axis, and it does not depend on inch/metric input.
- 2 When P is specified, bit 7 (IPR) of parameter No. 1004 exercises no influence.

While dwell is being executed, “1” is set for diagnosis display No. 0002.

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.

Diagnosis display

2	Dwell execution status

While dwell is being executed, “1” is displayed.

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.....	52

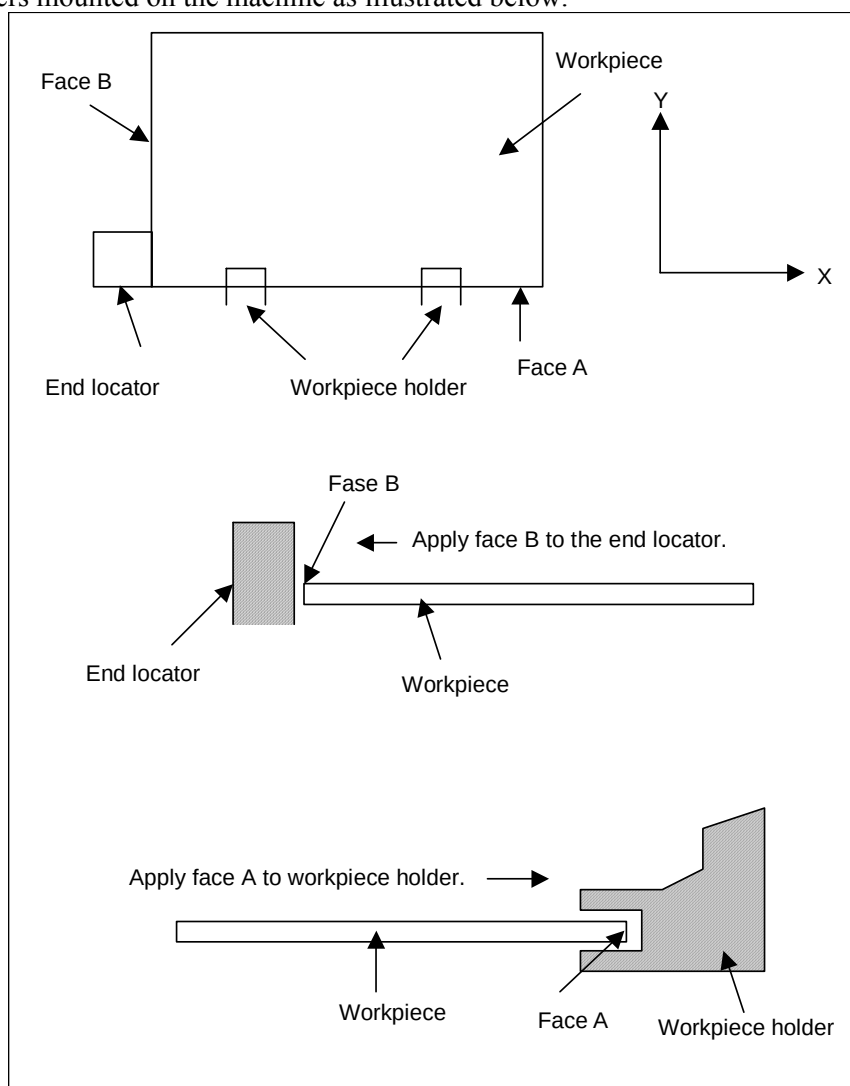
6.1 REFERENCE POSITION RETURN

Overview

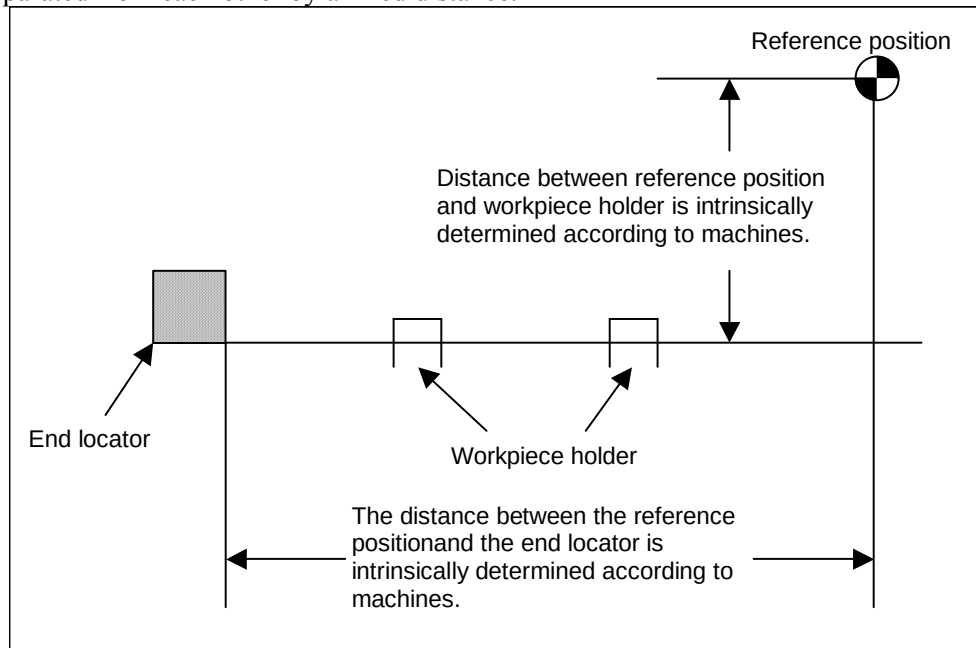
- Reference position

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.

When setting a workpiece to be machined to general turret punch press, the workpiece is held by the workpiece holders after positioning it by applying the end face of the workpiece to the end locator and workpiece holders mounted on the machine as illustrated below.



Generally, the distance between the reference position and the end locator as well as the distance between the reference position and the workpiece holders are intrinsically determined according to machines, and they are separated from each other by a fixed distance.



For example, if the start point is at the reference position and the point located at the left lower side of the workpiece is presumed as the zero point of the workpiece coordinate system. The tool position at the start point can be taught to NC as a position in the workpiece coordinate system by giving the following command at the initial stage of programming.

G92X x_R Y y_R ;

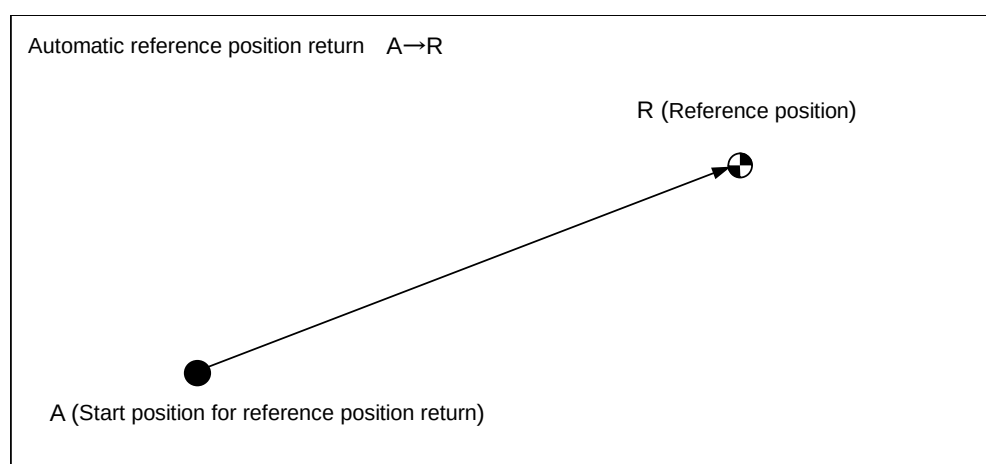
where, x_R : Distance from end locator to reference position along X-axis

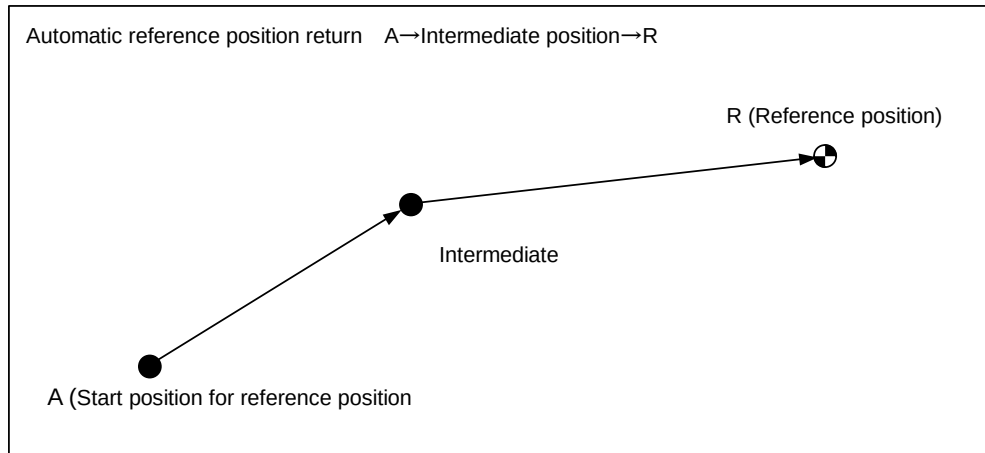
y_R : Distance from workpiece holder to reference position along Y-axis

- Reference position return

Tools are automatically moved to the reference position. When reference position return is completed, the lamp for indicating the completion of return goes on.

Setting bit 0 (SZR) of parameter No.16205 allows specifying whether to pass through the intermediate point or not when the axis is returned to reference position.





Format

- **Return to the reference position without passing through the intermediate point**

G28_; Reference position return

- **Return to the reference position via the intermediate point**

G28 IP_; Reference position return

IP : Specify the intermediate position in the absolute coordinate system.
(absolute/incremental programming)

Explanation

- **Return to the reference position without passing through the intermediate point (G28)**

Reference positions are performed at the rapid traverse rate of each axis.

Therefore, for safety, the compensation functions, such as the tool radius compensation, should be cancelled before executing this command.

(Example)

G28 M30 ;

- **Return to the reference position via the intermediate point**

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 tool radius compensation, should be cancelled before executing this command.

(Example)

G28 X100.0 Y150.0 ;

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

- **First return to the reference position after the power has been turned on (without an absolute position detector)**

When the G28 command is specified when manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position.

In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006).

- **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 reference position return lamp may also light up even if the programmed position is shifted from the reference position by 1 μ . This is because the least input increment of the machine tool system is smaller than its least command increment.

Reference item

- **Manual reference position return**

See III-3.1 MANUAL REFERENCE POSITION RETURN.

7 COORDINATE SYSTEM

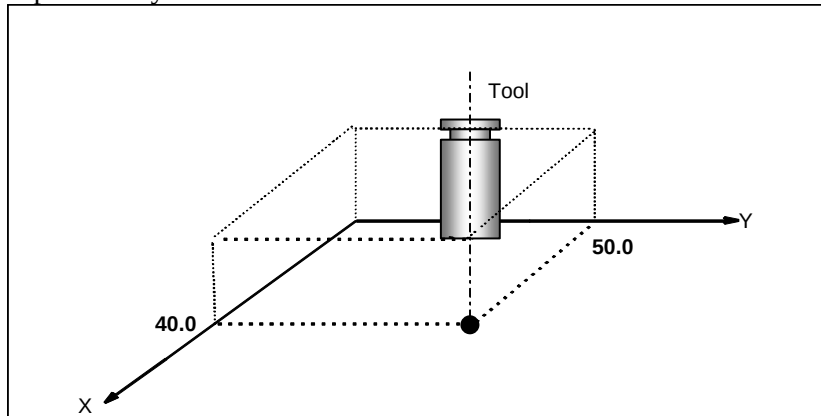
By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.

When two program axes, the X-axis and Y-axis are used, coordinates are specified as follows:

X_Y_

This command is referred to as a dimension word.

- Tool position specified by X40.0Y50.0



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 Section, "MANUAL REFERENCE POSITION RETURN"). 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 "Reference - Setting a machine coordinate system" described later.)

Format

(G90)G53 IP _ P1;

IP_ : Absolute command 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 tool radius compensation beforehand.

- G53 command immediately after power supply turning on

The Reference point by the manual Reference point return or G28 command should return once after turning on the power supply because it should set the machine coordinate system before it instructs in G53.

However, the operation is unnecessary for the machine with the absolute-position detector.

- Specification in the same block

Commands G50/G51 (Scaling), and G84/G85 (Rotation) cannot be specified in the same block where the G53 command is specified.

Note

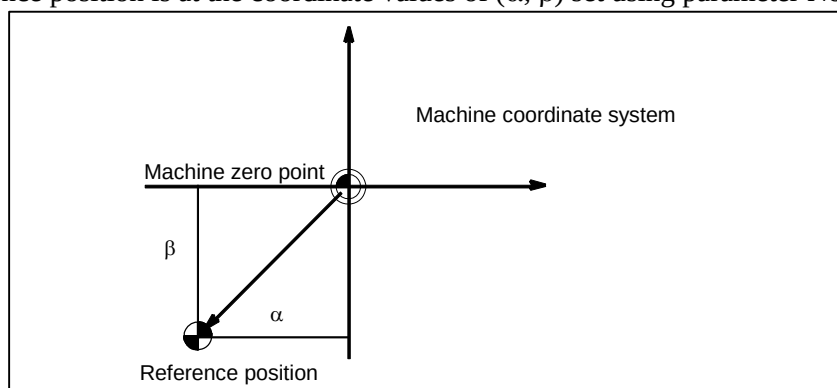
NOTE

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 (ZPR) of parameter No. 1201 is set to 1, a workpiece coordinate system is automatically set when manual reference position return is performed. (See Section, “MANUAL REFERENCE POSITION RETURN”.)

This function is, however, disabled when the workpiece coordinate system option is being used.

- (3) Method using a workpiece coordinate system selection G code

Six workpiece coordinate systems can be set beforehand using the MDI unit. Program commands G54 to G59 can be used to select the workpiece axis to be used. (See Subsection, “Displaying and Setting the Workpiece Origin Offset Value”.)

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

Format

- **Setting a workpiece coordinate system**

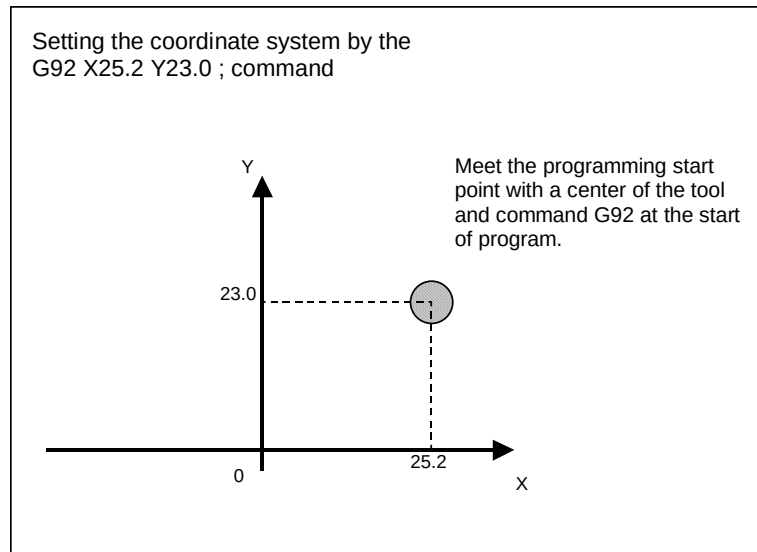
(G90) G92 IP_

Explanation

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

Cutter compensation is cancelled temporarily with G92.

Example



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 Subsection, “Setting a Workpiece Coordinate System”.)

- (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 unit
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.

Example

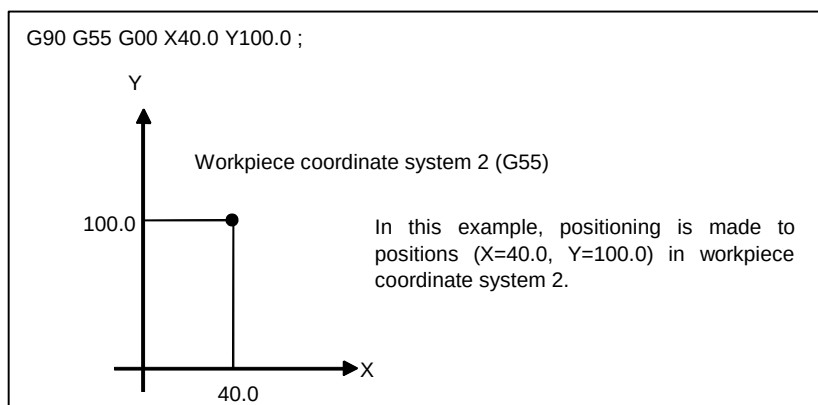


Fig. 7.2.2 (a)

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 unit (see Subsection, “Displaying and Setting the Workpiece Origin Offset Value”.)
- (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.

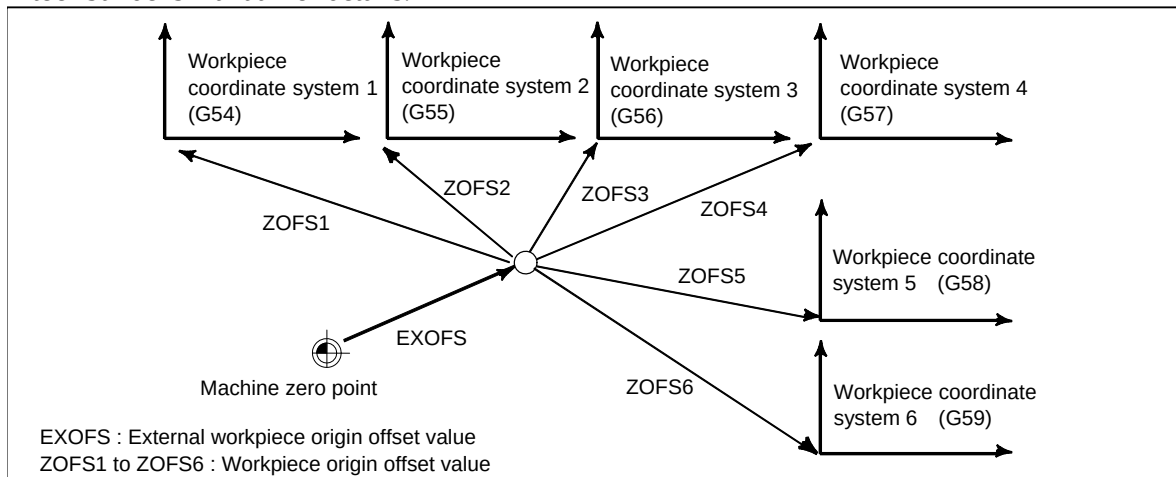


Fig. 7.2.3 (a) Changing an external workpiece origin offset value or workpiece origin offset value

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

G92 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.

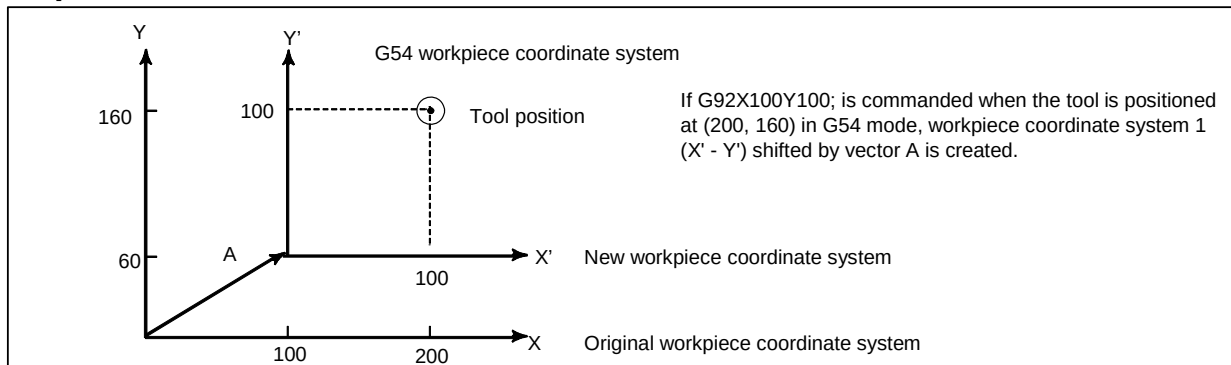
- Changing by setting a workpiece coordinate system

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 G92X100.0Y80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Y = 80.0 is set.

Example

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.

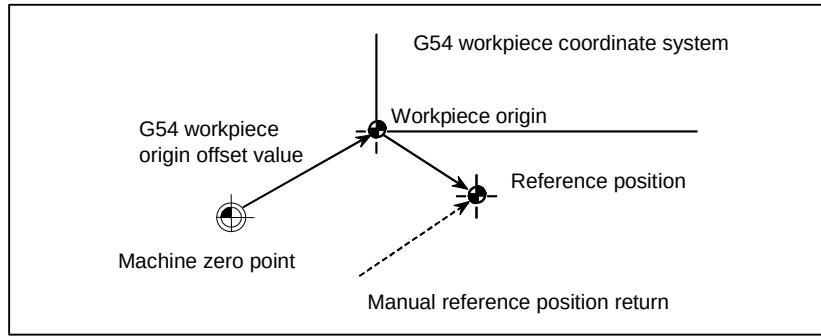
There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command. The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (see Subsection, “Workpiece Coordinate System Preset”).

Format**G92.1 IP 0 ;**

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

Explanation

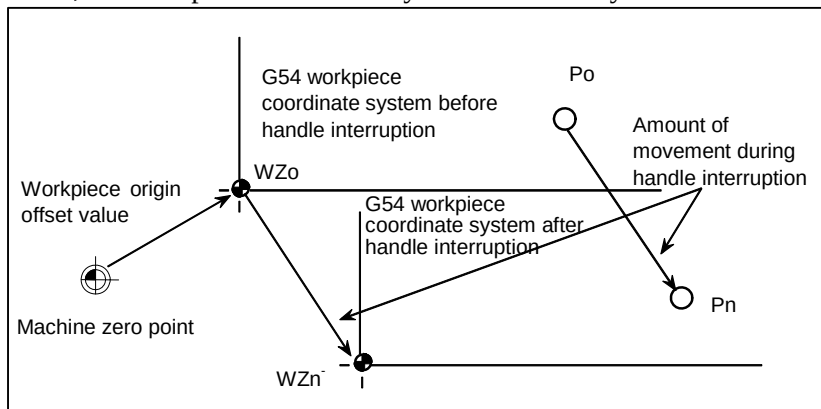
When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece origin offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate 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) Move command executed in the machine lock state
- (b) Movement by handle interruption
- (c) Operation using the mirror image function
- (d) Shifting the workpiece coordinate system by setting the local coordinate system or workpiece coordinate system

In the case of (b) above, the workpiece coordinate system is shifted by the amount of handle interruption.



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 (RELATIVE) as well as absolute coordinates.

When no workpiece coordinate system option (G54 to G59) is selected, the workpiece coordinate system is preset to the coordinate system with its origin placed at the reference position.

Limitation

- Tool radius compensation

When using the workpiece coordinate system preset function, cancel compensation modes: Tool radius 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 is set.

7.2.5 Automatic Coordinate System Setting

When bit 0 (ZPR) of parameter No. 1201 for automatic coordinate system setting is 1, a coordinate system is automatically determined when manual reference position return is performed.

Once α and β 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 $X = \alpha$ and $Y = \beta$.

This processing occurs as if the following are specified at the reference position:

G92X α Y β ;

This function cannot be used, however, when the workpiece coordinate system is applied.

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.

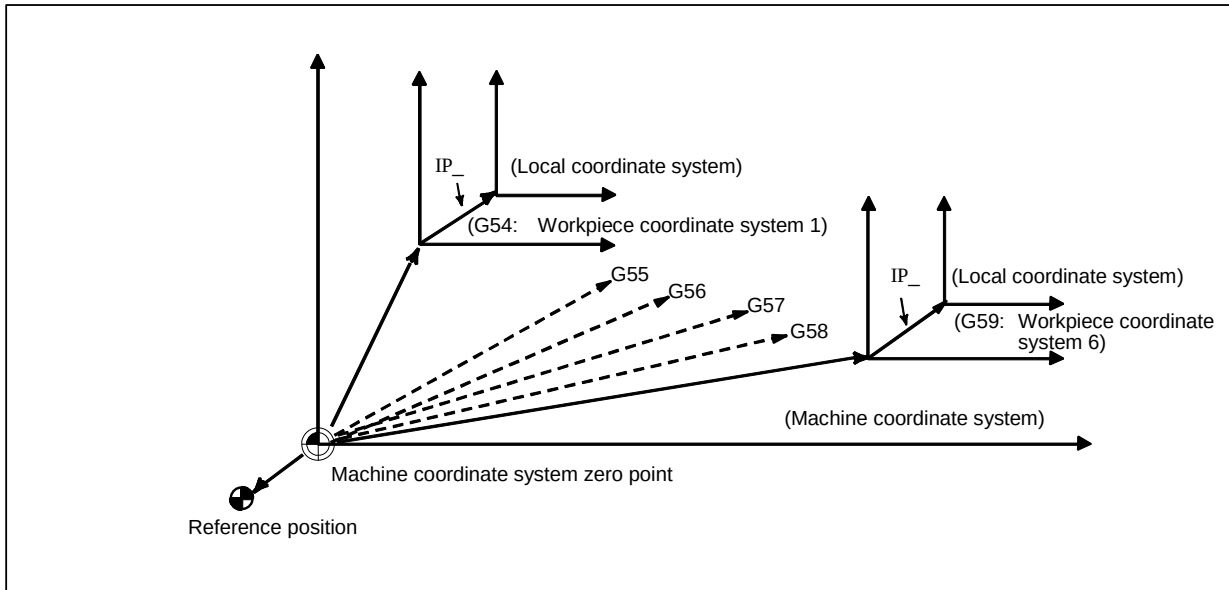


Fig. 7.3 (a) Setting the local coordinate system

⚠ 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:
 $G52_{\alpha}0;$
 α : Axis which returns to the reference position
- 2 The local coordinate system setting does not change the workpiece and machine coordinate systems.
- 3 Whether the local coordinate system is canceled at reset depends on the parameter setting. The local coordinate system is canceled when either bit 6 (CLR) of parameter No.3402 or bit 3 (RLC) of parameter No.1202 is set to 1
- 4 When a workpiece coordinate system is set with the G92 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 block remains unchanged.
- 5 G52 cancels the offset temporarily in tool radius compensation.
- 6 The incremental programming after the G52 block is not regarded as the incremental distance from the current tool position. This programming is regarded the incremental distance from origin of the local coordinate system specified.
 However, When the bit 0 (LIP) of parameter No. 16201 is set to 1, the incremental programming after the G52 block is regarded as the incremental distance from the current tool position.

7.4 PLANE SELECTION

Select the planes for circular interpolation, tool radius compensation, and coordinate system rotation. 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.

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).

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

G18Y_ ; 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 (G90, G91)	66
8.2 INCH/METRIC CONVERSION (G20, G21)	66
8.3 DECIMAL POINT PROGRAMMING	69

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

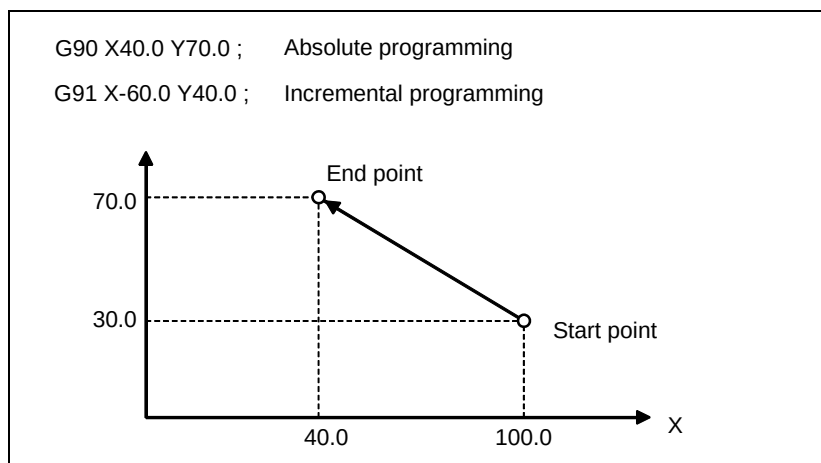
There are two ways to command travels of the tool; the absolute programming, and the incremental programming. In the absolute programming, coordinate value of the end position is programmed. The incremental programming is used to program the amount of a tool movement.

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

Format

Absolute programming	G90 IP_ ;
Incremental programming	G91 IP_ ;

Example



8.2 INCH/METRIC CONVERSION (G20, G21)

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

Format

G20 ;	Inch input
G21 ;	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 (see Section, "INCREMENT

SYSTEM”). The unit of data input for degrees remains unchanged. The unit systems for the following values are changed after inch/metric conversion:

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

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



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 Subsection, “Displaying and Entering Setting Data”).
- 3 If a function selected using bit 2 (IRF) of parameter No. 14000 or bit 0 (NIM) of parameter No. 11222 is not used, be sure to perform inch/metric conversion at the reference position (machine coordinate system origin).

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

Conventionally, inch/metric conversion must be performed at the machine coordinate system origin. 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, “CONVERT INCH/MM AT REF-POS” 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.

In addition, if the workpiece coordinate system has been shifted from the machine coordinate system, 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.

- Move command issued with the machine locked
- Move command issued using a handle interrupt
- Mirror image-based operation
- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92)

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, “ILLEGAL INCH/METRIC CONVERSION” being issued

- Positioning or linear interpolation
- Tool radius compensation cancel
- Normal direction control cancel

- Scaling cancel
- Macro modal call cancel
- Coordinate system rotation mode off

In addition, performing inch/metric conversion in any position other than the reference position requires the following options.

- Inch/metric conversion
- Workpiece coordinate system

Restrictions

When performing the following operations, make sure that the axis of interest is in the reference position (machine coordinate system origin).

- 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 from the machine coordinate system, 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.

- Move command issued with the machine locked
- Move command issued using a handle interrupt
- Mirror image-based operation
- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92)

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)

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.

- Positioning or linear interpolation
- Tool radius compensation cancel
- Normal direction control cancel
- Scaling cancel
- Macro modal call cancel
- Coordinate system rotation mode off

In addition, performing inch/metric conversion in any position other than the reference position requires the following options.

- Inch/metric conversion
- Workpiece coordinate system
- Workpiece coordinate system preset

Restrictions

When performing the following operations, make sure that the axis of interest is in the reference position (machine coordinate system origin).

- 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:

X, Y, Z, U, V, W, A, B, C, I, J, K, Q, 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 minimum unit is treated as described below.

Example 1)

When a value is specified directly at an address (in the case of 1/1000mm IS-B)

X1.2345 ; Treated as X1.235
X-1.2345 ; Treated as X-1.234

Example 2)

When a value is assigned to a macro variable (in the case of 1/1000mm IS-B)

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

NOTE

- 2 When more than eight 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:

X1.23456789;

Alarm PS0003, "TOO MANY DIGIT" occurs because more than eight digits are specified.

X123456.7;

If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an alarm occurs.

9 PRESS FUNCTION

Chapter 9, "PRESS FUNCTION", consists of the following sections:

9.1 PUNCH FUNCTION (1-CYCLE PRESSING)	71
9.2 POSITIONING & PRESSING OFF (G70).....	73
9.3 NIBBLING FUNCTION	73
9.4 NIBBLING BY M FUNCTION	82
9.5 EXTERNAL MOTION FUNCTION.....	86

9.1 PUNCH FUNCTION (1-CYCLE PRESSING)

This control sends a signal "Start press and punch" to the machine after moving a tool to the position commanded in a predetermined block.

When the machine receives this signal, it starts pressing. As a result, punching is made on a workpiece by the selected tool. After punching, the press motion stops, and a signal returns to the NC to indicate that "punch has finished".

Thus, NC proceeds to the execution of the next block. In this manner, punching on a workpiece by press motion is executed by data transfer between the NC and the machine, and it is necessary to know the blocks to be punched, in advance.

This description is made from the viewpoints of the NC side. Since details may differ depending upon the machine tool builders, refer to the machine tool builder's manual without fail.

NOTE

This section covers one-cycle punch only. For nibbling (punching by sequential repeated press motion), refer to "9.3 NIBBLING FUNCTION".

9.1.1 Block in which Punching is Made

Punching is made in a block where the X-axis or Y-axis is positioned at rapid traverse, in principle. In other words, punching is not done in a block where the X-axis or Y-axis is not positioned at rapid traverse. Blocks where punching is done are as follows:

- (1) Block where X-axis or Y-axis is positioned in the positioning mode (G00)

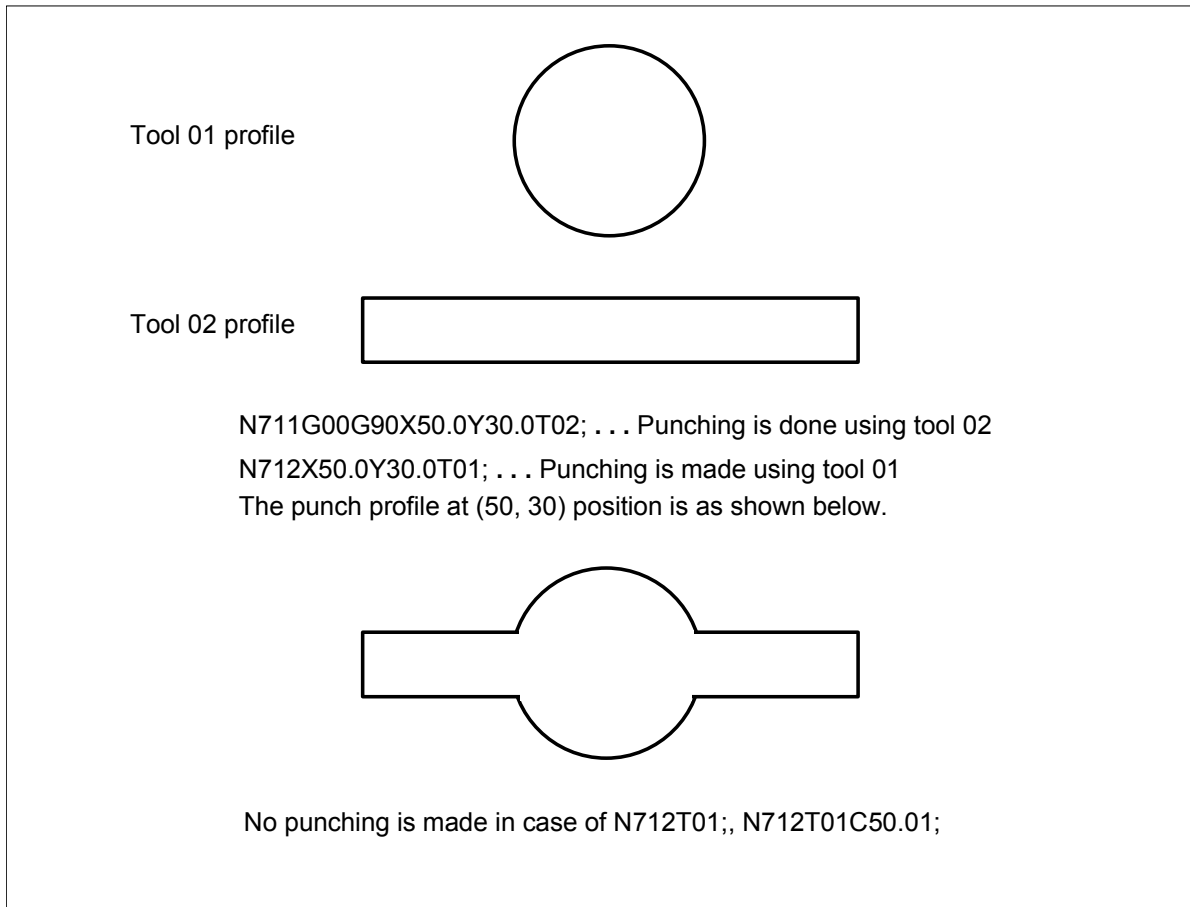


CAUTION

If the same position as the present tool position is commanded by address X or Y, positioning is not done, but punching is executed. (This is regarded as the positioning command with movement amount 0)

G00G91X0; . . . Punching is made.

This applies to such a case that punching is done at the same position using a different tool.

**CAUTION**

Punching is not done in T single block where the X-axis or Y-axis moves for tool offset.

- (2) Block where pattern function G26, G76, G77, G78, G79, G86, G87 or G89 was commanded
 Punching is made after positioning to respective points on a pattern.

Punching is not done in the following cases, even if the block corresponds to (1) or (2).

- (a) MDI mode is selected.
- (b) M code is commanded.
- (c) Blocks inserted between M code of workpiece clamp and M code of workpiece unclamp which are employed for repositioning of workpiece.
- (d) Block where positioning & punch off (G70) was commanded.

**CAUTION**

Punching is not done even in G00 mode if the block is irrespective of positioning such as coordinate system setting (G92), local coordinate system setting (G52), standard point command (G72), dwell (G04), etc.

9.2 POSITIONING & PRESSING OFF (G70)

Punching is made in a block where the X-axis or Y-axis is positioned at rapid traverse, in principle. Command the following code, if it is not desired to punch a workpiece after positioning a tool to the commanded position at rapid traverse.

G70X__Y__;



CAUTION

- 1 G70 is an one-shot G code.
- 2 Rapid traverse is made in a G70 block even if in G01, G02 or G03 mode.

9.3 NIBBLING FUNCTION

Nibbling means sequential repeated punching without stopping press motion.

Assume T_t be the time required for one-cycle press motion. The remaining time obtained by subtracting punching time T_p from T_t (or, $T_i = T_t - T_p$) is the time allowable for positioning.

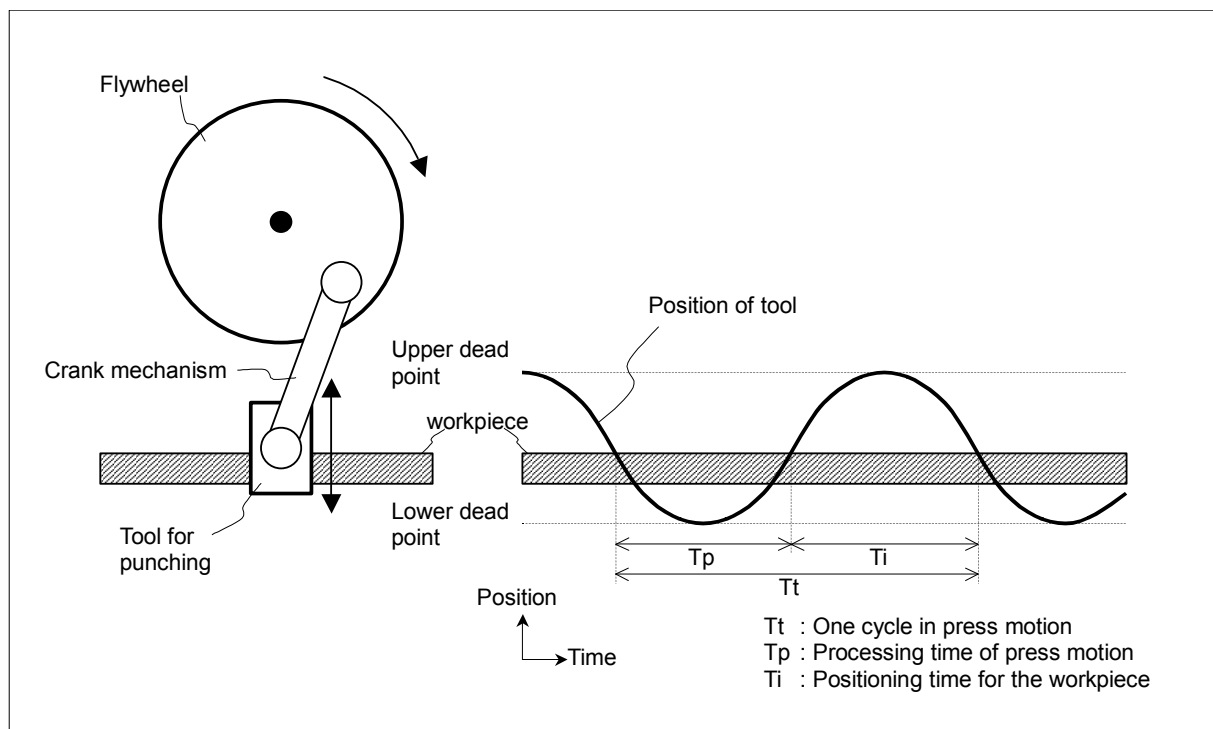
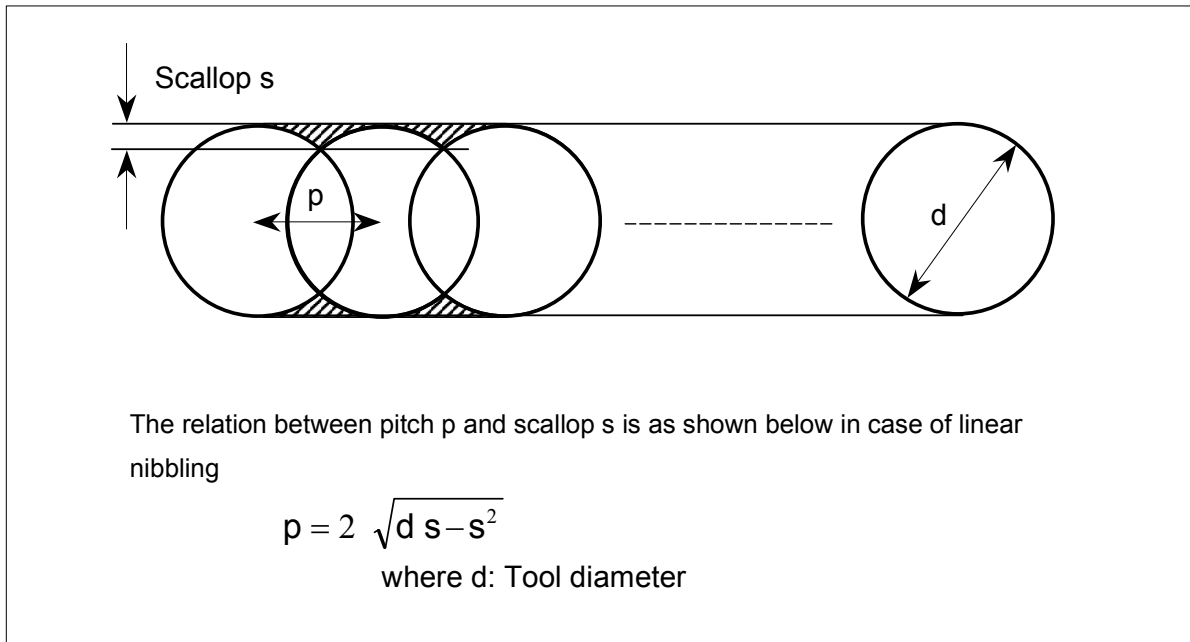


Fig.9.3

The maximum distance (maximum pitch) which can be positioned in time T_i is limited by various conditions, such as machine, servo motor, and others as well as time T_i .

In this NC, the maximum nibbling pitch determined by these conditions is preset as a parameter. On the other hand, the nibbling pitch is commanded by a program. If the commanded pitch exceeds the maximum pitch preset by the parameter, an alarm is produced.

Since this pitch can be specified directly, programming can be done, while taking the scallop into consideration.



The following functions are prepared for nibbling.

Function	Description
Circular nibbling (G68)	Punching at constant intervals (distance) along the Circular arc for a specified angle. G68 I <u>r</u> J <u>θ</u> K <u>Δθ</u> P <u>d</u> Q <u>p</u>
Linear nibbling (G69)	Punching at constant intervals (distance) along the straight line for a specified distance. G69 I <u>ℓ</u> J <u>θ</u> P <u>d</u> Q <u>p</u>
Nibbling by M function	M## ← M code of nibbling mode (Set value of parameter No.16183) } The route and the pitch are specified according to these command M\$\$ ← M code of nibbling mode cancel (Set value of parameter No.16184)

9.3.1 Circular Nibbling (G68)

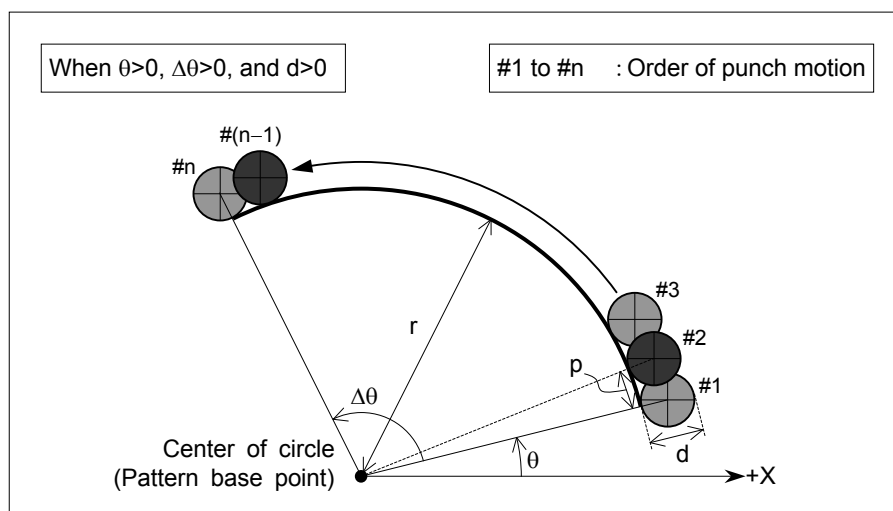
Circular nibbling (G68) specifies punching at constant pitch (distance) on the circular arc with a start point specified by G72 or the current tool position as the center of the circle.

Format

G68 I_r J_θ K_{Δθ} P_d Q_p

- r** : Radius of arc (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.
- θ** : Angle formed between the first punch point and the + X axis
(Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : minimum unit of data (deg)
-The counterclockwise direction is commanded by a positive number.
- Δθ** : Incremental angle from the first punch point to the least punch point
(Valid data range : -999999999~999999999)
Unit of data : minimum unit of data (deg)
-Counterclockwise nibbling is made when this angle is commanded by a positive number.
- d** : Tool diameter value (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-Nibbling is made outside the arc when this value is positive, inside the arc when this value is negative, and on the circumference when this value is 0.
-Actual tool path is the arc having radius $r+d/2$.
- p** : Pitch (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.
-This pitch is specified as an arc length.

Nibbling is made at pitch p using a tool having diameter d , starting with the point which forms angle θ with reference to the X-axis on the circumference having radius r , with the preset tool position or the position specified by G72 being set as the circle center, to the point which forms angle $\theta + \Delta\theta$ with reference to the X-axis.



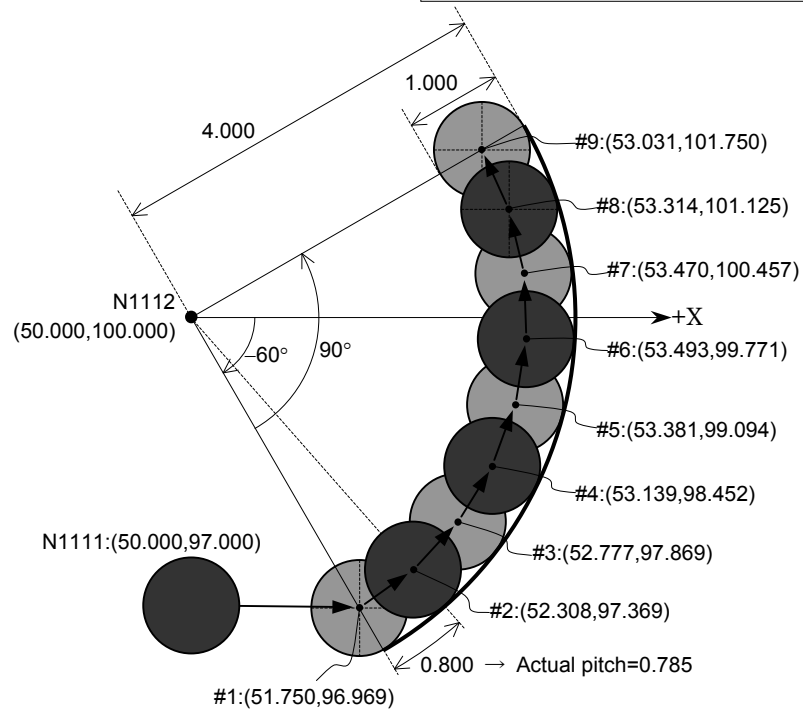
Example 1

```

N1111 G90 G00 X50. Y97.
N1112 G72 X50. Y100.
N1113 G68 I4. J-60. K90. P-1. Q0.8

```

Coordinates are expressed by IS-B
 #1 to #9 : Order of punch motion in
 N1113
 → : Path of tool



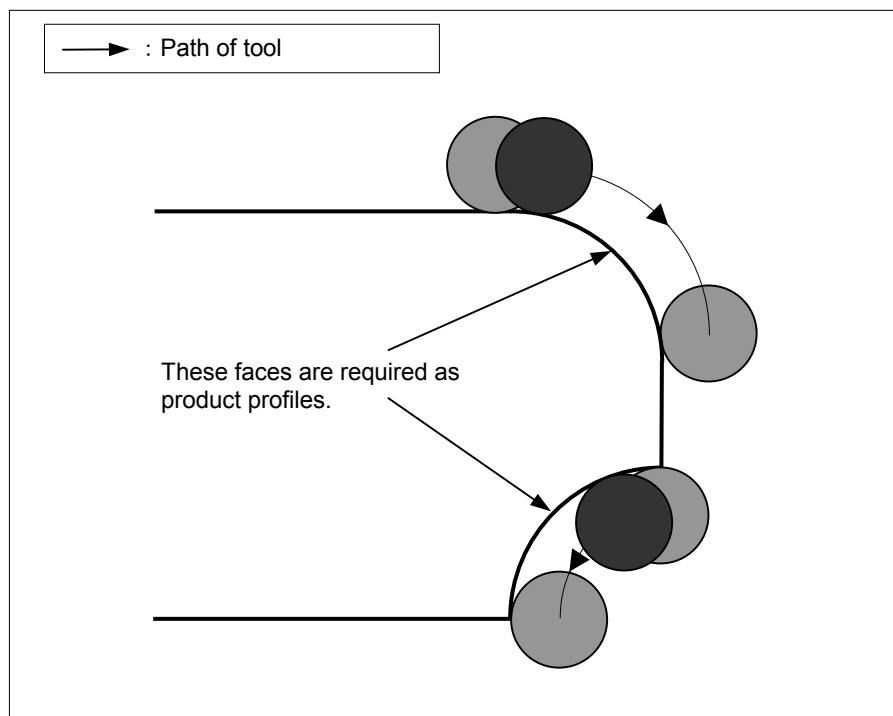
⚠ CAUTION

- 1 G68 is an one-shot G code.
- 2 The base point of G68 is the center of arc.
- 3 Pitch specification

The pitch is specified by the arc length.

The pitch is defined as the divided length of the arc having radius r specified in address I. The pitch does not mean the arc length between the centers of adjacent punch points, i.e., the divided length of the arc having a radius of $r + d/2$ (d : tool diameter).

This reason is shown below.



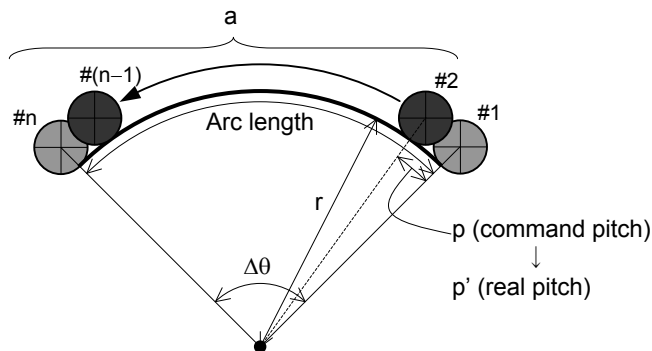
The commanded arc faces are required as a final product, irrespective of whether the tool moves along outside of the arc (right upper part) or inside of the arc (right lower part).

Accordingly, it is desirable that the pitch along the specified arc is equal to the commanded pitch, when taking the scallop into consideration.

4 Pitch compensation

In punching using the command pitch (p), it is generated a remainder that does not come up to 1 pitch. When the remainder is the pitch moving the last punching point, only the last pitch is different from the other pitch. That is why the pitch automatically corrected by the method of below figure is used as an actual pitch to keep a constant pitch from first to last. As a result, an actual pitch becomes the maximum below the instruction pitch and constant an interval.

Calculating method of actual pitch



- (1) The command pitch (p) is defined as a temporary pitch.
- (2) When the number of punching point without first punching point is defined as a, the number of punching point a is calculated by the following expression. (In the point, the number of punching point a must be integral number. However, when the number after the decimal point is cut off, tool does not arrive at the terminal punching point. So the number after the decimal point is rounded up.)

$$a = \frac{\text{Arc length}}{p} = \frac{2\pi \times r \times \frac{|\Delta\theta|}{360}}{p} = \frac{2\pi \times r \times |\Delta\theta|}{360 \times p}$$

(after the decimal point rounded up.)

- (3) And when the actual pitch is defined as p', pitch p' is the pitch to punch evenly by the number of punching point a rounded up. Therefore, pitch p is calculated by the following expression.

$$p' = \frac{\text{Arc length}}{a} = \frac{2\pi \times r \times |\Delta\theta|}{360 \times a}$$

- 5 If radius is 0, or if the pitch is 0 or more than a specified value, alarm (PS 4523) is produced.

9.3.2 Linear Nibbling (G69)

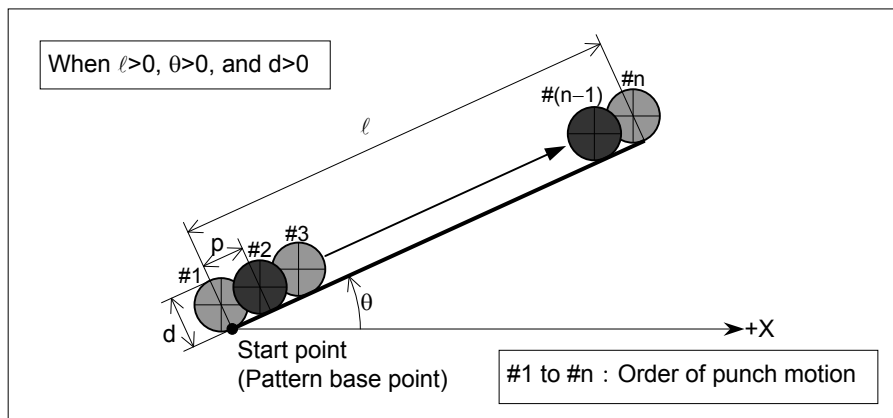
Linear nibbling (G69) specifies punching at constant pitch (distance) on the straight line which forms an specified angle with a start point specified by G72 or the current tool position as the start point.

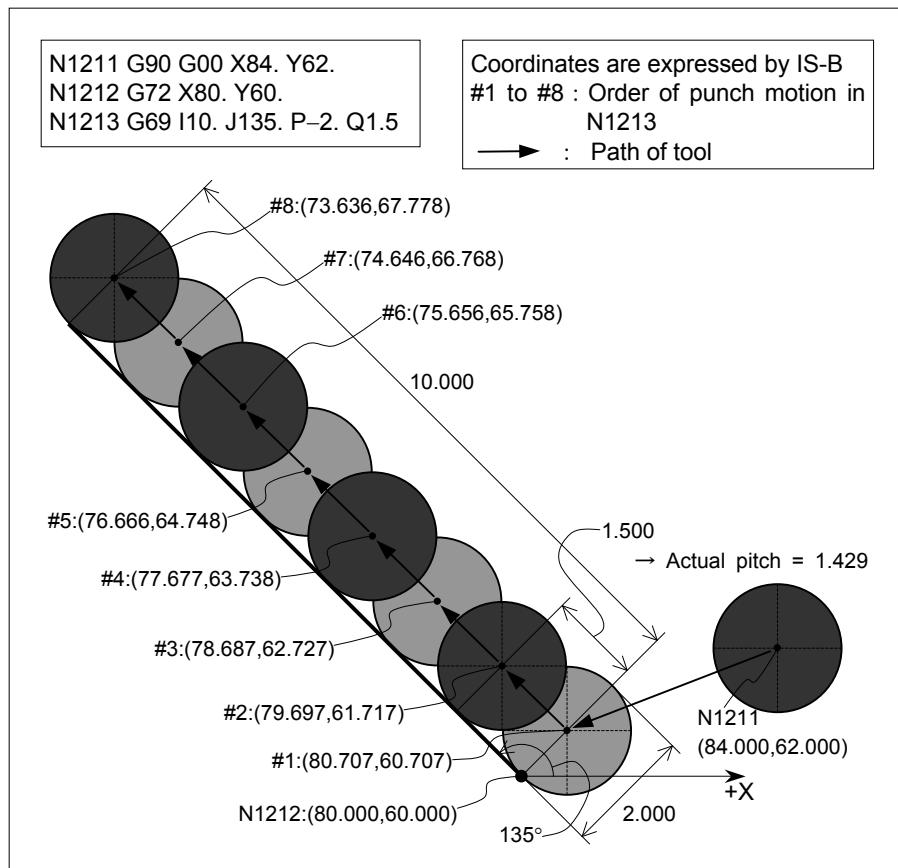
Format

G69 I ℓ J θ P d Q p

- ℓ : Length of straight line (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-If a negative number is commanded, the angle between the straight line and the +X axis becomes $\theta + 180$ deg.
- θ : Angle formed between straight line and the + X axis
(Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : minimum unit of data (deg)
-The counterclockwise direction is commanded by a positive number.
-The clockwise direction is commanded by a negative number.
- d : Tool diameter value (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-When this value is positive, nibbling is made leftwards as viewed from the start point to the end point of the straight line.
-When this value is negative, nibbling is made rightwards as viewed from the start point to the end point of the straight line.
-When this value is 0, nibbling is made along the straight line.
- p : Pitch (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.
-This pitch is specified as an straight line length.

By the above command, nibbling is made at pitch p using a tool having diameter d along a straight line of length which forms angle θ with reference to the X-axis, starting with the present tool position or the position specified by G72 as the start point.



Example 2**⚠ CAUTION**

- 1 G69 is a one-shot G code.
- 2 The base point of G69 is the start point.
- 3 The pitch compensation is the same as in circular nibbling (G68); except that the circumferential length of an arc in G68 is changed to the linear length (For details, refer to Note 4 in 9.3.1)
- 4 If pitch is 0 or more than specified, alarm (PS 4524) is produced.

9.3.3 Notes on Circular Nibbling (G68) and Linear Nibbling (G69)**⚠ CAUTION**

- 1 The maximum pitches in G68 and G69 are set by parameters No. 16186.
- 2 If T code is commanded in G68 or G69 block, nibbling is started after the X and Y axes have moved to the first punch point and also a tool has been selected.
- 3 M code is not commandable in G68 and G69 blocks.
- 4 For the rapid traverse to the first punch point, the rapid traverse override is effective when it is specified by the rapid traverse override switch on the machine operator's panel or by F1-digit specification. For the pitch movement up to the final point, the rapid traverse override is ineffective and fixed to 100%.
- 5 If G68 or G69 is commanded using the single block operation, nibbling is made up to the last punch point, and then, stopped.

- 6 If feed hold is applied halfway during the movement to the first punch point, the X and Y axes stop at once. These axes also stop immediately when the feed hold is applied halfway during the pitch movement from the first point to the last point. However, this can be changed by bit 2 (NSP) of parameter No. 16181 in such a way that the X and Y axes stop after pitch movement.
- 7 In a block just after G68 or G69, the tool does not move by the incremental amount from the tool position when nibbling ends, but moves from the programmed end point of the arc or straight line by the incremental amount.

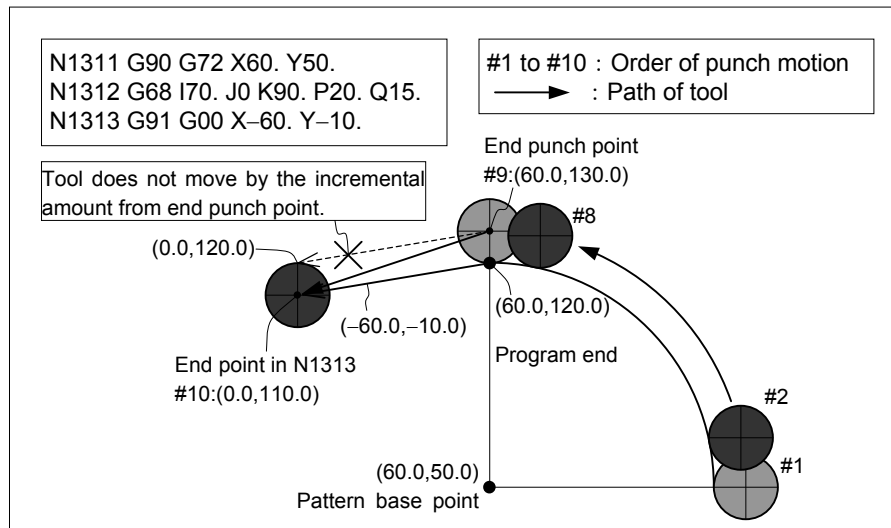


Fig.9.3.3 (a) Incremental programming Immediately after circular nibbling (G68)

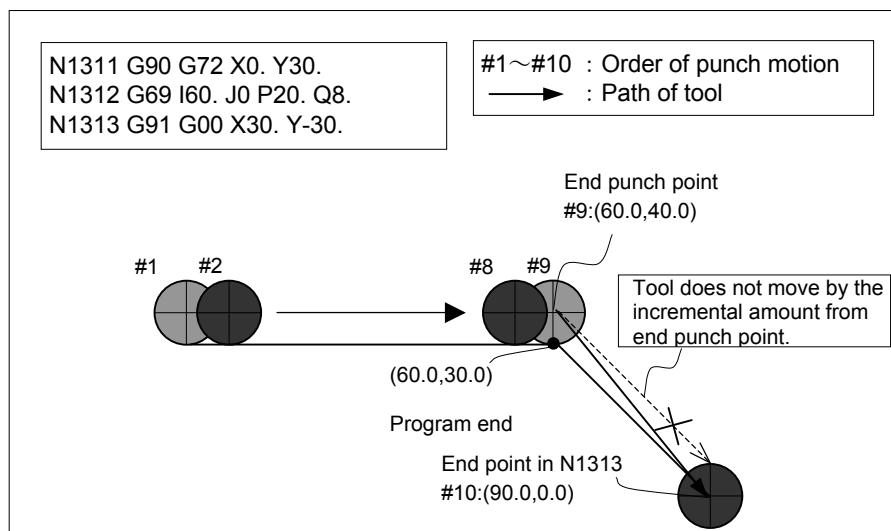
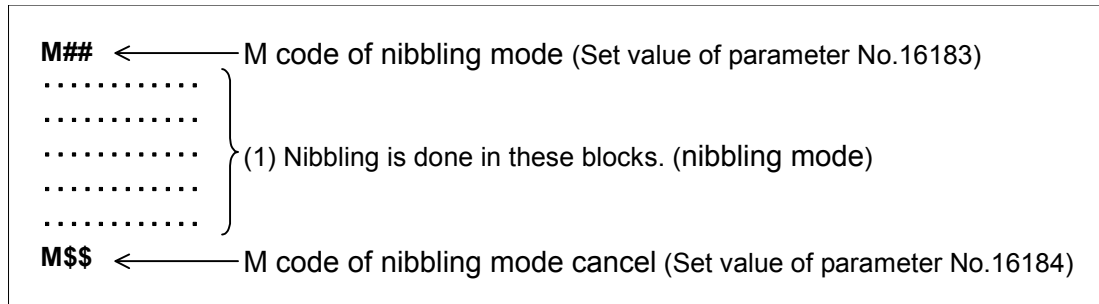


Fig.9.3.3 (b) Incremental programming Immediately after linear nibbling (G69)

- 8 Since radius, straight line length in G68 and G69 are not modal data, they must be commanded correctly in every block.
- 9 It is possible in certain machines that nibbling is not performed by G68 or G69 command, but one-cycle punching is executed at respective positioning points.

9.4 NIBBLING BY M FUNCTION

In addition to the circular or linear nibbling according to the G68 or G69 command, this control can perform nibbling by M function. In other words, it can execute nibbling in the blocks from a block with the M code of nibbling mode to a block with the M code of nibbling mode cancel as shown below.



In this manual, the M code of nibbling mode is described as M12, while the M code of the nibbling mode cancel is described as M13. However, since these M codes may be different from those specified above in certain machine tool builders, you are requested to read these M codes correspondingly according to the manual prepared by these machine tool builders. Don't use this nibbling by M function in a different way other than specified in this manual, since there are certain restriction about its use.



WARNING

Each of the M codes for nibbling mode and nibbling mode cancel must be commanded in a single block.

9.4.1 G00 Command in Nibbling Mode

Example 3

```
N100G00G90Xx1Yy1 ;
N110M12;
N120Xx2Yy2 T ;
N130Xx3Yy3 ;
N140Xx4Yy4 ;
N150Xx5Yy5 ;
N160Xx6Yy6 ;
N170Xx7Yy7 ;
N180M13;
```

- (1) The first punch point of nibbling is commanded in the block next to M12. A T code can also be commanded in this block concurrently.

The first punch point (x2, y2) of nibbling is previously commanded in N120 block, and nibbling is started when the positioning to (x2, y2) has been completed and the motion corresponding to the T code has been finished.

The movement amounts along the X-axis and Y-axis in this block are not restricted by the maximum positioning distances (parameter Nos. 16188, 16184) of the X-axis and Y-axis in the nibbling mode.

- (2) In the blocks after the first punch point of nibbling has been commanded, the positioning command is given to correspond to the pitches by G68 and G69 commands.

If the movement amounts along the X-axis and Y-axis in these blocks exceed the parameter set values (parameter No. 16188), alarm (PS 4521) results.

In N130 to N170 blocks, the positioning corresponding to the nibbling pitch is commanded. If the movement amounts along X-axis x_3-x_2 , x_4-x_3 , x_5-x_4 , x_6-x_5 , x_7-x_6 or the movement amounts of Y-axis y_3-y_2 , y_4-y_3 , y_5-y_4 , y_6-y_5 , y_7-y_6 exceed the parameter set values in each block, alarm (PS 4521) is issued.

- (3) M13 is commanded in the block next to the block where the last punch point of nibbling was commanded.

The last punch point (x_7 , y_7) of nibbling is commanded in the N170 block and M13 is commanded in the next block, i.e. in block N180. Punching is done at (x_n , y_n) and nibbling terminates.

The G00 command in the nibbling mode is given to execute the positioning to the commanded position, and the movement amounts along X-axis and Y-axis should not exceed the parameter set values (No. 16188), except for the command to the first punch point of nibbling.

9.4.2 G01, G02, and G03 Commands in Nibbling Mode

Linear nibbling can be done by commanding G01 in the nibbling mode, while circular nibbling can be done by commanding G02 and G03 in the nibbling mode.

The tool diameter cannot be offset by G01, G02, G03 commands. Accordingly, these commands are used together with cutter compensation commands (G40, G41, G42) when nibbling is done by offsetting a continuous straight line or circular arc leftwards or rightwards by the tool diameter.

(1) Straight line

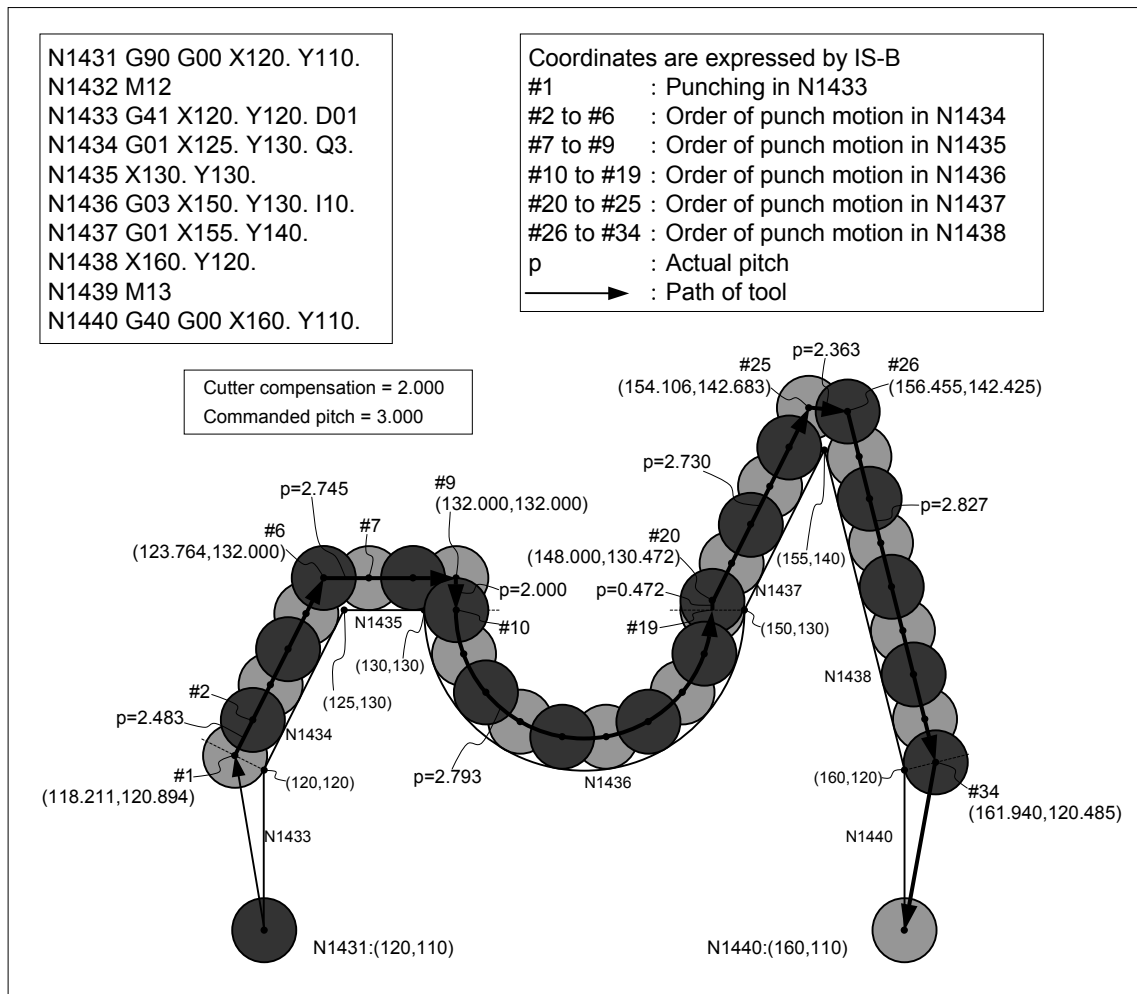
G01 X $\underline{x}$ Y $\underline{y}$ Q $\underline{p}$

- x, y : Specified in the same way as an ordinary G01.
 p : Pitch (Valid data range : 1 to 999999999)
 Unit of data : Input unit (inch/mm)
 -Should be specified by positive number.
 -This pitch is specified as a straight line length.

(2) Circular arc

{ G02 } X $\underline{x}$ Y $\underline{y}$ { I $\underline{i}$ J $\underline{j}$ } Q $\underline{p}$
{ G03 } X $\underline{x}$ Y $\underline{y}$ { R $\underline{r}$ } Q $\underline{p}$

- x, y, i, j, r : Specified in the same way as an ordinary G02 or G03.
 p : Pitch (Valid data range : 1 to 999999999)
 Unit of data : Input unit (inch/mm)
 -Should be specified by positive number.
 -This pitch is specified as an arc length.

Example 4**Fig.9.4.2**

The G40, G41, and G42 codes function as follows.

For details, refer to the section 16.4 “DETAILS OF TOOL RADIUS COMPENSATION”.

G code	Function
G40	Tool radius compensation cancel
G41	Leftward offset for moving direction of tool
G42	Rightward offset for moving direction of tool

Assume that the tool radius (2.0) is preset to offset No. 01.

- (1) The first punch point of nibbling and either G41 or G42 to offset the tool leftwards or rightwards as viewed from the moving direction of the tool are commanded in the block next to M12 block. The first punch point is commanded by G00.

The first punch point (120.0, 120.0) of nibbling is commanded and also G41 is commanded in the N1433 block.

Accordingly, positioning is done at rapid traverse to the point (118.211, 120.894) being offset leftwards to the commanded position by the tool diameter being preset to offset No. 01. Nibbling is started when the positioning has been completed.

- (2) The straight line is commanded by G01, the circular arc is commanded by G02, G03, and the pitch is commanded by Q, starting with the block next to the block where the first punch point was commanded.

Since pitch Q is a modal data, it is no longer needed to designate pitch Q after pitch Q was designated once.

The axis movement to pitch Q is done at rapid traverse.

The straight line and circular arc along which nibbling is done are commanded in N1434 to N1438 blocks. The straight line and circular arc obtained by offsetting the commanded straight line and circular arc leftwards by the tool diameter being preset to offset No. 01, are divided by pitch Q.

The axis movement to pitch is done at rapid traverse.

- (3) M13 is commanded in the block next to the block where the end point of nibbling was commanded. The end point (160.0, 120.0) of nibbling is commanded in N1438 block, and M13 is commanded in the N280 block next to the N1439 block. Nibbling is completed after punching was done at point (161.940, 120.485) being offset leftwards by the tool diameter at end point (160.0, 120.0).

For G01, G02, and G03 in the nibbling mode, the straight line and circular arc are divided by the pitch commanded by address Q, and the movement to respective pitches is done at rapid traverse. If the pitch commanded by address Q exceeds the parameter set value No. 16186: for mm input, No. 16187: for inch input, alarm (PS 4523) is issued.

Now, the difference of the motion will be described when the program shown in example 4 is presumed to have been programmed as shown in example 5.

Example 5

```
N1431 G90 G00 X120. Y110.
N1432 G41 X120. Y120. D01
N1433 M12
N1434 G01 X125. Y130. Q3.
```

```
.....
.....
.....
```

In the program shown in example 4, nibbling is started after the positioning has been completed to (118.211, 120.894). In case of the program shown in example 5, on the other hand, one-cycle punching only is done after the positioning has been completed to point (118.211, 120.894), and nibbling is started after the axis movement by pitch Q along the straight line in N1434 block.

9.4.3 Notes on Nibbling by M Function

CAUTION

- 1 The following commands only are executable in nibbling mode.
 - (i) X, Y positioning command by G00
Provided that the T code and F1-digit command can be included in the same block where the X, Y positioning is made by G00 to the first punch point of nibbling.
 - (ii) G26 (bolt hole circle), G76 (line at angle), G77 (arc), G78, G79 (grid), G86 (share proofs), G87 (square), G88 (radius), G89 (cut at angle)
The movement amounts along the X-axis and Y-axis to respective positioning points should not exceed the parameter set value (No. 16188), except when the first positioning point is equivalent to the first punch point of nibbling.
 - (iii) G01, G02, G03, G41 and G42 commands in the range described in 9.4.2.
- 2 The positioning distance commandable by the X-Y positioning command by G00 is not composite distance $\sqrt{\Delta x^2 + \Delta y^2}$ obtained by the movement amounts along X-axis and Y-axis.
If the absolute value of the movement amount along either X-axis or Y-axis exceeds the parameter set value (No. 16188), alarm (PS 4521) is issued.
This provision also applies to G26, G76, G77, G78, G79, G86, G87, G88 and G89, correspondingly.
- 3 In the Example 4, offset was made by the tool diameter by G41 or G42 to G01, G02 and G03. In that case, the offset straight line or circular arc is divided by the pitch commanded by address Q.
Be careful since the above division differs from such a case that a commanded circular arc is divided by a commanded pitch, like in G68 (circular nibbling).

9.5 EXTERNAL MOTION FUNCTION

Section 9.1 “PUNCH FUNCTIONS (1-CYCLE PRESSING)” explained the blocks, in which punching is made after positioning. In certain cases, no punching is made, but tapping and other mechanical motion may be executed in these blocks.

Example

M80;	}	Tapping is made instead of punching.
G00X__Y__T__;		
X__Y__;		
.....		
.....		
.....		
X__Y__;		
M81		

Since this function depends upon the machine tool builders, refer to the machine tool builder's manual.

10 S FUNCTION

Chapter 10, "S FUNCTION", consists of the following sections:

10.1 SPECIFYING THE S CODE WITH A BINARY CODE	87

10.1 SPECIFYING THE S CODE WITH A BINARY CODE

S code can be specified by address S followed by a binary code. 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.

11 TOOL FUNCTION (T FUNCTION)

Chapter 11, "TOOL FUNCTION (T FUNCTION)", consists of the following sections:

11.1 TOOL SELECTION FUNCTION	88
11.2 T COMMAND NEGLECT	89
11.3 TOOL OFFSET.....	90
11.4 CONTROLLING THE TURRET-AXIS (T-AXIS).....	90
11.5 MULTIPLE TOOL CONTROL.....	90
11.6 TOOL LIFE MANAGEMENT FUNCTION.....	93

11.1 TOOL SELECTION FUNCTION

By specifying an up to 8-digit numerical value following address T, tools can be selected 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 commandable 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:

- (1) Simultaneous execution of the move command and T function commands.
- (2) Executing T function commands upon completion of move command execution.

The selection of either (1) or (2) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

The T command must be given without fail to the block before a block where punching is first made by press motion in one program or to the same block where punching is first made by press motion.

If the T command is not given to these blocks, the press start signal which instructs "Punch by press motion", is not sent to the machine, and machining does not proceed to the next block.

Example

```
O1000G92.....;
N1G00G90X__Y__;
.....
.....
```

Punching should be made in N1 block.

However, since no T command is given to a block before N1 block or N1 block, no punching is made, and machining does not proceed to the next block.

⚠ CAUTION

- 1 The correspondence between commandable T codes and tools depends upon machine tool builders.
The commandable T codes are set in tool registering screen before shipment from factory. If a commanded T code was not registered, alarm (PS 4602) is produced.
- 2 No T code is commandable in the following blocks.
 - (i) G10 (Offset value setting)
 - (ii) G22 (Stored stroke limit function on)
 - (iii) G23 (Stored stroke limit function off)
 - (iv) G92 (Coordinate system setting)
 - (v) G52 (Local coordinate system setting)
 - (vi) G72 (Standard point command)
 - (vii) G75 (Automatic repositioning)
 - (viii) G98 (Base point command for multi-piece machining)
 - (ix) G73, G74 (Multi-piece machining command)
- 3 If DNC operation or memory operation is made in the T command neglect status, the T command is ignored, and the operation is made as if the T command were not given.
The press start signal is not sent to the machine side in a block to be punched, and processing does not proceed to the next block. If a program is checked by marking to a workpiece by using a marking tool, for example, select the marking punch tool by a T command in the MDI mode in advance, and perform the DNC operation or memory operation without reset operation in the T command neglect status. Now, punching is made using the tool selected in the MDI mode.
- 4 The T command must be given without fail to the block before a block where punching is first made by press motion in one program or to the same block where punching is first made by press motion.
If the T command is not given to these blocks, the press start signal which instructs "Punch by press motion", is not sent to the machine, and machining does not proceed to the next block.
- 5 The press start signal can be sent by setting bit 5 (TCF) of parameter No. 16003 even if no T command is given to a block before the block where the punching is made or the block in which punching is made. In this case, the integrated value of the number of punch times may differ from actual number of punch times about respective tools.

11.2 T COMMAND NEGLECT

This function ignores the T command. Whether the T command is ignored or not is generally selected by a switch on the machine operator's panel.

If the T command is ignored, it is treated, as if no T code command were present on a program. Accordingly, it is not checked that whether the T code be commandable or not.

⚠ CAUTION

The ignorance of T command is judged when the command is read from an I/O device or memory into buffer register. Accordingly, the selection of the T command ignorance switch is not effective for the block that has been read into the buffer register.

NOTE

By setting bit 5 (TNM) of parameter No. 16260, it is checked that whether the T code be commandable or not, only in the machine lock status.

11.3 TOOL OFFSET

Tool offset is applicable to respective T codes in the X-axis and Y-axis directions.

Since use of this tool offset function depends upon machine tool builders, refer to the machine tool builder's manual.

⚠ CAUTION

- 1 Tool offset compensation applies to tools numbered from 1 to 9999.
- 2 Tool offset values are set in the tool registering screen in the unit of the least command increment for each X axis and Y axis before shipment of the machine from the machine tool builder's factory.
- 3 If data are inputted by inch in a millimeter system machine, or if data are inputted by millimeter in an inch system machine, a tool offset error is produced within the sum of a half of the least input increment and half of the least command increment. This error is not accumulated.

11.4 CONTROLLING THE TURRET-AXIS (T-AXIS)

The CNC uses set parameters to control the turret which is indexed for a tool to be used. A specified T code is output, and at the same time, the turret is positioned at the location which was specified for the tool on the tool registration screen. Up to 136 tools can be registered on the screen. A T code consisting of one to four digits is used for controlling the turret. This control substantially reduces the time required to change tools because changing the tool, which was conventionally performed in the machine tool, is done by positioning the turret using the CNC. For detailed information, refer to the manual prepared by the machine tool builder.

NOTE

- 1 The T-axis can be controlled when TCL, bit 4 of parameter No. 16260, is 1.
- 2 When T axis control is valid, the T axis should be set as a rotation axis. (bit 0 (ROT_x) of parameter No. 1006 is set to 1)
- 3 Setting the following parameters of T axis allows the CNC to be compatible with FANUC Series 0i-PC, as follows:

Setting	Meaning
Bit 0 (ROT _x) of parameter No.1006=1	Rotary axis (A type)
Bit 1 (ROS _x) of parameter No.1006=0	
Bit 0 (ROA _x) of parameter No.1008=1	The rotary axis roll-over is valid.
Bit 1 (RAB _x) of parameter No.1008=0	The axis rotates in the direction in which the distance to the target is shorter.
Bit 2 (RRL _x) of parameter No.1008=1	Relative coordinates are rounded by the amount of the shift per one rotation

11.5 MULTIPLE TOOL CONTROL

In general, the tool holder of a punch holds one tool (die). To select a tool the tool holder is first moved to the position at which the tool is changed using the T command (cartridge indexing). Then, at that position, the tool holder is set on the punch holder.

In the case of the multiple-tool system, only the dies in the tool holder are changed because the tool holder keeps different types of dies (See Fig.11.5 (a)). The multiple-tool system enables the tool holder to change dies very quickly. Also, many dies can be used to widen the range of machining applications.

For details, refer to the manuals issued by machine tool builder.

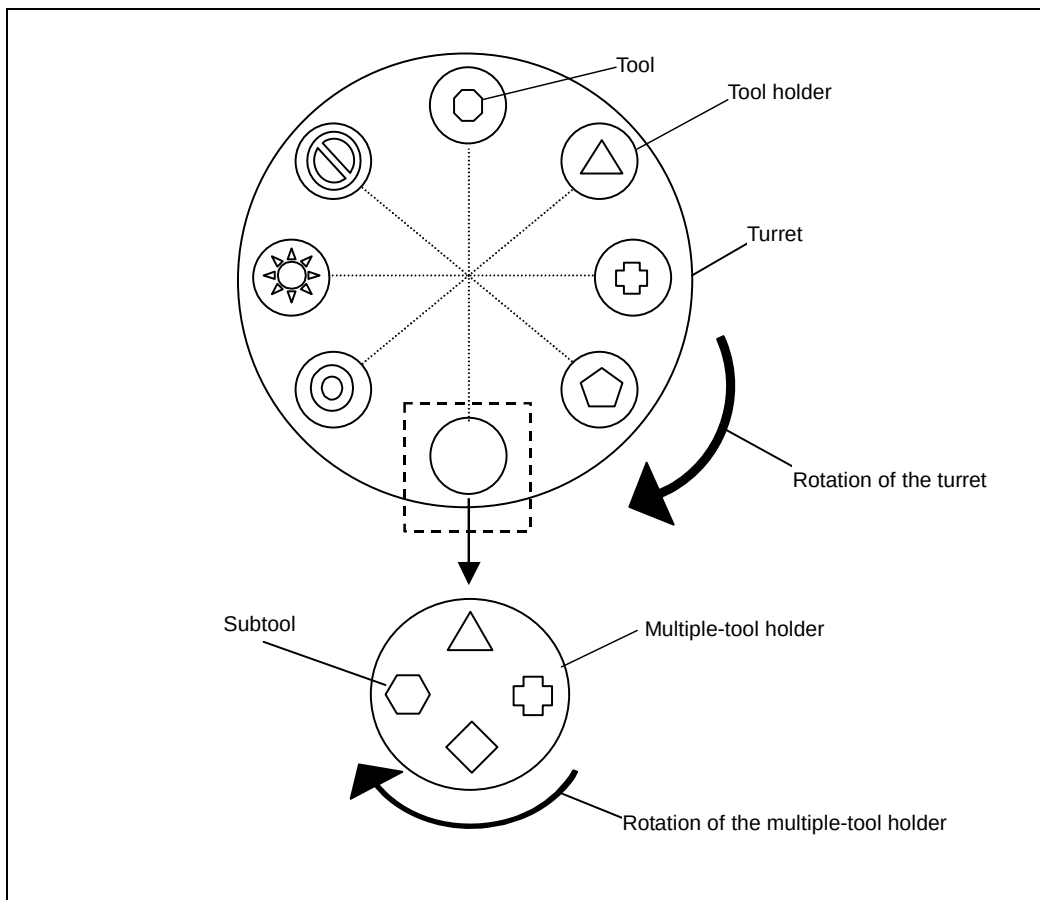
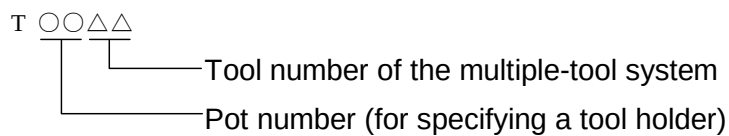


Fig.11.5 (a) Multiple-tool system

11.5.1 Tool Number

The pot numbers of a multiple-tool system are specified with T codes consisting of three or four digits, as follows:



The T codes used for controlling the indexing of the turret (tool holder) in a CNC are two-digit pot numbers from 00 to 99, including those for a multiple-tool system as shown above.

The T codes consisting of three or four digits are used to control a multiple-tool system. The first two digits are the pot numbers of the multiple-tool system and the last two digits are the tool numbers in the multiple-tool system.

NOTE

- 1 Tool numbers for controlling the turret setting in a CNC system are registered in tool registering screen. The number of tools is less than or equal to the number of the tools to be controlled.
- 2 Assign tool numbers for controlling a turret which is not that for a multiple-tool system as T codes consisting of one or two digits.
- 3 Assign tool numbers of a multiple-tool system on the tool registration screen for multiple tools.

11.5.2 Relationship between the Multiple-Tool System and the C Axis

The tools of a multiple-tool system are selected by turning the C axis. A tool is selected by placing it at the tool reference position. This position is parallel to the Y axis and on the center line of the tool holder of a multiple-tool system.

For each tool in a multiple-tool system, the angle the tool makes with the tool reference position is set in tool registering screen for. When the T code of a multiple-tool system is issued, the CNC system sends an internal command to the C axis, indicating the angle with the tool reference position corresponding to the selected tool. The CNC system places the selected tool in the tool reference position.

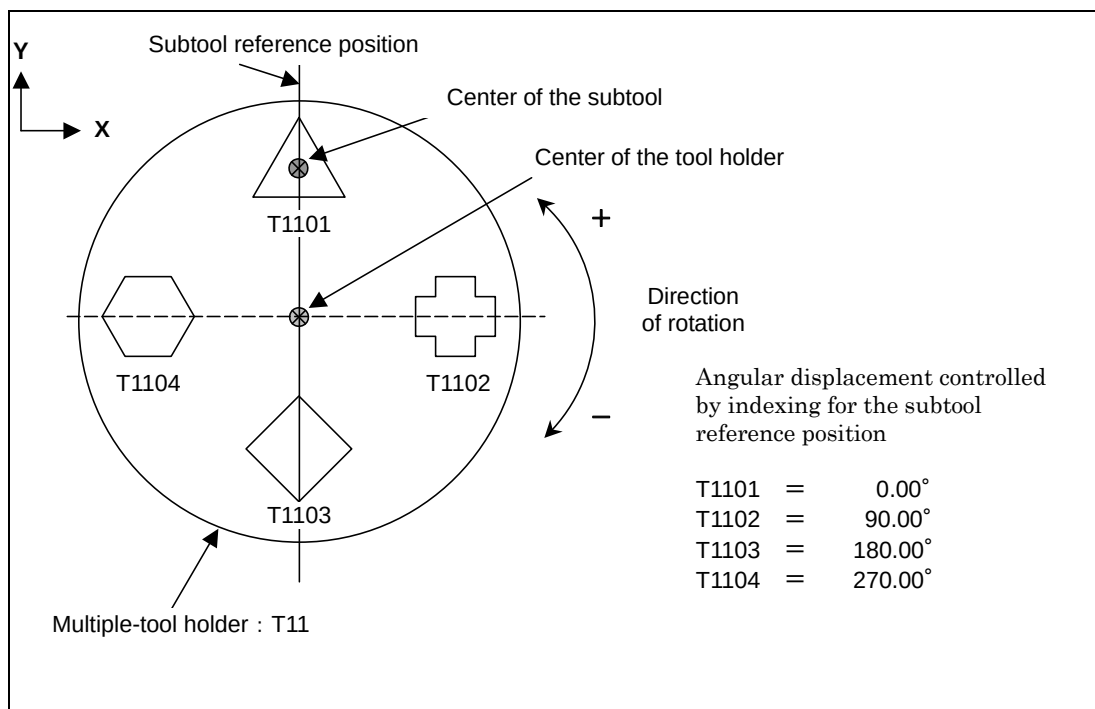


Fig.11.5 (b) Subtool reference position and angular displacement controlled by indexing

**CAUTION**

When the angular displacement for the tool reference position is set to 0, the tool that is selected when the tool holder has been indexed is used.

11.5.3 Tool Compensation

When a multiple-tool system is used, the centers of the dies in the multiple-tool system are not at the center of the tool holder. Therefore, tool compensation is necessary. Tool compensation for tools in a multiple-tool system works in the direction of the Y axis only. Specify the distance between the center of

the tool holder and each of the tools when the tool is located in the tool reference position in tool registering screen. The distance is the compensation value in the direction of the Y axis.

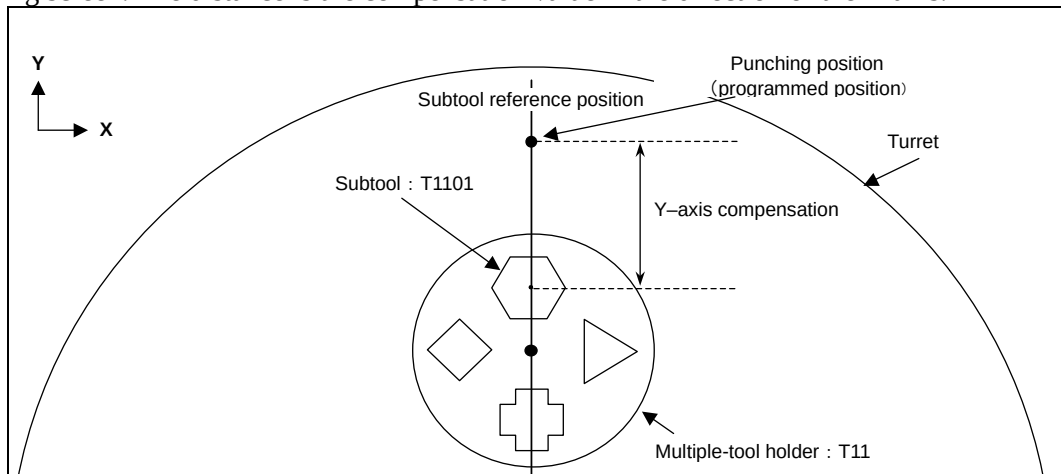


Fig.11.5 (c) Tool position compensation

11.5.4 Operational Notes



CAUTION

- 1 Do not issue any C axis command when multiple-tool control is selected.
- 2 For indexing a multiple-tool system, T codes consisting of at least three digits must be used. For using regular tools, T codes consisting of one or two digits must be used.
- 3 When using multiple-tool control, set bit 5 (CRM) of parameter No. 16362 to 0.

11.6 TOOL LIFE MANAGEMENT FUNCTION

Tools are classified into various groups, with the tool life (frequency of use) for each group being specified.

11.6.1 Tool Life Management Data

Tool life management data consists of number of punches and tool life value.

Explanation

- Number of punches

The number of punches count is incremented by 1 for each tool used punching.

11.6.2 Register and Change of Tool Life Management Data

In a program, tool life management data can be registered in the CNC unit, and registered tool life management data can be changed.

Format

G10L30 ;	Tool data entry mode setting
N_P7R_ ;	tool life management data setting
:	
:	
G11 ;	Tool data entry mode cancel
N_ :	Tool data No.
P7 :	Tool life data setting
R_ :	Tool data setting value

Explanation**- Types**

For tool life management, the type used varies from one machine to another. For details, refer to the appropriate manual of each machine tool builder.

11.6.3 Tool Life

The life of a tool is specified by a usage frequency (count).

12 AUXILIARY FUNCTION

Overview

There are two types of auxiliary functions; auxiliary function (M code) for specifying program end and so on, and secondary auxiliary function (B code).

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 12, "AUXILIARY FUNCTION", consists of the following sections:

12.1 AUXILIARY FUNCTION (M FUNCTION)	95
12.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK	97
12.3 SECOND AUXILIARY FUNCTIONS (B CODES)	98
12.4 M COMMAND FOR SWITCHING THE PUNCH AND LASER MODE.....	100

12.1 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 following:

- Subprogram call and end of subprogram (M98/M99)
- External subprogram call (M198 or parameter No.6030)
- M codes to call subprogram (parameter No.6044 to 6046, and No.6071 to 6079)
- M codes to call custom macro program (parameter No.6047 to 6049, and 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 Section 14.3, "SUBPROGRAM (M98, M99)" 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 Section 14.3, "SUBPROGRAM (M98, M99)" for details.

- **M08, M09 (Forming mode and forming mode cancel)**

If punching (excluding nibbling) is executed in a block between M08; and M09; it is executed when the preset time by parameter (No. 16032) has passed after completion of positioning, and machining proceeds to the next block when the preset time by parameter (No. 16033) has passed after receiving the punch finish signal from the machine side. (Other M codes may be used for these functions depending upon the machine tool builders.)

- **M10, M11 (Workpiece clamp and workpiece unclamp)**

The movement amounts of the X and Y axes are not added to the absolute coordinate value in a block between by M10; and M11;. No punching is executed, even if punching is specified in the block.

They are used to reposition a workpiece. (Other M codes may be used for these functions, depending upon machine tool builders.)

- **M12, M13 (Nibbling mode and nibbling mode cancel)**

Nibbling is executable in a block between M12; and M13;. (Other M codes may be used for these functions depending upon machine tool builders)

**WARNING**

M08, M09, M10, M11, M12 and M13 must be commanded without any other commands in a block.

⚠ CAUTION

- 1 No punching is made in a block where an M code is commanded. But punching may be made by some machine tool builder.
- 2 No M code is commandable in the following blocks.
 - (i) G10 (Offset amount setting, Programmable parameter input)
 - (ii) G22 (Stored stroke limit function ON)
 - (iii) G23 (Stored stroke limit function OFF)
 - (iv) G26 (Bolt hole circle)
 - (v) G76 (Line at angle)
 - (vi) G77 (Arc)
 - (vii) G78, G79 (Grid)
 - (viii) G68 (Circular nibbling)
 - (ix) G69 (Linear nibbling)
 - (x) G72 (Base point command)
 - (xi) G75 (Automatic repositioning)
 - (xii) G86 (Share proofs)
 - (xiii) G87 (Square)
 - (xiv) G88 (Radius)
 - (xv) G89 (Cut at angle)
 - (xvi) G98 (Base point command of Multi-piece machining)
 - (xvii) G73, G74 (Multi-piece machining command)
 - (xviii) G52 (Local coordinate system setting)

NOTE

- 1 If there is a block following M00, M01, M02, or M30, it is not read into the buffer storage. Similarly, ten M codes which cause the block following them not to enter the buffer storage are available by parameter setting (Nos. 3411 to 3421). As for these M codes, refer to the machine tool builder's manual.
- 2 For M98 code and M99 code, their code signals and strobe signals are not output to the machine.

12.2 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; G28G91X0Y0Z0; : : :	M40M50M60; G28G91X0Y0Z0; : : : :

12.3 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

- 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, "ILLEGAL USE OF DECIMAL POINT" is issued.

When the second auxiliary function is specified with a negative value, alarm PS0006, "ILLEGAL USE OF MINUS SIGN" 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 12.3 (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 12.3 (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 12.3 (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		Bit 0 (AUX) of parameter No.3405 = 0	Bit 0 (AUX) of parameter No.3405 = 1
Metric input system	Reference axis: IS-A	100×	100×
	Reference axis: IS-B	1000×	1000×
Inch input system	Reference axis: IS-A	100×	1000×
	Reference axis: IS-B	1000×	10000×



CAUTION

If a decimal fraction remains after multiplying the specified value with a decimal point by a magnitude in Table 12.3 (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, "TOO MANY DIGIT" is issued.

When the specified value is multiplied by a magnitude in Table 12.3 (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.

12.4 M COMMAND FOR SWITCHING THE PUNCH AND LASER MODE

The punch mode and laser mode can be switched by specifying M codes in parameters. An M code is specified in the first block for punching and for laser machining in a machining program. This will improve processing precision in the interpolation mode of the laser mode.

In the laser mode, punching is not performed even in blocks where positioning is done by rapid traverse along the X and Y axes. Manual punching, however, is possible. Note that pattern commands such as G26 and nibbling commands such as G68 cannot be specified. (These commands cause alarm (PS4630).)

(Example)

G92X1000. Y800. ;	
M40 ;	
G72X200. Y200. ;	} Punch mode
G26I50. J0K10. T101;	
M41;	
G90G00X100. Y100. ;	} Laser mode
G01X300. F300;	
Y300. ;	
X100. ;	
Y100. ;	
M40 ;	
G28M30 ;	

⚠ WARNING

- 1 The punch mode and laser mode can be switched only when bit 0 (ALA) of parameter No. 16242 is set to 1.
Set the M code to the parameter (No. 16244, 16245).
- 2 The punch mode is entered immediately after the power is turned on or the system is reset. (Laser mode is entered when bit 5 (RLM) of parameter No. 16240 is set.)
- 3 The M codes for switching the punch and laser modes must be specified in a single block.
- 4 In punch mode, AI Advanced Preview Control function cannot be used.

13 PROGRAM MANAGEMENT

Chapter 13, "PROGRAM MANAGEMENT", consists of the following sections:

13.1 PROGRAM ATTRIBUTES.....	101
13.2 RELATED PARAMETERS	101
13.3 PART PROGRAM STORAGE SIZE / NUMBER OF REGISTERABLE PROGRAMS	102

13.1 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.

13.2 RELATED PARAMETERS

This subsection lists the meanings of parameters related to program numbers and the folders and programs to be manipulated or executed.

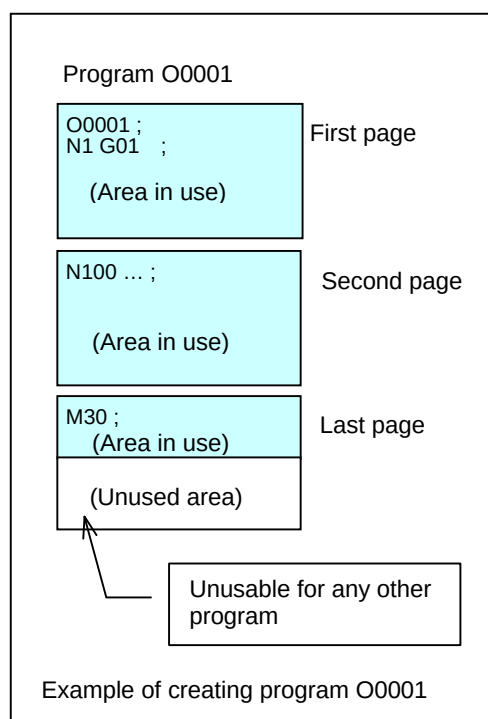
Parameter No.	Bit No.	Description
3202	0 (NE8)	Disables or enables editing of programs O8000 to O8999.
	4 (NE9)	Disables or enables editing of programs O9000 to O9999.
3210/3211	-	Password/keyword for protecting programs in the nine thousands
3404	2 (SBP)	In the subprogram call function, address P in the M198 block specifies a file/program number.
6001	5 (TCS)	Calls or does not call a custom macro by T code.
6038 to 6040	-	Start G code used to call custom macros, Start program number of custom macros called by G code, and Number of G codes used to call custom macros
6044 to 6046	-	Start M code used to call subprograms, Start program number of subprograms called by M code, and Number of M codes used to call subprograms
6047 to 6049	-	Start M code used to call custom macros, Start program number of custom macros called by M code, and Number of M codes used to call custom macros
6050 to 6059	-	G code for calling a custom macro with program No. 9010 to 9019
6071 to 6079	-	M code for calling a subprogram with program No. 9001 to 9009
6080 to 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

13.3 PART PROGRAM STORAGE SIZE / NUMBER OF REGISTERABLE PROGRAMS

The following table lists the combinations of program storage sizes and the total number of registerable programs.

Part program storage size	Number of registerable programs	0i-PD
512Kbyte	400	○
2Mbyte	400	☆

○: Standard package (1) ☆: Optional



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 0i-PD manages 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 0i-PC uses a similar way of management, but the unit of pages in it differs from that in the Series 0i-PD. So, if more than one program is registered in the Series 0i-PD, the total program size of registerable programs differs from that in the Series 0i-PC.

14 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.

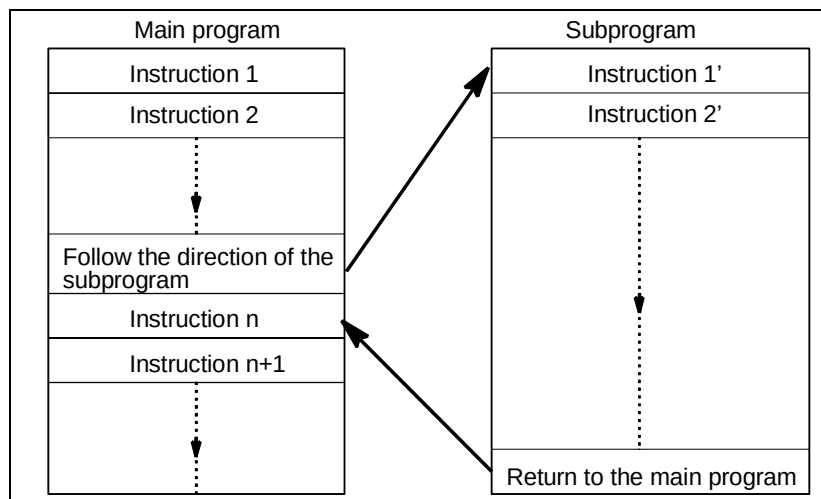


Fig. 14 (a) Main program and Subprogram

The CNC memory can hold up to 400 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. See Chapter III-9, “CREATING PROGRAMS” and Chapter III-10, “EDITING PROGRAMS” for the methods of registering and selecting programs.

- Program components

A program consists of the following components:

Table 14 (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

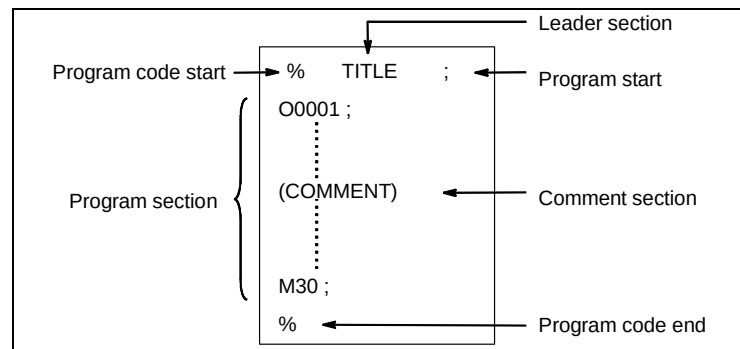


Fig. 14 (b) Program configuration

- Program section configuration

A program section consists of several blocks. A program section starts with a program number or file name 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 G00 X00. T02 ;
:	:
Block n	Nn M2 ;
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-14.2).

14.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See Section II-14.2, "PROGRAM SECTION CONFIGURATION" for a program section.

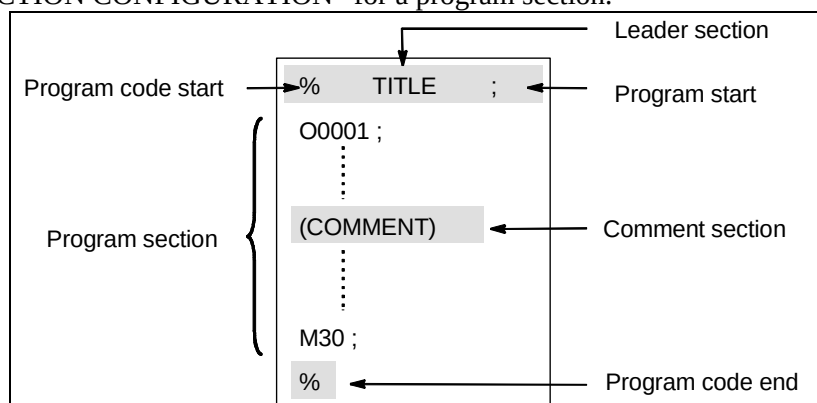


Fig. 14.1 (a) Program configuration

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 14.1 (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 a leader section is skipped, even a TV parity check is not made. So a leader section can contain any codes except the EOB code.

- 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 14.1 (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 14.1 (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 B are ignored, and thus are not read into memory.

When data in memory is output on external I/O device (See Chapter III-8, "DATA INPUT/OUTPUT"), 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.

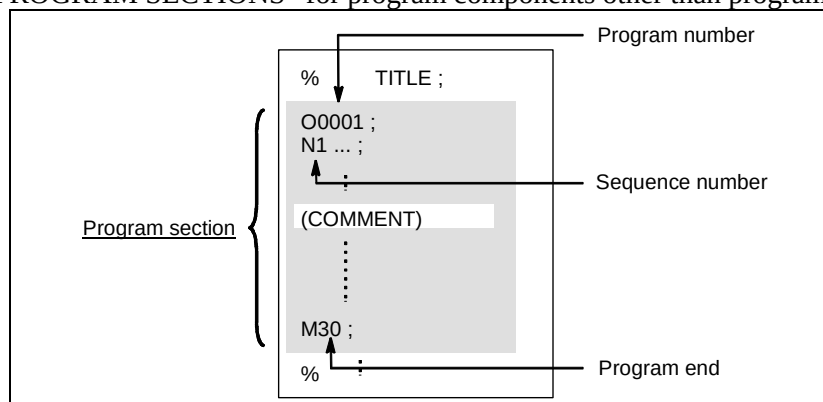
If an attempt is made to execute % when M02 or M30 is not placed at the end of the program, the alarm PS5010, "END OF RECORD" is occurred.

Table 14.1 (d) Code of a program code end

Name	ISO code	EIA code	Notation in this manual
Program code end	%	ER	%

14.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section, "PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS" for program components other than program sections.

**Fig. 14.2 (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 Section III-8.2, "INPUT/OUTPUT ON EACH SCREEN" or Section III-9.1, "CREATING PROGRAMS USING THE MDI UNIT")

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 14.2 (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.)

N300X200.OZ300.0 ; A sequence number is underlined.

Fig. 14.2 (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, an alarm PS0002, "TV ERROR" 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 panel (See Subsection III-12.3.2, "Displaying and Entering Setting Data").

- 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 14.2 (b) indicates the usable addresses and their meanings.

The same address may have different meanings, depending on the preparatory function specification.

Table 14.2 (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
S function	S	S code
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, K	Parameters for pattern function and Nibbling function
Offset number	D	Offset number
Dwell	P, X	Dwell time
Macro function	A,B,U,V,W	Macro number

NOTE

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

<u>N</u>	<u>G</u>	<u>X</u> <u>Y</u>	<u>F</u>	<u>S</u>	<u>T</u>	<u>M</u> ;
Sequence number	Preparatory function	Dimension word	Feed-function	S function	Tool function	Auxiliary function

Fig. 14.2 (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 14.2 (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
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.
Feed per minute	Increment system IS-A	F	0.01 to 999000.00mm/min	0.001 to 96000.000inch/min
	Increment system IS-B		0.001 to 999000.000mm/min	0.0001 to 9600.0000inch/min
S 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		D	0 to 32	0 to 32
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
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
Angle of pattern function / nibbling	Increment system IS-A	J, P, K	±999999.99 deg.	±999999.99 deg.
	Increment system IS-B		±999999.999 deg.	±999999.999 deg.
Number of punch points of pattern function		K, P	1 to 9999	1 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

*4 The maximum value of addresses M, T, and B is 99999999, and, the maximum value of address S is 99999.

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

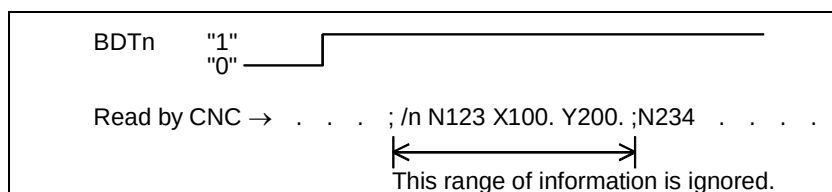
Input signal	Start code to be ignored
BDT1	/ or /1 ^(NOTE)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

NOTE

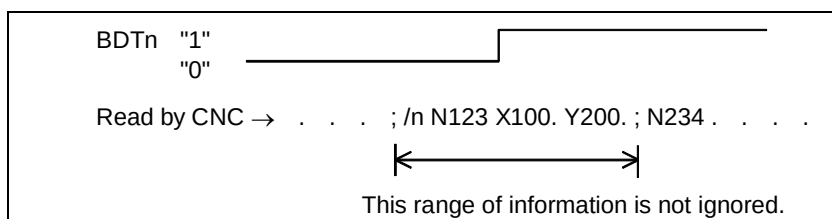
- 1 Number 1 for /1 can be omitted. However, when two or more optional block skips are specified for one block, number 1 for /1 cannot be omitted.
- 2 Depending on the machine tool, all optional block skip signals (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which 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.

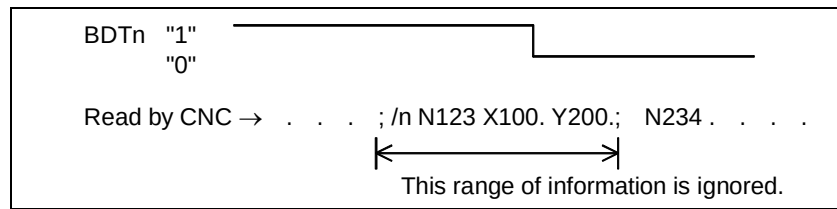
1. When the signal BDTn is set to 1 before the CNC starts reading a block that contains /n, the block is ignored.



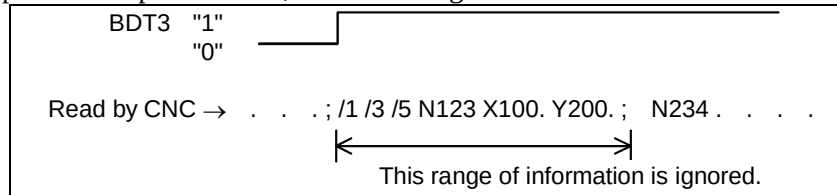
2. When the signal BDTn is set to 1 while the CNC is reading a block that contains /n, the block is not ignored.



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 block skip slash (/) must be specified at the beginning of a block. Note that 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 14.2 (d) Code of a program end

Code	Meaning usage
M02	For main program
M03	
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".)

14.3 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 ; : M99;	Subprogram number or subprogram file name (or the colon (:)) optionally in the case of ISO) Program end

M99 need not constitute a separate block as indicated below.

Example) X100.0Y100.0M99;

- Subprogram call

- When a subprogram with a 4-digit or shorter program number is called

M98 Pxxxx xxxx ;



or

M98 Pxxxx Lxxxxxxxx ;

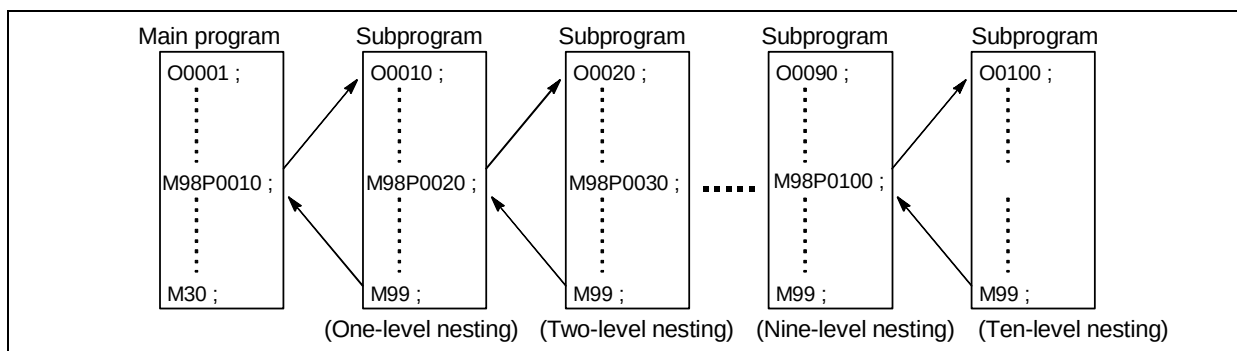


NOTE

- 1 When calling a subprogram of a 4-digit or shorter subprogram number repeatedly (P8 digit), make the subprogram number 4 digits by prefixing it with "0" if the subprogram number is shorter than 4 digits.
Example)
 P100100: Call subprogram No. 100 ten times.
 P50001: Call subprogram No. 1 five times.
- 2 If the number of times the subprogram is called repeatedly is omitted, the subprogram is called just once.
 In this case, it is not necessary to adjust the subprogram number length to 4 digits as described in Item 1 above.
- 3 When calling a subprogram repeatedly (P8 digit), 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, "SEQUENCE NUMBER NOT FOUND" is output.

Example

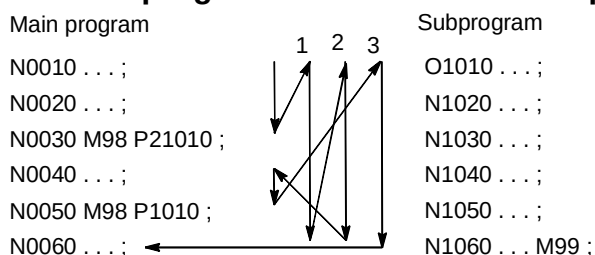
M98 P51002 ;

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 movement.

Execution sequence of subprograms called from a main program



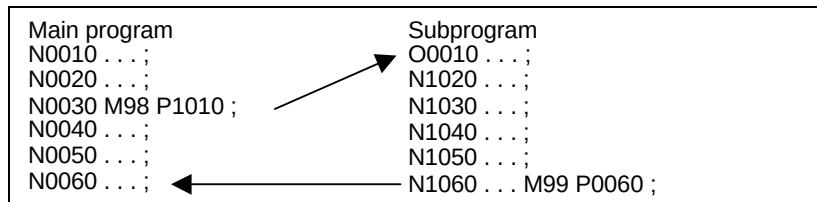
A subprogram can call another subprogram in the same way as a main program calls a subprogram.

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 operating 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.



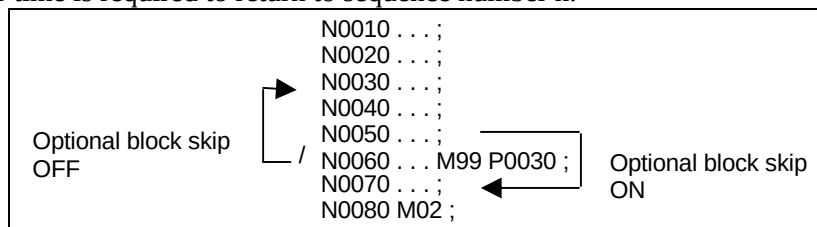
- Using M99 in the main program

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.

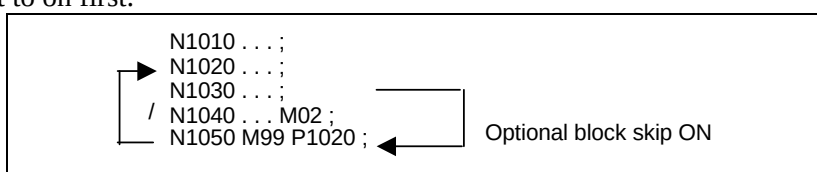


- Using a subprogram only

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.

(See Section III-10.4, "PROGRAM SEARCH" 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.

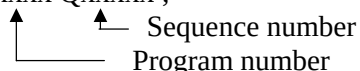


- Subprogram call with sequence number

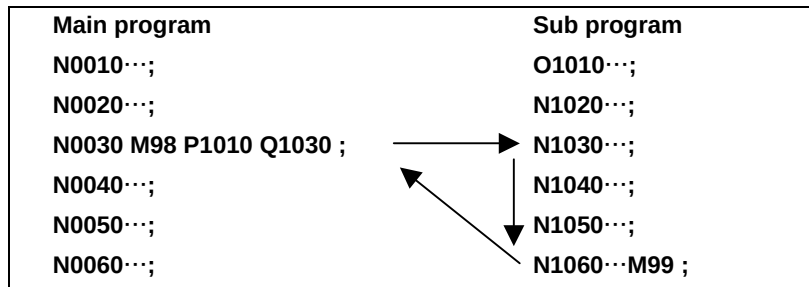
Setting bit 0 (SQC) of parameter No. 6005 to 1 can call a specified sequence number in the subprogram for execution.

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 ;



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.



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, “TOO MANY SUB,MACRO NESTING” is issued.



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.

NOTE

If bit 0 (SQC) of parameter No. 6005 is "0", and an M98 Pxxxx Qxxxxx command is specified, alarm PS0009, "IMPROPER NC-ADDRESS", is issued.

15 FUNCTIONS TO SIMPLIFY PROGRAMMING

Chapter 15, "FUNCTIONS TO SIMPLIFY PROGRAMMING", consists of the following sections:

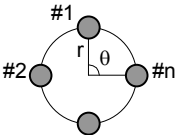
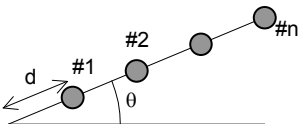
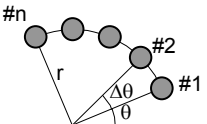
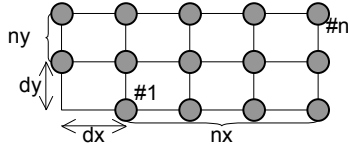
15.1 PATTERN FUNCTION.....	116
15.2 MEMORY AND CALL BY A/B MACRO.....	134
15.3 AUTOMATIC REPOSITIONING (G75).....	135
15.4 U/V/W MACRO FUNCTION.....	141
15.5 MULTI-PIECE MACHINING FUNCTION.....	145
15.6 BENDING COMPENSATION (G38, G39).....	152
15.7 LINEAR AND CIRCULAR PUNCH COMMAND.....	154
15.8 Y-AXIS CRACK CANCEL.....	159

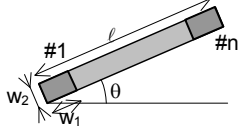
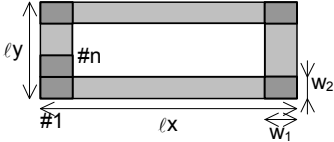
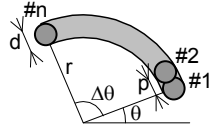
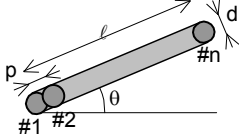
15.1 PATTERN FUNCTION

The pattern function means a function to punch multiple positions conforming to a certain format by one-block command including G function. This pattern function requires only one block command instead of several-block commands, and thus, its program is simplified.

This control system prepares the following patterns as the standard functions, which are used most frequently.

Table15.1 Pattern functions

G code system A	G code system B	Pattern function	Patterns
G26	G26	Bolt hole circle -Punching specified times at constant pitch (angle) on the circle. G26 I <u>r</u> J <u>θ</u> K <u>n</u>	
G76	G28	Line at angle -Punching specified times at constant pitch (distance) on the straight line. G76 I <u>d</u> J <u>θ</u> K <u>n</u>	
G77	G29	Arc -Punching specified times at constant pitch (angle) on the circular arc. G77 I <u>r</u> J <u>θ</u> P <u>Δθ</u> K <u>n</u>	
G78 G79	G36 G37	Grid -Punching specified times in grid points having specified interval. G78 I <u>dx</u> J <u>dy</u> P <u>nx</u> K <u>ny</u> G79 I <u>dx</u> J <u>dy</u> P <u>nx</u> K <u>ny</u>	

G code system A	G code system B	Pattern function	Patterns
G86	G66	Share proofs -Punching straight G86 I <u>ℓ</u> J <u>θ</u> P <u>w₁</u> Q <u>w₂</u>	
G87	G67	Square -Punching a rectangle G87 I <u>ℓx</u> J <u>ℓy</u> P <u>w₁</u> Q <u>w₂</u>	
G88	G78	Radius -Punching specified times at constant pitch (distance) along the circular arc that have specified angle. G88 I <u>r</u> J <u>θ</u> K <u>Δθ</u> P <u>d</u> Q <u>p</u>	
G89	G79	Cut at angle -Punching specified times at constant pitch (distance) along the straight line that have specified distance. G89 I <u>ℓ</u> J <u>θ</u> P <u>d</u> Q <u>p</u>	

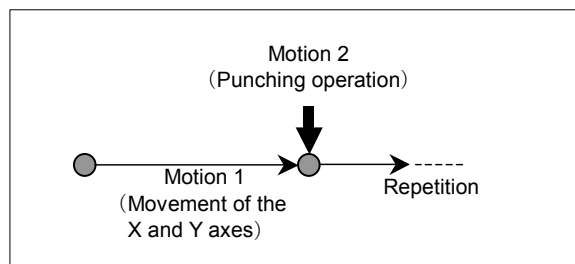
NOTE

G26, G76, G77, G78, G79, G86, G87, G88 and G89 are one-shot G codes.

In pattern function, the following two motions are repeatedly done to punch at respective positions.

Motion 1 ... Positioning of X, Y axes (rapid traverse)

Motion 2 ... Punch by press motion



15.1.1 Base Point Command (G72)

Of the pattern function, the center point of the bolt hole circle (G26), arc (G77) and radius (G88), and also the start point of line at angle (G76), grid (G78, G79), share proofs (G86), Square (G87) and cut at angle (G89) are called base point of pattern.

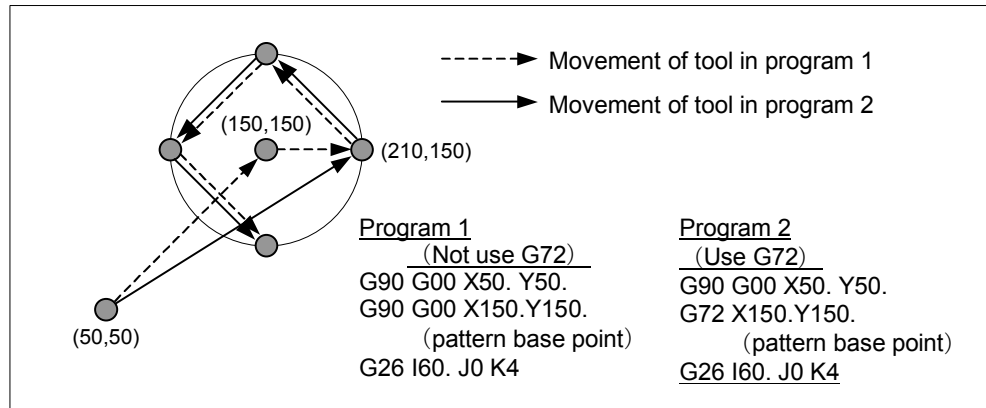
The present position of the tool when the pattern function was commanded, i.e. the tool position in the block just before the pattern function command, is the base point of pattern. If it is desired to set another position as the pattern base point, the base point of pattern is assignable by the following command.

G72X x Y y;

The (x, y) point in the work coordinate system is the pattern base point in absolute (G90) programming, while the point distant from the present position by (x, y) is the reference point in incremental (G91) programming.

G72 specifies the pattern base point, and the tool does not move.

Example



⚠ CAUTION

- 1 None of T code, M code, and F code is commandable in G72 block.
- 2 G72 is one-shot G code.

15.1.2 Bolt Hole Circle (G26)

Bolt Hole Circle (G26) specifies punching the specified times at the constant pitch (angle) on the circular arc with the pattern base point being set as the circle center.

Format

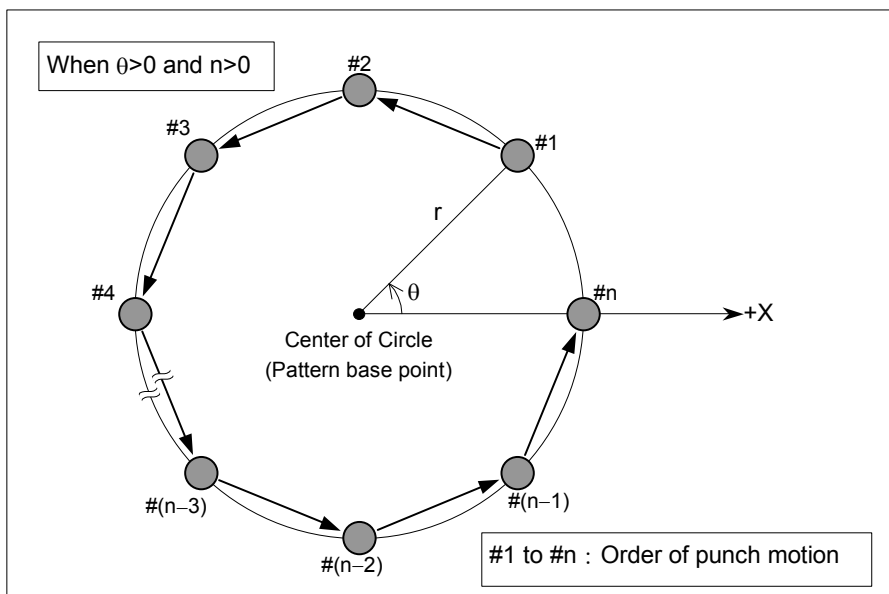
G26 I_r J_θ K_n

r : Radius of arc (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.

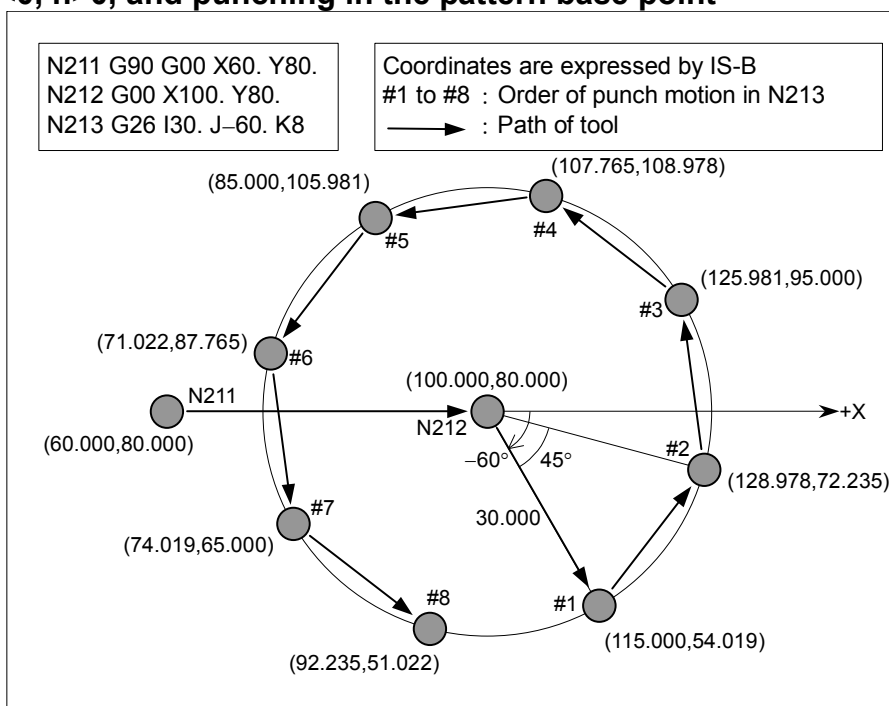
θ : Angle between the first punch point and +X axis (Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : Least input increment (deg)
-The counterclockwise direction is commanded as positive. While, clockwise direction is commanded as negative.

n : Number of punch points (Valid data range : ±1 to ±9999)
-Counterclockwise punching is made by a positive number, while clockwise punching is made by a negative number.

This G26 punches n pieces of equally divided points on the circumference, starting with the point which forms angle θ with the reference to X axis on the circumference having radius r with the pattern base point being set as the circle center.

**Example**

- When $\theta < 0$, $n > 0$, and punching in the pattern base point



If it is desired to punch the center of the circle, omit G72 of block N521.

NOTE

- 1 If the radius is 0 or the number of punch points is 0, an alarm (No. 4502) is issued.
- 2 T and C commands are possible in same block of G26.

15.1.3 Line at Angle (G76)

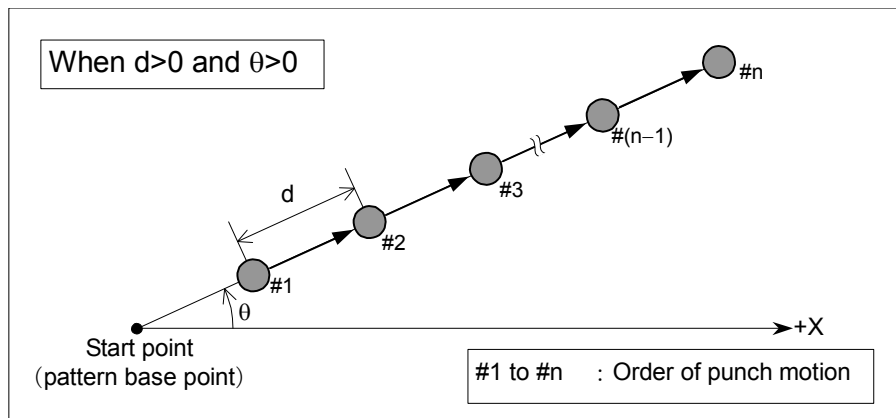
Line at angle (G76) specifies punching the specified times at the constant pitch (distance) on the straight line with the pattern base point as the start point.

Format

G76 I d J θ K n

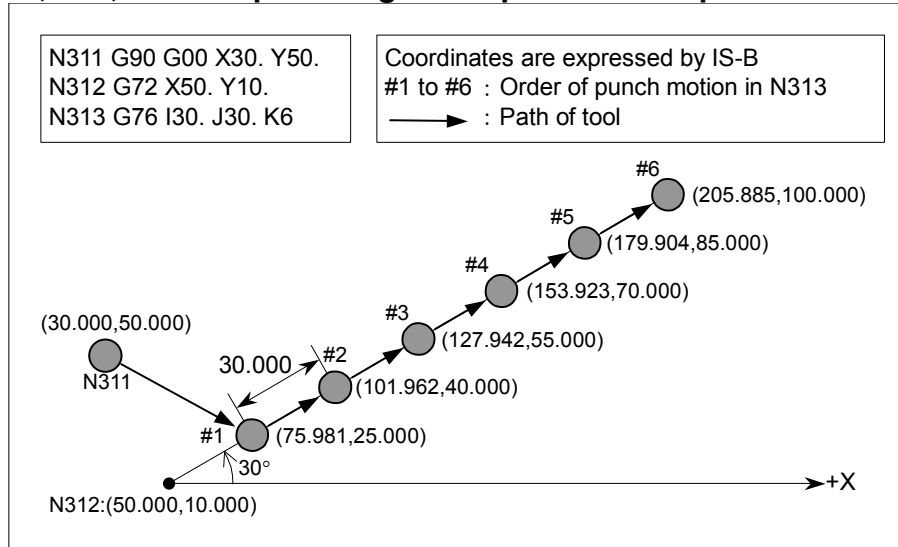
- d : Punch point interval (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-If this value is commanded with positive, punching is made in the θ direction.
-If this value is commanded with negative, punching is made in the $\theta + 180^\circ$ direction.
- θ : Angle which is formed between the +X axis and the line connecting the start point and punch points (Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : Least input increment (deg)
-Counterclockwise direction is commanded by a positive number.
-Clockwise direction is commanded by a negative number.
- n : Number of punch points (Valid data range : 1 to 9999)
-Should be specified by positive number.

By the above command, punching is made at n pieces of points which lie every intervals of d along the straight line which forms angle θ with reference to the X axis, starting with the pattern base point as the start point.



Example

- When $d>0$, $\theta>0$, and not punching in the pattern base point

**NOTE**

If the number of punch points is 0, alarm (PS4503) is produced.

15.1.4 Arc (G77)

Arc (G77) specifies punching the specified times at the constant pitch (angle) on the circular arc with the pattern base point as the circle center.

Format**G77 I_r J_θ P_{Δθ} K_n**

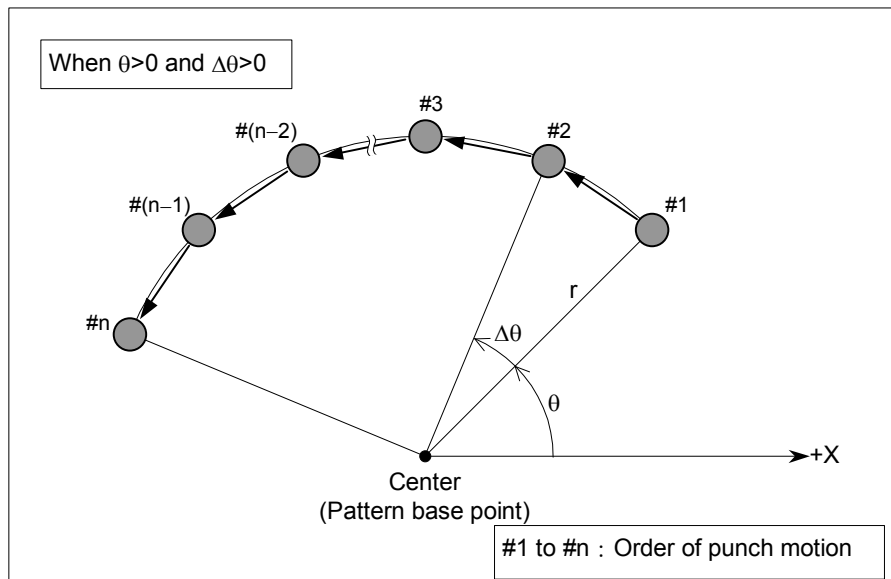
r : Radius of arc (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.

θ : Angle formed between the first punch point and the +X axis (Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : Minimum unit of data (deg)
-Counterclockwise punching is commanded by a positive number.
-Clockwise punching is commanded by a negative number.

Δθ : Angle formed between adjacent punch points axis (Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : Minimum unit of data (deg)
-Counterclockwise punching is commanded by a positive number.
-Clockwise punching is commanded by a negative number.

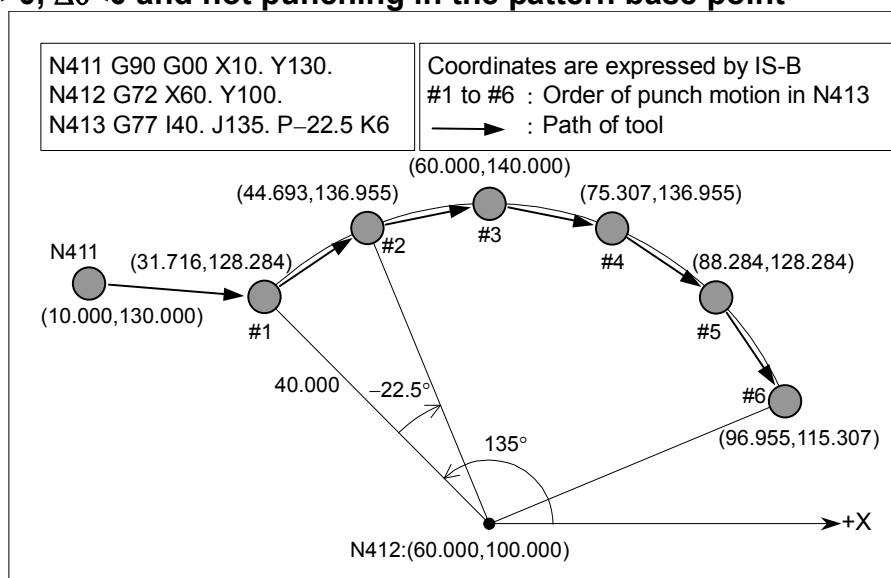
n : Number of punch points (Valid data range : 1 to 9999)
-Should be specified by positive number.

By the above command, punching is made at n pieces of points every incremental $\Delta\theta$ angle, starting with the point which forms θ angle with reference to the X-axis on the circumference of radius r, with the pattern base point being set as the circle center.



Example

- When $\theta > 0$, $\Delta\theta < 0$ and not punching in the pattern base point



NOTE

If the radius is 0 or the number of punch points is 0, alarm (PS 4504) is produced.

15.1.5 Grid (G78, G79)

Grid (G78, G79) specifies punching the specified times in the grid points having specified interval with the pattern base point as the start point.

G78 commands punching in the X-axis direction, while G79 commands punching in the Y-axis direction.

Format

G78 I dx J dy P nx K ny
G79 I dx J dy P nx K ny

dx : Punch point intervals in the X-axis direction (Valid data range : -999999999 to 999999999)

Unit of data : Input unit (inch/mm)

-This is commanded by a positive number when the first punch point in the X-axis direction is located in the +X direction as viewed from the start point.

-This is commanded by a negative number when the first punch point in the X-axis direction is located in the -X direction as viewed from the start point.

dy : Punch point intervals in the Y-axis direction (Valid data range : -999999999 to 999999999)

Unit of data : Input unit (inch/mm)

-This is commanded by a positive number when the first punch point in the Y-axis direction is located in the +Y direction as viewed from the start point.

-This is commanded by a negative number when the first punch point in the Y-axis direction is located in the -Y direction as viewed from the start point.

nx : Number of punch points in the X-axis direction (Valid data range : 1 to 9999)

-Should be specified by positive number.

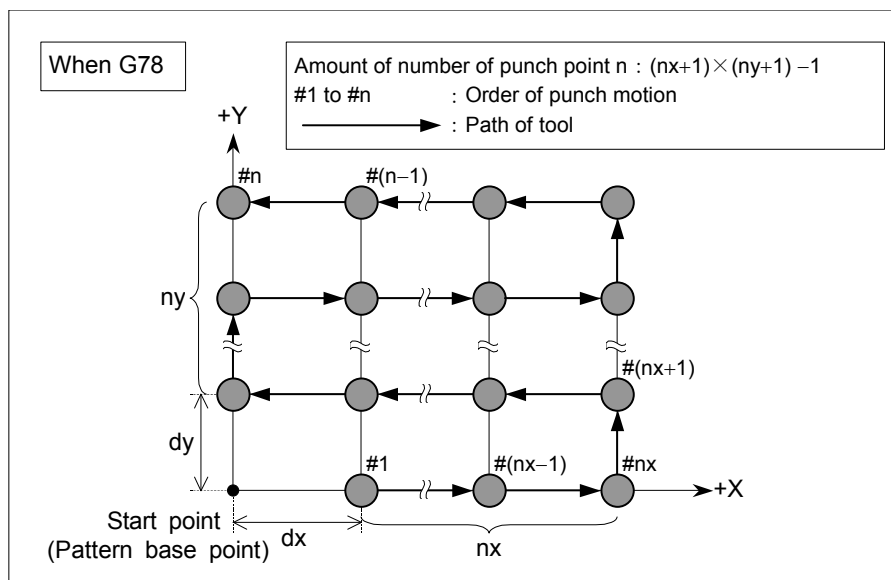
-The start point is not included in the number.

ny : Number of punch points in the Y-axis direction (Valid data range : 1 to 9999)

-Should be specified by positive number.

-The start point is not included in the number.

By the above command, punching is made at matrix points consisting of nx pieces at intervals of dx in the X-axis direction and ny pieces at intervals of dy in the Y-axis direction, starting with the pattern base point being set as the start point.



15.1.6 Share Proofs (G86)

Share proofs (G86) specifies punching for the specified distance along straight line starting with the pattern base point by using specified tool that figure is rectangle.

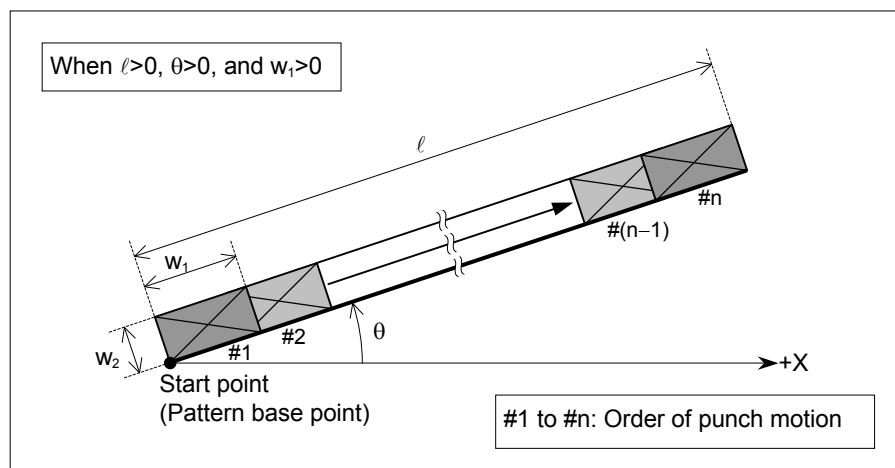
For this command, the CNC calculates the pitch automatically as follows to stack adjacent punch areas before punching in this pitch for the specified distance. Then, the product is stamped out .

Format

G86 I ℓ J θ P w_1 Q w_2

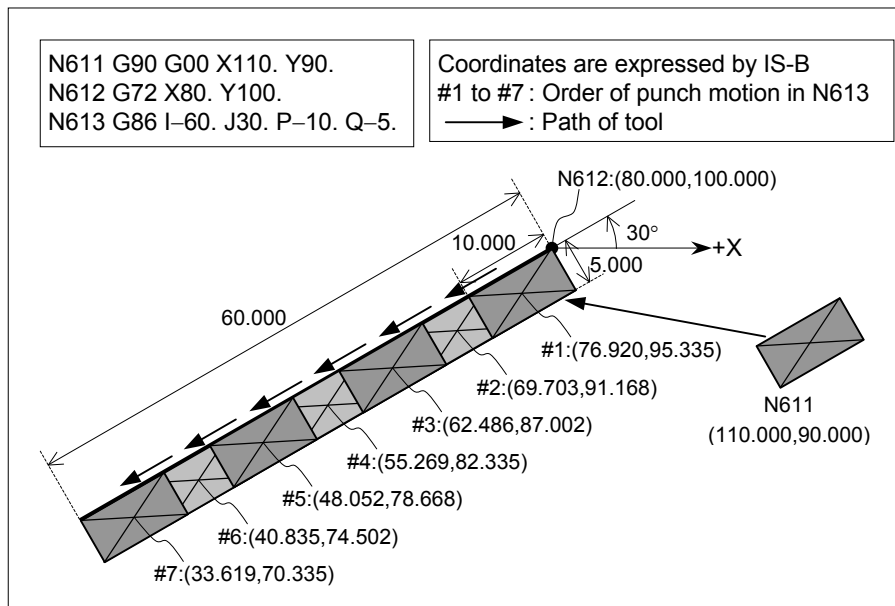
- ℓ : Length of punching (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-If this value is commanded with positive, punching is made in the θ direction.
-If this value is commanded with negative, punching is made in the $\theta+180^\circ$ direction.
- θ : The angle for the X-axis (Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : minimum unit of data (deg)
-The counterclockwise direction corresponds to positive.
-The clockwise direction corresponds to negative.
- w_1 : Tool width measured in parallel with the direction in which the tool advances
(Valid data range : ± 1 to ± 999999999)
Unit of data : Input unit (inch/mm)
-For positive and negative, punch the left and right sides in the forward direction, respectively.
- w_2 : Tool width measured vertically to the direction in which the tool advances
(Valid data range : ± 1 to ± 999999999)
Unit of data : Input unit (inch/mm)
-Both w_1 and w_2 must be designated when either positive or negative.
- For a square tool with $w_1 = w_2$, w_2 may be omitted.

With the pattern base point as a start point, this function allows to punch length $|\ell|$ in the direction of angle θ for the X-axis, using a rectangular tool with w_1 as the width and w_2 as the length.



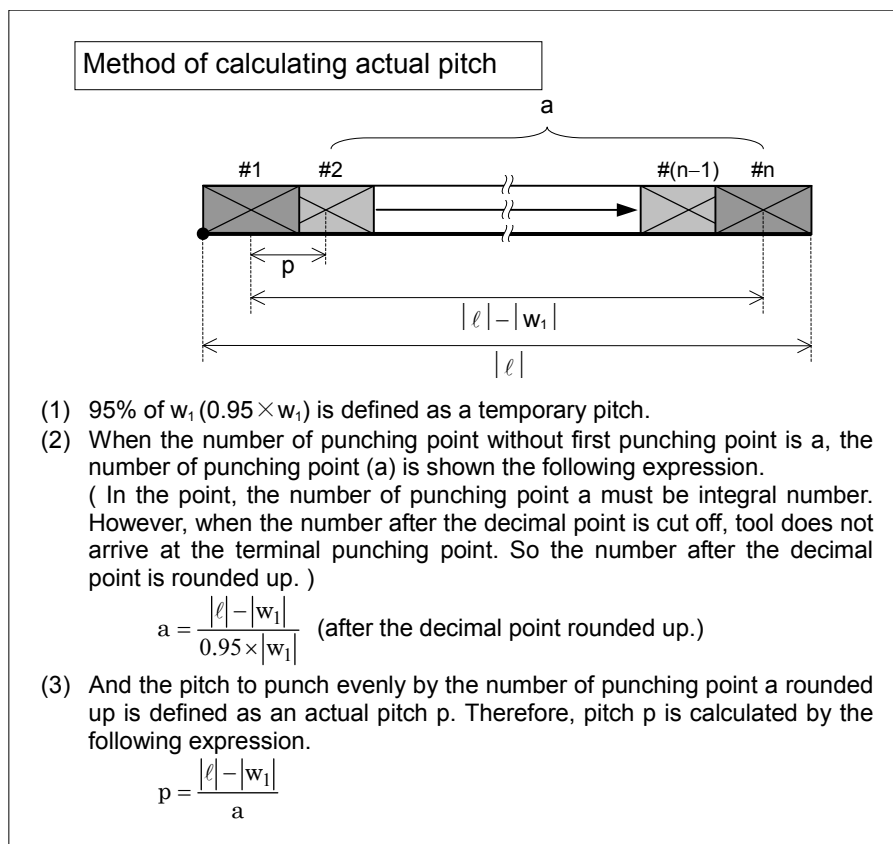
Example

- When $\ell < 0$, $\theta > 0$, $w_1 < 0$, and $w_2 < 0$



- Method of calculating actual pitch

The CNC calculates the pitch automatically as follows:



- **Relationship between length of punching ℓ (specified by I code) and tool with w_1 (specified by P code)**

Length ℓ must be; $\ell \geq 1.5 w_1$. If this limitation is not satisfied, the alarm (PS4506) "ILLEGAL COMMAND IN SHARE PROOFS" is issued. However, When bit 0 (AIP) of parameter No. 16202 is set to 1, length ℓ must be; $\ell \geq w_1$.

15.1.7 Square (G87)

Square (G87) specifies punching a rectangle with the pattern base point as the start point by using specified tool that figure is rectangle.

For this command, the CNC calculates the pitch automatically as well as Share proofs (G86) to stack adjacent punch areas before punching in this pitch along the edge of the specified rectangle. Then, the product is stamped out .

Format

G87 I ℓ_x J ℓ_y P w_1 Q w_2

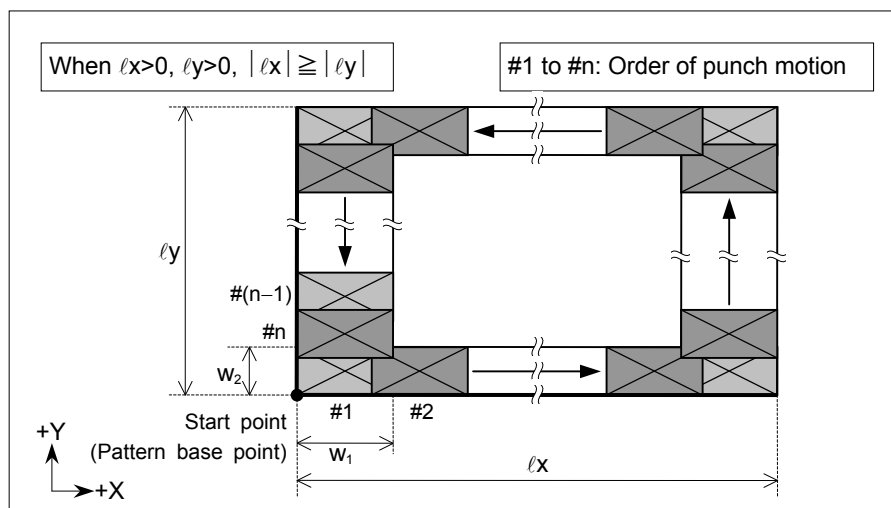
ℓ_x : Length of punching in the X-axis direction (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-For positive and negative, the length shall be in the +X and -X directions, respectively.

ℓ_y : Length of punching in the Y-axis direction (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-For positive and negative, the length shall be in the +Y and -Y directions, respectively.

w_1 : With of the tool in the X-axis direction (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.

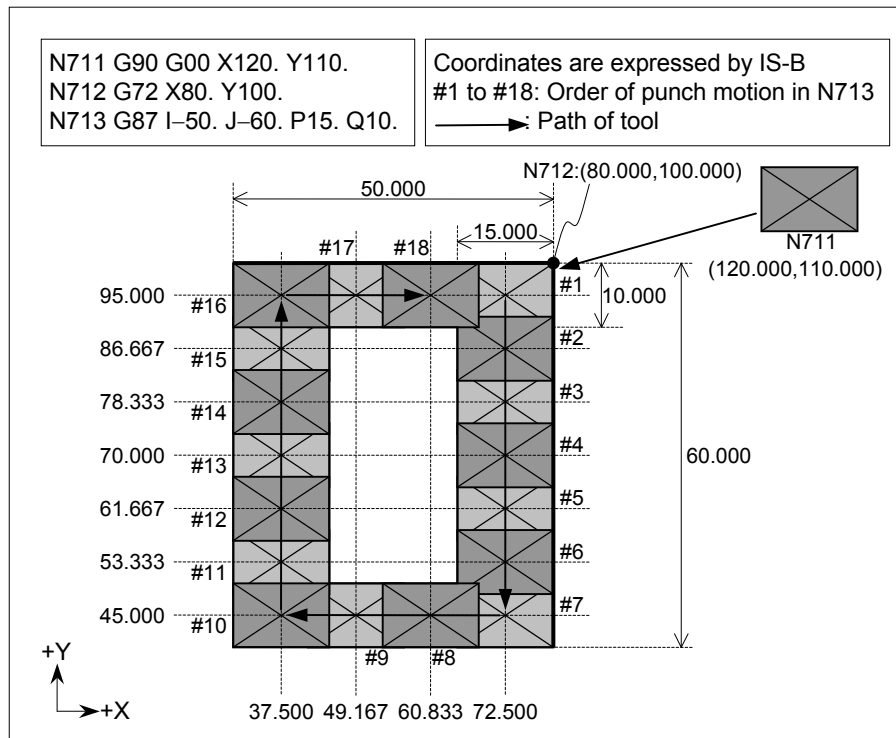
w_2 : With of the tool in the Y-axis direction (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.
- For a square tool with $w_1 = w_2$, w_2 may be omitted.

With the Pattern base point as a starting point, it allows to punch a rectangle with length $|\ell_x|$ in the X-axis direction and length $|\ell_y|$ in the Y-axis direction, using a tool with w_1 as the width and w_2 as the length. The punching operation is performed from the longitudinal direction (i.e.: direction with large length between $|\ell_x|$ and $|\ell_y|$). If $|\ell_x| = |\ell_y|$, punching operation will be performed in the X-axis direction.



Example

- When $lx < 0$, $ly < 0$, and $|lx| < |ly|$



15.1.8 Radius (G88)

Radius (G88) specifies punching at the constant pitch (distance) along the circular arc with the specified angle and the pattern base point as the circle center.

Format

G88 I_r J_θ K_{Δθ} P_d Q_p

r : Radius of arc (Valid data range : 1 to 999999999)

Unit of data : Input unit (inch/mm)

-Should be specified by positive number.

θ : Angle of the first punch point for the X-axis

(Valid data range : When the increment system is IS-B, -359999 to 359999)

Unit of data : Minimum unit of data (deg)

-Counterclockwise is commanded by a positive number.

-Clockwise is commanded by a negative number.

Δθ : Incremental angle from the first punch point to the last punch point

(Valid data range : -999999999 to 999999999)

Unit of data : Minimum unit of data (deg)

-Counterclockwise punching is commanded by a positive number.

-Clockwise punching is commanded by a negative number.

d : Tool diameter (Valid data range : -999999999 to 999999999)

Unit of data : Input unit (inch/mm)

-It allows to punch the circle outside, the circle inside, and the circumference, when positive, negative, and 0, respectively.

-Actual tool path is on the circular arc having the radius, $r+d/2$.

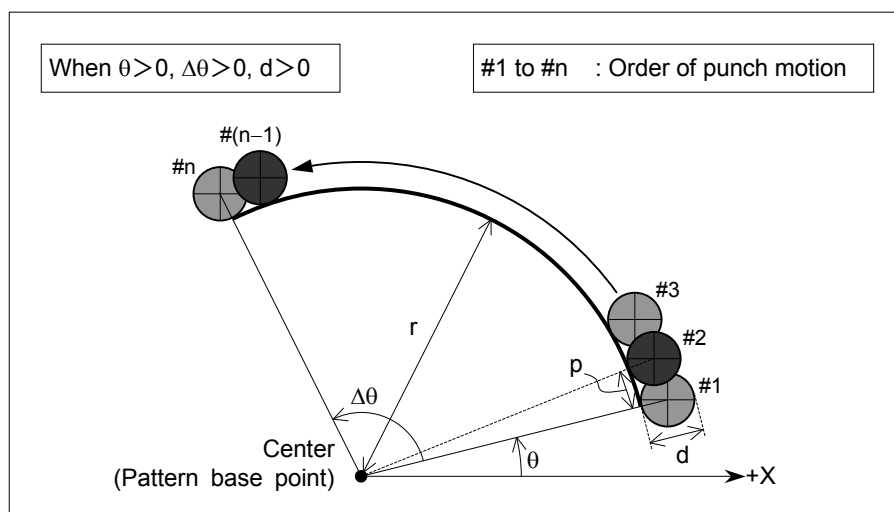
p : Pitch (Valid data range : 1 to 999999999)

Unit of data : Input unit (inch/mm)

-Should be specified by positive number.

-Designate the circular length, respectively.

The punching operation is performed at pitch P between a point having angle θ for the X-axis on the circumference (diameter r) and a point having angle $\theta + \Delta\theta$ for the X-axis with the pattern base point as a center according to the above, using a tool with diameter d.



Radius (G88) is the same as with circular nibbling (G68) except that there is no pitch limit value by parameter setting (No. 468 and 469), and it stops at each punch point for a single block. Therefore, also refer to the Circular Nibbling (G68) paragraph given under 9.3.1.

15.1.9 Cut at Angle (G89)

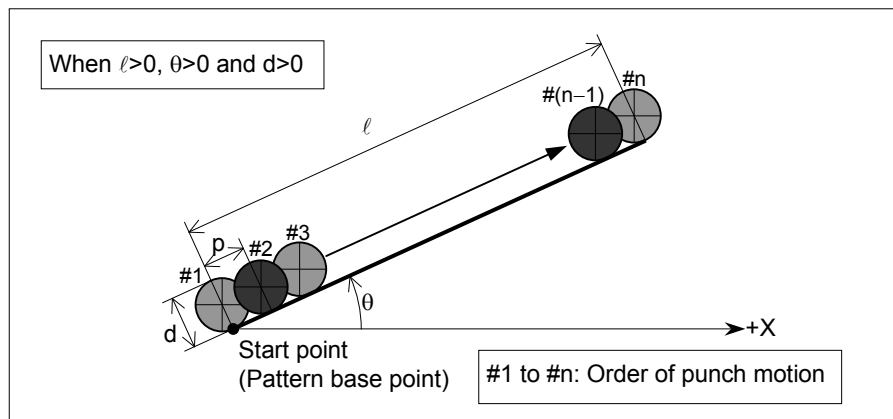
Cut at angle (G89) specifies punching at the constant pitch (distance) along the straight line starting with the pattern base point as a starting point.

Format

G89 I ℓ J θ P d Q p

- ℓ : Straight line length (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-If this value is commanded with positive, punching is made in the θ direction.
-If this value is commanded with negative, punching is made in the $\theta+180^\circ$ direction.
- θ : The angle for the X-axis (Valid data range : When the increment system is IS-B, -359999 to 359999)
Unit of data : minimum unit of data (deg)
-The counterclockwise direction corresponds to positive.
-The clockwise direction corresponds to negative.
- d : Tool diameter (Valid data range : -999999999 to 999999999)
Unit of data : Input unit (inch/mm)
-For positive and negative, punch the left and right sides in the forward direction, respectively.
-Perform punching operation on the straight line at 0.
- p : Pitch (Valid data range : 1 to 999999999)
Unit of data : Input unit (inch/mm)
-Should be specified by positive number.
-Designate the straight line length, respectively.

This function allows to punch a straight line with length ℓ having angle θ to X-axis with the current tool position or the position designated by G72 as a starting point, at pitch P , using a tool with diameter d .



Cut at angle (G89) is the same as with straight line nibbling (G69) except that there is no pitch limit value by parameter setting (No. 468 and 469) and it stops at each punching point for a single block; therefore, refer to 9.3.2 “Straight Line Nibbling (G69)” as well.

15.1.10 Incremental Command Just After Pattern Function

If an incremental command is given in a block just after the pattern function, the tool may not move by the incremental amount from the end point of the pattern function. In case of bolt hole circle (G26), share proofs (G86), square (G87), radius (G88) and cut at angle (G89).

The amount of movement attained by an incremental command just after the bolt hole circle is the commanded incremental amount from the center of the bolt hole circle.

In case of bolt hole circle (G26) and square (G87), the incremental amount is from the base point of pattern, namely, center and start point respectively.

In case of share proofs (G86), radius (G88) and cut at angle (G89), the incremental amount is from the end point of the program. The incremental amount is all started with the end point of the pattern function in line at angle (G76), arc (G77), and grid (G78, G79).

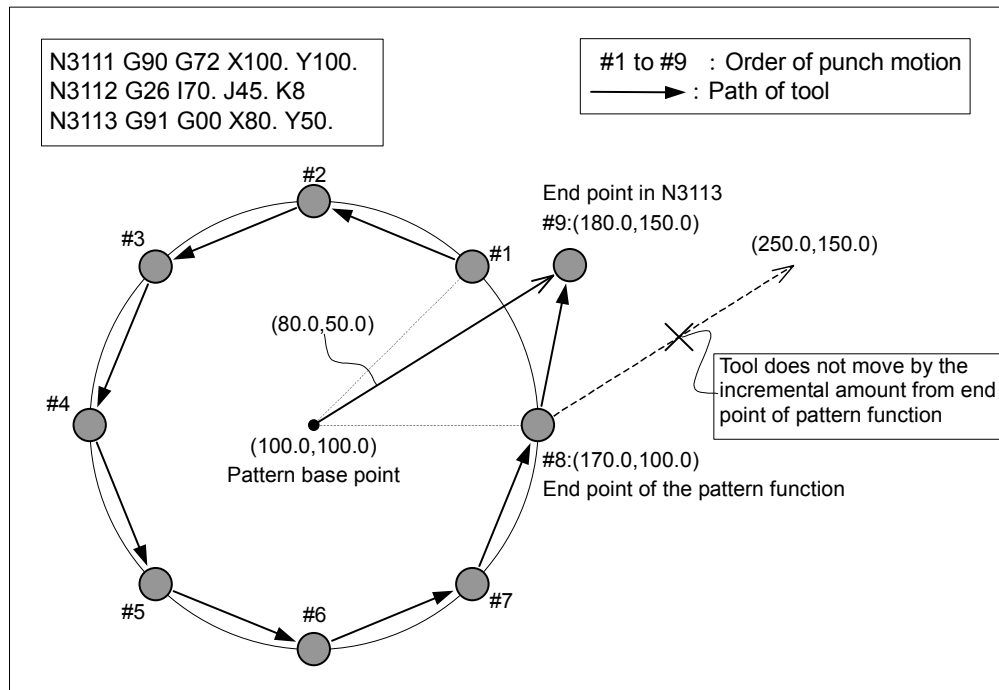


Fig.15.1.10 (a) Incremental command just after bolt hole circle (G26)

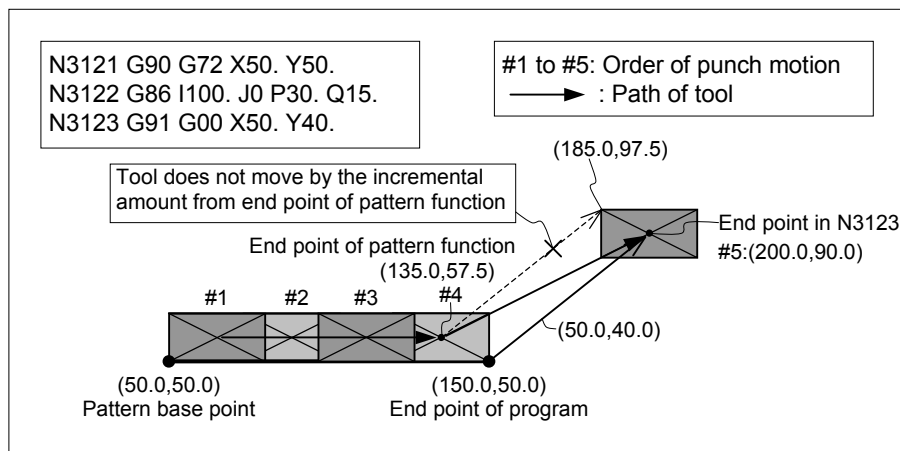


Fig.15.1.10 (b) Incremental command just after share proofs (G86)

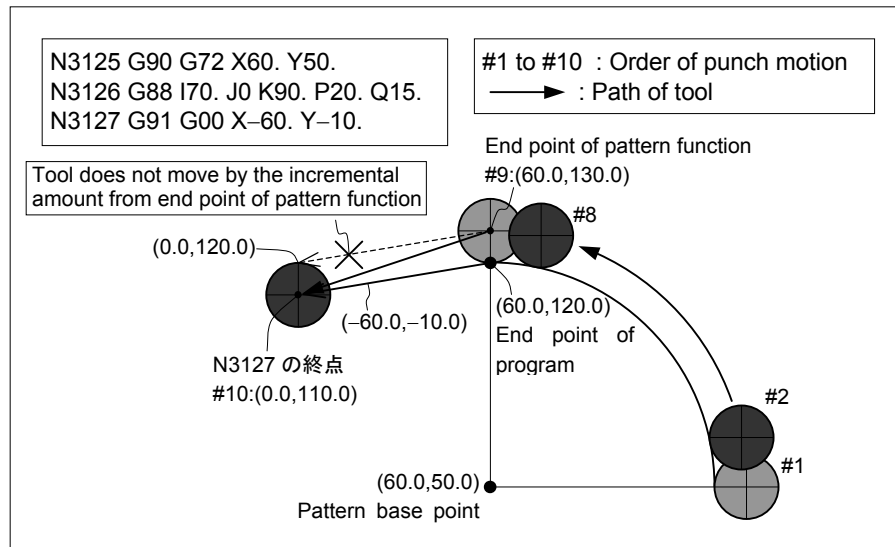


Fig.15.1.10 (c) Incremental command just after radius (G88)

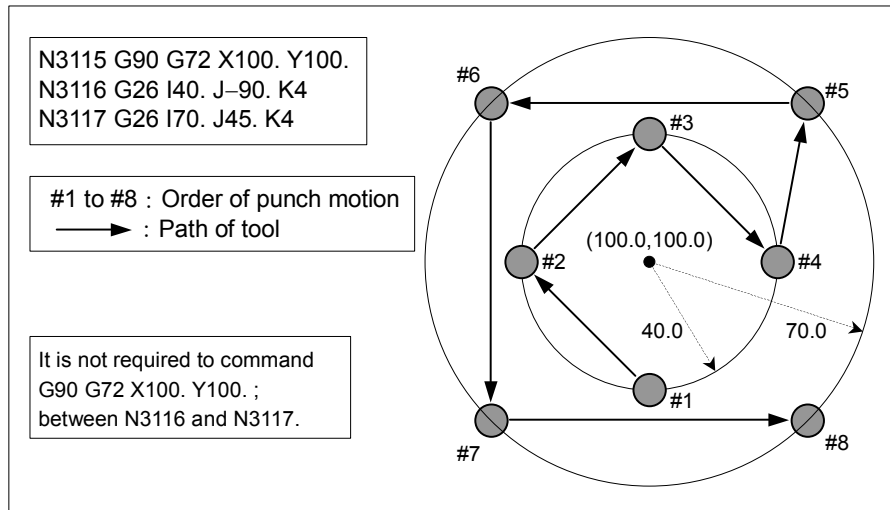
When the block execution of the bolt hole circle (G26) has been finished, the tool is located at the end point, in practice.

However, a programmer shall make a program assuming that the tool be located at the base point, i.e., the center of the pattern.

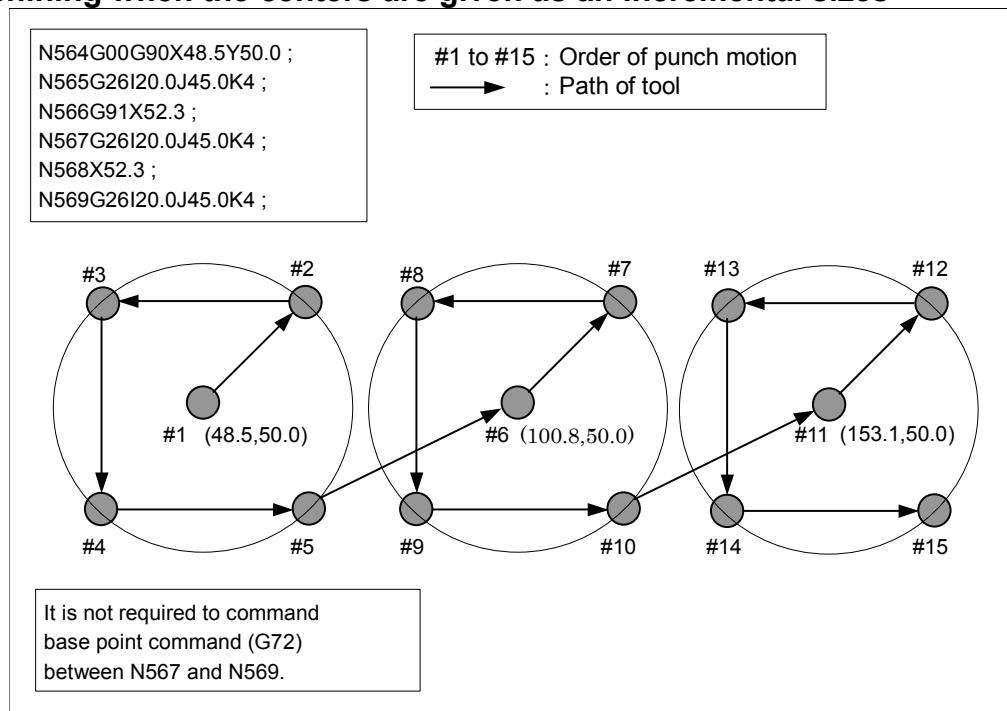
Thus, programming is simplified in machining, like the following example.

Example

- Concentric bolt hole circles



- Machining when the centers are given as an incremental sizes



15.1.11 Notes on Pattern Functions

⚠ CAUTION

- 1 Don't command M code in a block where the pattern function is commanded.
- 2 If a T code is commanded in a block where the pattern function is commanded, the X, Y axes are positioned to the first punch point and also a tool is selected concurrently.
 After positioning and tool selection, punching is made at the first punch point.
- 3 Even when the pattern function is commanded in G01, G02 or G03 mode, the X, Y axes are positioned at rapid traverse. However, since T code cannot be commanded in G01, G02, or G03 mode, such a command is not executable, in general.
- 4 If a pattern function is executed using the single block operation, the single block stop is made after punching has been finished at respective punch points.
- 5 If the pattern function is commanded in MDI mode, punching is not executed, but positioning only is made. Some machines may execute punching.
- 6 Since radius, number of punch points, and other parameters in the pattern function are not modal data, they must be commanded correctly in each block.
- 7 Operation for the C-axis can be specified in a block in which the pattern function is specified.

15.2 MEMORY AND CALL BY A/B MACRO

When it is desired to repeatedly use a pattern with the same figure among the pattern functions, this function can store it in memory with a given number and access it as needed. Programs other than those using the pattern functions can be stored in memory and can be called.

(1) Memory

When a one-digit numeral from 1 to 5 subsequent to address A is given before the G code of the pattern function, the pattern function according to the G-code is stored in the memory.

(Example) A2G76I300J3000K5;

(2) Call

By giving a one-digit numeral from 1 to 5 subsequent to address B, it is possible to call the pattern function stored previously with address A.

(Example) B2;

(3) Example

When bolt hole circles with the same figure having a center in (100, 100), (200, 100), (250, 180), and (150, 180) are present:

.....

N01G72X100.Y100.; Base point command

N02A2G26I40.J0K4; Bolt hole circle command

N03G72X200.Y100.; Base point command

N04B2; Equal to the block which commanded A2

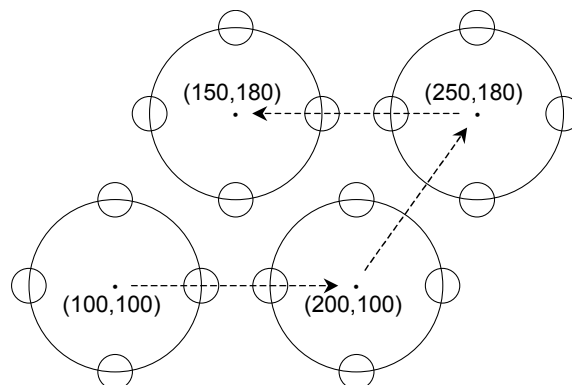
N05G72X250.Y180.; Base point command

N06B2; Equal to the block which commanded A2

N07G72X150.Y180.; Base point command

N08B2; Equal to the block which commanded A2

.....



⚠ CAUTION

- 1 Do not give the same number because the pattern function differs. At this time, the previous memory is erased.
- 2 Only a block with up to 192 characters can be stored in memory.

15.3 AUTOMATIC REPOSITIONING (G75)

By changing the hold position of a workpiece by the workpiece holders, a workpiece having a size larger than the stroke in X-axis direction of the machine can be machined.

If it is desired to punch a workpiece at the workpiece holder position when the workpiece was set to the machine, the hold position of the workpiece must be changed.

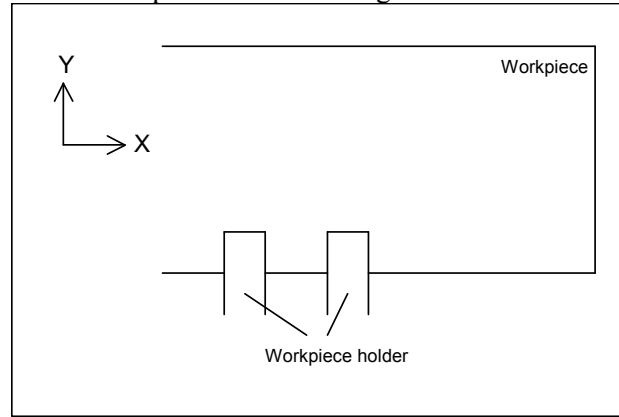


Fig.15.3 (a) Workpiece holder

Repositioning of a workpiece is generally done according to the following procedure, assuming that the workpiece is positioned at a location where the repositioning of the workpiece is executable.

- (1) The claw of the workpiece holder is opened, and also the clumper depresses the workpiece concurrently to fix the workpiece as a general procedure, so that the workpiece is not deviated from the table.

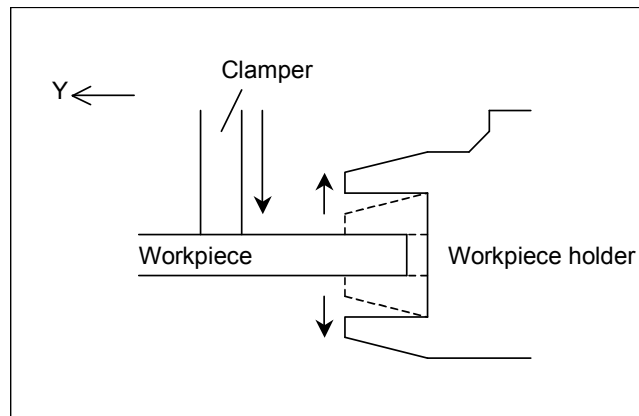


Fig.15.3 (b) Workpiece clamp

- (2) The workpiece holder moves in the Y-axis direction and separates from the workpiece.

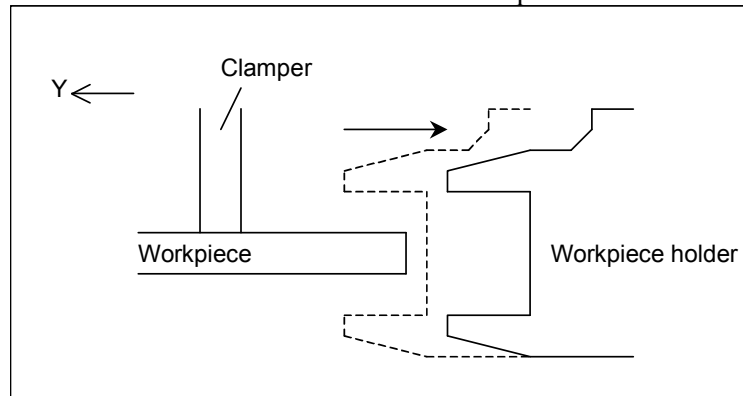


Fig.15.3 (c) Retreat of workpiece holder

- (3) The workpiece holder moves in the X-axis direction to relocate the hold position.

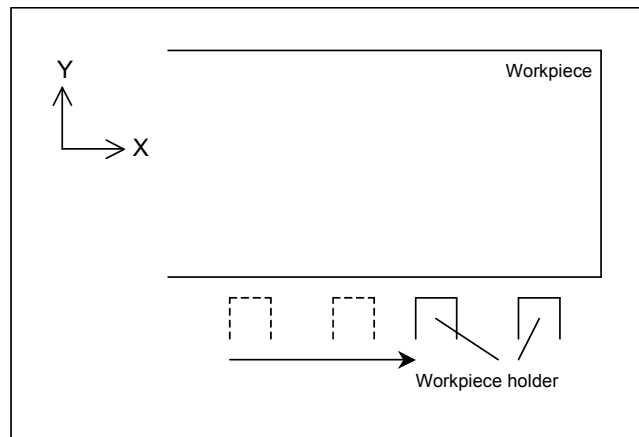


Fig.15.3 (d) Movement of workpiece holder

- (4) The workpiece holder moves in the Y-axis direction to return to the position where it can hold the workpiece.

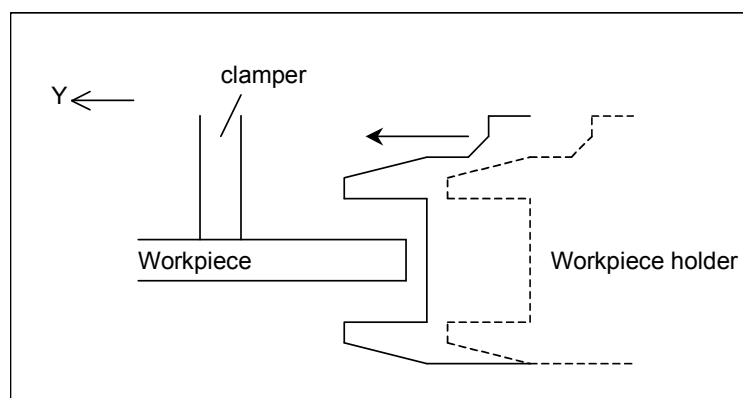


Fig.15.3 (e) Advancement of workpiece holder

- (5) The claw of the workpiece holder is closed to hold the workpiece, and the clamber lifts and separates from the workpiece concurrently.

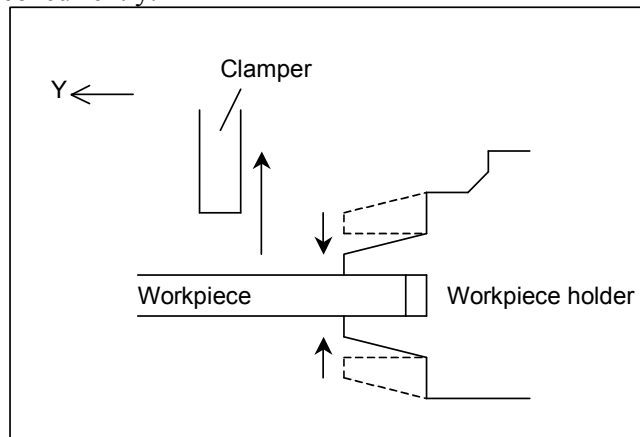


Fig.15.3 (f) Workpiece unclamp

A series of the above operation can be done by one-block command including G function.

G75 X_x

The above command is executed by being divided into the following 5 blocks.

- <1> M10 ;
- <2> G70G91Y_{y_R} ;
- <3> G70G91X_{-x} ;
- <4> G70G91Y_{-y_R} ;
- <5> M11 ;

Blocks <1>- <5> correspond to operation steps (1) – (5), respectively.

A relief or a return mount y_R in the Y-axis direction is preset by parameter (No. 16209).

By G75 command, the X and Y axes move.

The movement of the Y-axis is just cancelled by its relief and return. If it is assumed that the movement of the X-axis is the movement of a tool in the same manner as in ordinary block, for the X-axis command hereafter, X-axis command value must be subtracted by the value displaced by G75 or a coordinate system must be set by a G92 command with the tool at the position where the G75 was commanded in the block next to G75.

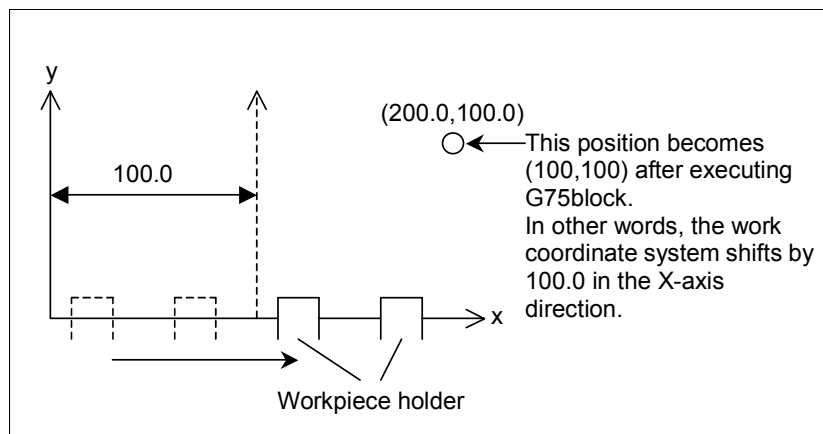


Fig.15.3 (g) The work coordinate system after executing automatic repositioning

The movement of the X-axis for repositioning changes the workpiece holder position. The tool position remains unchanged in the work coordinate system, in practice. Accordingly, internal processing is made in this control system so that the coordinate value of the tool is not changed by the axis movement performed by repositioning.

Thus, it is not necessary to take the repositioning of a workpiece into consideration in programming.

Now, the following description covers such a case that several workpieces are sequentially machined according to a program including the repositioning command.

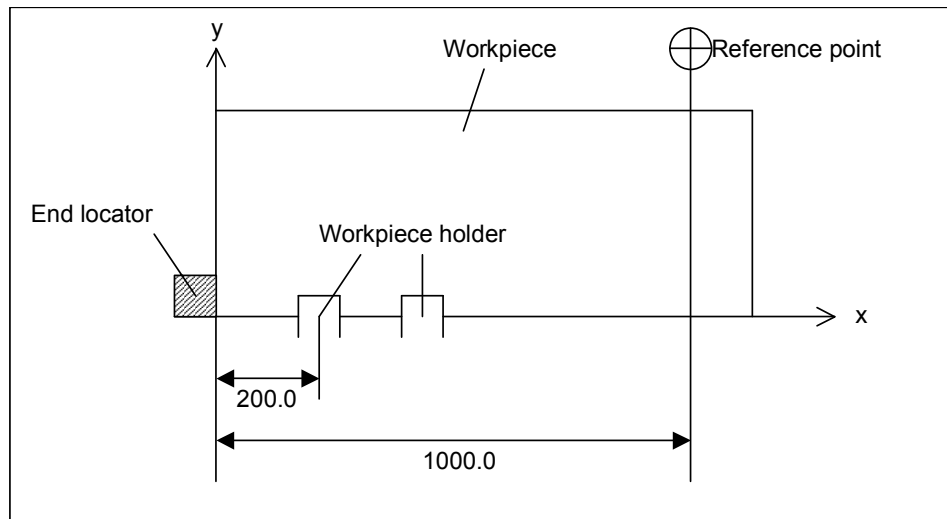


Fig.15.3 (h) The work position before executing automatic repositioning

Refer to the above figure as an example. The reference point is assumed as the start point of the tool. Assume that the distance between the reference point and the end locator is 1000 mm, and the left workpiece holder is mounted at a place distant from the end locator by 200 mm.

In order to machine the next workpiece by the same program after machining one workpiece, the start point of the tool must also be the reference point to the next workpiece. Accordingly, programming must be made to set the end point to the reference point.

```
G92X1000.0Y ... ;
```

```
.....
```

```
.....
```

```
.....
```

```
G75X300.0 ;
```

```
.....
```

```
.....
```

```
.....
```

```
G28M30 ;
```

“The end point is the reference point.” means that the position of the end point is just the same as the start point on the machine tool, and the workpiece holder position is the start point, i.e., the position distant from the end locator by 200 mm. However, the left lower point of the workpiece, i.e., the zero point of the work coordinate system, does not meet the end locator position, but it is distant from the end locator by 300 mm in the -X direction.

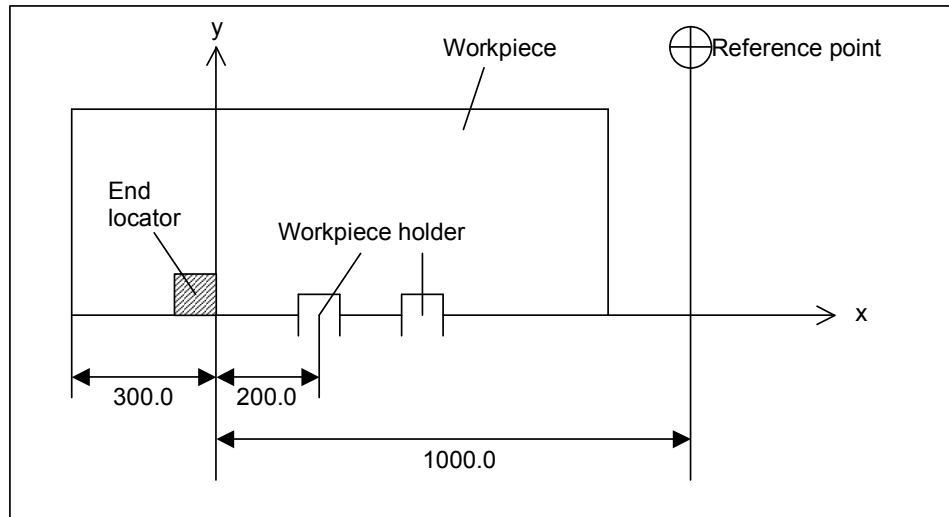


Fig.15.3 (i) The work position after executing automatic repositioning

When a new workpiece is set by attaching it to the end locator after removing the workpiece, the zero point of the work coordinate system must be positioned at the end locator.

Accordingly, the coordinate system must be set by G92 command at the start of the program without fail.

⚠ CAUTION

- 1 Neither T code nor M code should be commanded in G75 block.
- 2 The repositioning amount of the workpiece holder is specified by a numerical value following address X in G75 command. If repositioning is made in the +X direction of the work coordinate system, specify it by a positive numerical value.
- 3 Other M codes may be used for M10 and M11 depending upon machine tool builders.
- 4 Programming is made, assuming that the workpiece is fixed and a tool moves along the workpiece. However, the workpiece moves, while the tool is fixed generally. Accordingly, the movement direction of the workpiece, or, the movement direction of the workpiece holder, is contrary to the movement direction of the tool.

This means that, when the workpiece holder moves in the +X and +Y directions of the work coordinate system, the tool must be regarded to move in the -X and -Y direction respectively. Accordingly, the command in <2>, <3>, <4> blocks is opposite to movement direction in the work coordinate system of the workpiece holder.

 CAUTION

- 5 In G75 command, the relief quantity of the workpiece holder from the workpiece is equal to the return quantity of the workpiece holder to the workpiece in the absolute value of its movement amount.

However, if a workpiece is uneven, the workpiece holder depresses the workpiece, causing the workpiece to be deviated, assuming that the workpiece holder returns by the relief quantity as it is. For such a work, change the relief quantity and return quantity by the following programming.

```
M10;  
G91Y y1 ;  
X -x ;  
Y -y2 ;    y1 differs from y2.  
M11;
```

Even in the above command, internal processing is made in NC so that the tool position remains unchanged in the work coordinate system, irrespective of the movement of the X and Y axes.

Accordingly, it is not necessary for programming to take this movement into consideration in the subsequent commands. However, since the workpiece cannot be held if such a command is given again, it is necessary to take these circumstances into consideration when performing repositioning again.

- 6 It is not recommended for cycle time and machining accuracy to repeatedly execute repositioning by a program.
- 7 The single block stop is made after five sequential motion, if G75 is executed by a single block operation.

15.4 U/V/W MACRO FUNCTION

The macro function enables commands consisting of several blocks to be stored in the NC memory as a single macro and to be called when necessary.

15.4.1 Storage of Macros

To store several blocks as a macro, attach numerals of 2 digits (01 to 89) following address word U before commanding several blocks to be stored and attach the same numerals following address word V after them, then the several blocks sandwiched in between U and V are stored.

We call the numerals of 2 digits following U or V “macro number”. If the macro number is any of 01 to 59, then blocks sandwiched in between U and V are stored with their execution. If the macro number is any of 60 to 89, only storage of blocks is performed without their execution.

Example

```

;
G92X1830.0Y1270.0 ;
.....
.....
U02 ;
G90X100.0Y500.0T32;
G72X150.0Y700.0;
G87I100.0J300.0P10.0;
N100M00 ;
G72X500.0Y300.0T26;
A03G26I100.0J0K4;
G72X800.0Y300.0;
B03;
V02;
.....
.....
U70 ;
G90X200.0Y100.0T20;
G79I8.0J10.0P3K10;
G90X1000.0Y50.0T31;
G76I25.0J60.0K6 ;
V70 ;
.....
.....

```

} Storing while command executing

} Storing without command executing

In the blocks to be sandwiched in between U and V, all commands other than ones for storage of macros according to other macro numbers (see below Example (limitation)) may be specified. But controller resetting commands, such as M02, M30, etc., can not be specified. In addition, the part for custom macros cannot be specified. Address words U and V must be specified in independent blocks except for the sequence number (a numeric string of 5 digits following N).

Details of macro numbers 90 to 99 will be described later.

Example (limitation)

```

.....
.....
[ U10 ;
  G90X10000Y50000T32 ;
  G72X15000Y70000 ;
  G87I10000J30000P1000 ;
  N100M00 ;
  [ U20 ;
    G72X50000Y30000T26 ;
    A03G26I10000J0K4 ;
    G72X80000Y30000 ;
    B03 ;
    V20 ;
    G90X20000Y10000T20 ;
    V10 ;
  ]
]
```

As shown in the above example, another macro cannot be stored while a certain macro is being stored.

15.4.2 Macro Call

Several blocks stored previously as a single macro can be called by specifying numerals of 2 digits attached following address word W.

Example

```

.....
.....
U05 ;
G90X100.0Y500.0T32 ;
G72X150.0Y700.0 ;
G87I100.0J300.0P10.0 ;
M00 ;
V05 ;
.....
.....
G93G90X500.0Y0 ;
W05 ;
.....
```

Address word W must be specified in a independent block except for the sequence number. But as regards the commands G73 and G74 for the multi-piece machining function, W can be specified together with their G codes.

15.4.3 Nesting Call of Macros

A certain macro can call another macro and then further the latter macro can call any other macro.

Example

U05; (): Signifies a block number
; (1)
; (2)
 V05;
 U20;
; (3)
 W05;
; (4)
 V20;
 U70;
; (5) } These are not executed during storage because the macro number is 70.
 W20;
; (6)
 V70;
 W70;

The execution sequence of this example is as follows:

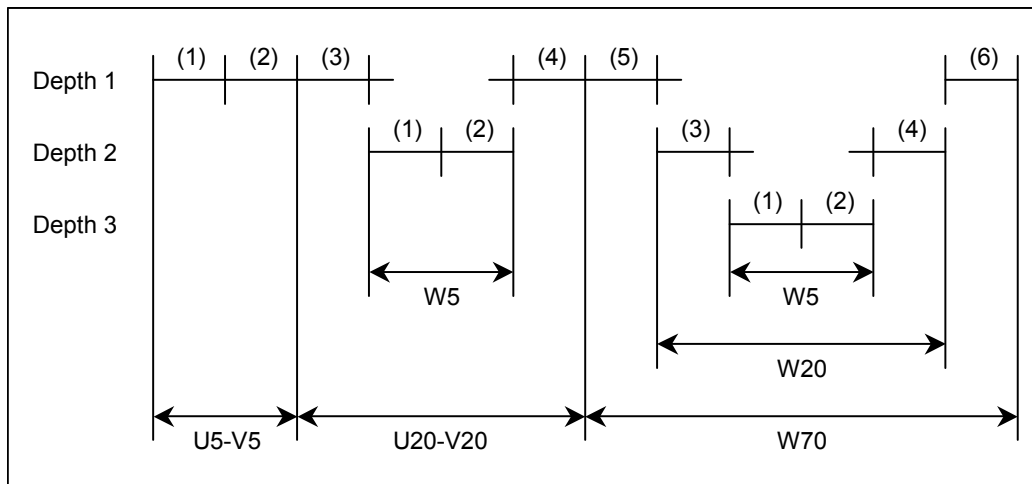


Fig.15.4.3 (a) The order of executing macro

15.4.4 Macro Storage Capacity

The storage capacity for U/V/W macro function is limited to 50kbyte. And storage method of U/V/W macro function is the same as a usual program

Effective use of the storage area for storing macros is guaranteed since previously stored macros are erased if they are changed to new macros with the same macro numbers as before when they are out of use.

Example

```
N100U10 ;  
  [G90X10000Y5000T20;  
N101A01G26I4000J9000K10;  
G91X10000;  
N102B01;]  
N105V10;
```

.....
Program codes in brackets [] are stored in memory.

15.4.5 Storage and Call of Multiple Macros (Macro Numbers 90 to 99)

With macro numbers 90 to 99, several macros can be stored and called as a single macro, though the item 15.4.1 “Storage of macros” describes that another macro cannot be stored while a certain macro is being stored. Macro numbers 90 to 99, however, only define several macros as a single macro, but they cannot store any executable commands by themselves.

Example

```
U90;  
U01;  
.....  
.....  
V01;  
U10;  
.....  
.....  
V10;  
U15;  
.....  
.....  
V15;  
G90X10000Y20000T15; ← This block is not stored.  
U02;                (such a use is no good)
```

```
.....  
.....  
V02;  
V90;  
.....  
.....  
W90; ← This command is equivalent to the following set of several commands.
```

```
W01 ;  
W10 ;  
W15 ;  
W02 ; } =W900
```


15.4.6 Deletion of Stored Macros

Macros stored therefore are all deleted in the following cases:

- (1) Reset (including reset due to M02, M30, etc.)
- (2) Controller power off

When the system is reset, the stored macros can be prevented from deletion by setting of bit 0 (UVC) of parameter No. 16200.

15.5 MULTI-PIECE MACHINING FUNCTION

The multi-piece machining function enables several sheets of product with the same punching shape to be produced from a single sheet of material at a time by simple commands.

This function allows so called “trial machining” that performs punching only on a sheet of product from the machining command tape for “multi-piece machining” by a simple setup method, therefore the machining command tape can be easily checked before full machining.

15.5.1 Base Point Command of Multi-Piece Machining (G98)

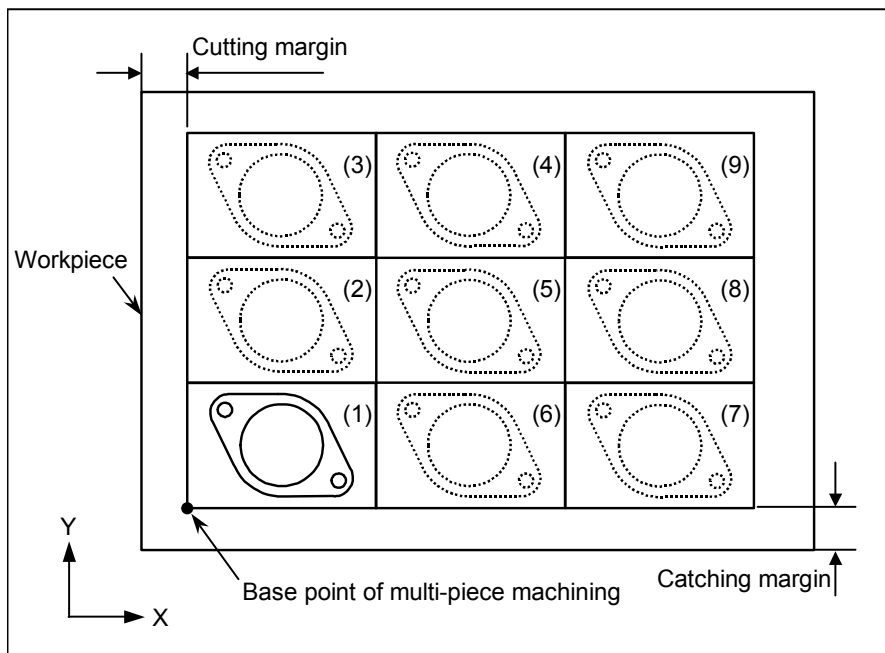


Fig.15.5.1 (a) Multi-piece machining

Parts (2) to (9) as shown above have the same punching shape as part (1).

Machining commands to punch on a sheet of material must be specified on the product part at the lower left (1). The point at the lower left of the set of multi-products (point B shown in Fig.15.5.1 (a); called as “Base point of multi-piece machining” hereafter) must be specified prior to the machining commands to punch on a product part by G98 as absolute co-ordinates under the system of co-ordinates specified using G92 command. In the G98 command, the X-axial and Y-axial lengths of one product and the numbers of products in the directions of X-axis and Y-axis must be specified.

Format**G98Xxb Yyb llx Jly Pnx Kny;**

xb : X-axis coordinate value of the base point of multi-piece machining
 yb : Y-axis coordinate value of the base point of multi-piece machining
 lx : X axial length of one product part (a positive number)
 ly : Y axial length of one product part (a positive number)
 nx : The number of products in the X axial direction (Note) (Command range is 0 to 9999)
 ny : The number of products in the Y axial direction (Note) (Command range is 0 to 9999)

NOTE

Product part (1) is not counted.

After command of G98, specify machining commands on the lower left product part ((1) in Fig.15.5.1 (a)) using the macro function (specify the machining commands between address word U and V).

In the multi-piece machining function, if a set of machining commands corresponding to a single tool is executed continuously on the all product parts, the tool change count is decreased so that the time required for full machining on a material sheet can be considerably decreased. The following illustrates this.

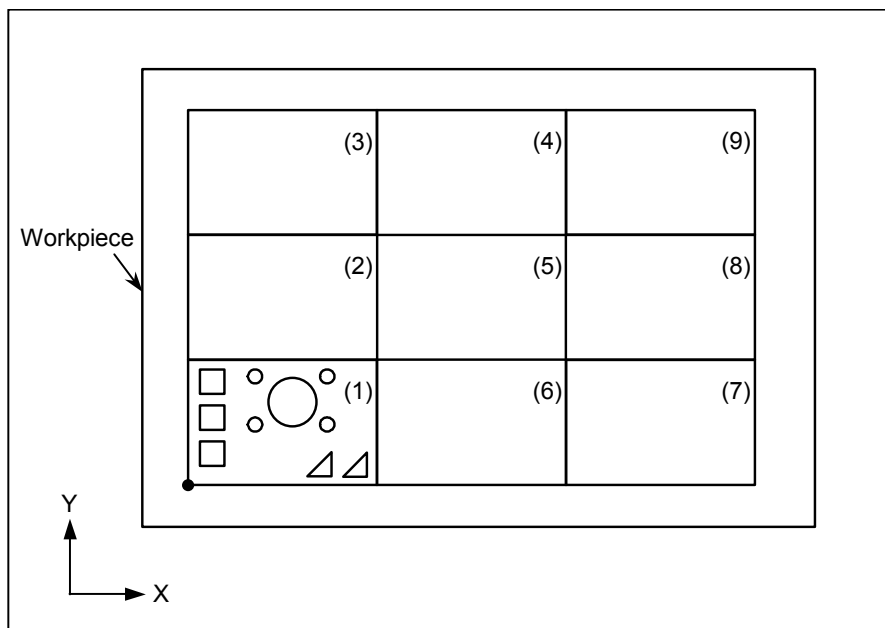


Fig.15.5.1 (b) Processing example

Let the following four types of tools be used to punch products for one sheet, as shown in Fig.15.5.1 (c).

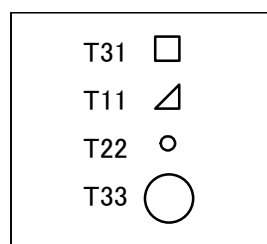


Fig.15.5.1 (c) Tools be used to punch products for processing example (Fig.15.5.1 (b))

Method 1

In this method, the machining commands to punch on one product part are stored as a single macro.

```

U01 ;
.....T31 ;
.....
.....
.....T22 ;
.....
.....
.....T33 ;
.....
.....
.....T11 ;
.....
.....
V01 ;

```

This method will run the risk of taking longer time for machining on a sheet of material because tool change is performed in the order, T31 → T22 → T33 → T11, while the respective product parts from 1 to 6 are machined if macro number 01 is called by the multi-piece machining command. (Tool change counts: $4 \times 6 = 24$)

Method 2

In this method, each set of machining commands to punch with a single tool is stored as a single macro.

```

U01 ;
.....T31 ;
.....
.....
V01 ;
U02 ;
.....T22 ;
.....
.....
V02 ;
U03 ;
.....T33 ;
.....
.....
V03 ;
U04 ;
.....T11 ;
.....
.....
V04 ;

```

If macro numbers 01, 02, 03 and 04 are called sequentially by the multi-piece machining command in this case, machining proceeds as follows.

- 1) Tool T31 performs full machining on all product parts.
- 2) Then, T22 performs full machining on all product parts.
- 3) Then, T33 performs full machining on all product parts.
- 4) Then, T11 performs full machining on all product parts.

Thus, its tool change is completed by only four steps so that tool change time is considerably decreased compared with method 1.

⚠ CAUTION

- 1 The G98 coordinate system is canceled by the G92 command.
- 2 If bit 6 (CLR) of parameter No. 3402 is set, the G98 coordinate system is canceled by a reset.
- 3 When the UV macro contains the G91 command, the tool moves according to the incremental value from the current position rather than the incremental position based on the G98 coordinate system.

15.5.2 Multi-Piece Machining Commands (G73, G74)

Specify any of the following commands, and then multi-piece machining is performed by calling machining commands stored using the macro function.

Format

G73(G74) W_{wn} Q_q ;

wn : A macro number

q : Machining start area specification

G73 goes on punching in the X axial direction, whereas G74 goes on punching in the Y axial direction in grid parts-line.

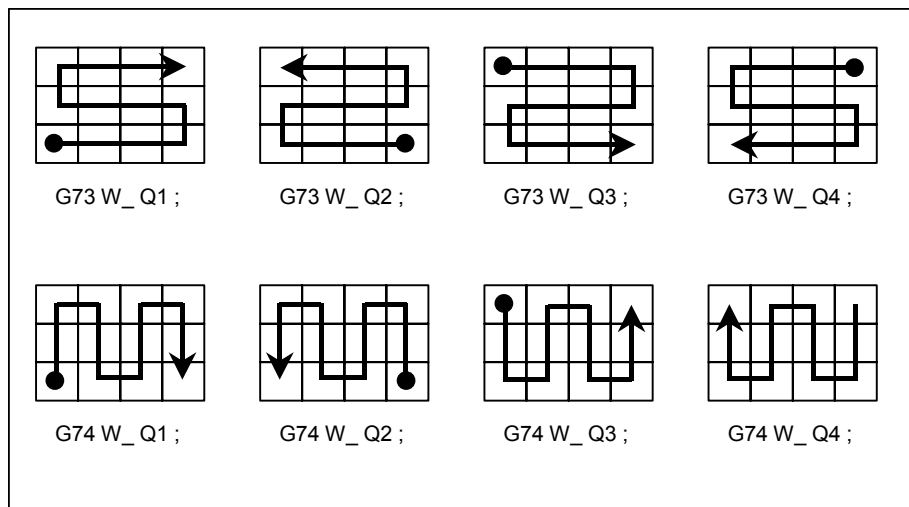


Fig.15.5.2 (a) Multi-piece machining

In the example shown in Fig.15.5.1 (b) to start machining from area (1) and to punch in the X axial direction, specify the following commands for the respective methods:

Method 1:G73W01Q1;

Method 2:G73W01Q1;
G73W02Q1;
G73W03Q1;
G73W04Q1;

Let the number of tool required to machine products for one sheet be n , and method 2 requires n times of commands G73/G74 to be specified. In such a case, several macro numbers can be represented by a single macro number if machining start area (specified by Q) and machining direction (G73/G74) are both the same with respect to the corresponding tools. Macro numbers to be provided for this purpose are 90 to 99. (These are called multiple macro storage numbers hereafter.) To store macro numbers 01, 02 and 04 as a single macro number 90 in the example shown in Fig.15.5.1 (b), specify as follows.

```

U90 ;
U01 ;
.....T31 ;
.....
V01 ;
U02 ;
.....T22 ;
.....
V02 ;
U04 ;
.....T11 ;
.....
V04 ;
V90 ;

```

In this case, the command of “G73 W90 Q1” becomes identical with the following series of commands.

```

G73W01Q1 ;
G73W02Q1 ;
G73W04Q1 ;

```

Though multi-piece machining commands (G73/G74) are generally used to punch products of the same shape arranged at intervals of grid, they can be also performed to punch exclusively in the X or Y axial direction. In such a case, specify the number of products in the X or Y axial direction (to be specified by address word P or K) in G98 to 0.

15.5.3 Setting of Machining Method for Multi-Piece Machining

When products are machined using a NC tape for multi-piece machining, any desired machining method can be selected.

- Select with parameter

When bit 3 of parameter No.16201 is set to 0, parameter No.16206 can set the method for multi-piece machining.

Parameter No.16206	Machining process when multiple workpieces are taken
0	Multi-piece machining is not used
1	Trial punching for multi-piece machining
2	Machining on the reset of material punched
3	Full machining for multi-piece machining

- Select with signal

Multi-piece machining setting signal MLP1, MLP2<Gn231.0,.1> can set the method for multi-piece machining.

Multi-piece machining setting signal		Machining process when multiple workpieces are taken
MLP1	MLP2	
0	0	Multi-piece machining is not used
1	0	Trial punching for multi-piece machining
0	1	Machining on the reset of material punched
1	1	Full machining for multi-piece machining

If trial punching is selected, only the lower-left product part of material ((1) in Fig.15.5.1 (a) and (b)). As a result, macros except 60 to 89 are executed while storing, and blocks specified in G73/G74 are all ignored.

Machining on the reset of material punched for trial signifies that after punching on only the lower-left product part of material, the remaining product parts of material are machined entirely. For this purpose, no machining is performed during the storage of macros, and the machining on the lower-left product part is skipped when commands of G73/G74 are executed.

For full machining on a material, no machining is performed during the storage of macros, but machining over the entire product parts is performed by G73/G74. Set to "0" if NC tape for the multi-piece machining is not used.

15.5.4 Command for Restarting Machining Multiple Products

Specifying the position from which machining multiple products restarts with address P in a block in which the G73 or G74 command for machining multiple products is specified enables machining multiple products to restart from the specified position.

Format

G73(G74) W_ Q_ P_ ;

W_ : UV macro number or subprogram number

Q_ : Machining sequence

P_ : Position from which machining restarts (Valid setting: $1 \leq P \leq \text{total number of products to be machined (Max. 99999999)}$)

- Example

Please specify a restart point by a P-code. Fig.15.5.4 (a) shows an example that restarting operation at point (6). In this case, specify P6.

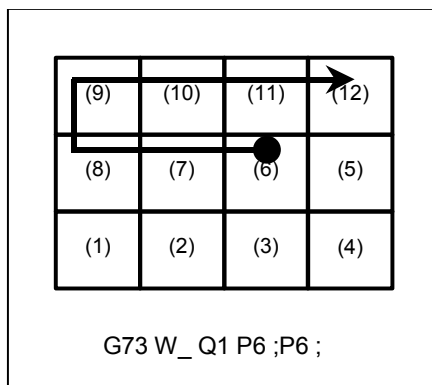


Fig.15.5.4 (a) Command for Restarting Machining Multiple Products

**CAUTION**

When restarting machining multiple products is specified, the reference position specified by the G98 command for setting the reference position used in machining multiple products must be used. Do not change the reference position.

NOTE

- 1 A restart command is invalid for trial machining.
- 2 When machining multiple products is restarted, specify the position from which machining restarts using address P so that the number of products includes the number of products to be used for the trial, though they are not machined.
- 3 If a setting specified with address P is not within the range from 1 to the total number of products inclusive, alarm (PS 4544) is issued.

15.6 BENDING COMPENSATION (G38, G39)

The hole position gap accompanied bending is compensated and the drilling is performed.

Format

Bending compensation for X axis direction

G38 Ix1 Jx2 Kx3 Px4 Qx5 R_ ;

Bending compensation for Y axis direction

G39 Iy1 Jy2 Ky3 Py4 Qy5 R_ ;

x1 to x5 : Bending position of X axis direction (Input unit)

y1 to y5 : Bending position of Y axis direction (Input unit)

R : The compensation amount attendant on a single bending is specified. (Input unit)

The reference point is specified by setting the local coordinate system.

G52X --- Y --- ;

In a block after a bending command is sent, it is judged to which area automatically designated the command value of the move command belongs, thus allowing compensation in accordance with the bending frequency.

When bending compensation cancel is required.

- (1) Specify the cancel of bending compensation.

G38R0 ; ... (Cancel of X axis direction)

G39R0 ; ... (Cancel of Y axis direction)

- (2) Specify M02, M30.

It is possible to cancel even in the reset state according to reset, clear, and emergency stop.

(Example) When the bending compensation of only X axis direction is performed.

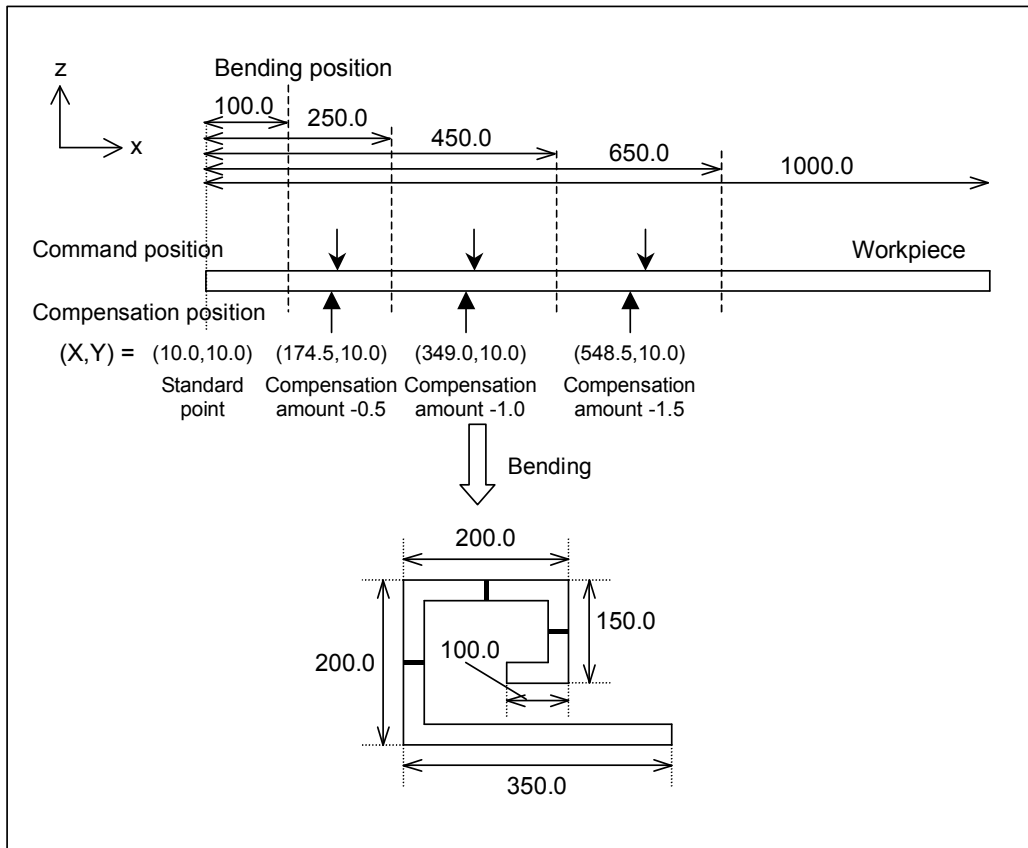
G52X10.0Y10.0

G38 I 100.0 J 250.0 K 450.0 P 650.0 Q 1000.0 R-0.5

G90 X175.0 Y0.0 ;

X350.0 ;

X550.0 ;



Program:

G52X100.Y0 ;	Specifications of standard point
G38I180.J260.K420.R-1. ;	Bending compensation command
G90X-50. ;	Absolute coordinate (X50.)
X270. ;	Absolute coordinate (X269.)
X300. ;	Absolute coordinate (X298.)
M02 ;	Cancel of bending compensation execution

In the above example, when the negative direction from the standard point, for example, X50. direction (X-50. since the coordinates system is offset in the program in the coordinates form the origin) is designated, bending compensation is not performed. At this time, it is regarded as area 1, that is, an area with no compensation by bending. As described above, when the number of bending is less than 4, it is possible to omit what is not required among X1-X5 and Y1-Y5 representing the distance from the standard point which indicates the area boundary.



CAUTION

In one block to which a pattern command, for example, is given during bending compensation, it is regarded as a move within the area, and no command over the area can be given.

15.7 LINEAR AND CIRCULAR PUNCH COMMAND

15.7.1 Linear Punch Command (G45)

Using linear punching command (G45) allows that workpiece is punched out along strait line from the pattern base point to the specified end point with the specified rectangle tool.

Format

G45 X_x Y_y P_{w₁} Q_{w₂} R_r D_{Δd}

(x , y) : End point (Valid data range : -999999999 to 999999999)

Unit of data : Input unit (inch/mm)

In G90 mode, please specify the position of the end point.

In G91 mode, please specify the amount of movement to the end point.

W₁ : Tool width measured in parallel with the direction in which the tool advances (Valid data range : 1 to 999999999)

Unit of data : Input unit (inch/mm)

-Should be specified by positive number.

W₂ : Tool width measured vertically to the direction in which the tool advances (Valid data range : 1 to 999999999)

Unit of data : Input unit (inch/mm)

-Should be specified by positive number.

-When this code is omitted, the CNC regards that W₂ is same as W₁.

r : Direction of tool offset with respect to the direction in which the tool advances (Valid data range : -1, 0, 1)

Unit of data : None

-Should be specified by positive number for offset to the left of the direction in which the tool advances.

-Should be specified by negative number for offset to the right of the direction in which the tool advances.

-Should specify R0 to punch out on programmed line (no offset). When R-code is omitted, the CNC regard that R0 is specified.

Δd : Microjoint width (Compensation value) (Valid data range : -999999999 to 999999999)

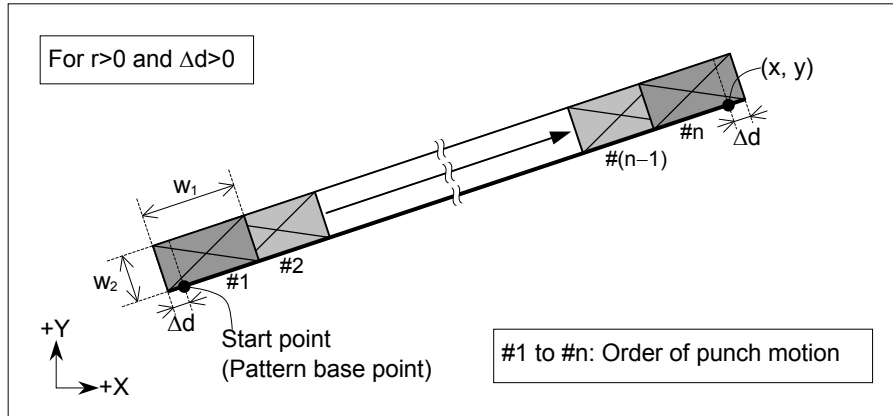
Unit of data : Input unit (inch/mm)

-Should be specified by positive number for compensation in such a way that the resultant workpiece dimensions increase.

-Should be specified by negative number for compensation in such a way that the resultant workpiece dimensions decrease.

-Should specify D0 to punch out with no compensation. When D-code is omitted, the CNC regard that D0 is specified.

Using above command allows workpiece is punched out along a straight line from the pattern base point as a start point to the end point (x, y) with a tool having dimensions (w₁, w₂). Moreover, specifying microjoint width Δd effects compensation at the punch start and end points.



• Calculating method of actual pitch

Pitch is calculated automatically as follows:

Calculating method of actual pitch

(1) When the length of punching is defined as ℓ , length ℓ is shown by the following express.

$$\ell = \sqrt{(x - x \text{ of base point})^2 + (y - y \text{ of base point})^2} + \Delta d \times 2$$

(In G90 mode)

$$\ell = \sqrt{x^2 + y^2} + \Delta d \times 2$$

(In G91 mode)

(2) 95% of w_1 ($0.95 \times w_1$) is defined as a temporary pitch.

(3) When the number of punching point without first punching point is a , the number of punching point (a) is shown the following expression.
(In the point, the number of punching point a must be integral number. However, when the number after the decimal point is cut off, tool does not arrive at the terminal punching point. So the number after the decimal point is rounded up.)

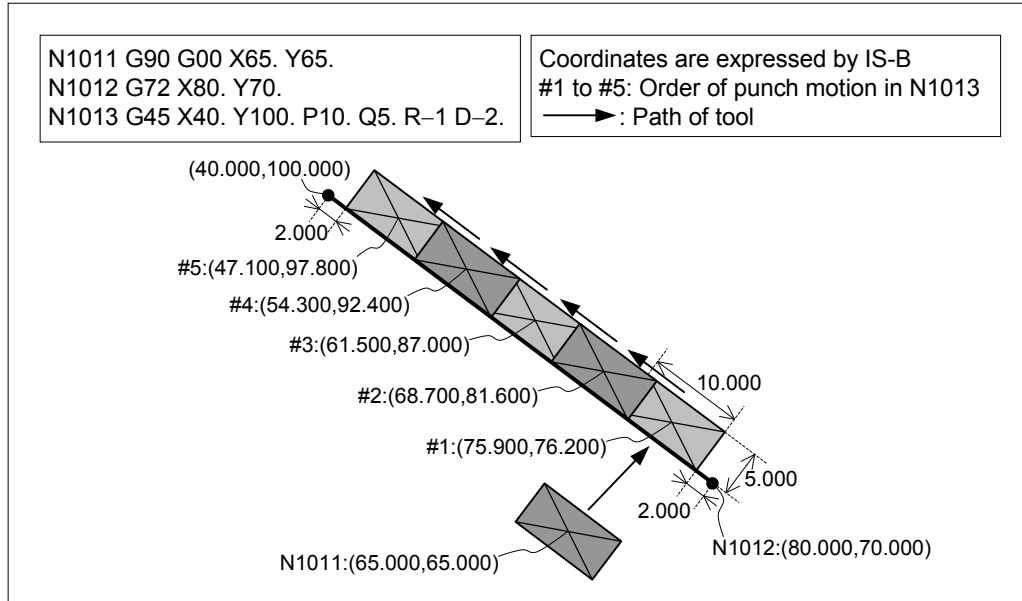
$$a = \frac{\ell - w_1}{0.95 \times w_1} \quad (\text{after the decimal point rounded up.})$$

(4) And the pitch to punch evenly by the number of punching point a rounded up is defined as an actual pitch p . Therefore, pitch p is calculated by the following expression.

$$p = \frac{\ell - w_1}{a}$$

Example

- When $r < 0$ and $\Delta d < 0$



NOTE

- 1 When the amount of movement is 0 or is smaller than 1.5 times of dx value during G45 block, the alarm (PS4500) is issued.
- 2 When the Tool width measured in parallel with the direction in which the tool advances (W_1) is not specified, the alarm (PS4510) is issued. Moreover, Tool width measured vertically to the direction in which the tool advances (W_2) is not specified, the CNC regards the tool shape as regular square.

15.7.2 Circular Punch Commands (G46 and G47)

Using circular punching commands (G46, G47) allows that workpiece is punched out on a circle with a radius of r between the pattern base point and the specified end point in constant pitches.

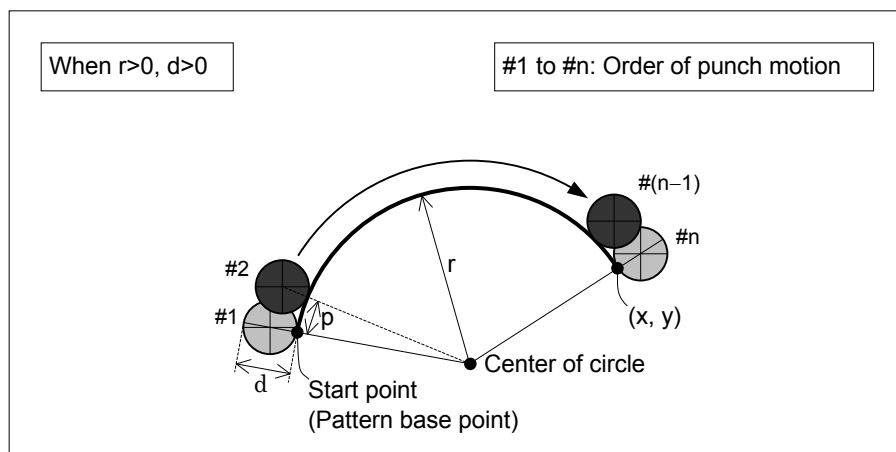
By using G46 command, the tool moves along the arc clockwise. And, by using G47 command, the tool moves along the arc counterclockwise.

Format

G46 X_x Y_y R_r P_d Q_p
G47 X_x Y_y R_r P_d Q_p

- (x , y) : End point (Valid data range : -999999999 to 999999999)
 Unit of data : Input unit (inch/mm)
 In G90 mode, please specified the position of the end point.
 In G91 mode, please specified the amount of movement to the end point.
- r : Radius of arc (Valid data range : ± 1 to ± 999999999)
 Unit of data : Input unit (inch/mm)
 -Should be specified by positive number to assign the arc with an angle of 180° or less.
 -Should be specified by negative number to assign the arc with an angle of 180° or larger.
 -R0 cannot be specified.
- d : Tool diameter (Valid data range : -999999999 to 999999999)
 Unit of data : Input unit (inch/mm)
 -Should be specified by positive number to punch outside the arc.
 -Should be specified by negative number to punch inside the arc.
 -Should specify P0 to punch on the arc.
- p : Nibbling pitch (Valid data range : 1 to 999999999)
 Unit of data : Input unit (inch/mm)
 -This pitch is specified as an arc length by positive number.

Using above command allows workpieces are punched out on a circle with a radius of r in pitches of q , using a tool with a diameter of d .



Moreover, when the distance between the start point and the end point is larger than a circular arc diameter ($2r$), workpiece is punched out with a start point on a circle with a radius of r . In this case, therefore, the tool cannot arrive in the end point specified.

Example

• When $r < 0$ and $d < 0$

N1111 G90 G00 X95. Y90.

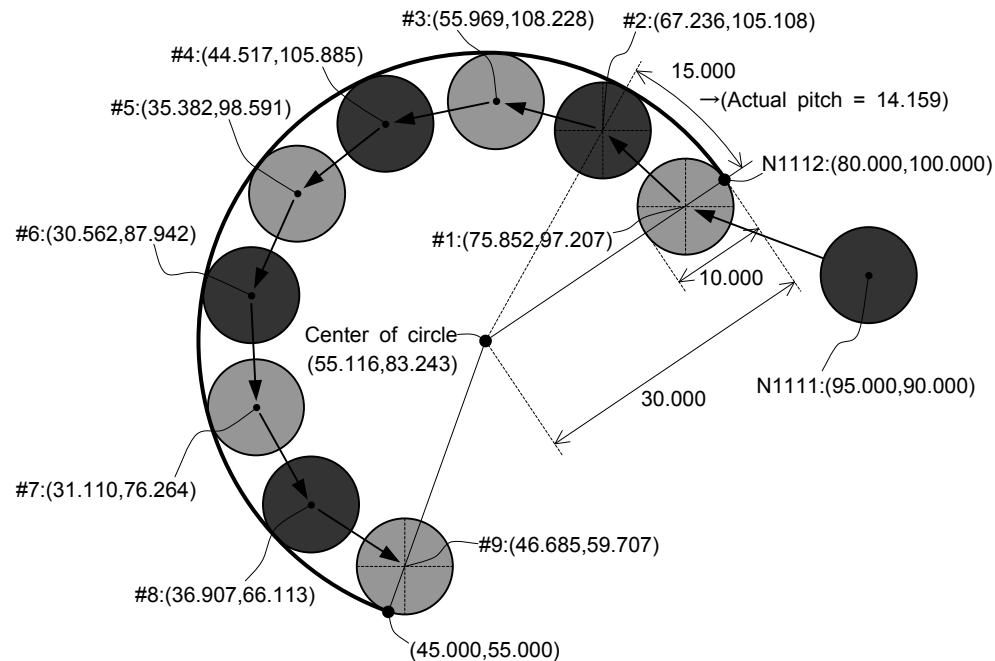
N1112 G72 X80. Y100.

N1113 G47 X45. Y55. R-30. P-10. Q15.

Coordinates are expressed by IS-B

#1 to #9 : Order of punch motion in N1113

→ : Path of tool



NOTE

- 1 When the start and end points of an arc are the same in a G46 or G47 command, if either the radius of the arc is zero or the interval exceeds the circumference, alarm (PS4511) is issued.
- 2 If address R and address Q are not specified, alarm (PS4511) is issued.

15.7.3 Controlling the C-axis

When any of the tools which can be rotated around the C-axis are specified in parameters No. 16370 to No. 16389 are selected in a block containing a G45, G46, or G47 command or a preceding block, the C-axis is automatically controlled in the following way in the block containing the G45, G46, or G47 command.

When a G45 command is specified, the C-axis is controlled at the first punch position so that the C-axis is oriented to the angle formed by the X-axis and the line connecting the start and end points.

When a G46 or G47 command is specified, the C-axis is controlled at each punch position so that one side of the tool is oriented to the center of the arc.

15.8 Y-AXIS CRACK CANCEL

The M-codes which is set by parameters (No. 16610 to 16614) are commanded, the crack between work coordinate system and machine coordinate system of Y-axis repositioning motion is canceled.

(Example 1)

Y1525 M30 ;

Y-axis moves including the crack of repositioning.

(Example 2)

Y1525 ;

M30 ;

The work coordinate system is preset to the machine coordinate system to cancel the crack of repositioning by M30.

Limitation

The cancel M-code must be different from another special meaning M-code.

But M02 and M30 are possible.

16

COMPENSATION FUNCTION

Chapter 16, "COMPENSATION FUNCTION", consists of the following sections:

16.1 SCALING (G50, G51)	160
16.2 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1, or G150,G151,G152)	166
16.3 OVERVIEW OF TOOL COMPENSATION (G40-G42)	170
16.4 DETAILS OF TOOL RADIUS COMPENSATION	175
16.5 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	223
16.6 COORDINATE SYSTEM ROTATION (G84, G85)	224

16.1 SCALING (G50, G51)

Overview

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

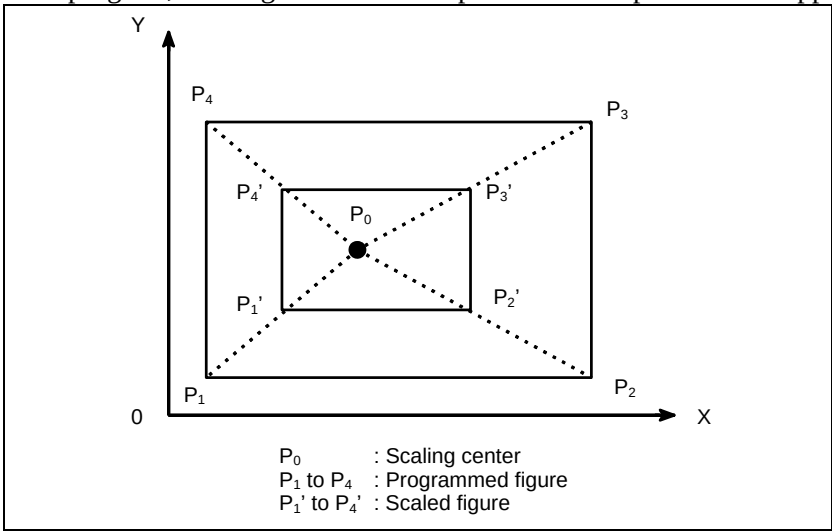


Fig. 16.1 (a) Scaling

Format

Scaling up or down along all axes at the same rate of magnification
(When bit 6 (XSC) of parameter No. 5400 = 0)

Format	Meaning of command
G51 IP_P_ ; Scaling start : : : } Scaling is effective. (Scaling mode)	IP_ : Absolute command for center coordinate value of scaling P_ : Scaling magnification
G50 ; Scaling cancel	

Scaling up or down along each axes at a different rate of magnification (mirror image)

(When bit 6 (XSC) of parameter No. 5400 = 1)

Format	Meaning of command
G51 IP_ I_ J_ K_; Scaling start	IP_ : Absolute command for center coordinate value of scaling
} Scaling is effective. (Scaling mode)	I_ J_ K_ : Scaling magnification for basic 3 axes (X, Y, and Z axes) respectively
G50 ; Scaling cancel	

CAUTION

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

NOTE

- 1 Entering electronic calculator decimal point input mode (bit 0 (DPI) of parameter No. 3401 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
- 2 Setting the least input increment equal to 10 times the least command increment (bit 7 (IPR) of parameter No. 1004 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
- 3 An attempt to specify 0 as a magnification rate causes alarm PS0142, "ILLEGAL SCALE RATE" to occur in a G51 block.

Explanation

- Axis for which scaling is to be enabled

For the axis for which scaling is to be enabled, set bit 0 (SCL) of parameter No. 5401 to 1.

- Minimum unit of scaling magnification

Least input increment of scaling magnification is: 0.001 or 0.00001.

It is 0.00001 (one hundred thousandth) if bit 7 (SCR) of parameter No. 5400 is 0 and 0.001 if it is 1.

- Scaling center

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

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

CAUTION

With the move command subsequent to the G51 block, execute an absolute (G90 mode) position command.

If no absolute position command is executed after the G51 block, the position assumed when G51 is specified is assumed the scaling center; once an absolute position command is executed, the scaling center assumes the coordinates specified in the G51 block, after that block.

- Scaling along each axis at the same rate of magnification

Set bit 6 (XSC) of parameter No. 5400 to 0.

If the scaling magnification P is not specified, the magnification set in parameter No. 5411 is used.

Decimal point input is not accepted as the magnification P. If decimal point input is made, alarm PS0007, "ILLEGAL USE OF DECIMAL POINT" will occur.

A negative value cannot be specified as the magnification P. If a negative value is specified, alarm PS0006, "ILLEGAL USE OF MINUS SIGN" will occur.

The allowable magnification range is from 0.00001 to 9999.99999.

- Scaling of each axis, programmable mirror image (negative magnification)

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. The axis subject to the mirror image is the one that contains the scaling center.

Set bit 6 (XSC) of parameter No. 5400 to 1 to validate each axis scaling (mirror image).

Using I, J, and K, specify the scaling magnifications for the basic 3 axes (X to Z axes). Use parameter No. 1022 to specify which axes to use as the basic 3 axes. For those of the X to Z axes for which I, J, and K are not specified and for axes other than the basic 3 axes, the magnification set with parameter No. 5421 is used.

A value other than 0 must be set to parameter No. 5421.

Decimal point programming can not be used to specify the rate of magnification (I, J, K).

Magnification can be set within the range of ± 0.00001 to ± 9999.99999 .

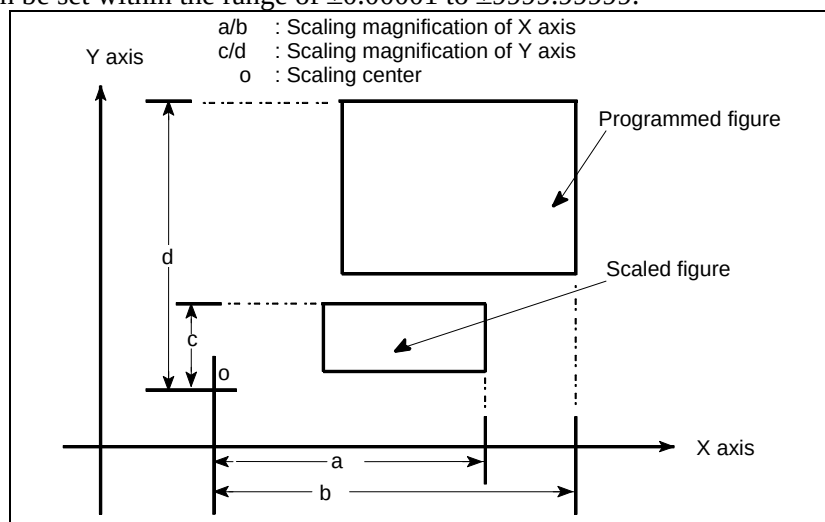


Fig. 16.1 (b) Scaling of each axis

⚠ CAUTION

Specifying the following commands at the same time causes them to be executed in the order indicated below:

- <1> Scaling (G51) (including a mirror image with a negative magnification)
- <2> Mirror image due to the external switch of the CNC or the settings of the CNC

- Scaling of circular interpolation

Even if different magnifications are applied to each axis in circular interpolation, the tool will not trace an ellipse.

```
G90 G00 X0.0 Y100.0 Z0.0;
```

```
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
```

(A magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.)

```
G02 X100.0 Y0.0 I0 J-100.0 F500;
```

Above commands are equivalent to the following command:

```
G90 G00 X0.0 Y100.0 Z0.0;
```

```
G02 X200.0 Y0.0 I0 J-100.0 F500;
```

(Because the end point is not on an arc, spiral interpolation is assumed.)

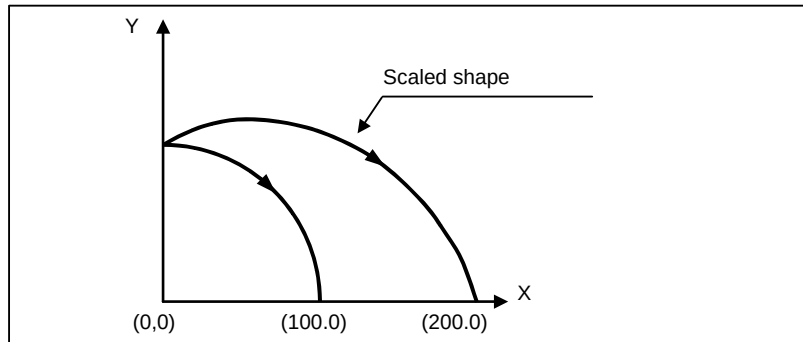


Fig. 16.1 (c) Scaling for circular interpolation1

Even for an R-specified arc, scaling is applied to each of I, J, and K after the radius value (R) is converted into a vector in the center direction of each axis.

If, therefore, the above G02 block contains the following R-specified arc, the operation will be same as that in which I and J are specified.

```
G02 X100.0 Y0.0 R100.0 F500 ;
```

- Scaling and coordinate system rotation

If both scaling and coordinate system rotation are specified at the same time, scaling is performed first, followed by coordinate system rotation. In this case, scaling is effective to the rotation center as well.

To specify both of them, specify scaling first and then coordinate system rotation. To cancel them, specify them in the reverse order.

Example

Main program

```
O1
G90 G00 X20.0 Y10.0 ;
M98 P1000 ;
G51 X20.0 Y10.0 I3000 J2000 ; (×3 in the X direction and ×2 in the Y direction)
M98 P1000 ;
G17 G84 X35.0 Y20.0 R30. ;
M98 P1000 ;
G85 ;
G50 ;
M30 ;
```

Subprogram

```
O1000 ;
G01 X20.0 Y10.0 F500 ;
G01 X50.0 ;
G01 Y30.0 ;
G01 X20.0 ;
G01 Y10.0 ;
M99 ;
```

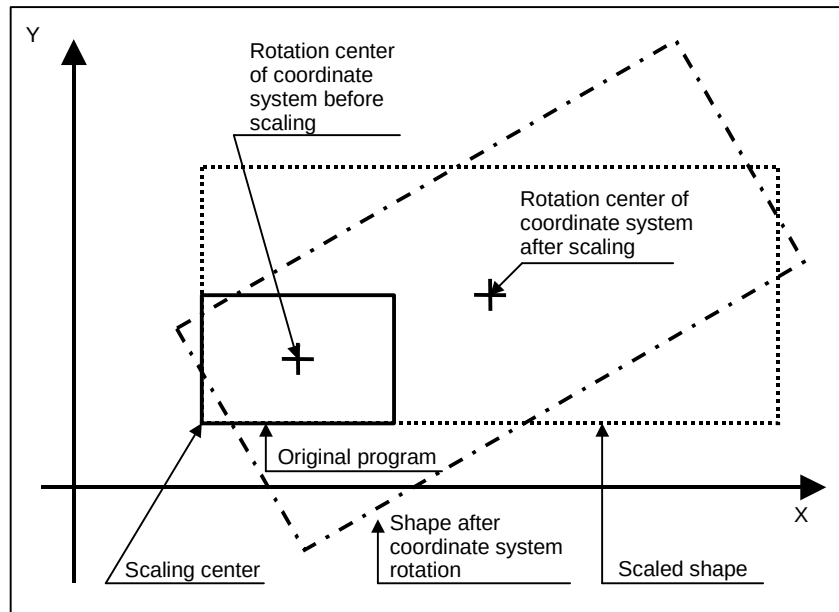


Fig. 16.1 (d) Scaling and coordinate system rotation

Limitation

- Tool compensation

This scaling is not applicable to tool radius compensation values (Fig. 16.1 (e)).

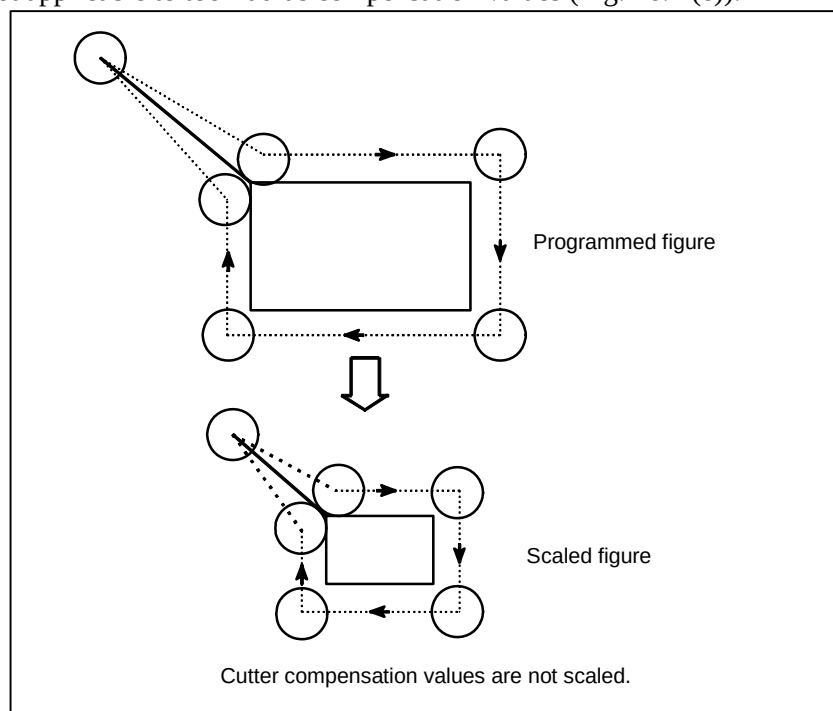


Fig. 16.1 (e) Scaling during tool compensation

⚠ CAUTION

- 1 If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
- 2 Before specifying the G code for reference position return (G28, etc.) or coordinate system setting (G52 to G59, G92, etc.), cancel the scaling mode.
- 3 If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In this case, the block is regarded as a no movement block, and therefore, it may affect the tool movement by tool compensation. See the description of tool compensation.
- 4 Refrain from scaling on a rotation axis for which the rollover function is enabled. Otherwise, the tool may rotate in a short-cut manner, possibly resulting in unexpected movement.

NOTE

- 1 The position display represents the coordinate value after scaling.
- 2 When a mirror image was applied to one axis of the specified plane, the following results:
 - (1) Circular command Direction of rotation is reversed.
 - (2) Tool radius compensation Offset direction is reversed.
 - (3) Coordinate system rotation Rotation angle is reversed.

Example

Sample program of a scaling in each axis

```

O1;
G51 X20.0 Y10.0 I750 J250; (× 0.75 in the X direction, × 0.25 in the Y direction)
G00 G90 X60.0 Y50.0;
G01 X120.0 F100;
G01 Y90;
G01 X60;
G01 Y50;
G50;
M30;
  
```

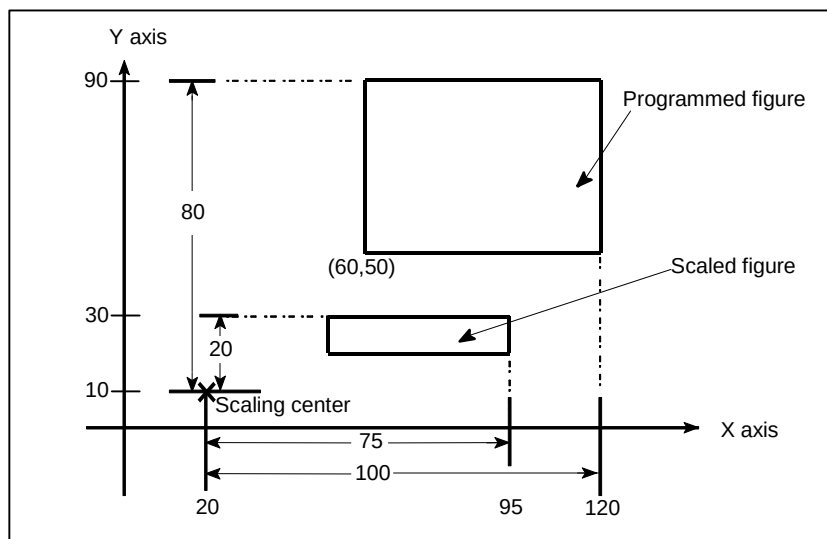


Fig. 16.1 (f) Program example of scaling in each axis

16.2 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1, or G150,G151,G152)

Overview

When a tool with a rotation axis (C-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C-axis is always perpendicular to the tool path (Fig. 16.2 (a)).

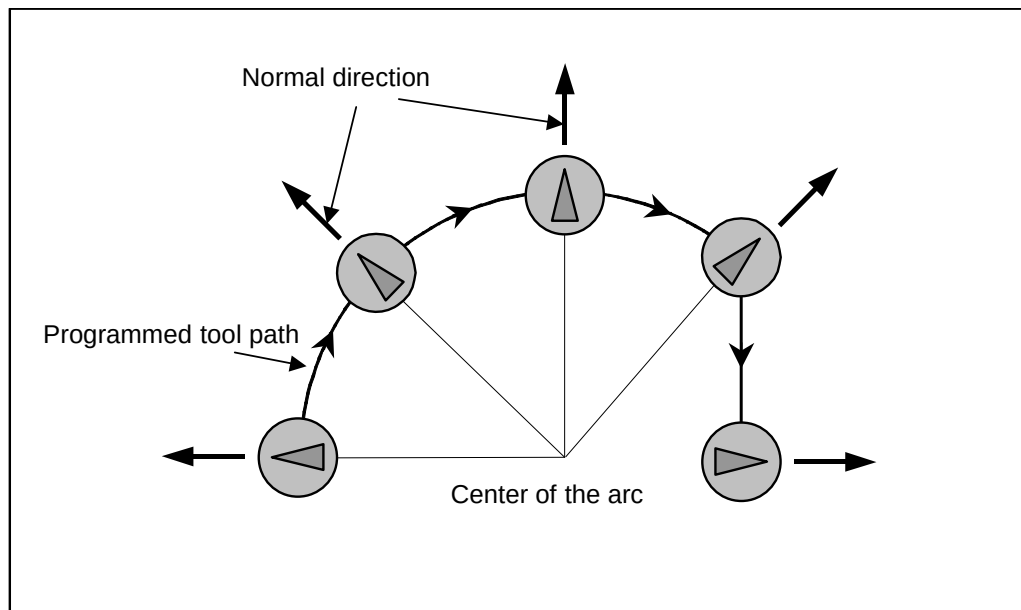


Fig. 16.2 (a) Sample Movement of the tool

Format

G41.1 (G151) ; Normal direction control, left
G42.1 (G152) ; Normal direction control, right
G40.1 (G150) ; Cancel normal direction control

The normal direction control, left (G41.1) command is used when the workpiece is on the right side of the tool as viewed while you are looking into the tool's way.

Once either G41.1 or G42.1 is issued, normal direction control is enabled (normal direction control mode).

Issuing G40.1 cancels the normal direction control mode.

Explanation

- Angle of the C axis

When viewed from the center of rotation around the C-axis, the angular displacement about the C-axis is determined as shown in Fig. 16.2 (b). The positive side of the X-axis is assumed to be 0°, the positive side of the Y-axis is 90°, the negative side of the X-axis is 180°, and the negative side of the Y-axis is 270°.

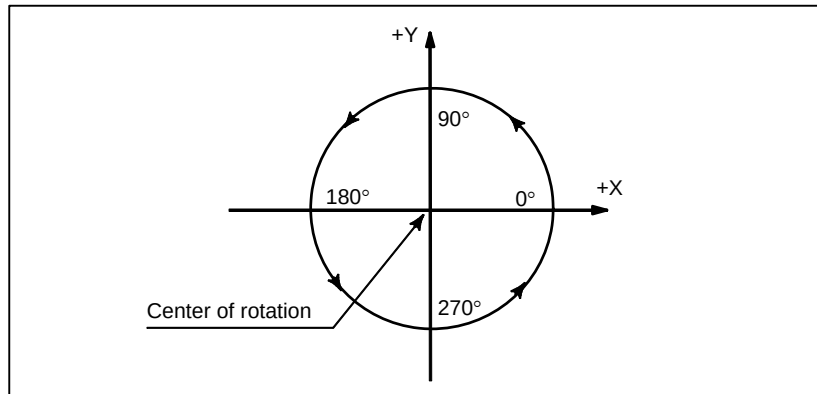


Fig. 16.2 (b) Angle of the C axis

- Normal direction control of the C axis

When the cancel mode is switched to the normal direction control mode, the C-axis becomes perpendicular to the tool path at the beginning of the block containing G41.1 or G42.1.

In the interface between blocks in the normal direction control mode, a command to move the tool is automatically inserted so that the C-axis becomes perpendicular to the tool path at the beginning of each block. The tool is first oriented so that the C-axis becomes perpendicular to the tool path specified by the move command, then it is moved along the X- and Y axes.

In the tool compensation mode, the tool is oriented so that the C-axis becomes perpendicular to the tool path created after compensation.

In single-block operation, the tool is not stopped between a command for rotation of the tool and a command for movement along the X- and Y-axes. A single-block stop always occurs after the tool is moved along the X- and Y-axes.

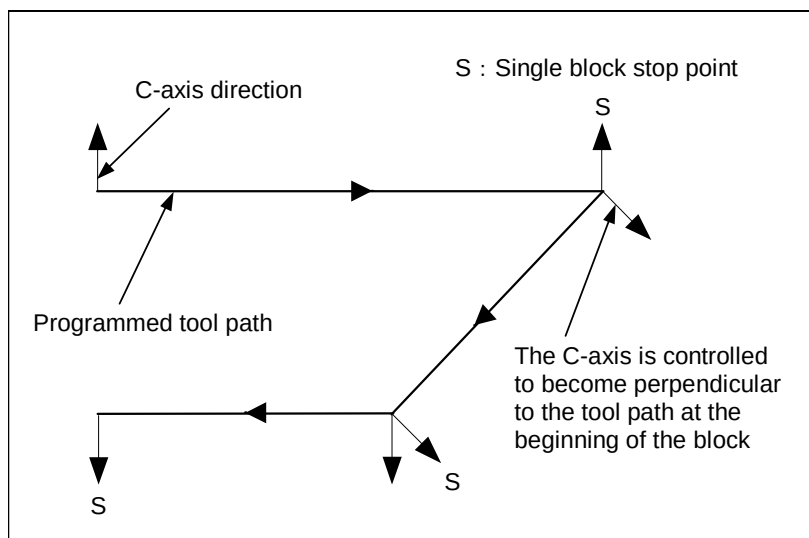


Fig. 16.2 (c) Point at which a single-block stop occurs in the normal direction control mode

Before circular interpolation is started, the C-axis is rotated so that the C-axis becomes normal to the arc at the start point. During circular interpolation, the tool is controlled so that the C-axis is always perpendicular to the tool path determined by circular interpolation.

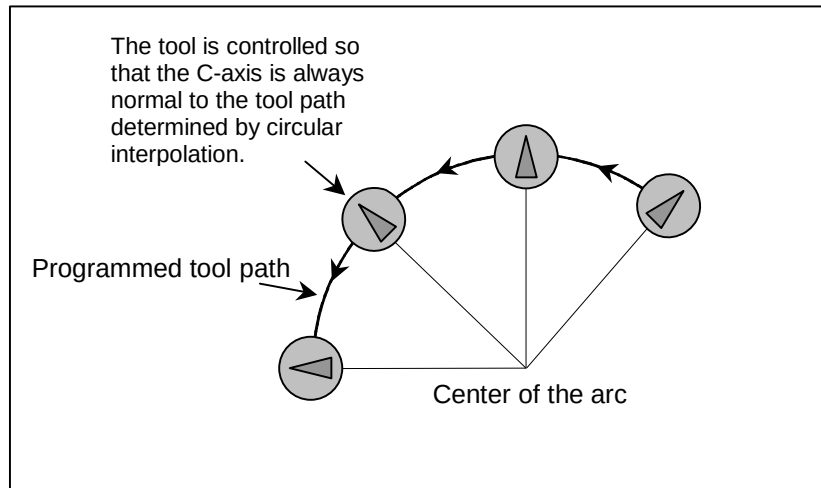


Fig. 16.2 (d) Normal direction control of the circular interpolation

NOTE

During normal direction control, the C axis always rotates through an angle less than 180 deg. I.e., it rotates in whichever direction provides the shorter route.

- C axis feedrate

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X-and Y-axes in rapid traverse (G00) mode, the rapid traverse feedrate is applied.

The feedrate of the C axis during circular interpolation is defined by the following formula.

$$F \times \frac{\text{Amount of movement of the C axis (deg)}}{\text{Length of arc (mm or inch)}} \quad (\text{deg/min})$$

F : Feedrate (mm/min or inch/min) specified by the corresponding block of the arc

Amount of movement of the C axis : The difference in angles at the beginning and the end of the block.

NOTE

If the feedrate of the C axis exceeds the maximum cutting speed of the C axis specified to parameter No. 1430, the feedrate of each of the other axes is clamped to keep the feedrate of the C axis below the maximum cutting speed of the C axis.

- Normal direction control axis

A C-axis to which normal-direction control is applied can be assigned to any axis with parameter No. 5480.

- Angle for which figure insertion is ignored

When the rotation angle to be inserted, calculated by normal-direction control, is smaller than the value set with parameter No. 5482, the corresponding rotation block is not inserted for the axis to which normal-direction control is applied. This ignored rotation angle is added to the next rotation angle to be inserted, the total angle being subject to the same check at the next block.

If an angle of 360 degrees or more is specified, the corresponding rotation block is not inserted.

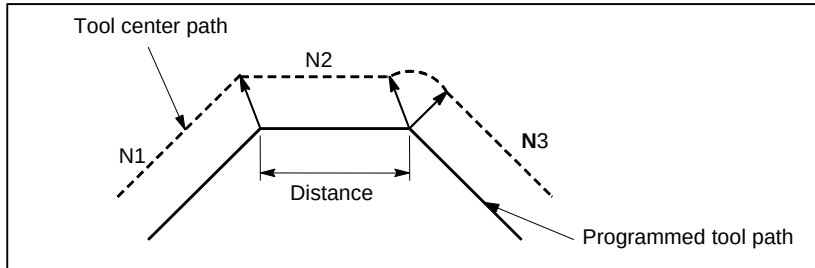
If an angle of 180 degrees or more is specified in a block other than that for circular interpolation with a C-axis rotation angle of 180 degrees or more, the corresponding rotation block is not inserted.

- Movement for which arc insertion is ignored

Specify the maximum distance for which machining is performed with the same normal direction as that of the preceding block.

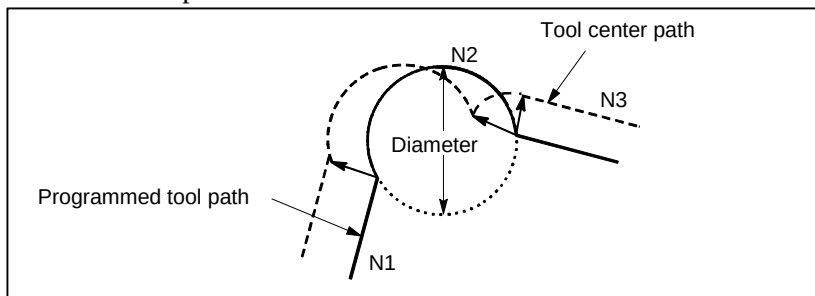
- Linear movement

When distance N2, shown below, is smaller than the set value, machining for block N2 is performed using the same direction as that for block N1.



- Circular movement

When the diameter of block N2, shown below, is smaller than the set value, machining for block N2 is performed using the same normal direction as that for block N1. And control as compensation along the circular arc is not performed.



NOTE

- 1 Do not specify any command to the C axis during normal direction control. Any command specified at this time is ignored.
- 2 Before processing starts, it is necessary to correlate the workpiece coordinate of the C axis with the actual position of the C axis on the machine using the coordinate system setting (G92) or the like.
- 3 Helical cutting cannot be specified in the normal direction control mode.
- 4 Normal direction control cannot be performed by the G53 move command.
- 5 The C-axis must be a rotation axis.

- T and C commands during normal-line direction control

- (1) During normal-line direction control, the T command results in an alarm (PS4606) except when the TANDC parameter (bit 7 of parameter No. 16263) is 1, in which case a single-tool command is valid.
- (2) During normal-line direction control, the C command is ignored. A C-axis offset command is also ignored if a C-axis index tool is specified.
- (3) A C-axis offset command specified before a normal-line direction control mode is entered remains valid after the normal-line direction control mode is entered.
- (4) Usually, a check is made on T commands during normal-line direction control. If a T command is already specified when a normal-line direction control mode is entered, a check is not made on the T command. Normal-line direction control is carried out no matter what tool type is specified.

Under normal direction control, a punch press shears a sheet while holding the sheet and moving the tool with interpolation. The cutting of an arc should be programmed as follows: Linear positioning is

performed with the C-axis set in the direction normal to the tangential direction at the start point of the arc; The sheet is held by the PF signal; A C-axis normal-direction movement is made with circular interpolation. The program should be created so that an arc and a straight line are connected by a tangent even in the normal direction control mode.

(Example)

G92 X1525 Y1525;

G90;

T201;

G70 X500 Y200; ← Positioning for creating a tangent at the start point of the arc
(G70: Positioning and press off command)

G41.1;

G00 X500 Y500; ← Positioning to the arc start point with the C-axis set in the
direction normal to the direction of the arc tangent

G03 X421.14 Y459.171-50F5000;

G40.1;

16.3 OVERVIEW OF TOOL COMPENSATION (G40-G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 16.3 (a)).

To make an offset as large as the radius of the tool, CNC first creates an offset vector with a length equal to the radius of the tool (start-up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head positions to the center of the tool.

If a linear interpolation or circular interpolation command is specified after start-up, the tool path can be shifted by the length of the offset vector during machining.

To return the tool to the start point at the end of machining, cancel the tool compensation mode.

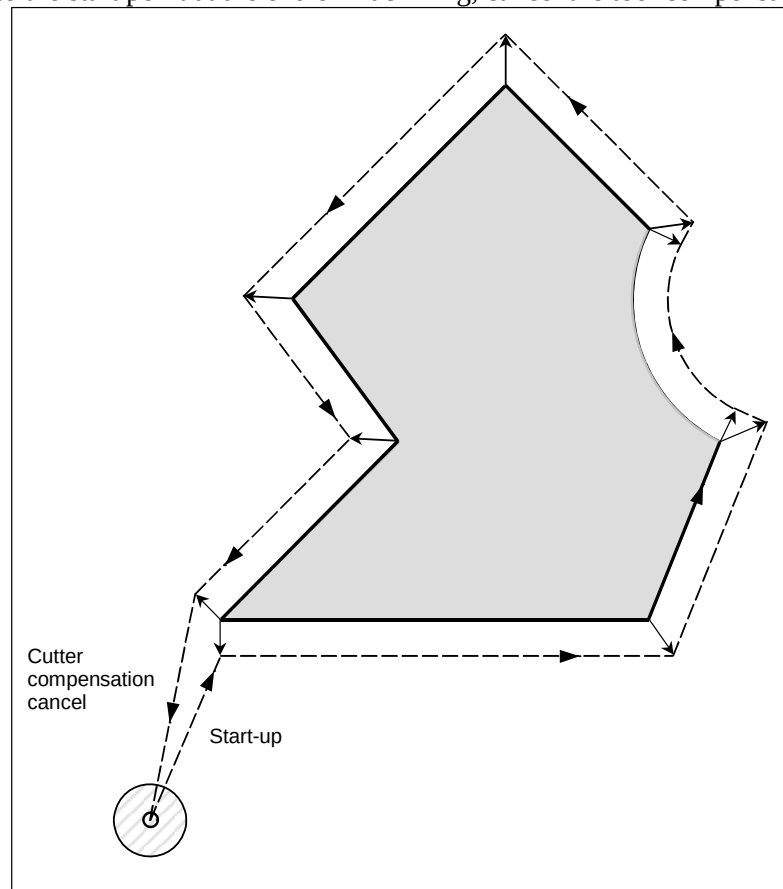


Fig. 16.3 (a) Outline of tool compensation

Format**- Start up (tool compensation start)****G00(or G01)G41(or G42) IP_D_;**

G41 : Tool compensation left (Group 07)

G42 : Tool compensation right (Group 07)

IP_ : Command for axis movement

D_ : Code for specifying as the tool compensation value (1 to 32) (D code)

- Tool compensation cancel (offset mode cancel)**G40 IP_;**G40 : Tool compensation cancel (Group 07)
(Offset mode cancel)

IP_ : Command for axis movement

- Selection of the offset plane

Offset plane	Command for plane selection	IP_
XpYp	G17 ;	Xp Yp_
ZpXp	G18 ;	Xp Zp_
YpZp	G19 ;	Yp Zp_

Explanation**- Offset cancel mode**

At the beginning when power is applied the control is in the cancel mode. In the cancel mode, the vector is always 0, and the tool center path coincides with the programmed path.

- Start-up

When a tool compensation command (G41 or G42, D code other than 0) is specified in the offset cancel mode, the CNC enters the offset mode.

Moving the tool with this command is called start-up.

Specify positioning (G00) or linear interpolation (G01) for start-up.

If circular interpolation (G02, G03) is specified, alarm PS0034 occurs.

For the start-up and subsequent blocks, the CNC prereads as many blocks as the number of preread blocks set in the parameter No. 19625.

- Offset mode

In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03).

If three or more blocks that move the tool cannot be read in offset mode, the tool may make either an excessive or insufficient cut.

If the offset plane is switched in the offset mode, alarm PS0037 occurs and the tool is stopped.

- Offset mode cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the CNC enters the offset cancel mode, and the action of this block is called the offset cancel.

1. G40 has been commanded.

2. 0 has been commanded as the offset number for tool compensation (D code).

When performing offset cancel, circular arc commands (G02 and G03) are not available. If these commands are specified, an PS0034 is generated and the tool stops. In the offset cancel, the control executes the instructions in that block and the block in the tool compensation buffer.

In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without reading the next block.

Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for tool compensation.

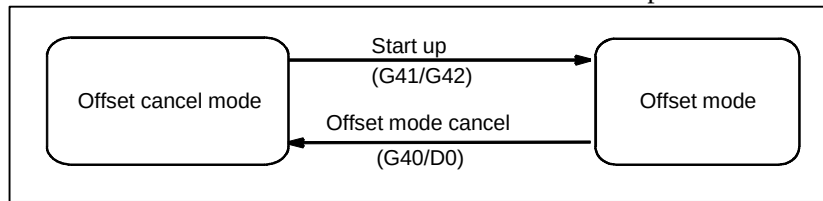


Fig. 16.3 (b) Changing the offset mode

- Change of the tool compensation value

In general, the tool compensation value shall be changed in the cancel mode, when changing tools. If the tool compensation value is changed in offset mode, the vector at the end point of the block is calculated for the new tool compensation value.

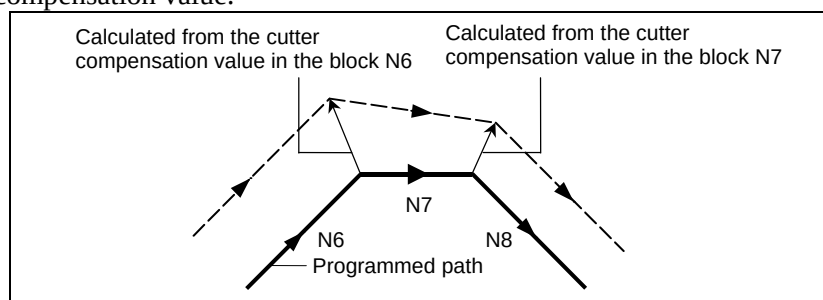


Fig. 16.3 (c) Changing the tool compensation value

- Positive/negative tool compensation value and tool center path

If the compensation value is negative (–), distribution is made for a figure in which G41's and G42's are all replaced with each other on the program. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vice versa.

Fig. 16.3 (d) shows one example.

Generally, the compensation value is programmed to be positive (+).

When a tool path is programmed as in <1>, if the compensation value is made negative (–), the tool center moves as in <2>, and vice versa. Consequently, the same program permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the compensation value.

Applicable if start-up and cancel is A type. (See the descriptions about the start-up of tool compensation.)

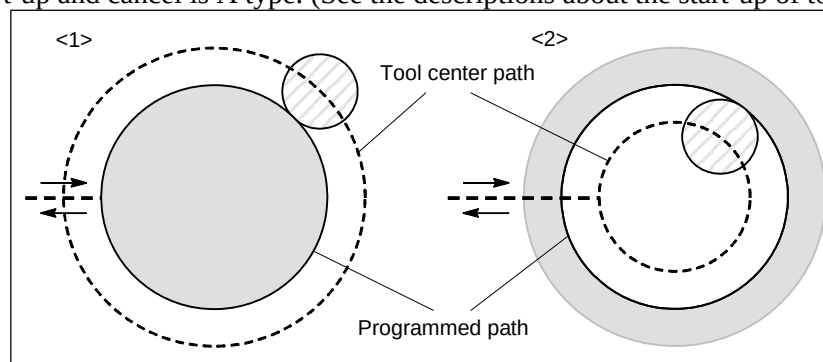


Fig. 16.3 (d) Tool center paths when positive and negative tool compensation values are specified

- Tool compensation value setting

Assign a tool compensation values to the D codes on the MDI panel.

NOTE

The tool compensation value for which the D code corresponds to 0 always means 0.

It is not possible to set the tool compensation value corresponding to D0.

- Valid compensation value range

The valid range of values that can be set as a compensation value is either of the following, depending on the bit 0 (OFA) of parameter No. 5042.

Valid compensation range (metric input)

OFA	Range
1	±9999.99 mm
0	±9999.999 mm

Valid compensation range (inch input)

OFA	Range
1	±999.999 inch
0	±999.9999 inch

The compensation value corresponding to offset No. 0 always means 0. It is not possible to set the compensation value corresponding to offset No. 0.

- Offset vector

The offset vector is the two dimensional vector that is equal to the tool compensation value assigned by D code. It is calculated inside the control unit, and its direction is up-dated in accordance with the progress of the tool in each block.

The offset vector is deleted by reset.

- Specifying a tool compensation value

Specify a tool compensation value with a number assigned to it. The number consists of 1 to 32 after address D (D code).

The D code is valid until another D code is specified.

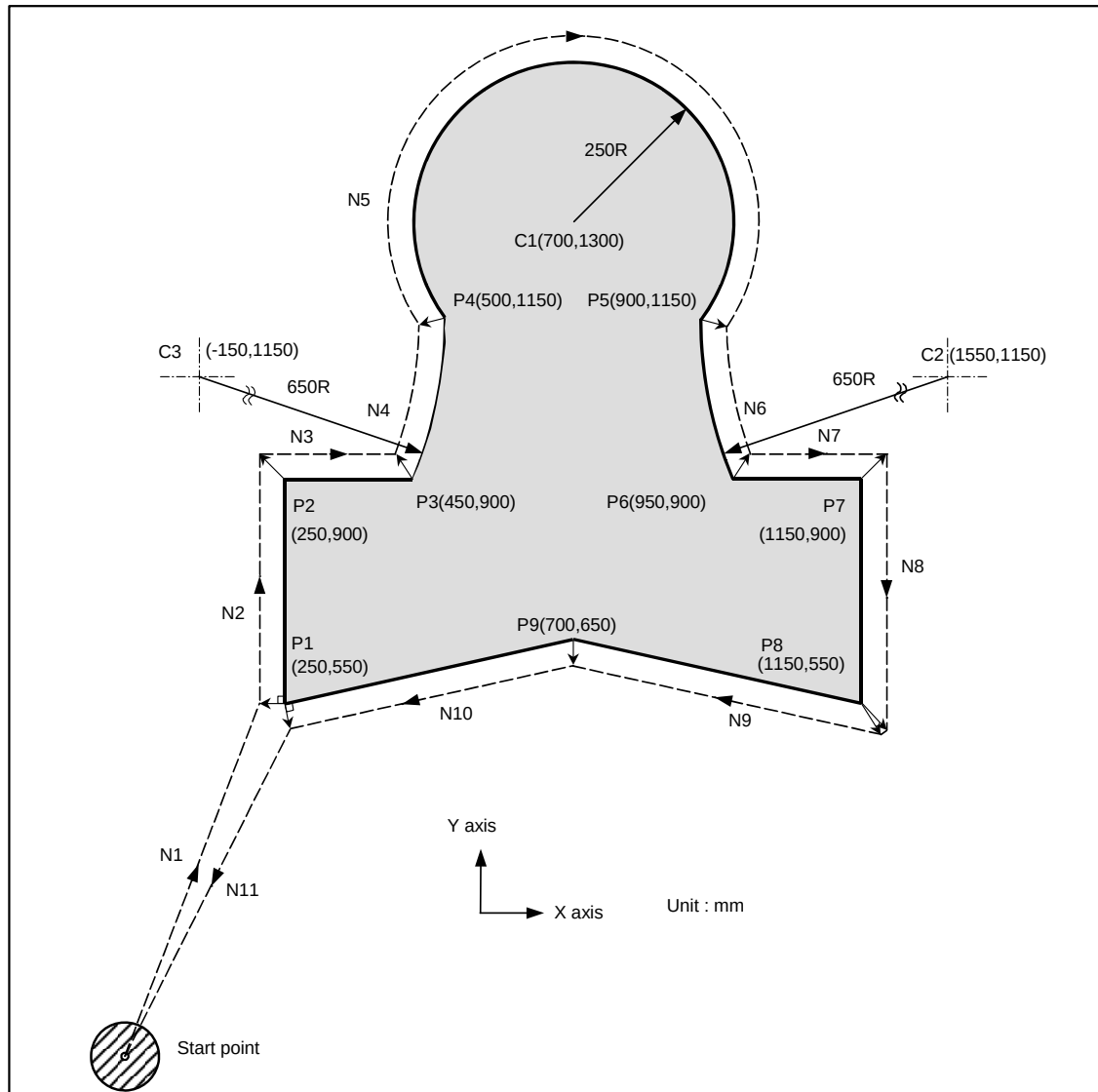
- Plane selection and vector

Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane.

Compensation is not executed for the coordinate of a position which is not in the specified plane. The programmed values are used as they are.

In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.

The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, an PS0037 is displayed and the machine is stopped.

Example

G17 G92 X0 Y0 Z0 ; Specifies absolute coordinates.

The tool is positioned at the start point (X0, Y0, Z0).

N1 G90 G00 G41 D07 X250.0 Y550.0 ; Starts tool compensation (start-up).

The tool is shifted to the left of the programmed path by the distance specified in D07.

In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15 mm).

N2 G01 Y900.0 F150 ; Specifies machining from P1 to P2.

N3 X450.0 ; Specifies machining from P2 to P3.

N4 G03 X500.0 Y1150.0 R650.0 ; Specifies machining from P3 to P4.

N5 G02 X900.0 R-250.0 ; Specifies machining from P4 to P5.

N6 G03 X950.0 Y900.0 R650.0 ; Specifies machining from P5 to P6.

N7 G01 X1150.0 ; Specifies machining from P6 to P7.

N8 Y550.0 ; Specifies machining from P7 to P8.

N9 X700.0 Y650.0 ; Specifies machining from P8 to P9.

N10 X250.0 Y550.0 ; Specifies machining from P9 to P1.

N11 G00 G40 X0 Y0 ; Cancels the offset mode.

The tool is returned to the start point (X0, Y0, Z0).

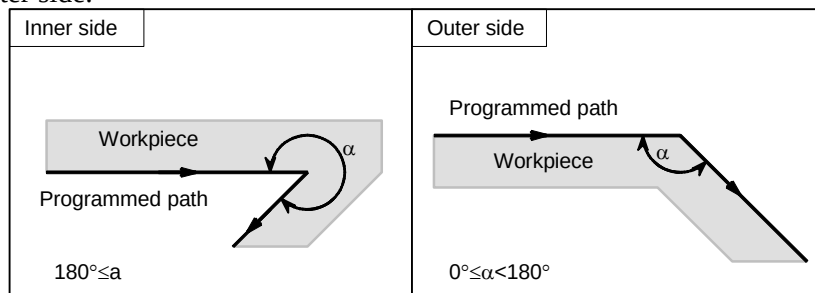
16.4 DETAILS OF TOOL RADIUS COMPENSATION

16.4.1 Overview

The following explanation focuses on the tool compensation.

- Inner side and outer side

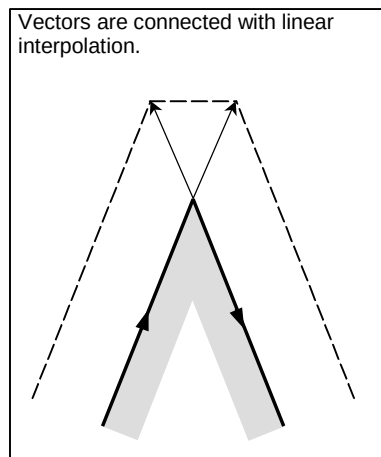
When an angle of intersection of the tool paths specified with move commands for two blocks on the workpiece side is over 180° , it is referred to as "inner side." When the angle is between 0° and 180° , it is referred to as "outer side."



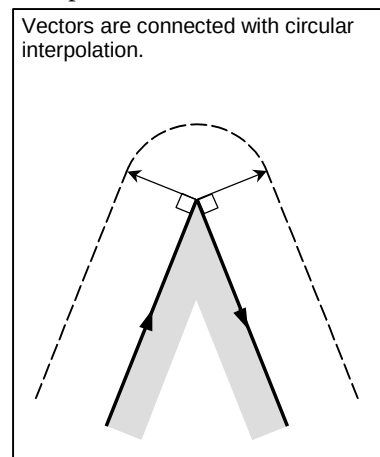
- Outer corner connection method

If the tool moves around an outer corner in tool compensation mode, it is possible to specify whether to connect compensation vectors with linear interpolation or with circular interpolation, using bit 2 (CCC) of parameter No. 19607.

<1> Linear connection type [Bit 2 (CCC) of parameter No. 19607 = 0]



<2> Circular connection type [Bit 2 (CCC) of parameter No. 19607 = 1]



- Cancel mode

The tool compensation enters the cancel mode under the following conditions. (The system may not enter the cancel mode depending on the machine tool.)

<1> Immediately after the power is turned on

<2> When the  key on the MDI panel is pushed

<3> After a program is forced to end by executing M02 or M30

<4> After the tool compensation cancel command (G40) is exercised

In the cancel mode, the compensation vector is set to zero, and the path of the center of tool coincides with the programmed path. A program must end in cancel mode. If it ends in the tool compensation mode, the tool cannot be positioned at the end point, and the tool stops at a location the compensation vector length away from the end point.

NOTE

The operation to be performed when a reset operation is performed during tool compensation differs depending on bit 6 (CLR) of parameter No. 3402.

- If CLR is 0
The system enters the reset state. G41/G42 are retained as the modal code of group 07, but to perform tool compensation, an offset number (D code) must be specified again.
- If CLR is 1
The system enters the clear state. The modal code of group 07 is G40, and to perform tool compensation again, G41/G42 and an offset number (D code) must be specified.

- Start-up

When a block which satisfies all the following conditions is executed in cancel mode, the CNC enters the tool compensation mode. Control during this operation is called start-up.

<1> G41 or G42 is contained in the block, or has been specified to place the CNC in the tool compensation mode.

<2> $0 < \text{compensation number of tool compensation} \leq \text{maximum compensation number}$

<3> Positioning (G00) or linear interpolation (G01) mode

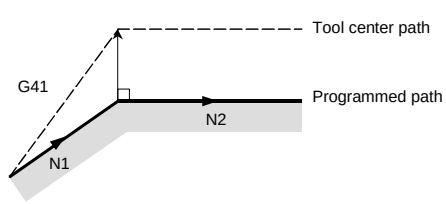
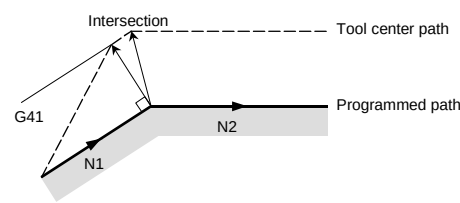
<4> A compensation plane axis command with a travel distance of 0 (except start-up type C) is specified.

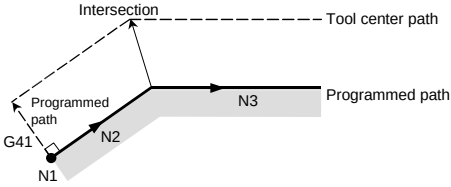
If start-up is specified in circular interpolation (G02, G03) mode, PS0034 will occur.

In a start-up block, workpiece coordinate system switching (G54 to G59) cannot be specified.

As a start-up operation, one of the three types A, B, and C can be selected by setting bit 0 (SUP) of parameter No. 5003 and bit 1 (SUV) of parameter No. 5003 appropriately. The operation to be performed if the tool moves around an inner side is of single type only.

Table 16.4.1 (a) Start-up/cancel operation

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector is output, which is vertical to the block subsequent to the start-up block and the block preceding the cancel block. 
0	1	Type B	A compensation vector is output, which is vertical to the start-up block and the cancel block. An intersection vector is also output. 

SUV	SUP	Type	Operation
1	0 1	Type C	When the start-up block and the cancel block are blocks without tool movement, the tool moves by the tool radius compensation value in the direction vertical to the block subsequent to the start-up block and the block preceding the cancel block.  For a block with tool movement, the tool follows the SUP setting: If it is 0, type A is assumed and if 1, type B is assumed.

- Reading input commands in tool compensation mode

In tool compensation mode, input commands are read from usually three blocks and up to eight blocks depending on the setting of parameter No. 19625 to perform intersection calculation or an interference check, described later, regardless of whether the blocks are with or without tool movement, until a cancel command is received.

To perform intersection calculation, it is necessary to read at least two blocks with tool movement. To perform an interference check, it is necessary to read at least three blocks with tool movement.

As the setting of parameter No. 19625, that is, the number of blocks to read, increases, it is possible to predict overcutting (interference) for up to more subsequent commands. Increases in blocks to read and analyze, however, cause reading and analysis to take more time.

- Ending (canceling) tool compensation

In tool compensation mode, tool compensation is canceled if a block that satisfies at least either one of the following conditions is executed:

<1> G40 is specified.

<2> D00 is specified as the compensation number of tool compensation.

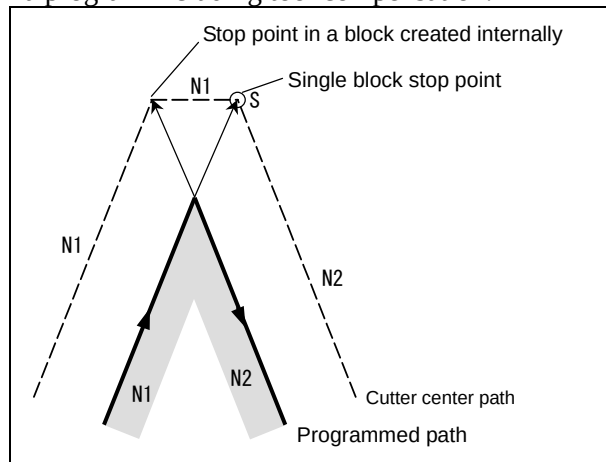
If tool compensation cancel is to be performed, it must not be by a circular command (G02, G03). Otherwise, an alarm will occur.

For a cancel operation, one of three types, A, B, and C, can be selected by appropriately setting bit 0 (SUP) of parameter No. 5003 and bit 1 (SUV) of parameter No. 5003. The operation to be performed if the tool turns around the inside is of a single type.

- Bit 0 (SBK) of parameter No. 5000

When bit 0 (SBK) of parameter No. 5000 is set to 1, a single block stop can be performed in a block created internally for tool compensation.

Use this parameter to check a program including tool compensation.



NOTE

When an M code (auxiliary function), S code, T code (tool function), or B code (second auxiliary function) is specified in the N1 block in the figure above, FIN is not accepted if the tool stops at the stop point in a block created internally (excluding the single block stop point).

- Meaning of symbols

The following symbols are used in subsequent figures:

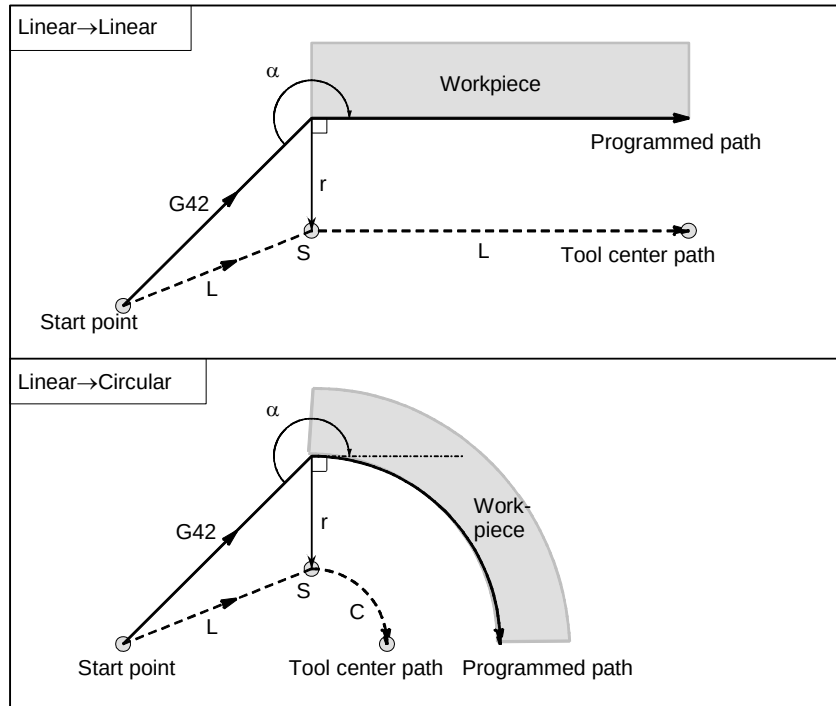
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves along an arc.
- r indicates the tool radius compensation value.
- An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by r.
- ○ indicates the center of the tool.

16.4.2 Tool Movement in Start-up

When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start-up):

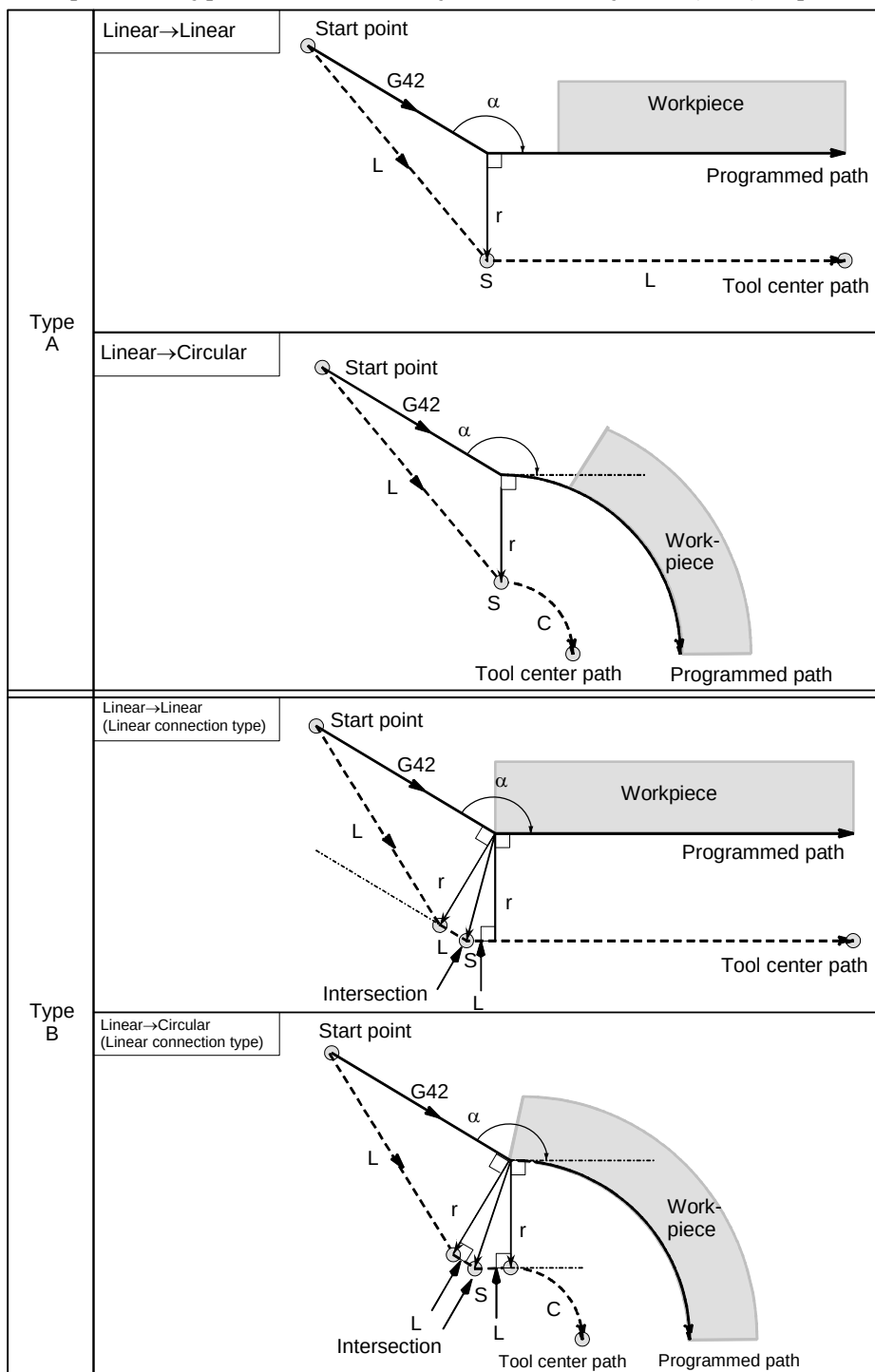
Explanation

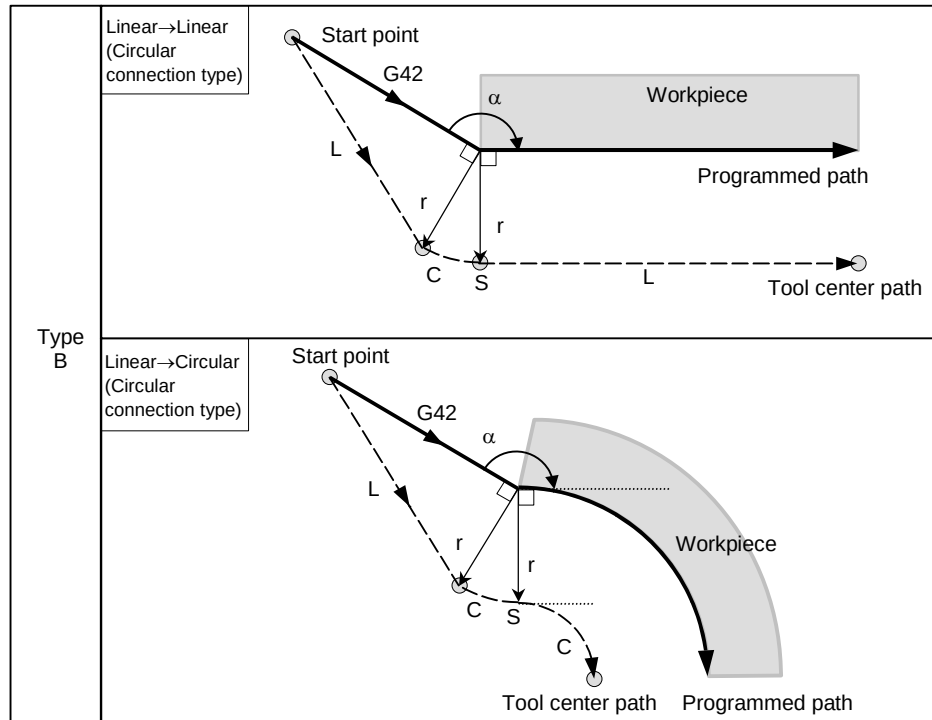
- Tool movement around an inner side of a corner ($180^\circ \leq \alpha$)



- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)

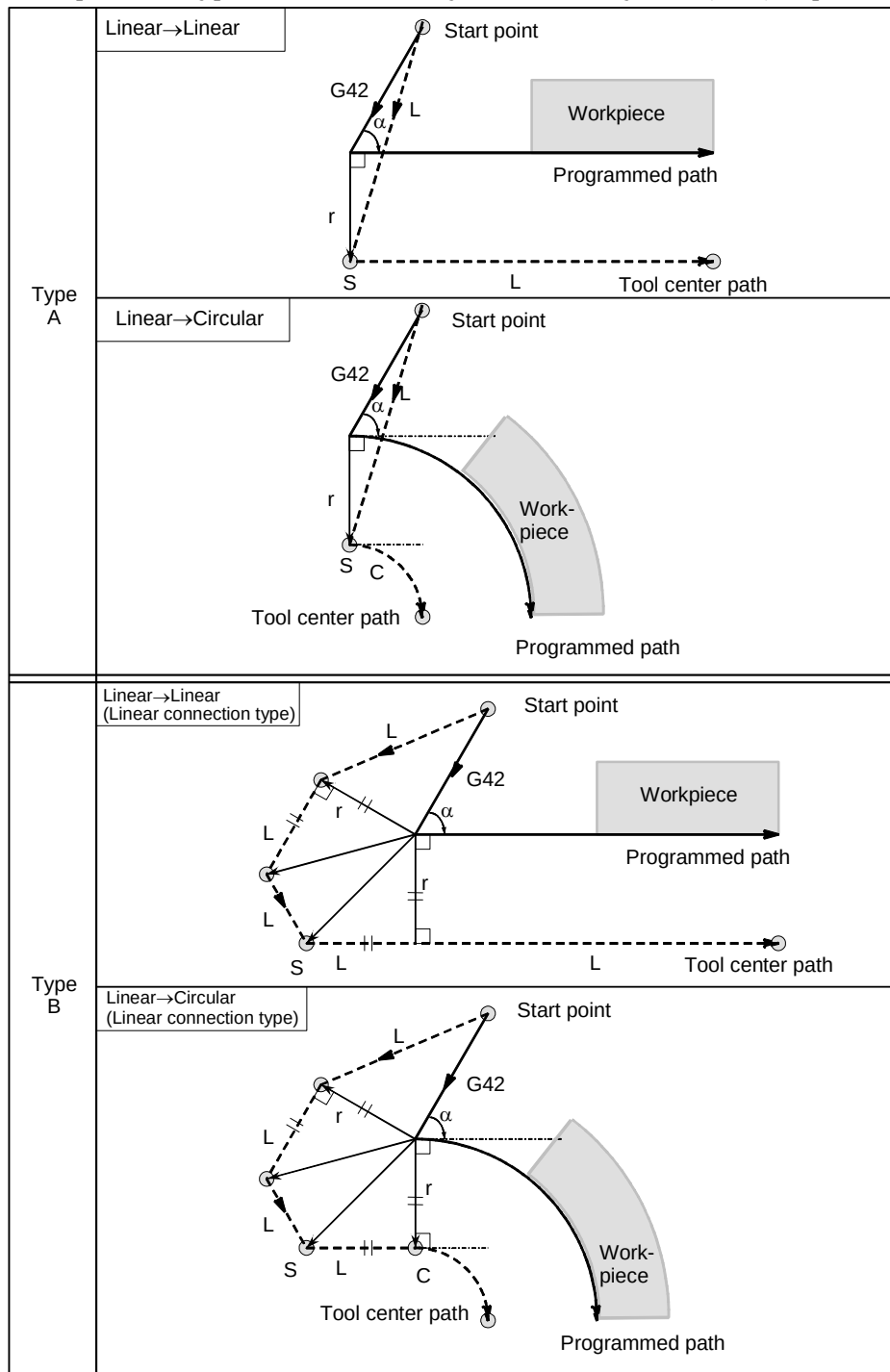
Tool path in start-up has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

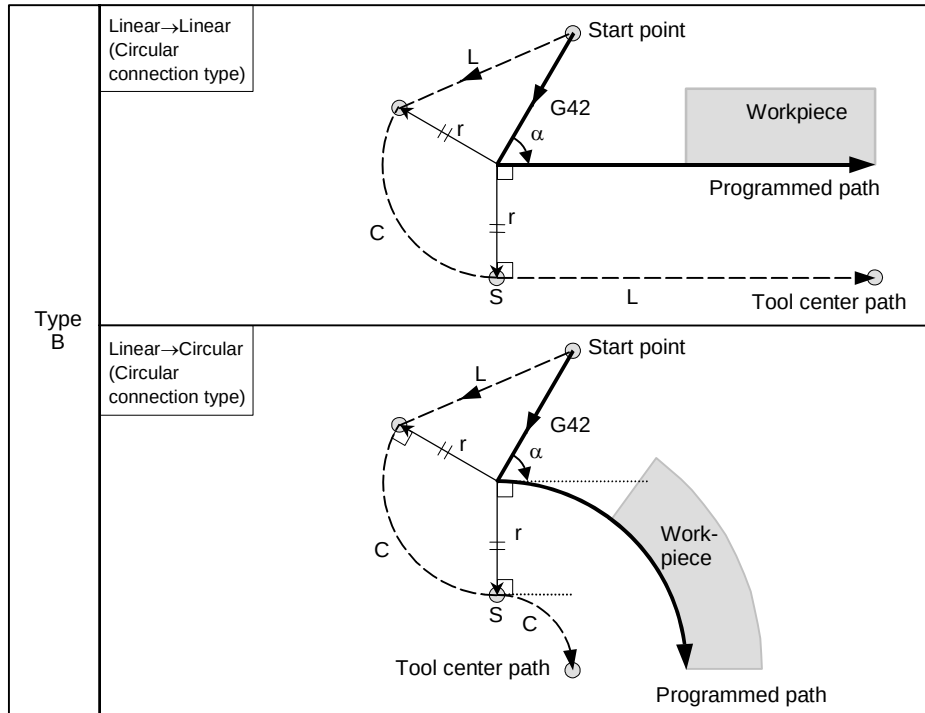




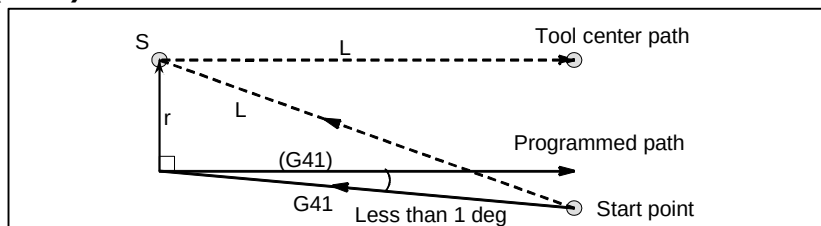
- **Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle ($\alpha < 90^\circ$)**

Tool path in start-up has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.





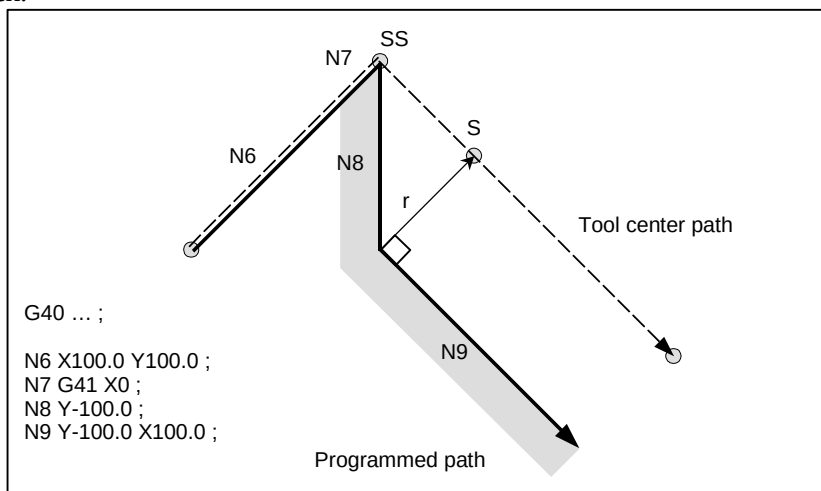
- Tool movement around the outside linear → linear at an acute angle less than 1 degree ($\alpha < 1^\circ$)



- A block without tool movement specified at start-up

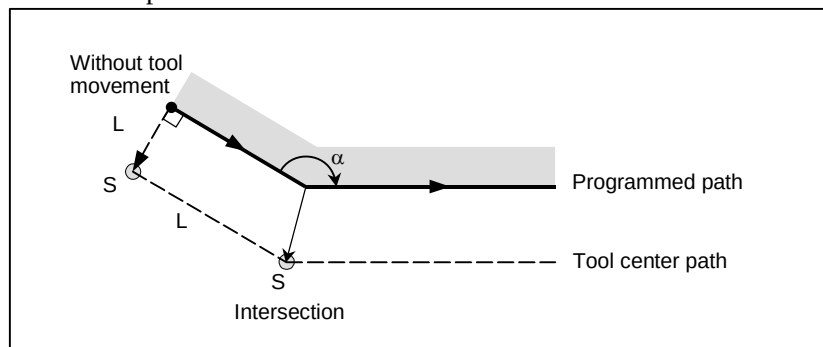
For type A and B

If the command is specified at start-up, the offset vector is not created. The tool does not operate in a start-up block.



For type C

The tool shifts by the compensation value in the direction vertical to the block with tool movement subsequent to the start-up block.



16.4.3 Tool Movement in Offset Mode

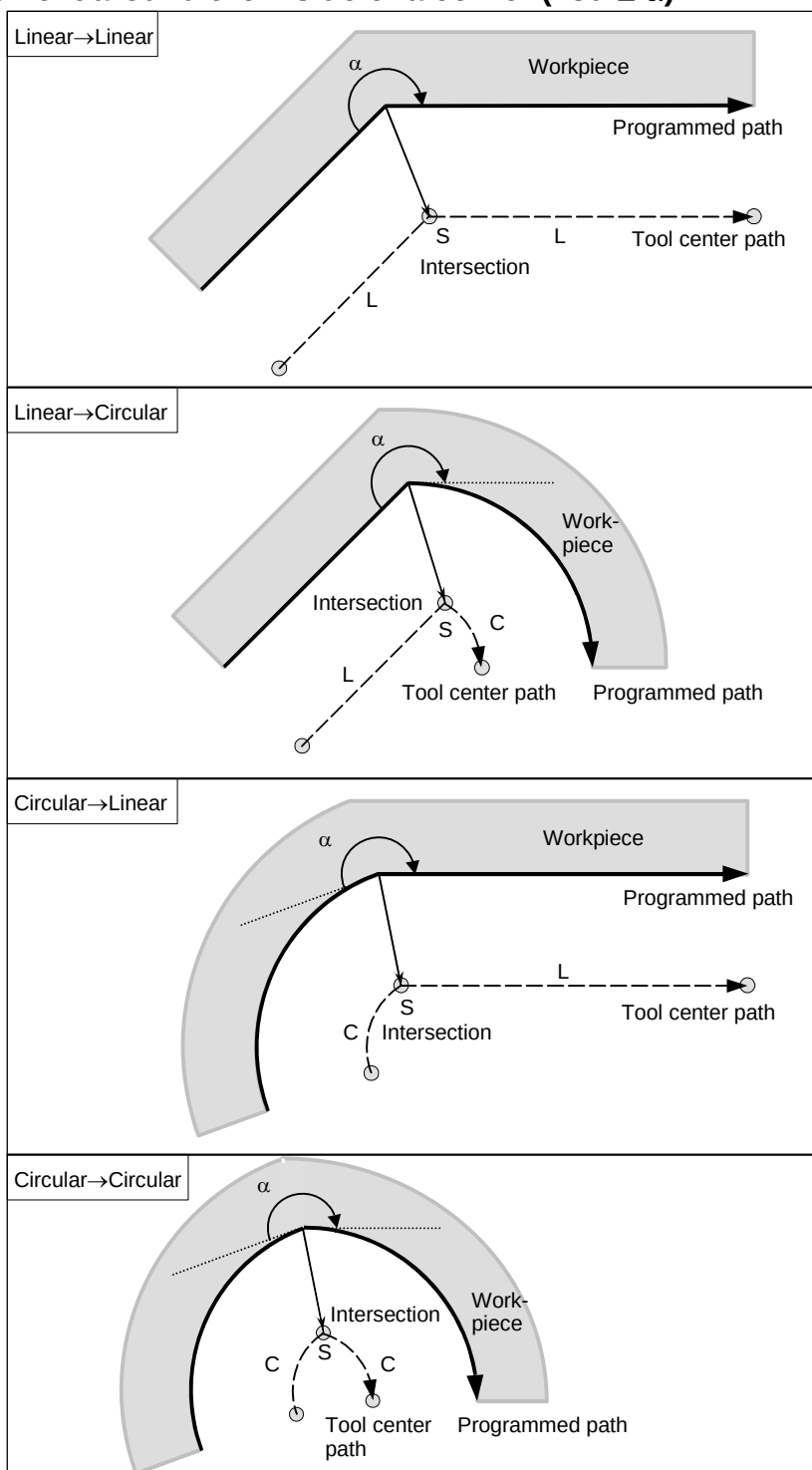
In offset mode, compensation is performed even for positioning commands, not to speak of linear and circular interpolations. To perform intersection calculation, it is necessary to read at least two blocks with tool movement. If, therefore, two or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as auxiliary function independent commands and dwell, are specified in succession, excessive or insufficient cutting may occur because intersection calculation fails. Assuming the number of blocks to read in offset mode, which is determined by parameter No. 19625, to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which intersection calculation is possible is $(N - 2) \geq M$. For example, if the maximum number of blocks to read in offset mode is 5, intersection calculation is possible even if up to three blocks without tool movement are specified.

NOTE

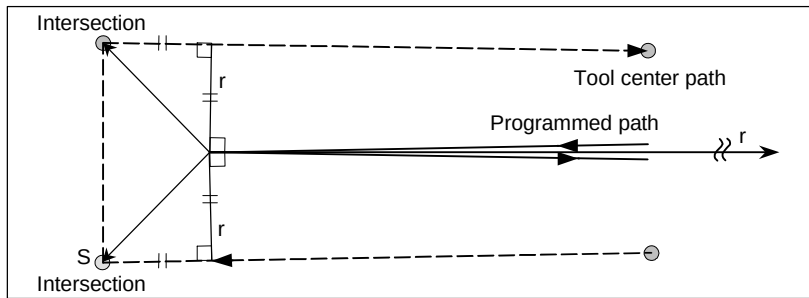
The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the interference check.

If a G or M code in which buffering is suppressed is specified, no subsequent commands can be read before that block is executed, regardless of the setting of parameter No. 19625. Excessive or insufficient cutting may, therefore, occur because of an intersection calculation failure.

- Tool movement around the inside of a corner ($180^\circ \leq \alpha$)

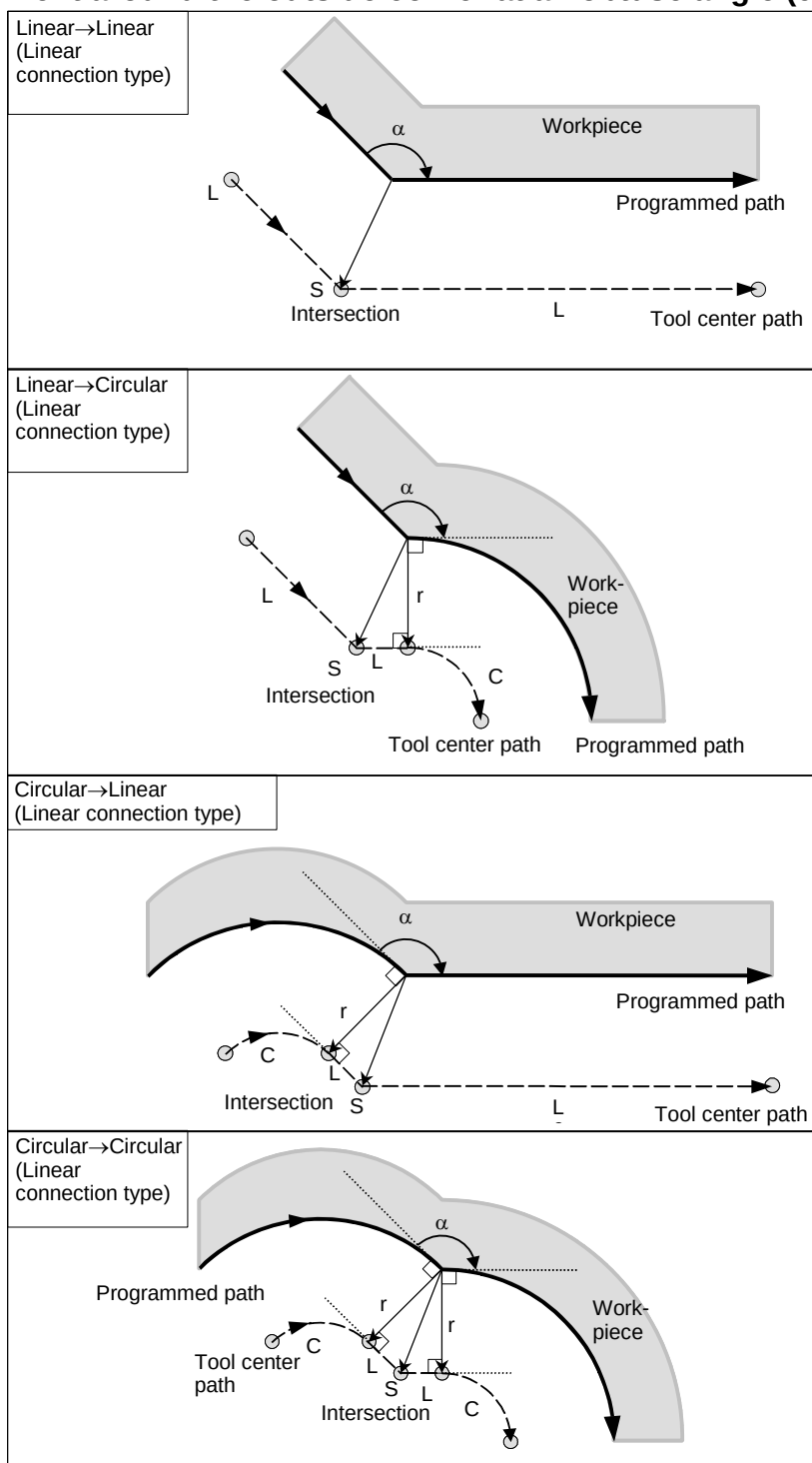


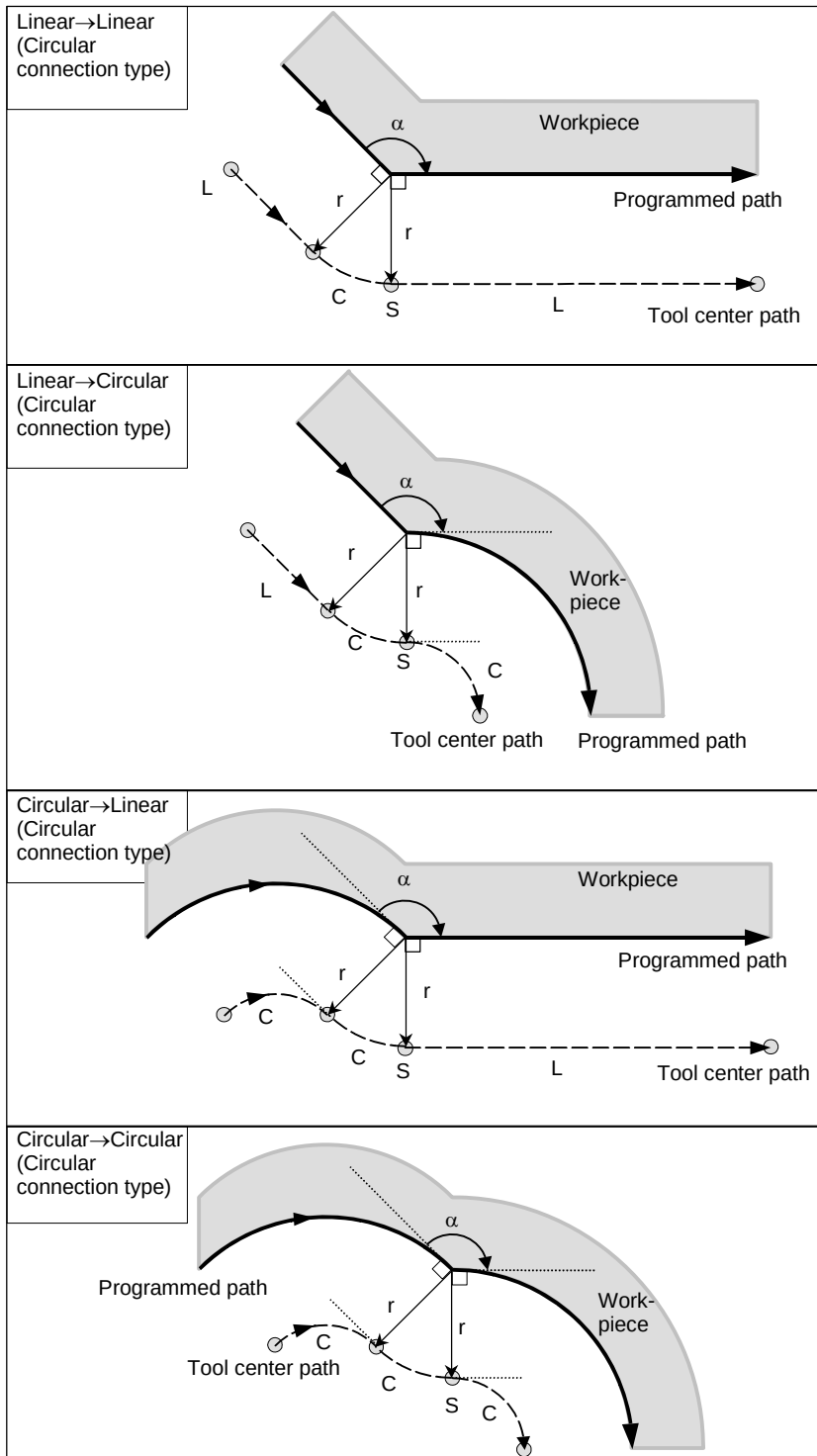
- Tool movement around the inside ($\alpha < 1^\circ$) with an abnormally long vector, linear $\rightarrow$ linear



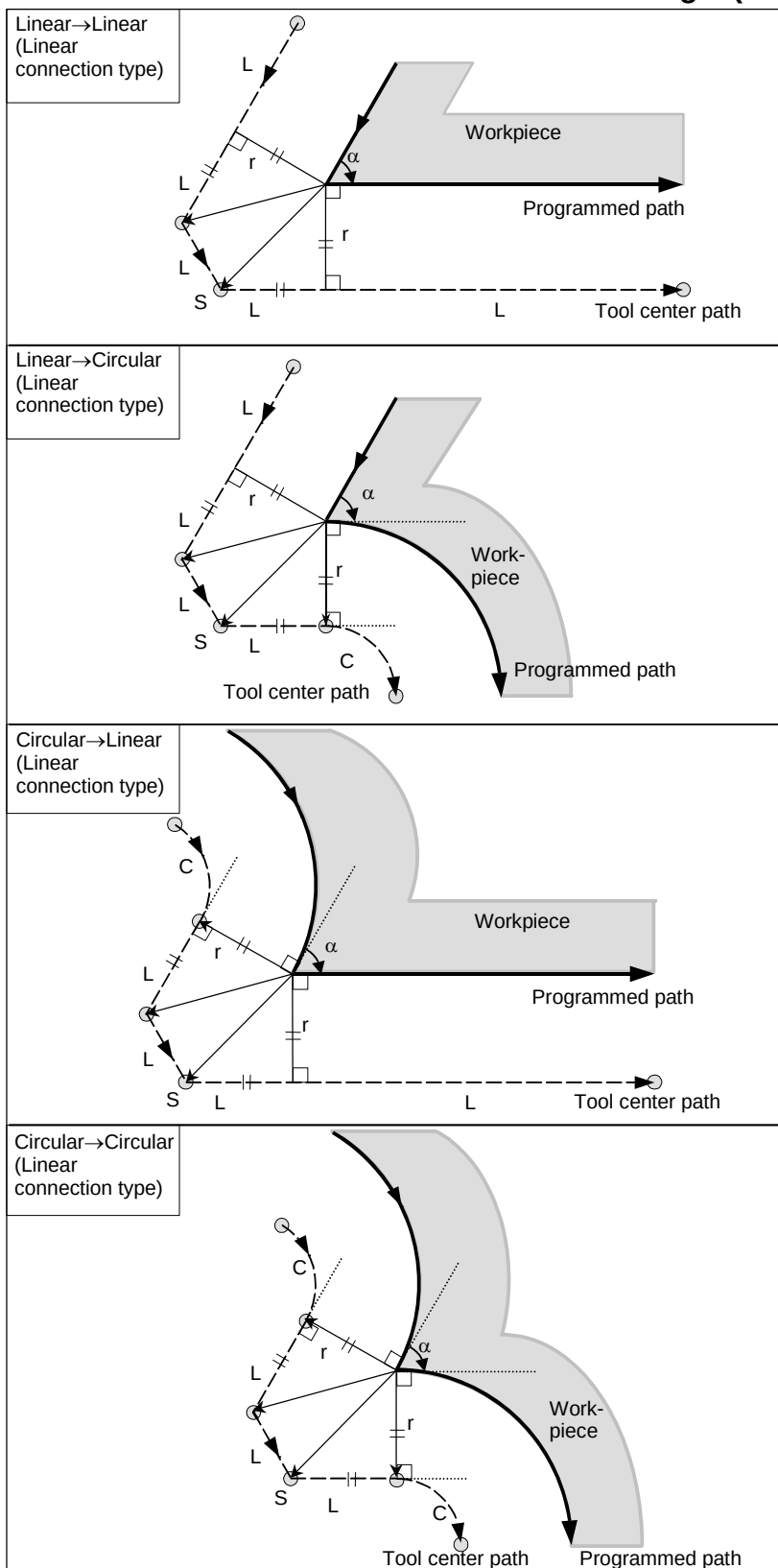
Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

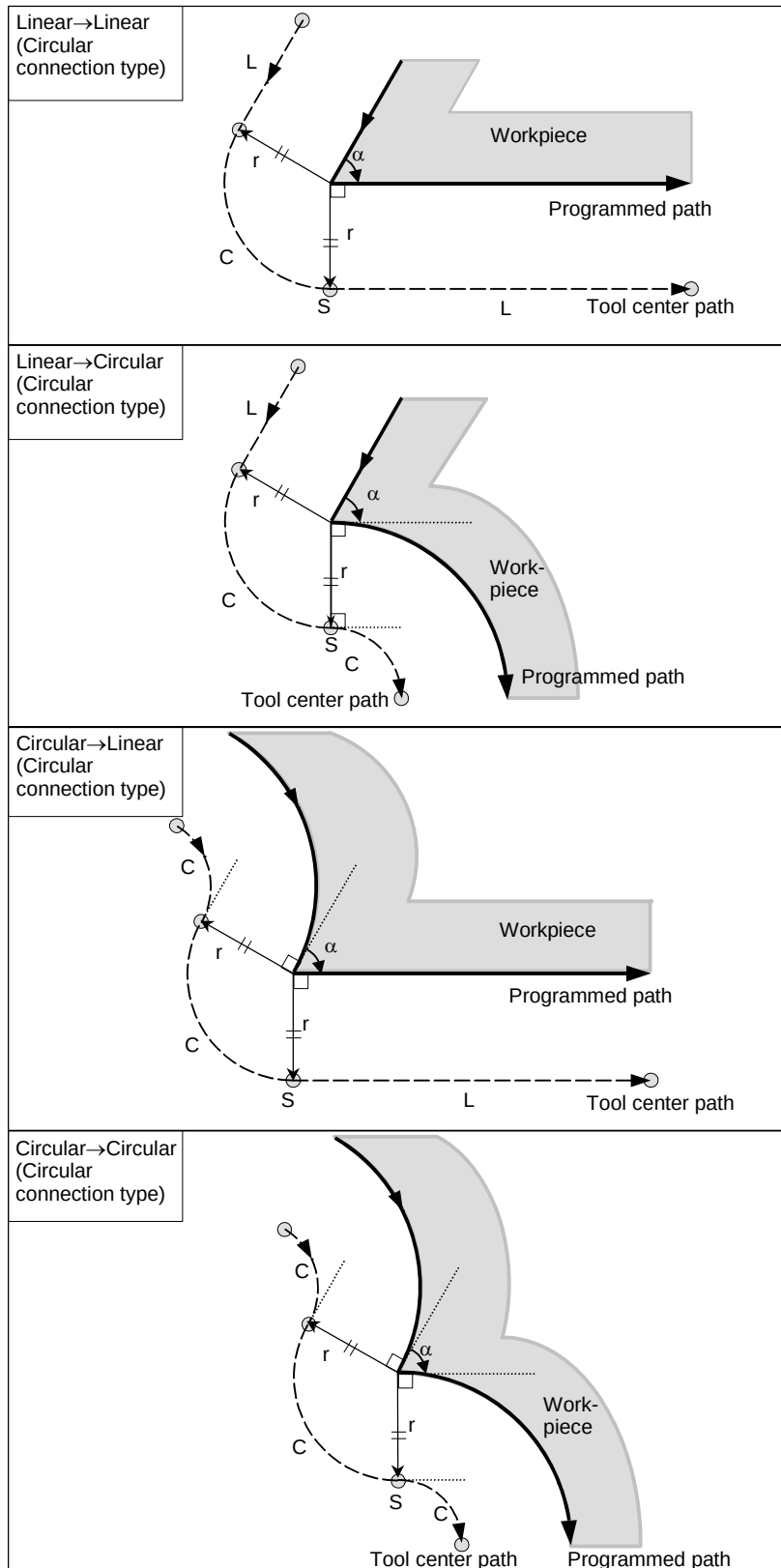
- Tool movement around the outside corner at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)





- Tool movement around the outside corner at an acute angle ($\alpha < 90^\circ$)

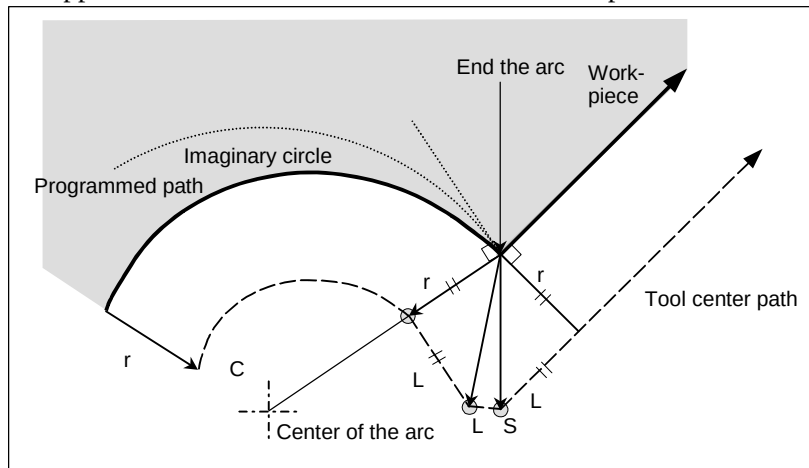




- When it is exceptional

End point for the arc is not on the arc

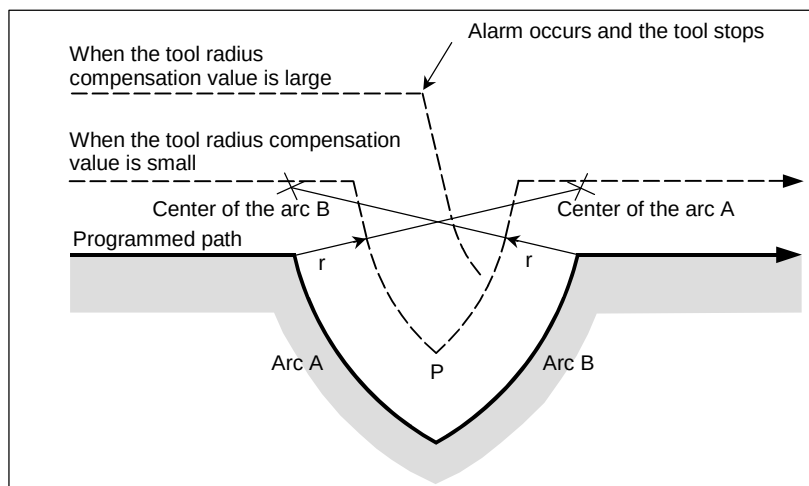
If the end of a line leading to an arc is not on the arc, the system assumes that the tool compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end point. Based on this assumption, the system creates a vector and carries out compensation. The same description applies to tool movement between two circular paths.



There is no inner intersection

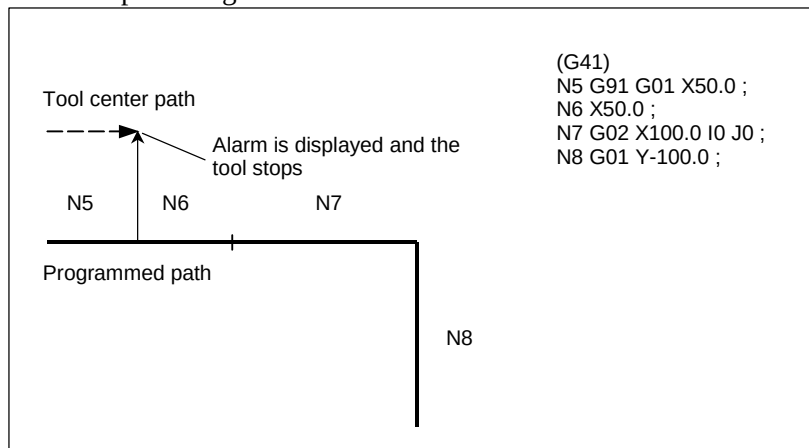
If the tool radius compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool radius compensation. When this is predicted, PS0033 occurs at the end of the previous block and the tool is stopped.

For example, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool radius compensation. If an excessively large value is specified, this intersection does not occur.



- When the center of the arc is identical with the start point or the end point

If the center of the arc is identical with the start point or end point, PS0041 is displayed, and the tool will stop at the start point of the preceding block of the arc.



- Change in the offset direction in the offset mode

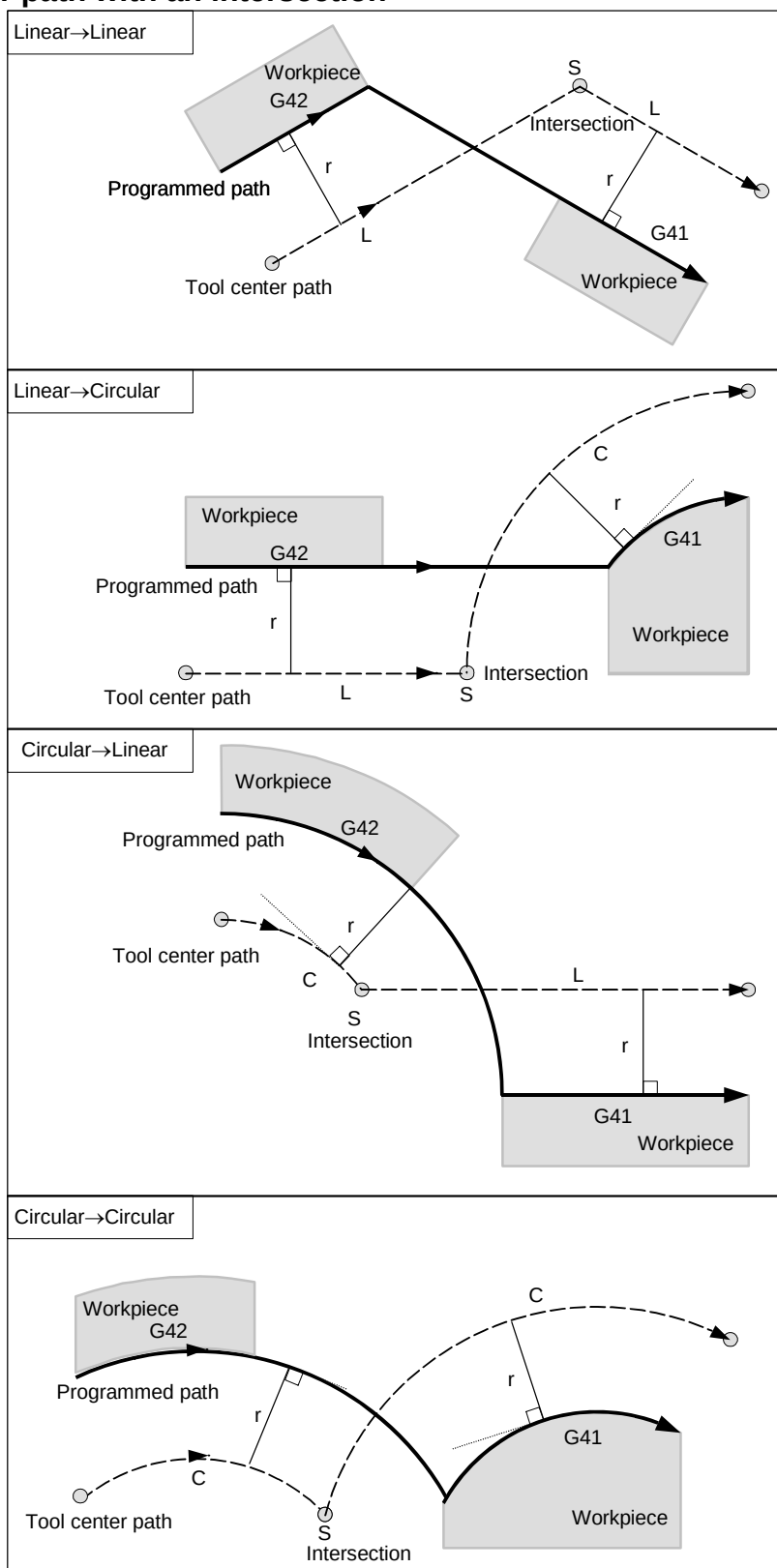
The offset direction is decided by G codes (G41 and G42) for tool radius compensation and the sign of the compensation value as follows.

G code	Sign of compensation	
	+	-
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block.

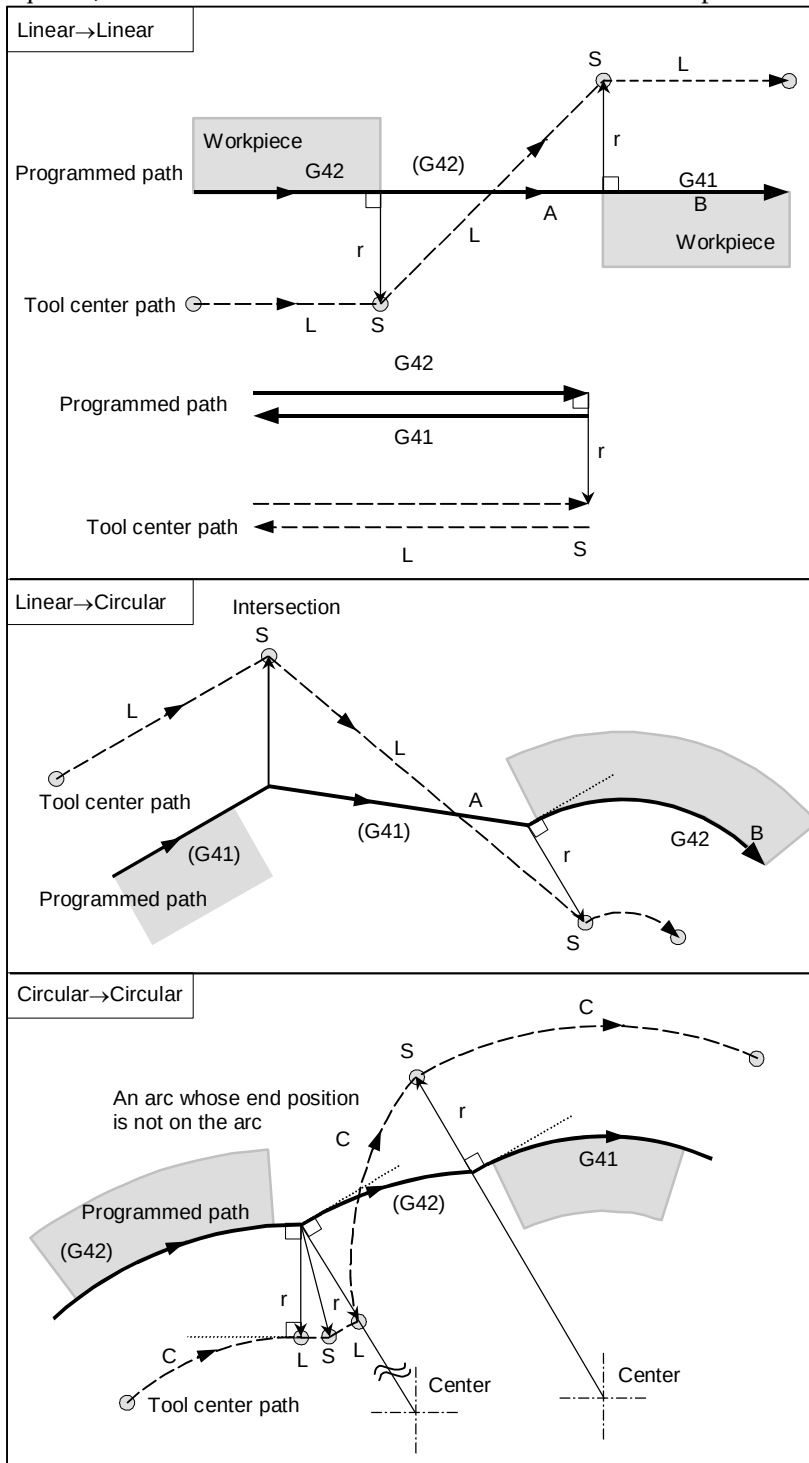
However, the change is not available in the start-up block and the block following it.

- Tool center path with an intersection



- Tool center path without an intersection

When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.

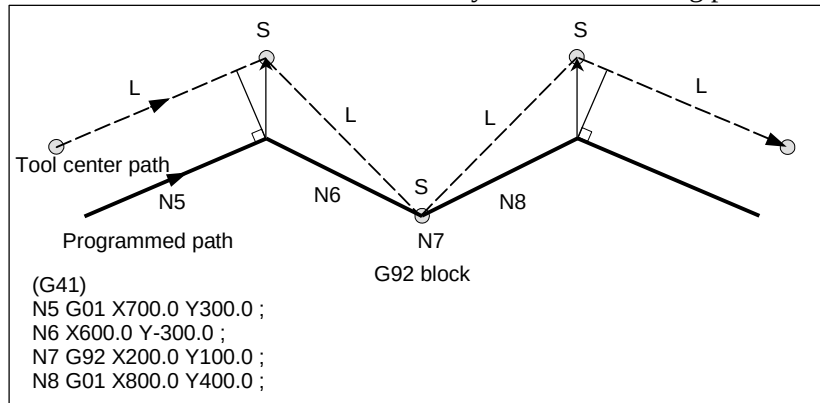


- Command canceling the offset vector temporarily

During offset mode, if G92 (workpiece coordinate system setting) or G52 (local coordinate system setting) is commanded, the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.

In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled.

Also when restored to offset mode, the tool moves directly to the intersecting point.



Before specifying G28 (reference position return), and G53 (machine coordinate system selection) commands, cancel offset mode, using G40. If an attempt is made to specify any of the commands in offset mode, the offset vector temporarily disappears.

- If I, J, and K are specified in a G00/G01 mode block

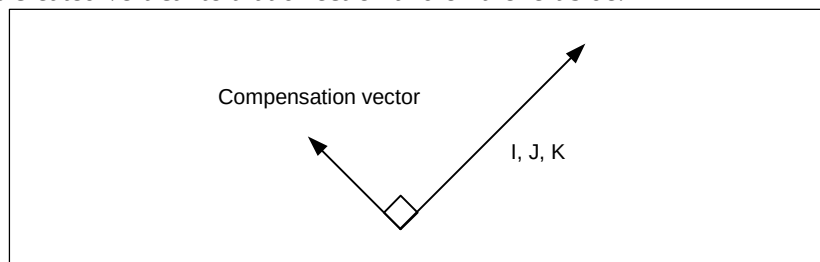
At the start of tool compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This makes it possible to change the compensation direction intentionally.

IJ type vector (XY plane)

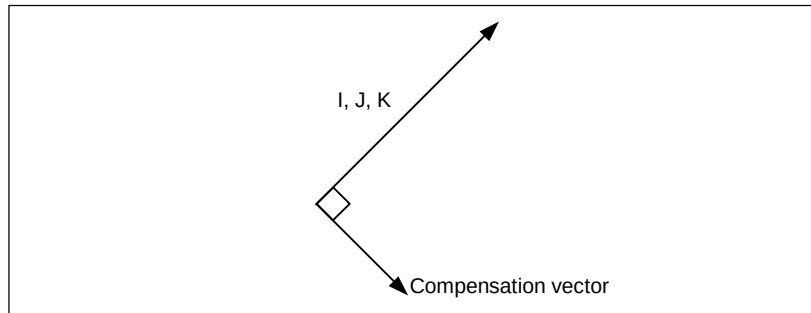
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) It is assumed that the compensation vector (IJ type vector) is the vector with a size equal to the compensation value and vertical to the direction specified by I and J, without performing intersection calculation on the programmed path. I and J can be specified both at the start of tool compensation and in that mode. If they are specified at the start of compensation, any start-up type set in the appropriate parameter will be invalid, and an IJ type vector is assumed.

Offset vector direction

In G41 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the left side.



In G42 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the right side.

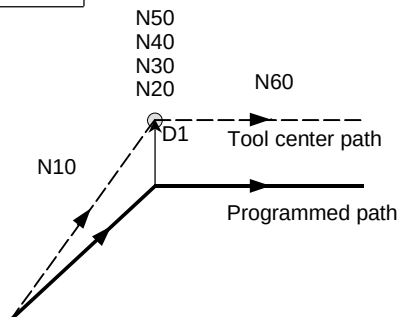


Example

If I and J are specified at the start of compensation (with tool movement)

```
(G40)
N10 G91 G41 X100.0 Y100.0
      I1 D1 ;
N20 G04 X1000 ;
N30 G01 F1000 ;
N40 S300 ;
N50 M50 ;
N60 X150. ;
```

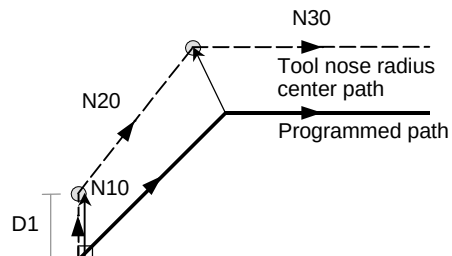
Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.



If I and J are specified at the start of compensation (without tool movement)

```
(G40)
N10 G41 I1 D1 ;
N20 G91 X100. Y100. ;
N30 X150. ;
```

Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.

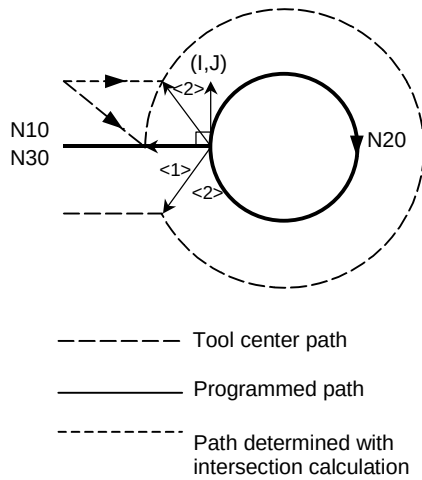


If I and J are specified at the start of compensation (with tool movement)

```
(G17 G41 G91 D1)
N10 G00 X150. J50. ;
N20 G02 I50. ;
N30 G00 X-150. ;
```

Note) In N10, a vector is specified with a size of D1 in the direction vertical to the Y axis, using J50.

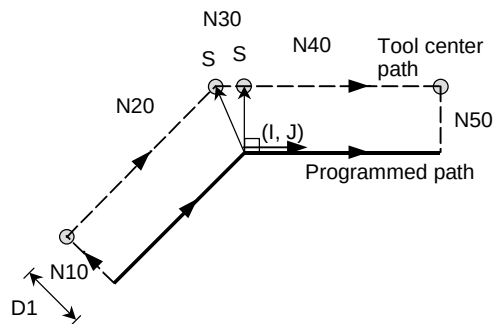
<1> IJ type vector
<2> Vector determined with intersection calculation



If I and J are specified in a block without tool movement in compensation mode

Start-up/cancel type C

```
N10 G41 D1 G01 F1000 ;
N20 G91 X100. Y100. ;
N30 I10. ;
N40 X150. ;
N50 G40 ;
```

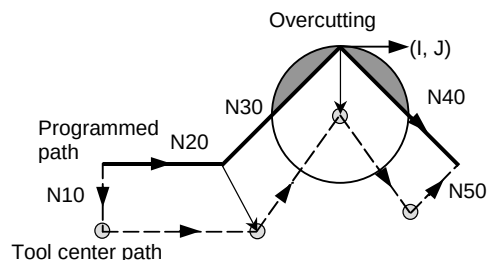


Limitation

If an IJ type vector is specified, tool interference may occur due to that vector alone, depending on the direction. If this occurs, no interference alarm will occur, or no interference avoidance will be performed. Overcutting may, therefore, result.

Start-up/cancel
Type C

```
N10 G42 D1 F1000 ;
N20 G91 X100. ;
N30 X100. Y-100. I10. ;
N40 X100. Y-100. ;
N50 G40 ;
```



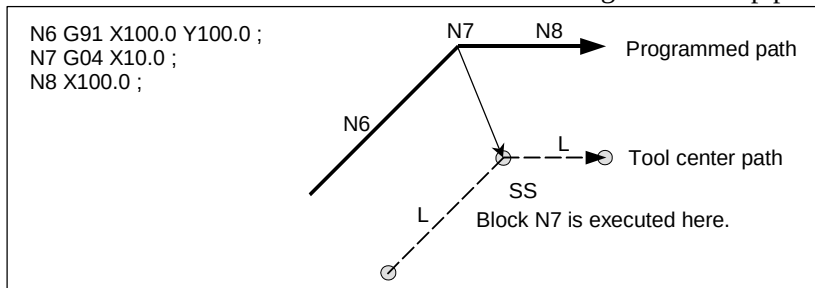
- A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if tool compensation is effected.

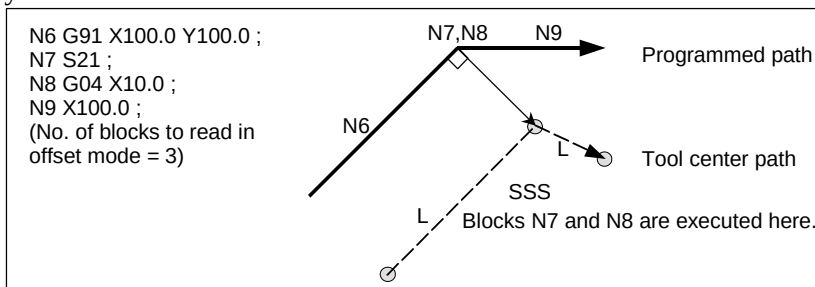
M05 ;	: M code output
S21 ;	: S code output
G04 X10.0 ;	: Dwell
G22 X100000 ;	: Machining area setting
G10 L11 P01 R10.0 ;	: Tool compensation value setting/changing
(G17) Z200.0 ;	: Move command not included in the offset plane.
G90 ;, O10 ;, N20 ;	: G, O, and N codes only
G91 X0 ;	: Move distance is zero.

- A block without tool movement specified in offset mode

Unless the number of blocks without movement consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)) in offset mode, the vector and the tool center path will be as usual. This block is executed at the single block stop point.

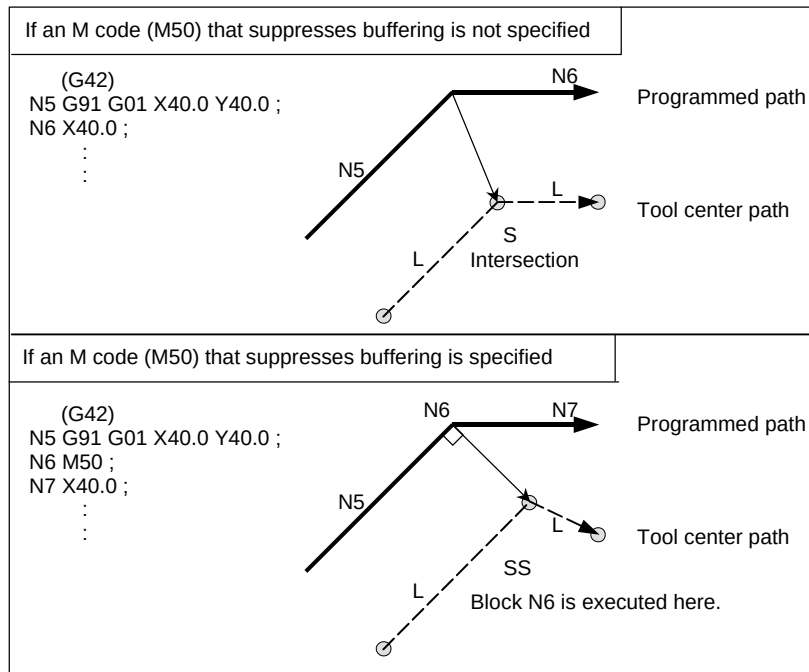


In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter No. 19625)). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.



- If an M/G code that suppresses buffering is specified

If an M/G code that suppresses buffering is specified in offset mode, it is no longer possible to read and analyze subsequent blocks regardless of the number of blocks to read in offset mode, which is determined by parameter No. 19625. Then, intersection calculation and a interference check, described later, are no longer possible. If this occurs, overcutting may occur because a vertical vector is output in the immediately preceding block.



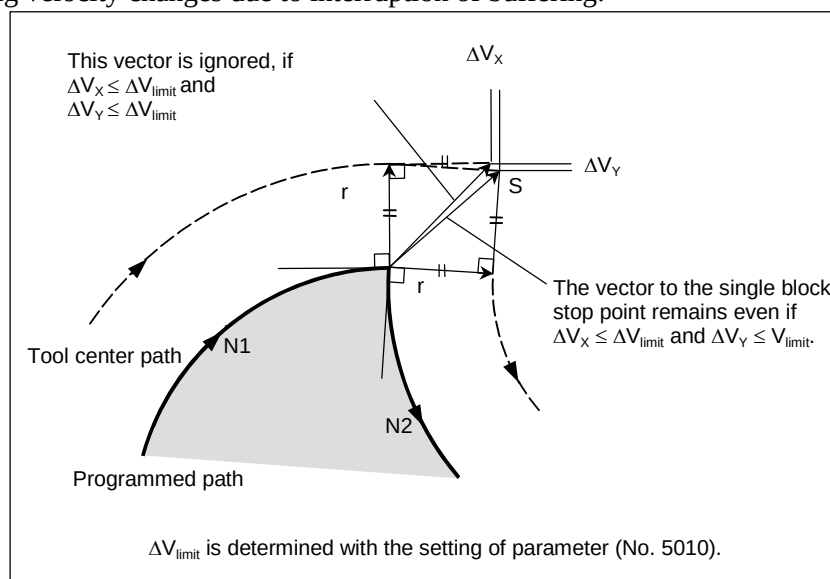
- Workpiece coordinate system or local coordinate system command in the offset mode

If the local coordinate system (G52) or workpiece coordinate system (G92) is specified in the tool compensation (G41 or G42) mode, G52 or G92 is assumed to be a buffering masked G code. The subsequent blocks are not executed until the G52 or G92 block is executed.

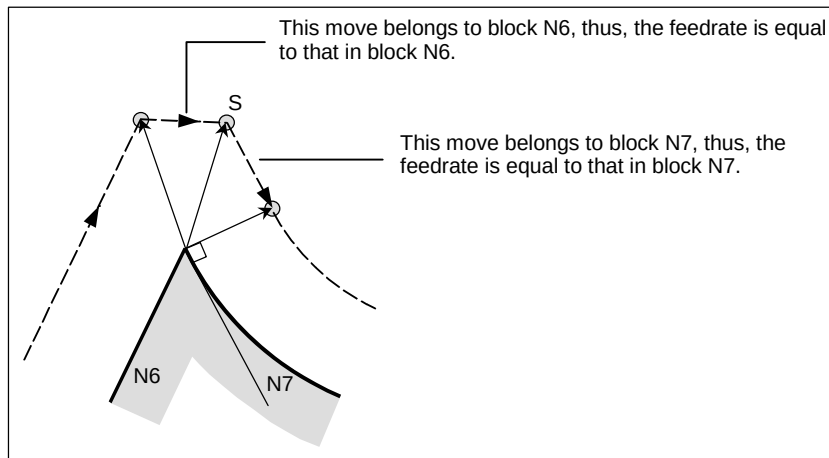
- Corner movement

When two or more offset vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.

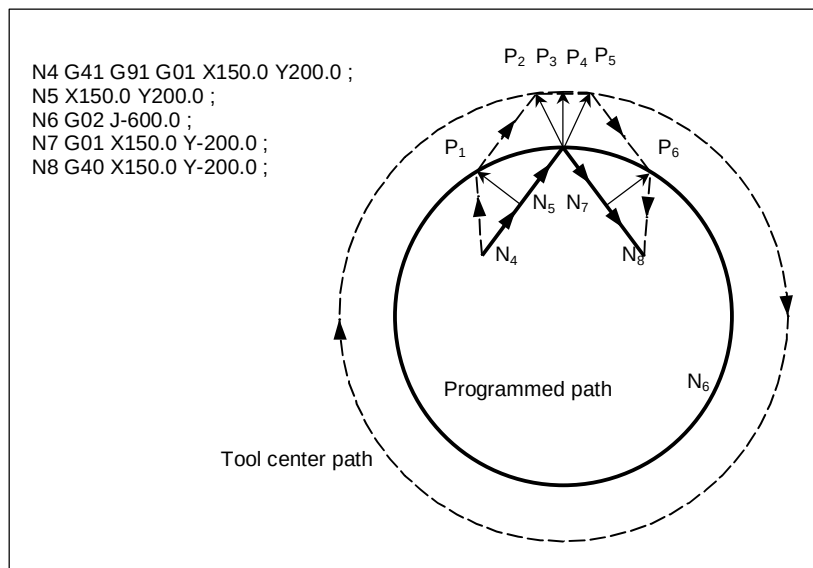
If these vectors almost coincide with each other (the distance of corner movement between the vectors is judged short due to the setting of parameter No. 5010), corner movement is not performed. In this case, the vector to the single block stop point takes precedence and remains, while other vectors are ignored. This makes it possible to ignore the very small movements arising from performing tool compensation, thereby preventing velocity changes due to interruption of buffering.



If the vectors are not judged to almost coincide (therefore, are not erased), movement to turn around the corner is performed. The corner movement that precedes the single block stop point belongs to the previous block, while the corner movement that succeeds the single block stop point belongs to the latter block.



However, if the path of the next block is semicircular or more, the above function is not performed. The reason for this is as follows:



If the vector is not ignored, the tool path is as follows:

$P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow (\text{Circle}) \rightarrow P_4 \rightarrow P_5 \rightarrow P_6$

But if the distance between P_2 and P_3 is negligible, the point P_3 is ignored. Therefore, the tool path is as follows:

$P_2 \rightarrow P_4$

Namely, circle cutting by the block N6 is ignored.

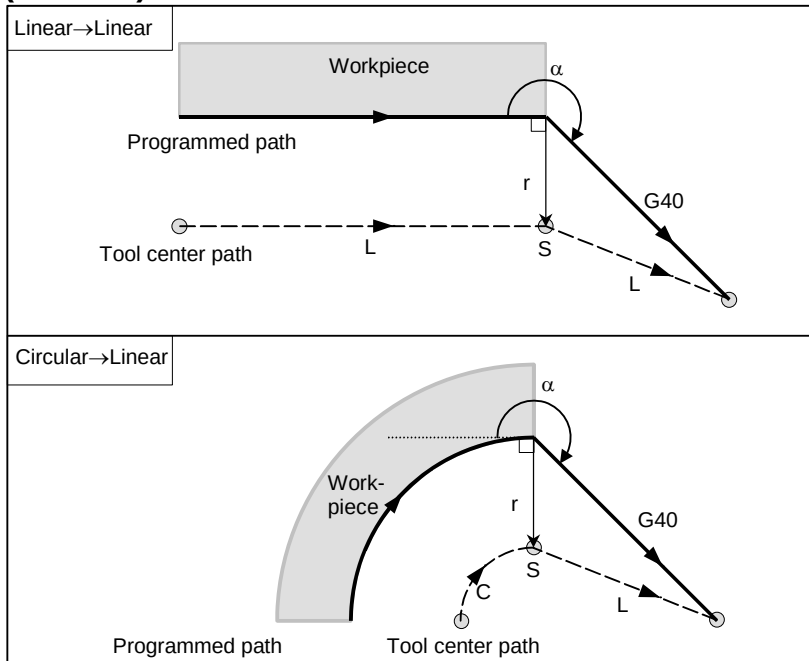
- Interruption of manual operation

For manual operation during the offset mode, see Section III-3.5 "Manual Absolute ON."

16.4.4 Tool Movement in Offset Mode Cancel

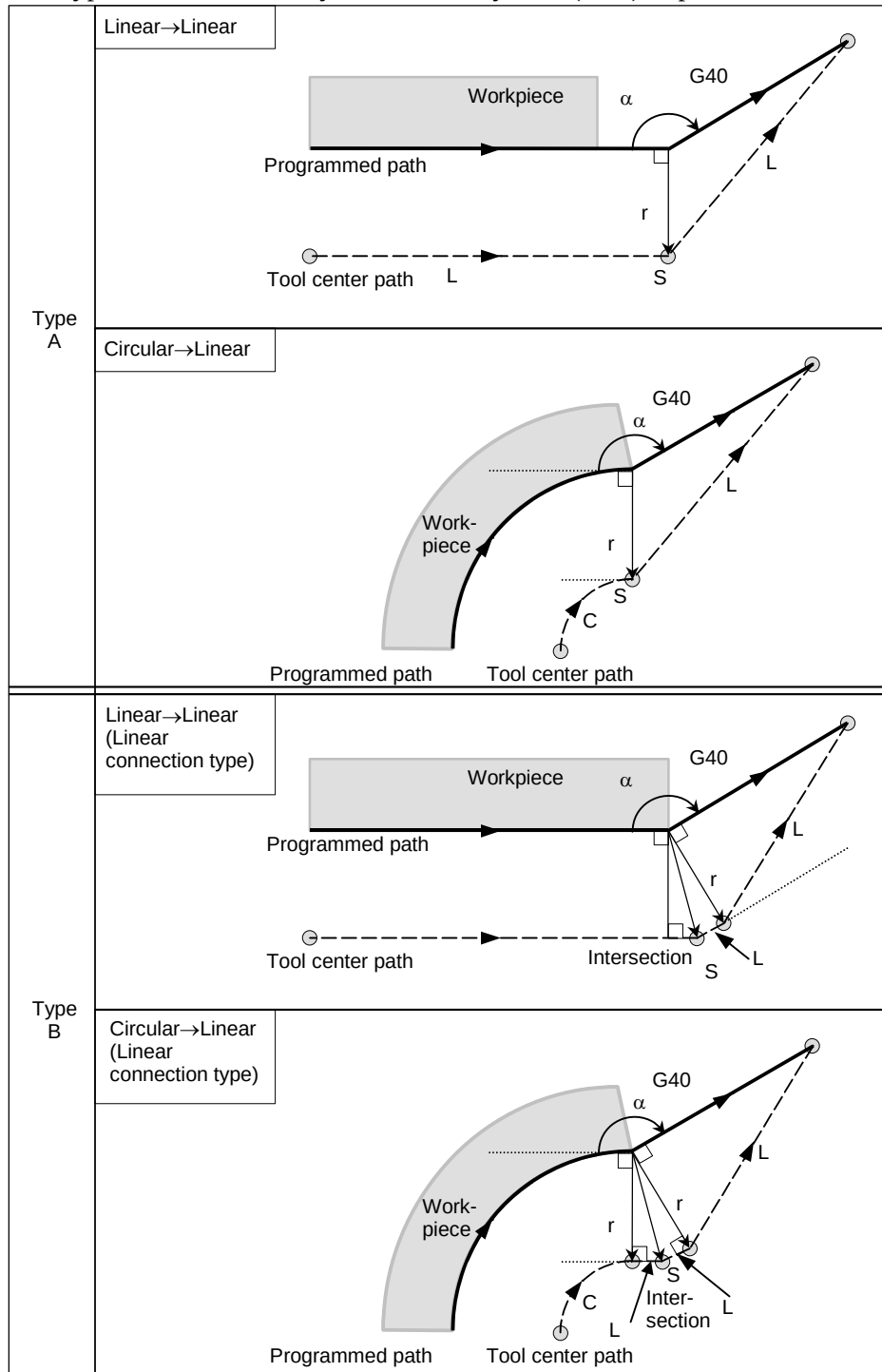
Explanation

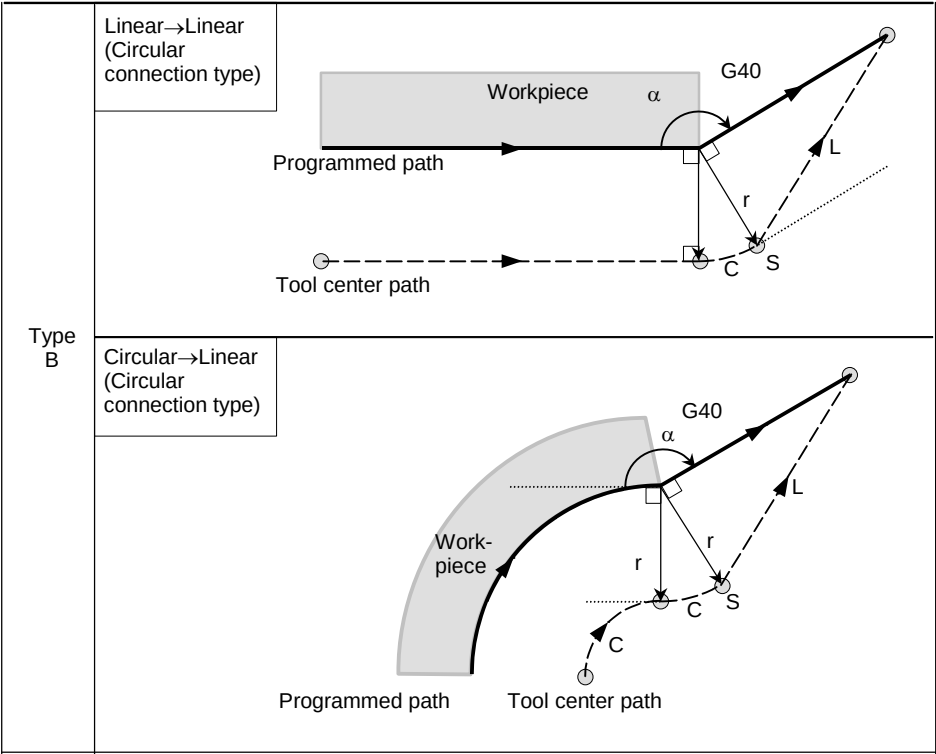
- If the cancel block is a block with tool movement, and the tool moves around the inside ($180^\circ \leq \alpha$)



- If the cancel block is a block with tool movement, and the tool moves around the outside at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)

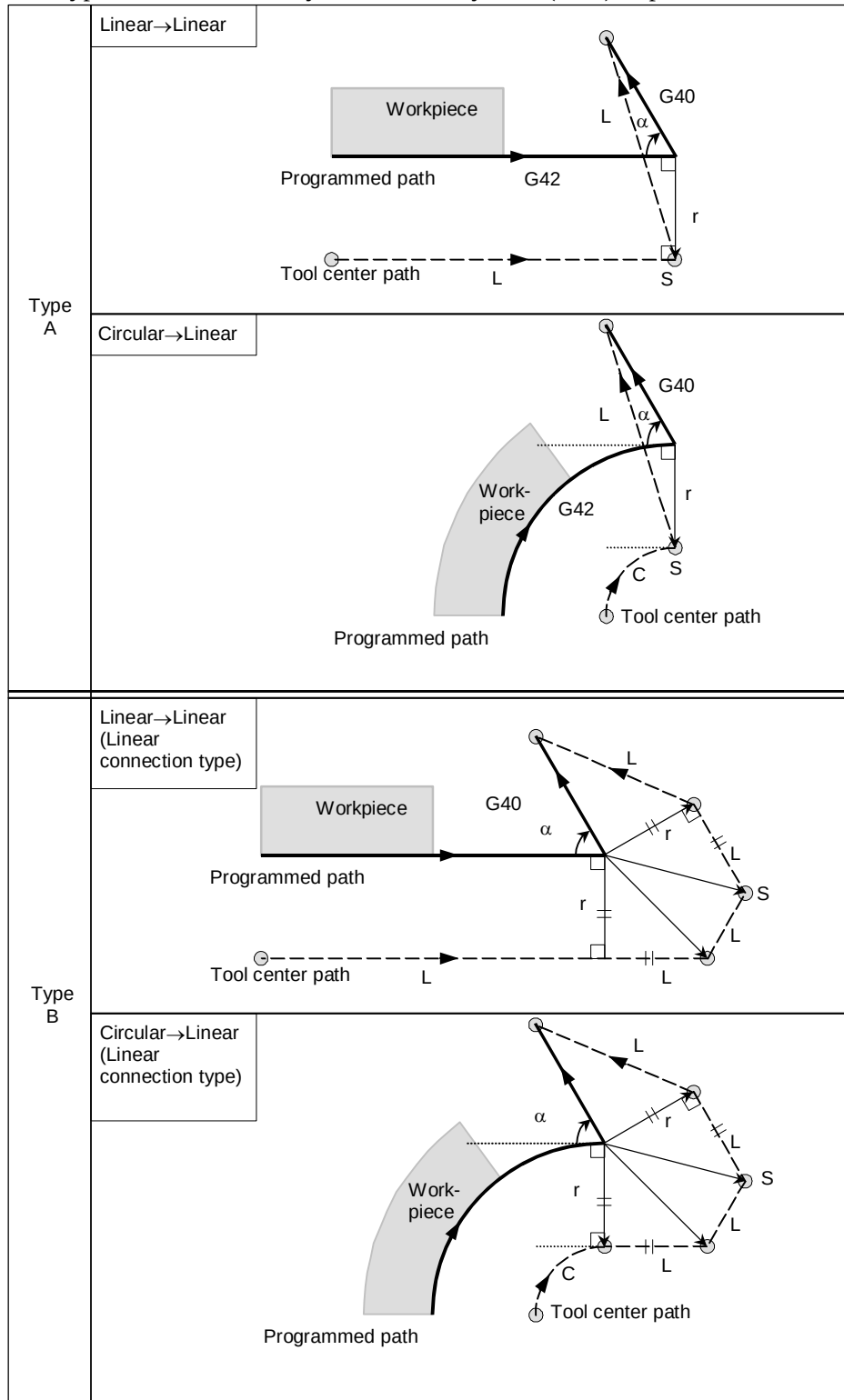
Tool path has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

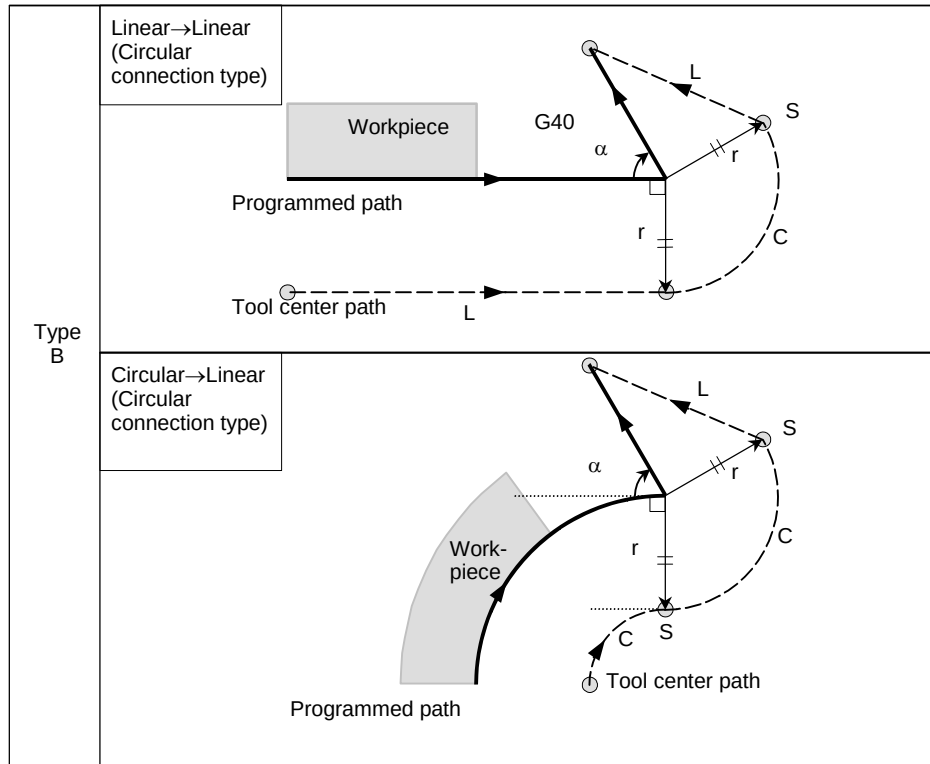




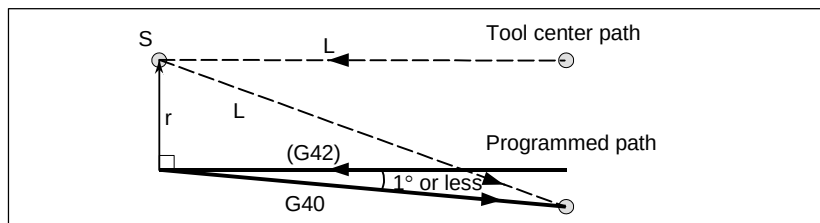
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle ($\alpha < 90^\circ$)

Tool path has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.





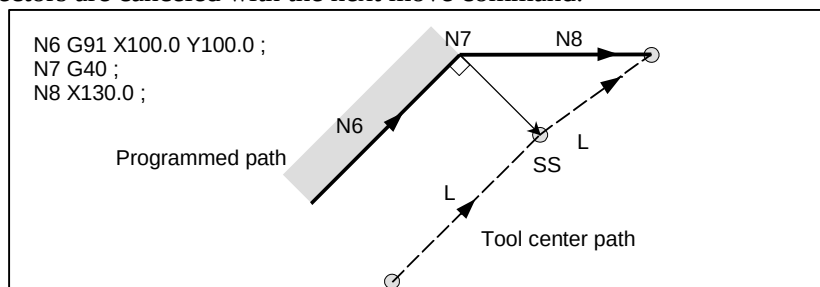
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle of 1 degree or less in a linear → linear manner ($\alpha \leq 1^\circ$)



- A block without tool movement specified together with offset cancel

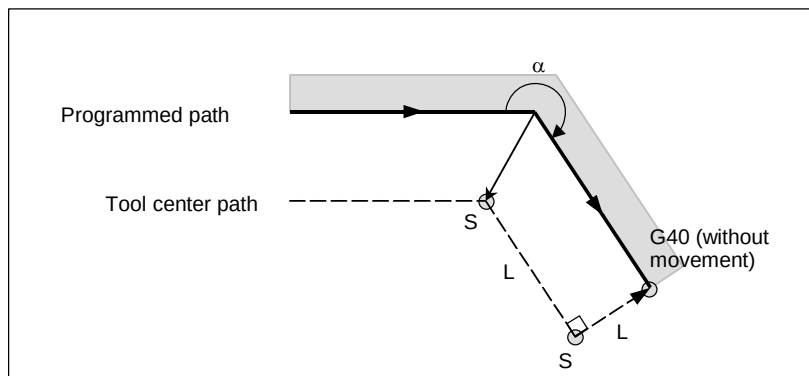
For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the tool radius compensation value in the vertical direction. The tool does not operate in the cancel block. The remaining vectors are canceled with the next move command.



For type C

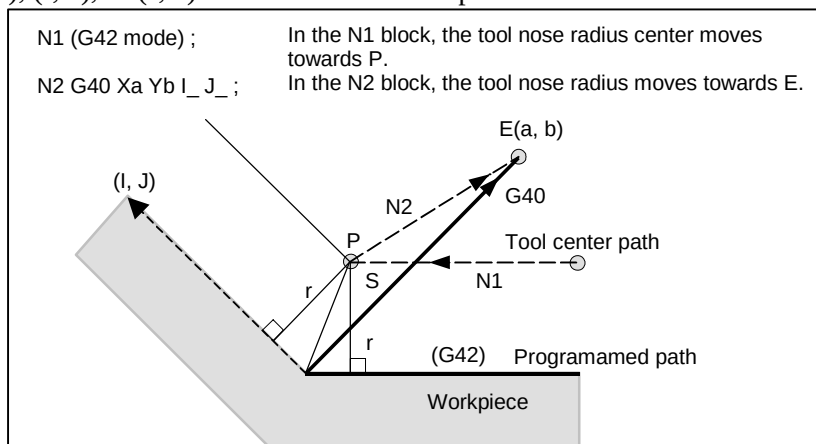
The tool shifts by the compensation value in the direction vertical to the block preceding the cancel block.



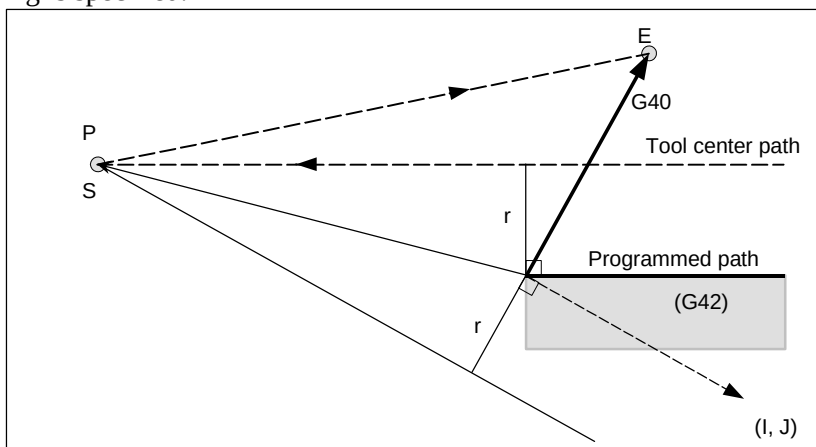
- Block containing G40 and I_J_K_

The previous block contains G41 or G42

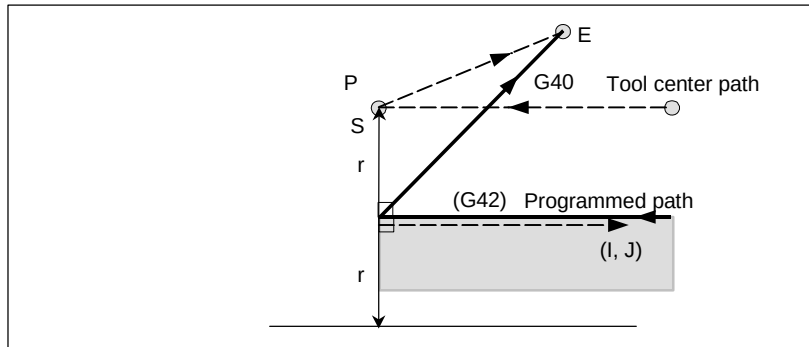
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end point determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified.



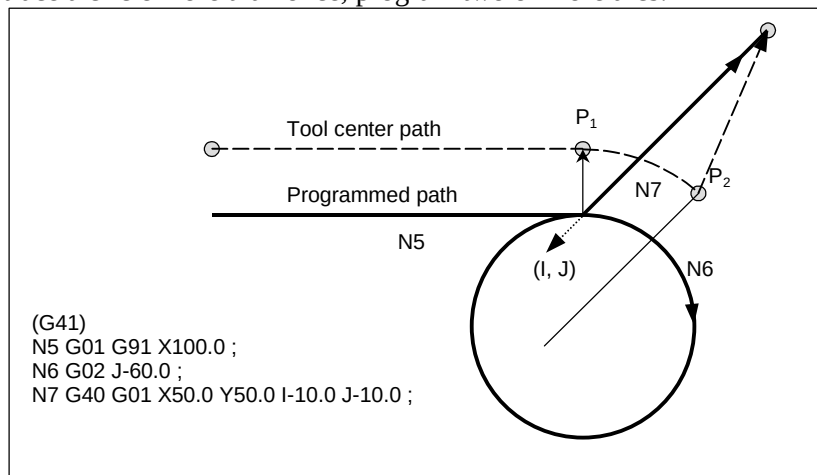
When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



- Length of the tool center path larger than the circumference of a circle

In the Example shown below, the tool does not trace the circle more than once. It moves along the arc from P_1 to P_2 . The interference check function described below may raise an alarm.

To make the tool trace a circle more than once, program two or more arcs.



16.4.5 Prevention of Overcutting Due to Tool Radius Compensation

Explanation

- Machining a groove smaller than the diameter of the tool

Since the tool compensation forces the tool center path to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.

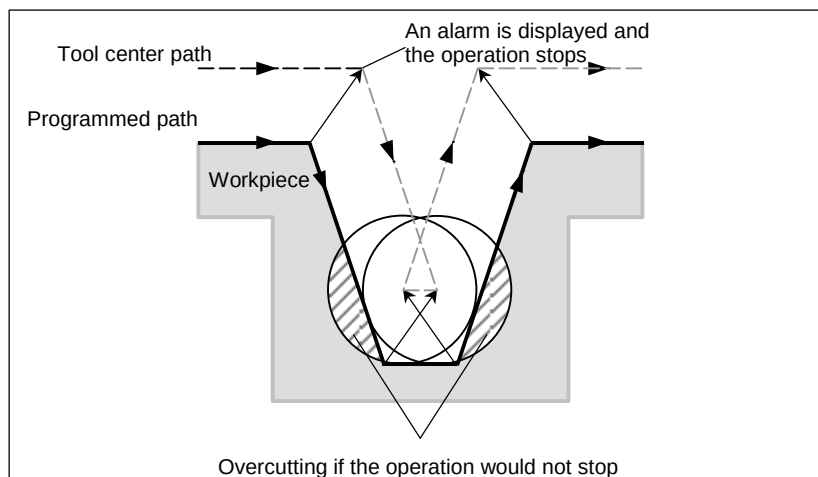


Fig. 16.4.5 (a) Machining a groove smaller than the diameter of the tool

- Machining a step smaller than the tool radius

For a figure in which a workpiece step is specified with an arc, the tool center path will be as shown in Fig. 16.4.5 (b). If the step is smaller than the tool radius, the tool center path usually compensated as shown in Fig. 16.4.5 (c) may be in the direction opposite to the programmed path. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued. If the step is of linear, no alarm will be generated and cut correctly. However uncut part will remain.

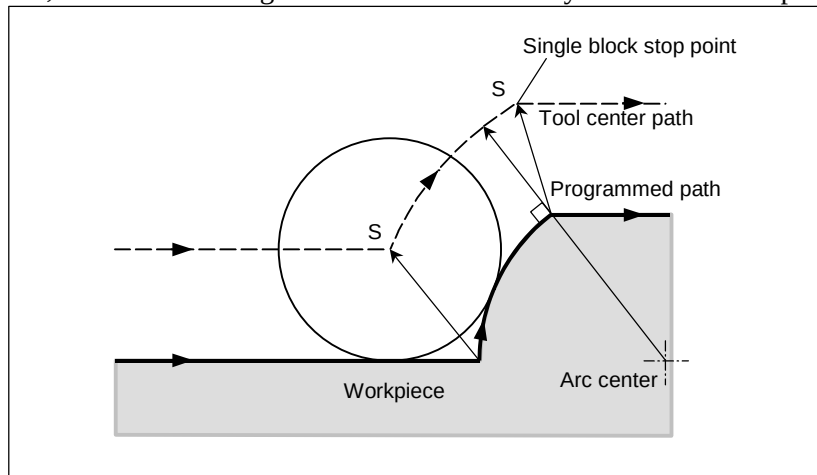


Fig. 16.4.5 (b) Machining a step larger than the tool radius

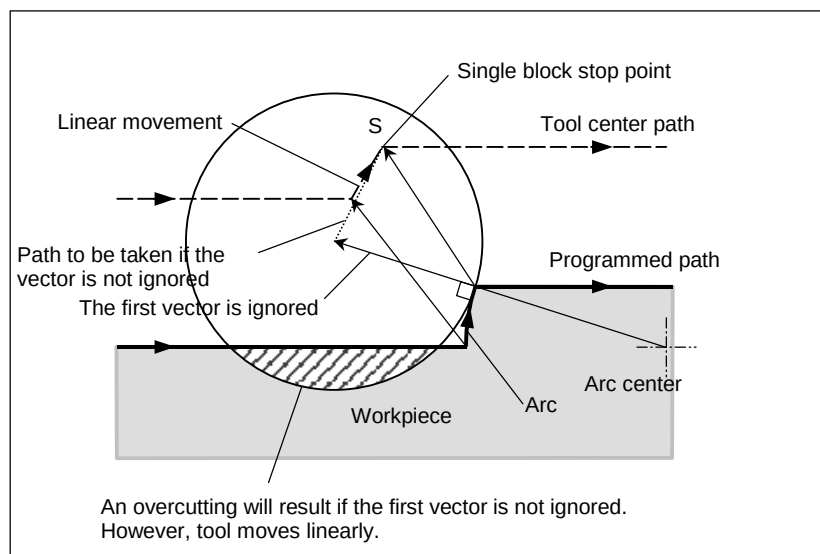


Fig. 16.4.5 (c) Machining a step smaller than the tool radius

- Starting compensation and cutting along the Z-axis

It is usually used such a method that the tool is moved along the Z axis after the tool compensation (normally XY plane) is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.

Let us consider the program in the Fig. 16.4.5 (d), assuming the number of blocks to read in tool compensation mode (parameter No. 19625) to be 3.

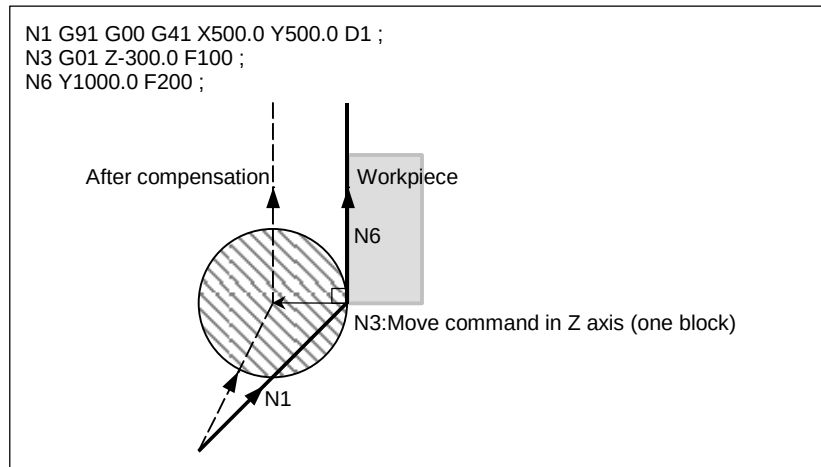


Fig. 16.4.5 (d)

In the program example in the Fig. 16.4.5 (d), when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the Fig. 16.4.5 (d).

Then, suppose that the block N3 (move command in Z axis) is divided into N3 and N5 in the Fig. 16.4.5 (e).

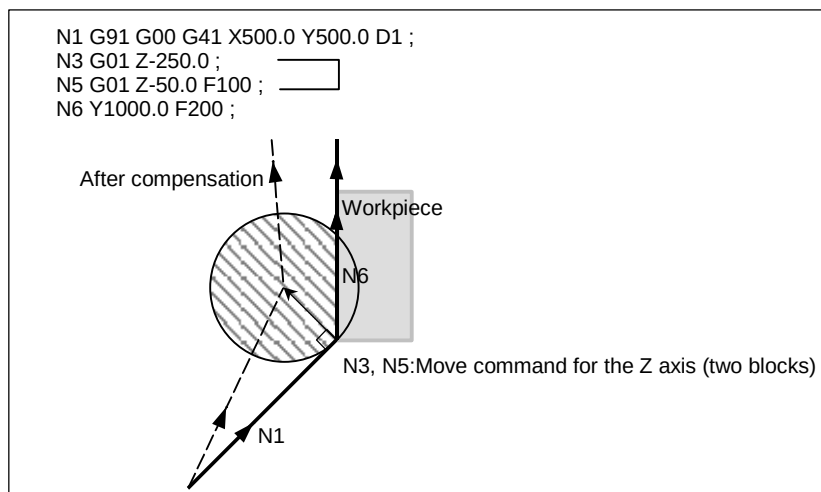


Fig. 16.4.5 (e)

At this time, because the number of blocks to read is 3, blocks up to N5 can be read at the start of N1 compensation, but block N6 cannot be read. As a result, compensation is performed only on the basis of the information in block N1, and a vertical vector is created at the end of the compensation start block. Usually, therefore, overcutting will result as shown in the Fig. 16.4.5 (e).

In such a case, it is possible to prevent overcutting by specifying a command with the exactly the same direction as the advance direction immediately before movement along the Z axis beforehand, after the tool is moved along the Z axis using the above rule.

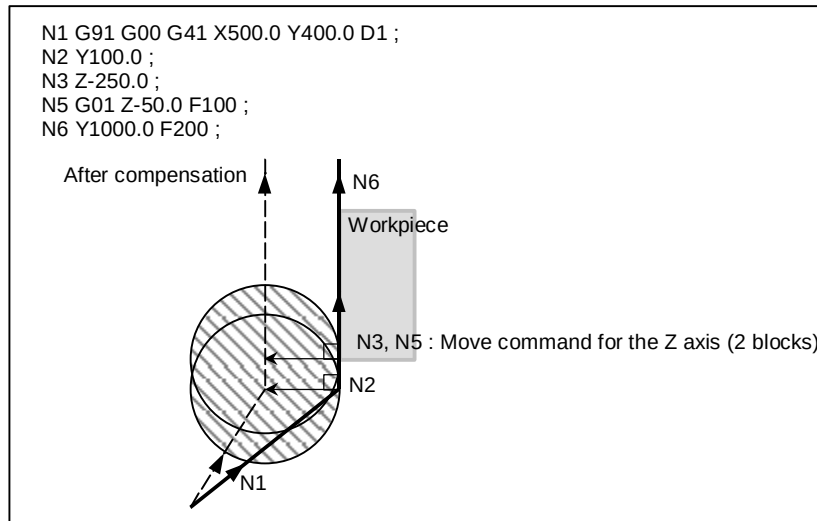


Fig. 16.4.5 (f)

As the block with sequence N2 has the move command in the same direction as that of the block with sequence N6, the correct compensation is performed.

Alternatively, it is possible to prevent overcutting in the same way by specifying an IJ type vector with the same direction as the advance direction in the start-up block, as in N1 G91 G00 G41 X500. Y500. I0 J1 D1;; after the tool has moved along the Z axis.

16.4.6 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

Explanation

- Condition under which an interference check is possible

To perform an interference check, it is necessary to read at least three blocks with tool movement. If, therefore, three or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as independent auxiliary function and dwell, are specified in succession, excessive or insufficient cutting may occur because an interference check fails. Assuming the number of blocks to read in offset mode, which is determined by parameter No. 19625, to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which an interference check is possible is

$$(N - 3) \geq M.$$

For example, if the maximum number of blocks to read in offset mode is 8, an interference check is possible even if up to five blocks without tool movement are specified. In this case, three adjacent blocks can be checked for interference, but any subsequent interference that may occur cannot be detected.

- Interference check method

Two interference check methods are available, direction check and circular angle check. Bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008 are used to specify whether to enable these methods.

Parameter CNV	Parameter CNC	Operation
0	0	An interference check is enabled, and a direction check and a circular angle check can be performed.
0	1	An interference check is enabled, and only a circular angle check is performed.
1	—	An interference check is disabled.

NOTE

There are no settings for performing a direction check only.

- Interference reference <1> (direction check)

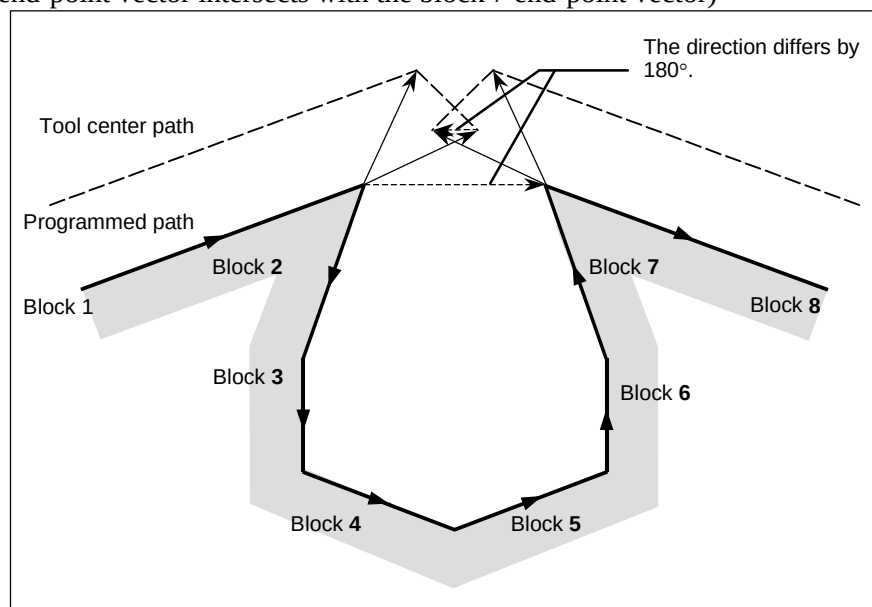
Assuming the number of blocks to read during tool compensation to be N, a check is first performed on the compensation vector group calculated in (block 1 - block 2) to be output this time and the compensation vector group calculated in (block N-1 - block N); if they intersect, they are judged to interfere. If no interference is found, a check is performed sequentially in the direction toward the compensation vector group to be output this time, as follows:

(Block 1 - block 2) and (block N-2 - block N-1)
 (Block 1 - block 2) and (block N-3 - block N-2)
 ⋮
 ⋮
 (Block 1 - block 2) and (block 2 - block 3)

Even if multiple number of compensation vector groups are generated, a check is performed on all pairs. The judgment method is as follows: For a check on the compensation vector group in (block 1 - block 2) and those in (block N-1 - block N), the direction vector from the specified (end point of block 1) to the (end point of block N-1) is compared with the direction vector from the (point resulting from adding the compensation vector to be checked to the end of block 1) to the (point resulting from adding the compensation vector to be checked to the end of block N-1), and if the direction is 90° or greater or 270° or less, they are judged to intersect and interfere. This is called a direction check.

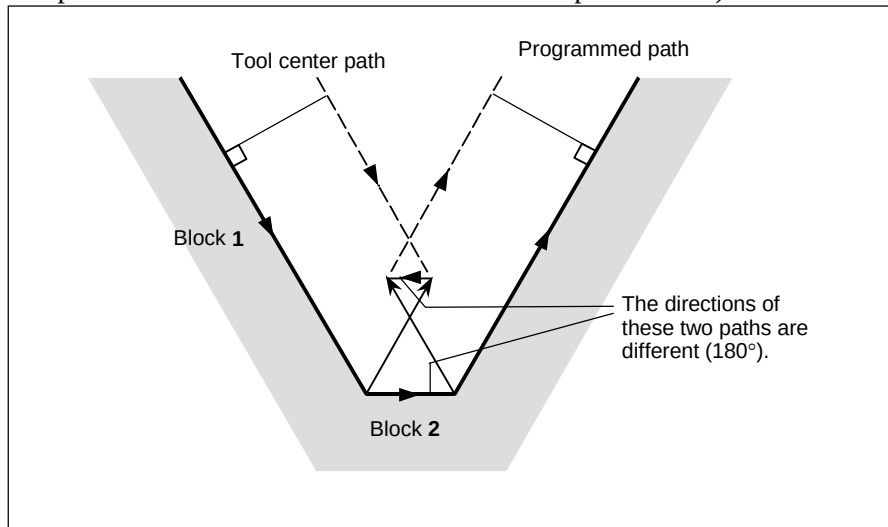
Example of interference standard <1>

(If the block 1 end-point vector intersects with the block 7 end-point vector)



Example of interference standard <1>

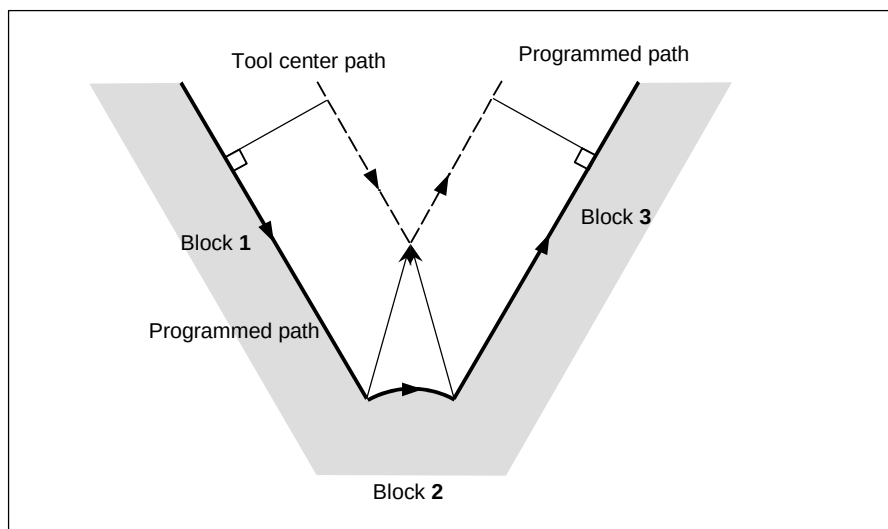
(If the block 1 end-point vector intersects with the block 2 end-point vector)



- Interference reference <2> (circular angle check)

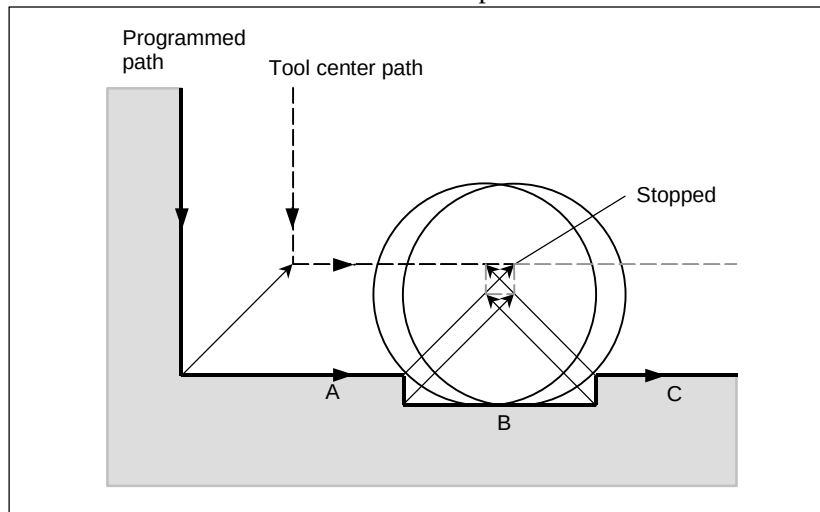
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed path and the circular angle of the start and end point of the post-compensation path, in addition to direction check <1>. If the difference is 180° or greater, the blocks are judged to interfere. This is called a circular angle check.

Example of <2> (if block 2 is circular and the start point of the post-compensation arc coincide with the end point)



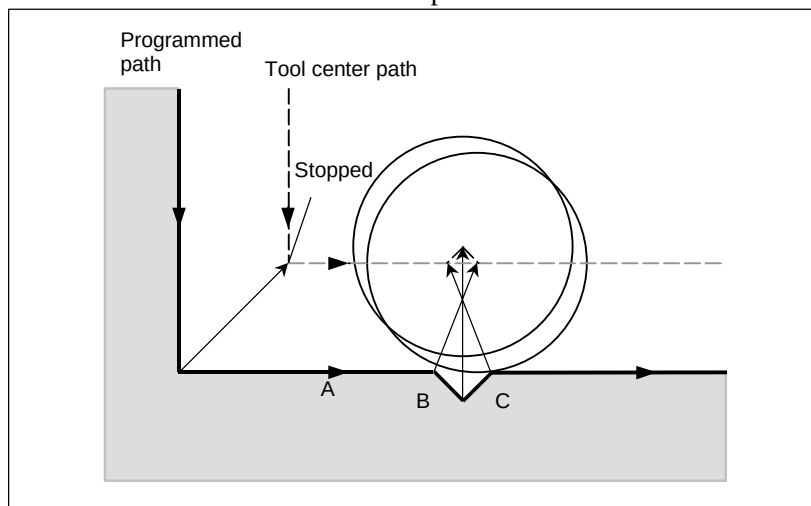
- **When interference is assumed although actual interference does not occur**

<1> Depression which is smaller than the tool radius compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after the tool compensation, the tool stops and an alarm is displayed.

<2> Groove which is smaller than the tool radius compensation value



Like <1>, an alarm is displayed because of the interference as the direction is reverse in block B.

16.4.6.1 Operation to be performed if an interference is judged to occur

The operation to be performed if an interference check judges that an interference (due to overcutting) occurs can be either of the following two, depending on the setting of bit 5 (CAV) of parameter No. 19607.

Parameter CAV	Function	Operation
0	Interference check alarm function	An alarm stop occurs before the execution of the block in which overcutting (interference) occurs.
1	Interference check avoidance function	The tool path is changed so that overcutting (interference) does not occur, and processing continues.

16.4.6.2 Interference check alarm function

- Interference other than those between adjacent three blocks

If the end-point vector of block 1 and the end-point vector of block 7 are judged to interfere as shown in the Fig. 16.4.6.2 (a), an alarm will occur before the execution of block 1 so that the tool stops. In this case, the vectors will not be erased.

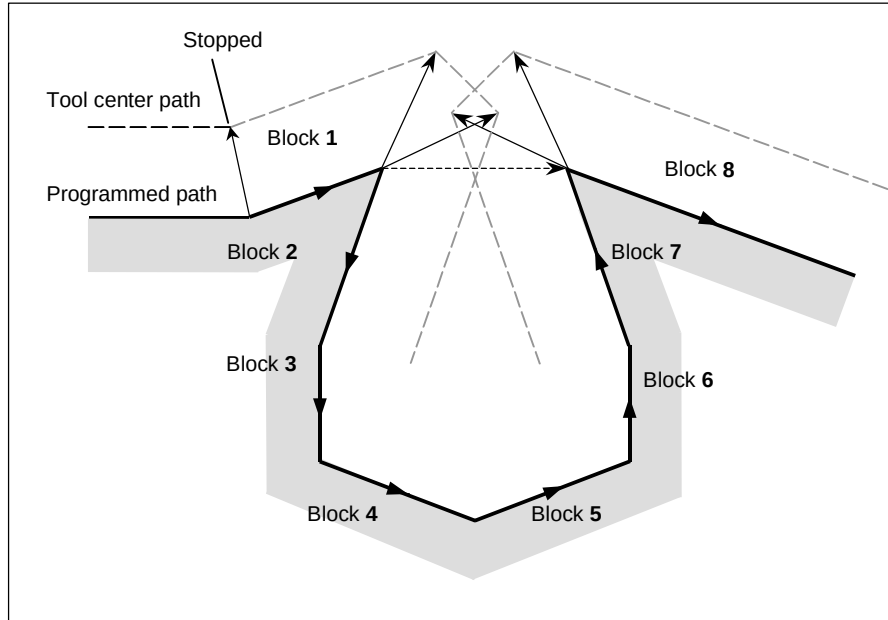


Fig. 16.4.6.2 (a)

- Interference between adjacent three blocks

If an interference is judged to occur between adjacent three blocks, the interfering vector, as well as any vectors existing inside of it, is erased, and a path is created to connect the remaining vectors. In the Example shown in the Fig. 16.4.6.2 (b), V_2 and V_5 interfere, so that V_2 and V_5 are erased, so are V_3 and V_4 , which are inside of them, and V_1 is connected to V_6 . The operation during this time is linear interpolation.

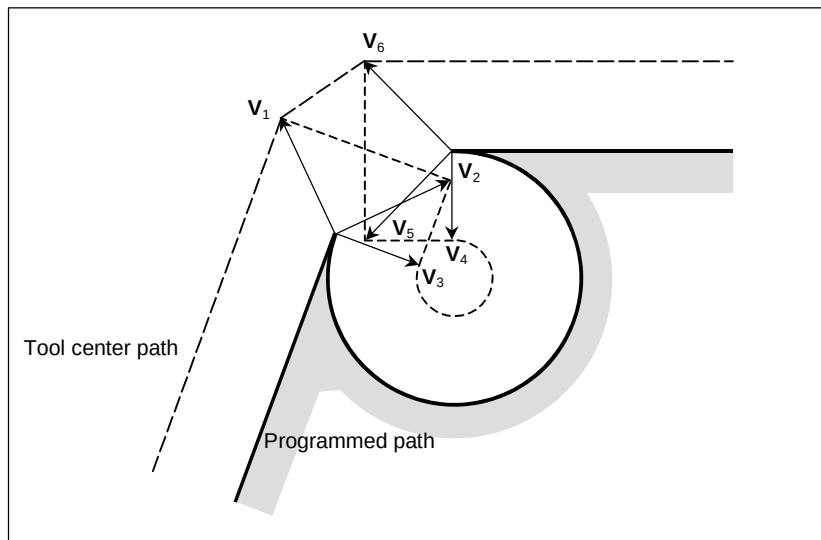


Fig. 16.4.6.2 (b)

If, after vector erasure, the last single vector still interferes, or if there is only one vector at the beginning and it interferes, an alarm will occur immediately after the start of the previous block (end point for a single block) and the tool stops. In the Example shown in the Fig. 16.4.6.2 (c), V_2 and V_3 interfere, but, even after erasure, an alarm will occur because the final vectors V_1 and V_4 interfere.

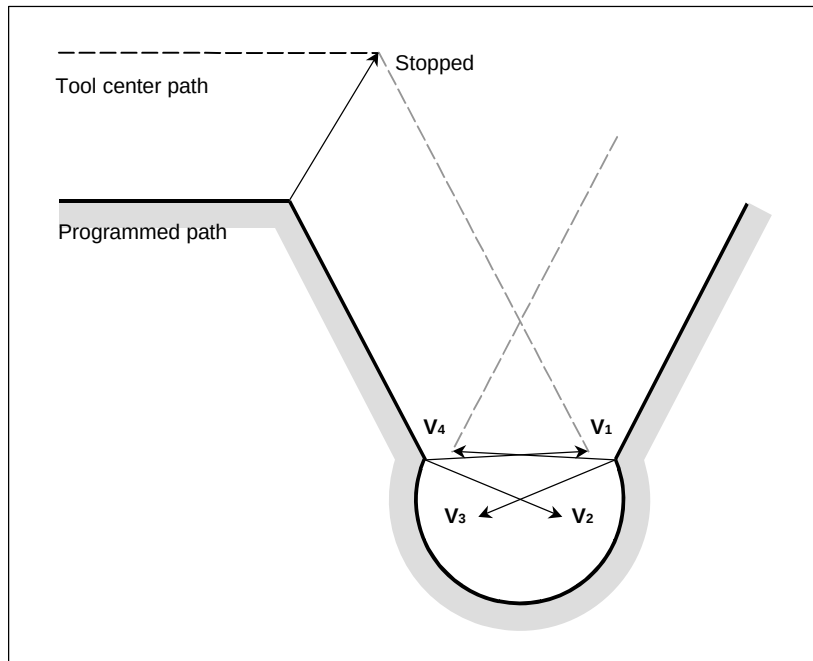


Fig. 16.4.6.2 (c)

16.4.6.3 Interference check avoidance function

Overview

If a command is specified which satisfies the condition under which the interference check alarm function generates an interference alarm, this function suppresses the generation of the interference alarm, but causes a new compensation vector to be calculated as a path for avoiding interference, thereby continuing machining. For the path for avoiding interference, insufficient cutting occurs in comparison with the programmed path. In addition, depending on the specified figure, no path for avoiding interference can be determined or the path for avoiding interference may be judged dangerous. In such a case, an alarm stop will occur. For this reason, it is not always possible to avoid interference for all commands.

- Interference avoidance method

Let us consider a case in which an interference occurs between the compensation vector between (block 1 - block 2) and the compensation vector between (block N-1 - block N). The direction vector from the end point of block 1 to the end point of block N-1 is called a gap vector. At this time, a post-compensation intersection vector between (block 1 - gap vector) and a post-compensation intersection vector between (gap vector - block N) is determined, and a path connecting them is created.

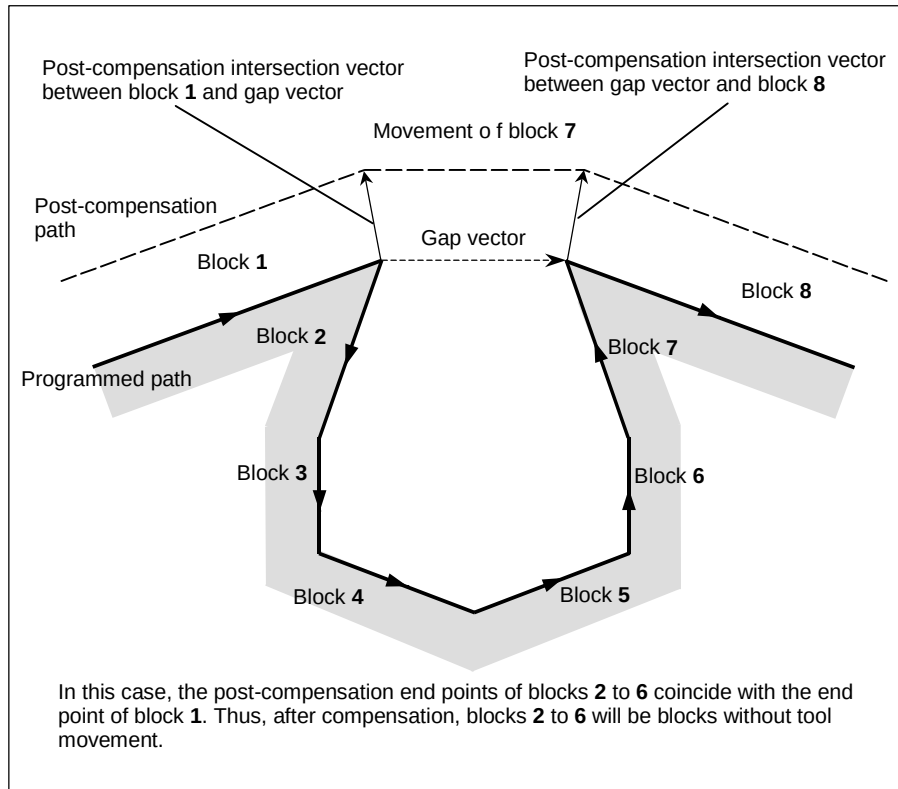


Fig. 16.4.6.3 (a)

If the post-compensation intersection vector of (block 1 - gap vector) and the post-compensation intersection vector of (gap vector - block N) further intersect, vector erasure is first performed in the same way as in "Interference between adjacent three blocks". If the last vectors that remains still intersects, the post-compensation intersection vector of (block 1 - block N) is re-calculated.

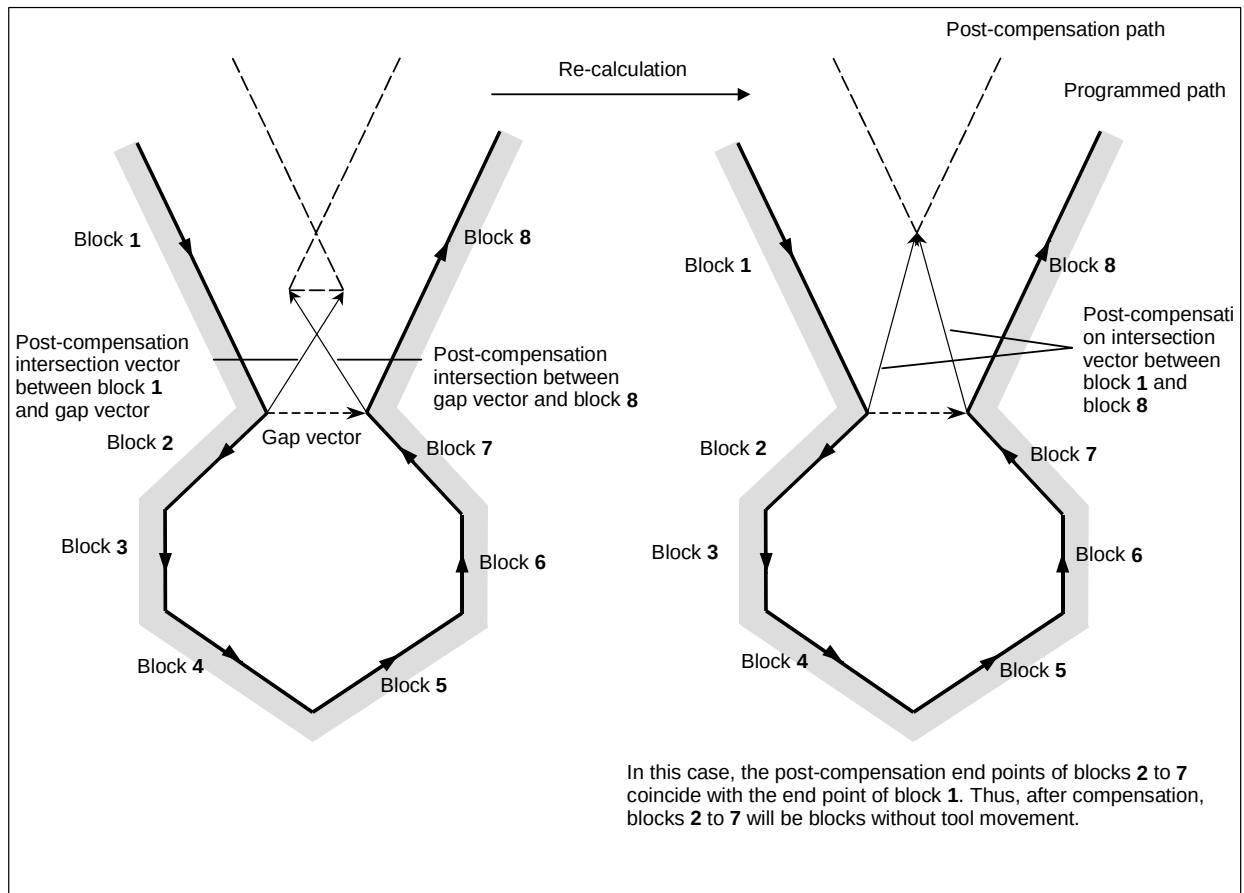


Fig. 16.4.6.3 (b)

If the tool radius compensation value is greater than the radius of the specified arc as shown in the Fig. 16.4.6.3 (c), and a command is specified which results in compensation with respect to the inside of the arc, interference is avoided by performing intersection calculation with an arc command being assumed a linear one. In this case, avoided vectors are connected with linear interpolation.

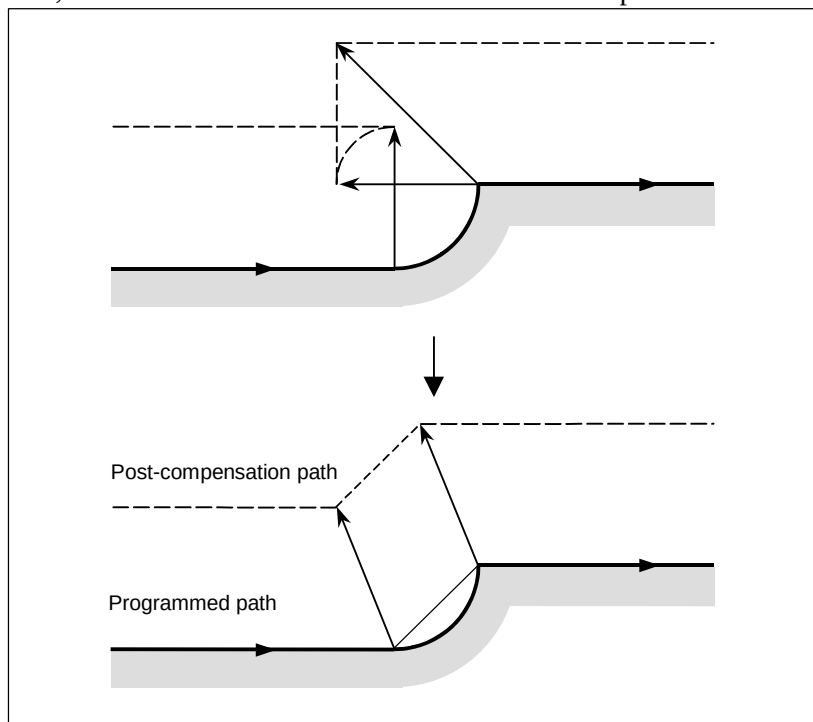


Fig. 16.4.6.3 (c)

- If no interference avoidance vector exists

If the parallel pocket shown in the Fig. 16.4.6.3 (d) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are parallel to each other, no intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop.

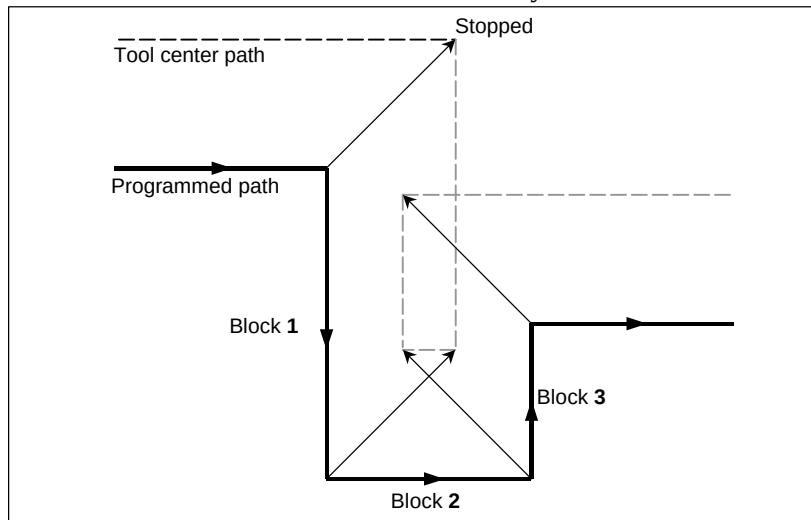


Fig. 16.4.6.3 (d)

If the circular pocket shown in the Fig. 16.4.6.3 (e) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are circular, no post-compensation intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop, as in the previous example.

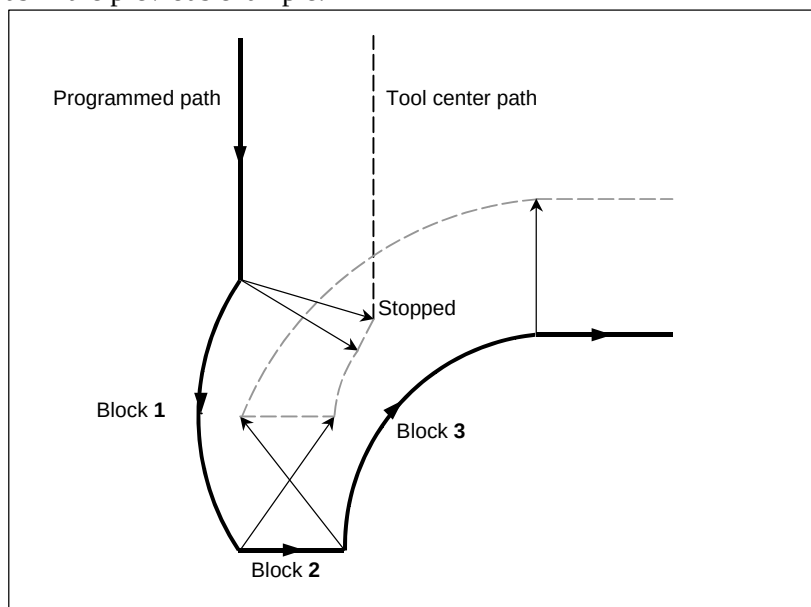


Fig. 16.4.6.3 (e)

- If it is judged dangerous to avoid interference

If the acute-angle pocket shown in the Fig. 16.4.6.3 (f) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the movement direction of the post-avoidance path extremely differs from the previously specified direction. If the post-avoidance path extremely differs from that of the original command (90° or greater or 270° or less), interference avoidance operation is judged dangerous; an alarm will occur immediately before block 1 and the tool will stop.

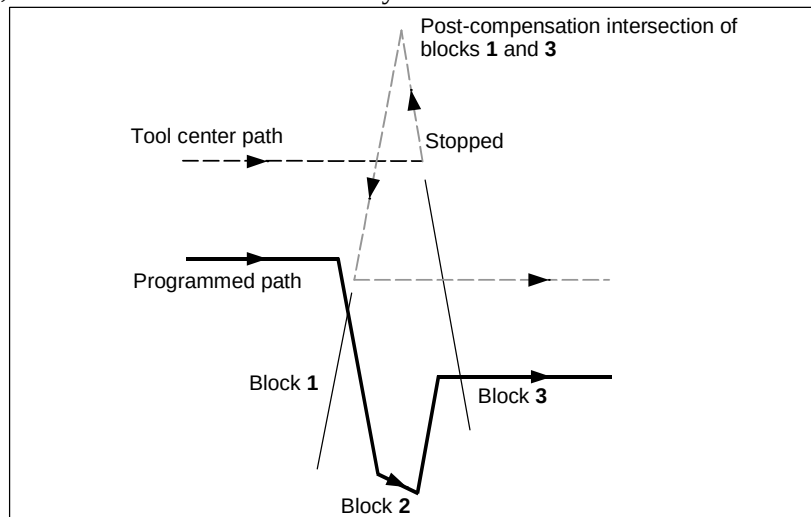


Fig. 16.4.6.3 (f)

If a pocket in which the bottom is wider than the top, such as that shown in the Fig. 16.4.6.3 (g), is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the relation between blocks 1 and 3 is judged an outer one, the post-avoidance path results in overcutting as compared with the original command. In such a case, interference avoidance operation is judged dangerous; an alarm will occur immediately before block 1 and the tool will stop.

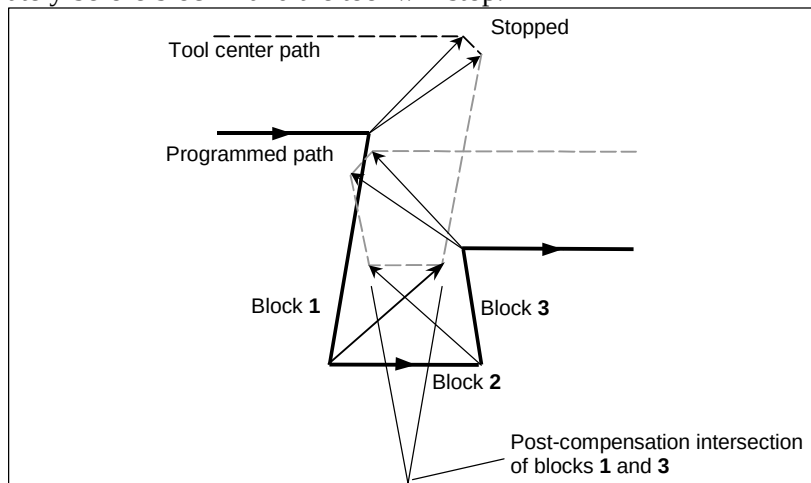


Fig. 16.4.6.3 (g)

- If further interference with an interference avoidance vector occurs

If the pocket shown in the Fig. 16.4.6.3 (h) is to be machined, if the number of blocks to read is 3, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, however, the end-point vector of block 3 that is to be calculated next further interferes with the previous interference avoidance vector.

If a further interference occurs to the interference avoidance vector once created and output, the movement in the block will not be performed; an alarm will occur immediately before the block and the tool will stop.

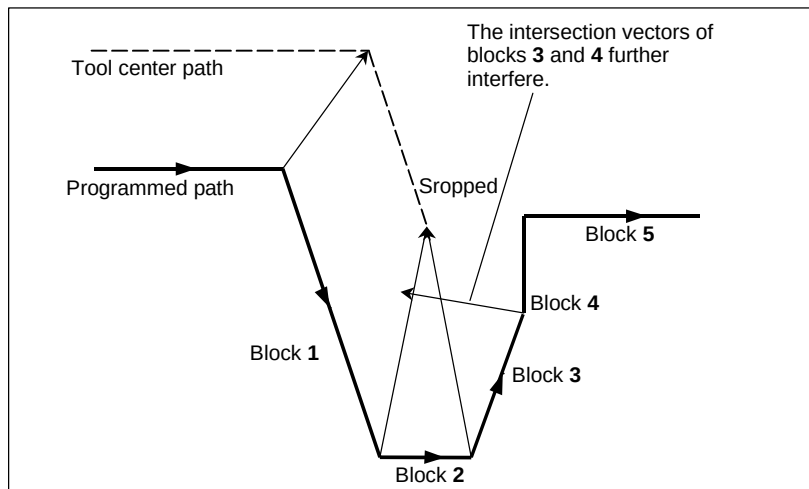


Fig. 16.4.6.3 (h)

NOTE

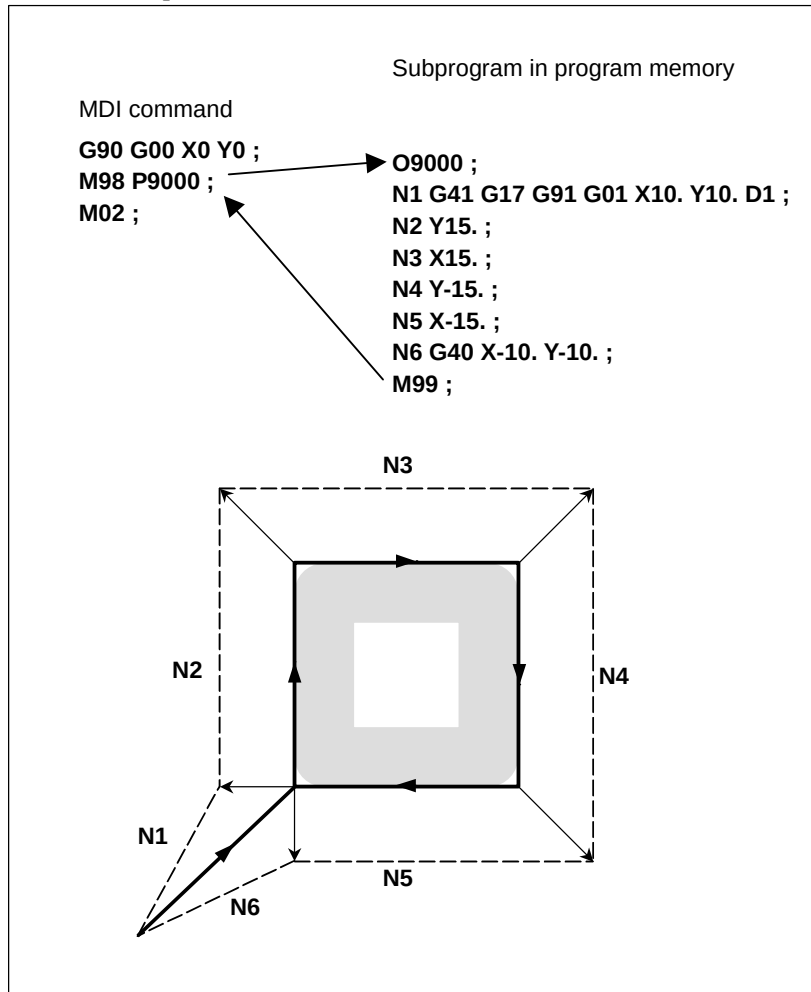
- 1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting bit 6 (NAA) of parameter No. 19607 appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", however, it is not possible to avoid an alarm regardless of the setting of this parameter.
- 2 If a single block stop occurs during interference avoidance operation, and an operation is performed which differs from the original movement, such as manual intervention, MDI intervention, tool radius compensation value change, intersection calculation is performed with a new path. If such an operation is performed, therefore, an interference may occur again although interference avoidance has been performed once.

16.4.7 Tool Radius Compensation for Input from MDI

Explanation

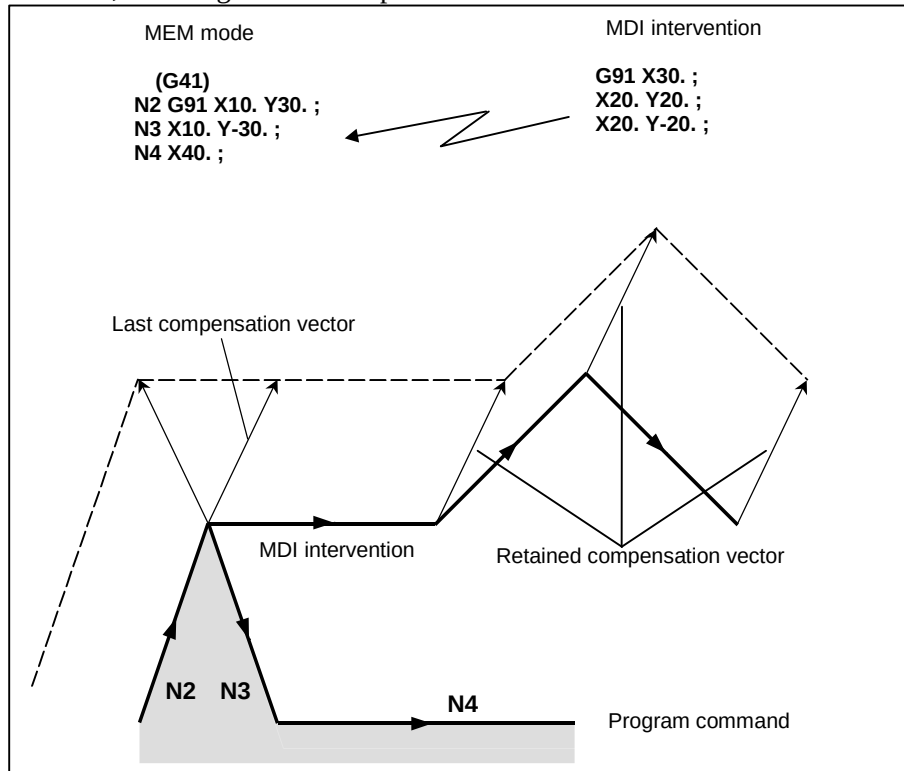
- MDI operation

During MDI operation, that is, if a program command is specified in MDI mode in the reset state to make a cycle start, intersection calculation is performed for compensation in the same way as in memory operation/DNC operation. Compensation is performed in the same way if a subprogram is called from program memory due to MDI operation.



- MDI intervention

If MDI intervention is performed, that is, if a single block stop is performed to enter the automatic operation stop state in the middle of memory operation, DNC operation, and the like, and a program command is specified in MDI mode to make a cycle start, tool compensation does not perform intersection calculation, retaining the last compensation vector before the intervention.



16.5 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Tool compensation values can be entered into CNC memory from the MDI panel or from a program. A tool compensation value is selected from the CNC memory when the corresponding code is specified after address D in a program.

The value is used for tool compensation.

Explanation

- Tool compensation memory A

The compensation value for tool compensation (for D code) is to be set in the compensation memory

Setting example

Compensation number	Compensation value
001	10.000
002	20.000
003	100.000
:	:

- Unit and valid range of tool compensation values

A unit and valid range of tool offset values can be selected from the following by parameter setting:

Unit and valid range of tool compensation values (metric input)

OFA	Unit	Valid range
1	0.01mm	±9999.99mm
0	0.001mm	±9999.999mm

Unit and valid range of tool compensation values (inch input)

OFA	Unit	Valid range
1	0.001inch	±999.999inch
0	0.0001inch	±999.9999inch

- Number of tool compensation data items

The number of tool compensation data is 32.

Format

```
G10 L11 P_ R_ ;
P_   : Tool compensation number
R_   : Tool compensation value
```

By specifying G10, a tool compensation value can be set or modified.

When G10 is specified by absolute input (G90), the specified value is used as the new tool compensation value.

When incremental input (G91) is used, a specified value added to the tool compensation value currently set is used as the new tool compensation value.

NOTE

- 1 Address R follows the increment system for tool offset values.
- 2 If L is omitted for compatibility with the conventional CNC format, or L1 is specified, the same operation as when L11 is specified is performed.

16.6 COORDINATE SYSTEM ROTATION (G84, G85)

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

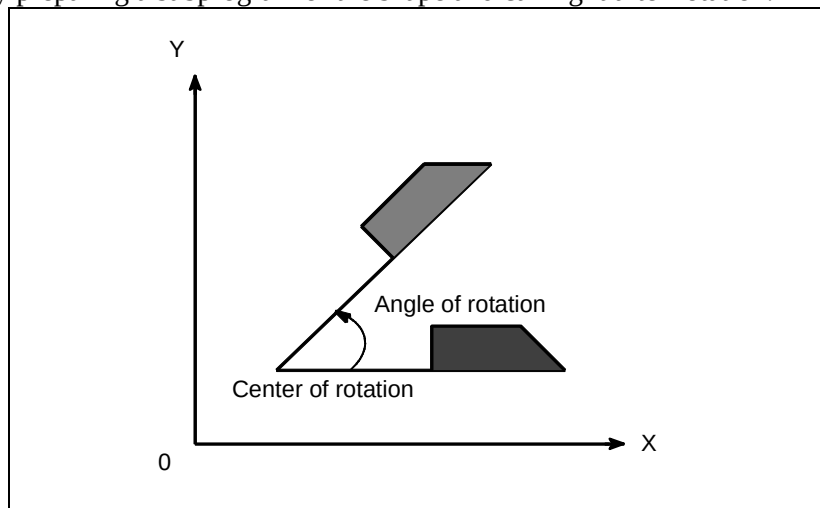


Fig. 16.6 (a) Coordinate system rotation

Format

$\left. \begin{array}{l} \text{G17} \\ \text{G18} \\ \text{G19} \end{array} \right\}$	G84 $\alpha\_ \beta\_ R\_ ;$	Start rotation of a coordinate system.
$\left. \begin{array}{l} : \\ : \end{array} \right\}$		Coordinate system rotation mode (The coordinate system is rotated.)
G85 ;		Coordinate system rotation cancel command

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

$\alpha\_ \beta\_ :$ Absolute programming for two of the X_, Y_, and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G84

$R\_ :$ Angular displacement with a positive value indicates counter clockwise rotation. Bit 0 (RIN) of parameter No. 5400 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).

Least input increment : 0.001 deg

Valid data range : -360,000 to 360,000

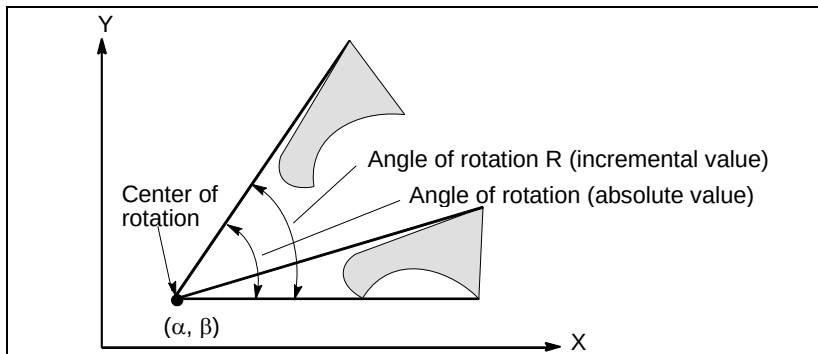


Fig. 16.6 (b) Coordinate system rotation

NOTE

When a decimal fraction is used to specify angular displacement ($R\_$), the 1's digit corresponds to degree units.

Explanation

- G code for selecting a plane: G17,G18 or G19

The G code for selecting a plane (G17,G18,or G19) can be specified before the block containing the G code for coordinate system rotation (G84). G17, G18 or G19 must not be designated in the mode of coordinate system rotation.

- Incremental programming in coordinate system rotation mode

The center of rotation for an incremental programming programmed after G84 but before an absolute programming is the tool position when G84 was programmed (Fig. 16.6 (c)).

- Center of rotation

When $\alpha\_ \beta\_$ is not programmed, the tool position when G84 was programmed is assumed as the center of rotation.

- Angular displacement

When R_ is not specified, the value specified in parameter No. 5410 is assumed as the angular displacement.

To specify angular displacement (R_) in 0.00001 degrees (one hundred-thousandth), set bit 0 (FRD) of parameter No. 11630 to 1. In this case, angular displacement R is specified within the range of -36000000 to 36000000.

- Coordinate system rotation cancel command

The G code used to cancel coordinate system rotation (G85) may be specified in a block in which another command is specified.

- Tool compensation

Tool radius compensation, and other compensation operations are executed after the coordinate system is rotated.

Limitation

- Commands related to reference position return and the coordinate system

In coordinate system rotation mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling coordinate system rotation mode.

- Incremental programming

The first move command after the coordinate system rotation cancel command (G85) must be specified with absolute values. If an incremental move command is specified, correct movement will not be performed.

- Note on the specification of one axis in coordinate system rotation

With the parameter below, a move position in the case where one axis is specified in the absolute mode can be selected. If two axes are specified, a movement is made to the same position, regardless of the setting of the parameter.

Bit 5 (AX1) of parameter No. 11600

If one axis is specified in the absolute mode when the coordinate system rotation mode is set:

0: The specified position is first calculated in the coordinate system before rotation then the coordinate system is rotated.

1: The coordinate system is first rotated then a movement is made to the specified position in the rotated coordinate system. (FS0i-PC compatible specification)

This parameter changes the handling of coordinates on axes not specified, so that a position to be reached by movement changes.

(Example)

G90 G0 X0 Y0

G01 X10. Y10. F6000

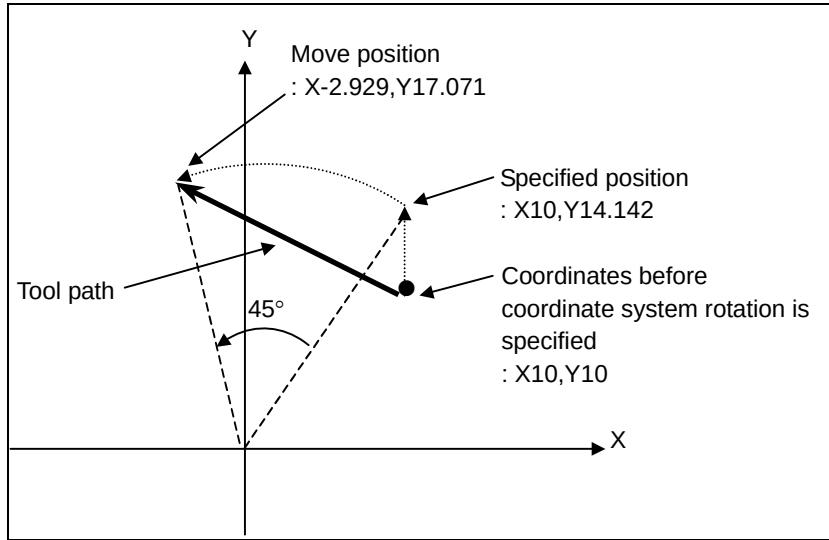
G84 X0 Y0 R45.... Specifies coordinate system rotation.

Y14.142..... Specifies one axis(1)

G85

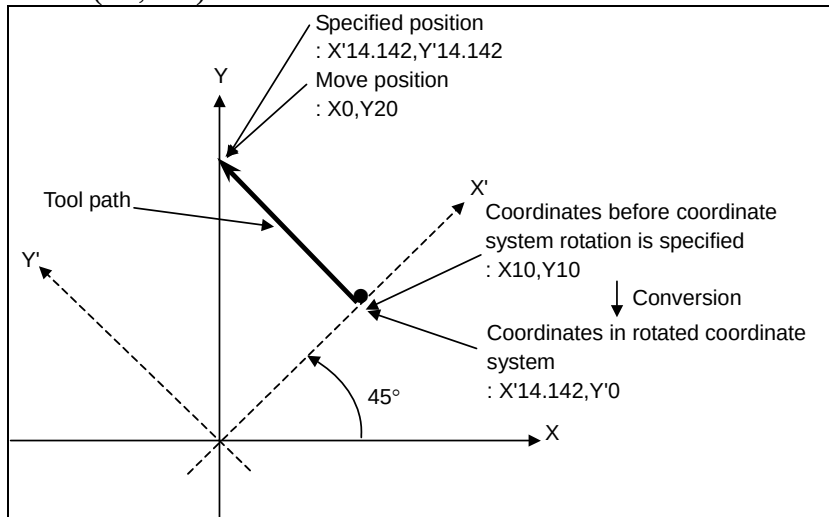
When bit 5 (AX1) of parameter No. 11600= 0:

The specified position is calculated in the coordinate system (XY) before rotation then the coordinate system is rotated. So, with the specification of (1), the position on the unspecified X axis is X10, and the specified position is (X10,Y14.142). Next, a movement is made to the move position (X-2.929,Y17.071) obtained by 45° rotation.



When bit 5 (AX1) of parameter No. 11600= 1:

With the specification of (1), coordinates $(X10, Y10)$ before coordinate system rotation are converted to coordinates $(X'14.142, Y'0)$ in the coordinate system $(X'Y')$ obtained by 45° rotation. Next, a movement is made to the specified position $(X'14.142, Y'14.142)$, that is, the move position $(X0, Y20)$.



Explanation

- Absolute/Incremental position commands

```

N1 G92 X-500.0 Y-500.0 G69 G17 ;
N2 G84 X700.0 Y300.0 R60.0 ;
N3 G90 G01 X0 Y0 F200 ;
    (G91X500.0Y500.0)
N4 G91 X1000.0 ;
N5 G02 Y1000.0 R1000.0 ;
N6 G03 X-1000.0 I-500.0 J-500.0 ;
N7 G01 Y-1000.0 ;
N8 G85 G90 X-500.0 Y-500.0 M02 ;

```

Tool path when the incremental command is designated in the N3 block (in parenthesis)

Originally programmed tool path

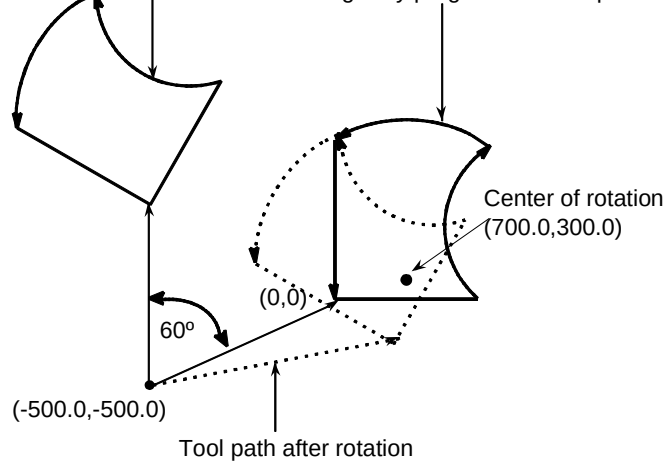


Fig. 16.6 (c) Absolute/incremental programming during coordinate system rotation

- Tool compensation and coordinate system rotation

It is possible to specify G84 and G85 in cutter compensation mode. The rotation plane must coincide with the plane of cutter compensation.

```

N1 G92 X0 Y0 G69 G01 ;
N2 G42 G90 X1000 Y1000 F1000 D01 ;
N3 G84 R-30000 ;
N4 G91 X2000 ;
N5 G03 Y1000 R1000 J500 ;
N6 G01 X-2000 ;
N7 Y-1000 ;
N8 G85 G40 G90 X0 Y0 M30 ;

```

Programmed shape before coordinate system rotation

Programmed shape after coordinate system rotation

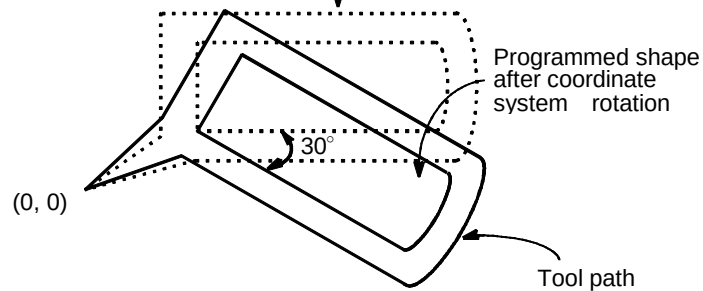


Fig. 16.6 (d) Tool compensation and coordinate system rotation

- Scaling and coordinate system rotation

If a coordinate system rotation command is executed in the scaling mode (G51 mode), the coordinate value (a, b) of the rotation center will also be scaled, but not the rotation angle (R). When a move command is issued, the scaling is applied first and then the coordinates are rotated.

A coordinate system rotation command (G84) should not be issued in tool compensation mode (G41, G42) on scaling mode (G51). The coordinate system rotation command should always be specified prior to setting the tool compensation mode.

- When the system is not in tool compensation mode, specify the commands in the following order :
 G51 ; Scaling mode start
 G84 ; Coordinate system rotation mode start
 :
 G85 ; Coordinate system rotation mode cancel
 G50 ; Scaling mode cancel
- When the system is in tool compensation, specify the commands in the following order (Fig. 16.6 (e)) :
 (tool compensation cancel)
 G51 ; Scaling mode start
 G84 ; Coordinate system rotation start
 :
 G41 ; Tool compensation mode start
 :

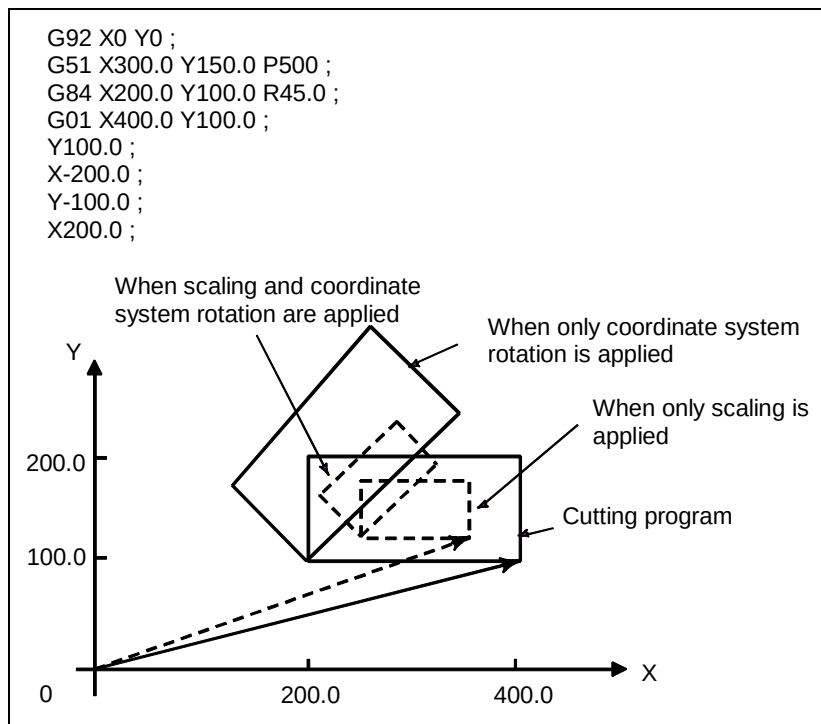


Fig. 16.6 (e) Scaling and coordinate system rotation in tool compensation mode

- Repetitive commands for coordinate system rotation

It is possible to store one program as a subprogram and recall subprogram by changing the angle.

Sample program for when parameter RIN (No. 5400#0) is set to 1.
The specified angular displacement is treated as an absolute or incremental value depending on the specified G code (G90 or G91).

```
G92 X0 Y0 G85 G17;  
G01 F200 H01 ;  
M98 P2100 ;  
M98 P072200 ;  
G00 G90 X0 Y0 M30 ;
```

```
O 2200 G84 X0 Y0 G91 R45.0 ;  
G90 M98 P2100 ;  
M99 ;
```

```
O 2100 G90 G01 G42 X0 Y-10.0 ;  
X4.142 ;  
X7.071 Y-7.071 ;  
G40 ;  
M99 ;
```

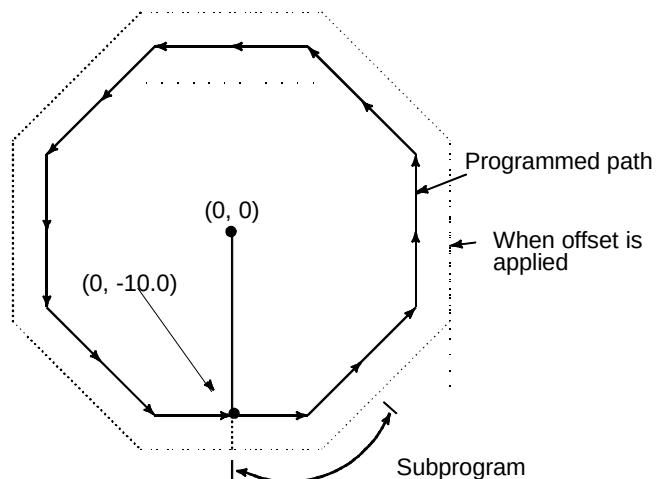
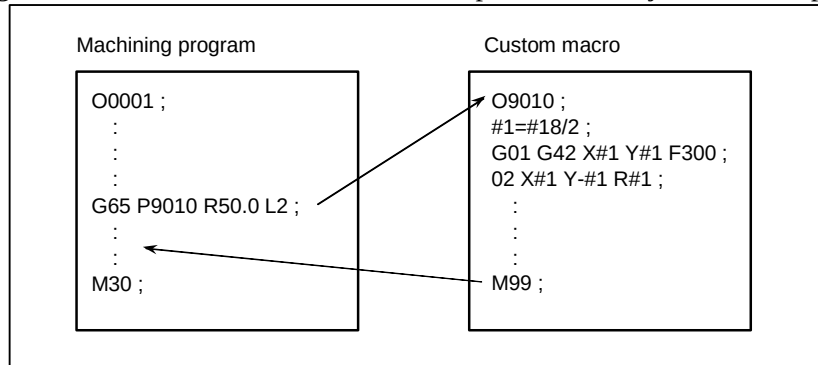


Fig. 16.6 (f) Coordinate system rotation command

17 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.



Chapter 17, "CUSTOM MACRO", consists of the following sections:

17.1	VARIABLES	231
17.2	SYSTEM VARIABLES.....	236
17.3	ARITHMETIC AND LOGIC OPERATION.....	256
17.4	READING PARAMETERS	262
17.5	MACRO STATEMENTS AND NC STATEMENTS	263
17.6	BRANCH AND REPETITION	264
17.7	MACRO CALL.....	268
17.8	PROCESSING MACRO STATEMENTS.....	281
17.9	REGISTERING CUSTOM MACRO PROGRAMS	283
17.10	CODES AND RESERVED WORDS USED IN CUSTOM MACROS	283
17.11	EXTERNAL OUTPUT COMMANDS	284
17.12	RESTRICTIONS	287
17.13	INTERRUPTION TYPE CUSTOM MACRO	288

17.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 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, "OVERFLOW :FLOATING" is issued.

When bit 0 (F16) 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 (F16) 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.

The 600 common variables #100 to #199 and #500 to #999 can be used. The variables #100 to #199 are cleared during power-down, but the variables #500 to #999 are retained during power-down.

- 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, "WRITE PROTECTED VARIABLE" 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 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 = 12345.0, 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 quoted, 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, numerals, 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]

```
[#_DATE]=20040117 ;      : 2004.01.17 is assigned to #3011 (year month date).
[#_TIME]=161705 ;       : 16:17:05 is assigned to #3012 (time minute second).
#101=[#_ABSMT[1]] ;     : #5021 (machine coordinate value of the 1st axis) is read off
                        : and assigned to #101.
#102=[#_ABSKP[#500*2]] ; : #506x (skip position of [#500*2]th axis) is read off and
                        : assigned to #102.
```

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, "NO VARIABLE NAME" is issued.
- 2 When a negative or other invalid subscript is specified, an alarm PS1099, "ILLEGAL SUFFIX []" 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 $\pi = 3.14159265358979323846$
#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.

17.2 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

Tool compensation memory A

System variable number	System variable name	Attribute	Description
#2001-#2032	[_OFS[n]]	R/W	Tool compensation value Note) Subscript n represents a compensation number (1 to 32).
#10001-#10032			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 32).

- 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)
#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 bit 0	[_M_SBK]	R/W	Enable or disable the suppression of single block stop.
#3003 bit 1	[_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 bit 0	[_M_FHD]	R/W	Enable or disable feed hold.
#3004 bit 1	[_M_OV]	R/W	Enable or disable feedrate override.

System variable number	System variable name	Attribute	Description
#3004 bit 2	[#_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)

- 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

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	[#_MAIN0]	R	Main program number

- Modal information

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)
#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)

System variable number	System variable name	Attribute	Description
#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.
#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)
#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)

- Position information

System variable number	System variable name	Attribute	Description
#5001-#5007	[#_ABSIO[n]]	R	End point position of the previous block (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 7).
#100001-#100007			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).
#5021-#5027	[#_ABSMT[n]]	R	Specified current position (machine coordinate system) Note) Subscript n represents an axis number (1 to 7).
#100051-#100057			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).
#5041-#5047	[#_ABSOT[n]]	R	Specified current position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 7).
#100101-#100107			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).
#5061-#5067	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 7).
#100151-#100157			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).

- Servo position deviation

System variable number	System variable name	Attribute	Description
#5101-#5107	[#_SVERR[n]]	R	Servo positional deviation Note) Subscript n represents an axis number (1 to 7).
#100251-#100257			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).

- Manual handle interruption

System variable number	System variable name	Attribute	Description
#5121-#5127	[#_MIRTP[n]]	R	Manual handle interruption Note) Subscript n represents an axis number (1 to 7).
#100651-#100657			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).

- Distance to go

System variable number	System variable name	Attribute	Description
#5181-#5187	[#_DIST[n]]	R	Distance to go Note) Subscript n represents an axis number (1 to 7).
#100801-#100807			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 7).

- Workpiece origin offset value

System variable number	System variable name	Attribute	Description
#5201-#5207	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 7).
#5221-#5227	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 7).
#5241-#5247	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 7).

System variable number	System variable name	Attribute	Description
#5261-#5267	[#_WZG56[n]]	R/W	G56 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#5281-#5287	[#_WZG57[n]]	R/W	G57 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#5301-#5307	[#_WZG58[n]]	R/W	G58 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#5321-#5327	[#_WZG59[n]]	R/W	G59 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#100301-#100307	[#_WZCMN[n]]	R/W	External workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#100351-#100357	[#_WZG54[n]]	R/W	G54 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#100401-#100407	[#_WZG55[n]]	R/W	G55 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#100451-#100457	[#_WZG56[n]]	R/W	G56 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#100501-#100507	[#_WZG57[n]]	R/W	G57 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#100551-#100557	[#_WZG58[n]]	R/W	G58 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).
#100601-#100607	[#_WZG59[n]]	R/W	G59 workpiece origin offset value (Note) Subscript n represents an axis number (1 to 7).

- Skip position (detection unit)

System variable number	System variable name	Attribute	Description
#5421-#5427	[#_SKPDTC[n]]	R	Skip position (detection unit) (Note) Subscript n represents an axis number (1 to 7).
#100701-#100707			The numbers to the left can also be used. (Note) Subscript n represents an axis number (1 to 7).

- Pattern function

System variable number	System variable name	Attribute	Description
#6251-#6257	[#_MPTREFP[n]]	R	Coordinate of pattern base point (NOTE) Subscript n is number of axis (1 to 7).
#154001-#154007			Following number is enabled. (NOTE) Subscript n is number of axis (1 to 7).

- Multi-piece machining function

System variable number	System variable name	Attribute	Description
#6261-#6267	[#_MHSREFP[n]]	R	Coordinate of multi-piece machining (NOTE) Subscript n is number of axis (1 to 7).
#154051-#154057			Following number is enabled. (NOTE) Subscript n is number of axis (1 to 7).

- Automatic repositioning

System variable number	System variable name	Attribute	Description
#6270	[#_MATREP]	R	Repositioning value

- Local coordinate system

System variable number	System variable name	Attribute	Description
#6271-#6277	[#_MLOCPOS[n]]	R	Origin of the local coordinate system (NOTE) Subscript n is number of axis (1 to 7).
#154101-#154107			Following number is enabled. (NOTE) Subscript n is number of axis (1 to 7).

- System constant

System constant number	System constant name	Attribute	Description
#0, #3100	[#_EMPTY]	R	Null
#3101	[#_PI]	R	Circular constant $\pi = 3.14159265358979323846$
#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 #1031.

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})

Variable number	Variable name	Point	Interface input signal
#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})

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.

Variable number	Variable name	Point	Interface input signal
#1032	[#_UIL[0]]	32	UI000-UI031
#1033	[#_UIL[1]]	32	UI100-UI131
#1034	[#_UIL[2]]	32	UI200-UI231
#1035	[#_UIL[3]]	32	UI300-UI331

$$\#1032 = \sum_{i=0}^{30} \# [1000 + i] \times 2^i - \#1031 \times 2^{31}$$

$$\# [1032 + n] = \sum_{i=0}^{30} \{ 2^i \times V_i \} - 2^{31} \times V_{31}$$

When $UIL_i = 0$, $V_i = 0$.

When $UIL_i = 1$, $V_i = 1$.

$n = 0-3$

[Output signal]

Interface output signals can be sent by assigning values to system variables #1100 to #1131 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})

Variable number	Variable name	Point	Interface input signal
#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)
#1130	[#_UO[30]]	1	UO030 (2 ³⁰)
#1131	[#_UO[31]]	1	UO031 (2 ³¹)

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.

Variable number	Variable name	Point	Interface input signal
#1132	[#_UOL[0]]	32	UO000-UO031
#1133	[#_UOL[1]]	32	UO100-UO131
#1134	[#_UOL[2]]	32	UO200-UO231
#1135	[#_UOL[3]]	32	UO300-UO331

$$\#1132 = \sum_{i=0}^{30} \#[1000 + i] \times 2^i - \#1131 \times 2^{31}$$

$$\#[1132 + n] = \sum_{i=0}^{30} \{2^i \times V_i\} - 2^{31} \times V_{31}$$

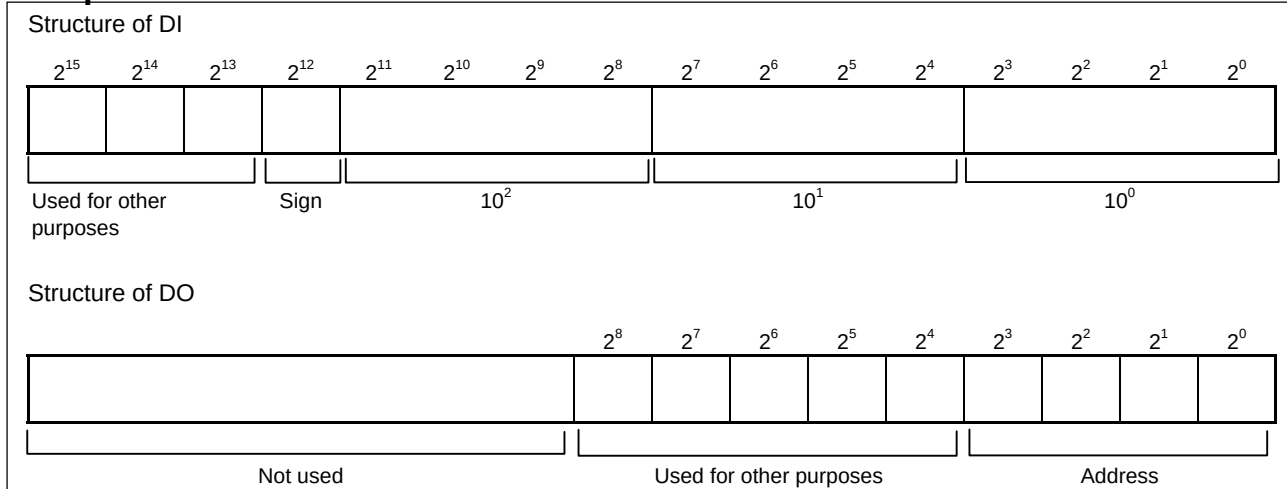
When $UIn_i = 0$, $V_i = 0$.

When $UIn_i = 1$, $V_i = 1$.

$n = 0-3$

NOTE

- 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.
- 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, bit 0 (MIF) of parameter No. 6001 must be set to 1.

Example

<1> The specified address D value is output to DO and signed BCD 3 digits are read into #100.

Macro calling instruction

G65 P9100 D (address);

A custom macro body is created as follows.

```
O9100 ;
#1132 = #1132 AND 496 OR #7 ;      : Address output
G65 P9101 T60 ;                    : Timer macro. For details, refer to the example of clock (#3001).
#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 specified address D values are output to DO and 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 P9111 D (data number);

A custom macro body is created as follows.

```
O9111 ;
G65 P9100 D[#7*2+1] ;              : Call the macro of <1> O9100
#101 = #100 ;
G65 P9100 D[#7*2] ;
#101 = #101 + #100/1000 ;
M99 ;
```

- Tool compensation value #2001-#2032, #10001-#10032 (Attribute: R/W)

The compensation values can be obtained by reading system variables #2001 to #2032 or #10001 to #10032 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Tool compensation memory A

Compensation number	Variable number	Variable name
1	#2001 or #10001	[#_OFS[1]]
2	#2002 or #10002	[#_OFS[2]]
3	#2003 or #10003	[#_OFS[3]]
:	:	:
31	#2031 or #10031	[#_OFS[31]]
32	#2032 or #10032	[#_OFS[32]]

- 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 after MC.

(Example) #3000 = 1 (ALARM MESSAGE);

→ "MC3001 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]

Punch cycle at corners of rectangle (for incremental programming)

Macro calling command

G65 P9081 Xxx Yyy ;

A custom macro body is created as follows.

O9081 ;

#3003 = 1 ;

G00 X0 Y0 ;

G00 X#24 ;

G00 Y#25 ;

G00 X-#24 ;

#3003 = 0 ;

M99 ;

Disable single block stop.
#24 corresponds to X and #25 to Y.

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

- These system variables are provided to maintain compatibility with conventional NC programs. It is not recommended to use control of feed hold and feedrate override. In addition, it is recommended that functions provided by G09, G61, and G64 is used to enable or disable exact stop.
- When the feed hold button is pressed during execution of a block for which feed hold is disabled:
 - 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.
 - 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.
- #3004 is cleared by a reset.
- 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								
	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC
#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	31 to 7	6	5	4	3	2	1	0
nth axis	-	7	6	5	4	3	2	1

For the 32 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 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.
- 2 When mirror image signals for axes other than the controlled axes are turned on, they are not read into system variable #3007.

- 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] Jan 20, 2011, PM 04:17:05

#3011 = 20110120

#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)

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

NOTE

Only tool compensation memory A can be used by Series 0i-PD.

- 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-#4120, #4201-#4320, #4401-#4520 (Attribute: R)

The modal information specified before the previous block of the macro statement that reads system variables #4001 to #4120 can be obtained in the block currently being looked ahead, by reading system variables #4001 to #4120.

The modal information of the block currently being executed can be obtained by reading system variables #4201 to #4320.

The modal information specified before the block interrupted by an interruption type custom macro by reading system variables #4401 to #4520.

The unit used when it was specified is applied.

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1>	#4001	[#_BUFG[1]]	Modal information (G code: group 1)
<2>	#4201	[#_ACTG[1]]	
<3>	#4401	[#_INTG[1]]	
<1>	#4002	[#_BUFG[2]]	Modal information (G code: group 2)
<2>	#4202	[#_ACTG[2]]	
<3>	#4402	[#_INTG[2]]	
:	:	:	:
<1>	#4030	[#_BUFG[30]]	Modal information (G code: group 30)
<2>	#4230	[#_ACTG[30]]	
<3>	#4430	[#_INTG[30]]	
<1>	#4102	[#_BUFB]	Modal information (B code)
<2>	#4302	[#_ACTB]	
<3>	#4502	[#_INTB]	
<1>	#4107	[#_BUFD]	Modal information (D code)
<2>	#4307	[#_ACTD]	
<3>	#4507	[#_INTD]	
<1>	#4108	[#_BUFE]	Modal information (E code)
<2>	#4308	[#_ACTE]	
<3>	#4508	[#_INTE]	
<1>	#4109	[#_BUFF]	Modal information (F code)
<2>	#4309	[#_ACTF]	
<3>	#4509	[#_INTF]	

Category	Variable number	Variable name	Description
<1>	#4111	[#_BUFH]	Modal information (H code)
<2>	#4311	[#_ACTH]	
<3>	#4511	[#_INTH]	
<1>	#4113	[#_BUFM]	Modal information (M code)
<2>	#4313	[#_ACTM]	
<3>	#4513	[#_INTM]	
<1>	#4114	[#_BUFN]	Modal information (sequence number N)
<2>	#4314	[#_ACTN]	
<3>	#4514	[#_INTN]	
<1>	#4115	[#_BUFO]	Modal information (sequence number O)
<2>	#4315	[#_ACTO]	
<3>	#4515	[#_INTO]	
<1>	#4119	[#_BUFS]	Modal information (S code)
<2>	#4319	[#_ACTS]	
<3>	#4519	[#_INTS]	
<1>	#4120	[#_BUFT]	Modal information (T code)
<2>	#4320	[#_ACTT]	
<3>	#4520	[#_INTT]	

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 #4120 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-#5067, #100001-#100157 (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 #5067 or #100001 to #100157.

Variable number	Variable name	Position information	Coordinate system	Tool position cutter compensation	Reading operation during movement
#5001 #5002 : #5007	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[7]]	1st axis block end point position 2nd axis block end point position : 7th axis block end point position	Workpiece coordinate system	Not included	Enabled
#100001 #100002 : #100007	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[7]]	1st axis block end point position 2nd axis block end point position : 7th axis block end point position			
#5021 #5022 : #5027	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[7]]	1st axis current position 2nd axis current position : 7th axis current position	Machine coordinate system	Included	Disabled
#100051 #100052 : #100057	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[7]]	1st axis current position 2nd axis current position : 7th axis current position			
#5041 #5042 : #5047	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[7]]	1st axis current position 2nd axis current position : 7th axis current position	Workpiece coordinate system	Included	Disabled
#100101 #100102 : #100107	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[7]]	1st axis current position 2nd axis current position : 7th axis current position			
#5061 #5062 : #5067	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[7]]	1st axis skip position 2nd axis skip position : 7th axis skip position	Workpiece coordinate system	Included	Enabled
#100151 #100152 : #100157	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[7]]	1st axis skip position 2nd axis skip position : 7th axis skip position			

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 (G33) 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.

- **Servo position deviation #5101-#5107, #100251-#100257 (Attribute: R)**

The servo position deviation for each axis can be obtained by reading system variables #5101 to #5120 or #100251 to #100300.

Variable number	Variable name	Position information	Read operation during movement
#5101	[_SVERR[1]]	1st axis servo position deviation	Disabled
#5102	[_SVERR[2]]	2nd axis servo position deviation	
:	:	:	
#5107	[_SVERR[7]]	7th axis servo position deviation	
#100251	[_SVERR[1]]	1st axis servo position deviation	
#100252	[_SVERR[2]]	2nd axis servo position deviation	
:	:	:	
#100257	[_SVERR[7]]	7th axis servo position deviation	

NOTE

When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.

- **Manual handle interruption #5121-#5127, #100651-#100657 (Attribute: R)**

The manual handle interruption for each axis can be obtained by reading system variables #5121 to #5127 or #100651 to #100657.

Variable number	Variable name	Position information	Read operation during movement
#5121	[_MIRTP[1]]	1st axis manual handle interruption	Disabled
#5122	[_MIRTP[2]]	2nd axis manual handle interruption	
:	:	:	
#5127	[_MIRTP[7]]	7th axis manual handle interruption	
#100651	[_MIRTP[1]]	1st axis manual handle interruption	
#100652	[_MIRTP[2]]	2nd axis manual handle interruption	
:	:	:	
#100657	[_MIRTP[7]]	7th axis manual handle interruption	

NOTE

When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.

- **Distance to go #5181-#5187, #100801-#100807 (Attribute: R)**

The distance to go value for each axis can be obtained by reading system variables #5181 to #5187 or #100801 to #100807.

Variable number	Variable name	Position information	Read operation during movement
#5181	[_DIST[1]]	1st axis distance to go value	Disabled
#5182	[_DIST[2]]	2nd axis distance to go value	
:	:	:	
#5187	[_DIST[7]]	7th axis distance to go value	
#100801	[_DIST[1]]	1st axis distance to go value	
#100802	[_DIST[2]]	2nd axis distance to go value	
:	:	:	
#100807	[_DIST[7]]	7th axis distance to go value	

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-#5327, #100301-#100607 (Attribute: R/W)

The workpiece origin offset value can be obtained by reading system variables #5201 to #5327 or #100301 to #100607. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#5201	[#_WZCMN[1]]	1st axis external workpiece origin offset value	External workpiece origin offset value (applied to all coordinate systems)
#5202	[#_WZCMN[2]]	2nd axis external workpiece origin offset value	
:	:	:	
#5207	[#_WZCMN[7]]	7th axis external workpiece origin offset value	
#5221	[#_WZG54[1]]	1st axis workpiece origin offset value	G54
#5222	[#_WZG54[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5227	[#_WZG54[7]]	7th axis workpiece origin offset value	
#5241	[#_WZG55[1]]	1st axis workpiece origin offset value	G55
#5242	[#_WZG55[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5247	[#_WZG55[7]]	7th axis workpiece origin offset value	
#5261	[#_WZG56[1]]	1st axis workpiece origin offset value	G56
#5262	[#_WZG56[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5267	[#_WZG56[7]]	7th axis workpiece origin offset value	
#5281	[#_WZG57[1]]	1st axis workpiece origin offset value	G57
#5282	[#_WZG57[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5287	[#_WZG57[7]]	7th axis workpiece origin offset value	
#5301	[#_WZG58[1]]	1st axis workpiece origin offset value	G58
#5302	[#_WZG58[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5307	[#_WZG58[7]]	7th axis workpiece origin offset value	
#5321	[#_WZG59[1]]	1st axis workpiece origin offset value	G59
#5322	[#_WZG59[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5327	[#_WZG59[7]]	7th axis workpiece origin offset value	

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#100301	[#_WZCMN[1]]	1st axis external workpiece origin offset value	External workpiece origin offset value (applied to all coordinate systems)
#100302	[#_WZCMN[2]]	2nd axis external workpiece origin offset value	
:	:	:	
#100307	[#_WZCMN[7]]	7th axis external workpiece origin offset value	
#100351	[#_WZG54[1]]	1st axis workpiece origin offset value	G54
#100352	[#_WZG54[2]]	2nd axis workpiece origin offset value	
:	:	:	
#100357	[#_WZG54[7]]	7th axis workpiece origin offset value	
#100401	[#_WZG55[1]]	1st axis workpiece origin offset value	G55
#100402	[#_WZG55[2]]	2nd axis workpiece origin offset value	
:	:	:	
#100407	[#_WZG55[7]]	7th axis workpiece origin offset value	
#100451	[#_WZG56[1]]	1st axis workpiece origin offset	G56
#100452	[#_WZG56[2]]	2nd axis workpiece origin offset	
:	:	:	
#100457	[#_WZG56[7]]	7th axis workpiece origin offset	
#100501	[#_WZG57[1]]	1st axis workpiece origin offset	G57
#100502	[#_WZG57[2]]	2nd axis workpiece origin offset	
:	:	:	
#100507	[#_WZG57[7]]	7th axis workpiece origin offset	

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#100551	[_WZG58[1]]	1st axis workpiece origin offset	G58
#100552	[_WZG58[2]]	2nd axis workpiece origin offset	
:	:	:	
#100557	[_WZG58[7]]	7th axis workpiece origin offset	G59
#100601	[_WZG59[1]]	1st axis workpiece origin offset	
#100602	[_WZG59[2]]	2nd axis workpiece origin offset	
:	:	:	
#100607	[_WZG59[7]]	7th axis workpiece origin offset	

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

NOTE

- 1 When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.
- 2 The variables #2500 to #2806, #5201 to #5327, and #100301 to #100607 are available when the workpiece coordinate system is applied.

- Skip position (detection unit) #5421-#5427, #100701-#100707 (Attribute: R)

The skip position with the detection unit can be obtained by reading system variables #5421 to #5427 or #100701 to #100707. The values of these variables are integers, and values in the workpiece coordinate system.

Variable number	Variable name	Position information	Read operation during movement
#5421	[_SKPDTC[1]]	1st axis skip position (detection unit)	Disabled
#5422	[_SKPDTC[2]]	2nd axis skip position (detection unit)	
:	:	:	
#5427	[_SKPDTC[7]]	7th axis skip position (detection unit)	
#100151	[_SKPDTC[1]]	1st axis skip position (detection unit)	
#100152	[_SKPDTC[2]]	2nd axis skip position (detection unit)	
:	:	:	
#100157	[_SKPDTC[7]]	7th axis skip position (detection unit)	

NOTE

- 1 When variables exceeding the number of control axes are specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs.
- 2 To specify these variables, set bit 2 (DSK) of parameter No. 6210 to 1. (If an attempt is made to specify them when DSK is 0, alarm PS0115 is issued.)

- **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 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 = 1	#10000	P-CODE variables (#10000)
	:	:
	#89999	P-CODE variables (#89999)
#8570 = 0	#10000	System variables (#10000)
	:	:
	#89999	System 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, "VARIABLE NO. OUT OF RANGE" occurs.

- **Base point of pattern function #6251 to #6257, #154001 to #154007 (Attribute: R)**

The coordinate of base point of pattern function in executing block can be obtained by reading system variables #6251 to #6257, and #154001 to #154007.

Variable number	Variable name	Description
#6251	[_MPTREFP[1]]	Pattern base point on 1st axis
#6252	[_MPTREFP[2]]	Pattern base point on 2nd axis
:	:	:
#6257	[_MPTREFP[7]]	Pattern base point on 7th axis
#154001	[_MPTREFP[1]]	Pattern base point on 1st axis
#154002	[_MPTREFP[2]]	Pattern base point on 2nd axis
:	:	:
#154007	[_MPTREFP[7]]	Pattern base point on 7th axis

- **Base point of multi-piece machining function #6261 to #6267, #154051 to #154057 (Attribute: R)**

The coordinate of base point of multi-piece machining function in executing block can be obtained by reading system variables #6261 to #6267, and #154051 to #154057.

Variable number	Variable name	Description
#6261	[_MHSREFP[1]]	Base point of multi-piece machining on 1st axis
#6262	[_MHSREFP[2]]	Base point of multi-piece machining on 2nd axis
:	:	:
#6267	[_MHSREFP[7]]	Base point of multi-piece machining on 7th axis
#154051	[_MHSREFP[1]]	Base point of multi-piece machining on 1st axis
#154052	[_MHSREFP[2]]	Base point of multi-piece machining on 2nd axis
:	:	:
#154057	[_MHSREFP[7]]	Base point of multi-piece machining on 7th axis

- **Repositioning value #6270 (Attribute: R)**

Repositioning value of automatic repositioning function in executing block can be obtained by reading system variables #6270.

Variable number	Variable name	Description
#6270	[_MPATREP]	Repositioning value

- **Origin of local coordinate system #6271 to #6277, #154101 to #154107 (Attribute: R)**

The offset from origin of the workpiece coordinate system to origin of local coordinate system can be obtained by reading system variables #6271 to #6277 and #154101 to #154107.

Variable number	Variable name	Description
#6271	[_MLOCPOS[1]]	Origin of local coordinate system on 1st axis
#6272	[_MLOCPOS[2]]	Origin of local coordinate system on 2nd axis
:	:	:
#6277	[_MLOCPOS[7]]	Origin of local coordinate system on 7th axis
#154101	[_MLOCPOS[1]]	Origin of local coordinate system on 1st axis
#154102	[_MLOCPOS[2]]	Origin of local coordinate system on 2nd axis
:	:	:
#154107	[_MLOCPOS[7]]	Origin of local coordinate system on 7th axis

17.3 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 17.3 (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.)
<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, "ARGUMENT VALUE OUT OF RANGE" 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, “ARGUMENT VALUE OUT OF RANGE” is issued.
 - A constant can be used instead of the #j variable.
- **ARCTAN #i = ATAN[#j]/[#k]; (two arguments)**
- $\text{ATAN}[\#j, \#k]$ is equivalent to $\text{ATAN}[\#j]/[\#k]$.
 - 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 = $\text{ATAN}[-1]/[-1]$; 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 = $\text{ATAN}[-1]/[-1]$; 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 ($-90^\circ \leq \text{ATAN}[\#j] \leq 90^\circ$). 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, $\text{ATAN}[\#j]/[\#k]$ is assumed.
Example:
#100 = $[\text{ATAN}[1]]/10$; : Divides ATAN with one argument by 10.
#100 = $\text{ATAN}[1]/[10]$; : Executes ATAN with two arguments.
#100 = $\text{ATAN}[1]/10$; : Assumes ATAN with two arguments, but issues an alarm PS1131, “MISSING OPEN BRACKET” 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, “ARGUMENT VALUE OUT OF RANGE” 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, “ARGUMENT VALUE OUT OF RANGE” 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

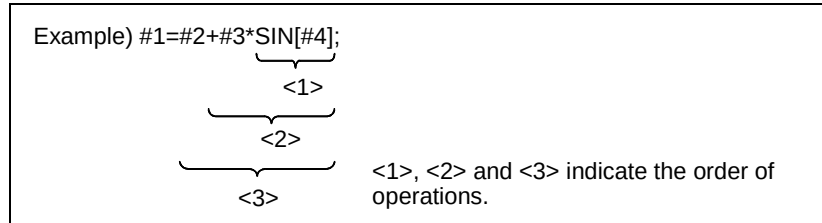
POW cannot be abbreviated.

- Priority of operations

<1> Functions

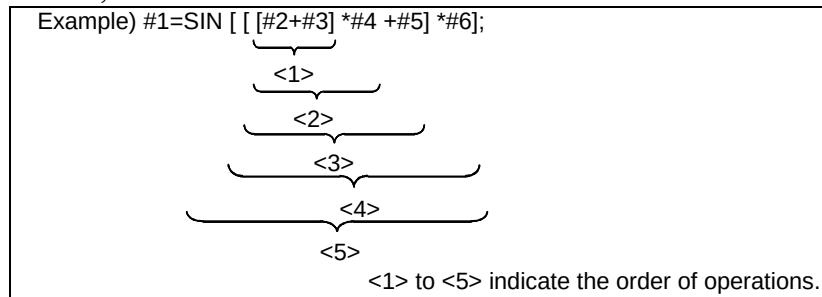
<2> Operations such as multiplication and division (*, /, AND)

<3> Operations such as addition and subtraction (+, -, OR, XOR)



- 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, "TOO MANY BRACKET NESTING" occurs.



Limitation

- **Caution concerning decreased precision**

When bit 0 (FOC) 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 17.3 (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(*1) $\left \frac{\varepsilon}{a} \right $
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}	MIN $\left \frac{\varepsilon}{b} \right $ ^(*2) $\left \frac{\varepsilon}{c} \right $
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error(*3) $\left \varepsilon \right $ 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] LT 0.001].

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, "ZERO DIVIDE" occurs.

17.4 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 #1, set the axis number of an axis type parameter in the range of 1 to 7. If an axis type parameter is to be read but #1 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) = 10010001
 #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=10010001
 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. 0012 (third axis) = 01010000
 #2=3404 ; Parameter number setting
 #3=2 ; Bit number setting
 If reading data with all bits
 #1=PRM[#2] ; #1=1010000
 If reading data with a specified bit
 #1=PRM[#2, #3] ; #1=0

17.5 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.
- 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

- NC statements that include a subprogram call command (such as subprogram calls by M98 or other M codes, or by T codes) and not include other command addresses except an O, N, P, or L address have the same property as macro statements.
- NC statements that include M99 and those do not include other command addresses except an O, N, P, or L address have the same property as macro statements.

17.6 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 ...)

17.6.1 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, "SEQUENCE NUMBER OUT OF RANGE" 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 destination of GOTOn, that is, the block with sequence number n, the sequence number must appear at the beginning of the block. If the sequence number is not at the beginning of the block, a branch cannot be made.

17.6.2 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.

As a "sequence number stored at previous execution of the corresponding block", a unique sequence number within the same program or the sequence number of a subprogram call with which the block was executed is stored.

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
<pre> : N10; : GOTO10; : N10; </pre> ← A branch to N10 before the GOTO statement occurs.	<pre> : N10; : GOTO10; : N10; </pre> ← A branch to N10 after the GOTO statement occurs.

Even if bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1 and the GOTO statement is executed, the branch to a block before GOTO statement may not occur, when the destination sequence number is not contained in the data stored at previous block execution. 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

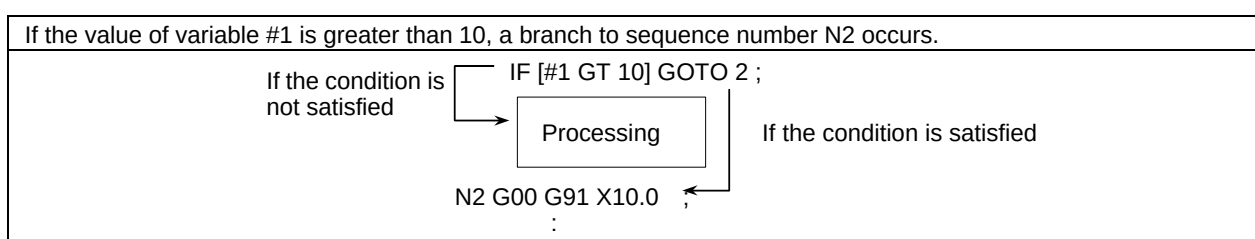
According to the restrictions on the GOTO statement, no branch to a sequence number within a DO-END loop cannot be made. If a program in which a branch to a sequence number within a loop occurs is executed, operation may differ depending on whether the GOTO statement using stored sequence numbers is used.

17.6.3 Conditional Branch (IF Statement)

Specify a <conditional expression> after IF.

IF[<conditional expression>]GOTO n

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[<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 Fig. 17.6.3 (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.

Fig. 17.6.3 (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(≤)

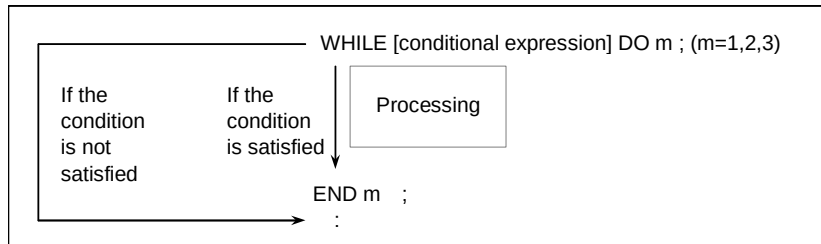
Sample program

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

17.6.4 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.

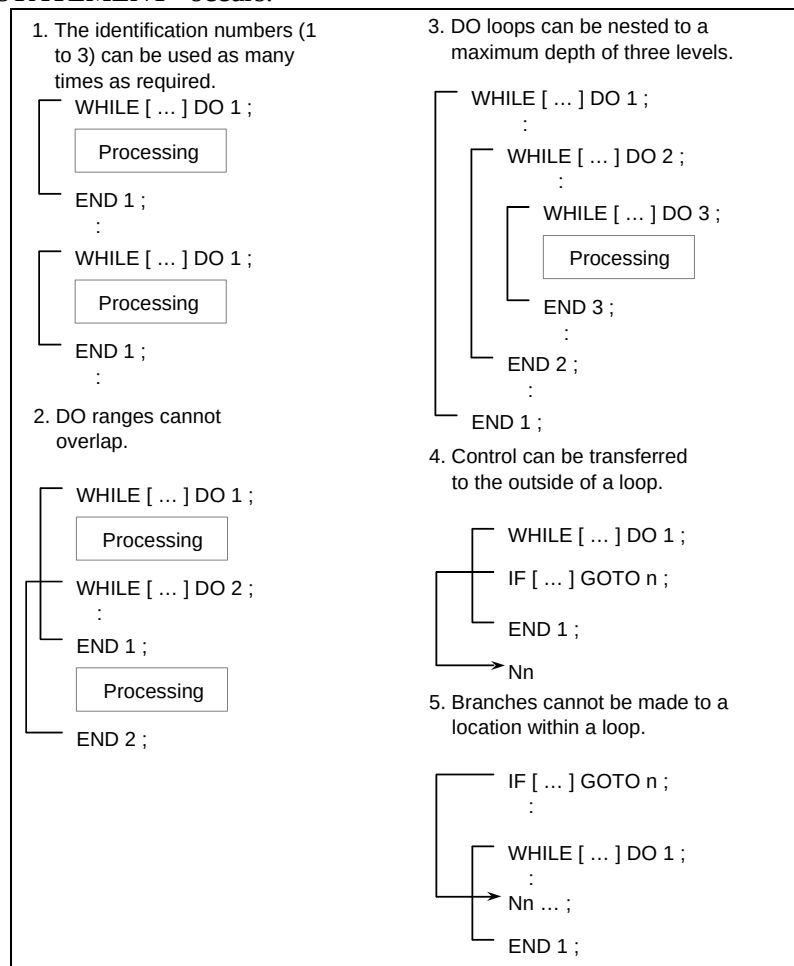


Explanation

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, "ILLEGAL LOOP NUMBER" 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, "MISSING END STATEMENT" occurs.



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

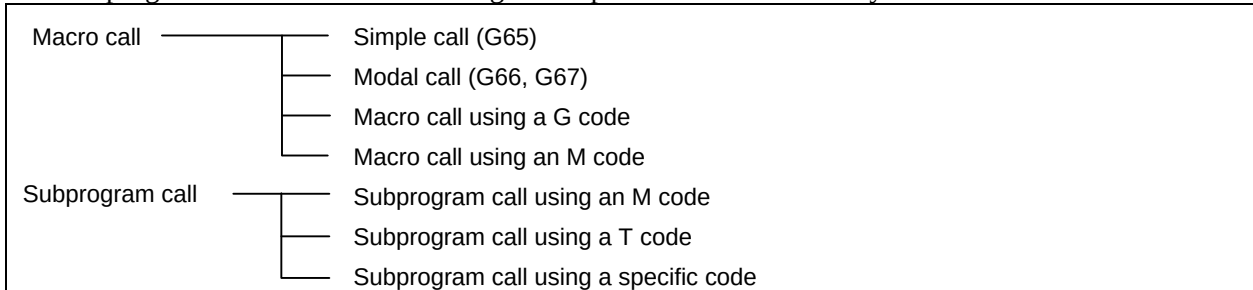
The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

17.7 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.



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 (for example M98, Mmm, or Ttt) as described below.

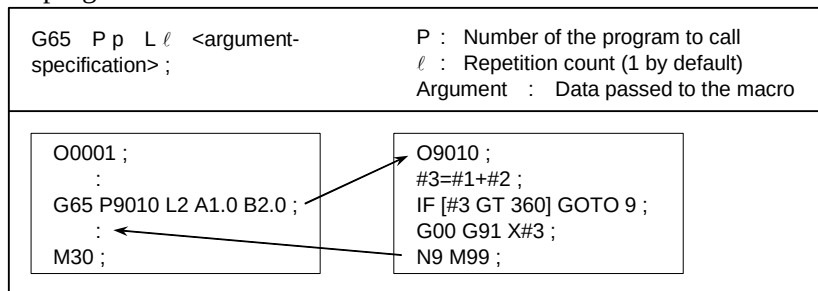
- With a macro call, an argument (data passed to a macro) can be specified. A subprogram call does not have this capability.
- If a macro call block contains another NC command (such as G01 X100.0 G65 Pp), an alarm PS0127, "DUPLICATE NC,MACRO STATEMENT" occurs.
- If a subprogram call block contains another 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, "Simple Call (G65)".)

17.7.1 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.



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
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
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 3-dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I ₁	#4
J ₁	#5
K ₁	#6
I ₂	#7
J ₂	#8
K ₂	#9
I ₃	#10
J ₃	#11

Address	Variable number
IK ₃	#12
I ₄	#13
J ₄	#14
K ₄	#15
I ₅	#16
J ₅	#17
K ₅	#18
I ₆	#19
J ₆	#20
K ₆	#21
I ₇	#22

Address	Variable number
J ₇	#23
K ₇	#24
I ₈	#25
J ₈	#26
K ₈	#27
I ₉	#28
J ₉	#29
K ₉	#30
I ₁₀	#31
J ₁₀	#32
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 II cannot be used.

Limitation

- **Format**

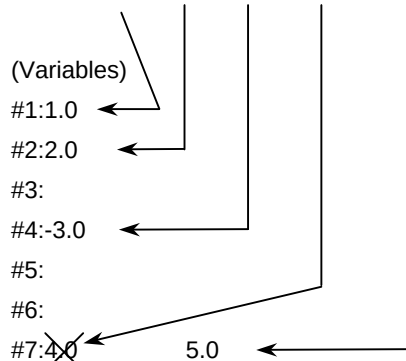
G65 must be specified before any argument.

- **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 ;



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.

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 (G94 mode)	0	2

NOTE

- When V is used in a call using a specific code, the number of decimal places is determined according to the setting for the reference axis.
- 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 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

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.

- 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.)

Increment system for the reference axis	AUP (No.3450#0) = 0	AUP(3450#0) = 1			
		AUX (No.3405#0) = 0		AUX (No.3405#0) = 1	
		Metric	Inch	Metric	Inch
IS-A	0	2		2	3
IS-B		3		3	4

- 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.

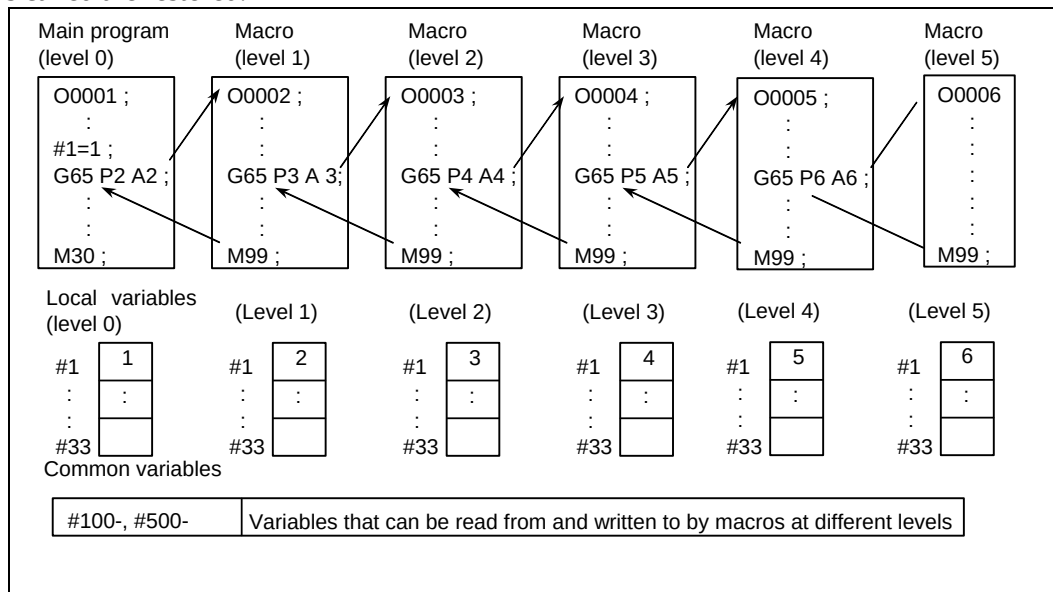
- Call nesting

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.

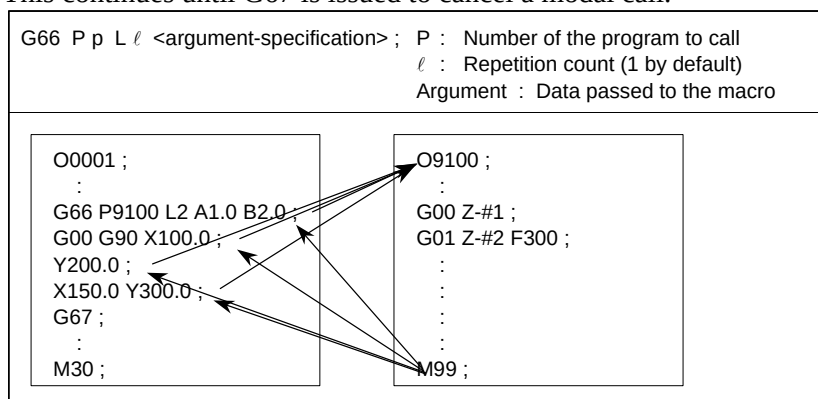
- Local variable levels

- 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.



17.7.2 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.



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 or subprogram 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.

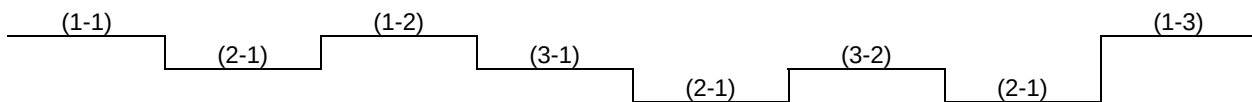
[Example]

```

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

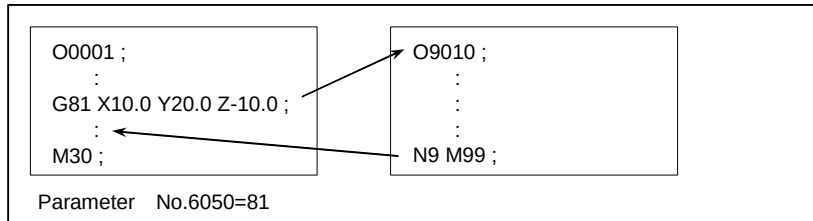
- G66 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66 mode, an alarm PS1100, "CANCEL WITHOUT MODAL CALL" occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- In a G66 block, no macros can 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 an 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.

17.7.3 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).



Explanation

By setting a G code number from -9999 to 9999 used to call a custom macro program (O9010 to O9019) in the corresponding parameters Nos.6050 to 6059, the macro program can be called in the same way as with G65.

When a negative G code is set, a modal call (G66 mode) is performed.

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 999999999 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, G66, or G66.1 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.

17.7.4 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.

When a negative G code is set in parameter No. 6038, modal calls are performed.

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

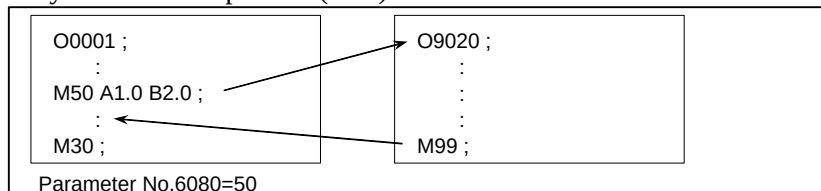
Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6038 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

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> $(\text{No. 6039} + \text{No. 6040} - 1) > 9999$
- 2 Simple and modal calls cannot be mixed in the specification.
- 3 If the G code set in parameters 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 parameters Nos. 6050 to 6059 is called.

17.7.5 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 parameters 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, G66, and M98 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.

17.7.6 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 9000, parameter No. 6048 to 4000, and parameter No. 6049 to 100.

M90000000 → O4000

M90000001 → O4001

M90000002 → O4002

:

M90000099 → O4099

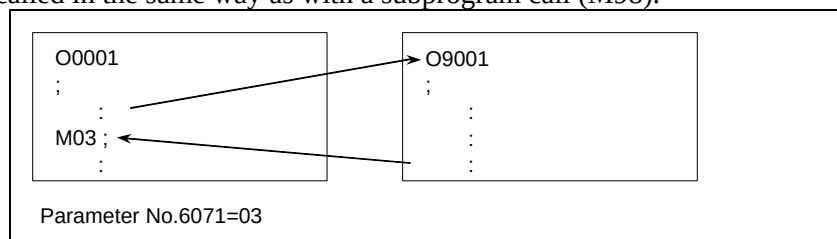
Custom macro calls (simple calls) for 100 combinations are defined as shown above.

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> $(\text{No. 6048} + \text{No. 6049} - 1) > 9999$
- 2 If the M code set in parameters 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 parameters Nos. 6080 to 6089 is called.

17.7.7 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, G66, and M98 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.

17.7.8 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 subprograms 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 numeric value set in parameter No. 6045 indicates the starting subprogram number. To disable this type of call, set 0 in parameter No. 6046.

[Example]

Set parameter No. 6044 to 80000000, parameter No. 6045 to 30000000, and parameter No. 6046 to 100.

M80000000 → O3000

M80000001 → O3001

M80000002 → O3002

⋮

M80000099 → O3099

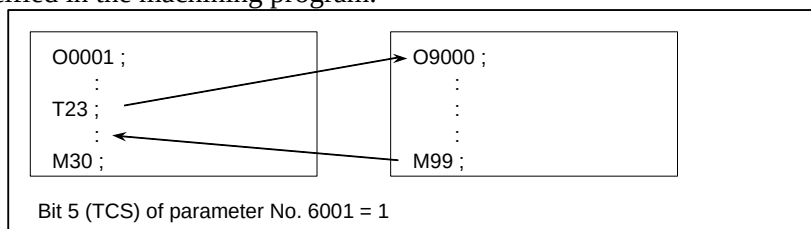
Subprogram calls for 100 combinations are defined as shown above.

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> $(\text{No. 6045} + \text{No. 6046} - 1) > 9999$
- 2 If the M code, set in parameters Nos. 6071 to 6079, to call the corresponding subprogram is within the range for M code multiple definitions of calling subprograms, the subprogram corresponding to the M code set in parameters Nos. 6071 to 6079 is called.

17.7.9 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

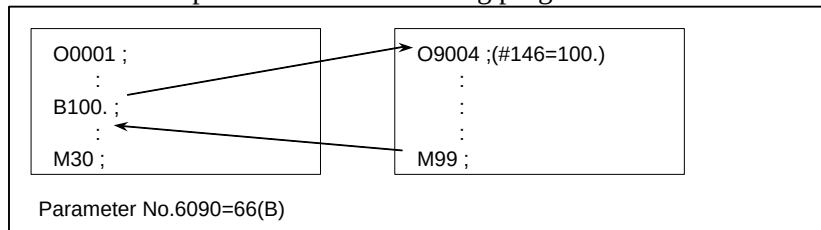
Argument specification is not allowed.

Limitation

- To call another program in a program called using a T code, only G65, G66, and M98 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.

17.7.10 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.

Address	Parameter setting	Address	Parameter setting
A	65	P	80
B	66	Q	81
D	68	R	82
F	70	S	83
H	72	T	84
I	73	V	86
J	74	X	88
K	75	Y	89
L	76	Z	90
M	77		

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	O9004	#146
6091	O9005	#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 code, only G65, G66, and M98 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 code.

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; ..... Starts tool action.
M99;

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 tool.
M99;

```

17.8 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, in the cutter compensation mode (G41 or G42), the CNC prereads the NC statements in at least three blocks ahead to calculate intersections.

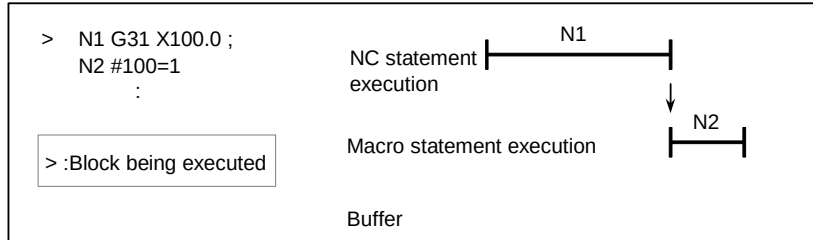
Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.

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 3421 to 3432, and blocks containing prevention buffering G codes such as G33 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

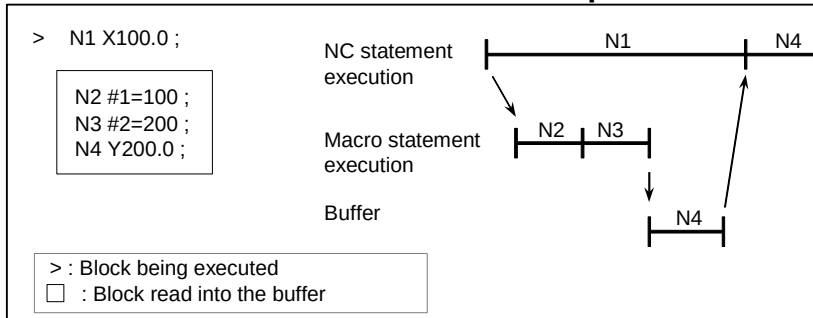
- When the next block is not buffered (M codes that are not buffered, G33, etc.)



⚠ CAUTION

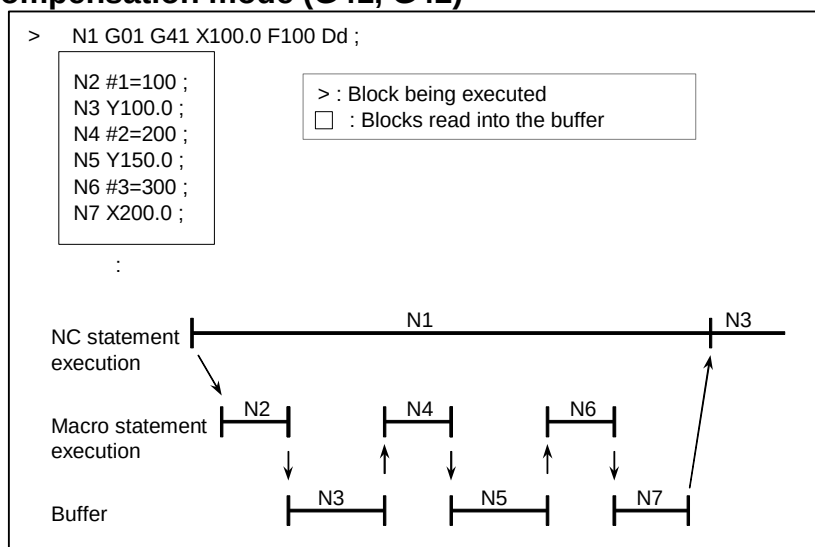
In case that you need to execute the macro statement after completing the block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, offset value, etc., it may differ 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)



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.

Diagnosis data

Diagnosis **1493**

Number of macro statement blocks executed by custom and execution macros

[Data type] 2-word

[Unit of data] Block

The number of macro statement blocks executed by custom and execution macros per 1024 ms is displayed.

This data provides an indication of the actual macro statement processing speed.

17.9 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.

17.10 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	Meaning	Code
*	0AAh	?	03Fh
=	0BDh	@	0C0h
#	0A3h	&	0A6h
[	0DBh	—	05Fh
]	0DDh	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	Meaning	Code
*	Code set in parameter No. 6010	?	Code set in parameter No. 6015
=	Code set in parameter No. 6011	@	Code set in parameter No. 6016
#	Code set in parameter No. 6012	&	Code set in parameter No. 6017
[	Code set in parameter No. 6013	_	Code set in parameter No. 6018
]	Code set in parameter No. 6014		

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 parameters 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, PRM

System variable (constant) names and registered common variable names are also used as reserved words.

17.11 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

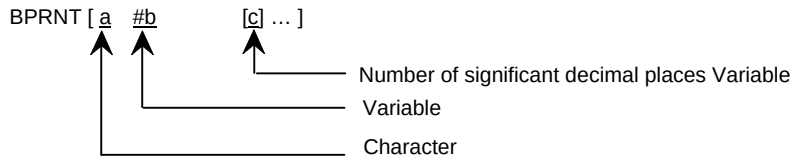
When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

- 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.



- (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 output 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

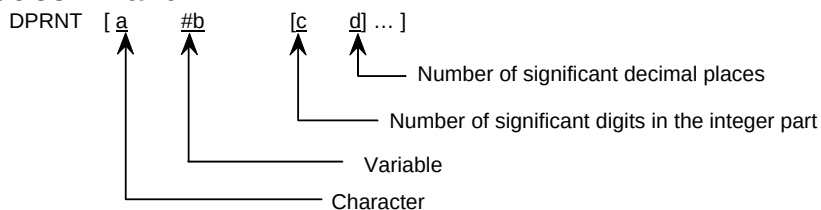
#10=12.34

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



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.

Never specify the output device FANUC Cassette or Floppy for output.

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.

NOTE

- 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. Therefore, when a reset operation is performed by a code such as M30 at the end of a program that performs data output, specify a close command at the end of the program so that processing such as M30 is not performed until all data is output.

17.12 RESTRICTIONS

- Sequence number search

A custom macro program cannot be searched for a sequence number.

- 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 (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.

		Bit 5 (SBM) of parameter No. 6000	
		0	1
Bit 7 (SBV) of parameter No. 6000	0	Not stopped when the single block mode is on.	Can be stopped in the single block mode. (Variable #3003 cannot be used to disable single block stop. Single block stop is always enabled.)
	1	Can be stopped in the single block mode. (Variable #3003 can be used to enable or disable single block stop.)	

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. They can be prevented from clearing by setting bit 6 (CCV) of parameter No.6001 to 1. System variables #100 to #199 are not 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.

- 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>

+0.000000000001 to +999999999999

-999999999999 to -0.000000000001

The number of significant digits is 12 (decimal).

If this range is exceeded, an alarm PS0004, "INVALID BREAK POINT OF WORDS" occurs.

17.13 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.

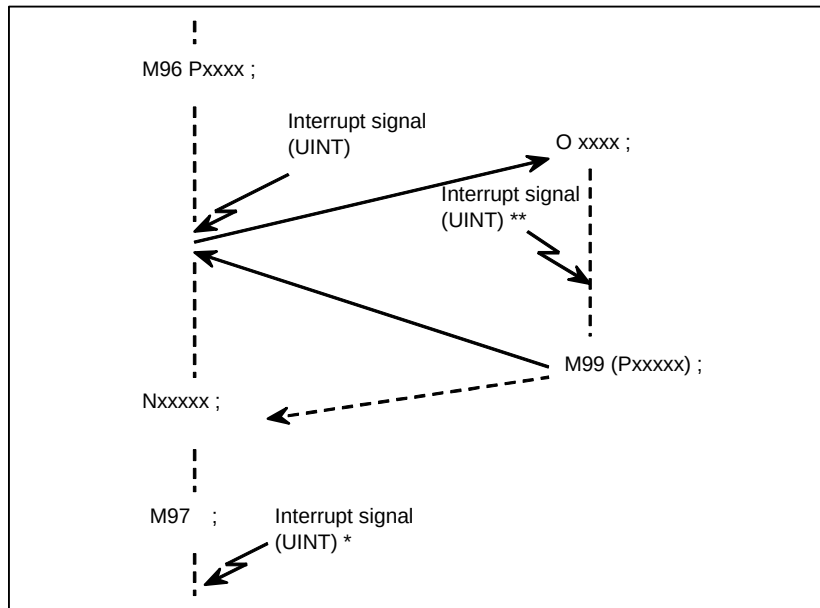


Fig 17.13 (a) Interruption type custom macro function

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 is input during execution of the interrupt program or after M97 (UINT, marked with asterisks (*) or (**)) in Fig 17.13 (a)), it is ignored.

17.13.1 Specification Method

Explanation

- Interrupt conditions

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

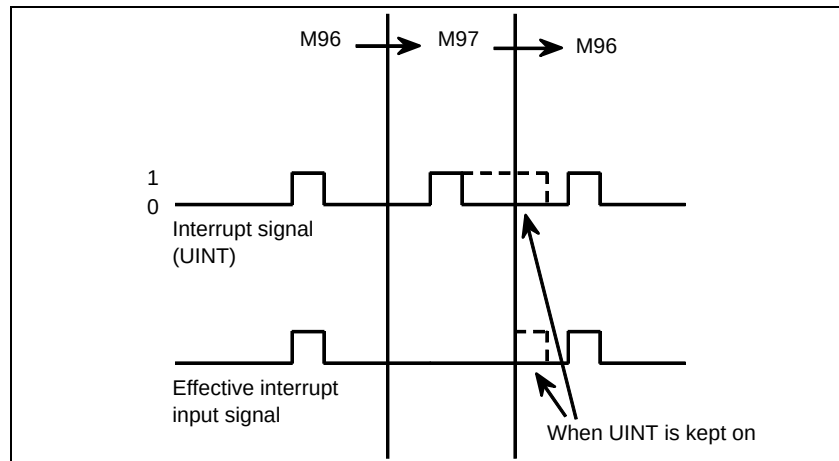
- When memory operation, DNC operation, or MDI operation is selected
- When STL (start lamp) is on
- When a custom macro interrupt is not currently being processed

A 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 the Subsection II-17.13.2, "Details of Functions".

17.13.2 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 macro M code 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)

```
O0013;
#101=#5041;
#102=#5042;
#103=#5043;
M99;
```

(Program containing no NC statement)

```
O0013#101=#5041;
#102=#5042;
#103=#5043;
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.

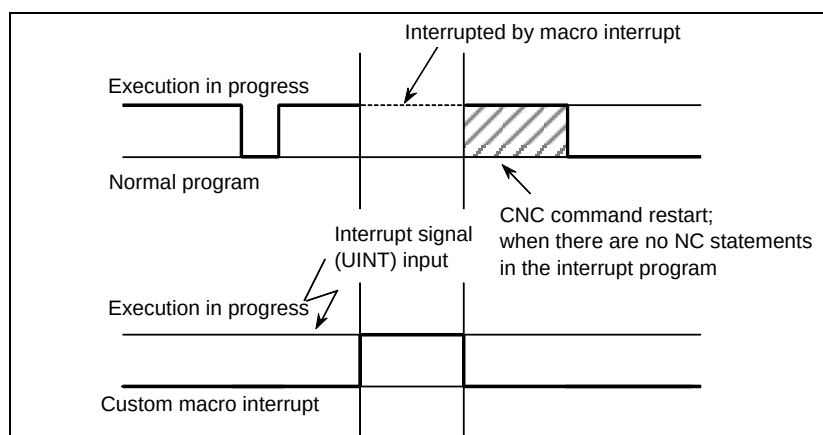


Fig. 17.13 (b) Custom macro interrupt and NC command (type I)

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 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.

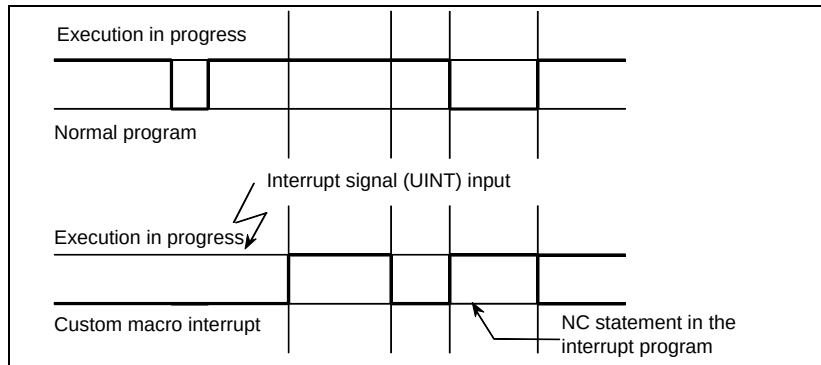


Fig. 17.13 (c) Custom macro interrupt and NC command (type II)

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 radius · 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
- <4> Automatic tool length measurement
- <5> Optional chamfering/corner R
- <6> Exponential interpolation
- <7> Normal direction control
- <8> Cutting point interpolation for cylindrical interpolation

- 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 becomes 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. 17.13 (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.

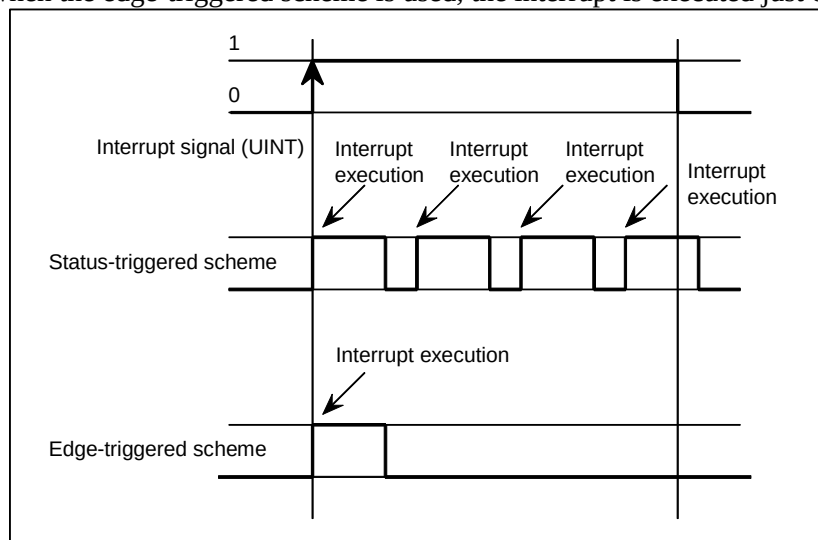


Fig. 17.13 (d) Custom macro interrupt signal

- 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

When an M99 block consists only of address O, N, P, L, or M, this block is regarded as belonging to the previous block in the program. 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 Pxxxxx, 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.

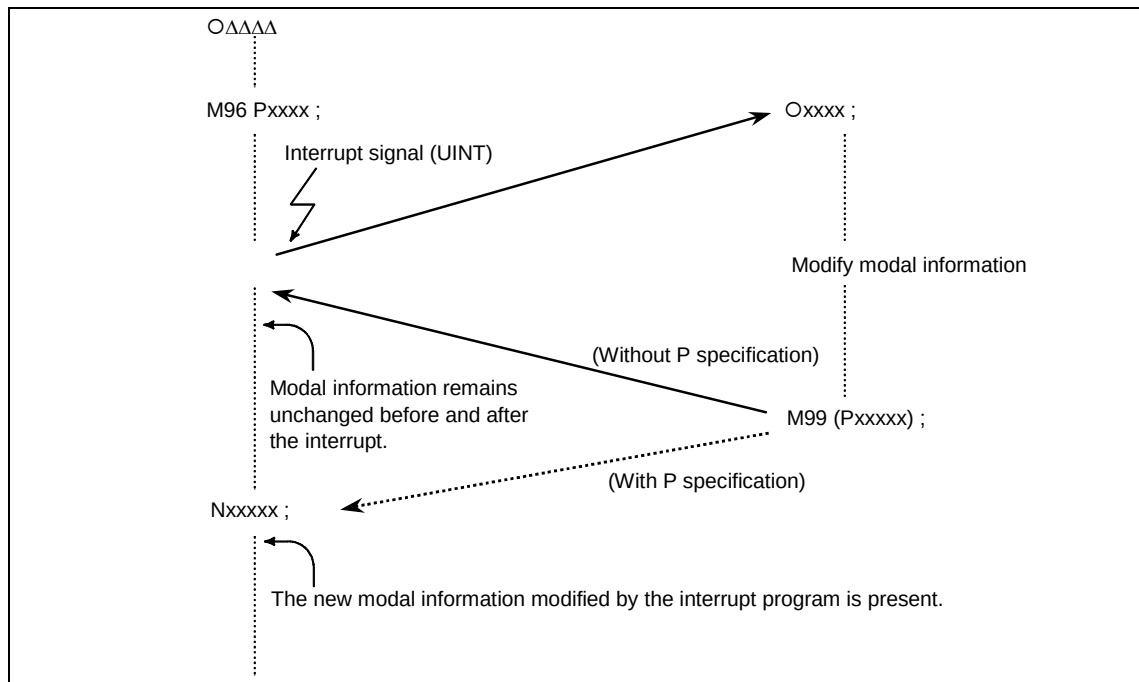


Fig. 17.13 (e) Custom macro interrupt and modal information

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 Pxxxxx

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

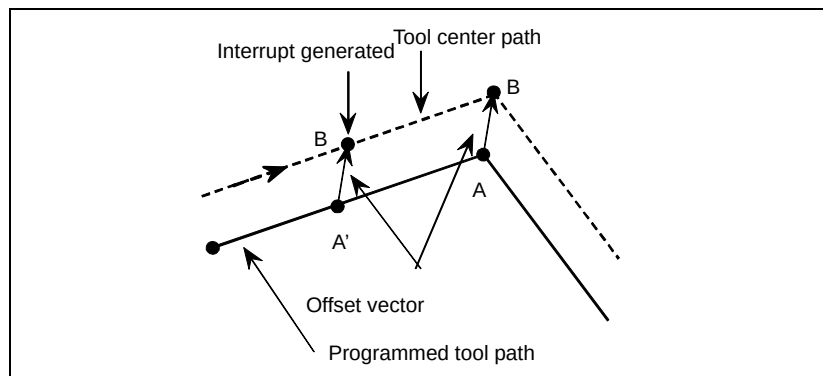
The old modal information which was valid in the interrupted block can be read using custom macro system variables #4401 to #4520.

System variable	Modal information which was valid when a custom macro interrupt was generated
#4401 :	G code (group 01) :
#4418	G code (group 18)
#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

- 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 Pxxxx ;, the modal call in the interrupt program remains valid.

NOTE

Alarm PS1101, "ILLEGAL CNC STATEMENT IRT." occurs in the following cases:

- <1>An interrupt is generated in the coordinate system rotation (G84) mode and another G84 is specified in the interrupt program.
- <2>An interrupt is generated in the scaling (G51) mode and another G51 is specified in the interrupt program.

18 PROGRAMMABLE PARAMETER INPUT (G10)

18.1 PROGRAMMABLE PARAMETER INPUT

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

G10L52 ; Parameter entry mode setting

N_(Q_)R_ ; For parameters other than the axis type

N_P_(Q_)R_ ; For axis 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 is specified)

NOTE

G10L52 cannot be used to enter pitch error compensation data.

- Pitch error compensation data entry mode

G10L50 ; Pitch error compensation data entry mode setting

N_R_ ; Pitch error compensation data entry

N_P_R_ ; For axis type parameters

:

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.

A custom macro variable can be used as the value of P.

**WARNING**

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.

**CAUTION**

Compatibility with the Series 0i-PC:

This model has parameters that are not compatible with the Series 0i-PC. So, before using this function, make a check according to the Parameter Manual (B64310EN, B-64560EN) of this model.

NOTE

Other NC statements cannot be specified while in parameter input mode.

Example

- Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 0)

G10L52 ;	Parameter entry mode
N3404 R 00000100 ;	SBP setting
G11 ;	Cancel parameter entry mode

- Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 1)

G10L52 ;	Parameter entry mode
N3404 Q2 R 1 ;	SBP setting
G11 ;	Cancel parameter entry mode

- 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)

G10L52 ;	Parameter entry mode
N1322P3R4500 ;	Change the value for the Z-axis to 4.500
N1322PP4R12000 ;	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.

G10L50 ;	Pitch error compensation data entry mode
N10010R1 ;	Change the compensation point number from 10 to 1
N10020R5 ;	Change the compensation point number from 20 to 5
G11 ;	Pitch error compensation data entry mode

18.2 TOOL DATA INPUT

Format

G10L30 ; Tool data entry mode setting

N_P_R_ ; Tool data entry

:

:

G11 ; Tool data entry mode cancel

N_ : Tool data No. or multiple tool data No. +200

R_ : Tool data setting value

P_ : Kind of tool data

▪ When N code is specified as tool data

P01 : Tool data No. setting

P02 : Turret position for indexing turret setting

P03 : X-axis tool offset setting

P04 : Y-axis tool offset setting

P05 : Tool change No. setting

P06 : Punch count setting

P07 : Tool life setting

P08 : Tool figure setting for graphic operation

P09 : X dimension of a tool setting for graphic operation

P10 : Y dimension of a tool setting for graphic operation

P11 : Tool angle setting for graphic operation

▪ When N code is specified as multiple tool data

P01 : Sub-tool No. setting

P02 : Angle for indexing turret of multiple tool setting

P03 : Y-axis tool offset setting

P04 : Multiple tool figure setting for graphic operation

P05 : X dimension of a multiple tool setting for graphic operation

P06 : Y dimension of a multiple tool setting for graphic operation

P07 : Multiple tool angle setting for graphic operation

Example

Change the value of tool number for the tool data No. 10.

G10L30 ; ← Tool data entry mode
N10P01R333 ; ← Modify tool number
G11 ; ← Tool data entry mode cancel

19 HIGH-SPEED CUTTING FUNCTIONS

Chapter 19, "HIGH-SPEED CUTTING FUNCTIONS", consists of the following sections:

19.1 AI ADVANCED PREVIEW CONTROL (G05.1) 300

19.1 AI ADVANCED PREVIEW CONTROL (G05.1)

Overview

AI advanced preview control is intended for high-speed, high-precision machining. The use of this function 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.

On the Series 0i-PD, the AI advanced preview control can be used during laser mode only.
The following table shows the functions included in the AI advanced preview control function.

	AI APC
Look-ahead block count	20
Look-ahead linear acceleration/deceleration before interpolation	○
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	○

Format

• AI advanced preview control

G05.1 Q_ ;

Q1 : AI advanced preview control mode on

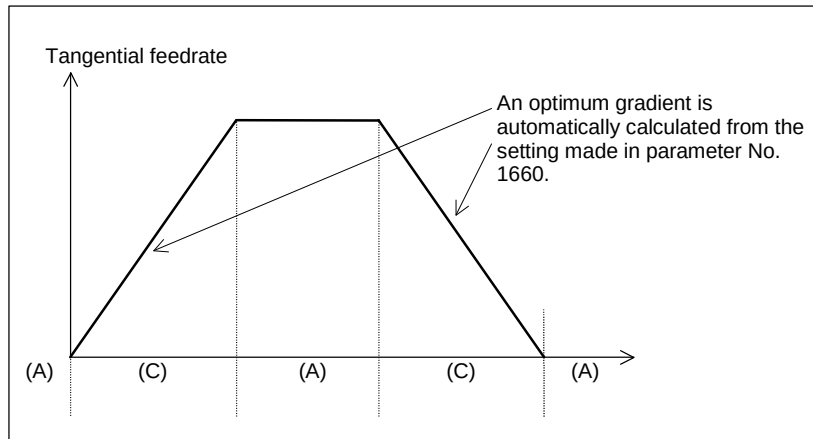
Q0 : AI advanced preview control mode off

NOTE

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

- Setting an acceleration

A permissible acceleration for the linear acceleration/deceleration of each axis is set in parameter No. 1660. 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.



- 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

Program:

N1 G01 G91 X20. F6000 ; (Move on the X-axis.)

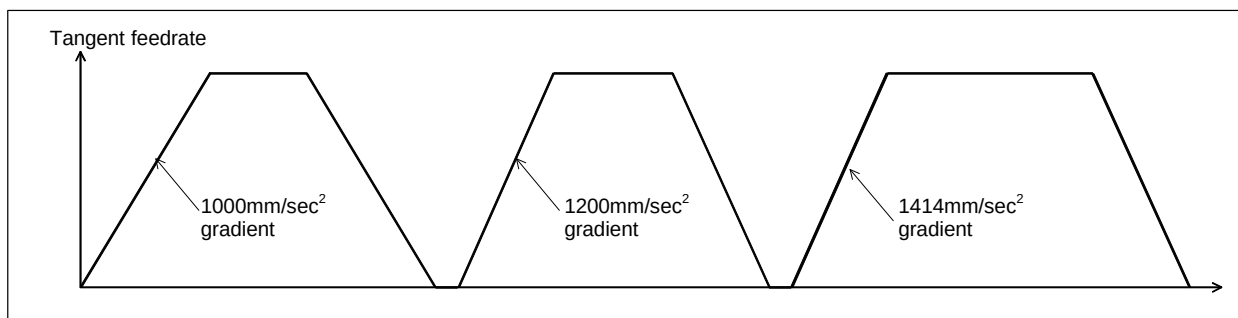
G04 X0.01 ;

N2 Y20. ; (Move on the Y-axis.)

G04 X0.01 ;

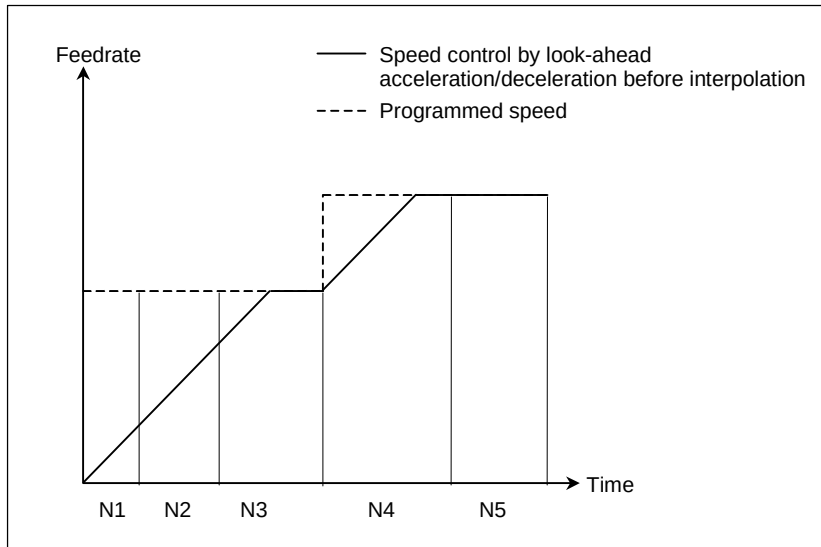
N3 X20. Y20. ; (Move in the XY direction (at 45 degrees).)

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

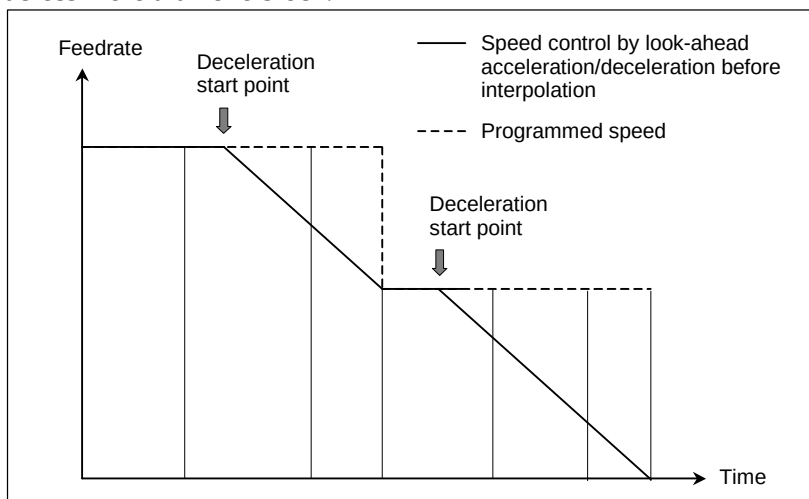
Acceleration is performed so that the feedrate programmed for a block is attained at the beginning of the block. When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, acceleration can be performed across more than one block.



- 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.

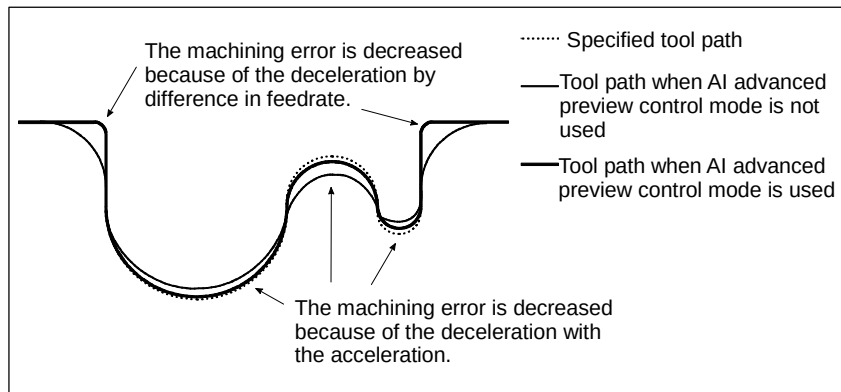
- Automatic feedrate control function

During the AI advanced preview control 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

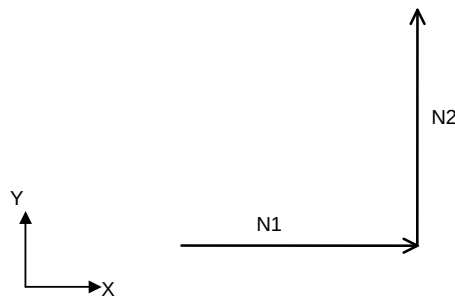


- Speed control based on the feedrate difference on each axis at a corner

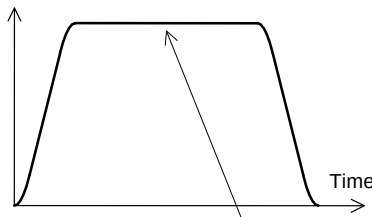
By using the speed control based on the feedrate difference on each axis at a corner, if a feedrate change occurs on each axis at a corner, the feedrate is determined so that any feedrate difference exceeding the permissible feedrate difference on that axis that has been set for parameter No. 1783 does not occur, and deceleration is automatically performed.

(Example)

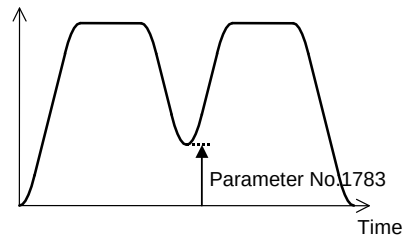
Program
N1 G01 G91 X100. F5000
N2 Y100.



Tangent feedrate

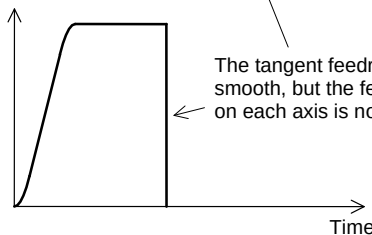


Tangent feedrate

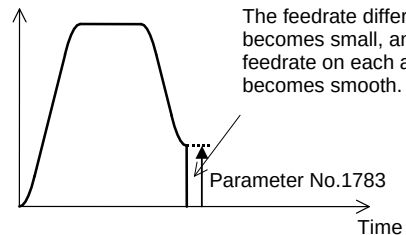


The deceleration based on the feedrate difference is used.

X-axis feedrate

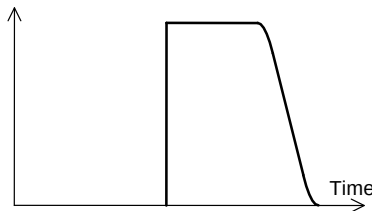


X-axis feedrate

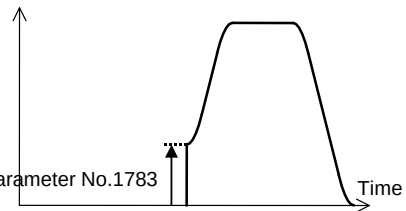


The feedrate difference becomes small, and the feedrate on each axis becomes smooth.

Y-axis feedrate



Y-axis feedrate



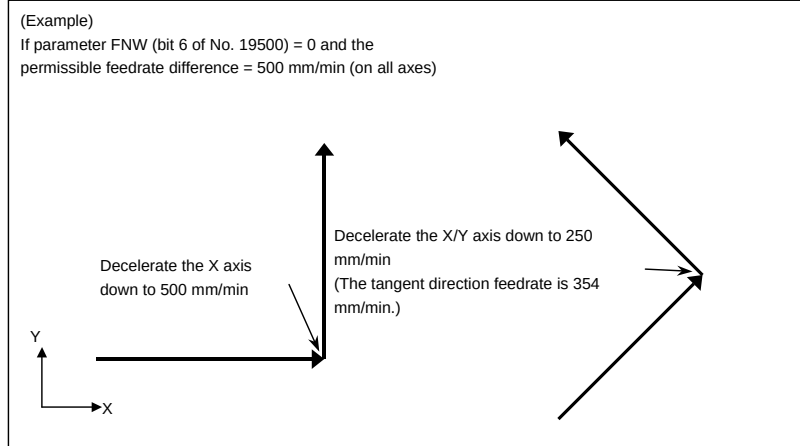
When speed control based on the feedrate difference is invalid

When speed control based on the feedrate difference is valid

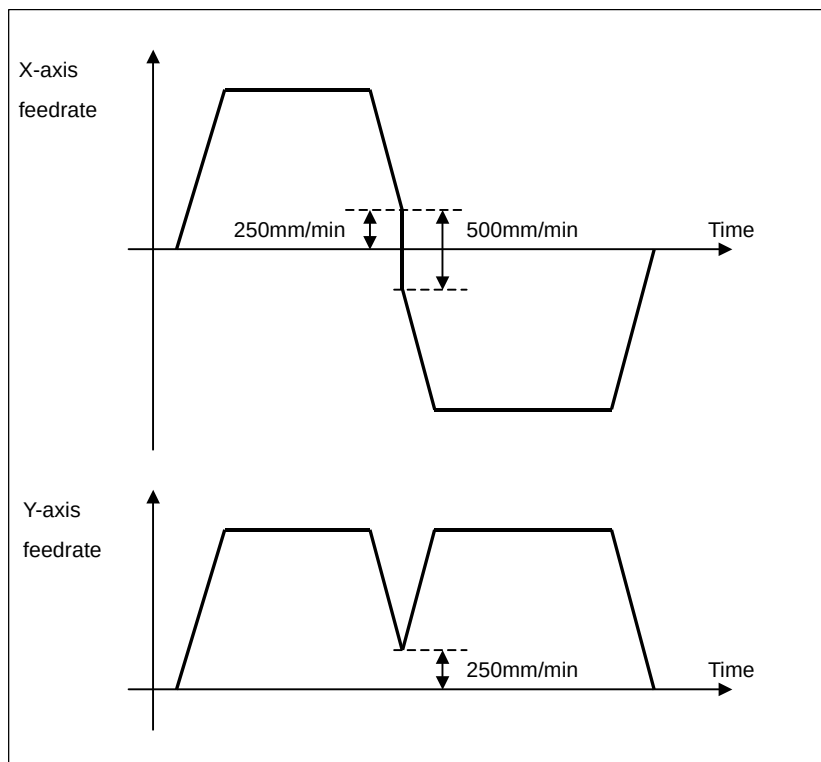
The method of deceleration based on the feedrate difference differs depending on the setting made for parameter FNW (bit 6 of No. 19500).

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.

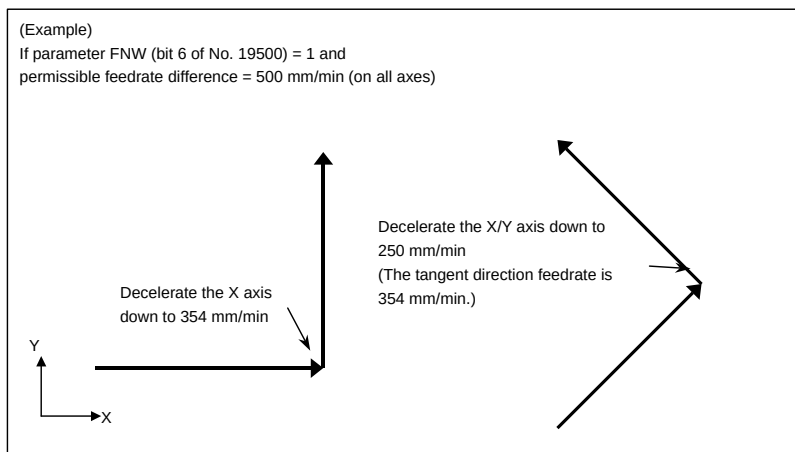
In this case, the deceleration feedrate differs if the travel direction differs, even if the shape is the same.



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.

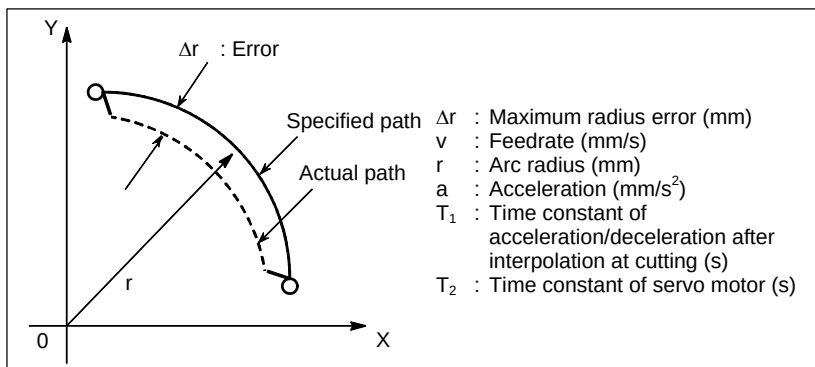


If "1" is set, the feedrate is determined not only with the condition that the permissible feedrate difference and permissible acceleration on each axis are not exceeded, but also that the deceleration feedrate is constant regardless of the travel direction if the shape is the same. If 1 is set for this 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.



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

In actual machining, the permissible error Δr is given as the machining accuracy. Therefore, the permissible acceleration a (mm/sec²) 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:

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

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.

- Speed control with the acceleration on each axis

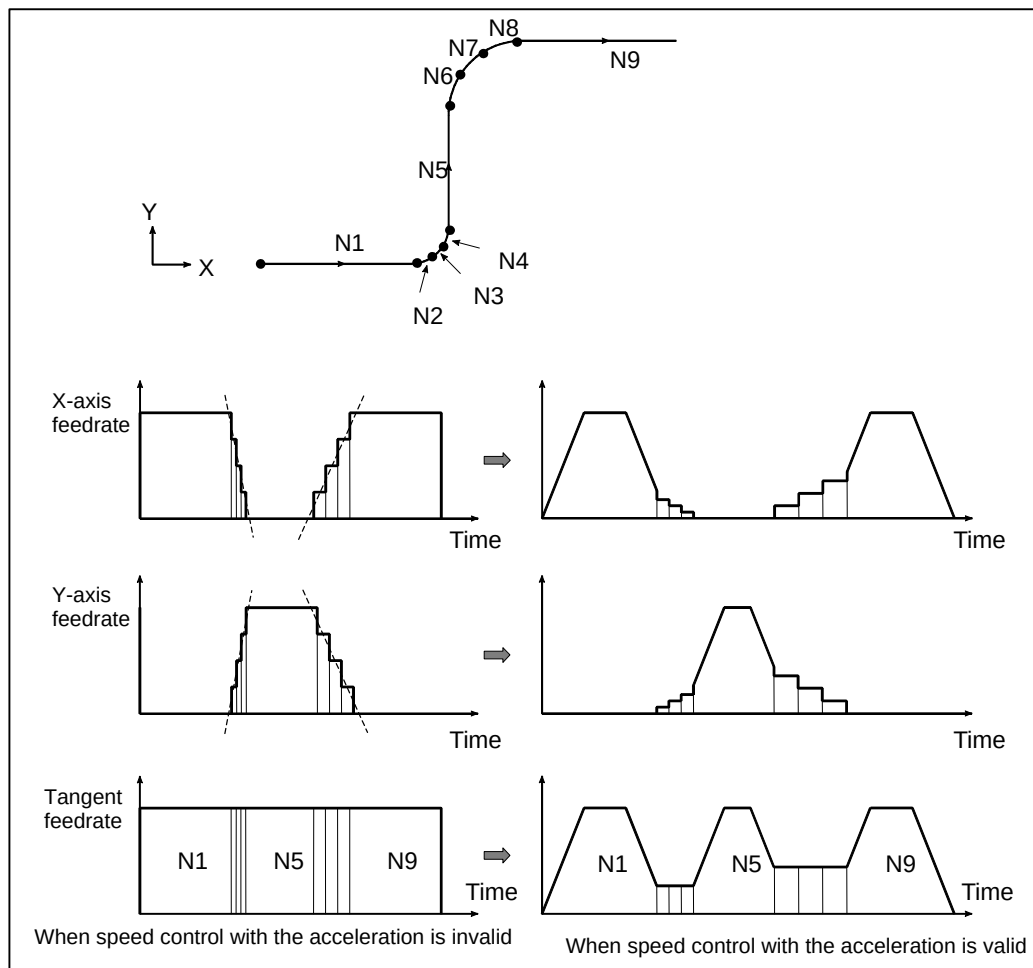
When consecutive small lines are used to form a curve, as in the example shown in the figure below, the feedrate differences on each axis at the individual corners are not very large. Thus, deceleration with the feedrate differences is not effective. Consecutive small feedrate differences, however, cause 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.

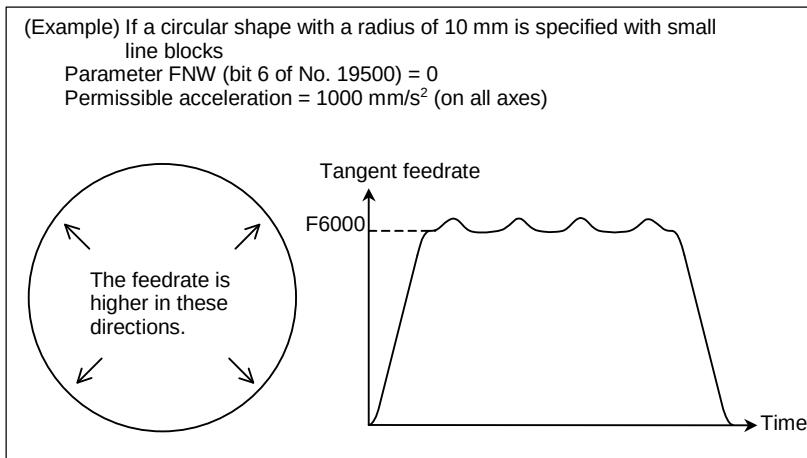
Depending on the specified figure, a very low deceleration feedrate may be calculated. In such a case, the lower feedrate limit can be set in parameter No. 1738 to prevent the feedrate from being decreased too much.

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.



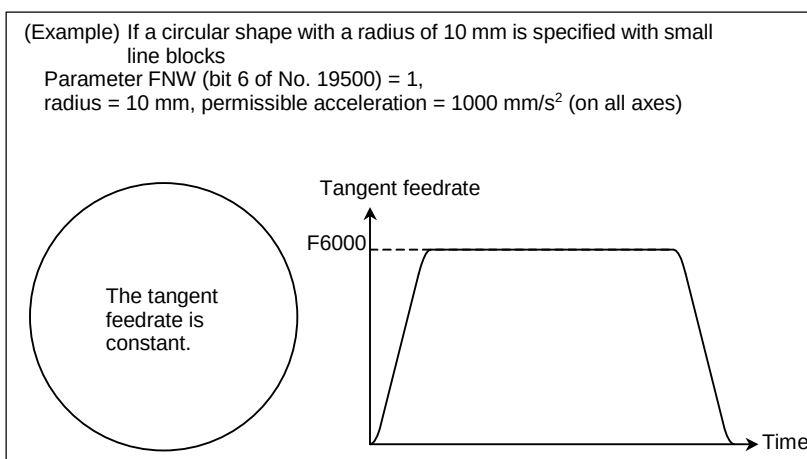
The method of determining the feedrate with the acceleration differs depending on the setting of parameter FNW (bit 6 of No. 19500).

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.



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.



NOTE

In circular interpolation, the tangent feedrate is constant regardless of the setting of the parameter.

- Another example of determining the feedrate

If a specified feedrate exceeds the upper feedrate limit of AI advanced preview control (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 AI advanced preview control mode

If any of the commands listed below is executed during AI advanced preview control mode, it is temporarily canceled. Note that the AI advanced preview control mode resumes as soon as it becomes available.

Function name	G code
Positioning (rapid traverse) ^(NOTE)	G00
When no move command is specified	—
One-shot G code other than those shown at right ^(NOTE)	G09

NOTE

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 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 and G53 commands are the interpolation type).
- (5) Bit 1 (AMP) of parameter No. 11240 is set to 1 (acceleration/deceleration before interpolation is valid for the G28 and G53 commands in the AI advanced preview control mode).

- **Parameter list**
Positioning

Parameter	Parameter No.
Positioning type (non-linear (0)/interpolation (1))	1401#1 LRP
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/deceleration before interpolation

Parameter	Parameter No.
Maximum permissible acceleration of acceleration/deceleration before interpolation	1660

Acceleration/deceleration after interpolation

Parameter	Parameter No.
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.
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.
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

Speed control with the acceleration on each axis

Parameter	Parameter No.
Permissible acceleration for the deceleration function with the acceleration	1737
Lower-limit feedrate for the deceleration function with the acceleration	1738
Method of determining the feedrate based on the feedrate difference or based on the acceleration	19500#6 FNW

Others

Parameter	Parameter No.
Maximum cutting feedrate during the mode of acceleration/deceleration before interpolation	1432
Upper-limit feedrate for AI advanced preview control	8465
Upper-limit feedrate for AI advanced preview control (when only the rotation axis is specified)	8466

20 AXIS CONTROL FUNCTIONS

Chapter 20, "AXIS CONTROL FUNCTIONS", consists of the following sections:

20.1 AXIS SYNCHRONOUS CONTROL.....	311
20.2 ROTARY AXIS ROLL-OVER	320
20.3 TANDEM CONTROL	321
20.4 C AXIS CONTROL (DIE ANGLE INDEXING).....	321
20.5 RAM-AXIS CONTROL	326

20.1 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. Moreover, by using a feedback signal from each motor, a positional difference (synchronous error) between the two motors is detected. 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).

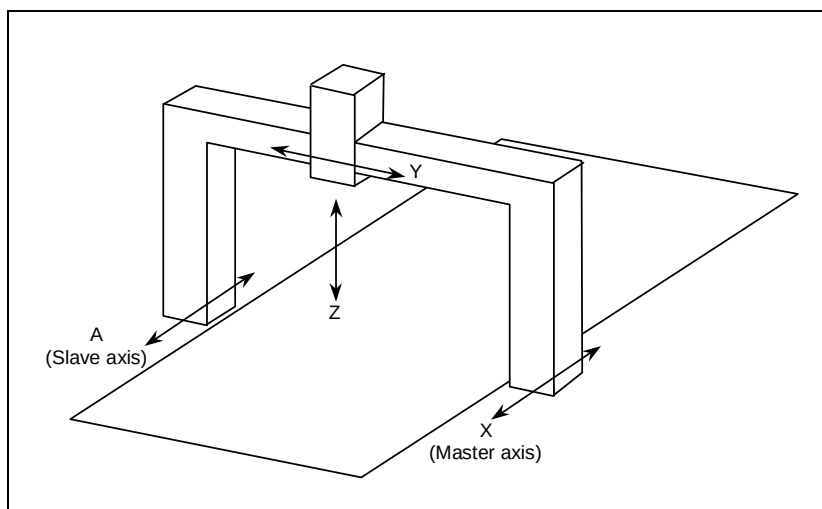


Fig. 20.1 (a) Example of machine with X and A being synchronous axes

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.

20.1.1 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 signals SYNCx for selecting the axis for axis synchronous control / signals SYNCJx for selecting the manual feed axis for axis synchronous control (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 mirror image signals M_{Ix} or mirror image check signals MM_{Ix}.

- 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.

20.1.2 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

To perform synchronous establishment, 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, "SYNC ALIGNMENT ERROR" 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.

- 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.

Until this synchronous establishment is completed, a synchronous error check is also made.

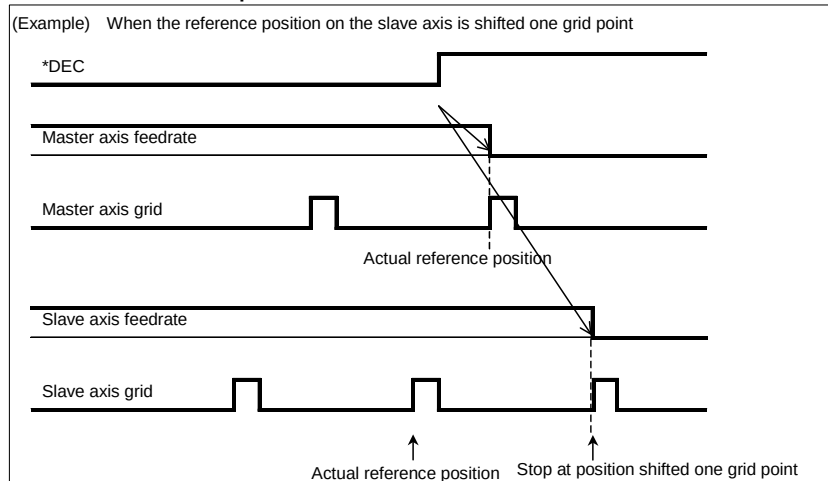
- Synchronous establishment based on manual reference position return operation

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 20.1.3, "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**

The 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.

20.1.3 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.

20.1.4 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

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 alarm SV0005, "SYNC EXCESS ERROR (MCN)" is issued, and the motor is stopped immediately.

A check can be made even in the emergency stop, servo off, and servo alarm states.

If a synchronous error check is made when the mode of operation is switched between synchronous operation and normal operation with an input signal, an error check is made even in normal 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.

- 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 alarm DS0001, "SYNC EXCESS ERROR (POS DEV)" is issued, and the axis synchronous control positional deviation error alarm signal SYNER is output.

The alarm DS0001 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.

20.1.5 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, synchronous error compensation and error check are not performed, so that an alarm DS0003, "SYNCHRONIZE ADJUST MODE" 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 alarm DS0003, "SYNCHRONIZE ADJUST MODE" is issued.

2. Reset the synchronous error excessive alarm.
In this state, error check is 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 is 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.

20.1.6 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, “SYNC TORQUE EXCESS” can be issued.

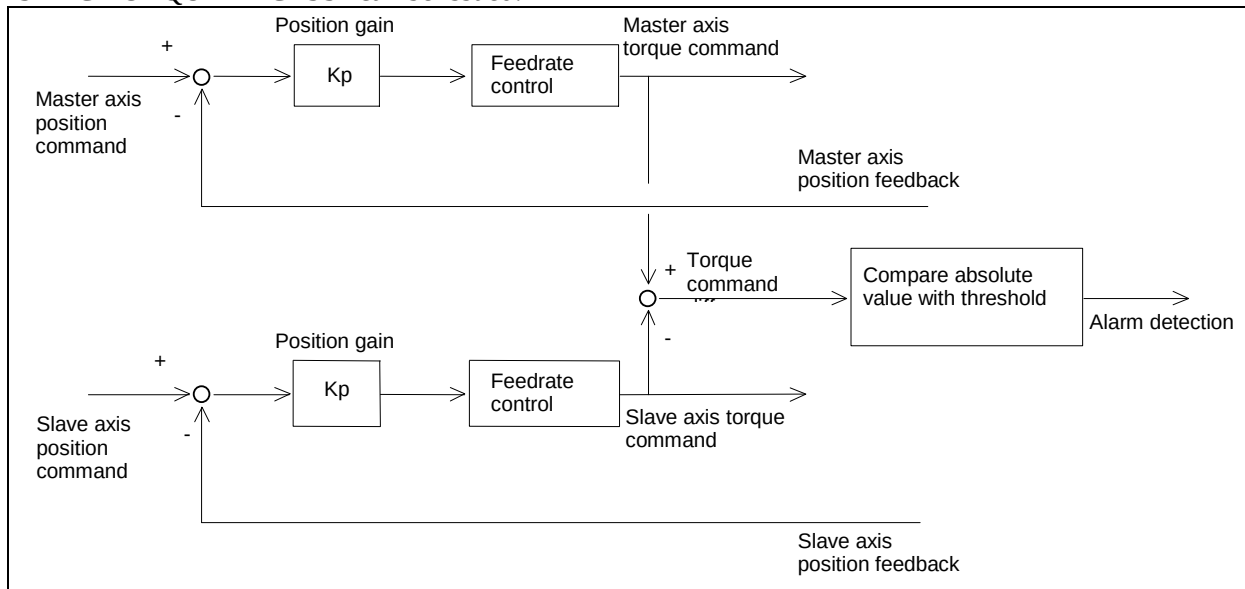


Fig. 20.1.6 (a) System configuration

[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 = 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 is set to 1. When the signal NSYNCA for disabling torque difference alarm detection for axis synchronous control is set to 1, alarm detection is disabled.

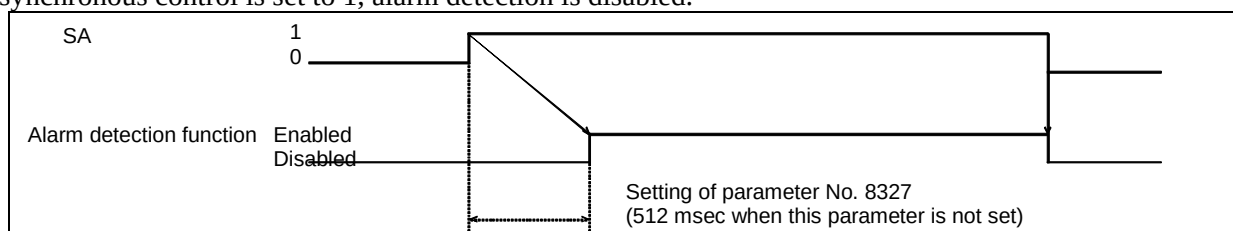


Fig. 20.1.6 (b) Timing chart

When the servo ready signal SA 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 automatic reference position return (G28) 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 (G92) and local coordinate system setting command (G52) is set with the master axis according to the master axis programming.
- 3 During synchronous operation, the axis-by-axis signals such as for external deceleration, interlock, and machine lock are enabled for the master axis only. During synchronous operation, those signals for the slave axis are ignored.
- 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, "ILLEGAL COMMAND IN SYNCHRO-MODE" 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.

20.2 ROTARY AXIS ROLL-OVER

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.

20.2.1 Rotary Axis Roll-over

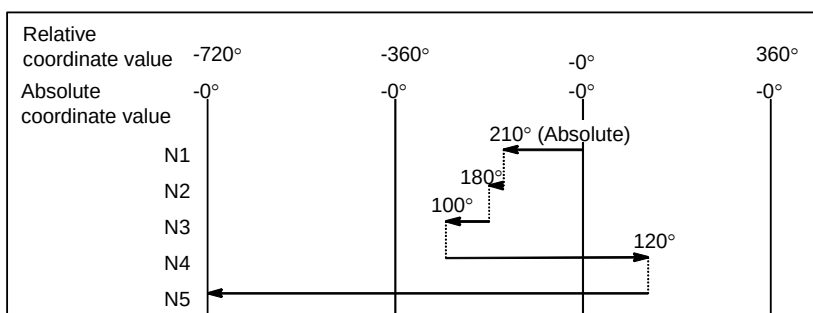
Explanation

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. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when 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 = 360000). When the following program is executed using the roll-over function of the rotary axis, the axis moves as shown below.

G90 A0 ; N1 G90 A-150.0 ; N2 G90 A540.0 ; N3 G90 A-620.0 ; N4 G91 A380.0 ; N5 G91 A-840.0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
	N1	-150	210
	N2	-30	180
	N3	-80	100
	N4	+380	120
	N5	-840	0



20.3 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis. Positioning is performed by the main motor only. The sub-motor is used only to produce torque. With this tandem control function, the torque produced can be doubled.

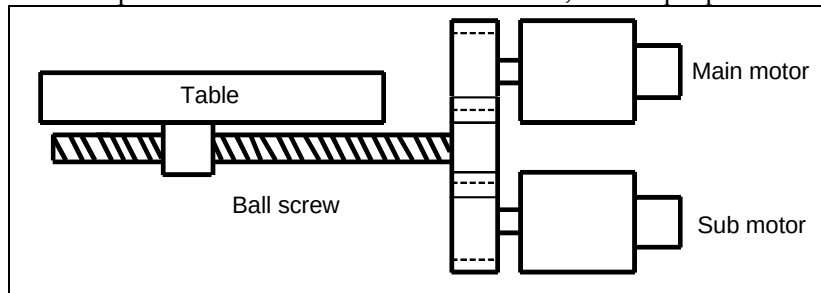


Fig. 20.3 (a) Example of operation

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.

20.4 C AXIS CONTROL (DIE ANGLE INDEXING)

For predetermined dies (tools) on a turret, the angular position of the die can be changed with a command from a DNC, a memory or MDI.

In the past, it was necessary to use many dies when the die shape is the same but the die arrangement is different. Even in such a case, this new function makes it possible to conduct the operations only with one die since the function can freely change the angular position of the die. Further, since chamfering of four corners of a workpiece can be performed only with one die, the time required for die change is reduced, resulting in shorter machining time.

Further, in pattern function for a circular geometry bolt hole circle, arc pattern, and arc nibbling function, the C axis is automatically controlled so that one side of the die always faces the center of arc at each punching position.

NOTE

- 1 The punch section and the die section of the die can be controlled by the same motors or C axis synchronously controlled by the respective servo motors.
- 2 Set 67(C) in parameter No.1020 as the name of the axis to control the axis under the C axis control.
- 3 The C axis should be set as a rotation axis. (bit 0 (ROT_x) of parameter No. 1006 is set to 1)
- 4 Setting the following parameters of C axis allows the CNC to be compatible with FANUC Series 0i-PC, as follows:

Setting	Meaning
Bit 0 (ROT _x) of parameter No.1006=1	Rotary axis (A type)
Bit 1 (ROS _x) of parameter No.1006=0	
Bit 0 (ROA _x) of parameter No.1008=1	The rotary axis roll-over is valid.
Bit 1 (RAB _x) of parameter No.1008=0	The axis rotates in the direction in which the distance to the target is shorter.
Bit 2 (RRL _x) of parameter No.1008=1	Relative coordinates are rounded by the amount of the shift per one rotation

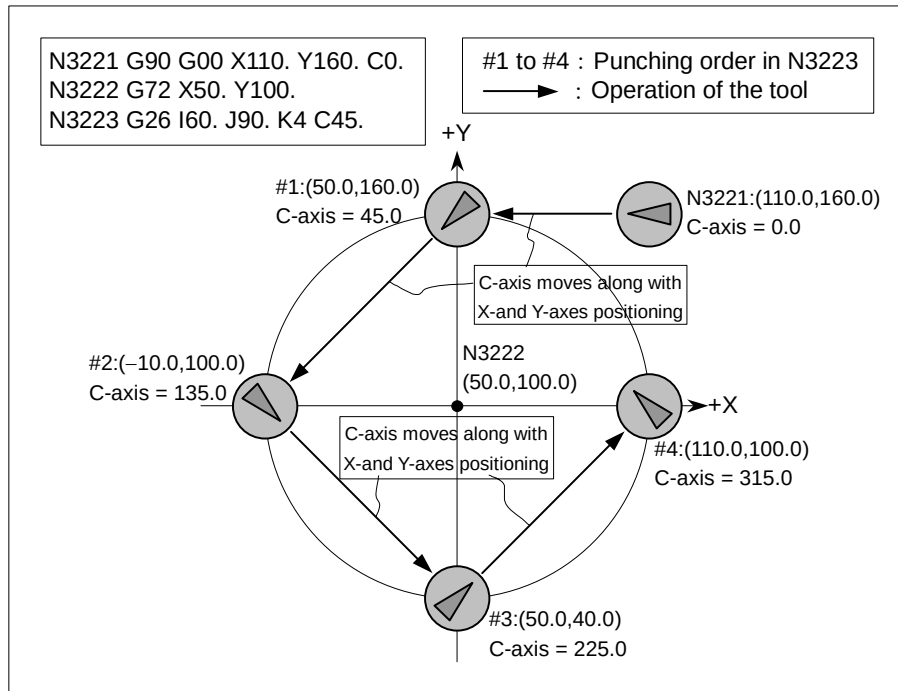


Fig.20.4 (a) Example of bolt hole circle (G26) and C command

There may be some restrictions of functions depending on the machine tools as shown in the following. Always refer to the manual issued by machine tool builder.

- Inhibition of simultaneous control of T and C axes in the manual mode.
- T axis can move only when C axis is at reference point (fixed position on the machine tool where the C axis can reach by the C axis manual reference point return.)

20.4.1 Simultaneously Controlled Axes

X, Y and T or X, Y and C when automatic operation.

20.4.2 Increment System

Least input increment IS-A : 0.01 deg, IS-B : 0.001 deg

Least command increment IS-A : 0.01 deg, IS-B : 0.001 deg

20.4.3 Maximum Programmable Dimension

IS-A : ± 999999.99 deg

IS-B : ± 99999.999 deg

20.4.4 Automatic Acceleration/Deceleration

Linear acceleration/deceleration is possible in manual and automatic rapid traverse rate.

20.4.5 Manual Continuous Feed, Incremental Feed, Manual Reference Point Return

Specifications are the same as X, Y and T axes.

20.4.6 Relationship with Absolute/Incremental Command (G90/G91)

The C-axis command is always regarded as absolute regardless of designation of absolute command (G90) and incremental command (G91).

20.4.7 Positioning in Smaller Angle Rotating Direction

When the C-axis command is designated, it is possible to automatically position the C axis from the present position to the commanded position in the smaller angle rotating direction. When the rotation angle is 180 deg, positioning occurs in the forward direction (CW or CCW according to the connections on the machine side) .

20.4.8 Blocks Where C-axis Command is Possible

C axis command can be specified in the following blocks:

- (1) A block where no one shot G code exists.
However, a block with U, V, W or B command is excluded:
(Example)
X _ Y _ C _ ;
- (2) G70 command
(Example)
G70X _ Y _ C _ ;
- (3) Pattern function (Including nibbling function) G26, G45, G46, G47, G68, G69, G76, G77, G78, G79, G86, G87, G88, G89
(Example)
G26I _ J _ K _ C _ ;
- (4) G01, G02 or G03 command in nibbling mode
(Example)
M12 ;
G01X _ Y _ Q _ C _ ;
X _ Y _ C _ ;
:
M13;

No C-axis command is allowed in a block other than above.

If C axis command is specified in a block other than above, no alarm will be generated. However, if a C-axis command (excluding the nibbling mode) is specified during the linear interpolation or the circular interpolation mode, an alarm (PS4600) is generated.

20.4.9 C-axis Command and its Operation

In 20.4.8, the blocks which C-axis command can be performed were listed. However, unless the die (tool) which allows C-axis control has been selected, C-axis commands cannot be made. Therefore, if the die which does not allow C-axis control is selected, it is necessary to select a die which allows C-axis control before the C-axis command block or to select a die which allows C-axis control to the same block as the C-axis command with the T-command. In the following description of functions, the die (tool) which allows C-axis control is assumed to be T15. Namely, it is assumed that the axis transfer of the C-axis is possible only when die T15 has been selected. In the case of a machine which allows axis transfer of the T-axis only when the C-axis is located at the reference point (fixed location on the machine which can be reached by manual reference point return of C-axis).

- (a) When there was T command

After the C-axis was positioned to the reference point, execution for the T-command is started.

Namely, after the C CR; (CR: setting value of parameter No. 1250, 1251) has been executed, the program command of the block where there was T-command is executed.

- (b) G28 command (automatic reference point return)

After the C-axis has been positioned to the reference point, the X- and Y-axis reference point return is executed by the G28 command. The following shows the program examples of the C-axis command and its operation details:

- (1) Sample program 1

```

:
N200 X_ Y_ T21;
N210 X_ Y_ C_ ;

```

In N210 block, C-axis control cannot be conducted since T15 is not selected.

- (2) Sample program 2

```

:
N200 X_ Y_ T01;
N210 X_ Y_ T15C_ ;

```

In N210 block, C-axis motion starts when positioning of the X, Y and T axes is over and T-command completion signal FIN is returned. Press start signal PF is sent out when positioning of the C axis is over.

When the N210 block is programmed as follows:

```
N210T15C – 1;
```

C axis starts to move when T-axis positioning is over and T-command completion signal FIN is returned. Press start signal PF is not sent out after positioning is finished.

Namely,

- (a) Signal PF is sent out when there is the X or Y command, and

- (b) Signal PF is not sent out when there are no X and Y commands.

20.4.10 Pattern Function, Nibbling Function and C-axis Command

The C-axis command in blocks of G26 (Bolt Hole Circle), G76 (Line At Angle), G77 (Arc), G78/G97 (Grid), G86 (Shear Proof), G87 (Square), G68 (Nibbling Arc), G69 (Nibbling Lin), G88 (Radius) and G89 (Cut At Angle) are described below. Of these, the movement specifications by G26, G77, G68 and G88 concerning the circular geometry differ from those of the other functions.

- (1) G45, G78, G79, G86, G87, G69, G89

```

:
:
N200 G72 X__Y__;
N210 G76 I__J__K__T15 C__ ;

```

In N210 block, T-axis positioning starts simultaneously when the X and Y axes start to move at the first positioning point. When the positioning of the X, Y and T axes is finished and T-command completion signal FIN is returned, positioning of the C axis starts.

After positioning of the C axis is over, press start signal PF is sent out, and punch operation is performed at the first positioning point. Thereafter, the X and Y axes are sequentially positioned following the commanded geometry without positioning of the C axis.

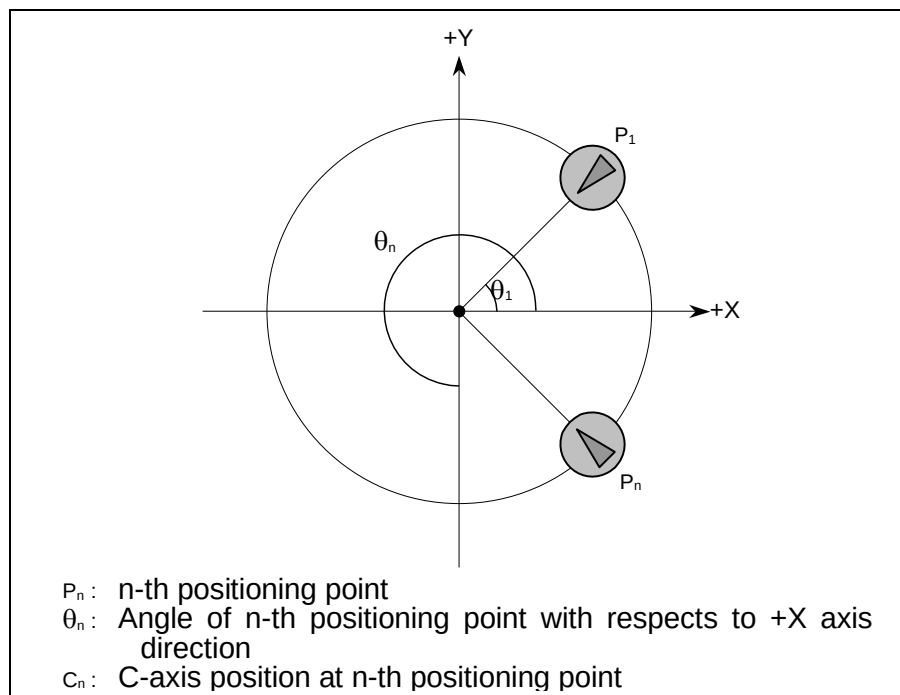
(2) G26, G46, G47, G77, G68, G88

```

      :
      :
N200  G72  X  Y  ;
N210  G26  I  J  θ  K  T15C  cl  ;

```

In N210 block, positioning to the first point is conducted in the same way as described in 19.3.10 (1). Thereafter, X and Y axes are sequentially positioned following the commanded geometry while the C axis positioning is conducted.



C_n is calculated as follows:

$$C_n = C_1 + (\theta_n - \theta_1)$$

In case of G68, and alarm (PS 4522) occurs if the C-axis move value between adjacent positioning points exceeds the parameter value (No. 16194).

20.4.11 C-axis Command in Nibbling Mode

For the C-axis commands between the M code for nibbling mode and the M code for nibbling mode cancel, an alarm is indicated if the C-axis movement amount per block exceeds the value set with the parameter (No. 16194).

20.4.12 T-axis Command Ignore Signal TNG and C-axis Command

The C-axis command is ignored when T command neglect signal TNG <Gn233.5> is on.

20.4.13 Compensating the Position of the C-axis

This function automatically compensates the position of the C-axis to orient each tool in the specified direction. This simplifies mechanical adjustment such as adjusting the reference position when installing tools. Up to 20 compensation settings are specified in parameters No. 16430 to No. 16449. Up to 20 tool numbers are specified in parameters No. 16370 to No. 16389.

The C-axis is compensated when the tool moves around the C-axis after T commands have been specified.



CAUTION

Tool numbers of tools must be specified with address T followed by up to four digits.

20.4.14 Compensating Backlash Along the C-axis for Each Tool Group

This function compensates backlash along the C-axis for each tool group. Changing the backlash compensation for each tool group enables higher-precision machining. Up to 20 compensation settings are specified in parameters No. 16390 to No. 16409. Up to 20 tool numbers are specified in parameters No. 16370 to No. 16389.



CAUTION

The tool numbers must be specified with address T followed by up to four digits.

20.5 RAM-AXIS CONTROL

The CNC controls press motion in the block for 1-cycle pressing by using RAM-axis control.

The CNC cannot move RAM-axis by program command. RAM-axis moves automatically at the end of the block executing 1-cycle press by setting RAM-axis and the RAM-axis motion patterns.

20.5.1 Setting RAM-axis Motion Pattern

RAM-axis motion pattern should be set on the C language executor variable area. After Setting the RAM-axis motion patterns, the CNC read specified the RAM-axis motion patterns when the punching is commanded. Specifying the address by the application software allows writing the data into C language executor variable area.

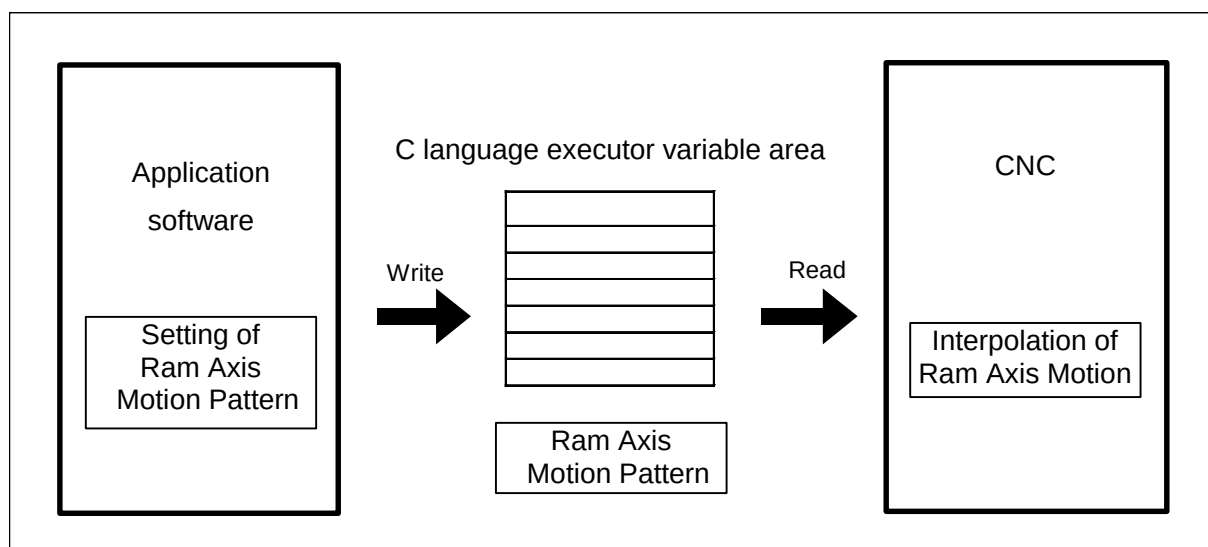


Fig.20.5.1 (a) Setting RAM-axis Motion Pattern

20.5.2 Press Motion by RAM-axis Control

RAM-axis moves automatically at the end of the block executing 1-cycle press by setting the RAM-axis motion patterns.

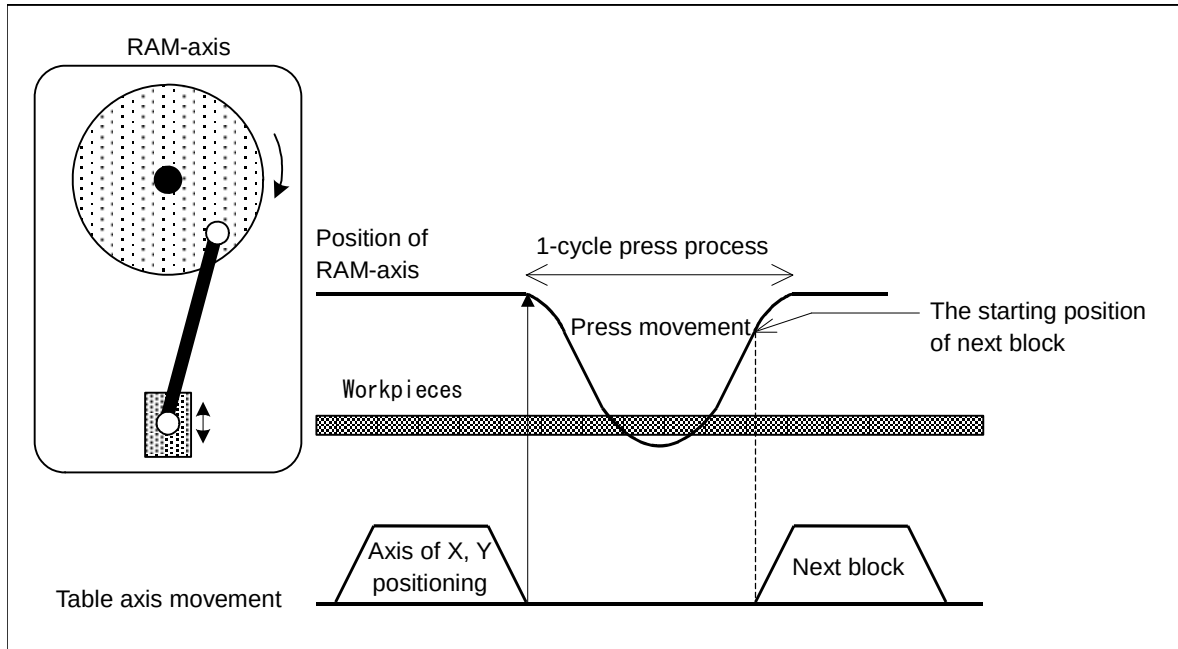


图20.5.2 (a) Press motion by RAM-axis control

20.5.3 RAM-axis Up Motion

Using the RAM-axis up signal RAMUP<Gn238.0> allows setting RAM-axis in the RAM up position specified.

After confirmed the RAM-axis up signal RAMUP<Gn238.0> = "1", the CNC sets RAM-axis in the RAM up position specified immediately.

The feedrate used for the RAM-axis up motion is set on the parameter No. 1420.

III. OPERATION

1 GENERAL

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

1.1	MANUAL OPERATION.....	331
1.2	TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION	332
1.3	AUTOMATIC OPERATION	333
1.4	TESTING A PROGRAM	334
1.5	EDITING A PROGRAM.....	336
1.6	DISPLAYING AND SETTING DATA	336
1.7	DISPLAY	339

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, "MANUAL REFERENCE POSITION RETURN".)

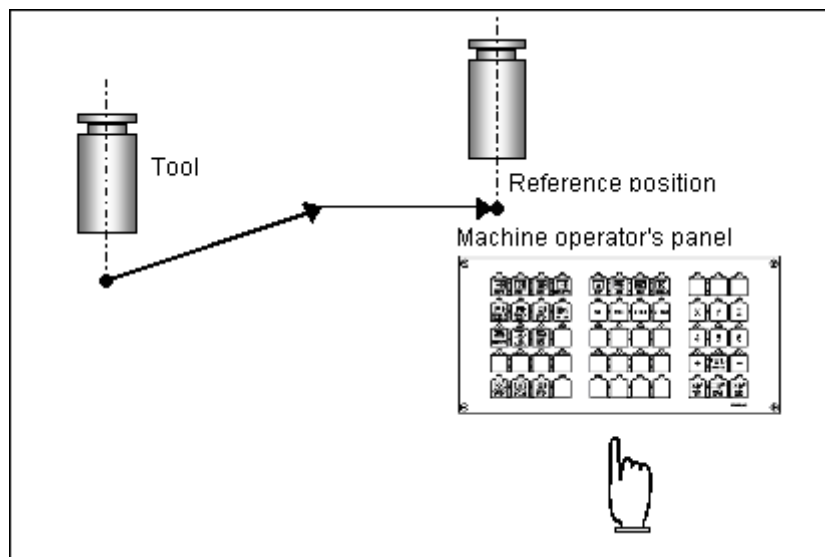


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.

This operation is called automatic reference position return. (See Chapter II-6, "REFERENCE POSITION".)

- 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.

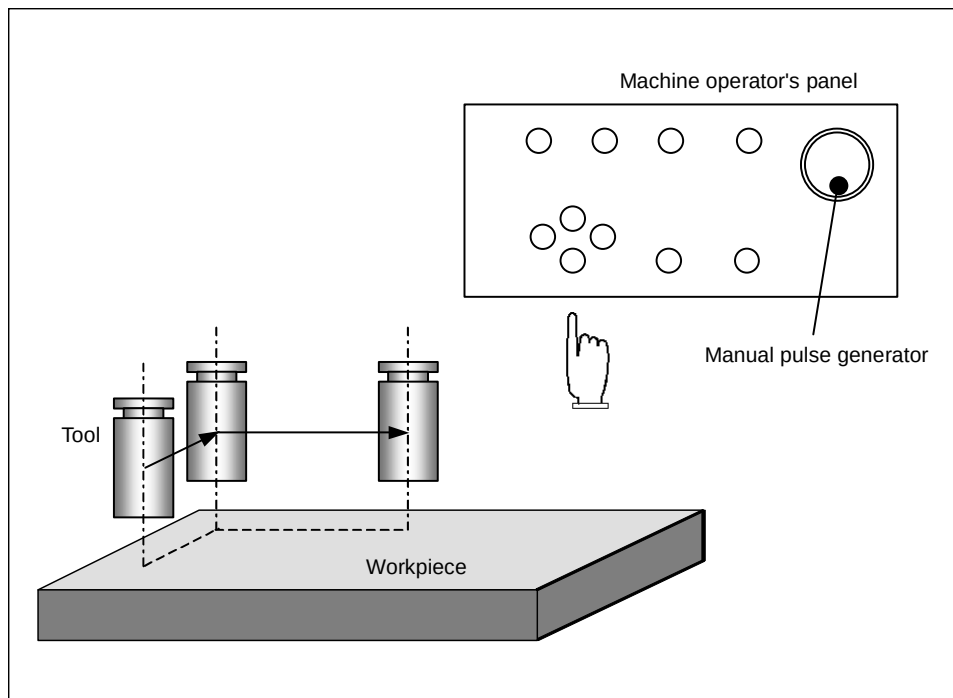


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

- (i) Jog feed (See Section III-3.2, "JOG FEED (JOG)".)
The tool moves continuously while a pushbutton remains pressed.
- (ii) Incremental feed (See Section III-3.3, "INCREMENTAL FEED".)
The tool moves by the predetermined distance each time a button is pressed.
- (iii) Manual handle feed (See Section III-3.4, "MANUAL HANDLE FEED".)
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 Chapter III-4, "AUTOMATIC OPERATION".)

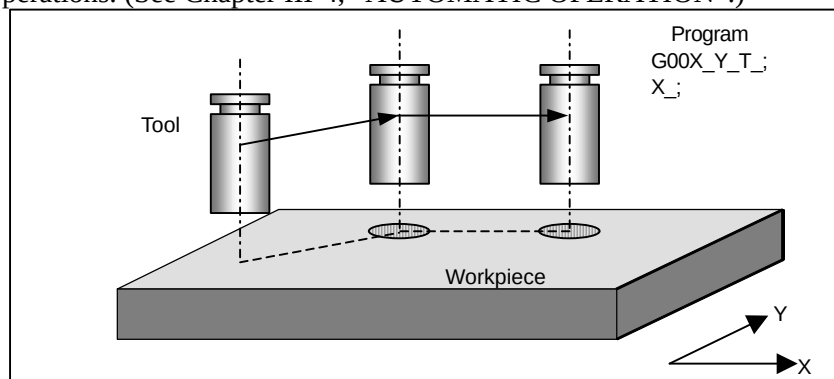


Fig. 1.2 (a) Tool Movement by programming

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.

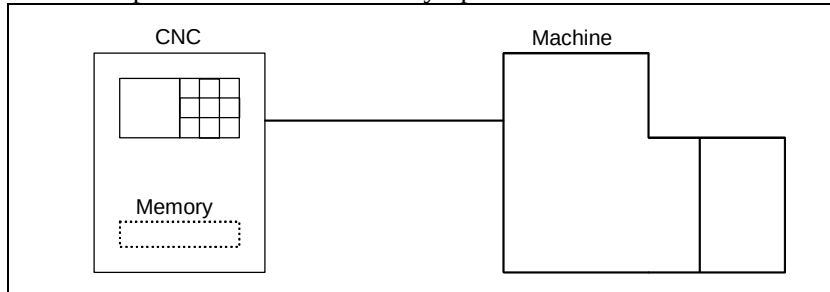


Fig. 1.2 (b) Memory operation

- MDI operation

After the program is entered, as a command group, from the MDI unit, the machine can be run according to the program. This operation is called MDI operation.

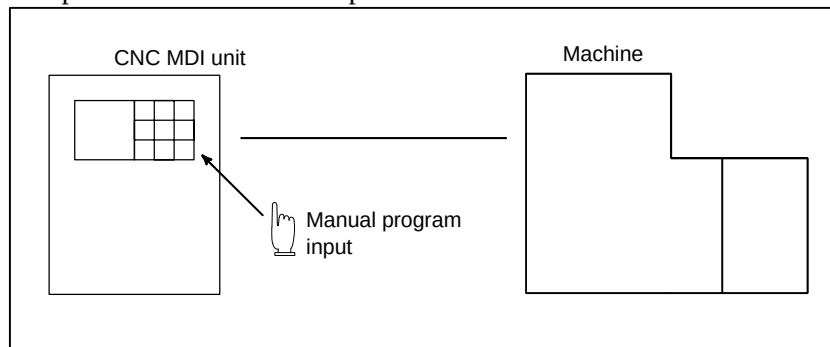


Fig. 1.2 (c) MDI operation

- 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. (See Section III-10.4, "PROGRAM SEARCH".)

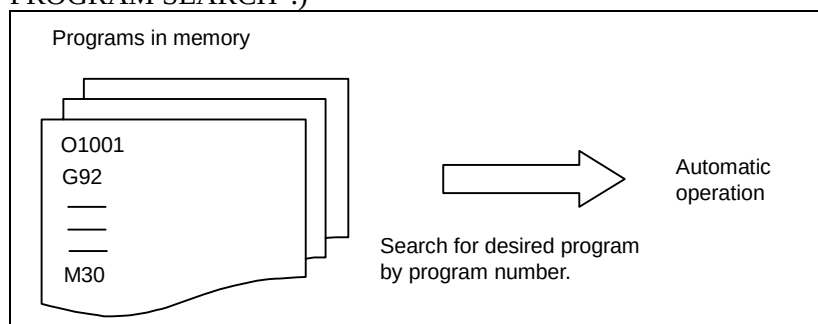


Fig. 1.3 (a) Program selection for automatic operation

- 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 Chapter III-4, “AUTOMATIC OPERATION”.)

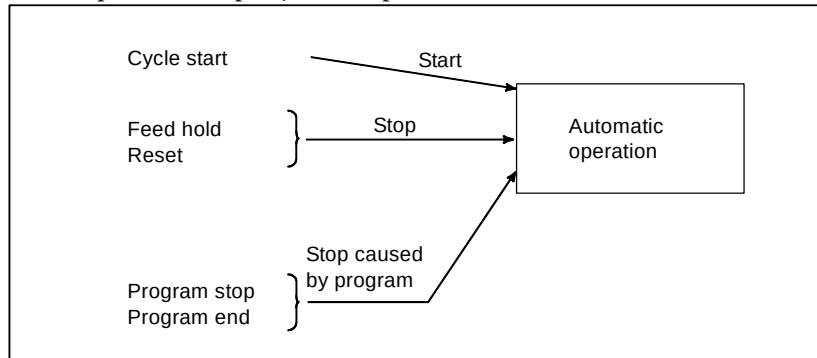


Fig. 1.3 (b) Start and stop for automatic operation

- Handle interruption

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle. (See Section III-4.6, “MANUAL HANDLE INTERRUPTION”.)

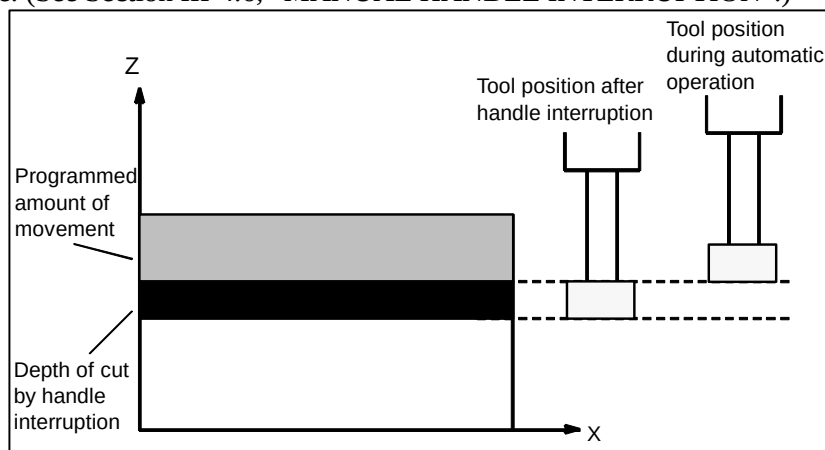


Fig. 1.3 (c) Handle interruption for automatic operation

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 Chapter III-5, “TEST OPERATION”).

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, “DRY RUN”.)

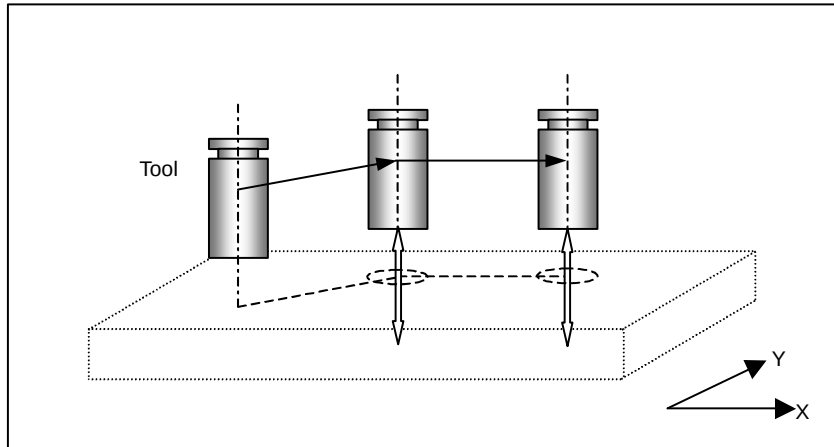


Fig. 1.4.1 (a) Dry run

- Feedrate override

Check the program by changing the feedrate specified in the program. (See Section III-5.2, “FEEDRATE OVERRIDE”.)

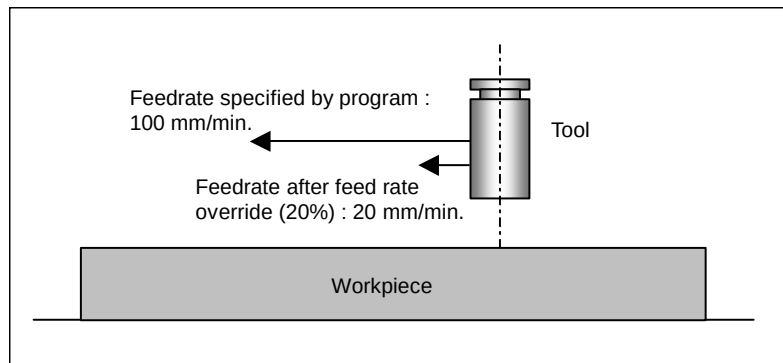


Fig. 1.4.1 (b) Feedrate override

- 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, “SINGLE BLOCK”.)

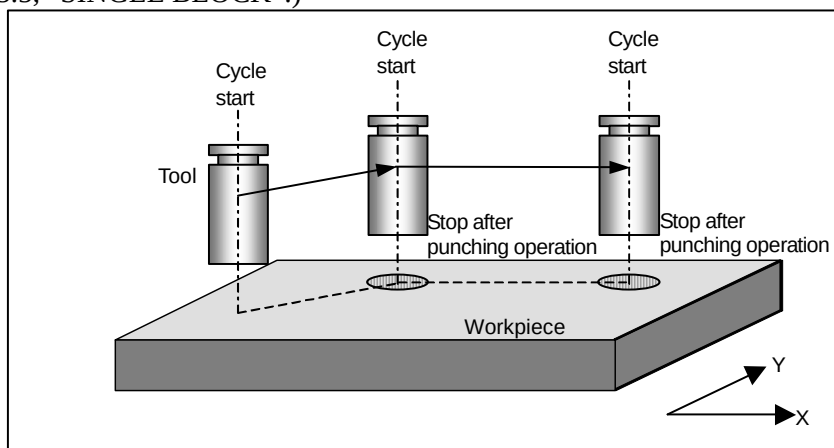


Fig. 1.4.1 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanation

- Machine Lock

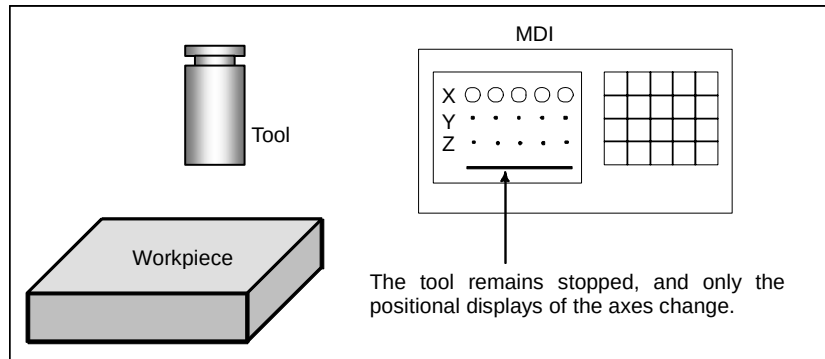


Fig. 1.4.2 (a) Machine Lock

- Auxiliary function lock

When automatic running is placed into the auxiliary function lock mode during the machine lock mode, all auxiliary functions are disabled. (See Section III-5.1, "MACHINE LOCK AND AUXILIARY FUNCTION LOCK".)

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 Chapter III-10, "EDITING PROGRAMS".)

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 Chapter III-12, "SETTING AND DISPLAYING DATA".)

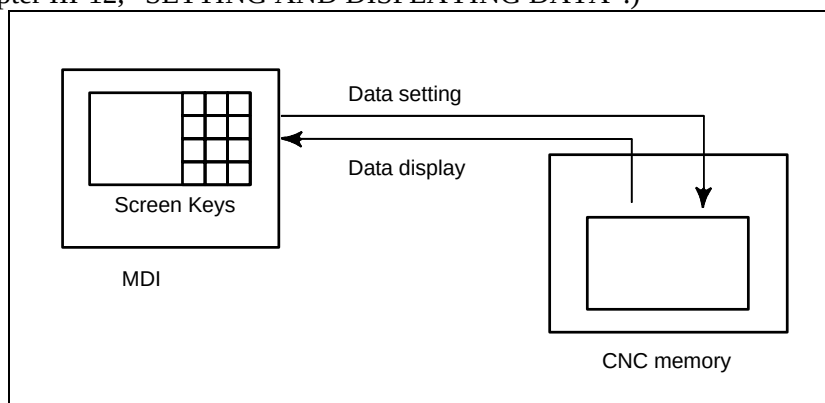


Fig. 1.6 (a) Displaying and setting data

Explanation

- Offset value

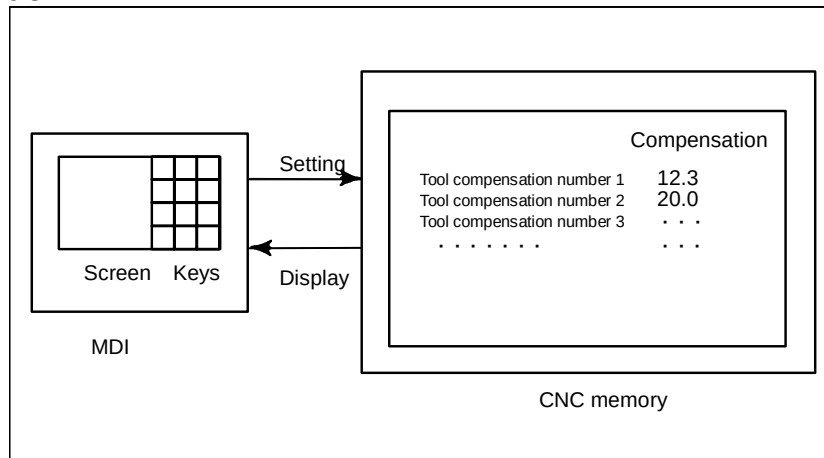


Fig. 1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (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. (See Subsection III-12.3.1, “Setting and Displaying the Tool Compensation Value”.)

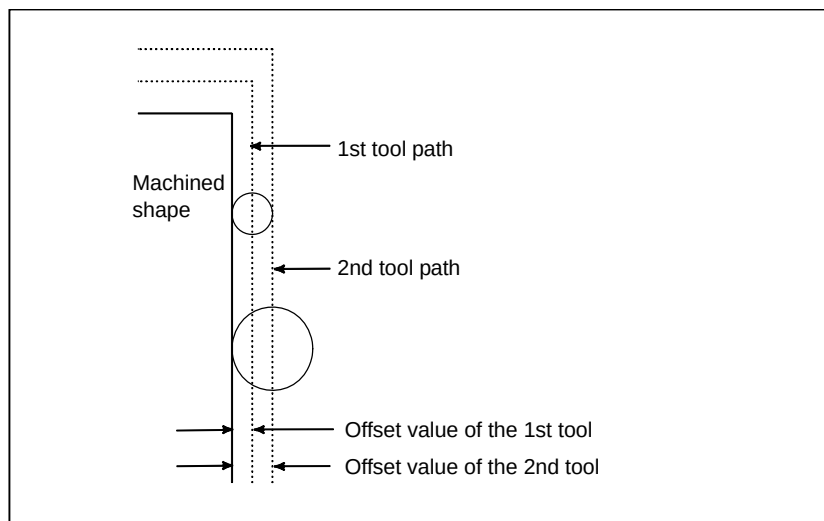


Fig. 1.6 (c) Offset value

- 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 Subsection III-12.3.2, “Displaying and Entering Setting Data”.)

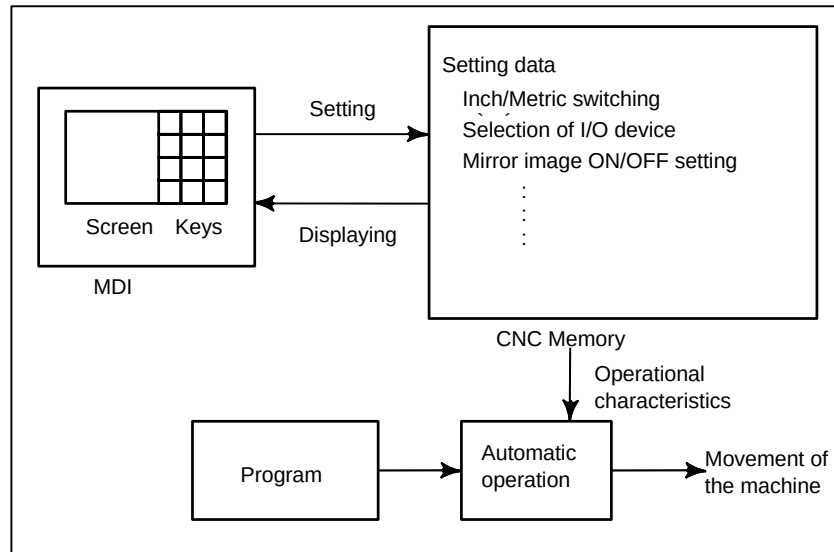


Fig. 1.6 (d) Displaying and setting operator's setting data

- 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 Subsection III-12.4.1, "Displaying and Setting Parameters".)

Parameters differ depending on machine tool.

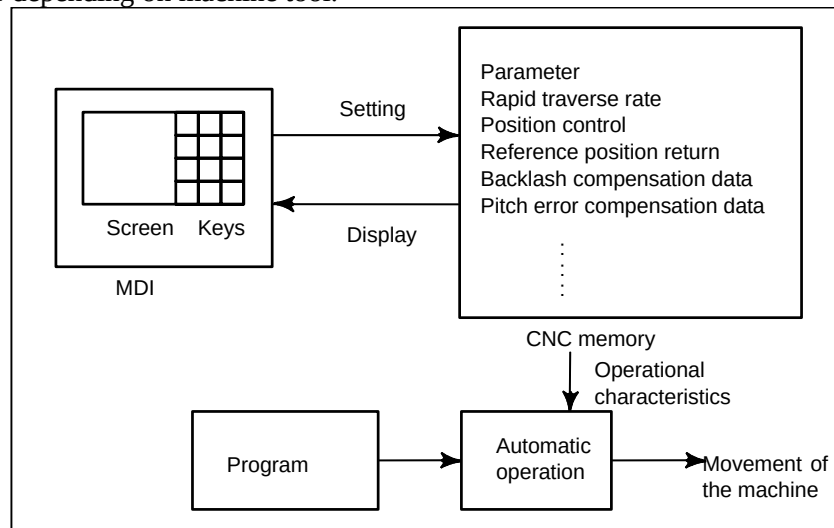


Fig. 1.6 (e) Displaying and setting parameters

- 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 Chapter III-12, “SETTING AND DISPLAYING DATA”.)

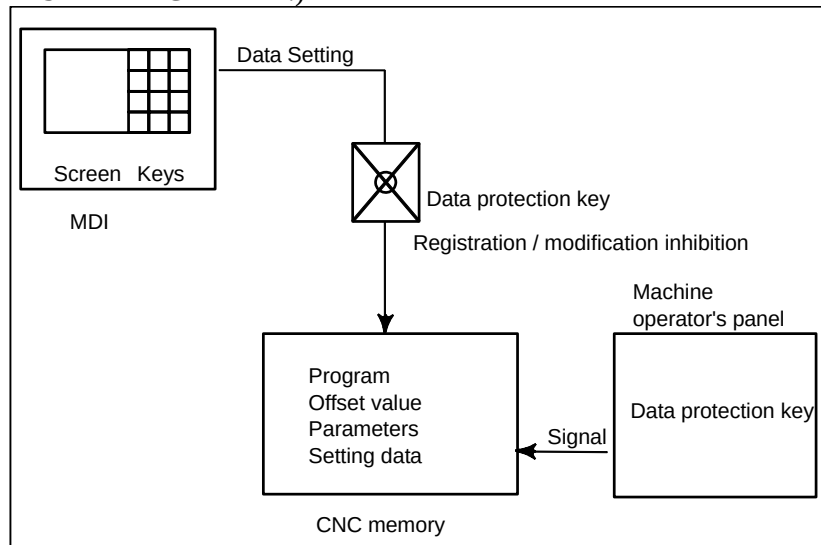


Fig. 1.6 (f) Data protection key

1.7 DISPLAY

1.7.1 Program Display

The contents of the currently active program are displayed.
(See Subsection III-12.2.1, “Program Contents Display”.)

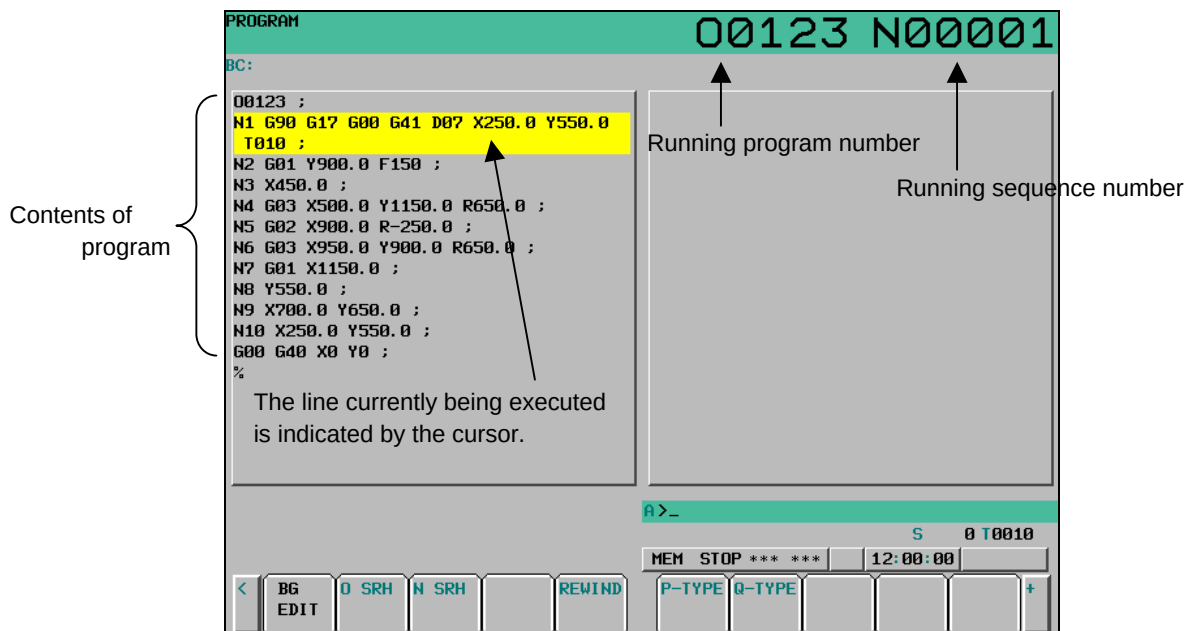


Fig. 1.7.1 (a)

The programs in the program memory are listed. (See Subsection III-12.2.4, “Program List Screen”.)

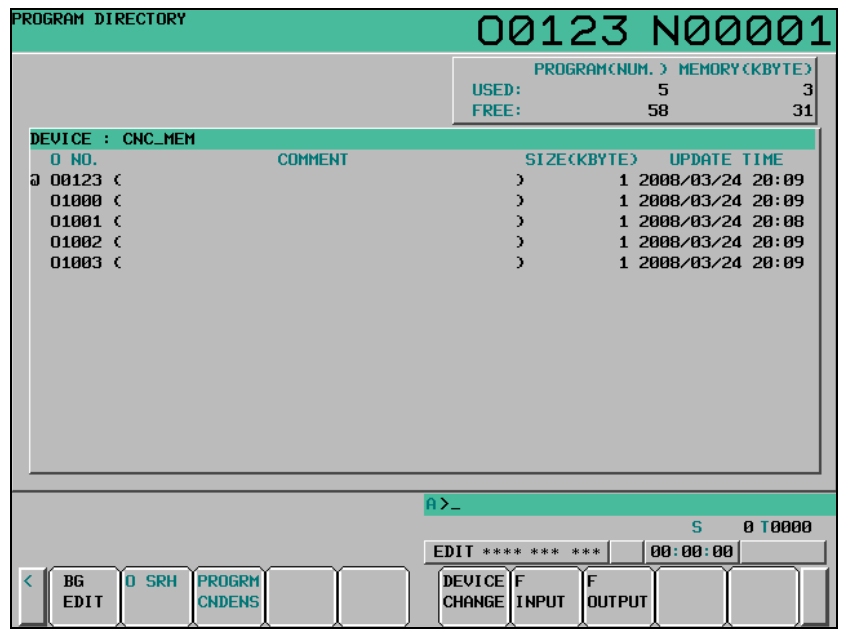


Fig. 1.7.1 (b)

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, “Position Display in the Workpiece Coordinate System”, “Position Display in the Relative Coordinate System”, and “Overall Position Display”.)

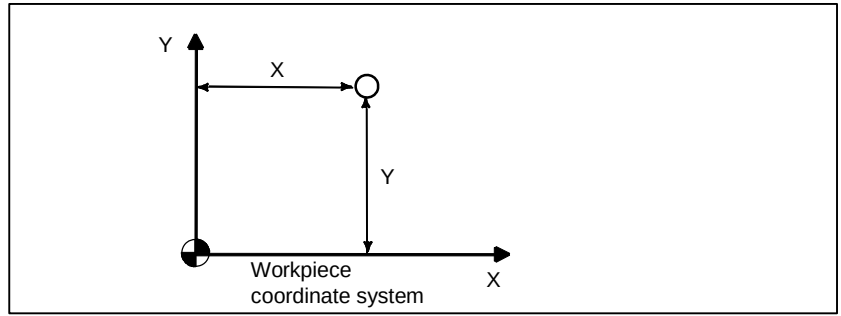


Fig. 1.7.2 (a)

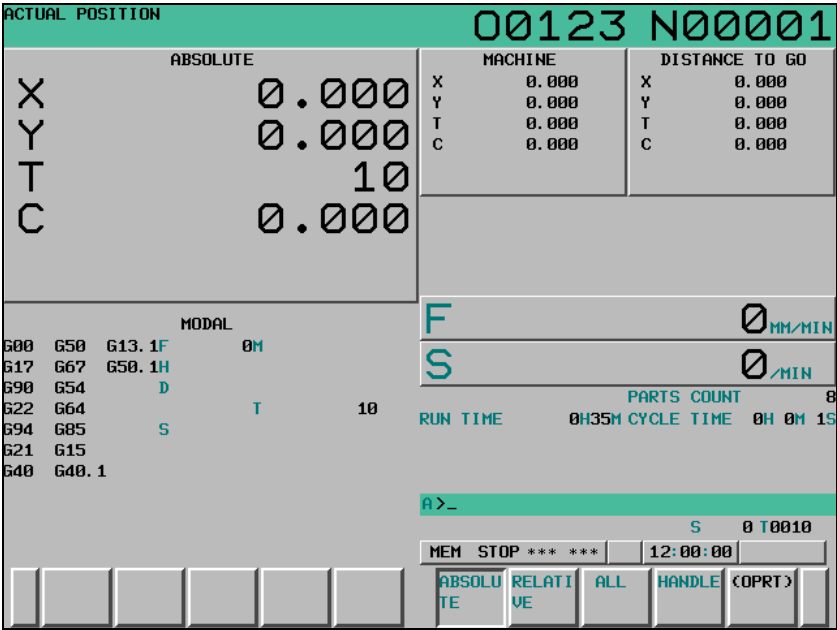


Fig. 1.7.2 (b)

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, “ALARM DISPLA”.)
See APPENDIX H 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 run time, cycle time, and parts count. (See Subsection III-12.1.6, “Displaying and Setting Run Time, Parts Count, and Time”.)

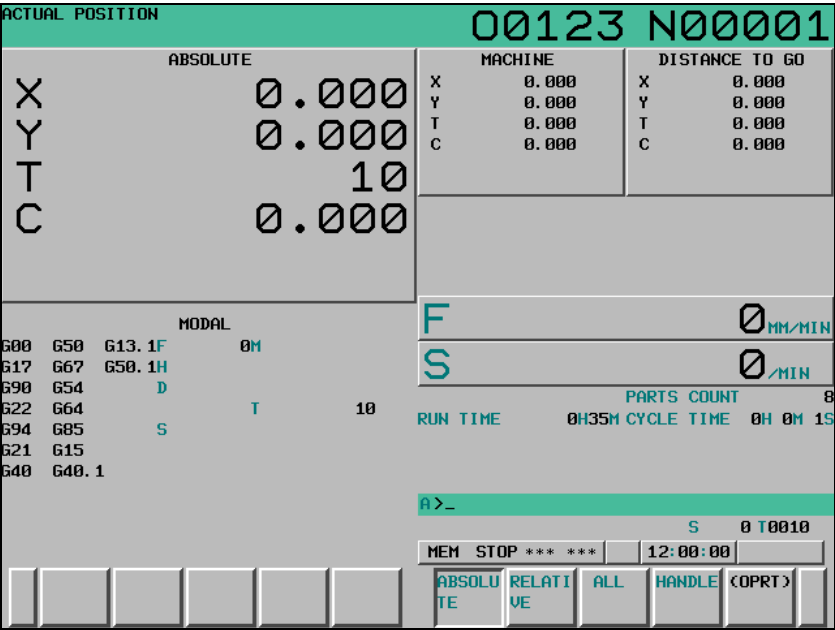


Fig. 1.7.4 (a)

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	POWER ON/OFF.....	343
2.2	SETTING AND DISPLAY UNITS.....	344
2.3	EXPLANATION OF THE MDI UNIT	349
2.4	FUNCTION KEYS AND SOFT KEYS	350
2.5	EXTERNAL I/O DEVICES	360

2.1 POWER ON/OFF

2.1.1 Turning on the Power

Procedure of turning on the power

Procedure

- 1 Check that the appearance of the CNC machine tool is 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.

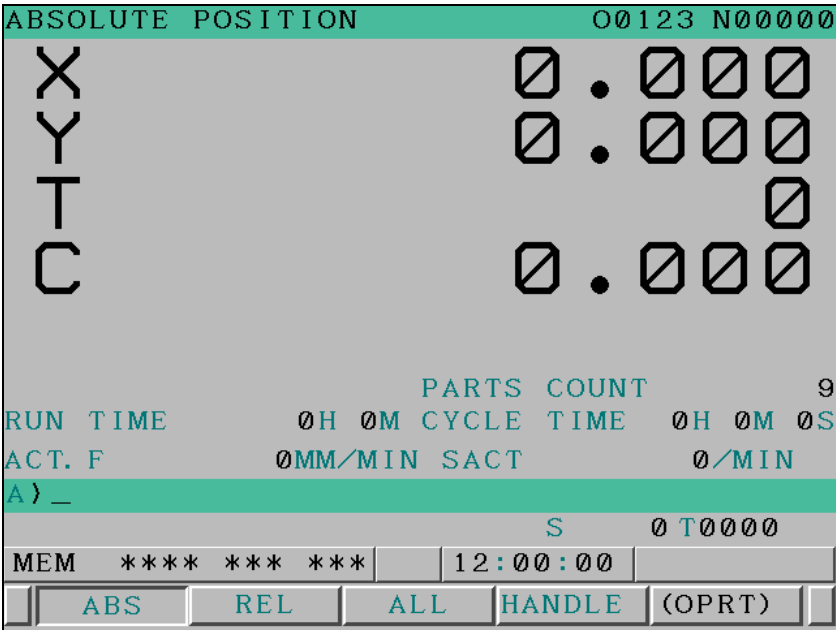


Fig. 2.1.1 (a) Position screen (for machining center system)

- 4 Check that the fan motor is rotating.



WARNING

Until the positional or alarm screen is displayed at the power on, do not touch a key on the MDI unit. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

2.1.2 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.

2.2 SETTING AND DISPLAY UNITS

The setting and display units are shown in Subsections III-2.2.1 to III-2.2.4.

8.4" LCD/MDI	III-2.2.1
10.4" LCD	III-2.2.2
Standard MDI Unit (ONG Key).....	III-2.2.3
Small MDI Unit (ONG Key).....	III-2.2.4

2.2.1 8.4" LCD/MDI

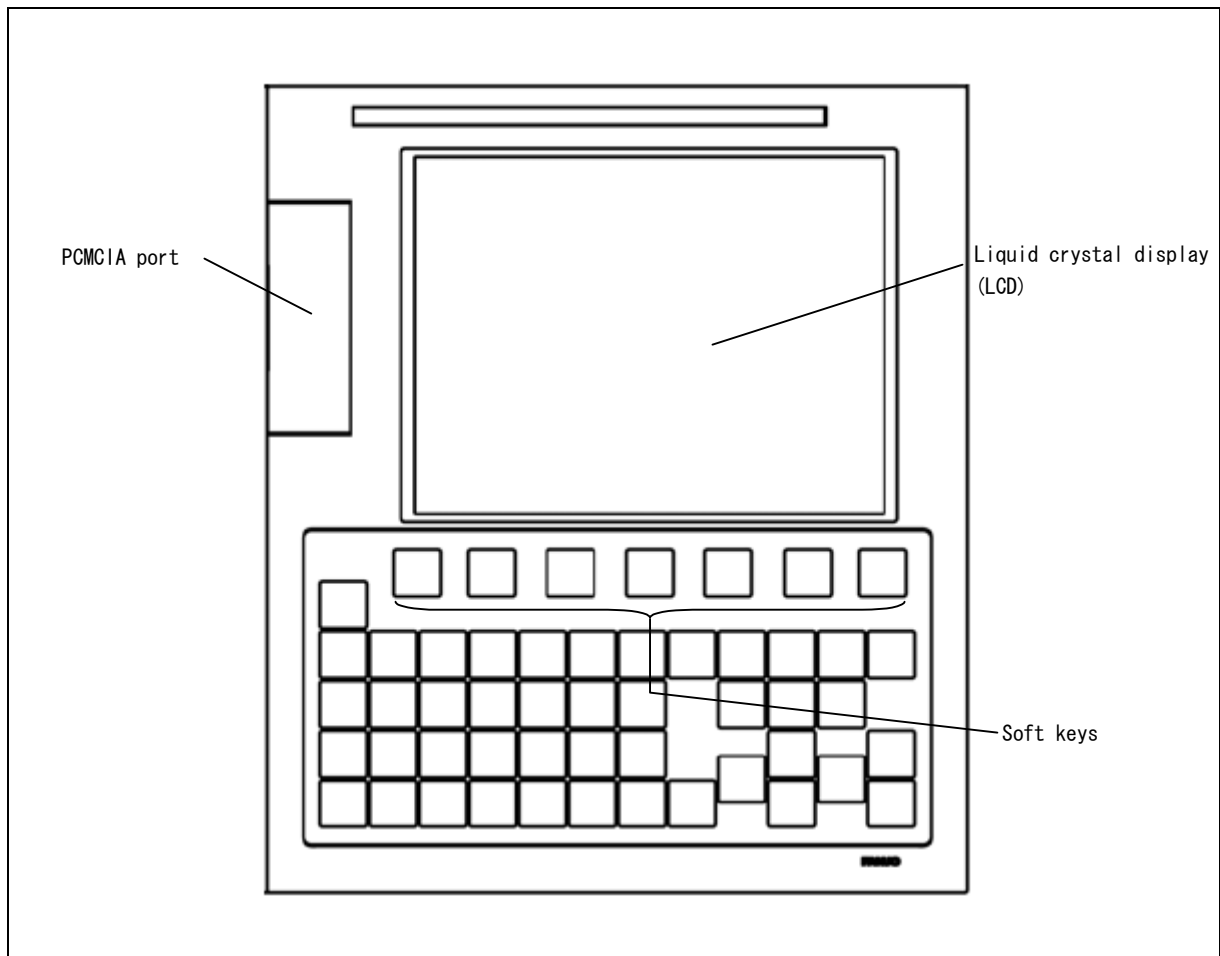


Fig. 2.2.1 (a) 8.4" LCD/MDI (vertical type)

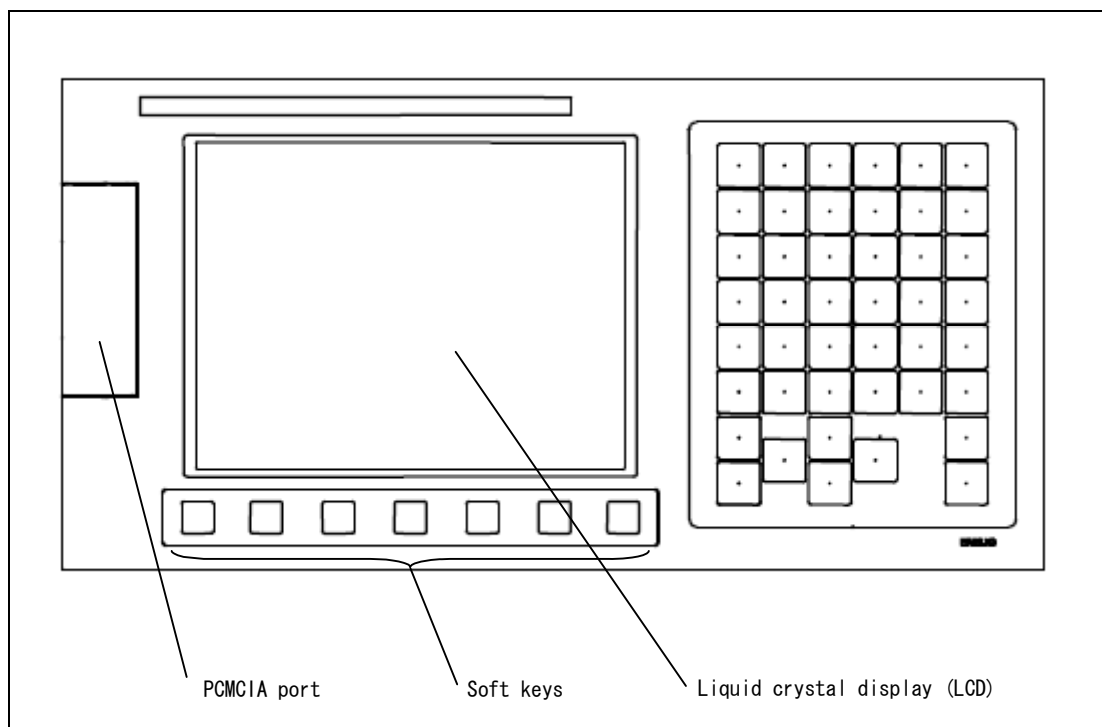


Fig. 2.2.1 (b) 8.4" LCD/MDI (horizontal type)

2.2.2 10.4" LCD

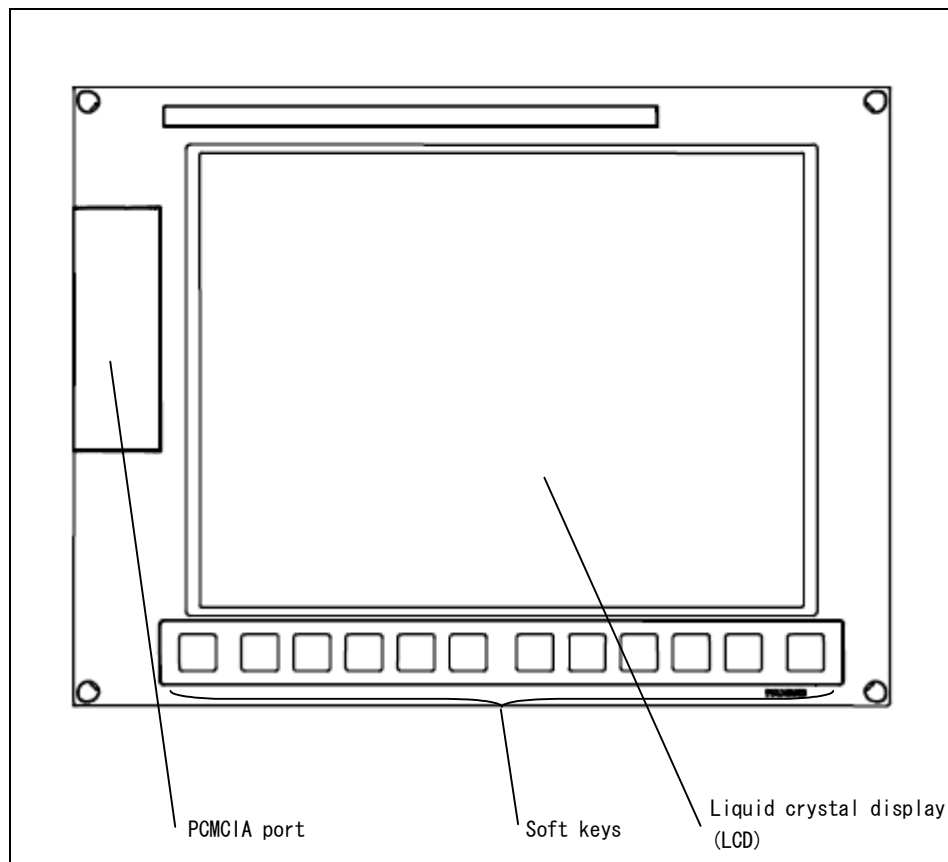


Fig. 2.2.2 (a) 10.4" LCD

(Note) The touch panel display unit has no soft keys.

2.2.3 Standard MDI Unit (ONG Key)

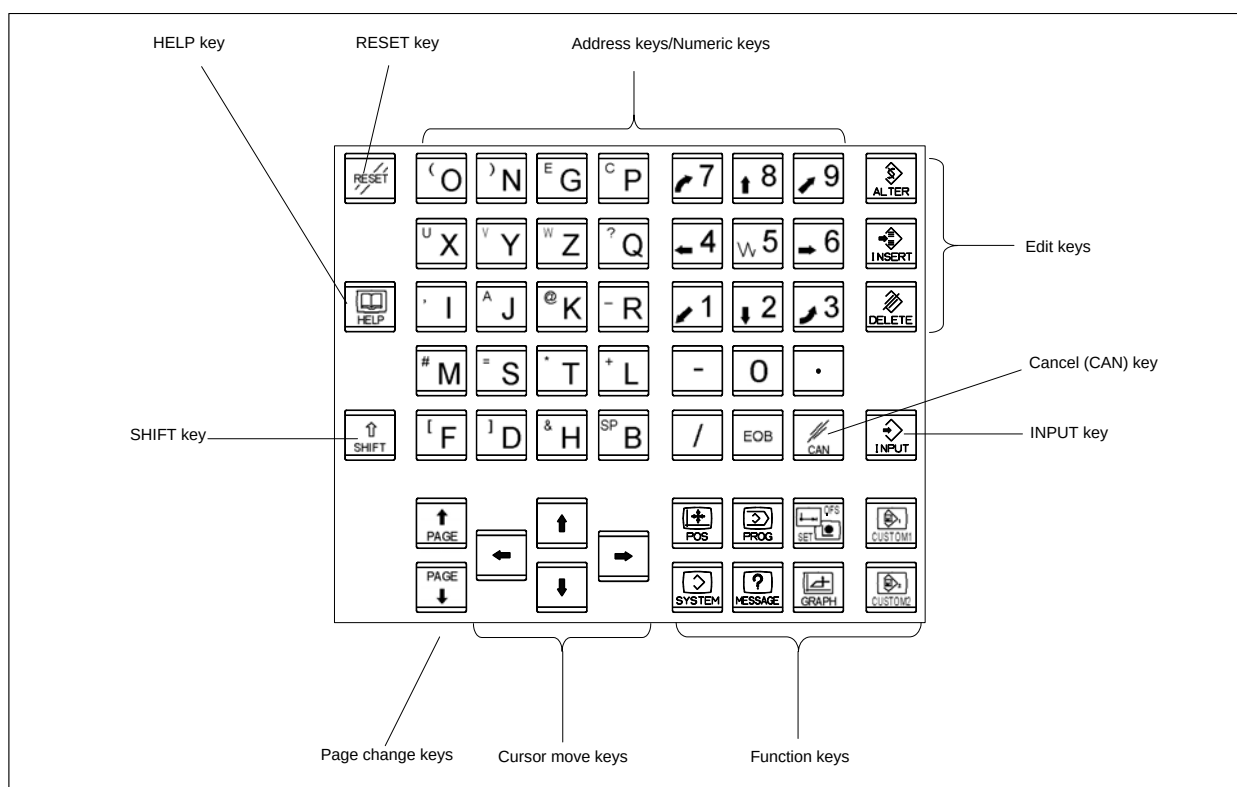


Fig. 2.2.3 (a) Standard MDI Unit (ONG Key)

2.2.4 Small MDI Unit (ONG Key)

Small MDI unit (ONG Key, horizontal type)

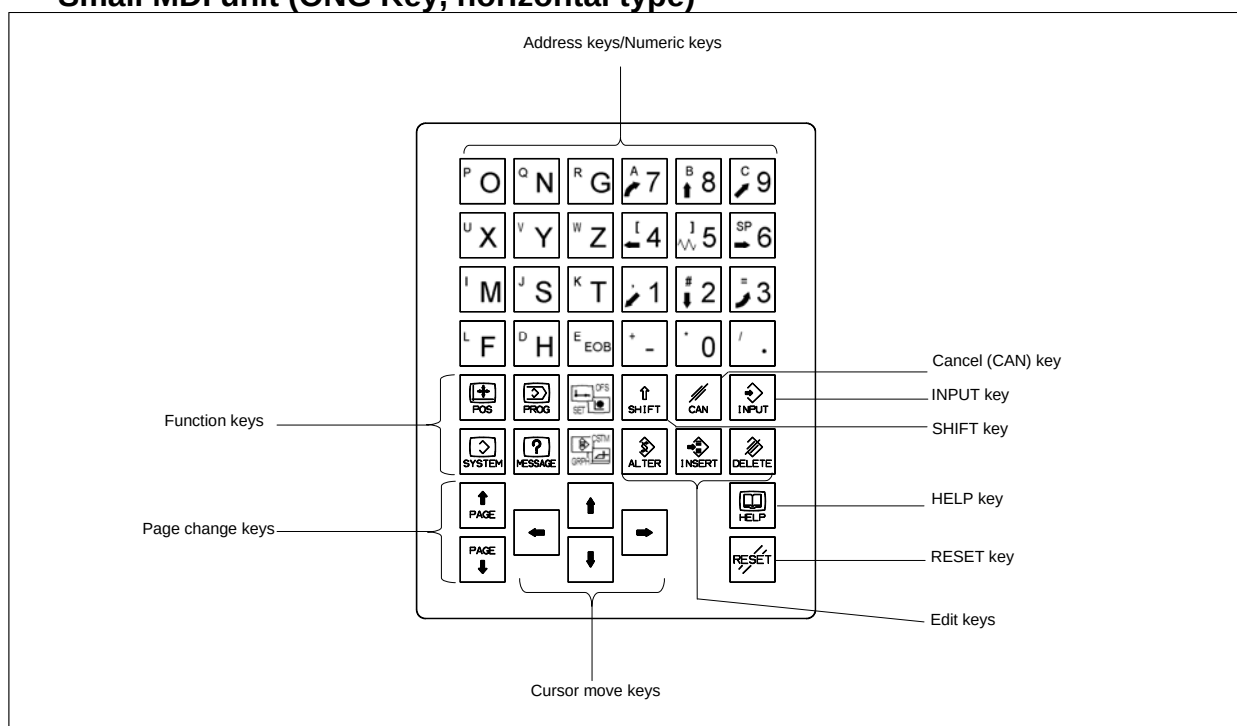
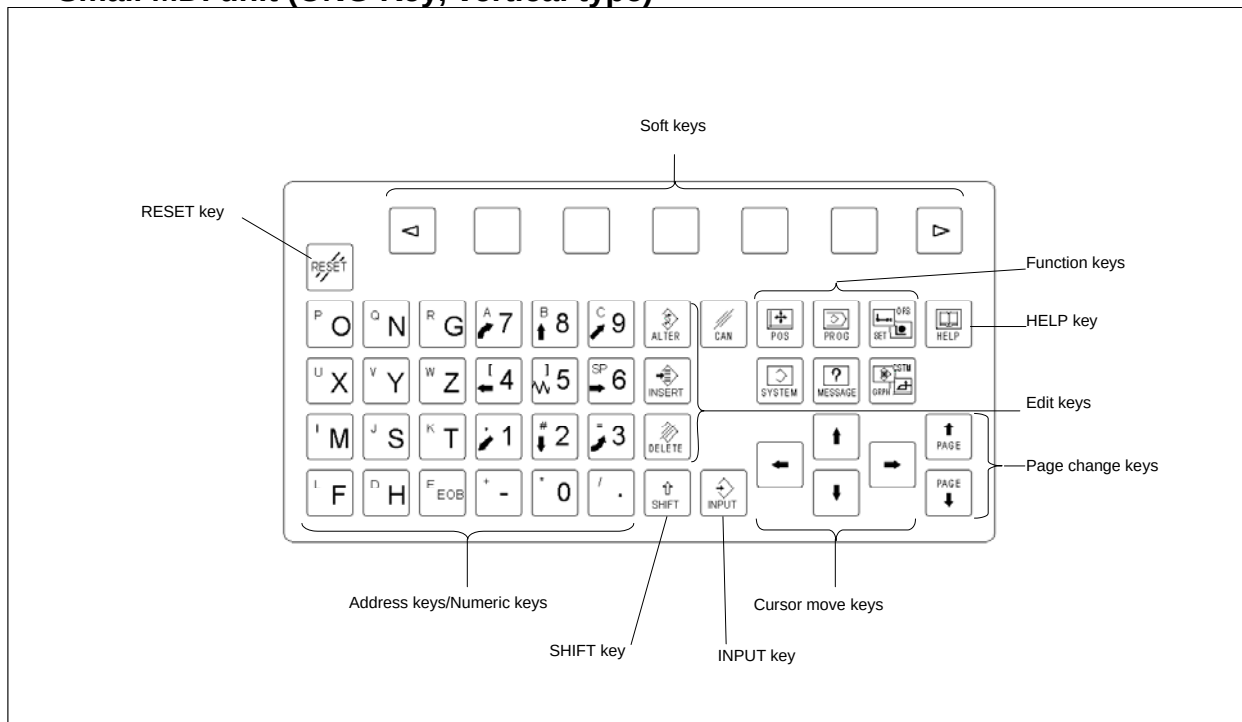
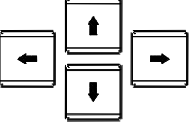


Fig. 2.2.4 (a) 8.4" Small MDI unit (ONG Key, horizontal type)

Small MDI unit (ONG Key, vertical type)**Fig. 2.2.4 (b) Small MDI unit (ONG Key, vertical type)**

2.3 EXPLANATION OF THE MDI UNIT

Table 2.3 (a) Explanation of the MDI unit

No.	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 keys have two characters on their key top. Pressing the <SHIFT> key switches the characters. Special character ^ is displayed on the screen when a character indicated at the top left corner on the key top can be entered.
6	INPUT key 	When an address 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  key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result.
7	CANCEL (CAN) key 	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  is pressed, Z is canceled and >N001X100_ is displayed.
8	Edit keys 	Press these keys when editing the program.  : ALTER  : INSERT  : IDELETE
9	Function keys 	Press these keys to switch display screens for each function. See Section III-2.4.2, "Function Keys" 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.

No.	Name	Explanation
11	Page change keys (Page keys)  	Two kinds of page change keys are described below.  : This key is used to changeover the page on the screen in the forward direction.  : This key is used to changeover the page on the screen in the reverse direction.

2.4 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.

2.4.1 General Screen Operations

- Procedure

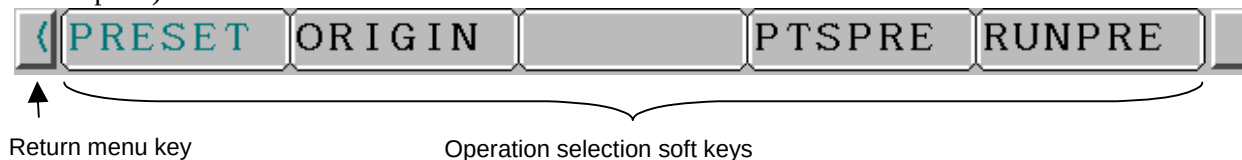
- 1 By pressing a function key on the MDI unit, the chapter selection soft keys that belong to the function are displayed.

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 to be performed.

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 assume one of the following states, depending on the selection target:

- 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

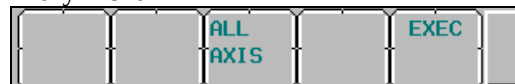
Chapter selection soft keys



Operation selection soft keys



Operation selection keys, auxiliary menu



2.4.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI unit:

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.

2.4.3 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 horizontal 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.

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-12.1.7, "Operating Monitor Display".
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 2.4.3 (a) Position display screen

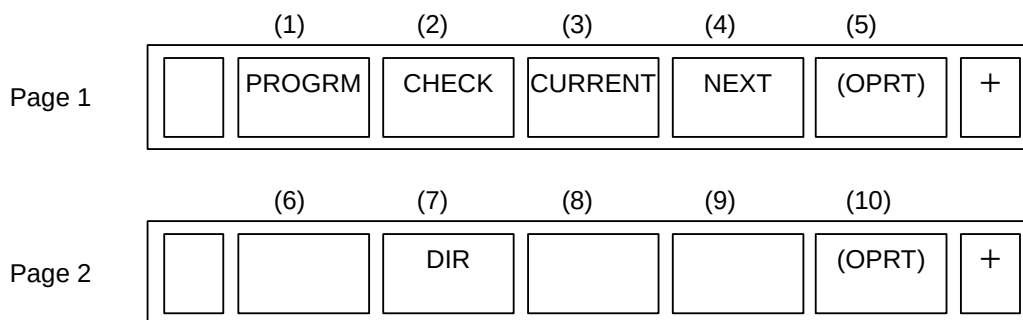
No.	Chapter menu	Description
(1)	ABS (ABSOLUTE)	Selects the absolute coordinate display screen.
(2)	REL (RELATIVE)	Selects the relative coordinate display screen.
(3)	ALL (ALL)	Selects the overall coordinate display screen.
(4)	HNDL (HANDLE)	Selects the operation screen for manual handle operation.
(6)	MONI (MONITOR)	Selects the screen for displaying the servo axis load meter.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

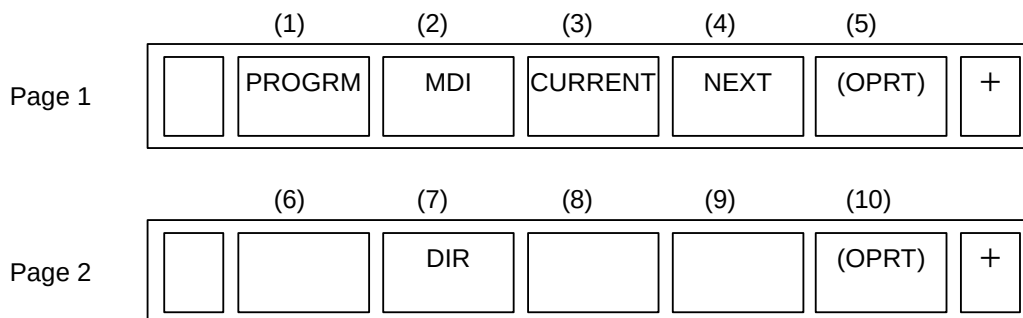
Program screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

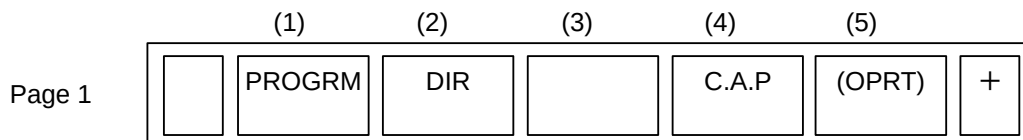
In the MEM/RMT mode



In the MDI mode



In the EDIT mode



In the JOG/HND/REF mode

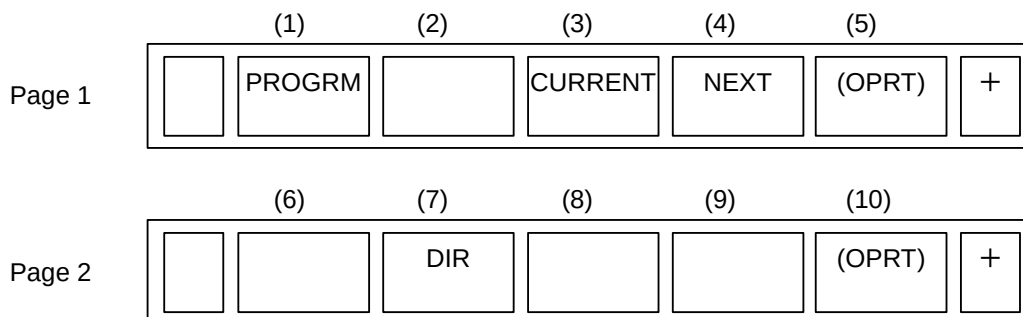


Table 2.4.3 (b) Program

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-inch display unit)
(2)	MDI (MDI)	Selects the screen for editing and displaying a program in the MDI mode. (Only in the MDI mode)

No.	Chapter menu	Description
(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.
(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.

Page 1	(1)	(2)	(3)	(4)	(5)
		OFFSET	SETTING	WORK	(OPRT) +
Page 2	(6)	(7)	(8)	(9)	(10)
		MACRO		OPR	(OPRT) +
Page 3	(11)	(12)	(13)	(14)	(15)
		TOOL	M.TOOL	SAFETY	(OPRT) +
Page 4	(16)	(17)	(18)	(19)	(20)
			LANG.	PROT.	GUARD (OPRT) +

Table 2.4.3 (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.
(8)	OPR (OPERAT PANEL)	Selects the screen for operating some operation switches on the machine operator's panel as soft switches.
(11)	TOOL (TOOL REGIST)	Selects the screen for setting tool data

No.	Chapter menu	Description
(12)	M.TOOL (MULTI TOOL)	Selects the screen for setting multiple tool data
(13)	SAFETY (SAFETY ZONE)	Selects the screen for setting safety zones
(17)	LANG. (LANGUAGE)	Selects the screen for setting a display language.
(18)	PROT. (PROTECT)	Selects the screen for setting data protection.
(19)	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		(OPRT) +
Page 3	(11)	(12)	(13)	(14)	(15)	
		W.DGNS	ALL IO		OPEHIS	+ (OPRT)
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	FL-net	(OPRT) +

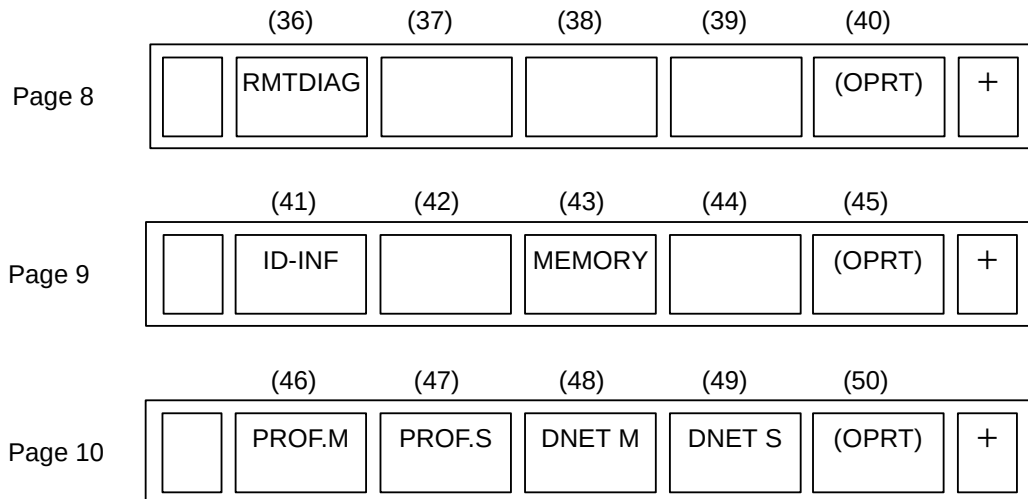


Table 2.4.3 (d) System

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.
(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 MAINTEN)	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.MATE MGR)	Select the screen of Power Mate CNC Manager.
(21)	COLOR (COLOR)	Selects the screen for setting colors to be used on the screen.
(22)	MAINTEN (PERIOD MAINTEN)	Selects the screen for setting maintenance items to be managed periodically.
(23)	M-INFO (MAINTEN 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.
(31)	EMBED (EMBED PORT)	Selects the screen for making settings related to the embedded Ethernet (embedded port).

No.	Chapter menu	Description
(32)	PCMCIA (PCMCIA LAN)	Selects the screen for making settings related to the embedded Ethernet (PCMCIA Ethernet card).
(33)	ETHBRD (ETHNET BOARD)	Selects the screen for making settings related to the fast Ethernet/fast data server.
(34)	FL-net (FL-net)	Selects the screen for making settings related to FL-net.
(36)	RMTDIAG (REMOTE DIAG)	Selects the screen for making settings related to remote diagnosis.
(41)	ID-INF (ID-INF)	Selects the screen for displaying the contents of ID information.
(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 slaver function.
(48)	DNET.M (DevNet MASTER)	Selects the screen for making settings related to the DeviceNet master function.
(49)	DNET.S (DevNet SLAVE)	Selects the screen for making settings related to the DeviceNet 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  and the function of each screen are described below.

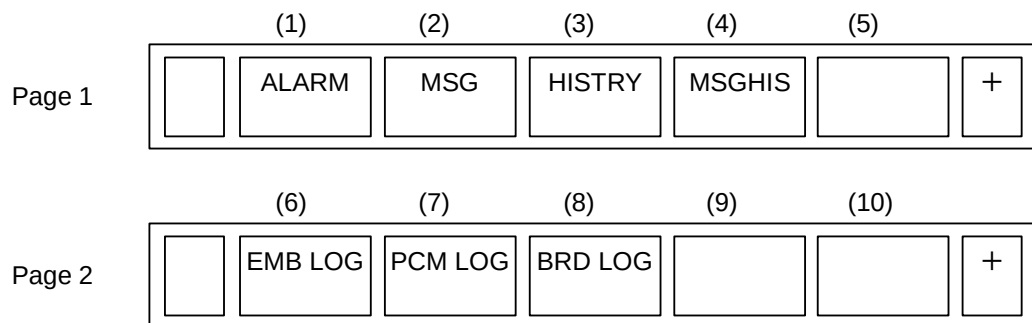


Table 2.4.3 (e) Message

No.	Chapter menu	Description
(1)	ALARM (ALARM)	Selects the alarm message screen.
(2)	MSG (MESSAGE)	Selects the operator message screen.
(3)	HISTRY (HISTORY)	Selects the screen for displaying the details of alarms issued so far.
(4)	MSGHIS (MESSAGE HISTRY)	Selects the external operator message screen.
(6)	EMB LOG (EMBED LOG)	Selects the screen for displaying error messages related to the embedded Ethernet (embedded port).
(7)	PCM LOG (PCMCIA LOG)	Selects the screen for displaying error messages related to the embedded Ethernet (PCMCIA Ethernet card).

No.	Chapter menu	Description
(8)	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:

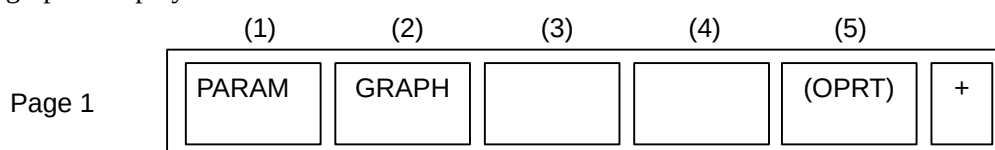


Table 2.4.3 (f) Graphic

No.	Chapter menu	Description
(1)	PARAM (PARAMETER)	Selects the screen for setting graphic parameters.
(2)	GRAPH (GRAPH)	Selects the screen for graphically displaying the tool path.

* The items enclosed by parentheses on the second line under "Chapter menu" are displayed in the 10.4-inch display unit.

2.4.4 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.

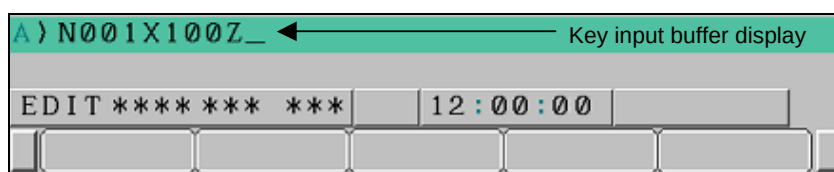


Fig. 2.4.4 (a) Key input buffer display

- 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.

2.4.5 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.

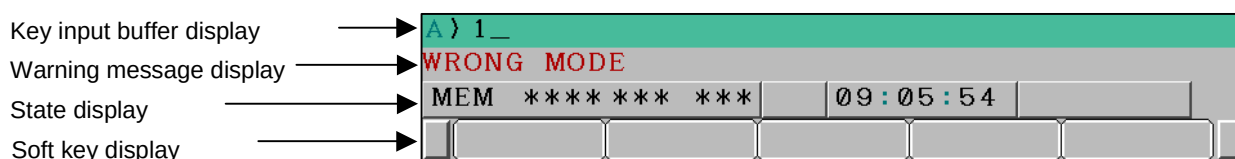


Fig. 2.4.5 (a) Warning message display

Table 2.4.5 (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.

2.5 EXTERNAL I/O DEVICES

External I/O devices such as a memory card are available.

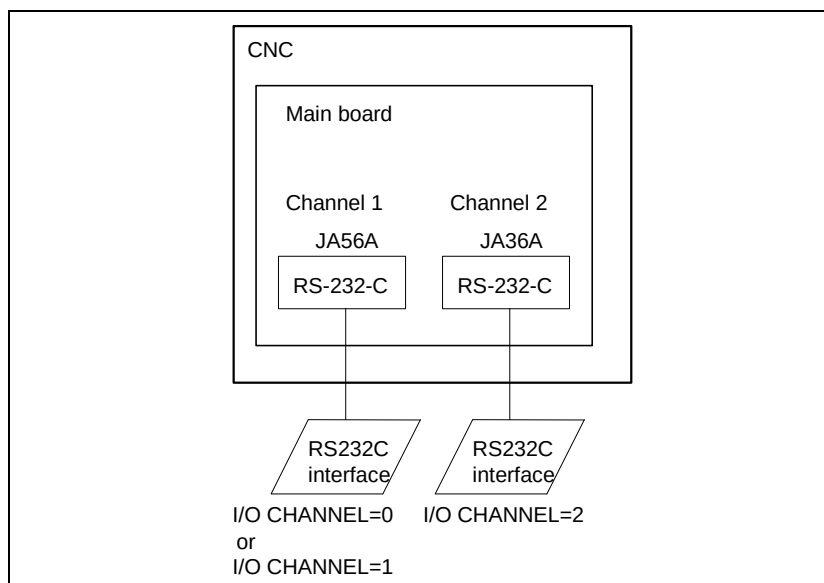
By using an external I/O device such as a memory card, the following data can be input or output:

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 Chapter III-8, "DATA INPUT/OUTPUT".

- Parameter setting

Before an external input/output device can be used, parameters must be set as follows.

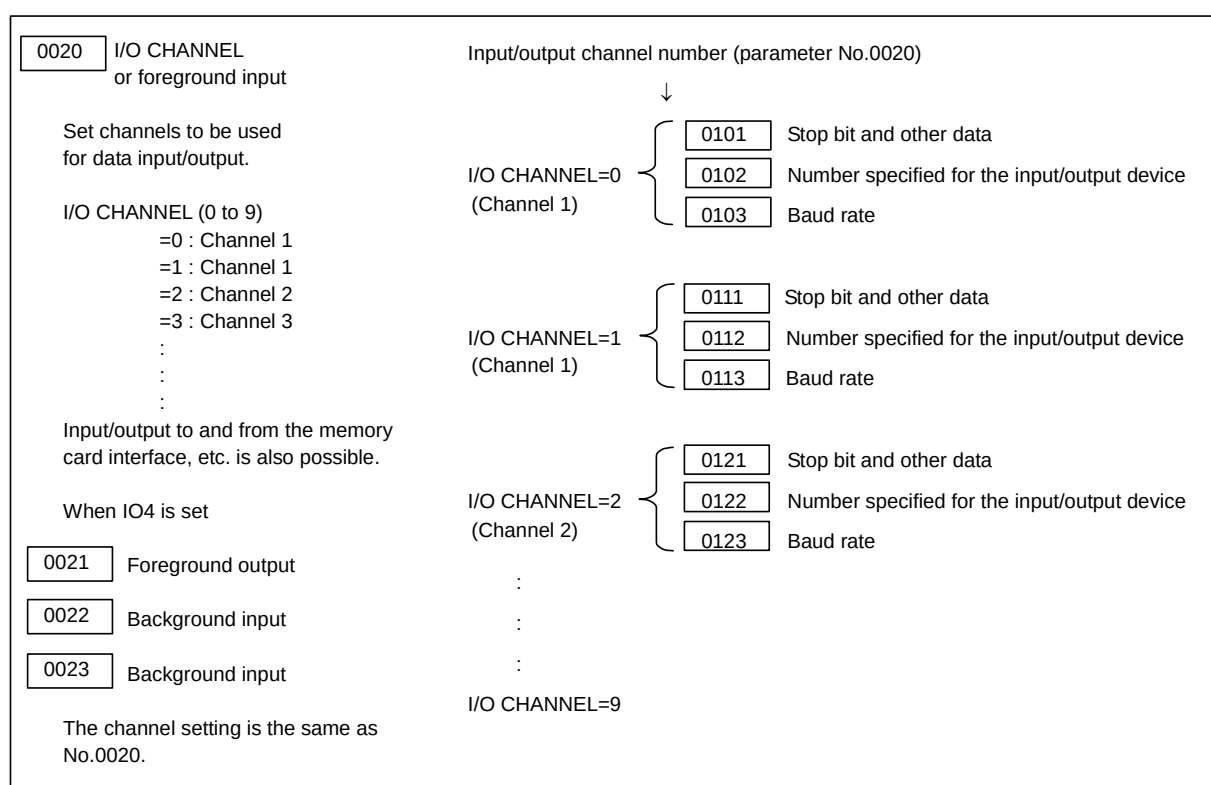


This CNC has a total of two channels of RS232C interfaces. It also has memory card interface. The input/output device to be used is specified by setting the channel (interface) connected to that device in setting parameter I/O CHANNEL.

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 RS232C interface parameters for the channels.



3 MANUAL OPERATION

MANUAL OPERATION are twelve kinds as follows :

3.1	MANUAL REFERENCE POSITION RETURN	362
3.2	JOG FEED (JOG)	363
3.3	INCREMENTAL FEED	364
3.4	MANUAL HANDLE FEED	365
3.5	MANUAL ABSOLUTE ON	368
3.6	DISTANCE CODED LINEAR SCALE INTERFACE	372
3.7	LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)	376

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.

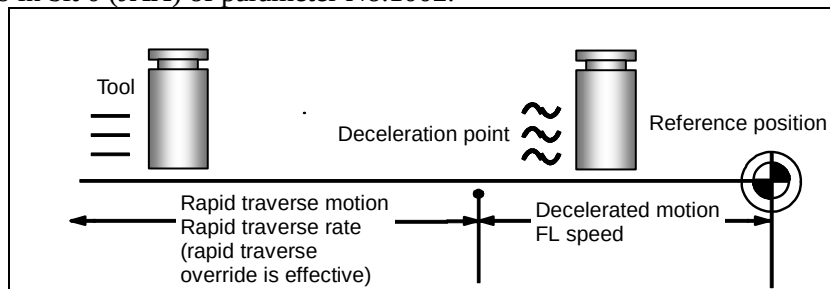


Fig. 3.1 (a) Manual reference position return

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.

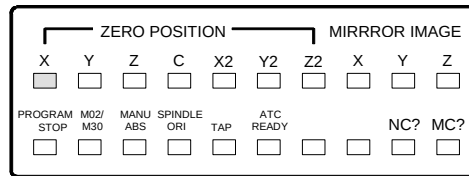


Fig. 3.1 (b)

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 $X = \alpha$, $Y = \beta$, $Z = \gamma$ when reference position return is performed. This has the same effect as specifying the following command for reference position return:

G92X α Y β Z γ ;

However, when the workpiece coordinate system is used (bit 0 (NWZ) of parameter No. 8136 is 0), it is not able to use.

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.

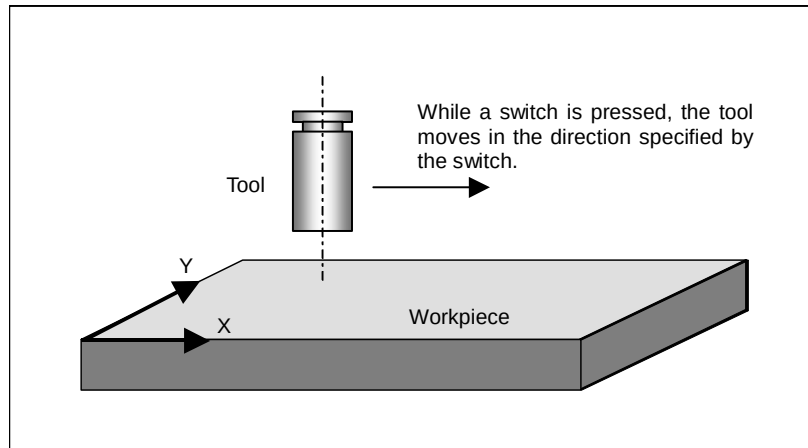


Fig. 3.2 (a) Jog Feed (JOG)

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.

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.

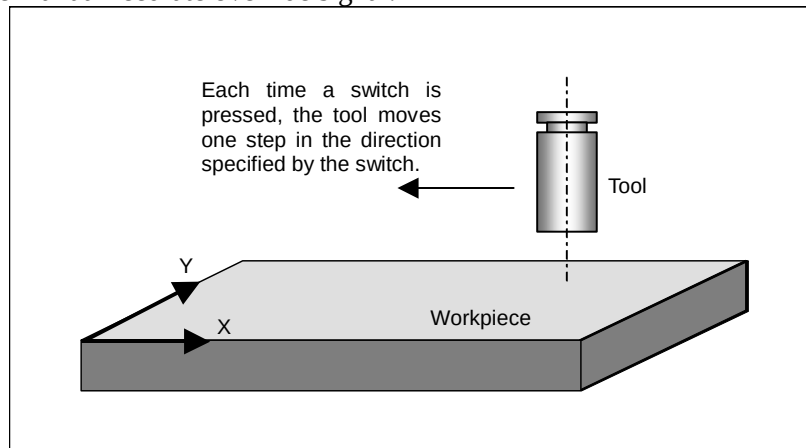


Fig. 3.3 (a) Incremental feed

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.

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. One of four types of magnifiers selected with the manual handle feed move distance select signal can be applied. With bit 2 (HNT) of parameter No. 7103, the minimum distance can be further 10 times greater.

The number of manual pulse generators is up to three. (Three axes can be moved at save time.)

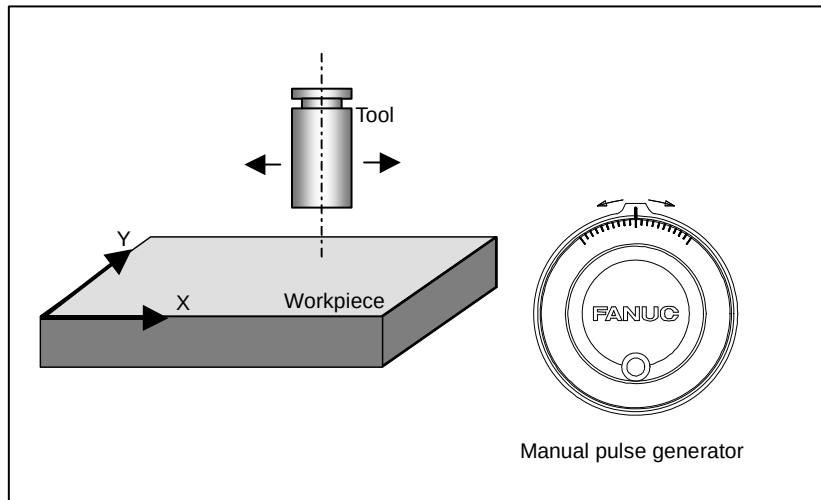


Fig. 3.4 (a) Manual handle feed

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.

- 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.

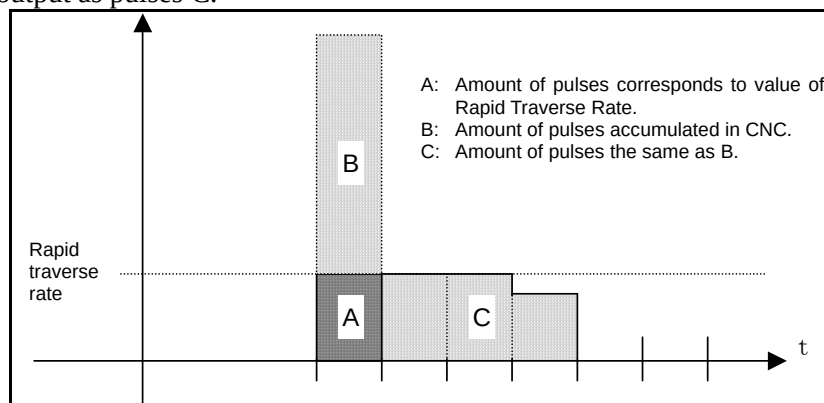


Fig. 3.4 (b) Amount of pulses output by CNC in Manual Handle Feed

Amount of pulses B is calculated in 2 cases as following:

- 1) In case of 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).
- 2) In case of 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 manual handle feed selection signals MP1 and MP2 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 Fig. 3.4 (c), 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.

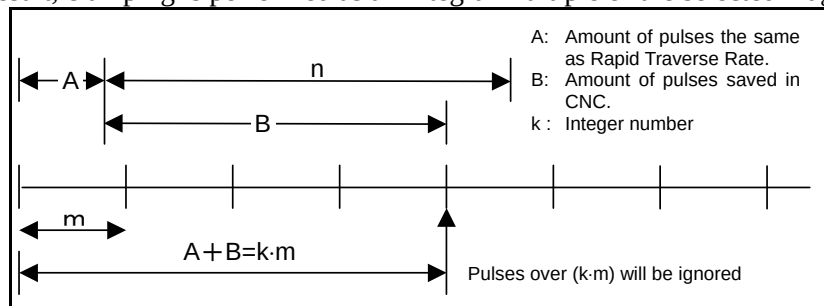


Fig. 3.4 (c) Amount of pulses exceeding the Rapid Traverse Rate ($n \geq m$)

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 $\times 100$ 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

The distance the tool is moved by manual operation is added to the coordinates under the manual absolute on state. And, it is not added to the coordinates under the manual absolute off state.

On the Series 0i-PD, only the manual absolute on state is available.

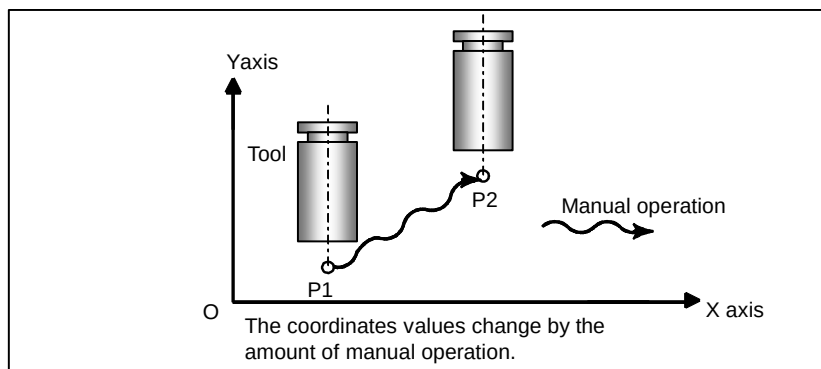


Fig. 3.5 (a) Manual absolute ON

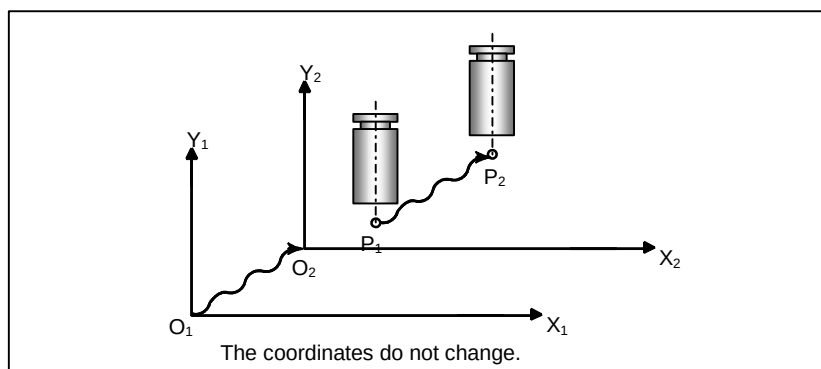


Fig. 3.5 (b) Manual absolute OFF

Explanation

The following describes the relation between manual operation and coordinates under the manual absolute ON state, using a program example.


```

G01G90 X100.0Y100.0F010 ;    <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 under the manual absolute ON

---→ Movement of the tool under the manual absolute 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>.

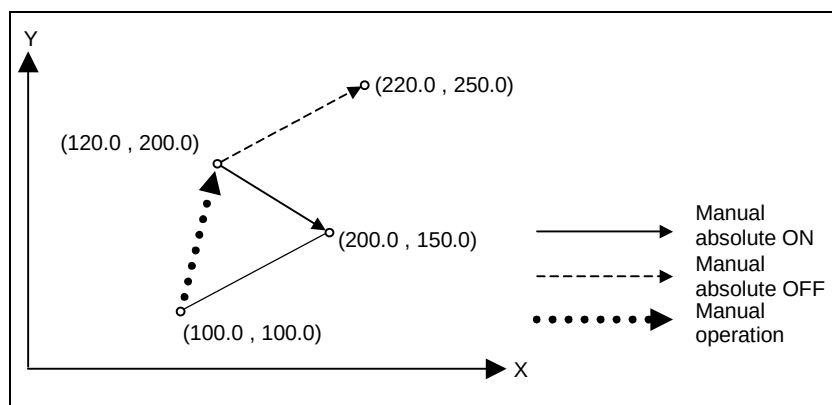


Fig. 3.5 (d) Manual operation after the end of block

- 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.

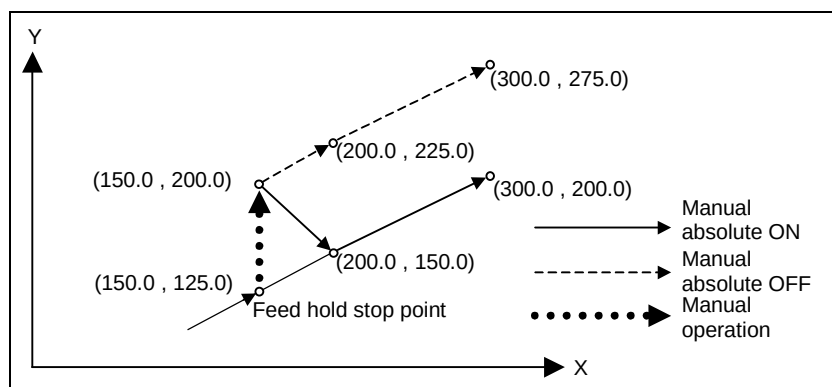


Fig. 3.5 (e) Manual operation after a feed hold

- 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.

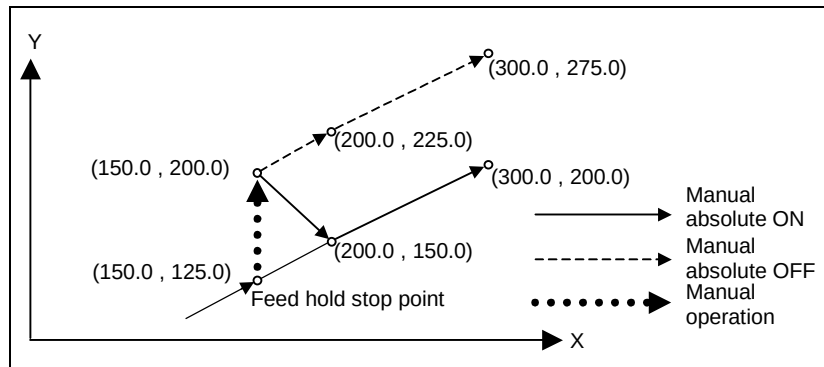


Fig. 3.5 (f) When reset after a manual operation following a feed hold

- **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.

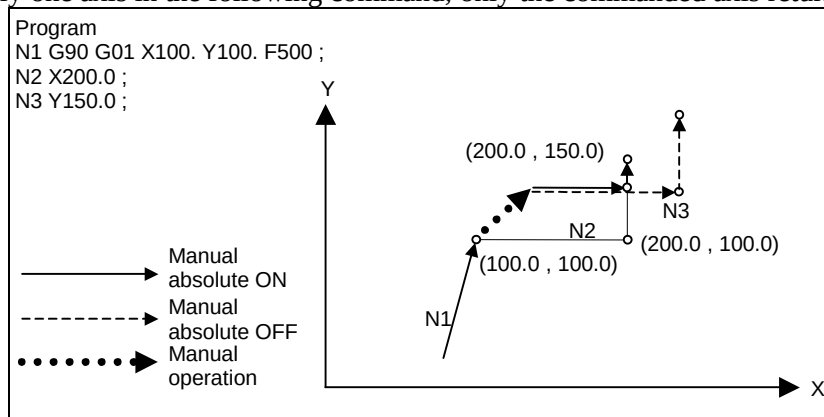


Fig. 3.5 (g) When a movement command in the next block is only one axis

- **When the next move block is an incremental**

When the following commands are incremental commands, operation is the same as the manual absolute OFF.

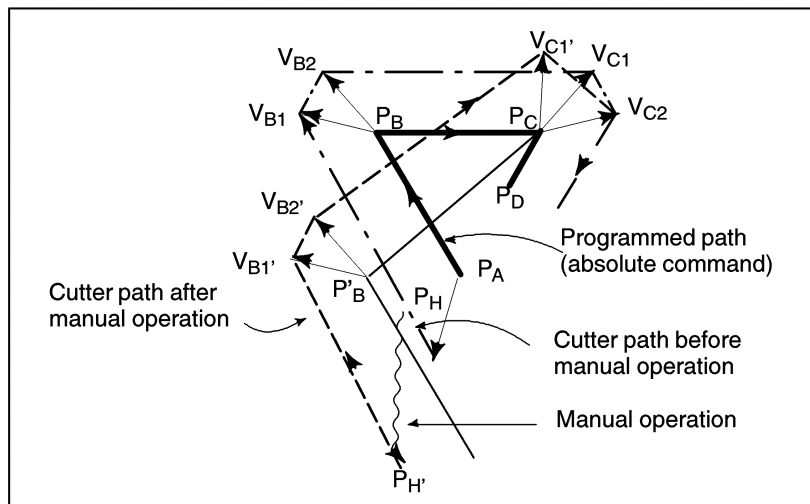
- **Manual operation during cutter radius compensation**

• Manual absolute ON during cutter radius compensation

Operation of the machine upon return to automatic operation after manual intervention with the manual absolute ON during execution with an absolute command program in the cutter 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.

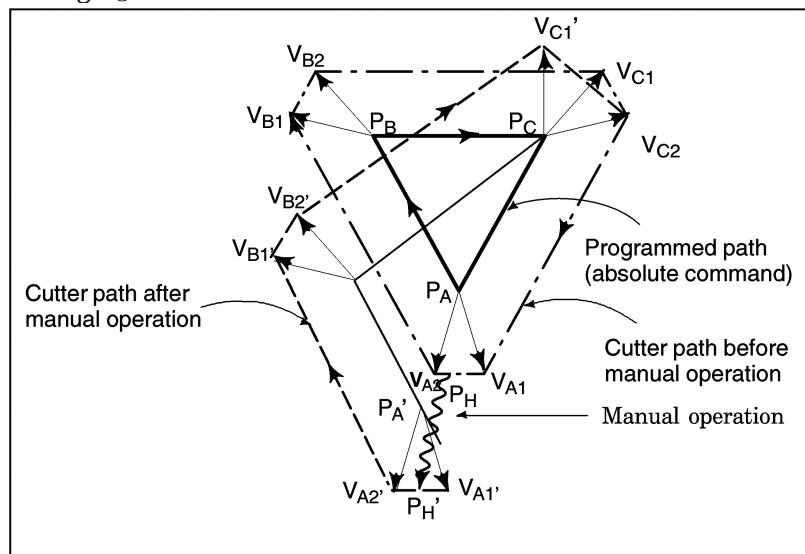
• Manual operation performed in other than cornering

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 P_H' . 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 .



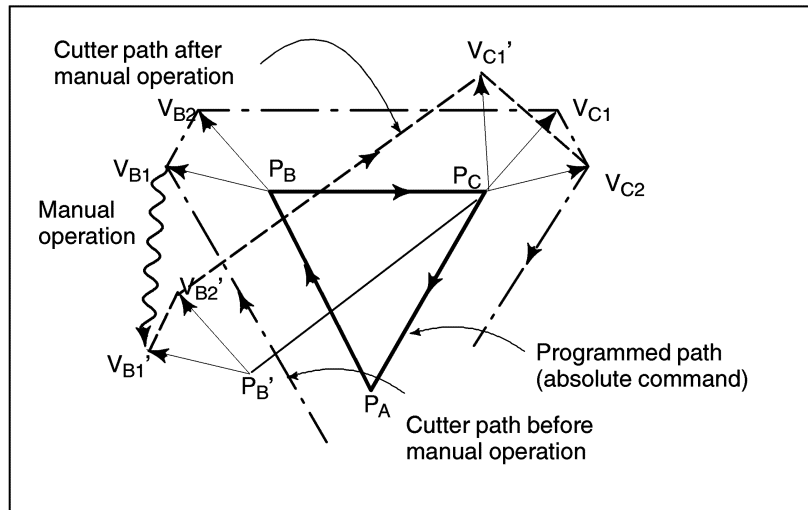
- Manual operation during cornering

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 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.

**NOTE**

Only manual absolute ON can be used by Series 0i-PD. Manual absolute on/off operation cannot be switched.

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.

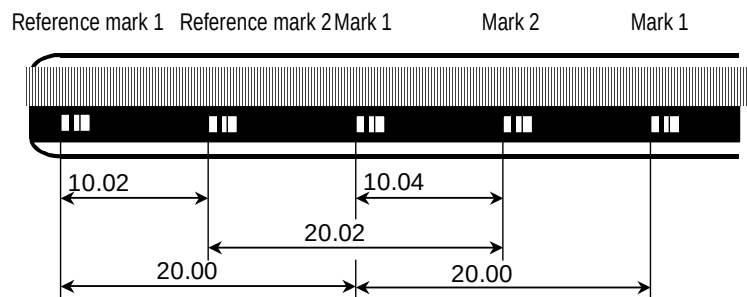


Fig. 3.6 (a) Example of distance coded linear scale

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.

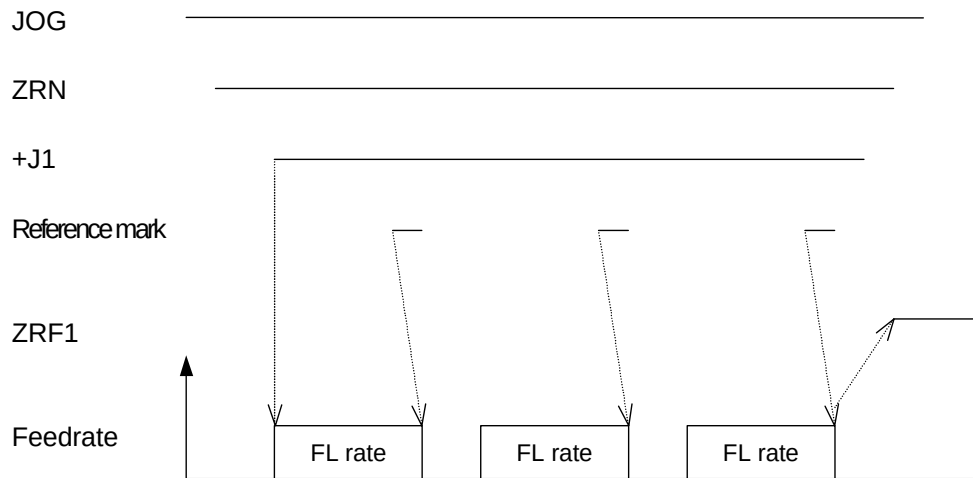


Fig. 3.6.1 (a) Timing chart for reference position establishment

- 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.

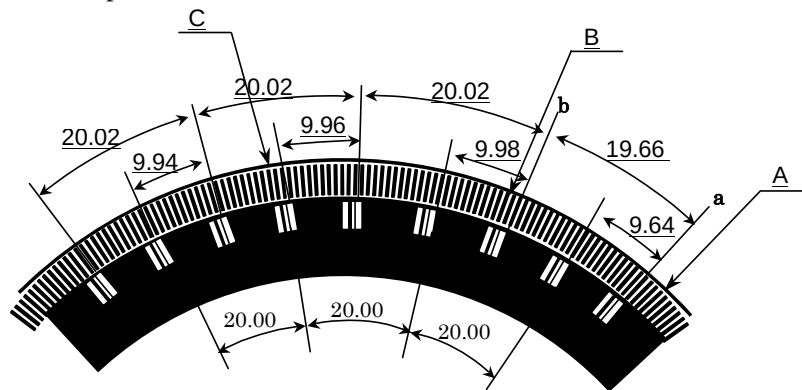
3.6.2 Reference Position Return

- (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 bit 0 (RFS) of parameter No. 1818.
- (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 the reference point return procedure is executed, the coordinate value are rounded in 0 to 360 degree, even if Machine coordinate values are linear axis type.
- In case of distance coded rotary encoder, only the measurement by three points or four points is possible. (bit 2 (DC2) of parameter No.1802 is disregarded as 0.)

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, "ILLEGAL SYNCHRONOUS AXIS" 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, SYNC2, ...) is 1. (During establishment of a reference position, the synchronization select signal status must be maintained.)

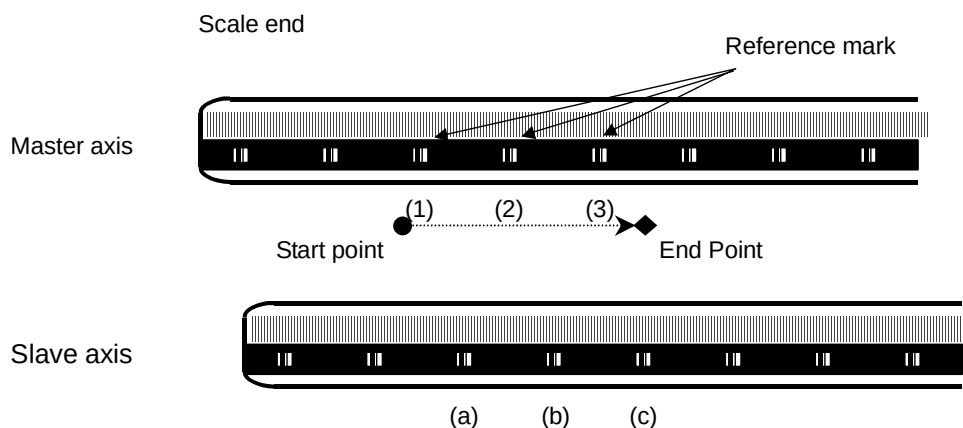
Reference position establishment with axis synchronization control axes

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, ... 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 the above example, the following sequence is executed.

- When the reference mark (1) of the master axis is detected, both master axis and slave axis stop.
- Both the axes begin to move again at a reference position return FL feedrate.
- When the reference mark (a) of the slave axis is detected, both axes stop again.
- Both the axes begin to move again at FL feedrate.
- Both axes repeat the operation until all point ((2) -> (b) -> (3) -> (c)) are detected.
- When the slave axis detects the third reference mark (c), both the axes end the reference position establishment.

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, ... 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 Note

- (1) In the case of the actual interval of reference marks is different from parameter setting value, the alarm DS1449, "REFERENCE MARKS ARE DIFFERENT FROM PARAMETER" 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 Nos. 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".

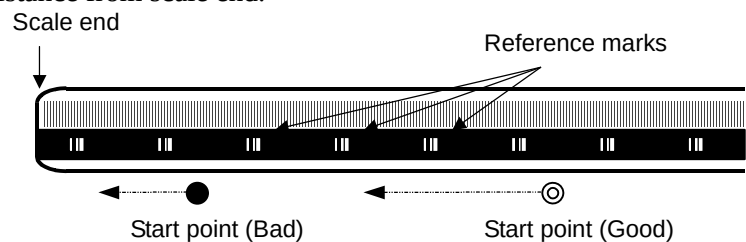
To use such a scale, please adjust the detection unit by modification of parameters No.1820 (CMR) and Nos.2084/2085(flexible feed gear) to make the difference of parameters Nos.1821 and 1882 more than 4 as following examples.

- (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.

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

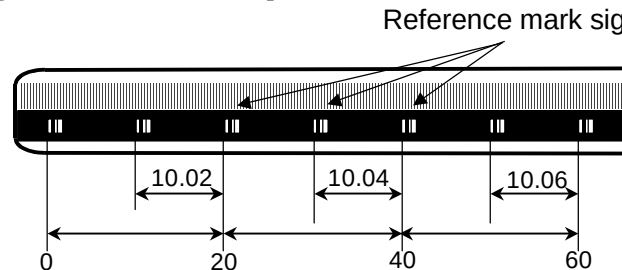
Overview

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.



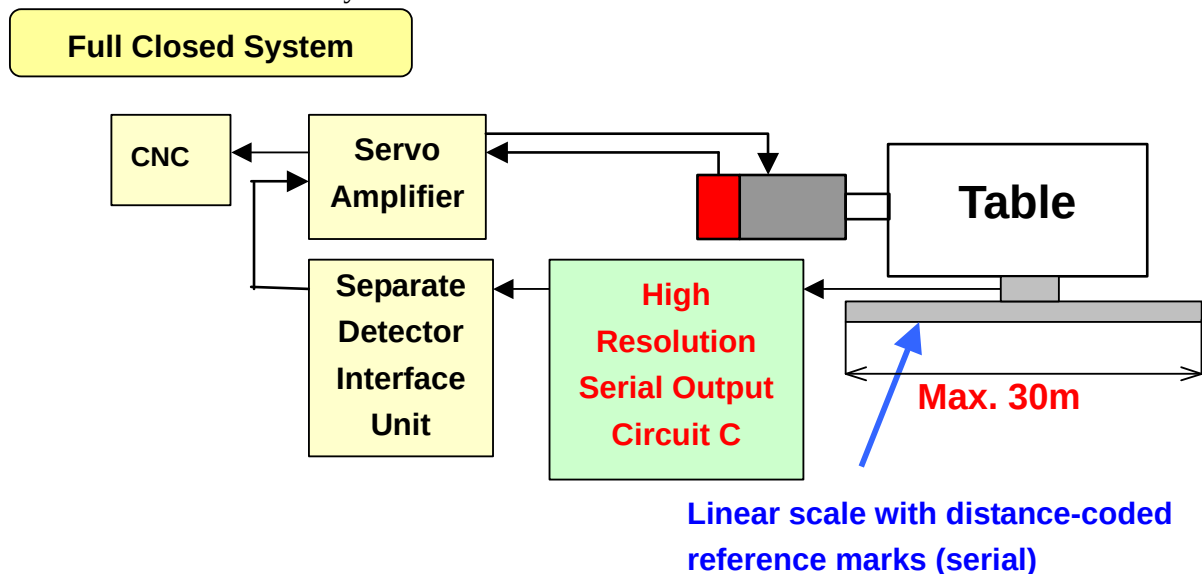
The CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position, because of the interval of each reference mark is different with regular interval.

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

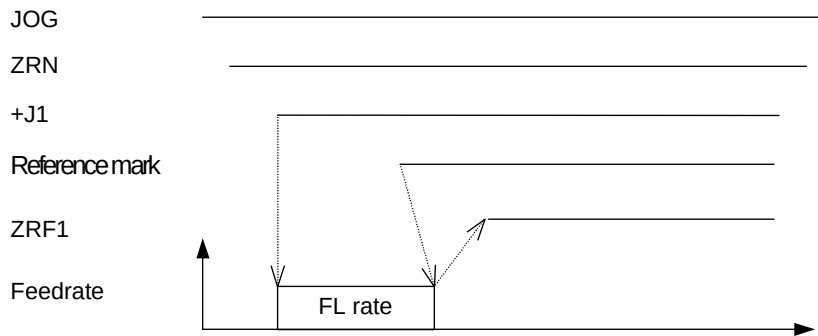
It is available under full closed system.



- 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 bit 0 (RFS) of parameter No. 1818.

- 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.)

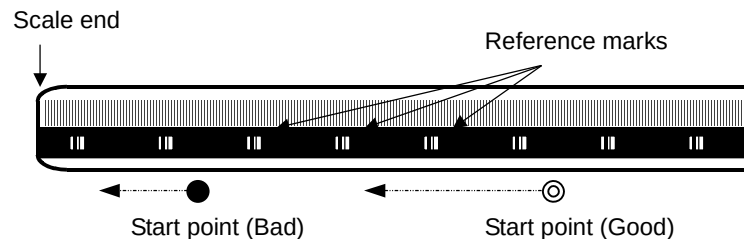
- To the parameters, which relate to this function (except No.1883, No.1884), the same value must be set for the master axis and for the slave axis.
- The linear scale with distance-coded reference marks (serial) should be applied for the master axis and the slave axis.
If either of the master axis or the slave axis is not the linear scale with distance-coded reference marks (serial), alarm DS0018, "SERIAL DCL:MISMATCH(SSYNC CTRL)" occurs when reference position establishment is tried.
- During operating the establishment of reference position, the state of signal (SYNCn or SYNCJn) for selecting synchronized axis 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".

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.



4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation. The chapter 4 explains the following types of AUTOMATIC OPERATION:

4.1	MEMORY OPERATION	380
	Operation by executing a program registered in CNC memory	
4.2	MDI OPERATION	382
	Operation by executing a program entered from the MDI unit	
4.3	DNC OPERATION.....	385
	Function for executing a program while reading the program from an input device connected to the RS232C interface or a memory card.	
4.4	SCHEDULE OPERATION	387
	Function for executing a program while reading the program from an input device connected to the RS232C interface or a memory card according to a schedule.	
4.5	EXTERNAL SUBPROGRAM CALL (M198).....	391
	Function for calling and executing subprograms (files) registered in an external input/output device during memory operation	
4.6	MANUAL HANDLE INTERRUPTION.....	393
	Function for performing manual feed during movement executed by automatic operation	
4.7	MIRROR IMAGE.....	399
	Function for enabling mirror-image movement along an axis during 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 unit is pressed, automatic operation terminates and the reset state is entered.

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Memory operation

Procedure

- 1 Press the MEMORY mode selection switch.
- 2 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.
- 3 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.
- 4 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 unit.
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  key on the MDI unit 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

A block skip slash (/) must be specified at the beginning of a block.

Note that if a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

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 unit.

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.
- 2 Press the  key to select the program screen. The following screen appears:

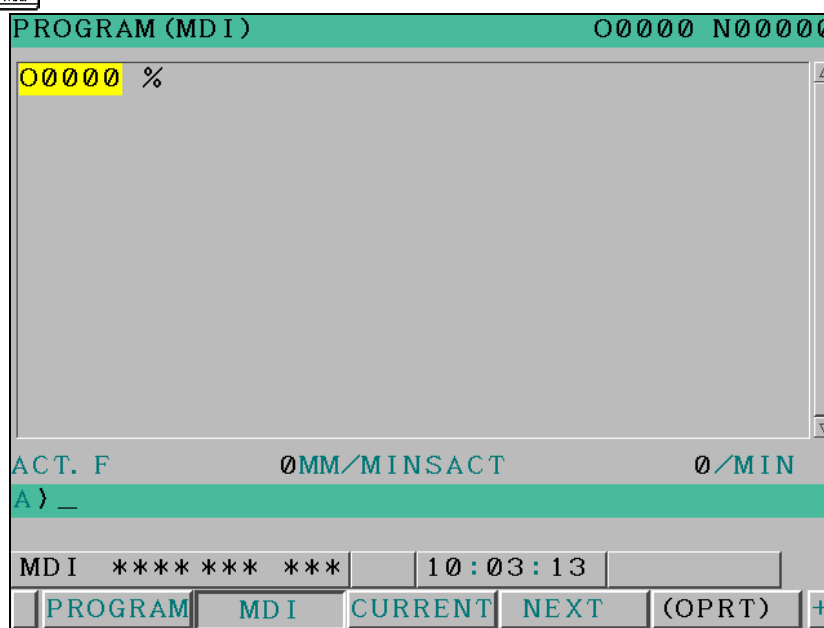


Fig. 4.2 (a) MDI program screen

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 bit 7 (MCL) of parameter No. 3203 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 ER(%) 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  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 ER(%) 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  and  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 G90Y200.0 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.

Bit 4 (MAB) of parameter No. 3401=0	Bit 4 (MAB) of parameter No. 3401=1	
	Bit 5 (ABS) of parameter No. 3401=0	Bit 5 (ABS) of parameter No. 3401=1
Absolute mode operation with G90 command, and incremental programming operation with G91 command	Incremental mode operation at all times, independent of G90/G91 command	Absolute mode operation at all times, independent of G90/G91 command

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, 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, "ILLEGAL MODE FOR GOTO/WHILE/DO" 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.)

MEMORY CARD		N00000	
DNC FILE	00123		
REG NUM	60		
DEVICE : M_CARD			
NO.	FILE NAME	SIZE (KBYTE)	UPDATE
D0017	00123	1	2008/01/23
0018	00567	1	2008/02/06
0019	01000	1	2008/01/30
0020	01001	1	2008/01/31
0021	01002	1	2008/01/31
0022	01003	1	2008/01/31
0023	01234	1	2008/02/19
0024	02222	1	2008/01/28
A) _			
RMT	**** **	10:07:34	
[<] DEVICE	[REFRESH]	[DNC SET]	[DNC CLR] [+]

- 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.

PROGRAM		O0123 N00456	
BC:			
O0123 ;			
N456 G90 G00 X-554378.000			
Y-154363.000 T10 ;			
X-554370.000 Y-154363.00 ;			
N622 G00 X-55378 Y-1543632 ;			
X-553484 Y-153996 C123.000 ;			
Y-151521 ;			
Y-152898 ;			
X-554378 Y-149452 ;			
X-554373 ;			
Y-154363 ;			
A) _			
		S	0 T0010
RMT	STRT	MTN	FIN 12:00:00
PROGRAM	CHECK	CURRENT	NEXT (OPRT) +

Fig. 4.3 (a) PROGRAM screen

PROGRAM (CHECK)		O0123 N00456	
O0123 ;			
N456 G90 G00 X-554378.000			
Y-154363.000 T10 ;			
X-554370.000 Y-154363.00 ;			
	ABSOLUTE	DISTANCE	G00 G94 G67
X	-21225.600	-533152.400	G17 G21 G54
Y	-21225.600	-133137.400	G90 G40 G64
T	0.000	0.000	G22 G50 G85
C	0.000	0.000	
			H M
T	10		D
F		S	
ACT. F	19395MM/MIN	SACT	0/MIN
A) _			
		S	0 T0010
RMT	STRT	MTN	FIN 12:00:00
ABS	REL		(OPRT)

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, “CAN NOT COMMAND M198/M99” 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, “ILLEGAL MODE FOR GOTO/WHILE/DO” 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.

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.

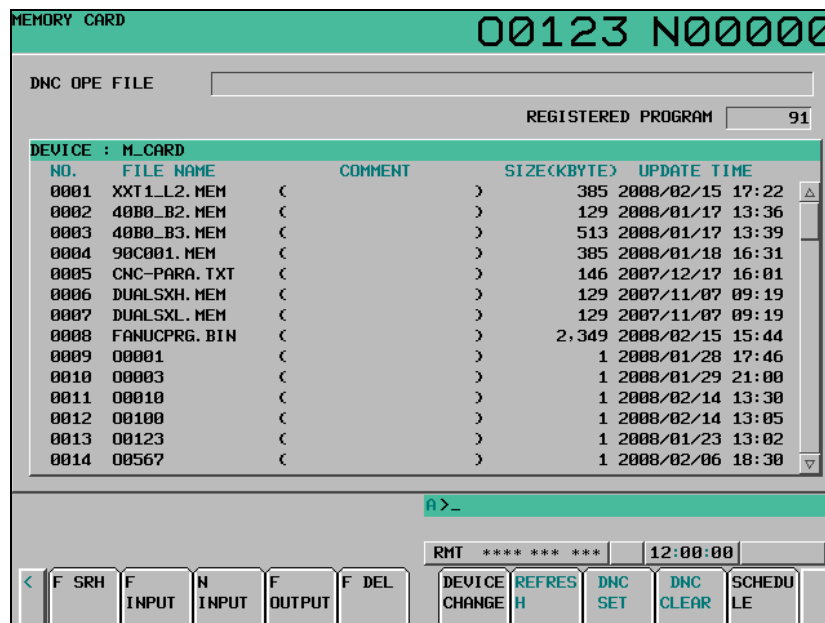


Fig. 4.4 (a) Program list screen (10.4-inch)

[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.

SCHEDULE LIST					00001 N00001	
NO.	FILE NO	FILE NAME	REQ.	CUR.		
1	0009	00001	1	0		
2	0013	00123	1	0		
3			0	0		
4			0	0		
5			0	0		
6			0	0		
7			0	0		
8			0	0		
9			0	0		
10			0	0		
11			0	0		
12			0	0		
13			0	0		
14			0	0		
15			0	0		
16			0	0		

A>_

RMT ***** 16:31:12

< FILE UP FILE DOWN DELETE INSERT ALL DELETE

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.

[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.

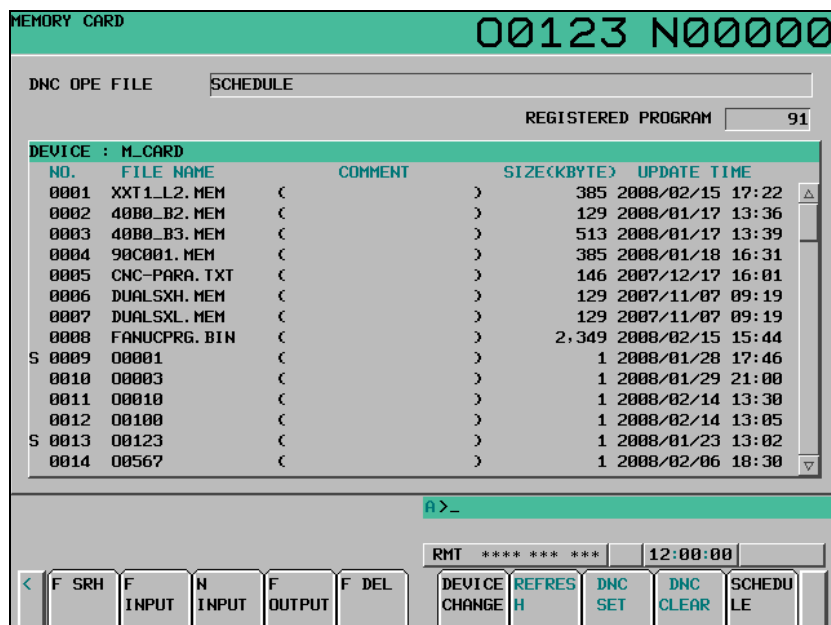


Fig. 4.4 (g) Program list screen (after setting schedule data) (10.4-inch display unit)

- 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

- Before setting schedule operation, release DNC operation files in the MDI mode. DNC operation and schedule operation cannot be specified at the same time.
- Before starting schedule operation, confirm that schedule data is set correctly on the schedule list screen.
- 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 folder display during execution of a file**
During schedule operation, folders 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.

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 Pxxxxxxx Lyyyyyyy ;

↑ ↑

Pxxxxxxx : Program number (or file number)
Lyyyyyyy : Number of repetitive calls

• **If calling with a file name**

M198 <xxxx> Lyyyyyyy ;

↑ ↑

xxxx : File name of an external subprogram
Lyyyyyyy : Number of repetitive calls

When address L is omitted, the number of repetitive calls is assumed to be 1.

FS0i-PC compatible command format

(The following command format is valid only when a 4-digit program number is used.)

M198 Pxxxx yyyv ;

↑ ↑

xxxv : Number of repetitive calls
yyv : 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.)

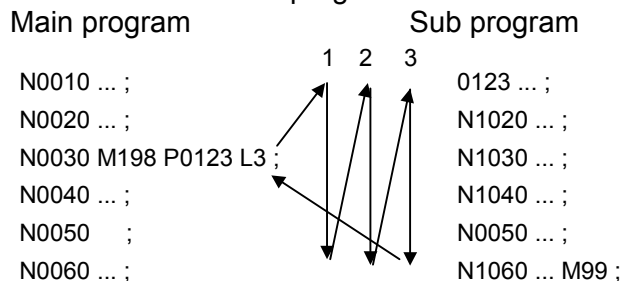
Specify a program number (file number) registered in an external device at address P. If the specified program number (file number) is not registered in the connected external device, an alarm PS1079, "PROGRAM FILE NOT FOUND" is issued. (When the memory card is used as external device, alarm (SR1966) is issued.)

Example)

M198 P0123 L3;

This command specifies that the subprogram having external subprogram number 00123 is to be called three times repeatedly.

The subprogram is called from the main program and executed as follows:



- 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

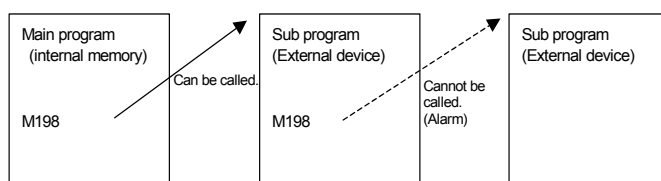
- 1 An external subprogram call can be specified during program operation in the MEM mode or MDI mode. To make an external subprogram call in the MDI mode, set bit 1 (MDE) of parameter No. 11630 to 1.
- 2 An external subprogram call is available for the following external devices:

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.

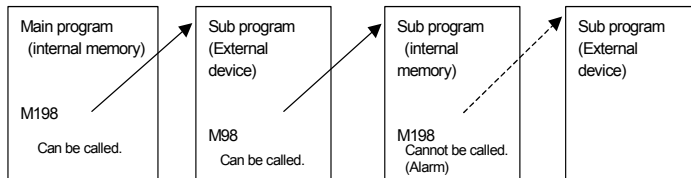
A program number call is always enabled regardless of the setting of bit 2 (SBP) of parameter No. 3404.

- 4 To perform a file name call using a Handy File or floppy cassette as the external device, set bit 2 (SBP) of parameter No. 3404 to 1.
- 5 called using another external device subprogram call. (An alarm PS1080, "DUPLICATE DEVICE SUB PROGRAM CALL" is issued.)



NOTE

- 6 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.)



- 7 A call using the external device subprogram call function is counted as one level of subprogram nesting.
- 8 In a multi-path system, an external device subprogram call cannot be performed simultaneously from multiple paths.
When external subprogram calls using the Data Server that are available in multi-path systems are used, however, external subprogram call commands using the Data Server can be specified simultaneously from multiple 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 signals HSnIA to HSnIE.

The minimum unit of travel distance per scale division is the least input increment. One of four types of magnifiers selected with manual handle feed selection signals MP1 and MP2 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-3.4, "MANUAL HANDLE FEED".)

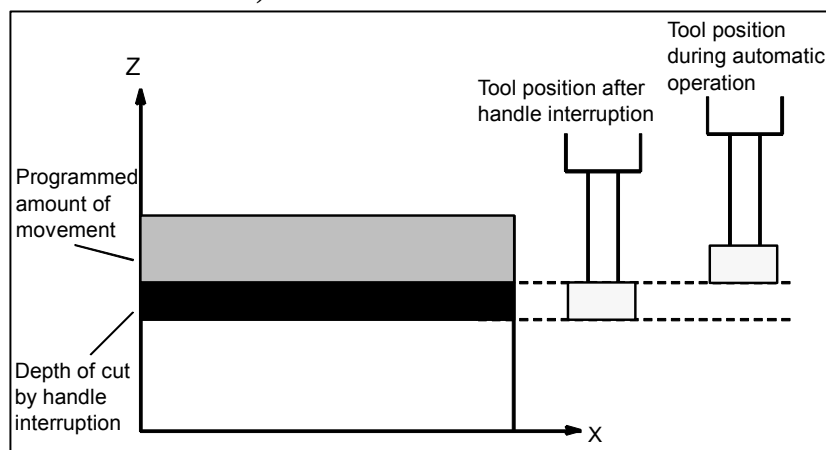


Fig. 4.6 (a) Manual handle interruption

**WARNING**

The travel distance per scale division by manual handle interruption is the least input increment as with manual handle feed. With a machine of metric input and inch output, for example, the travel distance per 254 scale divisions is 0.01 inch. With a machine of inch input and metric output, the travel distance per 100 scale divisions is 0.254 mm.

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

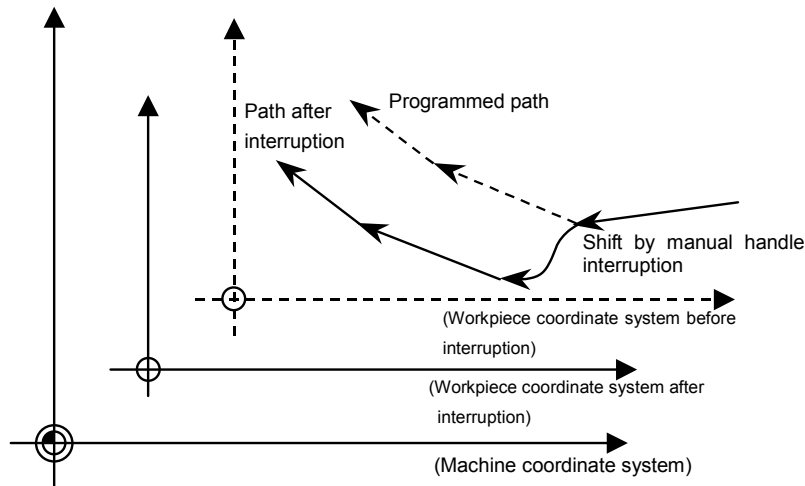
Suppose that the maximum allowable cutting feedrate for an axis is 5 m/min, and that a movement is made in the + direction at 2 m/min along the axis. In this case, manual handle interruption can be accepted even when the manual pulse generator is rotated up to a speed equivalent to 3 m/min. Manual handle 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.

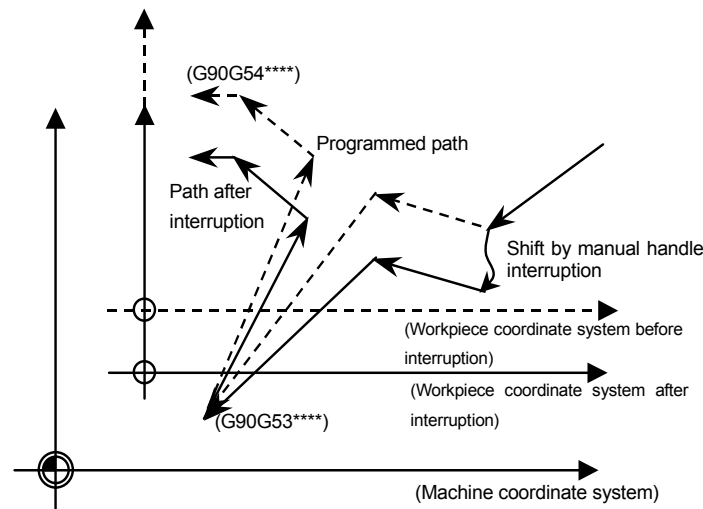
- Manual handle interruption and coordinate system

- 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.



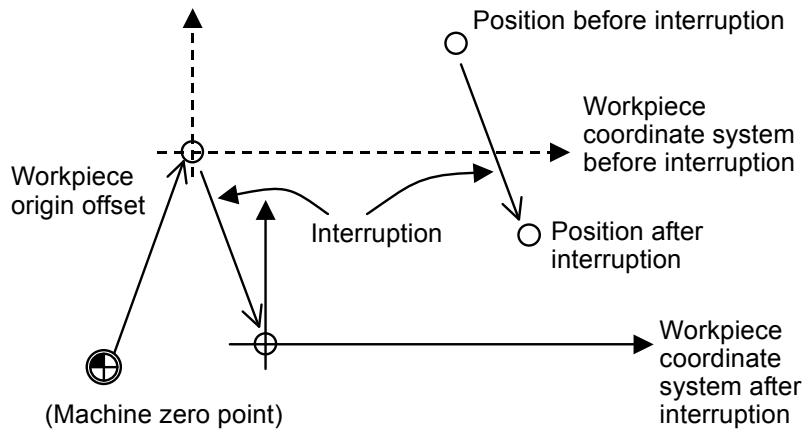
- 3 In automatic reference position return (G28), the end point (reference position) is not affected by manual handle interruption. However, the midpoint is in the workpiece coordinate system, so that the position shifted by the amount of interruption becomes the midpoint.

- Cancellation of the amount of interruption

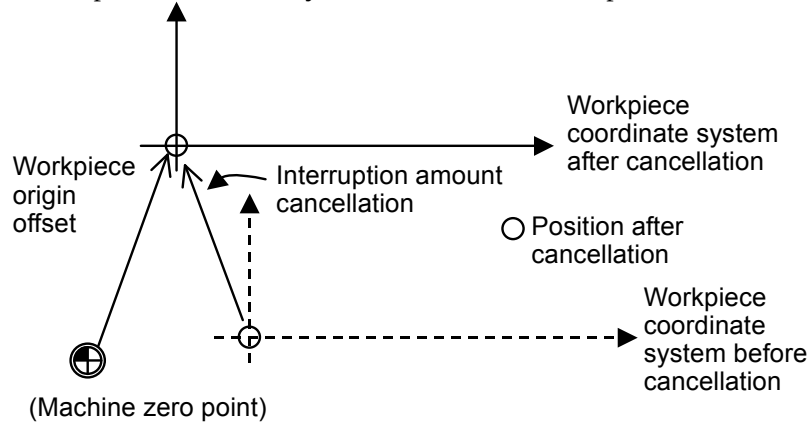
Operation by which the workpiece coordinate system shifted by manual handle interruption from the machine coordinate system is returned to the original workpiece coordinate system is referred to as cancellation of the amount of interruption.

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 for the amount of manual handle interruption.

When bit 3 (HLC) of parameter No. 7100 is set to 1, soft key [INTRPT CANCEL], which is used for this operation, appears. When HLC is set to 0, 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 unit.
- 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.

Table 4.6 (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 , 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.

HANDLE INTERRUPT				00123 N00000			
INPUT UNIT				OUTPUT UNIT			
X	0.000			X	0.000		
Y	0.000			Y	0.000		
T	0.000			T	0.000		
C	0.000			C	0.000		
RELATIVE				DISTANCE TO GO			
X	0.000			X	0.000		
Y	0.000			Y	0.000		
T	0.000			T	0.000		
C	0.000			C	0.000		
				PARTS COUNT			
RUN TIME				CYCLE TIME			
0H 5M				0H 0M 4S			
ACT. F				SACT			
0MM/MIN				0/MIN			
A) _				S 0 T0010			
MEM **** **				12:00:00			
ABS				REL			
ALL				HANDLE			
				(OPRT)			

Fig. 4.6 (b) Manual handle interruption screen

- (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
These values have no effect on 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.

- Display for five-axis systems or better

Systems having five or more axes provide the same display as the overall position display.

Note

NOTE

- 1 In a manual operation mode such as the jog feed mode or manual handle feed 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 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 unit.

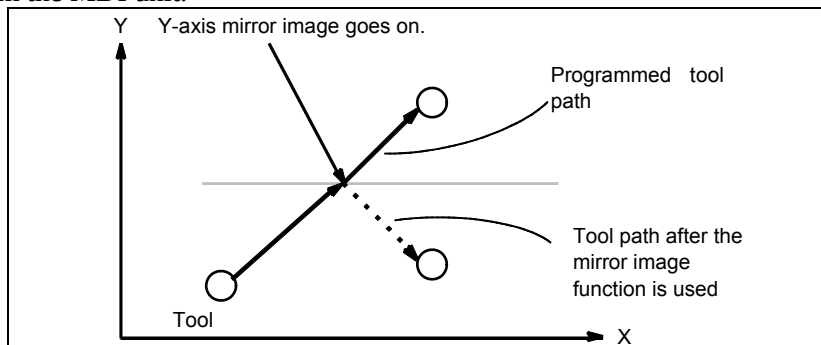


Fig. 4.7 (a) Mirror image

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 [SETTING] soft key for chapter selection to display the setting screen.

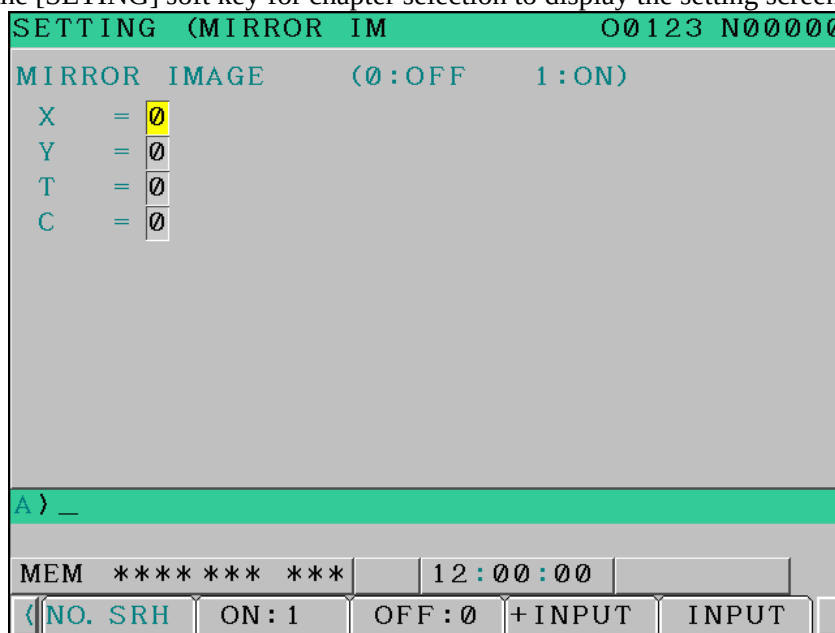


Fig. 4.7 (b) Setting screen

- 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

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) cannot be reversed.

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.....	401
5.2	FEEDRATE OVERRIDE	402
5.3	RAPID TRAVERSE OVERRIDE	403
5.4	DRY RUN.....	404
5.5	SINGLE BLOCK	405
5.6	PUNCH	406
5.7	MANUAL PUNCH.....	407

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.

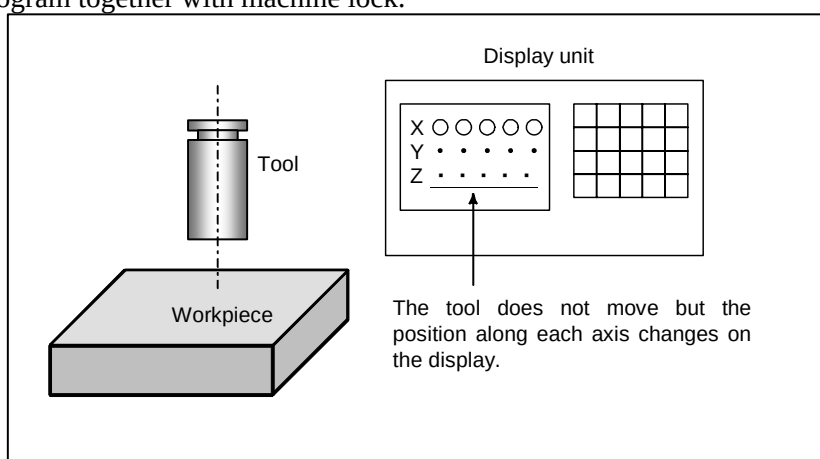


Fig. 5.1 (a) Machine lock

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 G28 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

The following M codes are executed even in the auxiliary function lock state:

- M00, M01, M02, M30
- M98, M99
- M198 (external subprogram call function)
- M codes for calling a subprogram (parameters Nos. 6071 to 6079, Nos. 6044 to 6046)
- M codes for calling a custom macro (parameters Nos. 6080 to 6089, Nos. 6047 to 6049)

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.

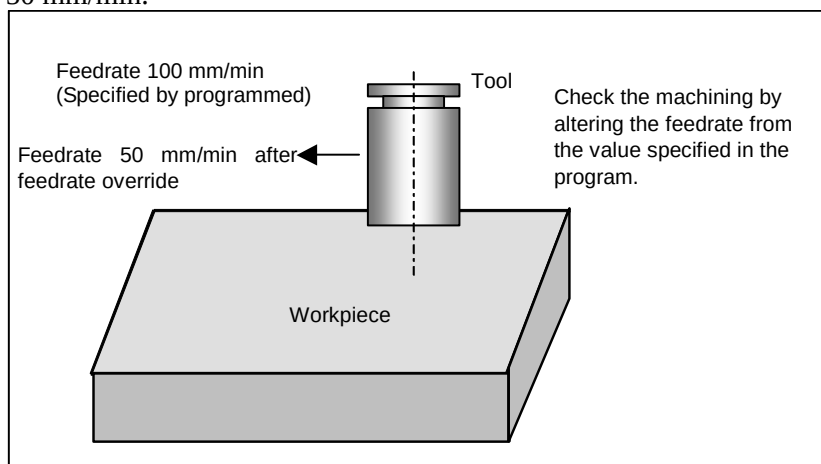


Fig. 5.2 (a) Feedrate override

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.

5.3 RAPID TRAVERSE OVERRIDE

An override of four steps (25%, 50%, 75%, 100%) can be applied to the rapid traverse rate.

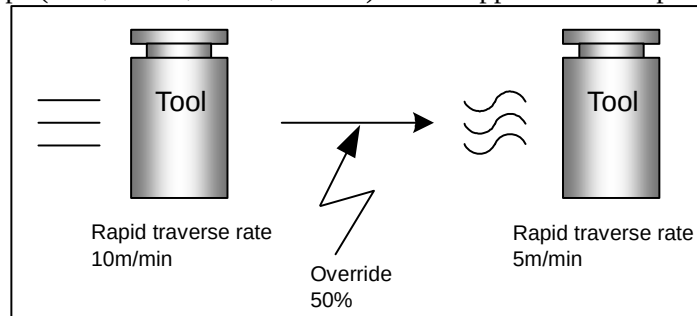


Fig. 5.3 (a) Rapid traverse override

Rapid traverse override

Procedure

Select one of the four feedrates with the rapid traverse override switch during rapid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Rapid traverse override switch on the machine operator's panel	Rapid traverse rate	
	Excluding T, C axes	T, C axes
100	100%	100%
75	75%	100%
50	50%	50%
25	25%	50%

When the bit (G0F) of parameter No. 16050 is set to 0, the number of one digit from 1 to 4 after F code stated allows the rapid traverse rate in automatic operation is overridden.

One-digit F feed code	Override by one-digit F feed code
F1	100%
F2	75%
F3	50%
F4	25%

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 using pattern function (G26, G76, G77, G78, or G79)
- (3) Rapid traverse during positioning first press point in nibbling mode (G68, G69, or, nibbling by M code).
- (4) Rapid traverse by positioning and no press start (G70)
- (5) Rapid traverse during using automatic repositioning (G75).
- (6) Rapid traverse during using automatic Reference Position Return (G28).

For the T-axis and C-axis, the feedrate is 100% of the rapid traverse rate when the rapid traverse override switch is set to 100 or 75, and, the feedrate is 50% of the rapid traverse rate when the switch is set to 50 or 25. However, the override is always set to 100% when bit 1(OVE) of parameter No.16844 is set to 1.

⚠ CAUTION

- 1 For the manual rapid traverse and rapid traverse in manual reference point return, the rapid traverse override function is ineffective.
- 2 The rapid traverse rates in manual and automatic operation modes are separately settable for each axis.
- 3 For the rapid traverse attained to each pitch from the first punch point to the last punch point in nibbling function, the rapid traverse override is ineffective, and it is always fixed to 100%.
- 4 Since whether the rapid traverse override is effected or not is judged when data are read from the reader/puncher interface or memory into buffer storage, the traverse rate in the block being executed or the block already in the buffer storage remain unchanged even when the rapid traverse override switch has been changed.
- 5 Rapid traverse rate by F code cannot be used together with rapid traverse override by F1-digit command

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.

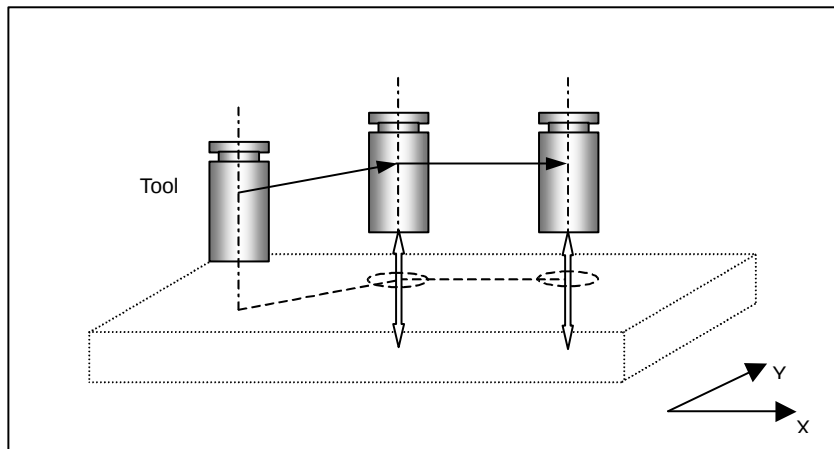


Fig. 5.4 (a) Dry run

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 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 5.4 (a) according to the rapid traverse switch and parameters.

Table 5.4 (a)

Rapid traverse switch	Program command	
	Rapid traverse	Cutting feed
ON	Rapid traverse rate	Dry run feedrate $\times$ Jvmax ^(*2)
OFF	Dry run feedrate $\times$ JV, or rapid traverse rate ^(*1)	Dry run feedrate $\times$ 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 $\times$ JV when bit 6 (RDR) of parameter No. 1401 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.

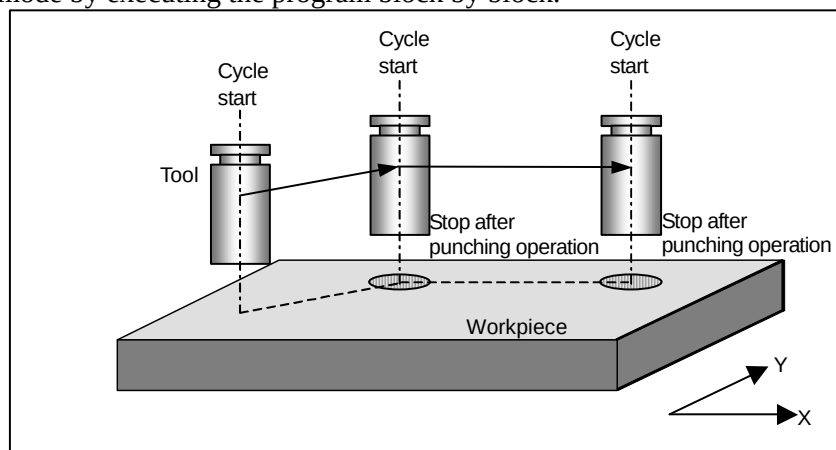


Fig. 5.5 (a) Single block

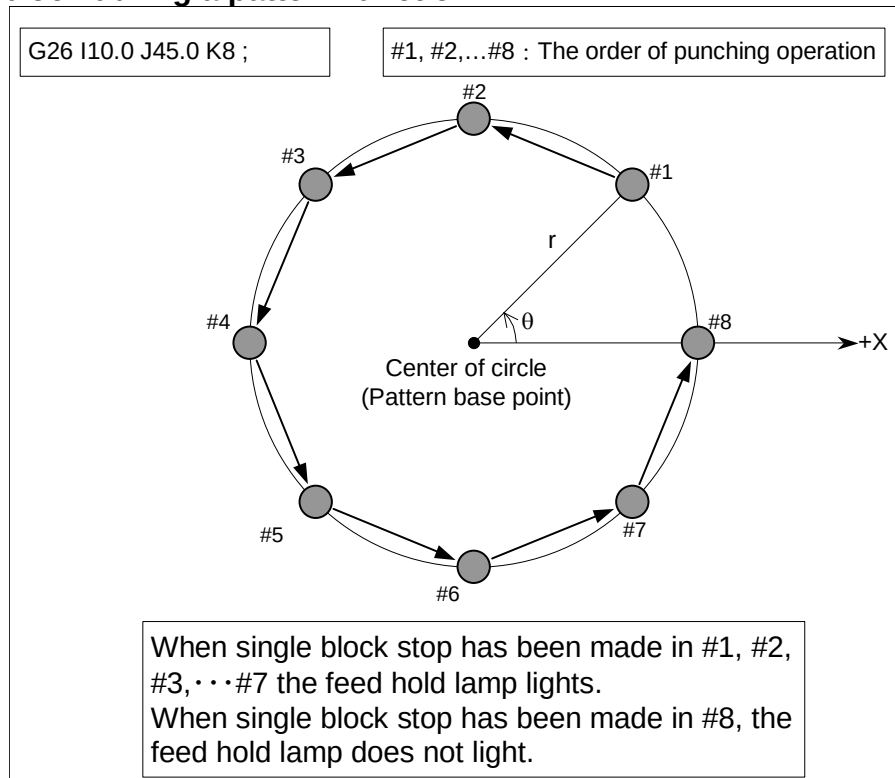
Single block

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

- Single block during a pattern function



5.6 PUNCH

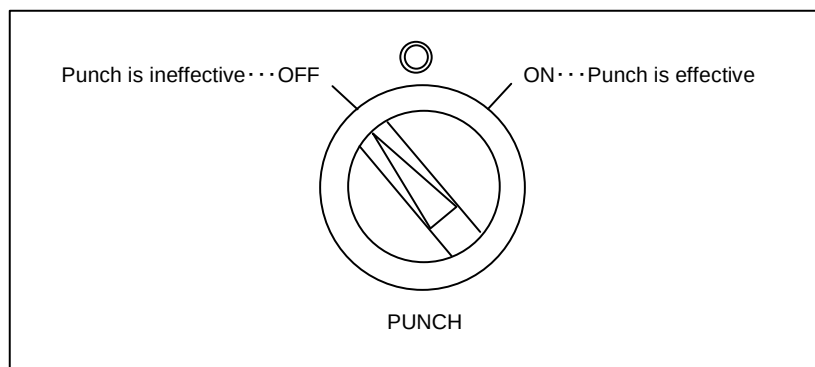


Fig.5.6 Punch On/Off

Explanation

This function makes punch (including nibbling) ineffective in a block where punching is made by press motion during the DNC or memory mode operation.

5.7 MANUAL PUNCH

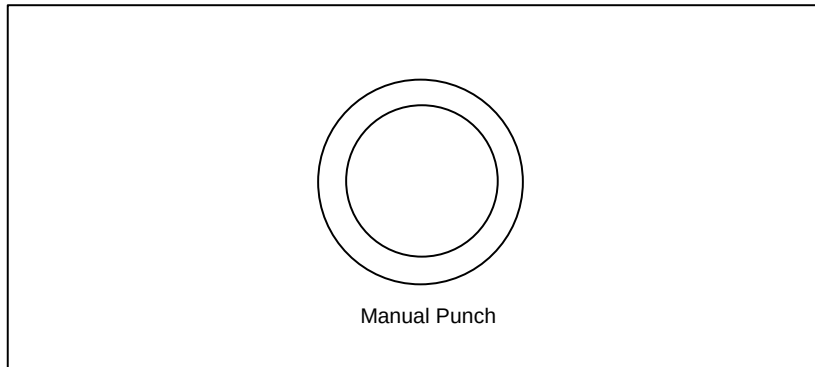


Fig.5.7 Manual punch

Explanation

When depressing this button, punching is made by press motion. When depressing this button again after releasing it once, punching is made again.

Generally, when this button is depressed, while the punch ON/OFF switch in Section 5.6 is being set to ON, punching executed, while if the switch is set to OFF, punching is not executed.



CAUTION

No punching is executed by depressing this button when the cycle start lamp is lighting or during manual axis movement. Since effective conditions of this button depend on the machine tool builders, refer to the machine tool builder's manual.

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	408
6.2	OVERTRAVEL	408
6.3	STORED STROKE CHECK	409
6.4	STROKE LIMIT CHECK BEFORE MOVE.....	412
6.5	WRONG OPERATION PREVENTION FUNCTIONS.....	414
6.6	SAFETY ZONE CHECK	427

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

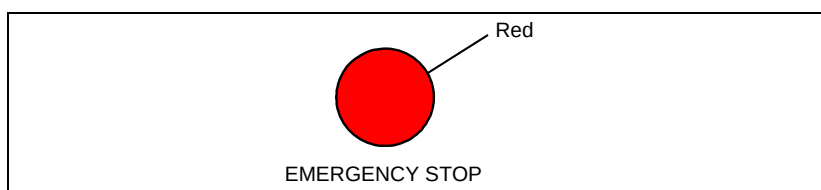


Fig. 6.1 (a) Emergency stop

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 reaches the stroke end set by the machine tool limit switch, it decelerates and stops because of working the limit switch and an OVERTRAVEL alarm is displayed.

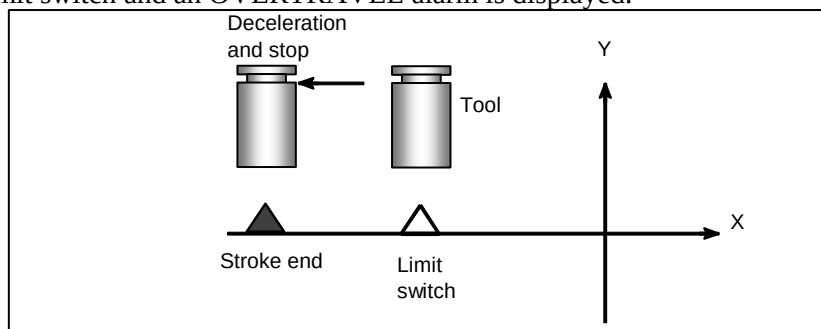


Fig. 6.2 (a) Overtravel

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

Table 6.2 (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.

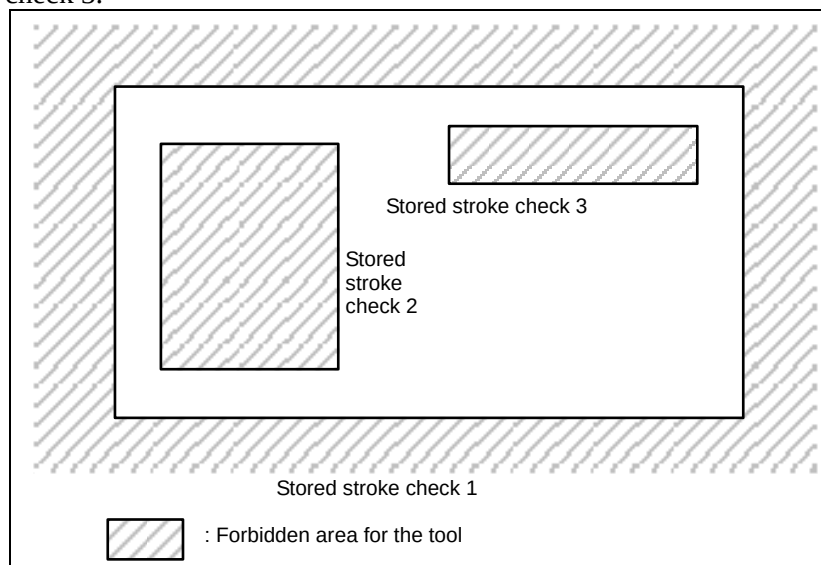


Fig. 6.3 (a) Stroke check

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. Bit 0 (OUT) of parameter No. 1300 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 in the Fig. 6.3 (b) creates or changes the forbidden area:

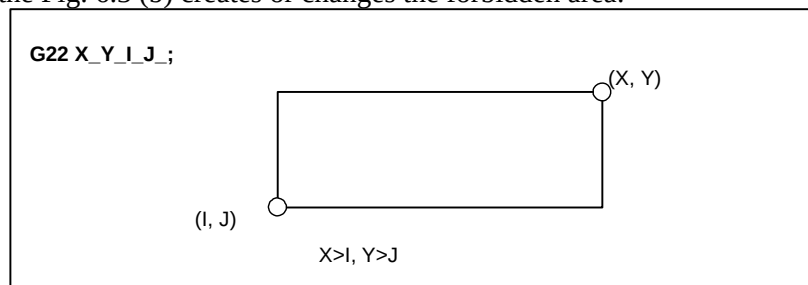


Fig. 6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the Fig. 6.3 (c) must be set.

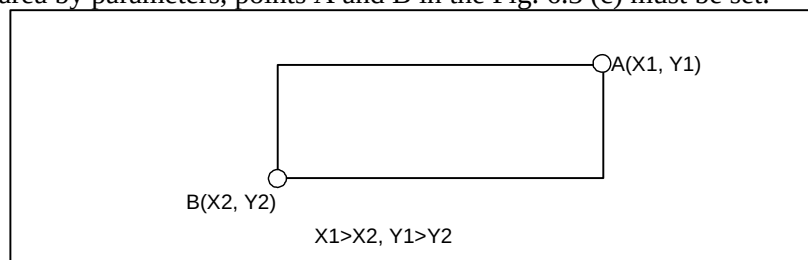


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

The values X1, Y1, X2, and Y2, which are set by parameters Nos. 1322 and 1323, must be specified by the distance from the machine coordinate system (machine unit). The values X, Y, I, and J, 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 Nos. 1324 and 1325. The area inside the boundary becomes the forbidden area. The values X1, Y1, X2, and Y2 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 two points for specifying a forbidden area are erroneously set, the rectangular parallelepiped having the points as vertices is assumed as a boundary.
- 3 Since an axis without the reference position return function has no forbidden areas, there are no alarms about forbidden areas for the axis.

- Forbidden area overlapping

Area can be set in piles.

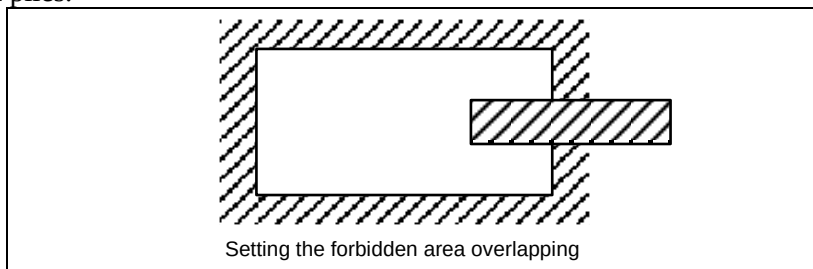


Fig. 6.3 (d) Setting the forbidden area overlapping

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 tool 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.

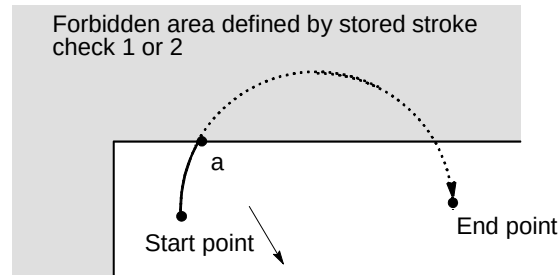
6.4 STROKE LIMIT CHECK BEFORE MOVE

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.

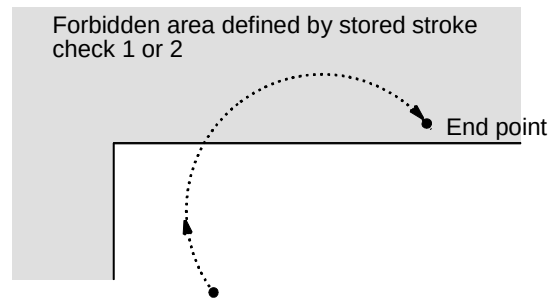
**WARNING**

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)

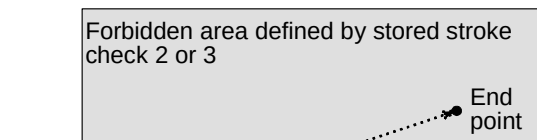
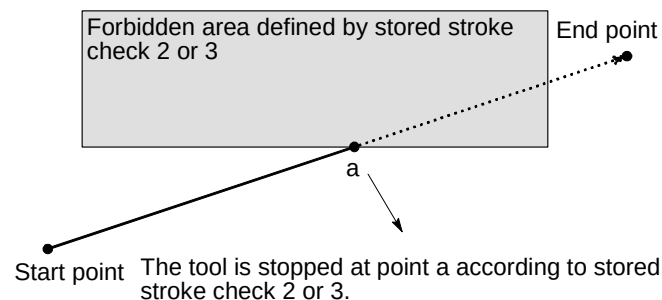


The tool is stopped at point a according to stored stroke check 1 or 2.



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)



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 G33 (skip) block can be determined using (bit 2 (NPC) of parameter No. 1301).

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.

- 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 pattern function and automatic reference position return) is executed, an alarm is issued at the start point of any operation whose end point falls within an inhibited area.

- 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 and /or 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

- 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 messages and warning messages in the Table 6.5.1.1 (a) or Table 6.5.1.1 (b) is displayed. No message is displayed when the input data range check is disabled.

When the set effective data range is valid

Table 6.5.1.1 (a) List of messages displayed 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

Table 6.5.1.1 (b) List of messages displayed 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

- 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

- 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 ?" 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

- 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 an 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
	C1	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
AXIS MOVING	: *
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.

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

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 AI contour control 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

ACTUAL POSITION				00123 N00000			
ABSOLUTE				MACHINE		DISTANCE TO GO	
X	250.000			X	250.000	X	0.000
Y	550.000			Y	550.000	Y	0.000
T	10			T	0.000	T	0.000
C	0.000			C	0.000	C	0.000
MODAL				F 0 MM/MIN			
G00	G50	G13.1	F 1000M	S 0 /MIN			
G17	G67	G50.1	H	PARTS COUNT 9			
G90	G54	D		RUN TIME 0H 0M CYCLE TIME 0H 0M 9S			
G22	G64	T 10					
G94	G05	S					
G21	G15						
G40	G40.1						
A>_				MODAL DATA IS CHANGED BY BLOCK STOP			
MEM **** * * * *				19:19:54			
ABSOLUTE		RELATIVE		ALL		HANDLE (OPRT)	

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

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 Press 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, press the [GUARD] soft key. The operation confirmation function setting screen is displayed.

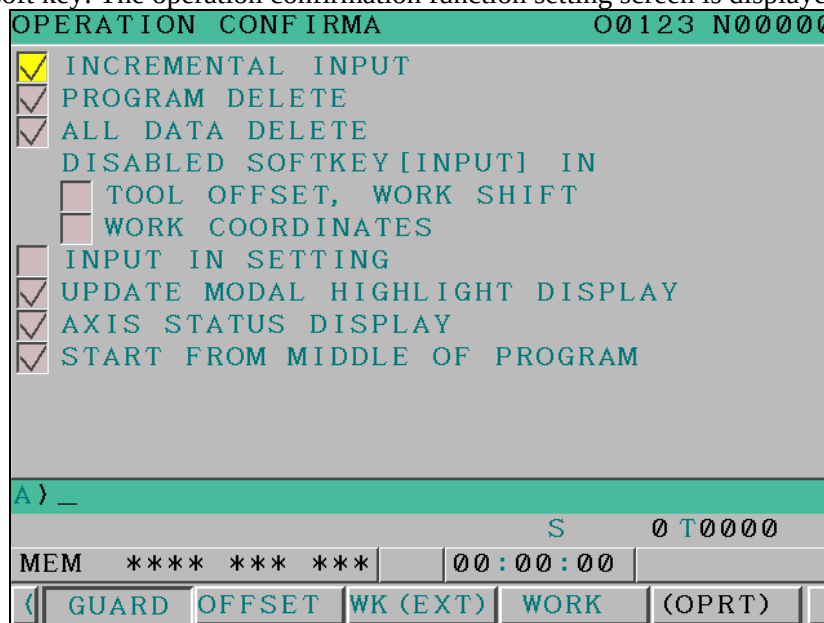


Fig. 6.5.3.1 (a) Operation confirmation function setting screen

- 5 Move the cursor to the check box of the item you want to change the setting, by pressing the , , , and  keys.

- 6 Press the soft key [(OPRT)].
- 7 Press the soft key [ON:1] or [OFF:0]. When you press the [ON:1] soft key, a check mark (✓) appears in the corresponding check box, indicating that the function is enabled. When you press 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 Table 6.5.3.1 (a) shows what is displayed for each item to be set and the corresponding functions. The items whose column of "Default" is marked by "○" is effective automatically after memory all clear.

Table 6.5.3.1 (a)

Displayed item	Default	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 (lathe system), and work shift (lathe system))
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 Press 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, press the [OFFSET] soft key. The tool offset range setting screen is displayed.

OPERATION CONFIRMA				00123 N00000	
TOOL OFFSET : (1/3)					
FROM	TO	LOW-LIMIT	UP-LIMIT		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
0	0	0. 000	0. 000		
A) _					
			S		0 T0000
MEM	****	***	***	00:00:00	
(	GUARD	OFFSET	WK (EXT)	WORK	(OPRT)

Fig. 6.5.3.2 (a) Tool offset range setting screen

- Press the soft key [(OPRT)].
- Move the cursor to the item you want to set, by using the  and  keys, , , , and  keys, or the [SWITCH] soft key.
- Press the MDI key, enter necessary data, and then press 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

- What to set with tool offset memory A

With tool offset memory A, an effective data range is specified using the four items in the Table 6.5.3.2 (a).

Table 6.5.3.2 (a)

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	

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.

- Example of setting an input data range

For example, suppose that the following values are set with offset memory A.

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 Press 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, press the [WORK] soft key. The workpiece origin offset range setting screen is displayed.

OPERATION CONFIRM				00123 N00000	
WORK COORDINATES (1/6)					
WK COORD : G54 ~ G59、G54. 1P1 ~ G54. 1P48					
NO. 1		FROM		TO	
WK	CO	(NONE)		(NONE)	
		LOW-LIMIT		UP-LIMIT	
X		0. 000		0. 000	
Y		0. 000		0. 000	
T		0. 000		0. 000	
C		0. 000		0. 000	
A) _					
				S	0 T0010
MEM	****	***	***	12:00:00	
[	GUARD	OFFSET	WK (EXT)	WORK	(OPRT)

Fig. 6.5.3.3 (a) Workpiece origin offset range setting screen

- 5 Press the soft key [(OPRT)].

- 6 Move the cursor to the item you want to set, by using the  and  keys, or , , , and  keys.
- 7 Press the MDI key, enter necessary data, and then press 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 four items in the Table 6.5.3.3 (a).

Table 6.5.3.3 (a)

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 two items in the Table 6.5.3.3 (b).

Table 6.5.3.3 (b)

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid external workpiece origin offset value range on each axis.
	UP-LIMIT	

6.6 SAFETY ZONE CHECK

This is the safety function to set the safety zone for protecting the workpiece holder that holds the workpiece set on the carriage, and disable punching in that area or forbid the tool to approach there into.

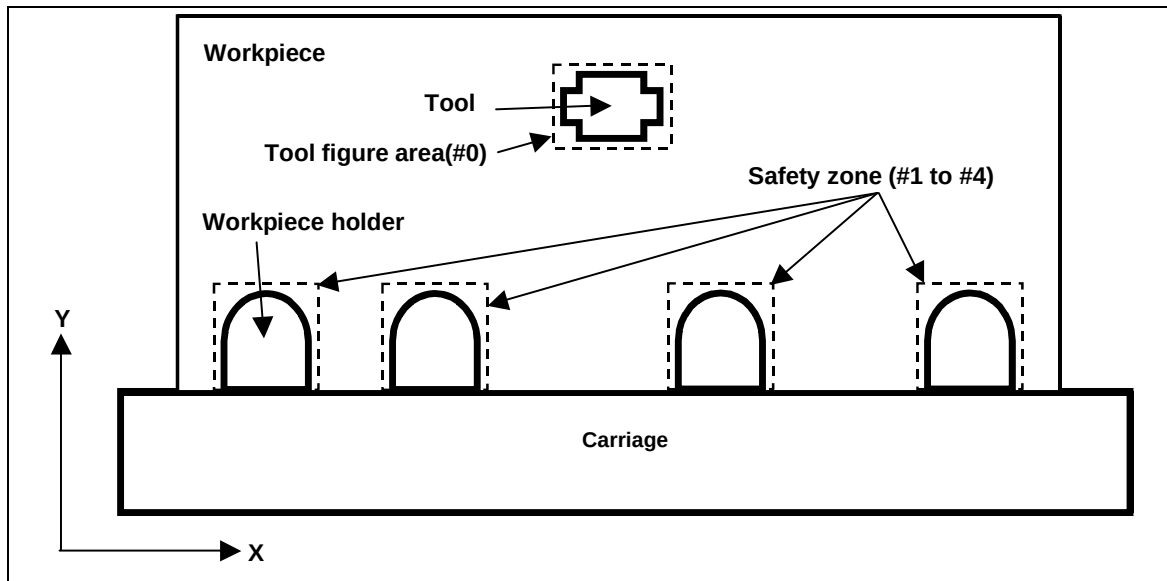


Fig.6.6 (a) Outline

This function permits to set tool figure area (#0) and up to four safety zones (#1 - #4), as shown above. Type A and type B of safety zone check are prepared.

6.6.1 Punch Forbidden Area and Approach Forbidden Area (Type A)

The safety zone is settable in two types, punch forbidden area and approach forbidden area, that are set by the parameter SZ1 to SZ4 (No. 16501#0 - #3) shown below.

(1) Punch forbidden area

When the tool figure area goes into the safety zone and the punching is commanded, an alarm (DS4800 to DS4803) is given to disable punching. In the case of positioning & punching command in the automatic operation mode, when the end point of positioning is in the punch forbidden area, an alarm is given without moving the axis.

(Previous check)

In the case of move command without punching, the tool figure area can go into the punch forbidden area, but manual punching is impossible after going into this area.

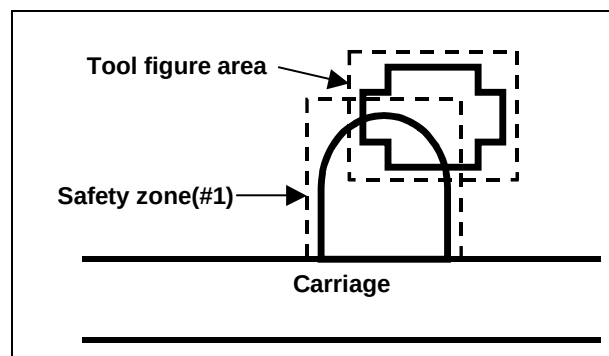


Fig.6.6 (b) Punch forbidden area (Type A)

2) Approach forbidden area

The tool figure area can not go into the safety zone. When the tool figure area approaches into the safety zone by the move command, the axis is immediately stopped and an alarm (DS4810 to DS4817, DS4830 to DS4837) is given. This is valid in either manual or automatic operation mode.

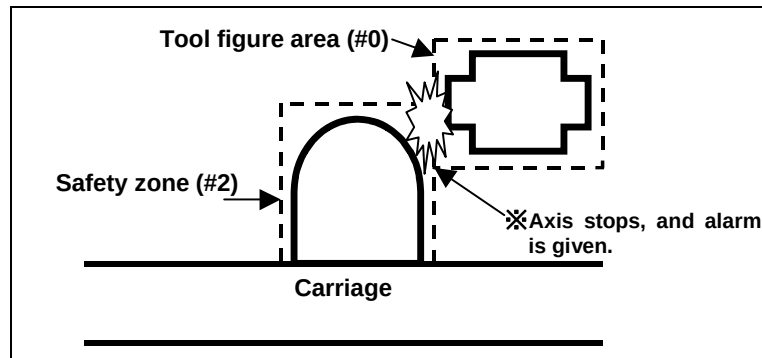


Fig.6.6 (c) Approach forbidden area (TypeA)

6.6.2 Punch Forbidden Area and Approach Forbidden Area (Type B)

Overview

By setting bit 0 (SF0) of parameter No. 16500, the type B safety zone check can be selected. With type B, no alarm is issued even if a tool enters a safety zone; after confirming the safety of the situation, the operator can perform a punch operation, or can position the tool to the next punching position without performing punching.

With type B, all safety zones are handled as punch forbidden areas.

- Punch forbidden area

If the tool enters a safety zone, the punch-forbidden area-entered signal WDD0 <Fn231.7>, used to provide notification that the tool has entered a punch forbidden area, is set to 1 in the punch block. When punching is to be performed in a punch-forbidden area, punch-enable signal WDC <Gn232.4> is set to 1 after the operator confirms that punching can be performed safely. After detecting the WDC signal, the CNC sets the PF signal to 1.

When punching is not to be performed in a punch forbidden area, the press activation lock signal PFL <Gn230.0> is set to 1. Upon detecting the PFL signal, the CNC moves on to execute the next block.

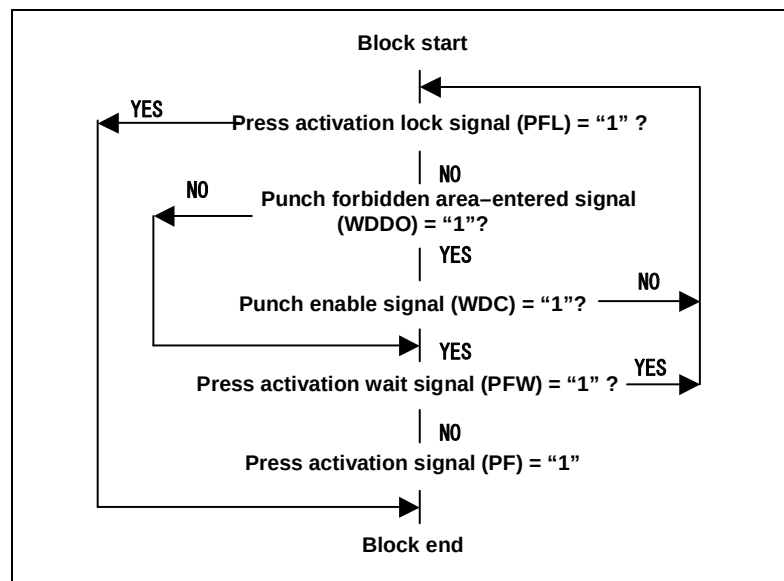


Fig.6.6 (d) Type B Punch forbidden area

⚠ CAUTION

- 1 If the tool enters a punch forbidden area during nibbling, the WDDO signal is set to 1 one punching position before the tool enters punch forbidden area.
- 2 With type B, the setting of SZ1 to SZ4 (bits 0 to 3 of parameter No. 16501) is ignored.

6.6.3 Setting the Safety Zone

Set the machine coordinate value when the workpiece holder is positioned at the tool center (punching position), in the parameters 16505 - 16516 in output units.

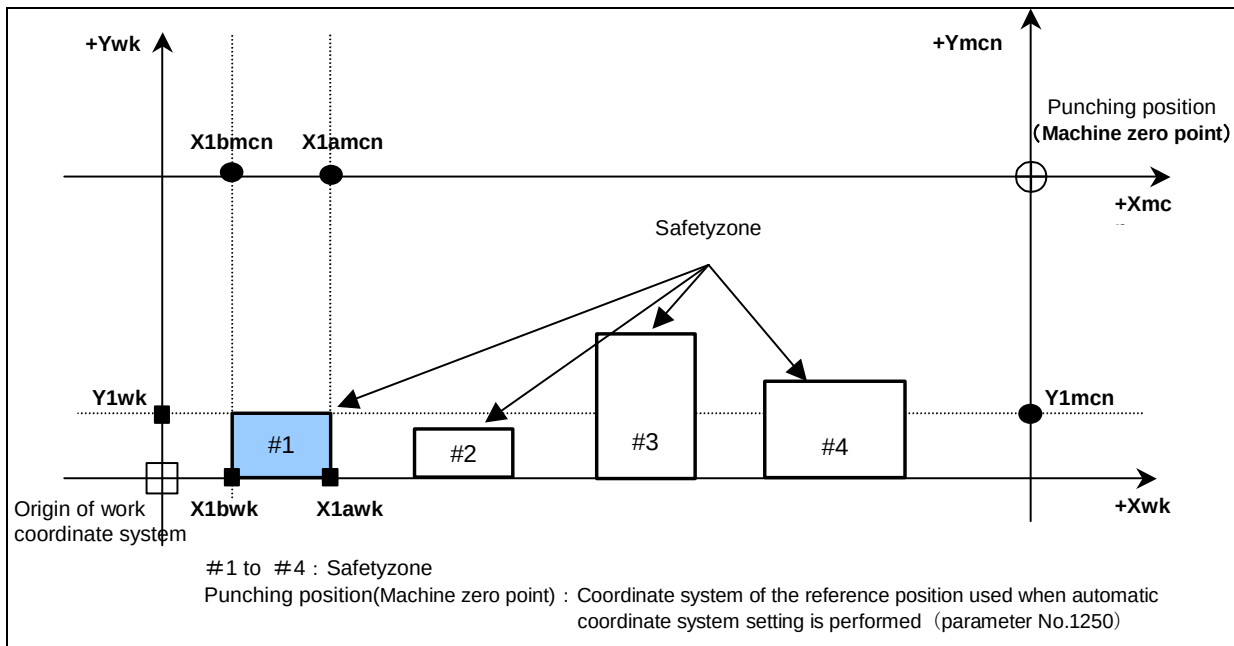


Fig.6.6 (e) Set the value of Safety zone

Regarding #1, the safety zone is specified at both ends (X1amcn, X1bmcn) for the X-axis direction by parameter No.16505 to 16507, and at the forward end (Y1mcn) of the workpiece holder for the Y-axis direction by parameter No.16507. Farther, width of workpiece holder along the X-axis is specified by parameter No.16540.

The specifying method is the same as for #2, #3 and #4.

6.6.4 Setting the Tool Shape Area

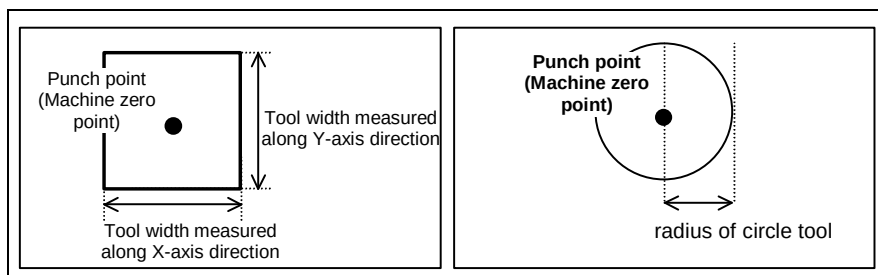


Fig.6.6 (f) The value of setting tool shape area

The specification of the area of tool figure sets the size in the X direction and Y direction of the tool by the parameter (No.16517 to 16532 and No.16551 to 16558).

The setting unit is output unit.

12 kinds of or less tool figure can be set.

Moreover, setting bit 1 (RTZ) of parameter No.16540 to 1 allows to set area of the circle tool figure. When setting area of the circle tool figure, the size in the X direction should be set to the radius of the circle tool, and the size in the Y direction should be set to 0.

The tool shape area can be changed by using the programmable parameter input function (G10). Therefore, when multiple tools are used, it is possible to specify the tool shape area meeting the tool No. (Txx).

6.6.5 Automatic Setting of the Safety Zone

The detector on the machine automatically detects the positions of the workpiece holders mounted on the carriage. Values representing the detected positions are then set in the safety zone parameters.

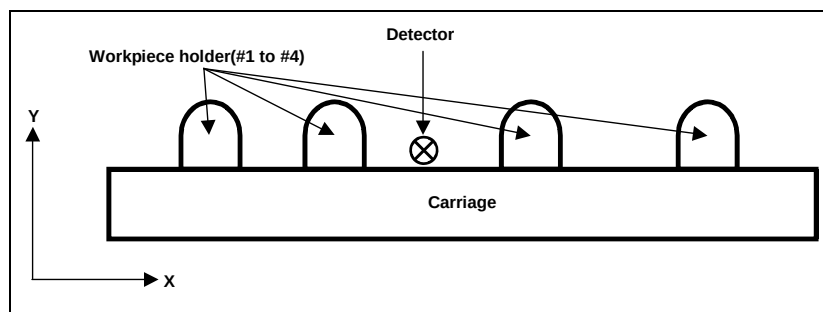


Fig.6.6 (g) AUTOMATIC SAFETY-ZONE SETTING

The signal for detecting the position of the workpiece holders turns on and off as workpiece holders #1 to #4 pass the detector as shown in Fig.6.6 (g). The safety zone is determined from the signal. The position on the X axis when the signal goes high is regarded as one end of a workpiece holder, and the position when the signal goes low is regarded as the other end.

Workpiece holder detection command format

G32X__x__F__f__P__p__Q__q__;

G32 is specified to detect the positions of workpiece holders. Before G32 can be specified, the workpiece holder position detector must be activated.

Before detection is started, the X-axis move command must be specified following address X. Either the positive or negative move direction is allowed. If a positive move command is specified, the position at which detection starts must be in the negative side with respect to workpiece holder #1. Sufficient distance must be kept between the position at which detection starts and workpiece holder #1 to enable stability of speed before workpiece holder #1 is reached. Workpiece holders #1, #2, #3, and #4 are detected in that order. If a negative move command is specified, the workpiece holders are detected in the reverse order starting from #4. The other conditions are the same as those used for a positive move command.

The feed rate during detection is specified with F, in the same way as for ordinary interpolation. The position of a workpiece holder is obtained from the machine positions indicated by the rising and falling edges of the workpiece holder detection signal. There is a difference between the indicated machine position and actual machine position because of an error such as a lag in the servo system. This error must be compensated with numeric values following addresses P and Q. The compensating value for the position on the rising edge of the workpiece holder detection signal is set following address P, and the compensating value for the position on the falling edge is set following address Q.

The lag in the servo system is obtained from the following equations:

$$\Delta E = T1 \times F + T2 \times F \quad (\text{exponential function acceleration/deceleration})$$

$$\Delta E = 1/2 T1 \times F + T2 \times F \quad (\text{linear acceleration/deceleration})$$

where,

ΔE : Lag in the servo system

$T1$: Time constant for automatic acceleration/deceleration

$T2$: Servo time constant

F : Feed rate

The sign of the compensating value is positive when compensation is made in the opposite direction to the move direction specified by the detection command.

If address P/Q is omitted in the G32 block, the error between coordinates and actual machine position is calculated by the CNC.

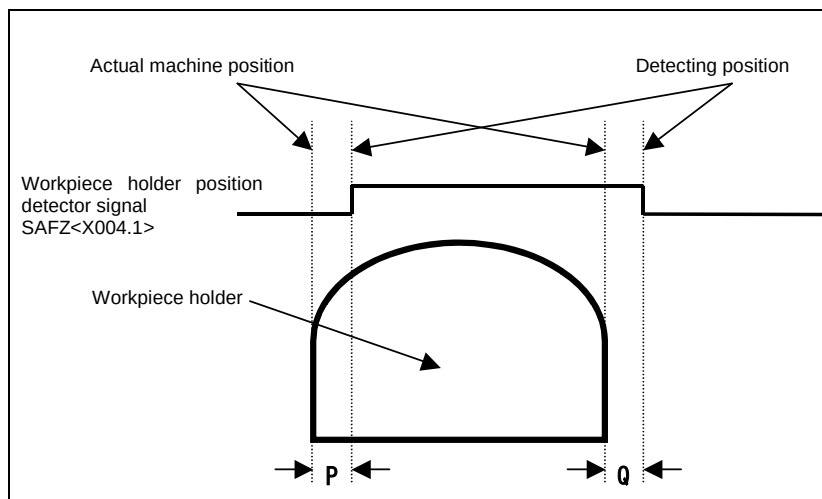


Fig.6.6 (h) Workpiece holder position detector signal SAFZ<X004.1> and compensation of detecting position

6.6.6 Displaying the Safety Zones and Tool Zone

After safety zone values are set automatically, they can be displayed on the safety zone screen as shown below. With this screen, the user can check whether the set values are valid.

Screen

- Type A

ACTUAL POSITION 00000 N00000

ABSOLUTE

X 0.000
Y 0.000
T 10
C 0.000

MODAL

G00 G50 G13.1F M
G17 G67 G50.1H
G90 G54 D
G22 G64
G94 G85 S T 10
G21 G15
G40 G40.1

SACT 0/MIN

SAFETY ZONE (ABSOLUTE)

AREA #1

X2 100.000
X1 200.000
Y 50.000

AREA #2

X2 300.000
X1 400.000
Y 50.000

AREA #3

X2 100.000
X1 200.000
Y 50.000

AREA #4

X2 700.000
X1 800.000
Y 50.000

TOOL ZONE #1

X 50.000
Y 50.000

ZONE NUMBER

N 2

MDI ***** 12:00:00

TOOL REGIST MULTI SAFETY ZONE (OPRT) +

Fig.6.6 (i) Safety Zone Setting Screen Type A (10.4-inch display unit)

- Type B

ACTUAL POSITION 00000 N00000

ABSOLUTE

X 0.000
Y 0.000
T 10
C 0.000

MODAL

G00 G50 G13.1F M
G17 G67 G50.1H
G90 G54 D
G22 G64
G94 G85 S T 10
G21 G15
G40 G40.1

SACT 0/MIN

SAFETY ZONE (ABSOLUTE)

AREA #1

W 100.000
X 150.000
Y 50.000

AREA #2

W 100.000
X 350.000
Y 50.000

AREA #3

W 100.000
X 150.000
Y 50.000

AREA #4

W 100.000
X 750.000
Y 50.000

TOOL ZONE #1

X 50.000
Y 50.000

ZONE NUMBER

N 2

MDI ***** 12:00:00

TOOL REGIST MULTI SAFETY ZONE (OPRT) +

Fig.6.6 (j) Safety Zone Setting Screen Type B (10.4-inch display unit)

NOTE

The display items of type B are as follows:

W : Workpiece holder width

X : Workpiece holder central position relative to the tool center

Y : Workpiece holder tip position relative to the tool center

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 60 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 begins at the start position on the next line.

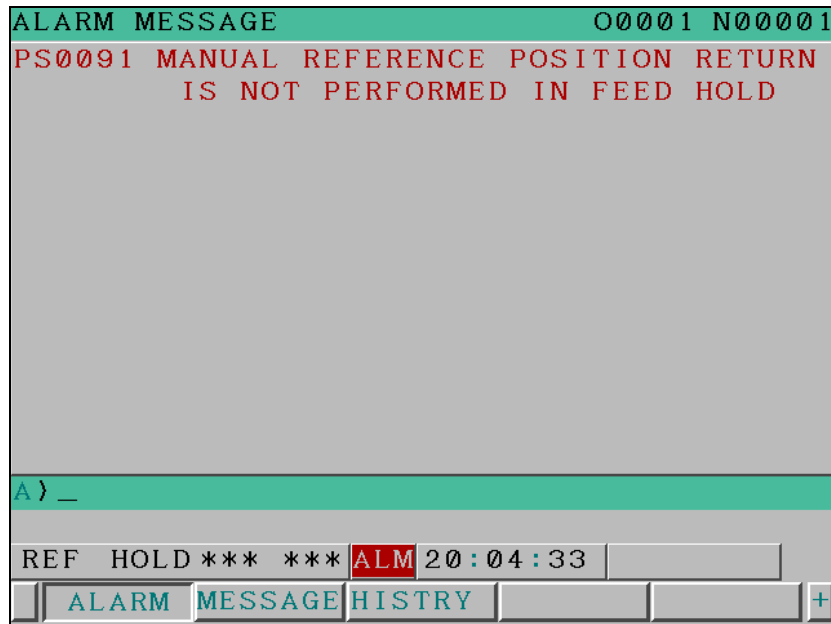


Fig. 7.1 (b) Display wrapping (example for the 8.4-inch display unit)

- **Displaying an 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).

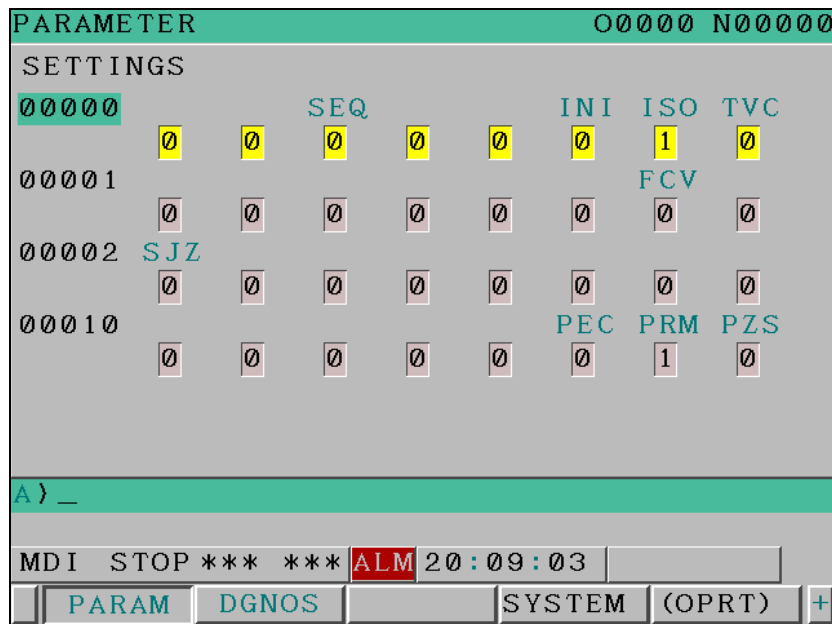


Fig. 7.1 (c) Parameter screen (example for the 8.4-inch display unit)

In this case, display the alarm screen by following the steps below.

- 1 Press the function key .
- 2 Press the soft key [ALARM].
- 3 You can change pages by using the page key.

- **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 H, "ALARM LIST".

- 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.

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 [HISTORY].
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> Page No.
- 3 You can change pages by using the page key.

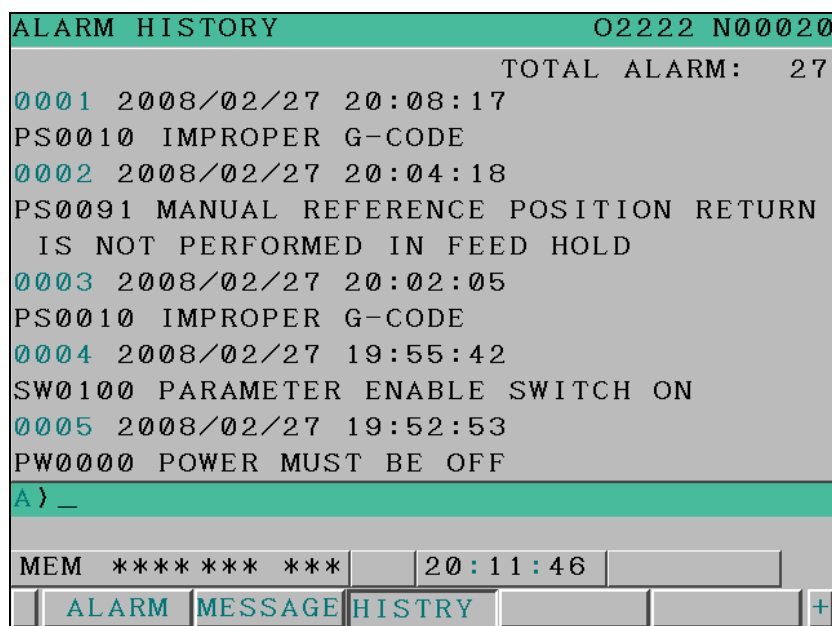


Fig. 7.2 (a) Alarm history screen (example for the 8.4-inch display unit)

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].

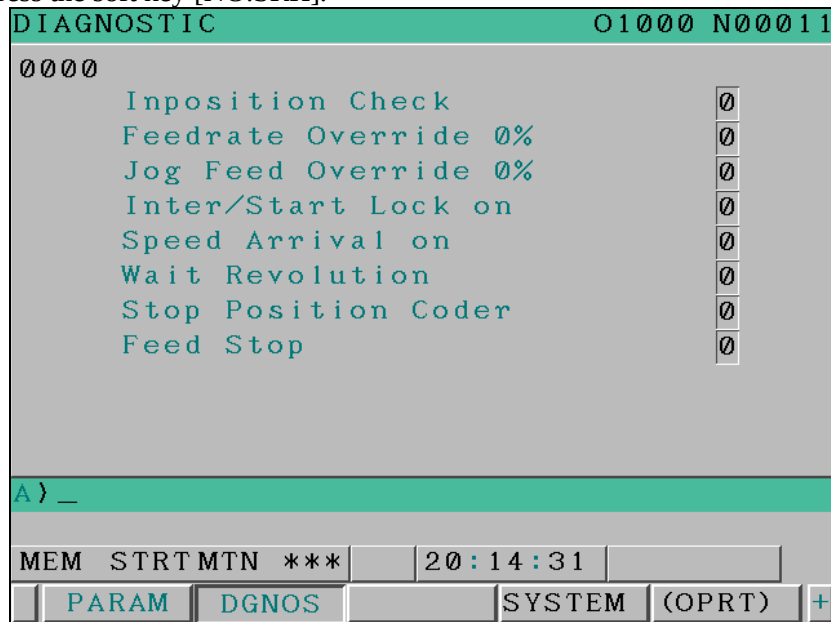


Fig. 7.3 (a) Diagnostic display (example for the 8.4-inch display unit)

Diagnostic numbers 990 to 998 indicate Press condition. (See Table7.3 (a))

Table7.3 (a) Press condition

No.	Display	Internal status when 1 is displayed
990	WAITING FOR SIGNAL PFW OFF	Waiting for turning off press start wait signal PFW.
991	WAITING FOR 1-CYCLE *PE OFF	Waiting for press stop to an one-cycle press motion.
992	WAITING FOR 1-CYCLE *PFIN OFF	Waiting for completion of punching to an one-cycle press motion.
993	WAITING FOR NIBBLING *PE OFF	Waiting for press stop to a nibbling motion.
994	WAITING FOR NIBBLING *NFIN OFF	Waiting for completion of punching to a nibbling motion.
995	WAITING FOR T COMMAND	Press operation is not executable, because the punch command was given before executing the T command once after the cycle operation signal OP has been turned from off to on.
996	MANUAL PRESS	Manual press is executing.
997	WAITING FOR PF	Waiting that absolute value of X and Y axis

No.	Display	Internal status when 1 is displayed
		position deviation value becomes the value of parameter No. 16010 and below.
998	REMAINING U/V MACRO STORAGE SIZE	Remaining U/V macro storage size (Kbyte).

7.4 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.

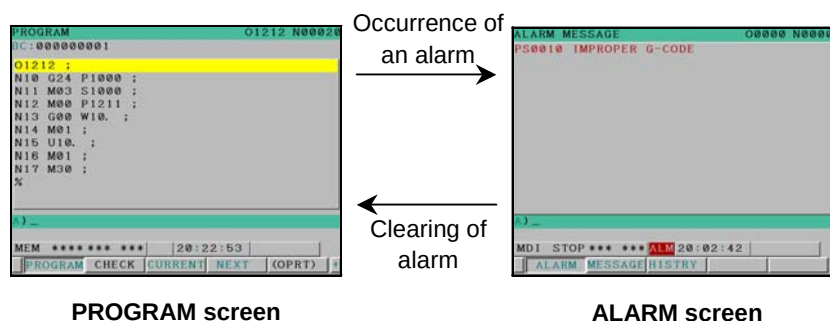
Switching between screens when alarms are cleared

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)



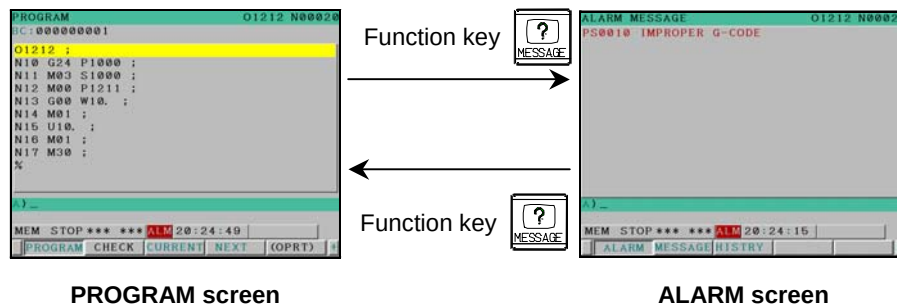
NOTE

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)



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 custom 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.

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 in the Table 8 (a) can be input and output.

Table 8 (a)

Data type	Default file name	Reference item
Program	ALL-PROG.TXT	8.2.1
Offset data	TOOLOFST.TXT	8.2.3
Parameter	CNC-PARA.TXT	8.2.2
Pitch error compensation data	PITCH.TXT	8.2.4
Custom macro common variable	MACRO.TXT	8.2.5
Workpiece coordinate system data	EXT_WKZ.TXT	8.2.6
Operation history data	OPRT_HIS.TXT	8.2.7
Tool data	TOOLDATA.TXT	8.2.8
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)	MAITEMC.TXT	
Servo waveform diagnosis	WAVE-DGN.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.

This function is enabled by setting bit 1 (COW) of parameter No. 11308.

The external I/O device set in NC parameter No. 0020 is selected. See the Table 8 (b) below for details.

Table 8 (b)

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. (Bit 1 (COW) of parameter No. 11308 = 1)

PARAMETER										00123 N00020									
SETTINGS																			
00000		0		0		SEQ		0		0		INI		ISO		TVC		0	
00001		0		0		0		0		0		0		FCV		0		0	
00002		SJZ		0		0		0		0		0		0		0		0	
00010		0		0		0		0		0		PEC		PRM		PZS		1	
A) _																			
OVERWRITE?																			
EDIT *****										20:27:21									
REWRITE										CAN									

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

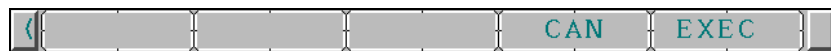


Fig. 8.1 (d) Soft key display after [ALL] or [NON-0] is pressed

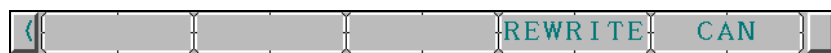


Fig. 8.1 (e) Soft key display after [EXEC] is pressed



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, 3-dimensional error compensation, macro variable, workpiece coordinate system data, operation history, and tool management.

Section 8.2, "INPUT/OUTPUT ON EACH SCREEN", consists of the following subsections:

8.2.1	Inputting and Outputting a Program.....	443
8.2.1.1	Inputting a program	443
8.2.1.2	Outputting a program	444
8.2.2	Inputting and Outputting Parameters.....	445
8.2.2.1	Inputting parameters.....	445
8.2.2.2	Outputting parameters	445
8.2.3	Inputting and Outputting Offset Data.....	446
8.2.3.1	Inputting offset data.....	446
8.2.3.2	Outputting offset data.....	447
8.2.4	Inputting and Outputting Pitch Error Compensation Data	447
8.2.4.1	Inputting pitch error compensation data.....	447
8.2.4.2	Outputting pitch error compensation data	448
8.2.4.3	Input/output format of pitch error compensation data.....	448
8.2.5	Inputting and Outputting Custom Macro Common Variables.....	449
8.2.5.1	Inputting custom macro common variables.....	449
8.2.5.2	Outputting custom macro common variables	450
8.2.6	Inputting and Outputting Workpiece Coordinates System Data	451
8.2.6.1	Inputting workpiece coordinate system data	451
8.2.6.2	Outputting workpiece coordinate system data.....	452
8.2.7	Inputting and Outputting Operation History Data.....	452
8.2.7.1	Outputting operation history data.....	452
8.2.8	Inputting Outputting Tool Data	453
8.2.8.1	Inputting tool data.....	453
8.2.8.2	Outputting tool data.....	453
8.2.8.3	Output format of tool data	454

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 8.2.1.1 (c).

- 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.

Table 8.2.1.1 (c)

[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 output.
- 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 8.2.1.2 (a).
- 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].

Table 8.2.1.2 (a)

[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]

[F-NAME]	[O SET]	Output file name	Output program
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 output.

- 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 parameters in the Table 8.2.2.2 (a) are not output:

Table 8.2.2.2 (a)

	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 output.
- 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:

- Tool compensation memory A

```
%
G10 G90 P01 R_ Q_
G10 G90 P02 R_ Q_
...
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.

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 .
- 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 output.
- 2 Press the function key .
- 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:

Table 8.2.4.3 (a)

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 and 7.

The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example

```
N10001Q0P5;
    Pitch error compensation data number  1
    Pitch error compensation data value   5
```

- Beginning and end of a record

A pitch error compensation data record begins with % and ends with %.

Example

```
% ; .....Beginning of record
N10000Q0P1;
N10001Q0P3;
:
N11279Q0P0;
% .....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 output.
- 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.

```
%
G10L85P100(0000000000000000)
G10L85P101(0000000000000000)
G10L85P102(FFFFFFFFFFFFFFFF)
```

```

:
G10L85P500(4024000000000000)
G10L85P501(4021000000000000)
G10L85P502(0000000000000000)
.
SETVN500[ABC]
SETVN501[DEF]
SETVN502[GHI]
:
M02
%

P_ : Macro variable number
SETVN_ [ _ ] : Common variable name

```

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 output.
- 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 from the memory of the CNC to an external device.

Outputting operation history data

Procedure

- 1 Make sure the output device is ready for output.
- 2 Press the function key .
- 3 Press the continuous menu key  until soft key [OPERAT HISTRY] appears.
Press the soft key [OPERAT HISTRY].
- 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 soft key [F OUTPUT].
- 7 Type the file name that you want to output.
If the file name is omitted, default file name “OPRT_HIS.TXT” is assumed.
- 8 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].

8.2.8 Inputting Outputting Tool Data

8.2.8.1 Inputting tool data

Tool data is input from external I/O devices such as a memory card to the memory of the CNC.

Inputting tool data

Procedure

- 1 Make sure the input device is ready for reading.
- 2 Press the EDIT switch on the machine operator's panel.
- 3 Press the function key .
- 4 Press the continuous menu key , and press the chapter selection soft key [TOOL].
- 5 Press the continuous menu key , and press soft key [TOOL NUMBER], [PUNCH COUNT], or [TOOL SHAPE].
- 6 Press the soft key [(OPRT)].
- 7 Press the soft key [F INPUT].
- 8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “TOOLDATA.TXT” is assumed.
- 9 Press the soft key [EXEC].
This starts reading the tool 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].

8.2.8.2 Outputting tool data

Tool data is output the memory of the CNC to external I/O devices such as a memory card.

Outputting tool data

Procedure

- 1 Make sure the output device is ready for output.
- 2 Press the EDIT switch on the machine operator's panel.
- 3 Press the function key .
- 4 Press the continuous menu key , and press the chapter selection soft key [TOOL].
- 5 Press the continuous menu key , and press soft key [TOOL NUMBER], [PUNCH COUNT], or [TOOL SHAPE].
- 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 “TOOLDATA.TXT” is assumed.
- 9 Press the soft key [EXEC].
This starts outputting the tool 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].

8.2.8.3 Output format of tool data

Output format of tool data is as follows.

%												
N001	T	M	X	Y	S	P	C	I	J	K	B	;
(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)	(11)	(12)	
N002	T	M	X	Y	S	P	C	I	J	K	B	;
.	.	.	.	.	.	.	.	.	.	.	.	;
.	.	.	.	.	.	.	.	.	.	.	.	;
N136	T	M	X	Y	S	P	C	I	J	K	B	;
N201	T	A	Y	C	I	J	K					;
(1)	(2)	(13)	(5)	(8)	(9)	(10)	(11)					
.	.	.	.	.	.	.	.	.	.	.	.	;
.	.	.	.	.	.	.	.	.	.	.	.	;
.	.	.	.	.	.	.	.	.	.	.	.	;
N264	T	A	Y	C	I	J	K					;
%												

Tool data

Tool data for multiple tools

Items (1) to (13) are as follows :

- (1) Tool registration number
The tool numbers registered for multiple tools are output with N200 to N299.
- (2) Tool number
- (3) Turret position
- (4) X-axis offset
- (5) Y-axis offset
- (6) Number of a tool to be substituted
- (7) Number of punch operations
- (8) Tool figure for graphic operation
- (9) X dimension of a tool for graphic operation
- (10) Y dimension of a tool for graphic operation
- (11) Tool angle for graphic operation
- (12) Tool life value
- (13) Angle for indexing the turret for multiple tool

- Output file name

If the output file name is omitted, default file name "TOOLDATA.TXT" is assumed.

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, workpiece coordinate system data, operation history data, and tool management data to an external I/O device.



CAUTION

While an external I/O device is being accessed, do not turn the power to the CNC off or remove the device, or the device may be damaged.

NOTE

The ALL IO screen can be operated only if a memory card is selected as an external I/O device. Use NC parameter No. 20 to select 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	455
8.3.2 Inputting and Outputting Parameters	456
8.3.3 Inputting and Outputting Offset Data	457
8.3.4 Inputting/Outputting Pitch Error Compensation Data	458
8.3.5 Inputting/Outputting Custom Macro Common Variables.....	459
8.3.6 Inputting and Outputting Workpiece Coordinates System Data.....	460
8.3.7 Inputting and Outputting Operation History Data	460
8.3.8 File Format and Error Messages	461

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 8.3.1 (a) for details.
- 6 Set the program number to be used after the input.
Type a program number, and press the soft key [P 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].

Table 8.3.1 (a)

[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 [P SET].
If O-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 O-9999 is specified, the output file name is assumed to be "ALL-PROG.TXT".
See the Table 8.3.1 (b) for details.
(For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6.)
- 7 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen.
When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Table 8.3.1 (b)

[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 read 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 read 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 read 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 read 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 read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

8.3.7 Inputting and Outputting Operation History Data

Operation history data can be output on the ALL IO screen.

Only output operation is permitted for operation history data.

The output data is of text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

Outputting operation history data

Procedure

- 1 Press the soft key [OPERAT HISTORY] 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 "OPRT_HIS.TXT" is assumed.
- 6 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 read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

8.3.8 File Format

Explanation

- File format

All files that are read from and written to an external device are of text format. The format is described below.

A file starts with % or LF, followed by the actual data. A file always ends with %. In a 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 read, 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."

Limitation

- Memory card specification

Use memory cards that comply with PCMCIA Ver2.0 or JEIDA Ver4.1.

- Attribute memory

Memory cards that have no attribute memory or contain no device information in the attribute memory cannot be used.

- Flash ROM card

Flash ROM cards can be used for input only.

8.4 MEMORY CARD SCREEN

8.4.1 Displaying the Memory Card 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 [MEMORY CARD], then the memory card screen appears.

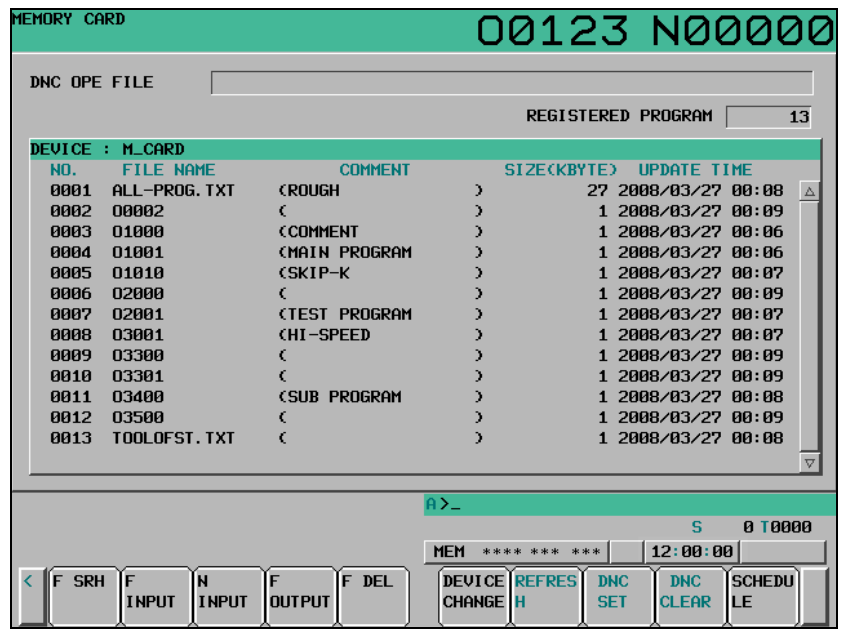


Fig. 8.4.1 (a) Memory card screen

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
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 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 [EMB ETHER], and the EMBEDDED ETHERNET HOST FILE LIST screen appears.

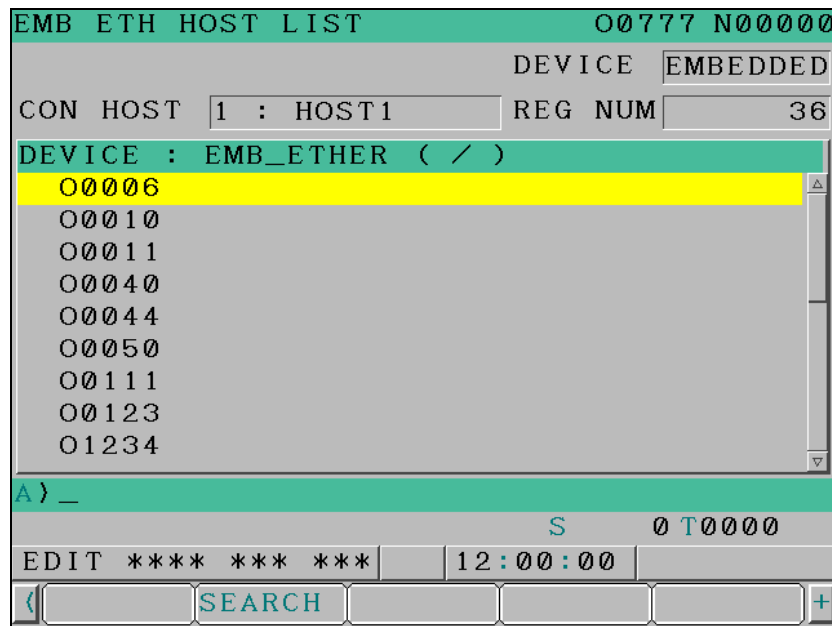


Fig. 8.5.1 (a) Embedded Ethernet host file list screen

NOTE

- 1 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. Make a selection in "DEVICE SELECTION" on the Ethernet setting screen.
 - (2) A host computer can be selected from connection destinations 1, 2, and 3. Make a selection according to soft key [HOST CHANGE] described later.
- 2 File names including 2-bytes characters cannot be displayed correctly.

- 6 When a list of files is larger than one page, the screen display can be switched using the page keys  .
- 7 Press the soft key [REFRESH] to update the screen display.
- 8 Press the soft key [DETAIL OFF] to display the host file list (file names only) screen. Press the soft key [DETAIL ON] to display the host file list (detail) screen.
- 9 When selecting a folder with cursor and pressing the MDI key , the selected folder is changed to the new current folder.

Display item

AVAILABLE DEVICE

"EMBEDDED" or "PCMCIA" is displayed.

CONNECT HOST

The currently selected connection host number and host name are displayed.

REGISTERED PROGRAM

The number of files registered in the work folder of the connected host is displayed. Up to 8 digits can be displayed.

CURRENT FOLDER

The current folder name of the connected host is displayed.

If the folder-path is long compared with the display-item, characters: “...” and only the first and last ten letters of the folder name are displayed.

FILE LIST

There is no distinction between file names and folder names.

Although the maximum number of displayed characters is 127, characters are displayed as far as they can be displayed in one line.

Operation list

DETAIL ON, DETAIL OFF

The screen display can be switched between the display of file names only and the display of details.

REFRESH

Display data can be updated.

INPUT

A file can be input from the host computer to the part program storage memory of the CNC. This item is displayed only when 9 is set as the I/O device number of the CNC.

OUTPUT

A file can be output from the part program storage memory of the CNC to the host computer. This item is displayed only when 9 is set as the I/O device number of the CNC.

DELETE

A folder or file can be deleted from the host computer.

RENAME

A file or folder on the host computer can be renamed.

CREATE FOLDER

A folder can be created on the host computer.

HOST CHANGE

The connected host can be changed.

NC program input

A file (NC program) stored on the host computer can be input into the part program storage memory of the CNC.

Procedure

- 1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
- 2 Press the EDIT switch on the machine operator's panel.
- 3 Press the soft key [(OPRT)].
- 4 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
- 5 Position the cursor on the file to input, and press [F GET] or type the file name to input.
Press the soft key [F SET].
If the input file name is omitted, default input file name “ALL-PROG.TXT” is assumed.

- 6 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 name [O SET] are omitted, see the Table 8.5.1 (a).
- 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].

Table 8.5.1 (a)

[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]

NC program output

A file (NC program) stored in the part program storage memory of the CNC can be output to the host computer.

Procedure

- 1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
- 2 Press the EDIT switch on the machine operator's panel.
- 3 Press the soft key [(OPRT)].
- 4 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
- 5 Type a program number of which you want to output, and press the soft key [O SET].
To specify an output file name, type a file name, and press the soft key [F-NAME].
For an explanation of the operations to be performed if an output file name [F NAME] and an output program number [O SET] are omitted, see the Table 8.5.1 (b).
- 6 Press the soft key [EXEC].
This starts outputting the program, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output of the program, press the soft key [CAN].

Table 8.5.1 (b)

[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]

8.6 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.

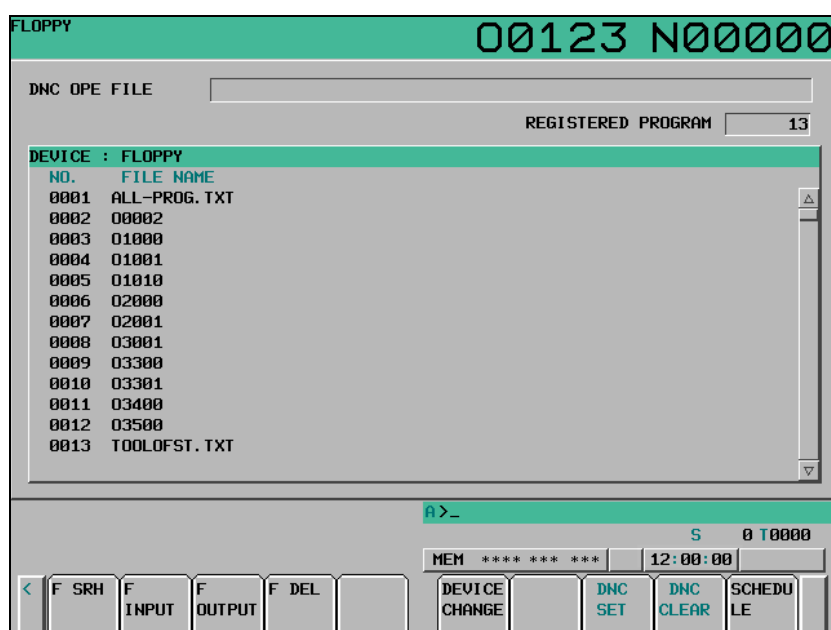


Fig. 8.6.1 (a) Floppy cassette screen

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

- Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
- Press the soft key [(OPRT)].
- Press the soft key [F OUTPUT].
- 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.
- 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.
- 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]	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  for five seconds. The function can be canceled by pressing key .

- 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.

In either case, screen data is not output, so that either the existing file needs to be deleted or the memory card needs to be replaced 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.

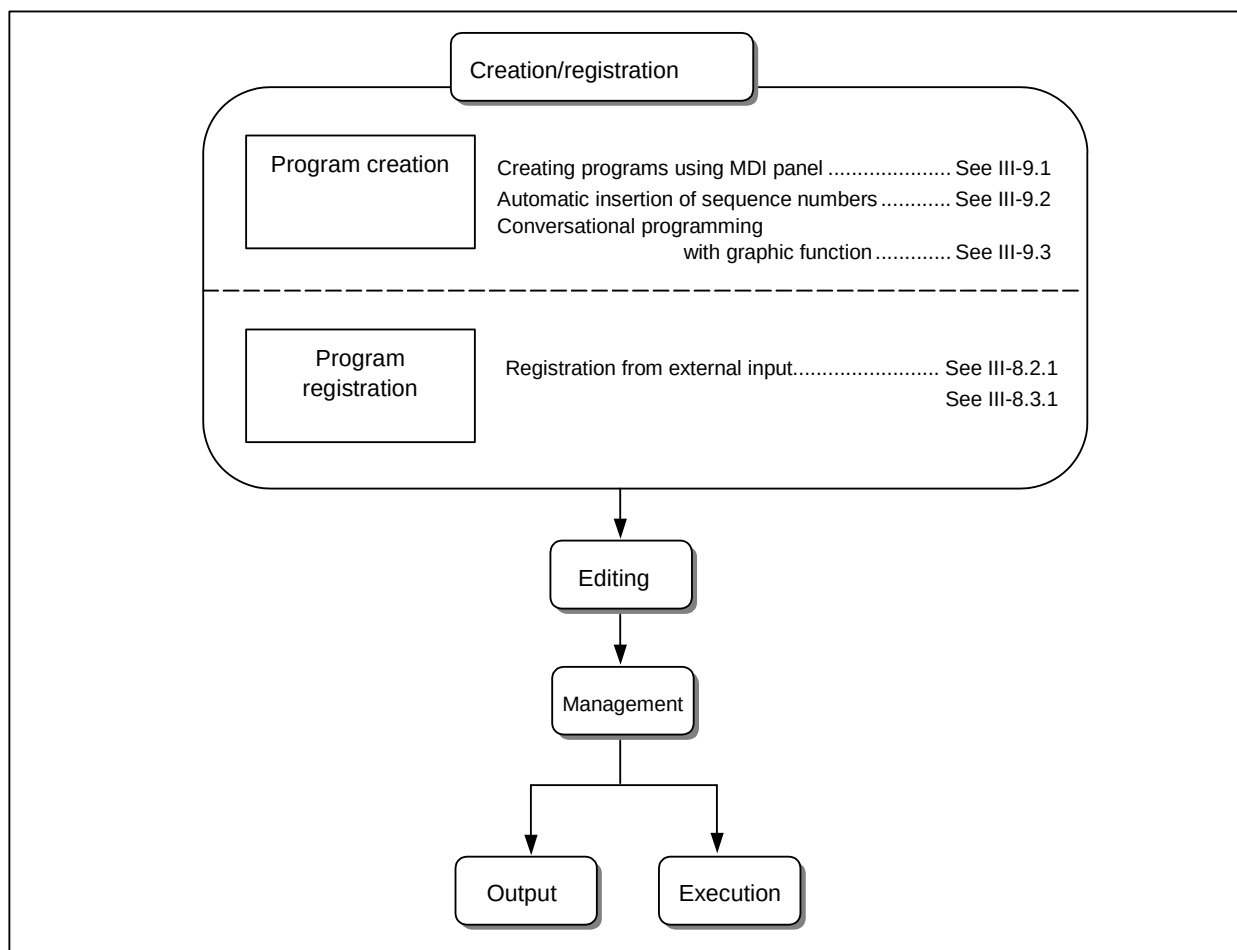
- Canceling the hard copy function

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 conversational programming with graphic function.



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  key is pressed after the control-out code "(", comments, and control-in code ")" have been typed, the typed comments are registered.
- When the  key is pressed midway through comments, to enter the rest of comments later, the data typed before the  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-12.3.2).
- 2 Enter the EDIT mode.
- 3 Press  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  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  and enter the initial value of N.
- 6 Press  key.
- 7 Enter each word of a block.
- 8 Press  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.

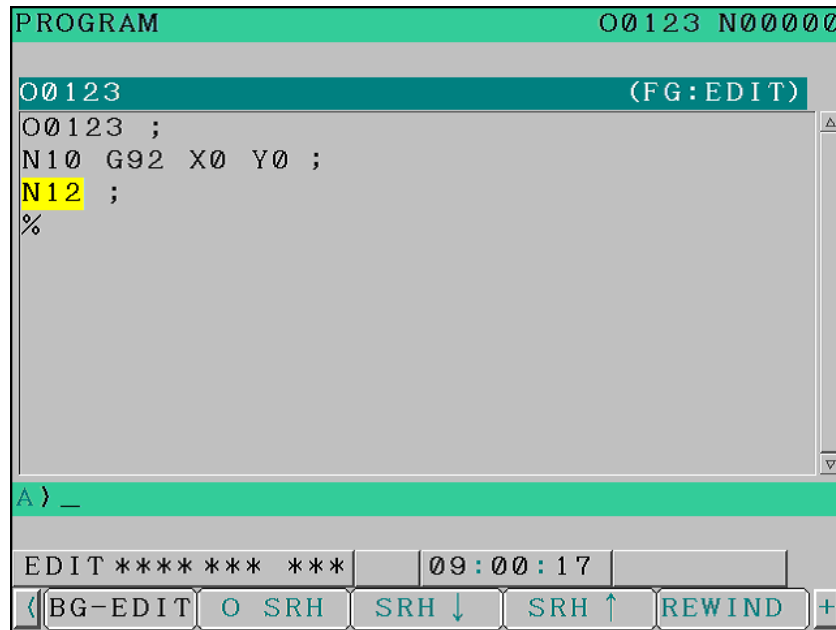


Fig. 9.2 (a)

- 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 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.

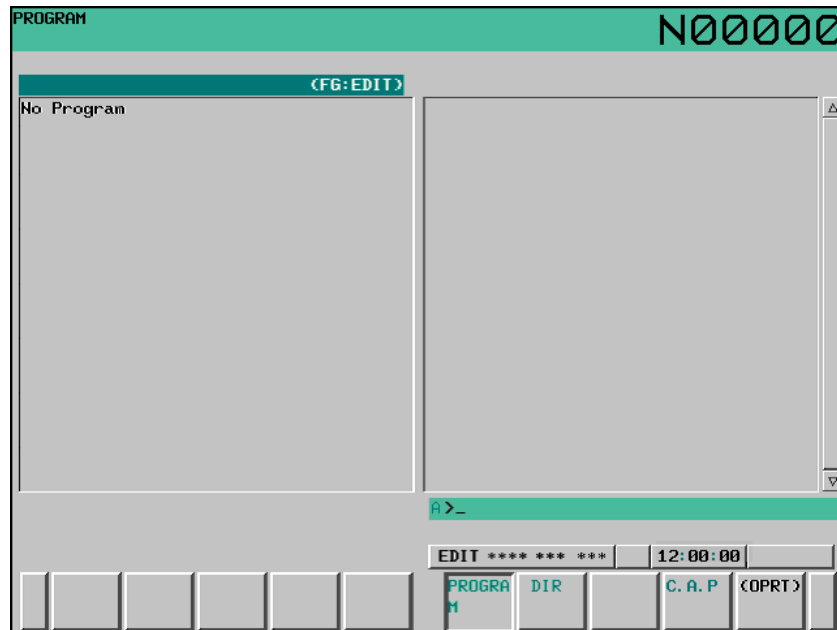


Fig. 9.3 (a) Program screen (with no program registered)

If a program is registered, the program currently selected is displayed.

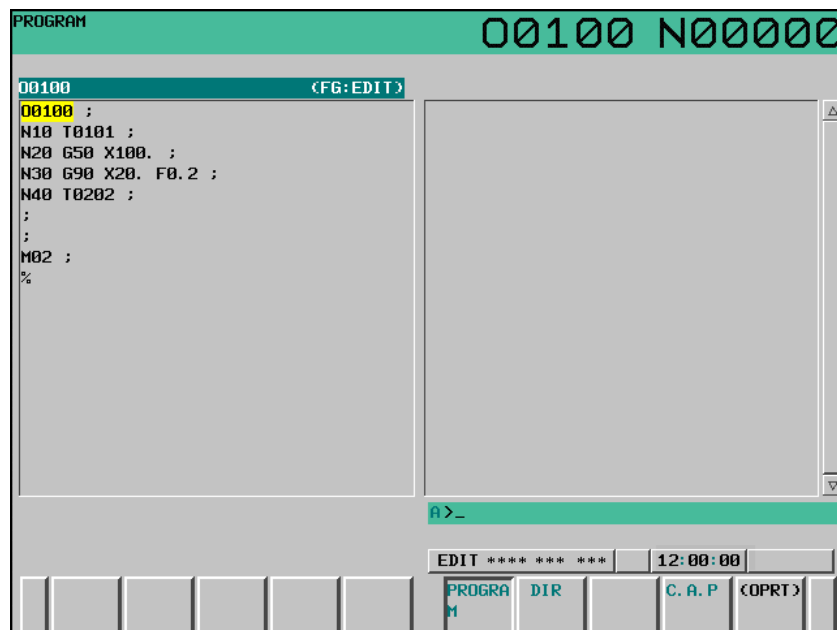


Fig. 9.3 (b) Program screen (with a program registered)

- 3 Key in the program number of a program to be registered after keying in address O, then press  key. For example, when a program with program number 10 is to be registered, key in O10 , then press  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.)

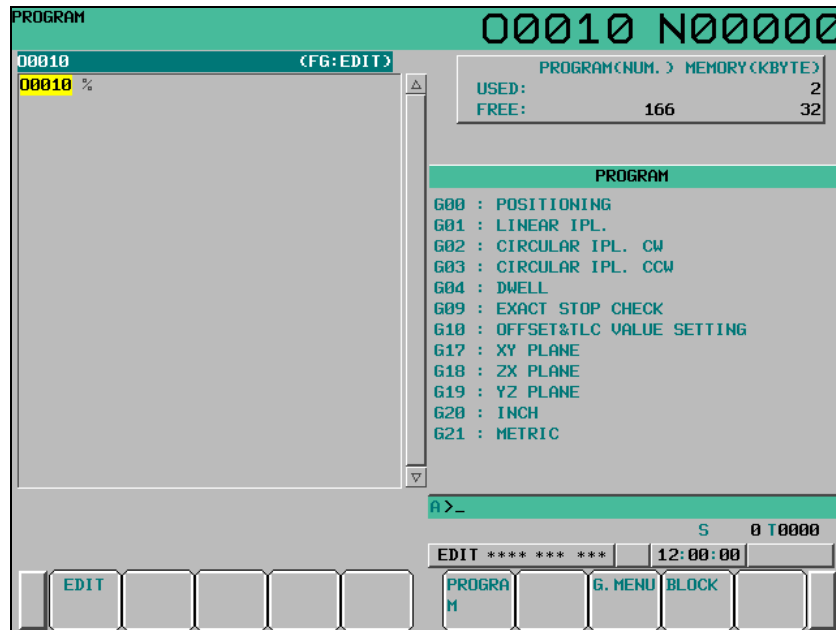


Fig. 9.3 (c) G code menu screen

- 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.

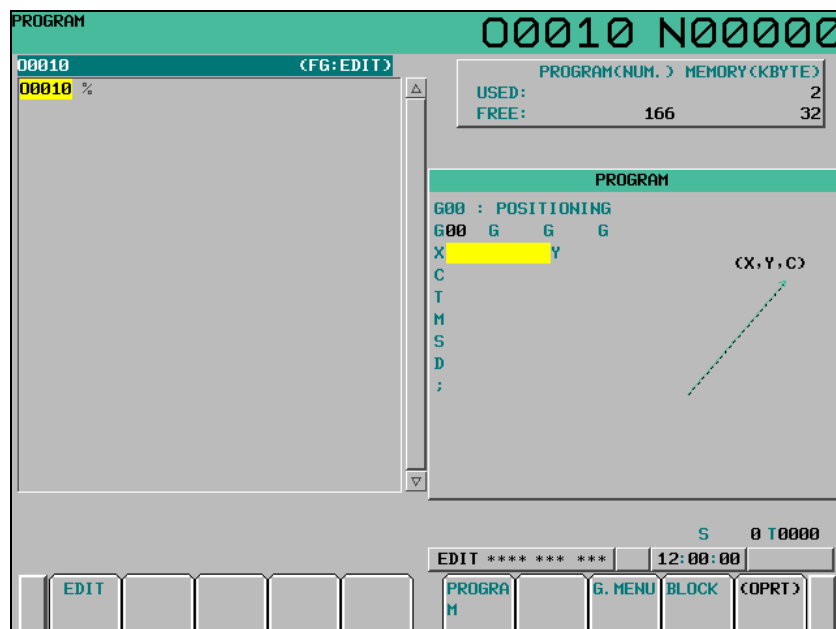


Fig. 9.3 (d) G code details screen (G00)

When no keys are pressed, the standard details screen is displayed.

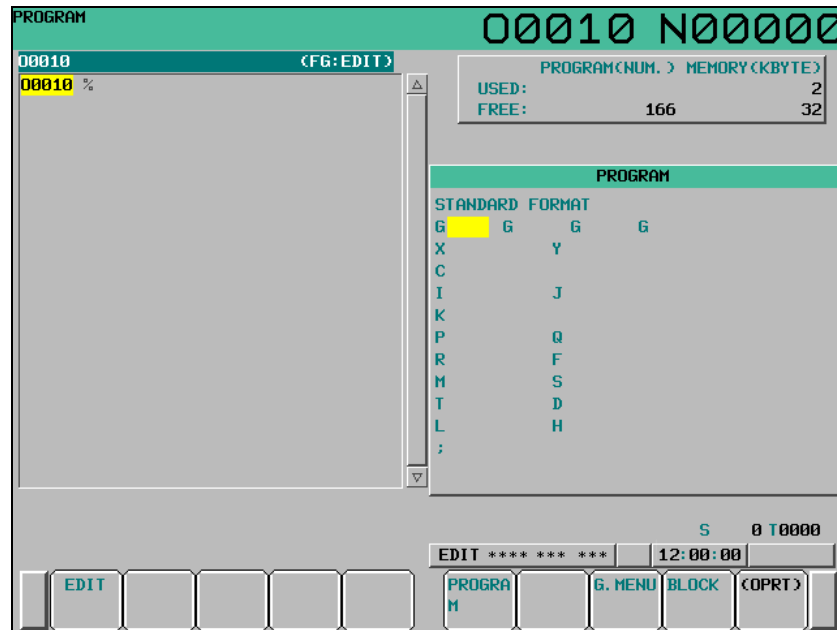


Fig. 9.3 (e) G code details screen (standard details screen)

- 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  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  or  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  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  key. This operation replaces an entire block of a program.

Procedure 3: Inserting a block

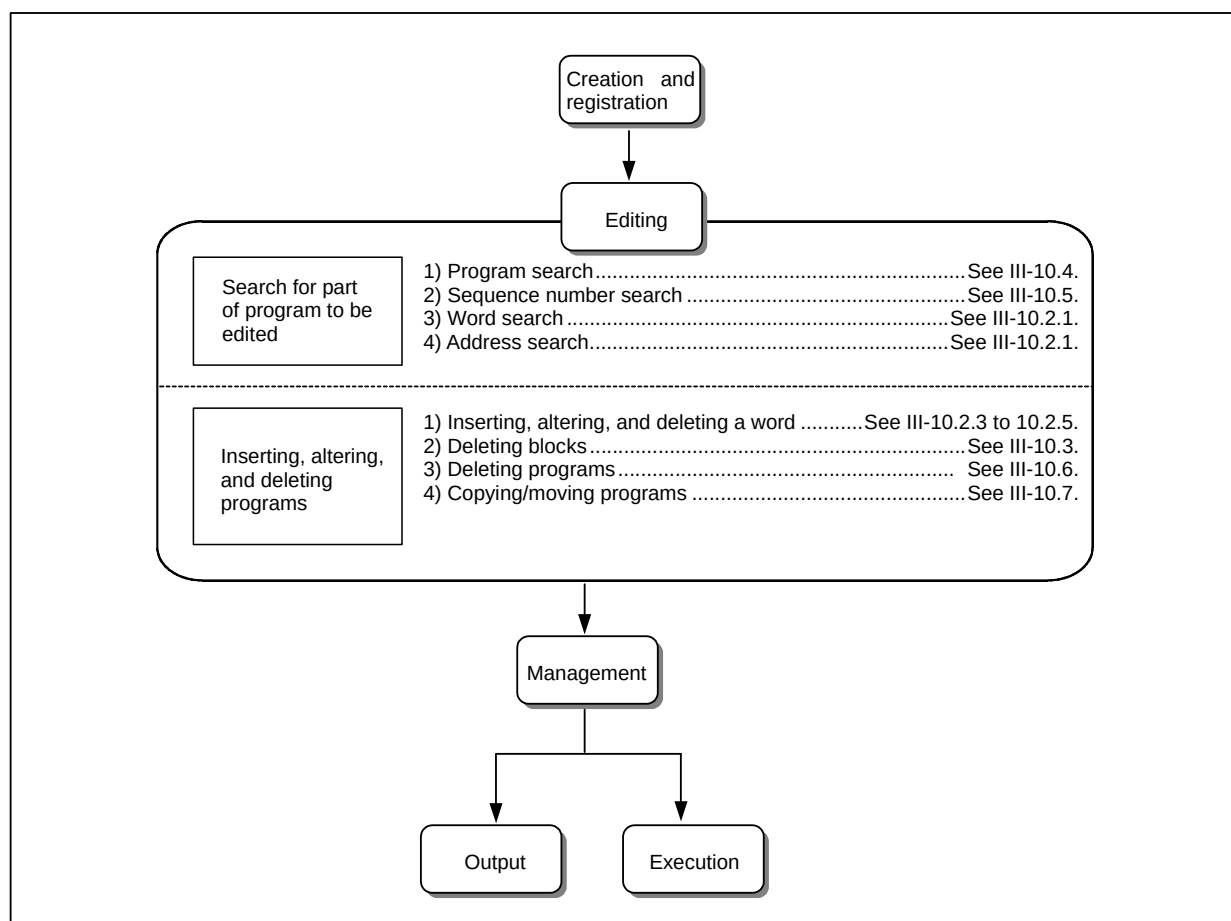
- 1 On the conversational screen, display the block immediately before a new block is to be inserted, by using the page keys. On the program screen, move the cursor with the page keys and cursor keys to immediately before the point where a new block is to be inserted.
- 2 Press the soft key [G.MENU] 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  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  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.

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. 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 .
- 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-10.2.1.
- 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.

The word "word," when used in the description of editing, means an editing unit according to the precise definition.

**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

A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

Procedure for scanning a program

- 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: Scanning X100.



Fig. 10.2.1 (a)

- 3 Holding down the cursor key or scans words continuously.
- 4 The first word of the next block is searched for when the cursor key is pressed.
- 5 The first word of the previous block is searched for when the cursor key is pressed.
- 6 Holding down the cursor key or moves the cursor to the head of a block continuously.
- 7 Pressing the page key displays the next page and searches for the first word of the page.
- 8 Pressing the page key displays the previous page and searches for the first word of the page.
- 9 Holding down the page key or displays one page after another.

Procedure for searching a word

Example of searching for S12

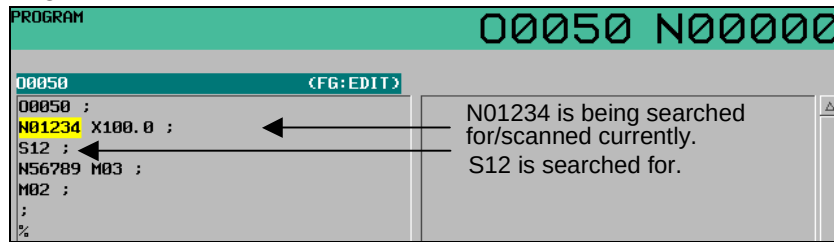


Fig. 10.2.1 (b)

- 1 Press soft key [SEARCH].
- 2 Key in address S.
- 3 Key in 1 2.
 - S12 cannot be searched for if only S1 is keyed in.
 - S09 cannot be searched for by keying in only S9.
To search for S09, be sure to key in S09.
- 4 Pressing soft key [$\downarrow$ SEARCH] starts searching forward from the cursor position. Pressing [$\uparrow$ SEARCH] starts searching backward.
- 5 To search for the same word successively, press [$\downarrow$ SEARCH] or [$\uparrow$ SEARCH].

Procedure for searching an address

Example of searching for M03

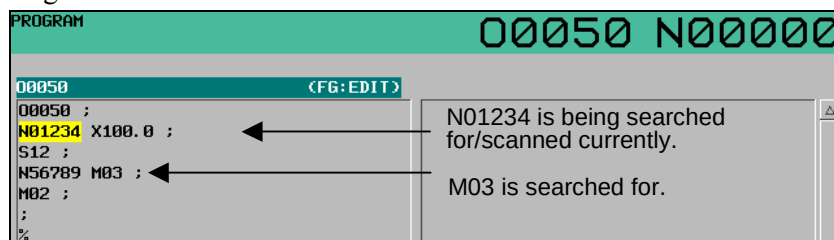


Fig. 10.2.1 (c)

- 1 Pressing the [SEARCH] key
- 2 Key in address M.
- 3 Pressing soft key [$\downarrow$ SEARCH] starts searching forward from the cursor position. Pressing [$\uparrow$ SEARCH] starts searching backward.
- 4 To search for the same word successively, press [$\downarrow$ SEARCH] or [$\uparrow$ SEARCH].

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 RESET 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 Search for the program number.

- 1 When the program screen is selected in MEMORY or EDIT mode, enter a program number or program name.

When entering a program number, press address key  then type the program number.

- 2 Press soft key [PROGRM 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 [LINE SEARCH].
- 4 Press soft key [TOP].

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  key.

Example of Inserting T15

- 1 Search for or scan X100.

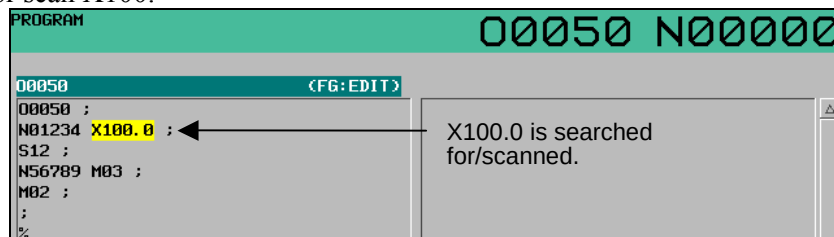


Fig. 10.2.3 (a)

- 2 Key in   .
- 3 Press the  key.

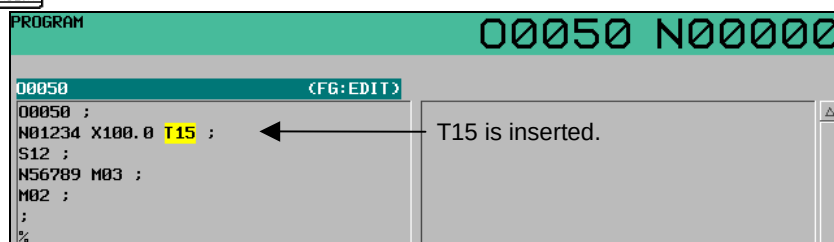


Fig. 10.2.3 (b)

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  key.

Example of changing T15 to M15

- 1 Search for or scan T15.

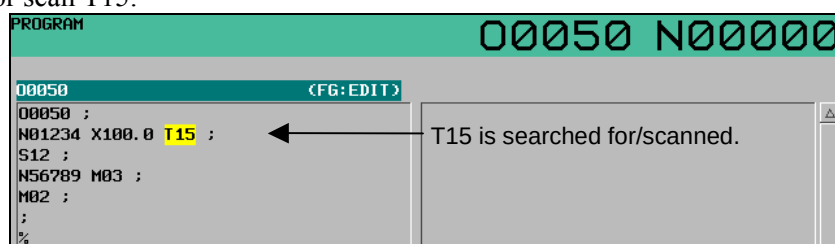


Fig. 10.2.4 (a)

- 2 Key in   .
- 3 Press the  key.

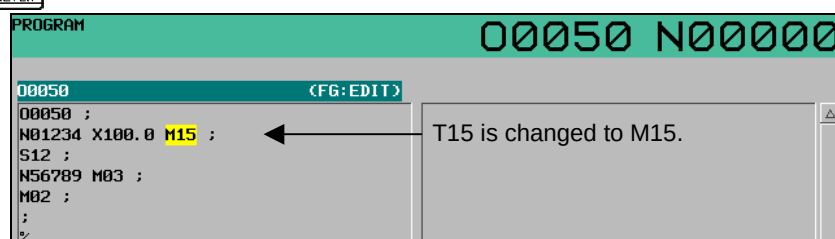


Fig. 10.2.4 (b)

10.2.5 Deleting a Word

Procedure for deleting a word

- 1 Search for or scan a word to be deleted.
- 2 Press the  key.

Example of deleting X100.0

- 1 Search for or scan X100.0.

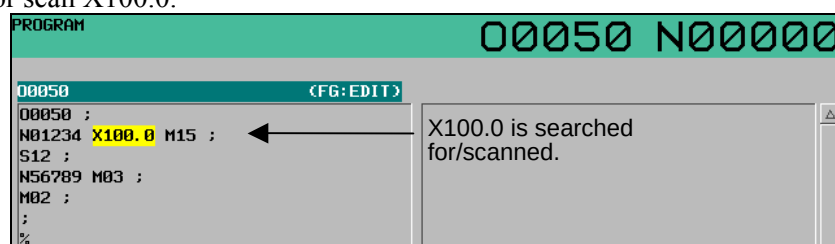


Fig. 10.2.5 (a)

- 2 Press the  key.

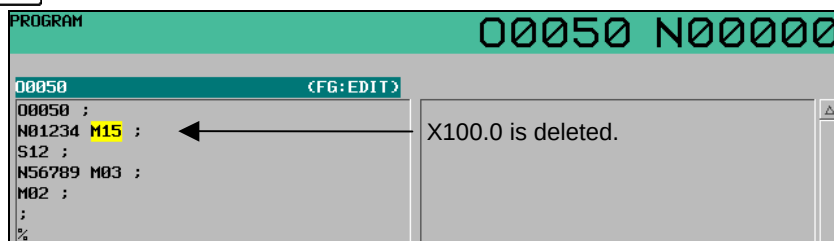


Fig. 10.2.5 (b)

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  key.
- 3 Press the editing key .

Example of deleting a block of N01234

- 1 Search for or scan N01234.

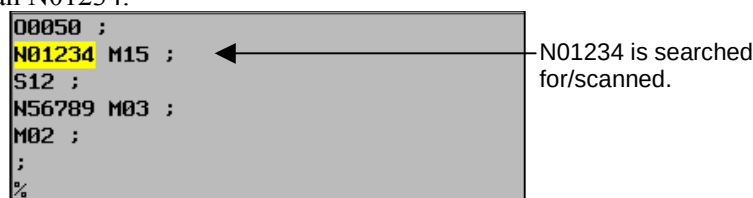


Fig. 10.3.1 (a)

- 2 Press the  key.

- 3 Press the editing key .

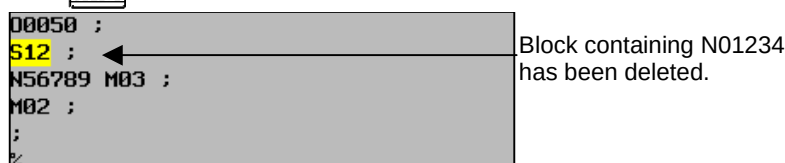


Fig. 10.3.1 (b)

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  key as many times as the number of blocks that you want to delete.
- 3 Press the editing key .

Example of deleting blocks from N01234 to the EOB of a block which is two blocks ahead

- 1 Search for or scan N01234.

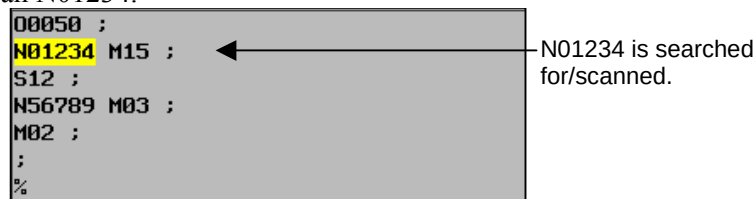


Fig. 10.3.2 (a)

- 2 Press  .
- 3 Press the editing key .

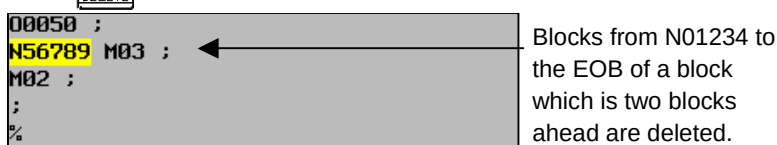


Fig. 10.3.2 (b)

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
- 1 Select EDIT or MEMORY mode.
 - 2 Press function key  to display the program screen.
 - 3 Enter a program number or program name. When entering a program number, press address key  then type the program number.
 - 4 Press soft key [PROGRM SEARCH]. Alternatively, press the cursor key  or .
 - 5 Upon completion of search operation, the program number searched for is displayed in the upper-right corner of the screen
If the program is not found, alarm PS0071 occurs.
- Method 2
- 1 Select EDIT or MEMORY mode.
 - 2 Press function key  to display the program screen.
 - 3 Press soft key [PROGRM SEARCH].

- 4 Then, press soft key [PREV PROGRM] or [NEXT PROGRM].
Pressing [PREV PROGRM] searches for the previous program on the folder, and pressing [NEXT PROGRM] searches for the next program.

Method 3

This method searches for the program number (0001 to 0031) corresponding to a signal on the machine tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

- 1 Select MEMORY mode.
- 2 Set the reset state.
 - The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)
- 3 Set the program number selection signal on the machine tool side to a number from 01 to 31.
- 4 Press the cycle start button.
 - When the signal on the machine tool side represents 00, program number search operation is not performed.
 - If the program corresponding to a signal on the machine tool side is not registered, alarm DS0059 is raised.

Method 4

When bit 4 (PGS) of parameter No. 11308 is 1, search for an O number program by specifying digits only.

- 1 Select EDIT or MEMORY mode.
- 2 Press function key  to display the program screen.
- 3 Enter the digits of a program number. Address key  need not be pressed before the digits.
- 4 Press soft key [PROGRM SEARCH]. Alternatively, press the cursor key  or .

Example

- When bit 4 (PGS) of parameter No. 11308 is 1
Specify "1000" → Search for program O1000.
Specify "ABC" → Search for program ABC.
- When bit 4 (PGS) of parameter No. 11308 is 0
Specify "1000" → Search for program 1000.
Specify "ABC" → Search for program ABC.

When bit 4 (PGS) of parameter No. 11308 is 1, it is not possible to search for a program with any name consisting of digits only.

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.

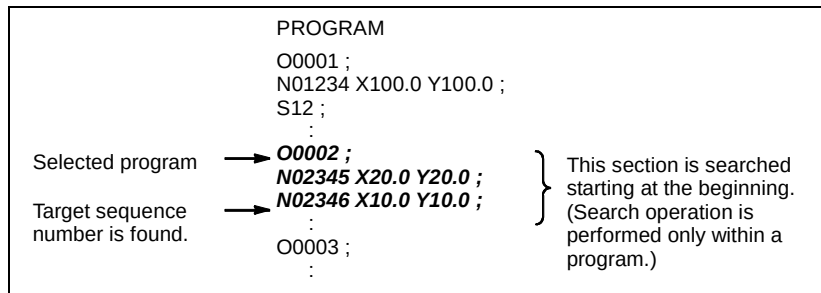


Fig. 10.5 (a)

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.

- Checking during search

During search operation, the following checks are made:

- Optional block skip

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.

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.

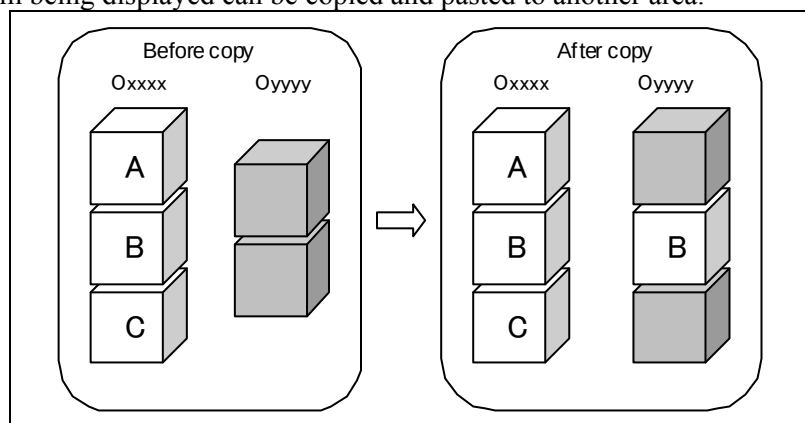
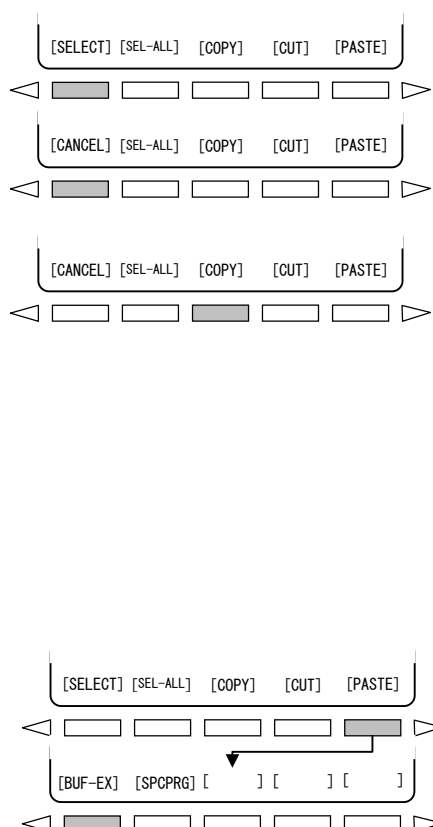


Fig. 10.7.1 (a) Copying a part of a program

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.

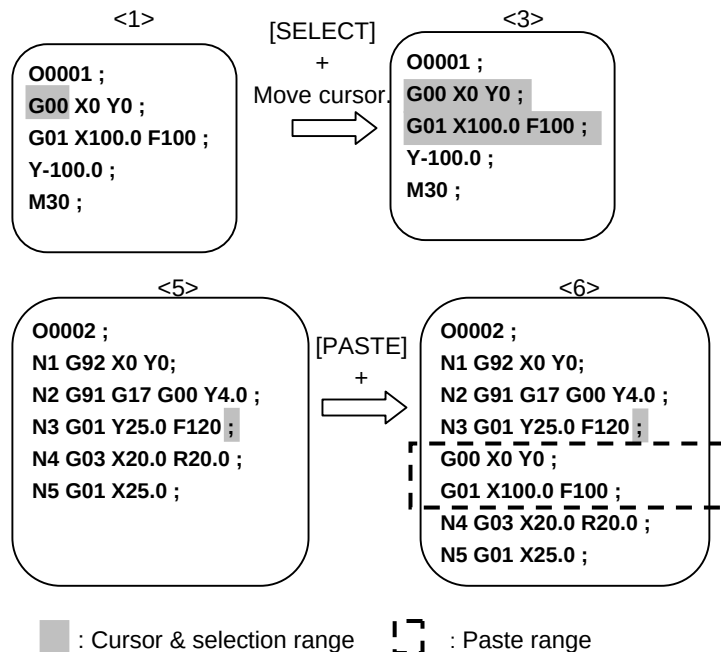
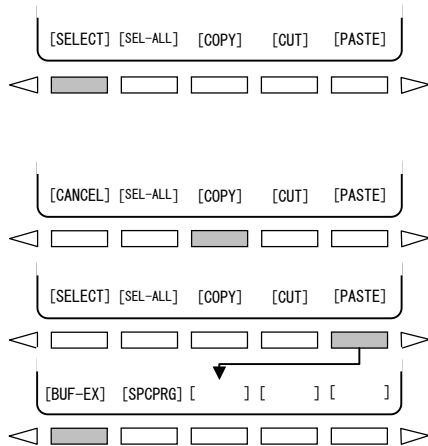
Procedure for copying a part of a program



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.

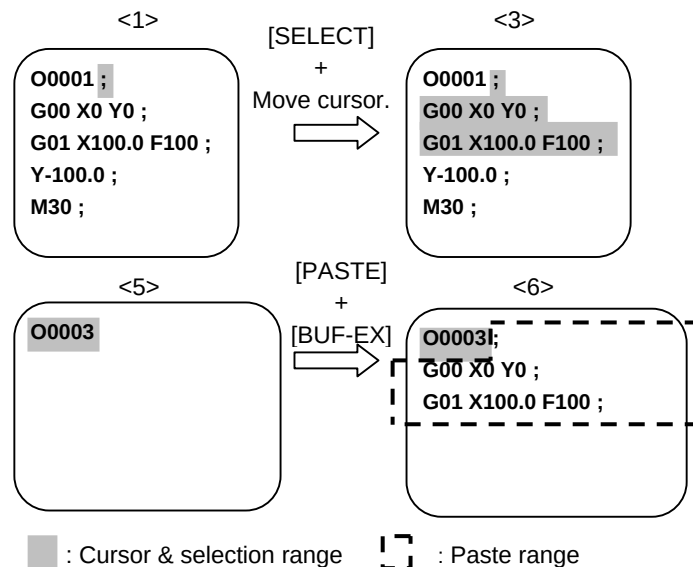
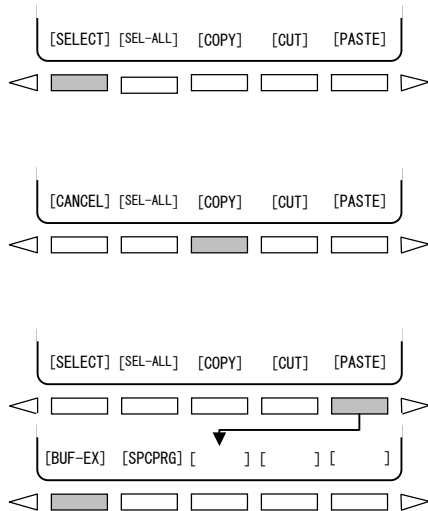
1. Display O0001 then move the cursor to a desired copy start position. (<1>)
2. Press soft key [SELECT].
3. When the cursor is moved, the range from the copy start position to the current cursor position is selected and displayed in the same color as the cursor color. (<3>)
4. Press soft key [COPY]. 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.

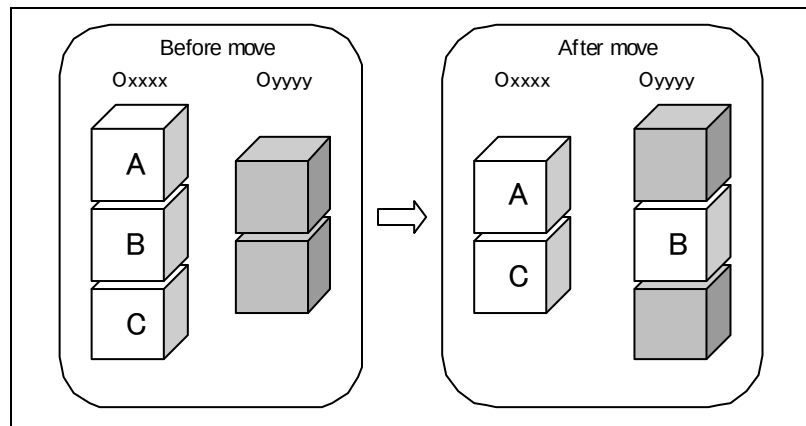
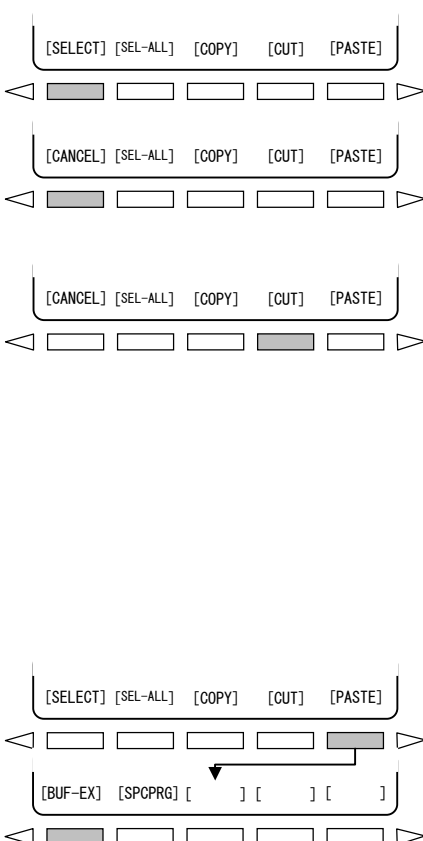


Fig. 10.7.2 (a) Moving a part of a program

In Fig. 10.7.2 (a), program B of program number xxxx is cut and pasted into the program of program number yyyy.

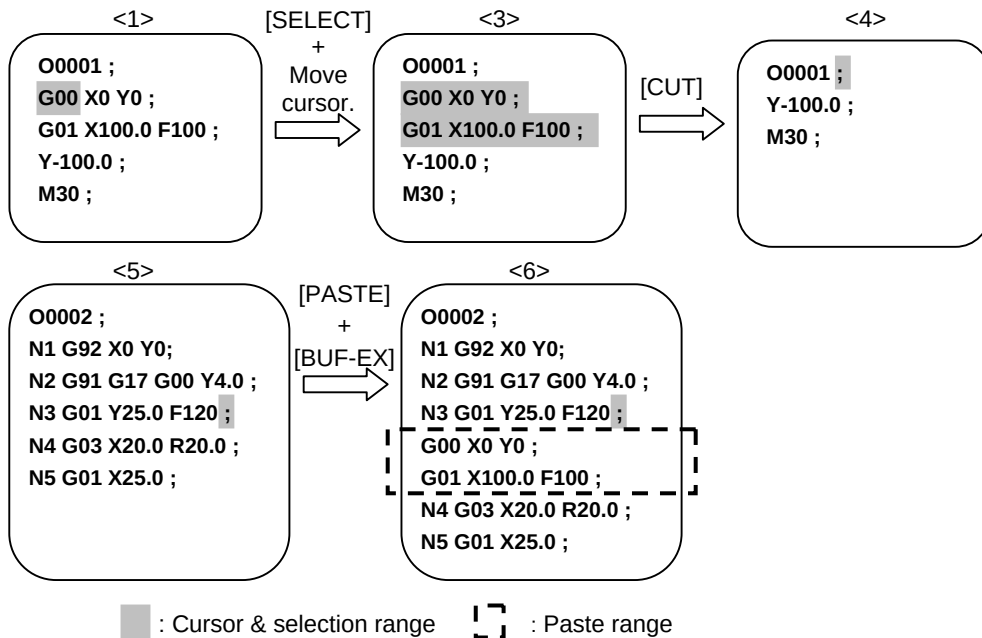
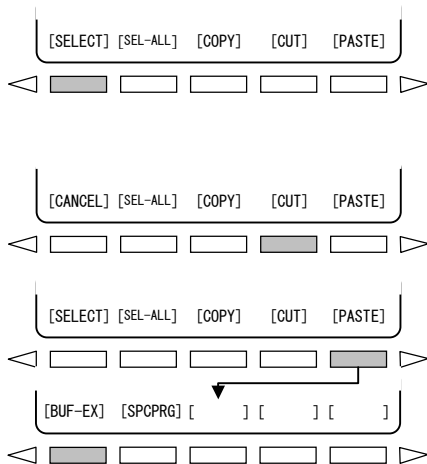
Procedure for moving a part of a program



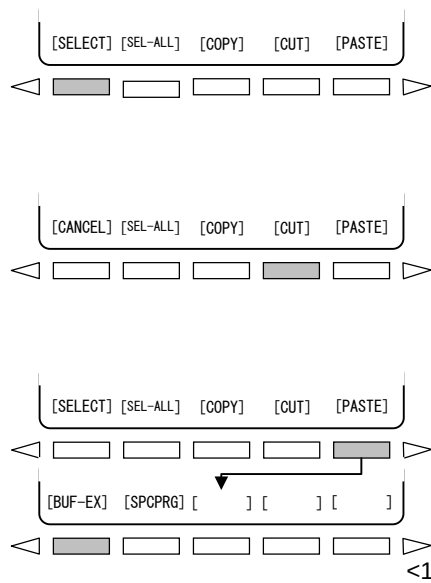
1. Move the cursor to a desired cut start position.
2. Press soft key [SELECT].
3. When the cursor is moved, the range from the cut start position to the current cursor position is selected and displayed in the same color as the cursor color.
To cancel the selection, press soft key [CANCEL].
4. Press 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.

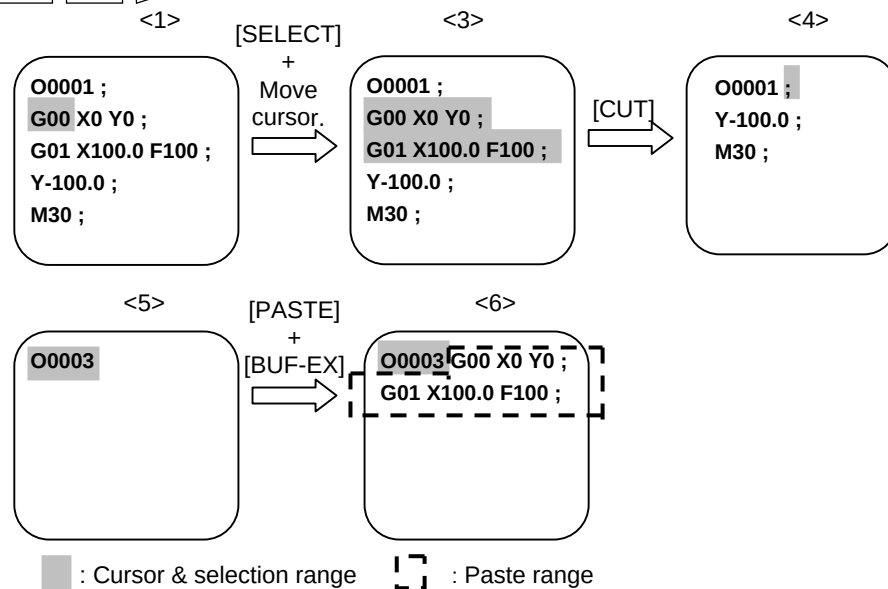
1. Display O0001 then move the cursor to a desired cut start position. (<1>)
2. Press soft key [SELECT].
3. When the cursor is moved, the range from the cut start position to the current cursor position is selected and displayed in the same color as the cursor color. (<3>)
4. Press soft key [CUT]. The 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.

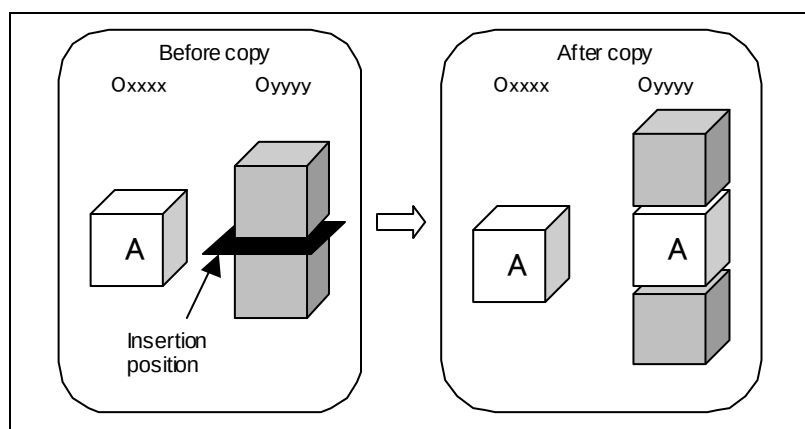
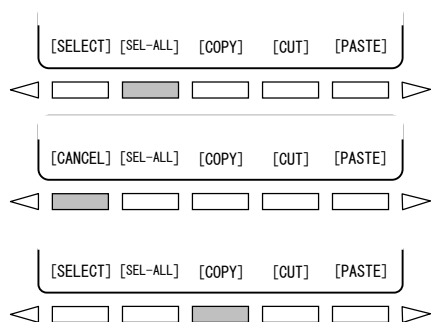


Fig. 10.7.3 (a)

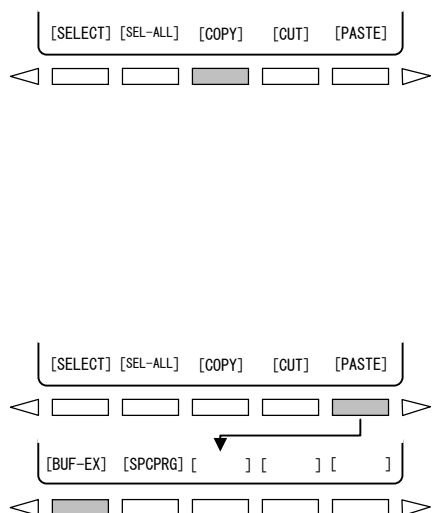
In Fig. 10.7.3 (a), the program of program number xxxx is inserted into the program of program number yyyy. The program of program number xxxx remains unchanged before and after the insertion.

Procedure for copying an entire program



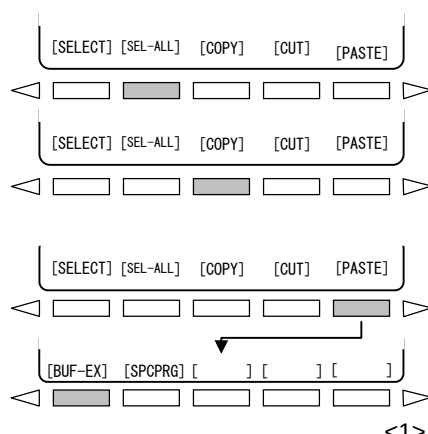
1. Display a program to be copied.
2. Press soft key [SEL-ALL]. The entire program except its O number is selected and displayed in the same color as the cursor color.

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>)

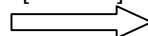
<1>

```
O0001 ;
G00 X0 Y0 ;
G01 X100.0 F100 ;
Y-100.0 ;
M30 ;
```

<3>

```
O0002 ;
N1 G92 X0 Y0;
N2 G91 G17 G00 Y4.0 ;
N3 G01 Y25.0 F120 ;
N4 G03 X20.0 R20.0 ;
N5 G01 X25.0 ;
```

[PASTE]
+
[BUF-EX]



<4>

```
O0002 ;
N1 G92 X0 Y0;
N2 G91 G17 G00 Y4.0 ;
N3 G01 Y25.0 F120 ;
G00 X0 Y0 ;
G01 X100.0 F100 ;
Y-100.0 ;
M30 ;
N4 G03 X20.0 R20.0 ;
N5 G01 X25.0 ;
```

■ : Cursor & selection range □ : Paste range

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.

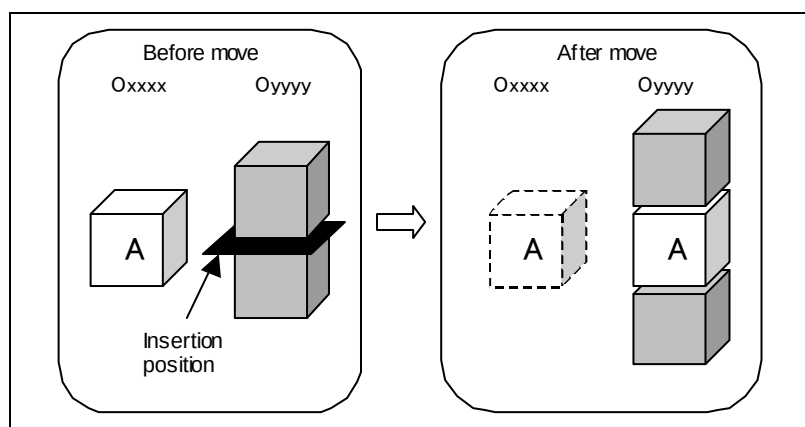
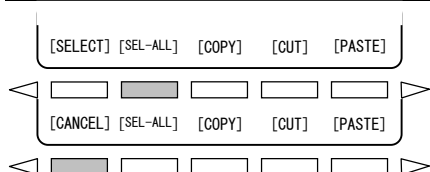


Fig. 10.7.4 (a)

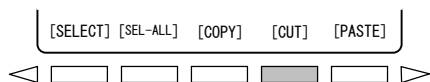
In Fig. 10.7.4 (a), the program of program number xxxx is inserted into the program of program number yyyy. The contents of the program of program number xxxx are deleted. However, the deleted program remains as an empty program.

Procedure for moving an entire program

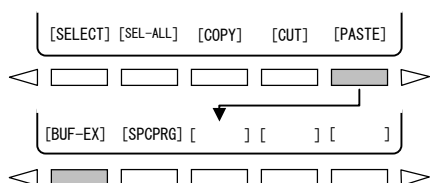


1. Display a program to be cut.
2. Press soft key [SEL-ALL]. The entire program except its O number is selected and displayed in the same color as the cursor color.

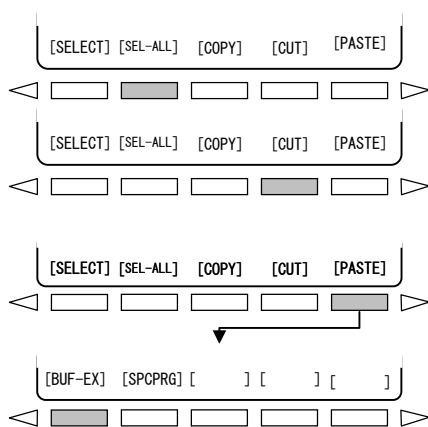
To cancel the selection, press soft key [CANCEL].



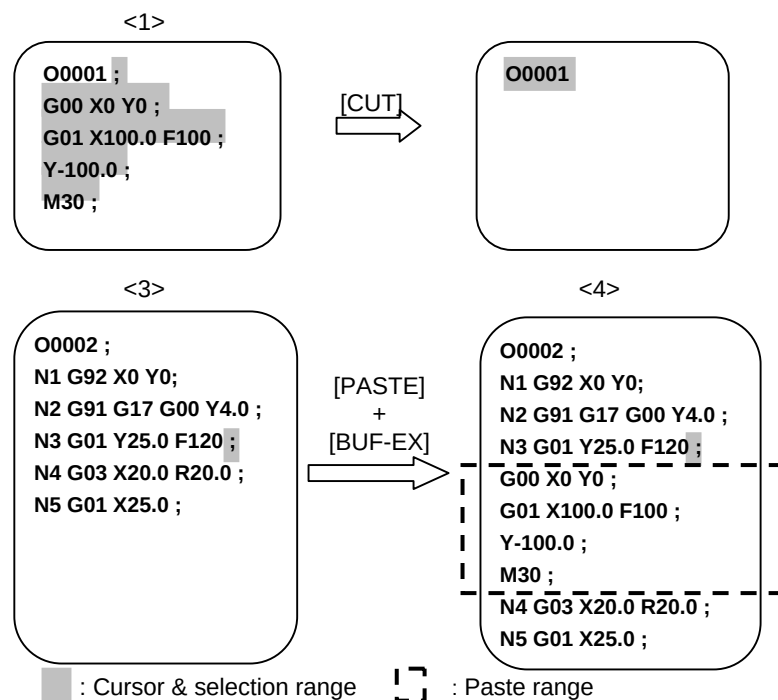
3. Press soft key [CUT]. The entire program is stored in the copy buffer. At the same time, the contents of the program are cut. If soft key [CUT] is pressed with no program range selected, the warning message "WORDS ARE NOT SELECTED" is displayed and the copy buffer is cleared. If the program size of 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.

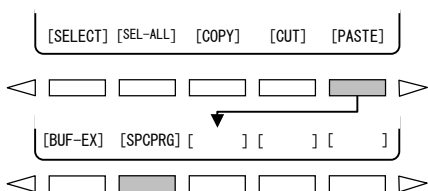
10.7.5 Copy Specifying a Program Number

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.

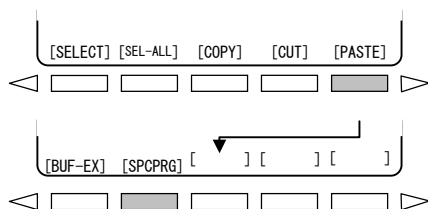
Even when the size of a program exceeds the capacity of the copy buffer, the entire program can be copied.

Procedure for copying a program by specifying its program number

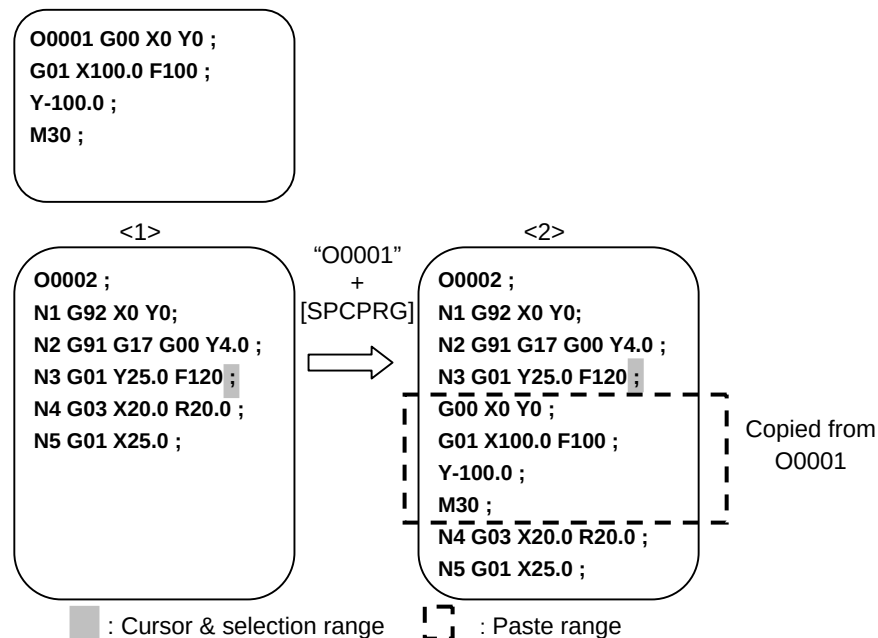


1. Display a paste target program then move the cursor immediately before the desired paste position.
2. Press soft key [PASTE] then key in the program number of a program to be pasted. Next, press [SPCPRG]. The program of the program number keyed in is pasted after the cursor. When [SPCPRG] is pressed with no program number selected, the warning message "PROGRAM IS NOT SPECIFIED" is displayed.

Example) O0001 is copied and pasted to O0002.



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>)

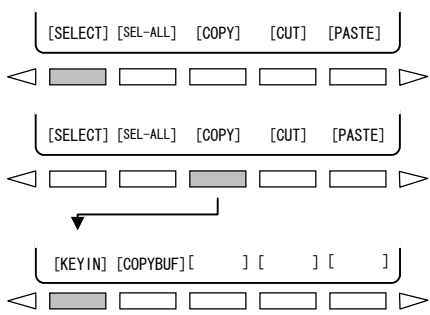


For convenience of explanation, the cursor of <2> is placed at the same position as for <1>. Actually, the cursor moves to the last word of pasted contents.

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.

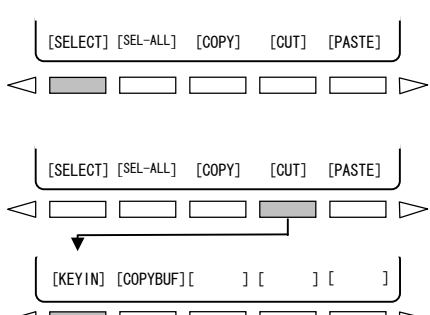
Procedure: Copying to the key-in buffer

- 
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.

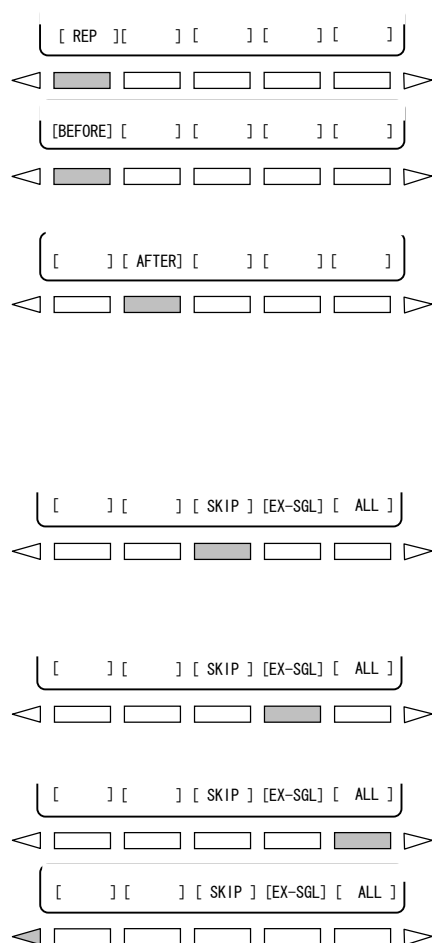
NOTE

- 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 .
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) Head positions where the cursor is placed

```

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.

10.10 PASSWORD FUNCTION

The password function locks bit 4 (NE9) of parameter No. 3202, which protects programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute, according to the settings in two parameters, PASSWORD (parameter No. 3210) and KEYWD (parameter No. 3211). In the locked state, parameter NE9 cannot be set to 0. In this case, the protection of programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute cannot be released unless a 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 PS0100 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 parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD ≠ 0 and PASSWD ≠ KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

- 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 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. 10.11 (a), characters such as "@" and "(" appear as soft keys.



Fig. 10.11 (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 10.11 (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.

() ? * & @ _

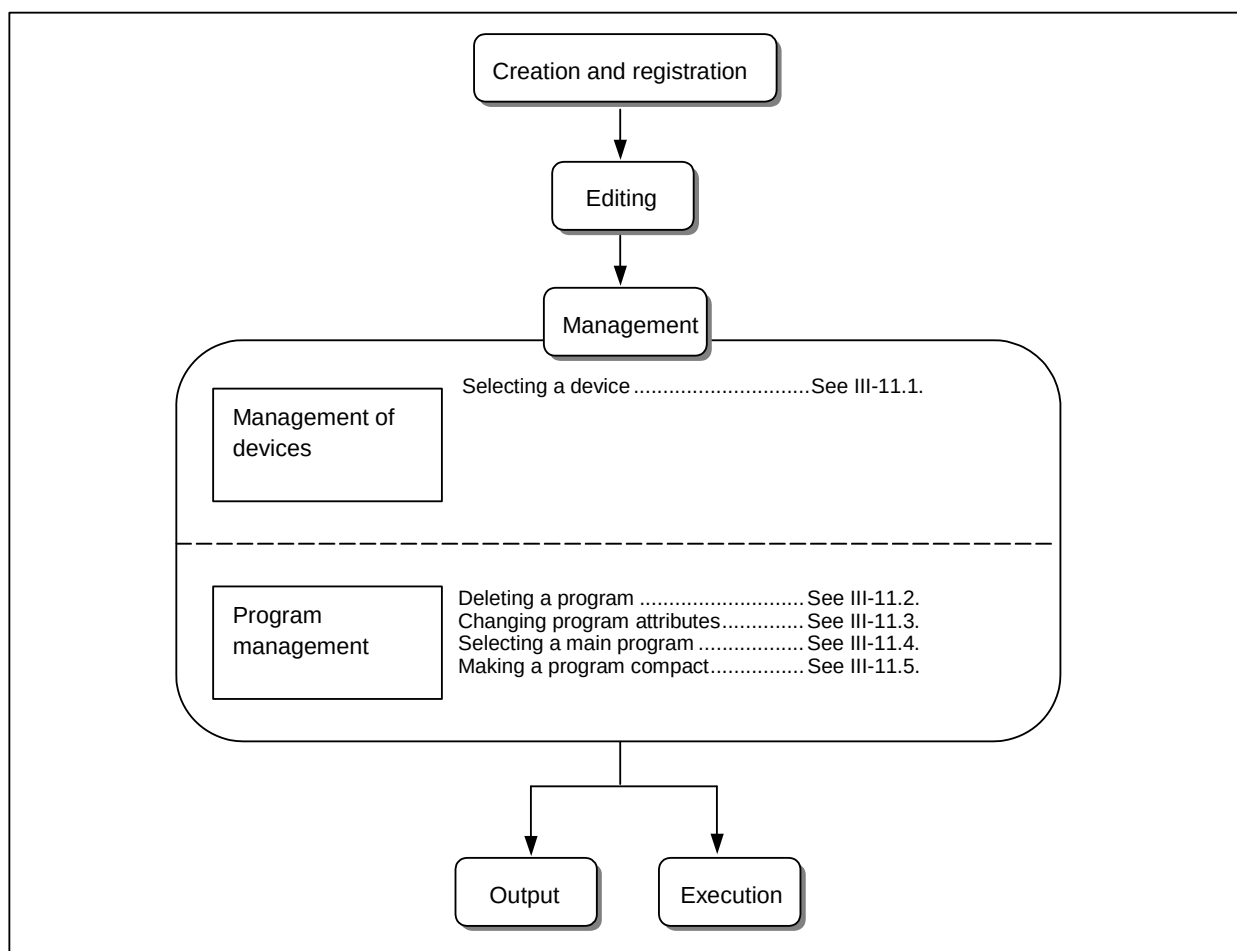
11 PROGRAM MANAGEMENT

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 [DETACH].

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.
- 2 This operation is enabled only in EDIT mode or MEM mode.
- 3 When the default folder is a folder in a program storage file stored on the memory card, it is changed to "//CNC_MEM/" by a removal 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/T code/particular address)**- Macro program (call using G code/M code)**

The following subprogram/macro program calls a program from the CNC_MEM device (CNC program storage memory):

- Subprogram call using M code/T code/particular addresses
- Macro call using G code/M code

NOTE

For a memory card program, subprogram call using M code/T code/particular addresses 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 input/output data to and from a memory card
- PMC data I/O screen
Display of the contents of a memory card, and input/output to and from a memory card
- Program directory screen
Input/output 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:
 - When the alarm (SR1964) is issued
Reset the alarm after performing a "removal" operation.
 - When the alarm (IO1030) is issued
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 (Table 11.1.1 (a)):

Table 11.1.1 (a)

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	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

11.1.2 Selecting a From Cassette as a Device

The Handy File can be connected to the NC to display a list of programs in a floppy cassette.

Procedure for selecting a device

- 1 Set the relevant parameters so that communication with the Handy File is enabled.
- 2 Press the function key .
- 3 Press the soft key [DIR].
- 4 Press the operation selection soft key [(OPRT)].
- 5 Hold down the continuous menu key  until the soft key [DEVICECHANGE] (or [DEVICE] when an 8.4-inch display unit is used) is displayed.
- 6 Press the soft key [DEVICECHANGE].
- 7 Press the soft key [FLOPPY].

Limitation

- Floppy disk directory display during schedule operation

During schedule operation, directories in a floppy disk cannot be displayed in a background edit.

- Operation of creation, edition, and management of a program

When "floppy cassette as a device" is selected, operation of creation, edition, and management of a program is below (Table 11.1.2 (a)):

Table 11.1.2 (a)

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable
Inserting, alteration, and deletion a Word	Unusable

Item	Usable
Deletion of a block	Unusable
Program search	Usable
Sequence number search	Unusable
Deletion of a program	Usable
Editing a custom macro	Unusable
Password function	Unusable
Selecting a device	Usable
Selecting a main program	Unusable
Making a program compact	Unusable
Input/output of program	Usable

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 .
- 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 .
- 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  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 .
- 3 Press the soft key [DIR].
- 4 Key in the program number of a program to be made compact.
(Press the address key  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:

12.1	SCREENS DISPLAYED BY FUNCTION KEY		526
12.2	SCREENS DISPLAYED BY FUNCTION KEY		535
12.3	SCREENS DISPLAYED BY FUNCTION KEY		547
12.4	SCREENS DISPLAYED BY FUNCTION KEY		582
12.5	SCREENS DISPLAYED BY FUNCTION KEY		633
12.6	DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION 633		
12.7	SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION 636		

Explanation

- Screen transition chart

The screen transition for when each function key on the MDI unit 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  is pressed. In general, function key

 or  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  or  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

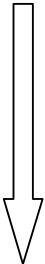
(10.4")

(1)	(2)	(3)	(4)	(5)	
ABSOLU TE	RELAT I VE	ALL	HANDLE	(OPRT)	+

(8.4")

ABS	REL	ALL	HNDL	(OPRT)	+

↓	↓	↓	↓	
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

(10.4")

(6)	(7)	(8)	(9)	(10)	
MONITO R				(OPRT)	+

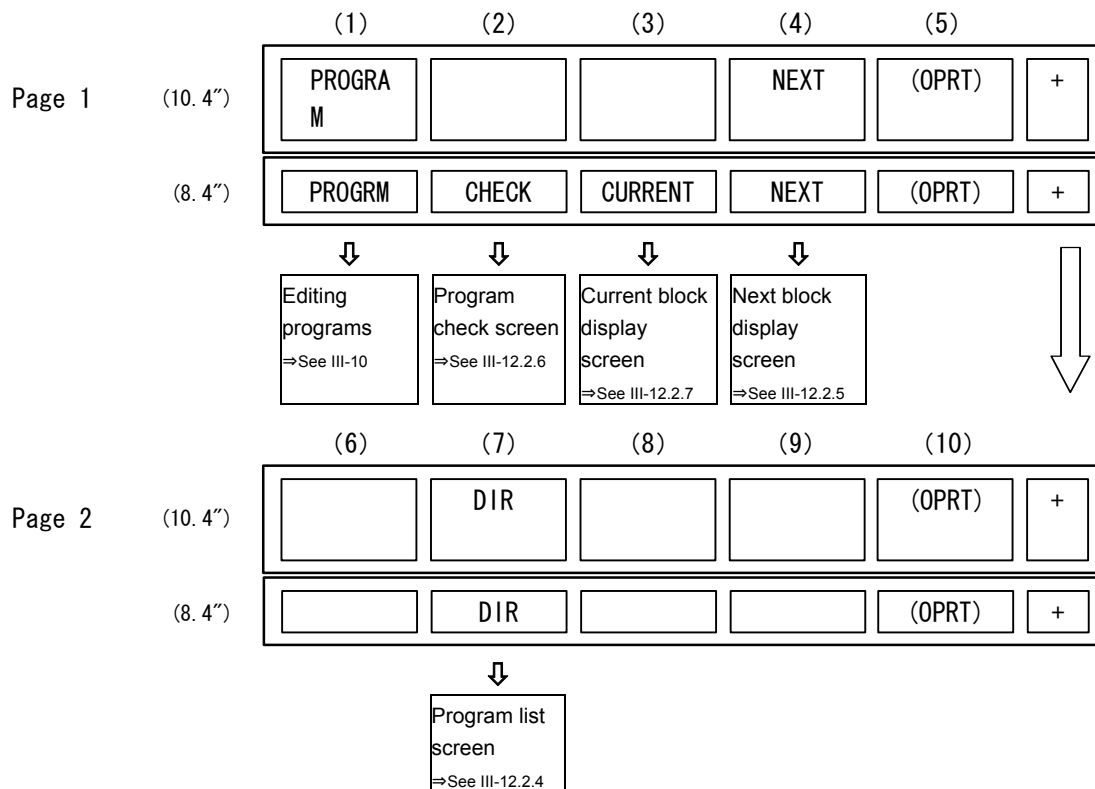
(8.4")

MONI				(OPRT)	+

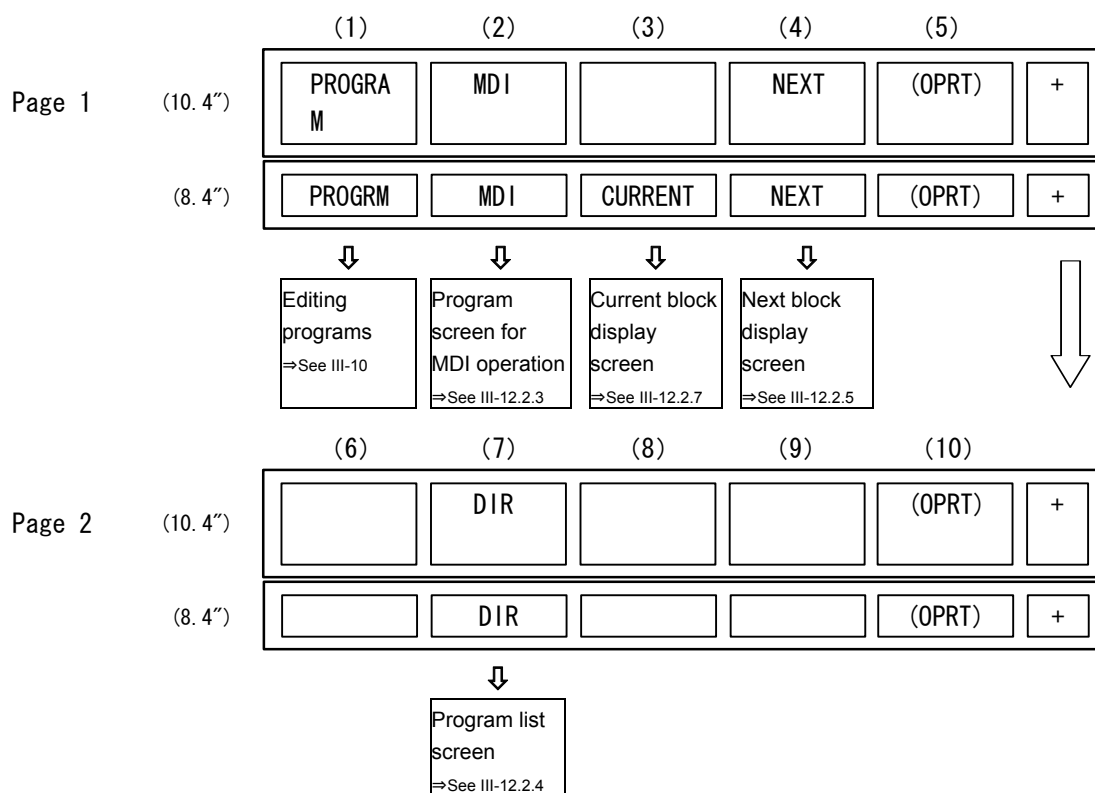
↓
Operating monitor display ⇒ See III-12.1.7

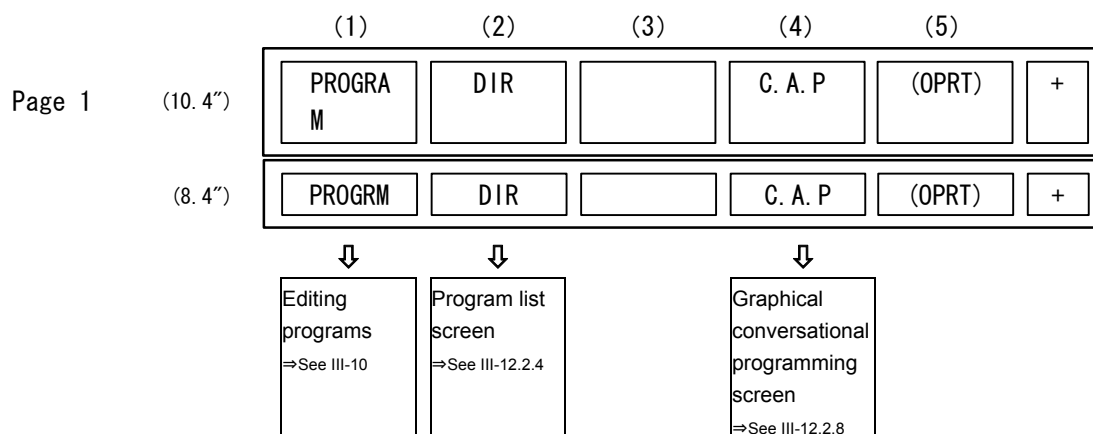
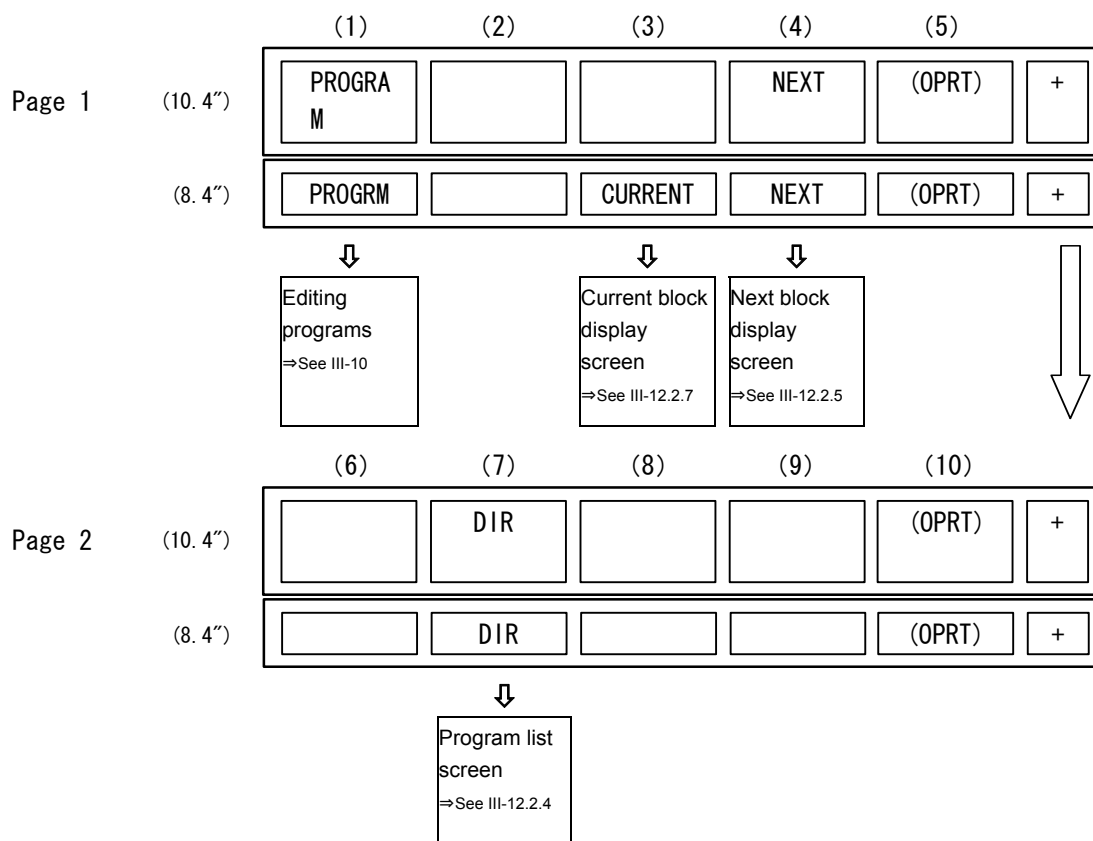
Screen displayed when the function key  is pressed (for 8.4/10.4-inch display unit)

In the MEM/RMT mode

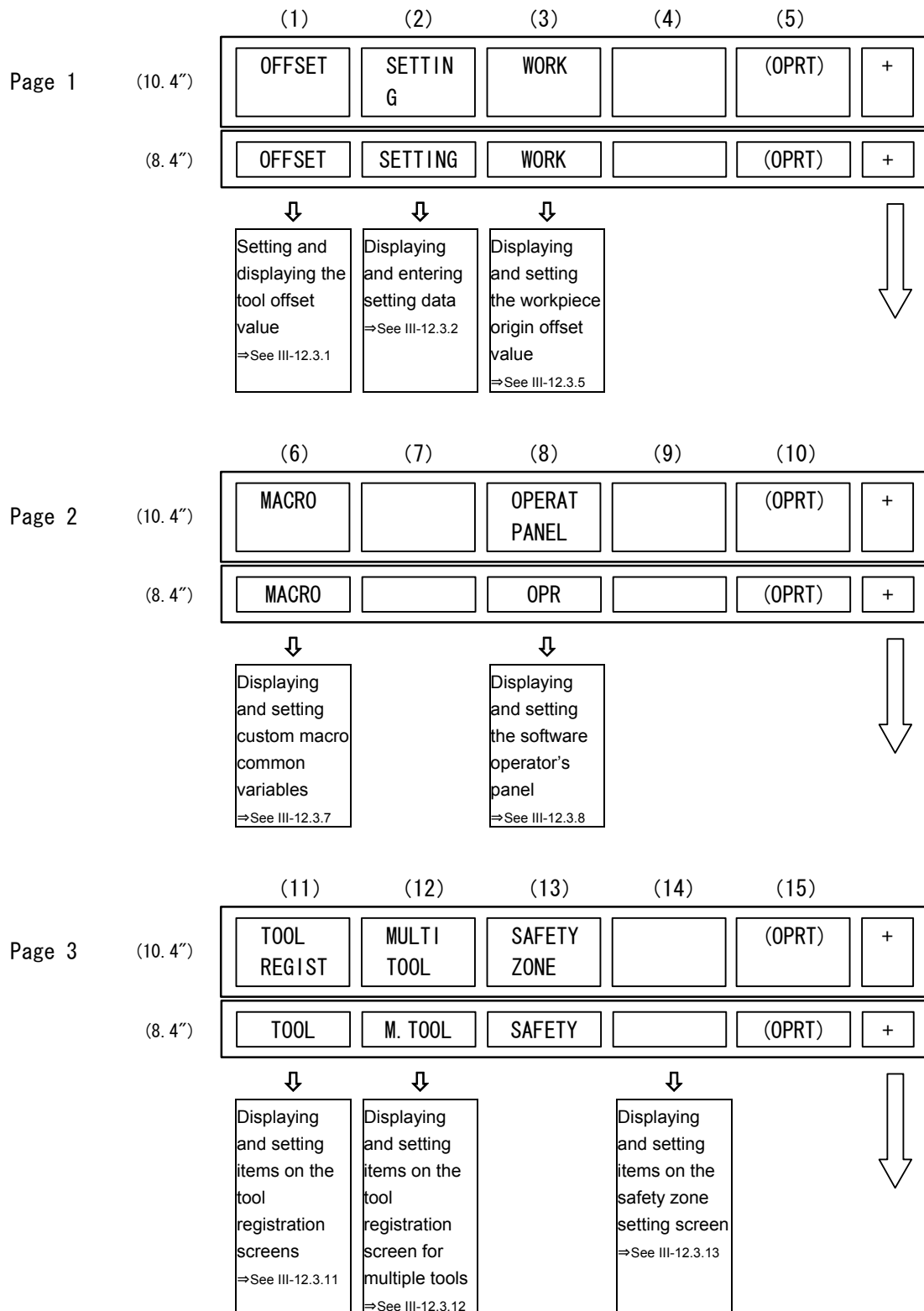


In the MDI mode



In the EDIT mode**In the JOG/HND/REF mode**

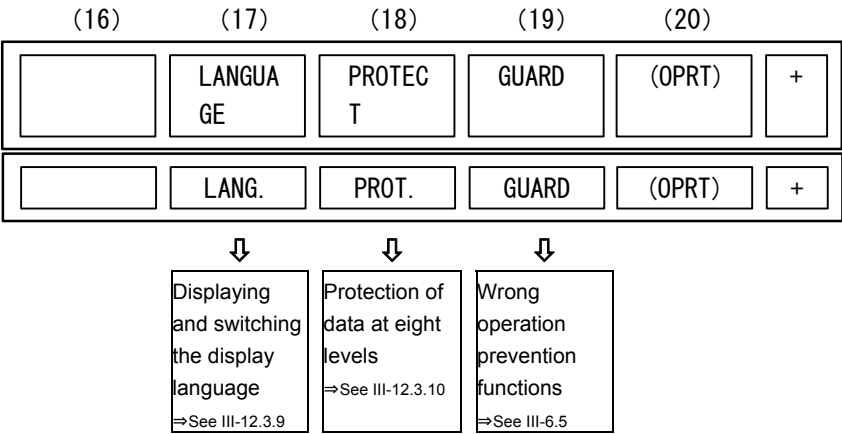
Screen displayed when the function key  is pressed (for 8.4/10.4-inch display unit)



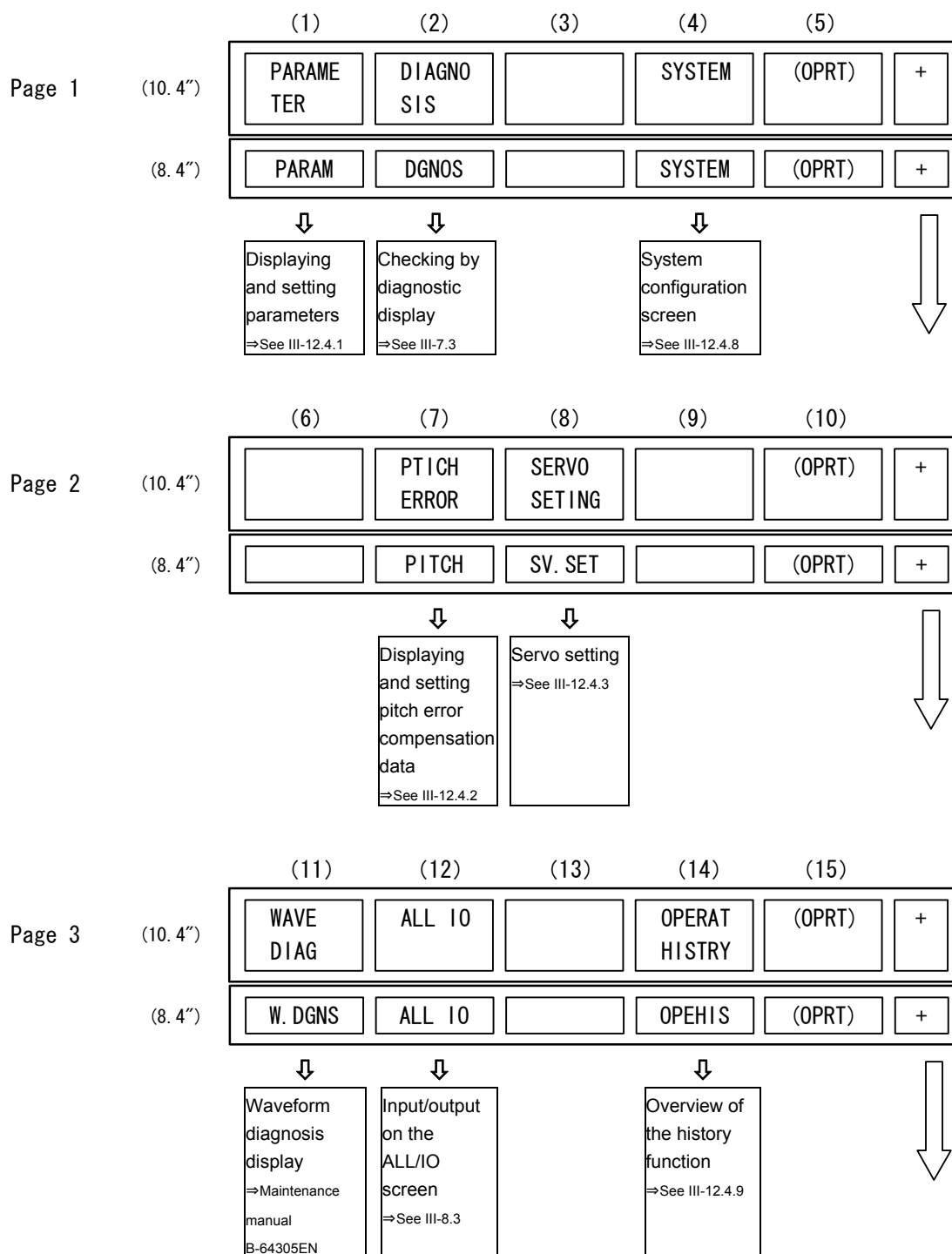
Page 4

(10.4")

(8.4")

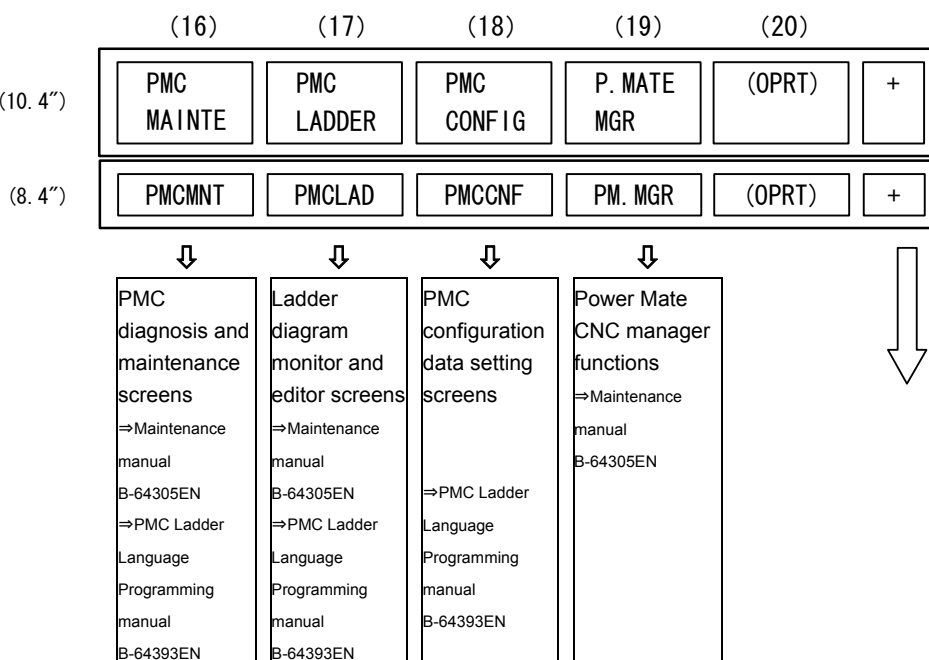


Screen displayed when the function key  is pressed (for 8.4/10.4-inch display unit)



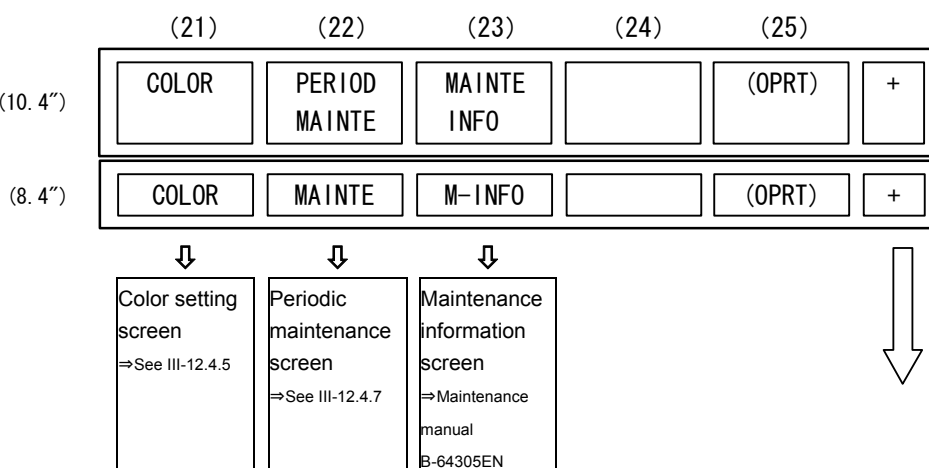
Page 4

(10.4")



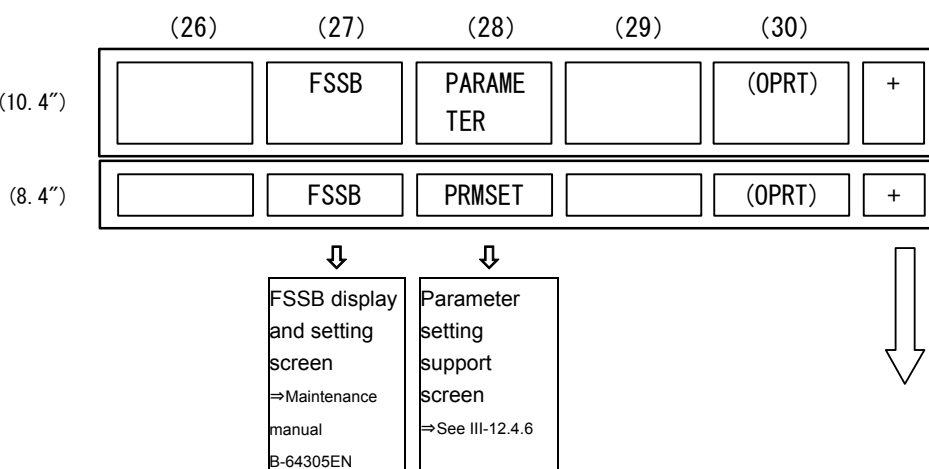
Page 5

(10.4")



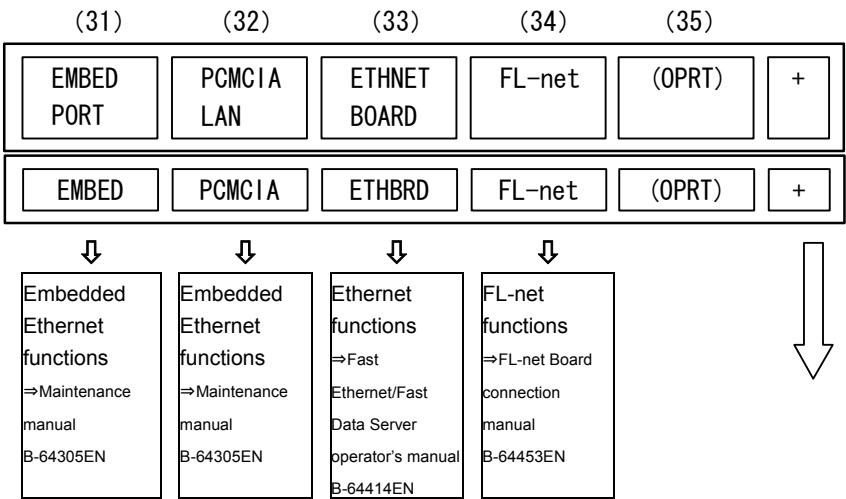
Page 6

(10.4")



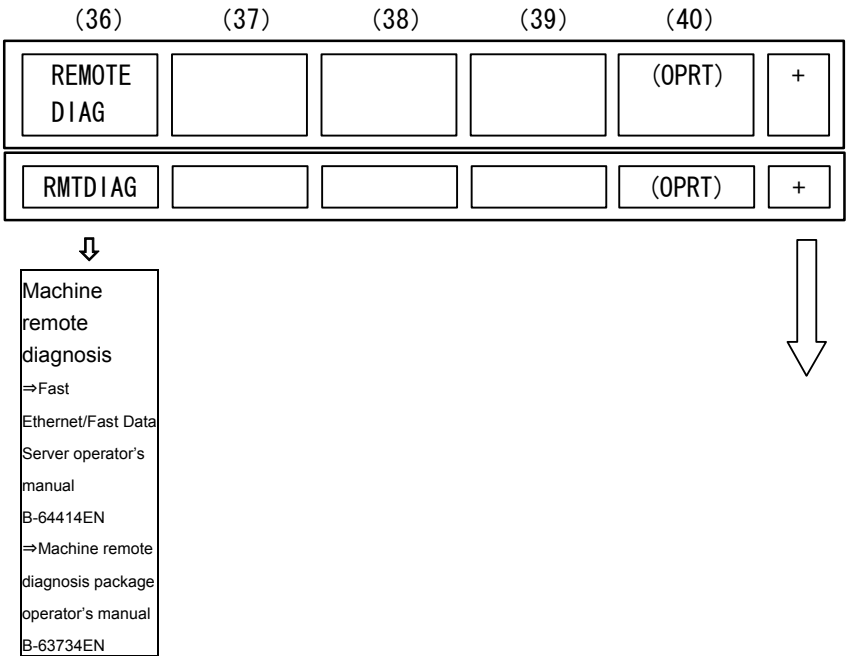
Page 7

(10.4")



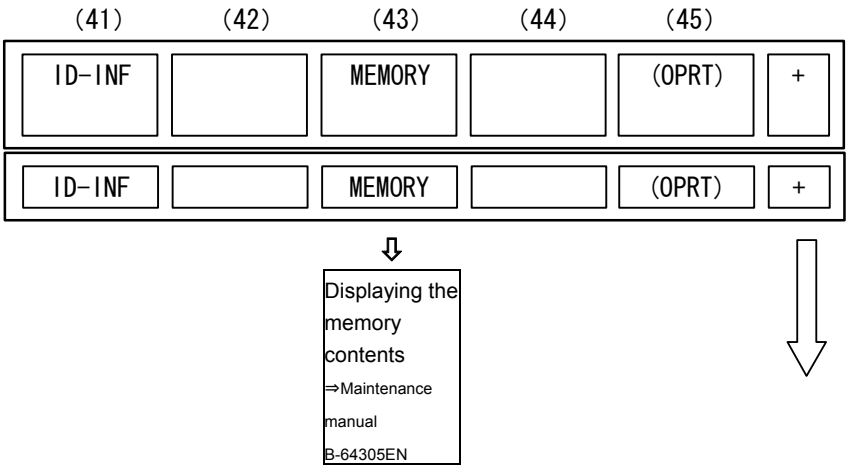
Page 8

(10.4")



Page 9

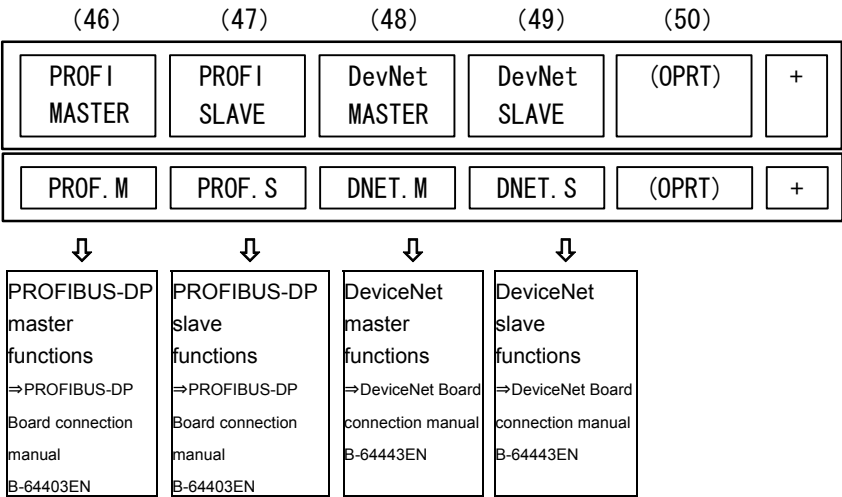
(10.4")



Page 10

(10.4")

(8.4")



12.1 SCREENS DISPLAYED BY FUNCTION KEY

Section 12.1, "SCREENS DISPLAYED BY FUNCTION KEY ", consists of the following subsections:

12.1.1	Position Display in the Workpiece Coordinate System.....	526
12.1.2	Position Display in the Relative Coordinate System.....	527
12.1.3	Overall Position Display.....	529
12.1.4	Workpiece Coordinate System Preset	530
12.1.5	Actual Feedrate Display	531
12.1.6	Display of Run Time and Parts Count.....	533
12.1.7	Operating Monitor Display.....	534

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 (operating monitor display).

Function key  can also be used to display the screen for displaying the distance moved by handle interruption. See Section III-4.6, "MANUAL HANDLE INTERRUPTION" 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 III-6.5.2.3, "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 soft key [ABSOLUTE].
- 3 Press soft key [ABSOLUTE] again. The sixth and subsequent axes are displayed.

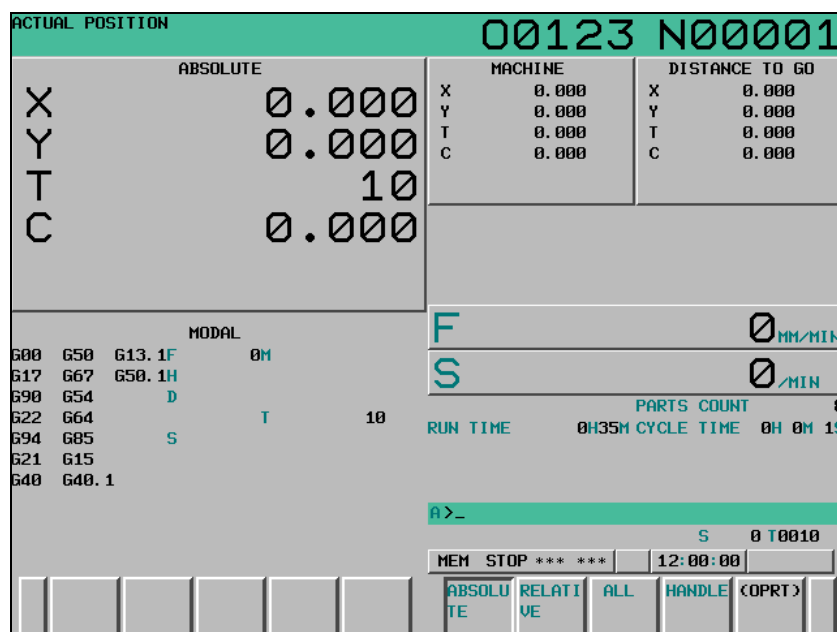


Fig. 12.1.1 (a) Current position (absolute) screen (10.4-inch display unit)

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 III-12.1.4, "Workpiece Coordinate System Preset".

- Display including compensation values

Bit 7 (DAC) of parameter No. 3104 can be used to select whether the displayed values include cutter 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 soft key [RELATIVE].
- 3 Press soft key [RELATIVE] again. The sixth and subsequent axes are displayed.

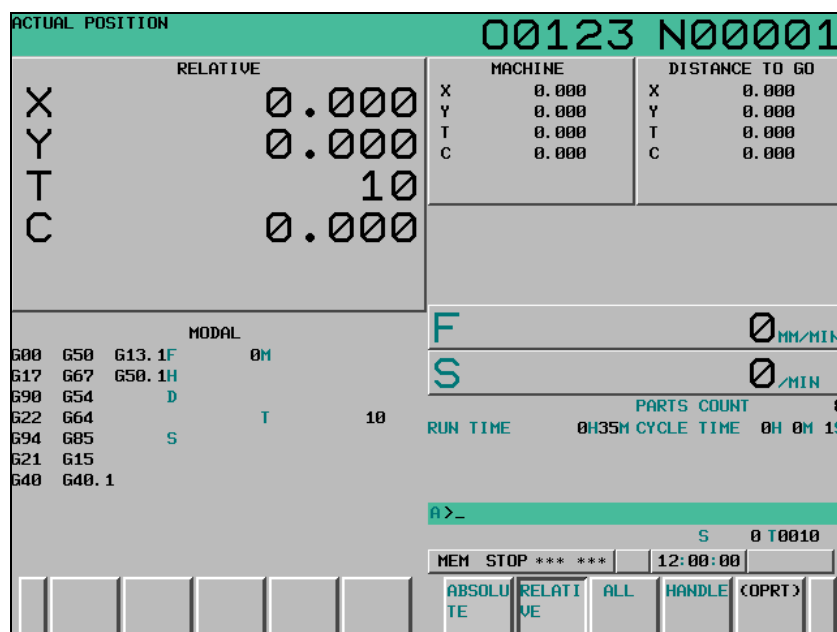


Fig. 12.1.2 (a) Current position (relative) screen (10.4-inch display unit)

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

Bit 5 (DRC) of parameter No. 3104 can be used to select whether the displayed values include cutter compensation.

- Presetting by setting a coordinate system

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.

If parameter bit PPD is set to 0, by setting bit 5 (PWR) of parameter No. 11277 to 1, it is possible to preset the relative coordinates at power-on in the machine coordinate system.

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 Subsection, “Position Display in the Relative Coordinate System” for the procedure.

Procedure for displaying overall position display screen

Procedure

- 1 Press function key .
- 2 Press soft key [ALL].
- 3 Press soft key [ALL] again. The sixth and subsequent axes are displayed.

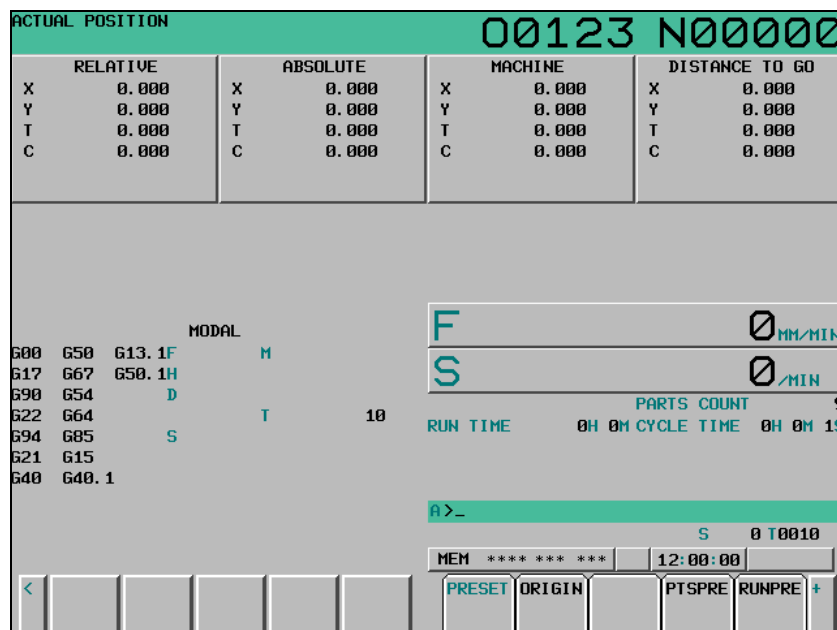


Fig. 12.1.3 (a) Current position (overall) screen (10.4-inch display unit)

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 MEMORY 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 See Subsection, "Position Display in the Relative Coordinate System".

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 (RELATIVE).

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.

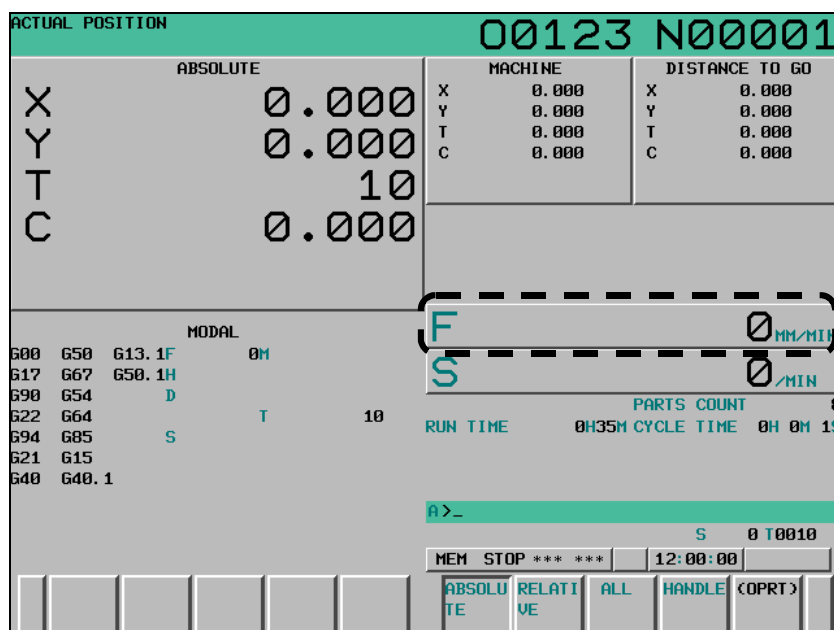


Fig. 12.1.5 (a) Current position (absolute) screen (10.4-inch display unit)

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:

$$\text{Fact} = \sqrt{\sum_{i=1}^n (f_i)^2}$$

n : Number of axes
 f_i : 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)

- 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 also displayed on the program check 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.

Table 12.1.5 (a) Displaying the number of decimal places

Parameter No.3135	Metric input INI(No.0#2)=0	Inch input INI(No.0#2)=1
0	With no decimal point	Two digits to the right of the decimal point
1	One digit to the right of the decimal point	Three digits to the right of the decimal point
2	Two digits to the right of the decimal point	Four digits to the right of the decimal point
3	Three digits to the right of the decimal point	Five digits to the right of the decimal point

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  to display a current position display screen. "PARTS COUNT", "RUN TIME", and "CYCLE TIME" are displayed in the area enclosed by .

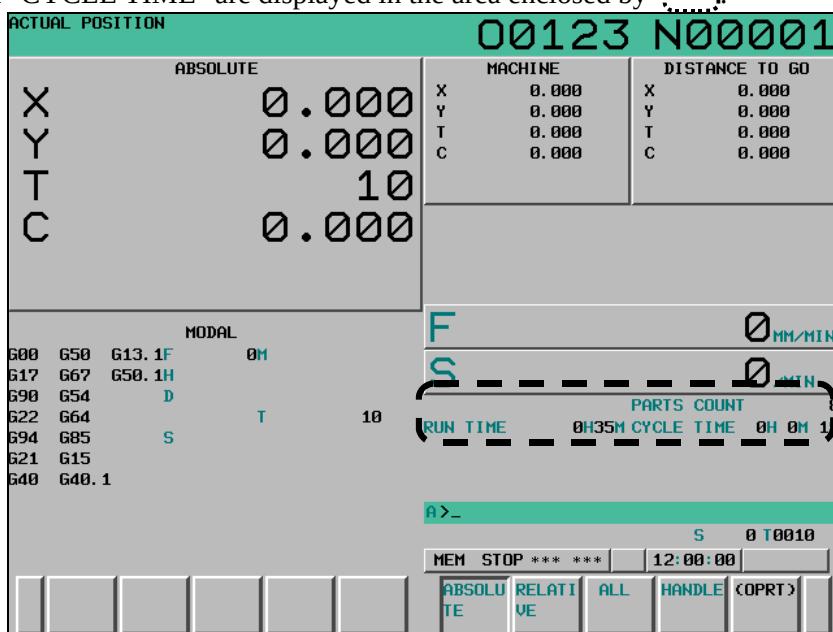


Fig. 12.1.6 (a) Current position (absolute) screen (10.4-inch display unit)

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-12.3.4, "Displaying and Setting Run Time, Parts Count, and Time".

- 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.

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].

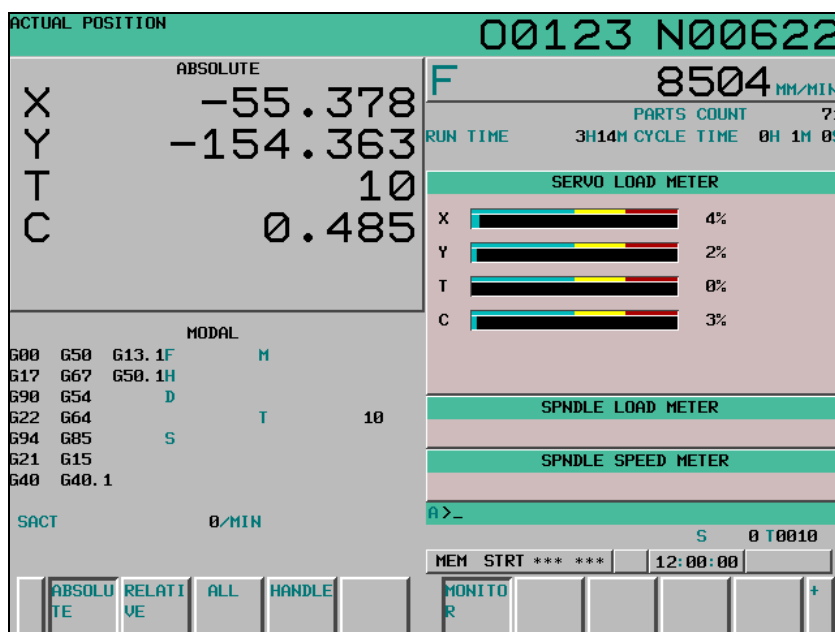


Fig. 12.1.7 (a) Operating monitor (10.4-inch display unit)

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. One screen displays load meters for up to five axes at a time. By pressing the soft key [MONITOR], load meters for the 6th axis and up are displayed.

- Unit of graph

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%).

- Load meter

The reading on the load meter depends on servo parameter No. 2086.

- Color of graph

If the value of a load meter exceeds 100%, the bar graph turns purple.

12.2 SCREENS DISPLAYED BY FUNCTION KEY

Section 12.2, "SCREENS DISPLAYED BY FUNCTION KEY ", consists of the following subsections:

12.2.1	Program Contents Display	535
12.2.2	Editing a Program	536
12.2.3	Program Screen for MDI Operation	538
12.2.4	Program List Screen	538
12.2.5	Next Block Display Screen	539
12.2.6	Program Check Screen	540
12.2.7	Current Block Display Screen (Only for the 8.4-Inch Display Unit)	541
12.2.8	Graphical Conversational Programming Screen	541
12.2.9	Background Editing	543

This section describes the screens displayed by pressing function key . The screens include a program editing screen, program list display screen, and screens for displaying the command states of the program currently being executed.

1. Program screen
2. Program list screen
3. Next block display screen
4. Program check screen

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 MEMORY 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.

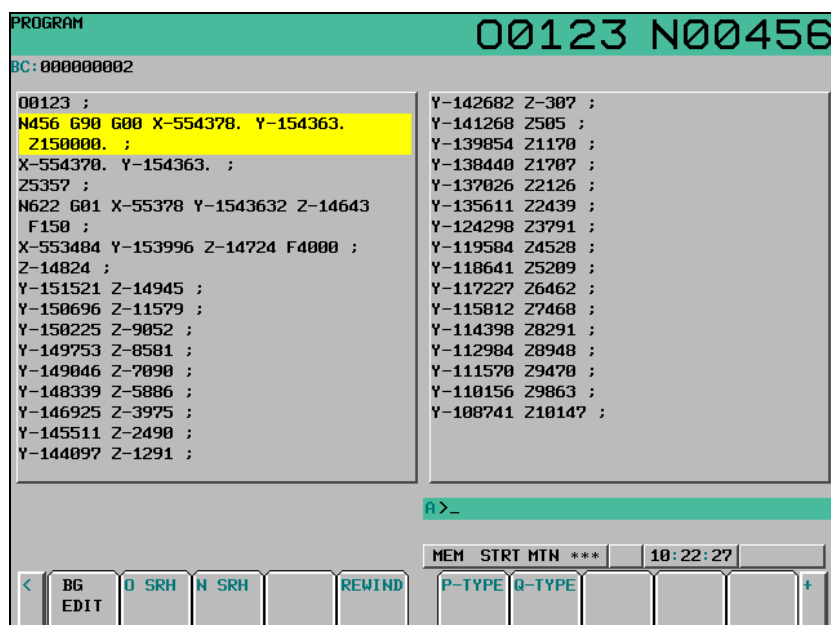


Fig. 12.2.1 (a) Screen for displaying the program being executed (full screen display) (10.4-inch)

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.

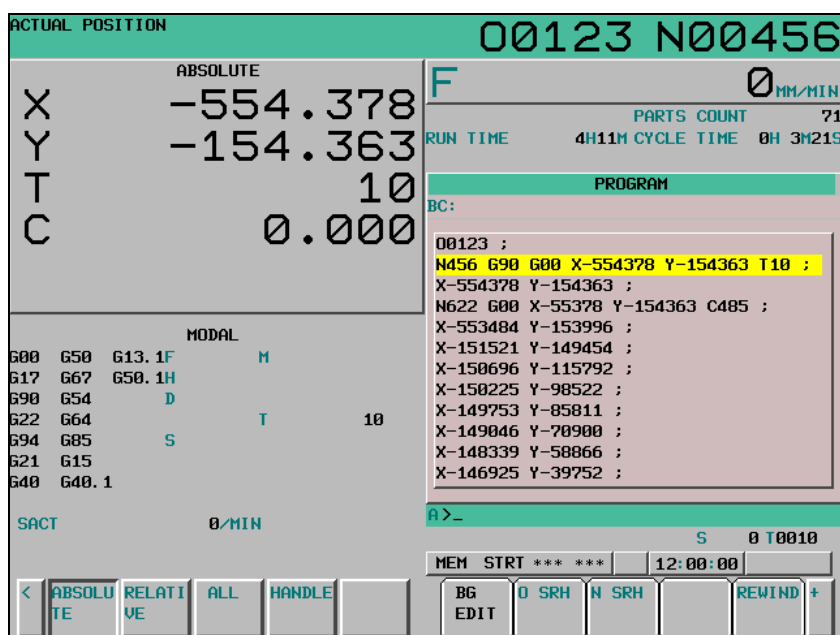


Fig.12.2.1 (b) Screen for displaying the program being executed (small screen display) (10.4-inch)

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  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.

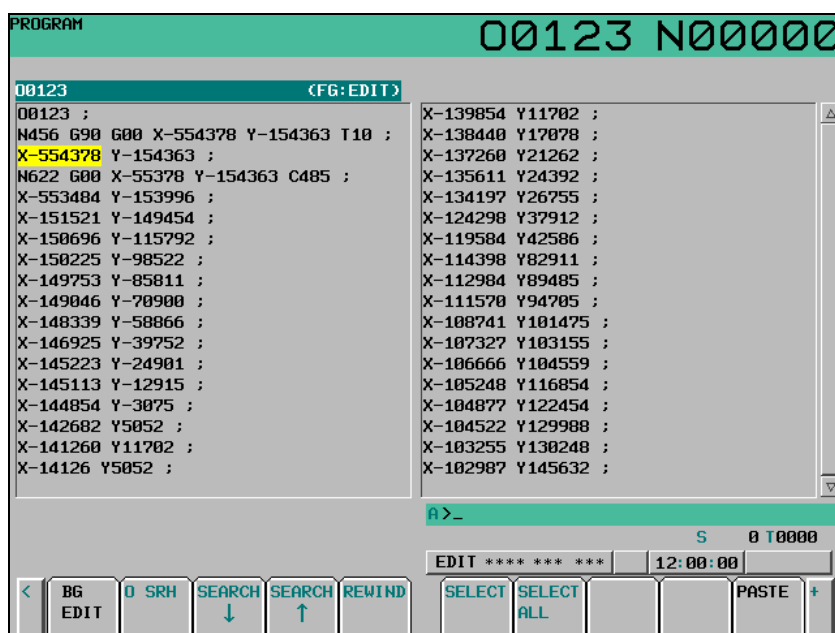


Fig. 12.2.2 (a) Program editing screen (quarter screen display) (10.4-inch)

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.

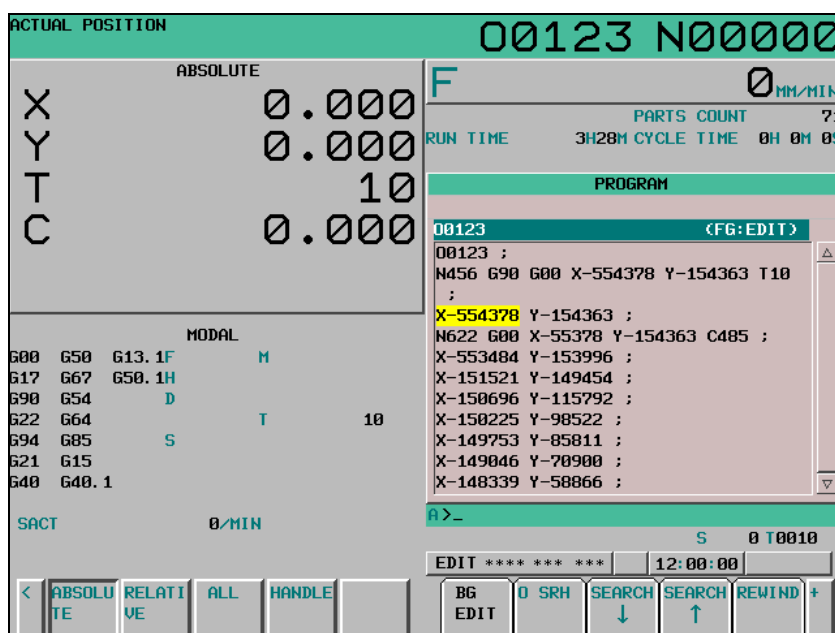


Fig. 12.2.2 (b) Program editing screen (small screen display) (10.4-inch)

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.

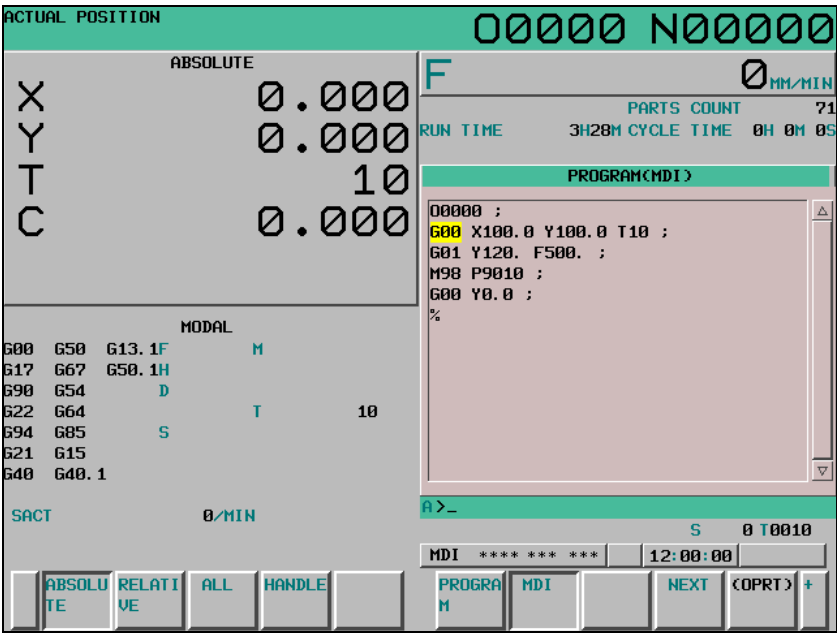


Fig. 12.2.3 (a) MDI operation program screen (10.4-inch)

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 folder screen

Procedure

- 1 Press function key .
- 2 Press continuous menu key .
- 3 Press chapter selection soft key [LIST].

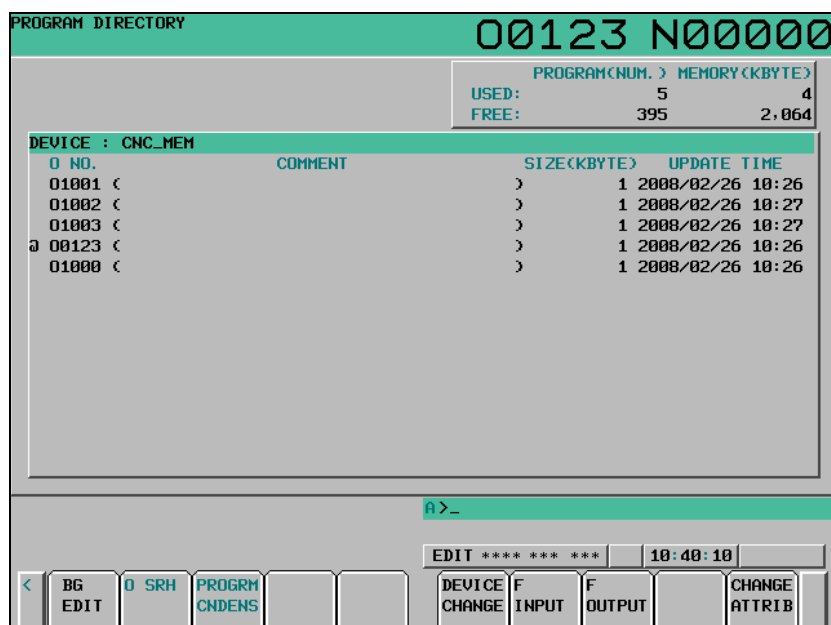


Fig. 12.2.4 (a) Program list screen (10.4-inch)

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.

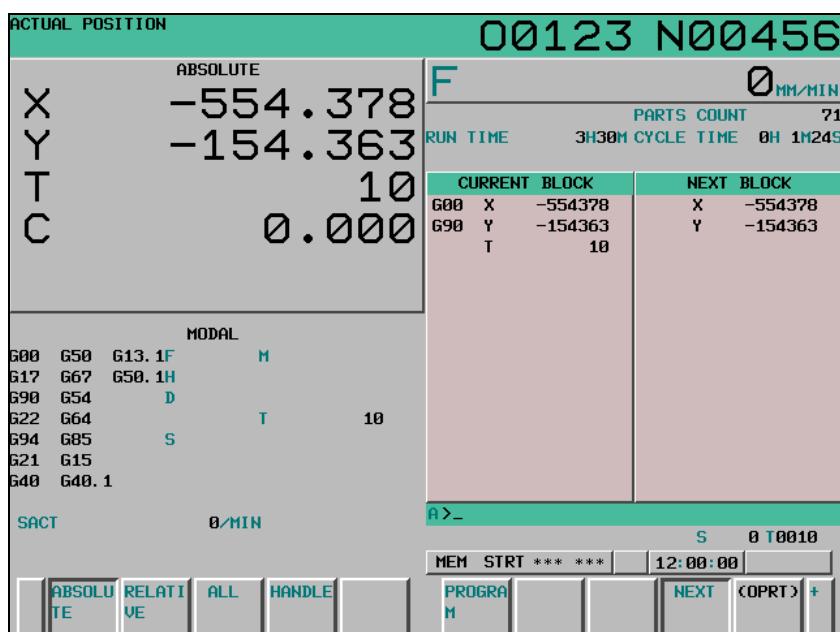


Fig. 12.2.5 (a) Next block display screen (10.4-inch)

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.

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.

PROGRAM (CHECK)				O0123 N00456	
O0123 ;					
N456 G90 G00 X-554378 Y-154363 T10 ;					
X-554378 Y-154363 ;					
N622 G00 X-55378 Y-154363 C485 ;					
ABSOLUTE		DISTANCE		G00	G94 G67
X	-554.378	0.000	G17	G21	G54
Y	-154.363	0.000	G90	G40	G64
T	0.000	0.000	G22	G50	G85
C	0.000	0.000			
			H	M	
T	10	D			
F	S				
ACT. F	0MM/MIN SACT		0/MIN		
A) _					
				S	0 T0010
MEM	STRT	***	***	12:00:00	
(	ABS	REL			(OPRT)

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.

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.

PROGRAM			O0123 N00456		
CURRENT BLOCK			MODAL		
G00	X	-554378	G00	G15	F
G90	Y	-154363	G17	G40. 1	M
	T	10	G90	G13. 1	
			G22	G50. 1	
			G94		H D
			G21		
			G40		T 10
			G50		
			G67		
			G54		S
			G64		
			G85		
A) _			S 0 T0010		
MEM	STRT	*** **	12:00:00		
PROGRAM	CHECK	CURRENT	NEXT	(OPRT)	+

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 Section III-9.3, "CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION".

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].

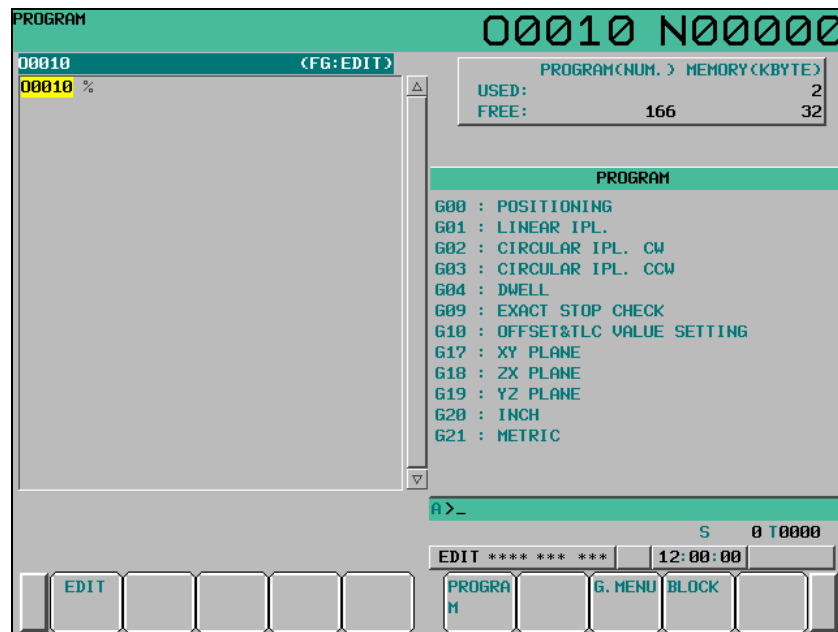


Fig. 12.2.8 (a) Graphical conversational programming screen (G code menu screen) (10.4-inch)

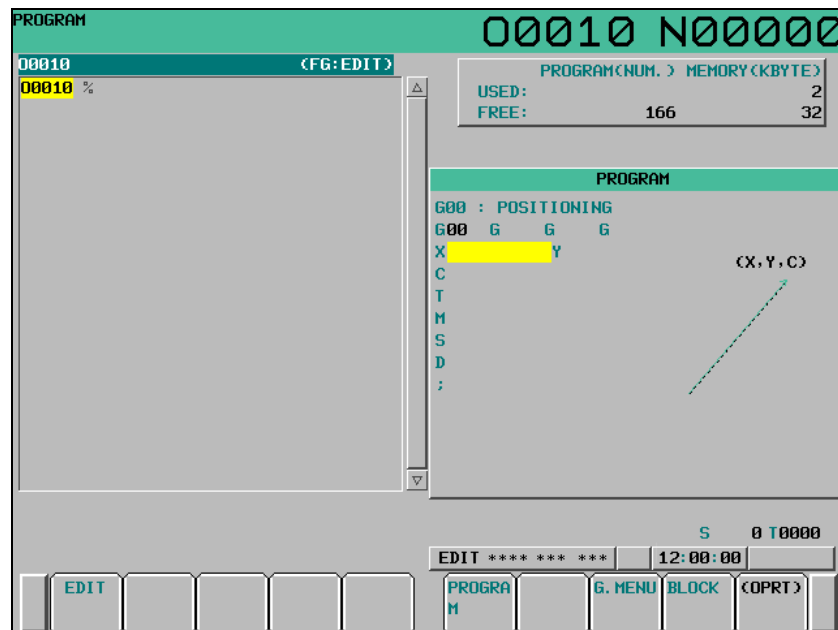


Fig. 12.2.8 (b) Graphical conversational programming screen (G code detail screen) (10.4-inch)

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.

- 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.

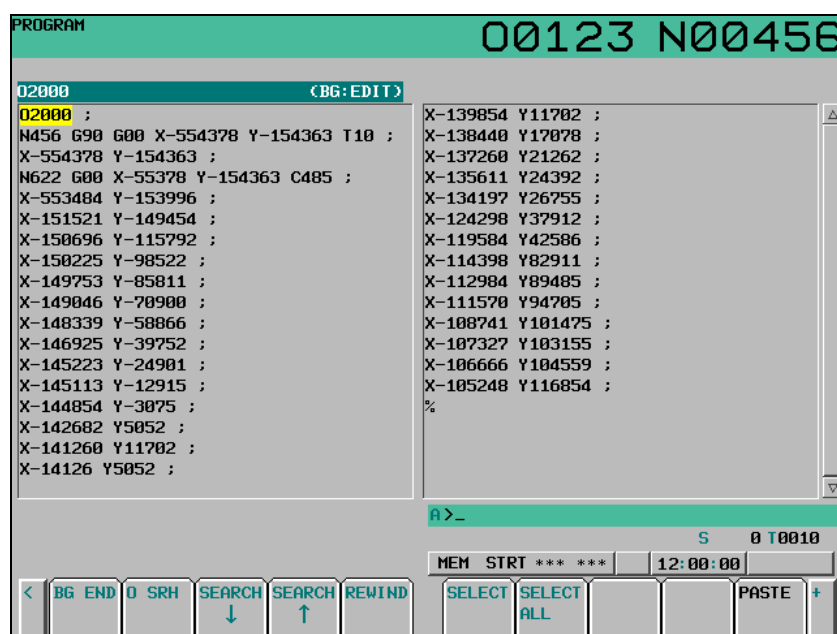


Fig. 12.2.9 (a) Screen being edited in the background (editing mode) (10.4inch)

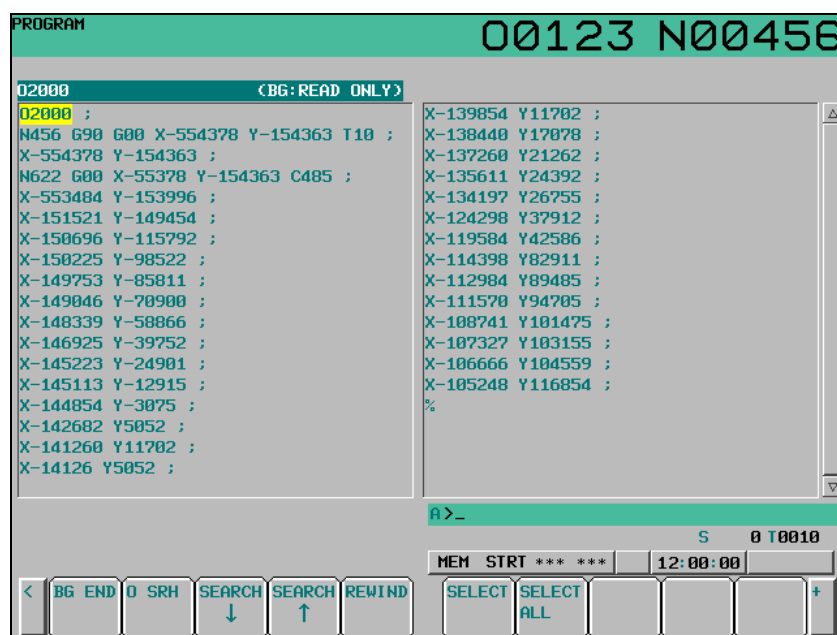


Fig. 12.2.9 (b) Screen being edited in the background (reference mode) (10.4inch)

- 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.

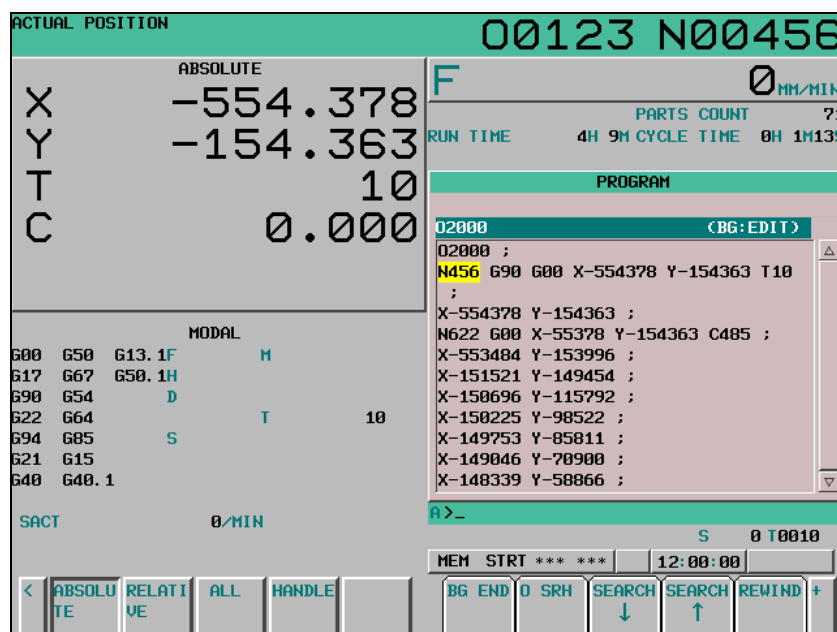


Fig. 12.2.9 (c) Program screen (small screen) (10.4-inch)

Starting of background editing

Background editing can be started from the following screens.

PROGRAM
PROGRAM LIST
MDI PROGRAM
PROGRAM CHECK
CURRENT BLOCK
NEXT BLOCK

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 .
- 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 .
- 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:

12.3.1	Setting and Displaying the Tool Compensation Value	547
12.3.2	Displaying and Entering Setting Data.....	549
12.3.3	Sequence Number Comparison and Stop.....	551
12.3.4	Displaying and Setting Run Time, Parts Count, and Time.....	553
12.3.5	Displaying and Setting the Workpiece Origin Offset Value.....	555
12.3.6	Direct Input of Workpiece Origin Offset Value Measured.....	555
12.3.7	Displaying and Setting Custom Macro Common Variables	557
12.3.8	Displaying and Setting the Software Operator's Panel.....	558
12.3.9	Displaying and Switching the Display Language	561
12.3.10	Protection of Data at Eight Levels	562
12.3.11	Displaying and Setting Items on the Tool Registration Screens.....	567
12.3.12	Displaying and Setting Items on the Tool Registration Screen for Multiple Tools.....	574
12.3.13	Displaying and Setting Items on the Safety Zone Setting Screen.....	578

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. Settings
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. Tool data
11. Safety zone

The software operator's panel, and display language switching depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

12.3.1 Setting and Displaying the Tool Compensation Value

Tool offset values are specified by D codes in a program. Compensation values corresponding to D codes are displayed or set on the screen.

Procedure for setting and displaying the tool compensation value

Procedure

- 1 Press function key .
- 2 Press chapter selection soft key [OFFSET] or press function key  several times until the tool compensation screen is displayed.

OFFSET			00123 N00000		
NO.	DATA	NO.	DATA	NO.	DATA
001	0.000	018	0.000		
002	0.000	019	0.000		
003	0.000	020	0.000		
004	0.000	021	0.000		
005	0.000	022	0.000		
006	0.000	023	0.000		
007	0.000	024	0.000		
008	0.000	025	0.000		
009	0.000	026	0.000		
010	0.000	027	0.000		
011	0.000	028	0.000		
012	0.000	029	0.000		
013	0.000	030	0.000		
014	0.000	031	0.000		
015	0.000	032	0.000		
016	0.000				
017	0.000				

RELATIVE	
X	0.000
Y	0.000
T	0.000
C	0.000

ABSOLUTE	
X	0.000
Y	0.000
T	0.000
C	0.000

MACHINE	
X	0.000
Y	0.000
T	0.000
C	0.000

A>_	
S 0 T0010	
MEM STOP *** ** 12:00:00	
< NO.SRH	INP.C. +INPUT INPUT
ERASE	F INPUT F OUTPUT

Fig. 12.3.1 (a) Tool compensation memory A (10.4-inch display unit)

- 3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].
- 4 To set a compensation value, enter a value and press soft key [INPUT].
To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [+INPUT]. Or, enter a new value and press soft key [INPUT].

Explanation

- Decimal point input

A decimal point can be used when entering a compensation value.

- Other setting method

An external input/output device can be used to input or output a tool offset value. See Chapter III-8 in OPERATOR'S MANUAL.

- Tool compensation memory

The compensation value for tool compensation (for D code) is to be set in the compensation memory.

- Disabling entry of compensation values

The entry of compensation values may be disabled by setting bit 0 (WOF) of parameter No.3290.

And then, the input of tool compensation values from the MDI can be inhibited for a specified range of offset numbers. The first offset number for which the input of a value is inhibited is set in parameter No. 3294. The number of offset numbers, starting from the specified first number, for which the input of a value is inhibited is set in parameter No. 3295.

Consecutive input values are set as follows:

- 1) When compensation values are input consecutively from offset numbers for which the input of values is enabled to offset numbers for which the input of values is inhibited, a warning is issued, but the compensation values in the range of the offset numbers for which the input of values is enabled are set.
- 2) When compensation values are input consecutively from offset numbers for which the input of values is inhibited to offset numbers for which the input of values is enabled, a warning is issued and the compensation values are not set.

12.3.2 Displaying and Entering Setting Data

Data such as the TV check flag and output 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-12.3.3 for the sequence number comparison and stop function. This subsection describes how to set data.

Procedure for setting the setting data

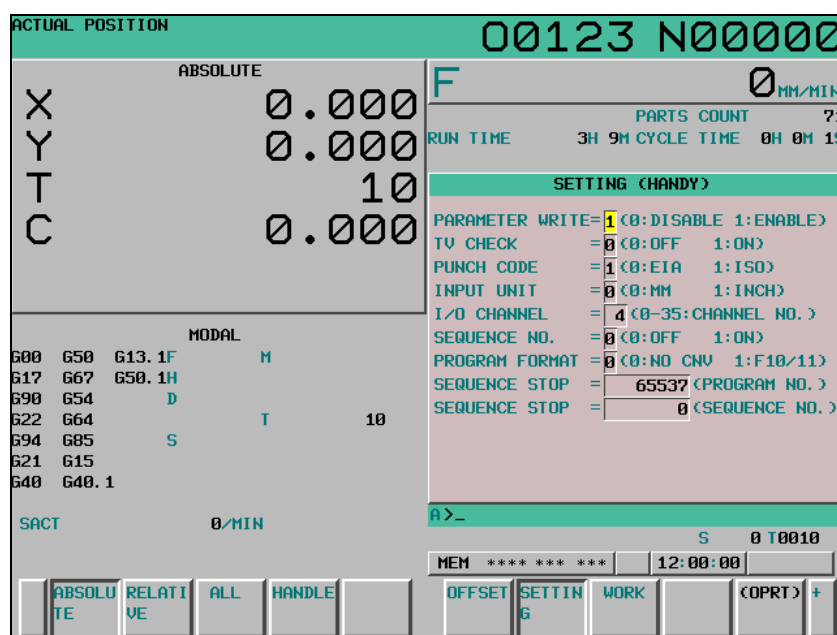
Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press soft key [SETTING] to display the setting data screen.

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.



ACTUAL POSITION 00123 N00000

ABSOLUTE

X	0.000
Y	0.000
T	10
C	0.000

MODAL

G00	G50	G13.1F	M
G17	G67	G50.1H	
G90	G54	D	
G22	G64		T 10
G94	G85	S	
G21	G15		
G40	G40.1		

SETTING (HANDY)

PARAMETER WRITE= 1 (0:DISABLE 1:ENABLE)

TV CHECK = 0 (0:OFF 1:ON)

PUNCH CODE = 1 (0:EIA 1:ISO)

INPUT UNIT = 0 (0:MM 1:INCH)

I/O CHANNEL = 4 (0-35:CHANNEL NO.)

SEQUENCE NO. = 0 (0:OFF 1:ON)

PROGRAM FORMAT = 0 (0:NO CNV 1:F10/11)

SEQUENCE STOP = 65537 (PROGRAM NO.)

SEQUENCE STOP = 0 (SEQUENCE NO.)

STATUS BAR

MEM **** * 12:00:00 S 0 T0010

ABSOLUTE RELATIVE ALL HANDLE OFFSET SETTING WORK COPY +

Fig. 12.3.2 (a) SETTING (HANDY) screen (10.4-inch display unit)

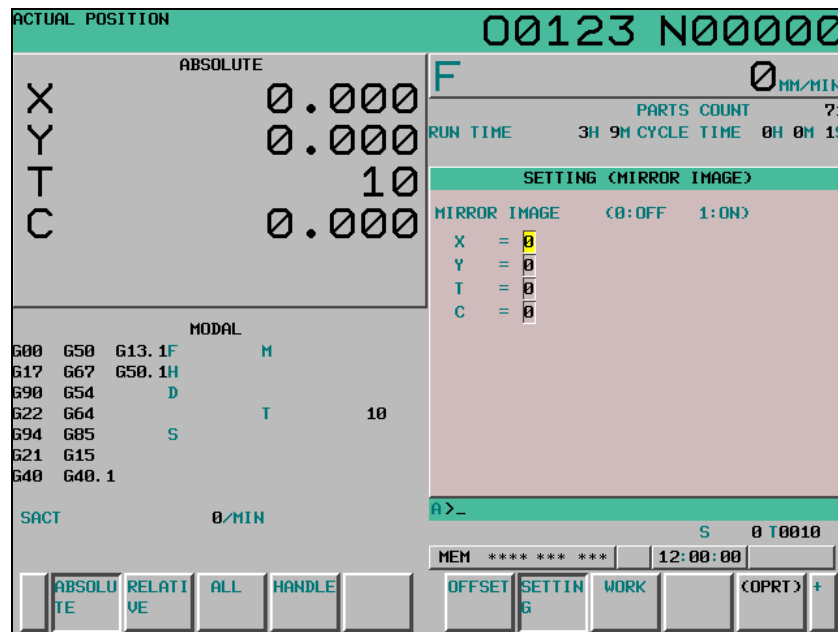


Fig. 12.3.2 (b) SETTING (MIRROR IMAGE) screen (10.4-inch display unit)

- 4 Move the cursor to the item to be changed by pressing cursor keys     .
- 5 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

- OUTPUT 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 (RS232-C port 1)

1 : Channel 1 (RS232-C port 1)

2 : Channel 2 (RS232-C port 2)

4 : Memory card interface

5 : Data Server interface

9 : Embedded ethernet interface

- 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

This data is not available.

- 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-12.3.4, “Displaying and Setting Run Time, Parts Count, and Time” for this screen.

12.3.3 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. 12.3.3 (a) SETTING (HANDY) screen (10.4-inch display unit)

- 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.

- 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.

12.3.4 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.

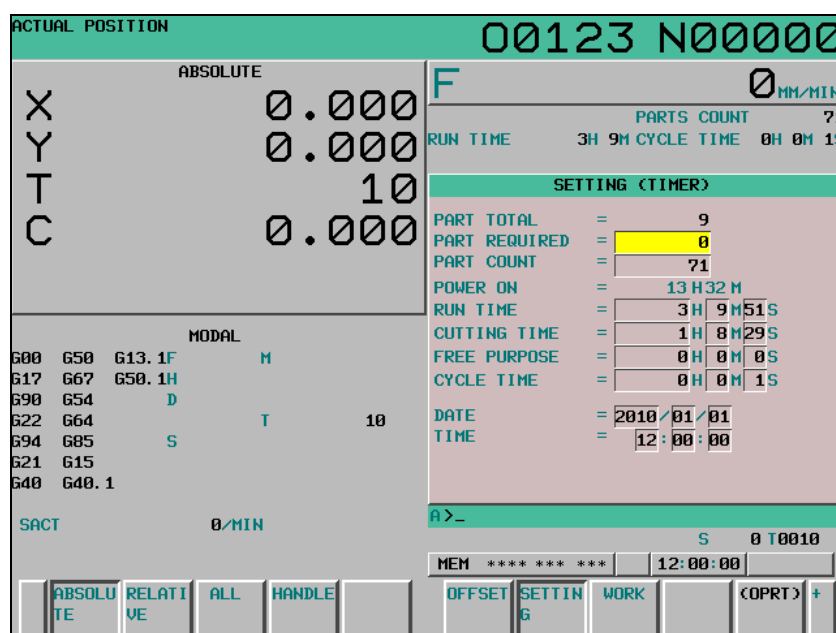


Fig. 12.3.4 (a) SETTING (TIMER) screen (10.4-inch display unit)

- 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 12.3.4 (a)

Item	Maximum value	Item	Maximum value
Year	2096	Hour	23
Month	12	Minute	59
Day	31	Second	59

12.3.5 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.

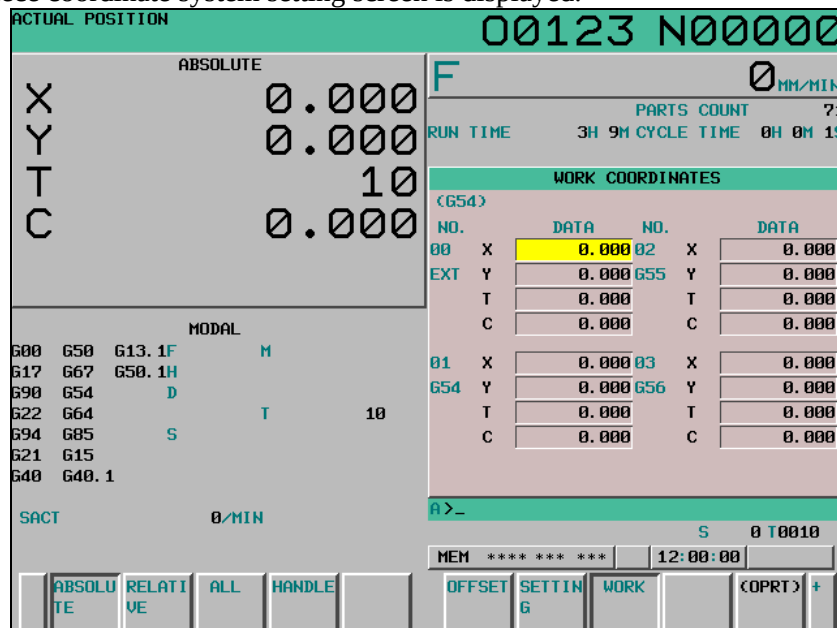


Fig. 12.3.5 (a) WORK COORDINATES screen (10.4-inch display unit)

- 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 and 6 to change other offset values.
- 9 Turn on the data protection key to disable writing.

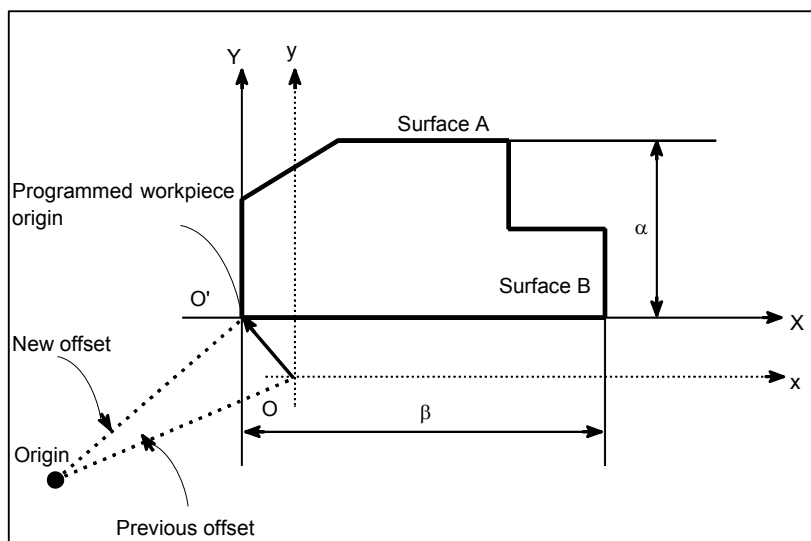
12.3.6 Direct Input of Workpiece Origin Offset Value Measured

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 When the workpiece is shaped as shown above, position the reference tool manually until it touches surface A of the workpiece.
- 2 Retract the tool without changing the Y coordinate.
- 3 Measure distance α 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].

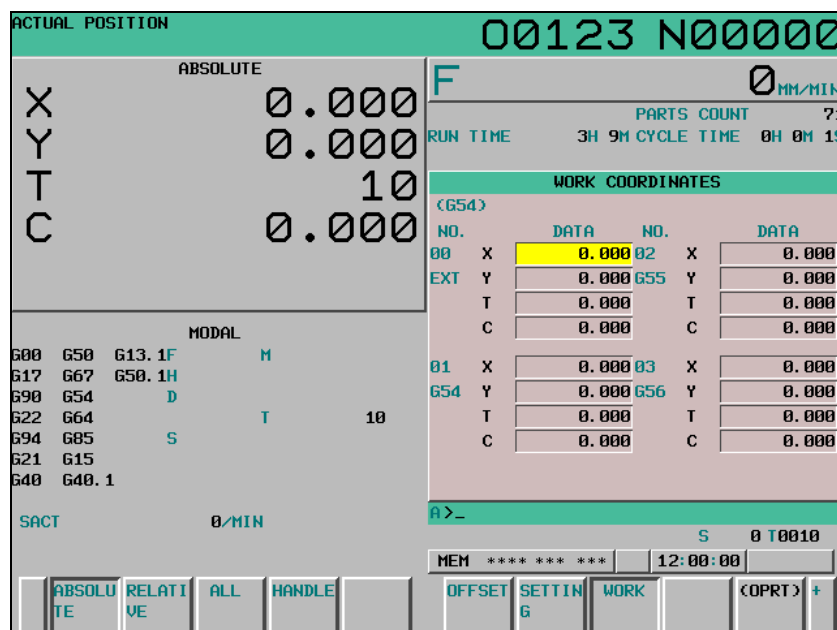


Fig. 12.3.6 (a) WORK COORDINATES screen (10.4-inch display unit)

- 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 (Y-axis in this example).

- 8 Enter the measured value (α) then press soft key [MEASUR].
- 9 Move the reference tool manually until it touches surface B of the workpiece.
- 10 Retract the tool without changing the X coordinate.
- 11 Enter the measured value of distance (β) to surface B as an X value as with 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.

12.3.7 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #199, and #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.

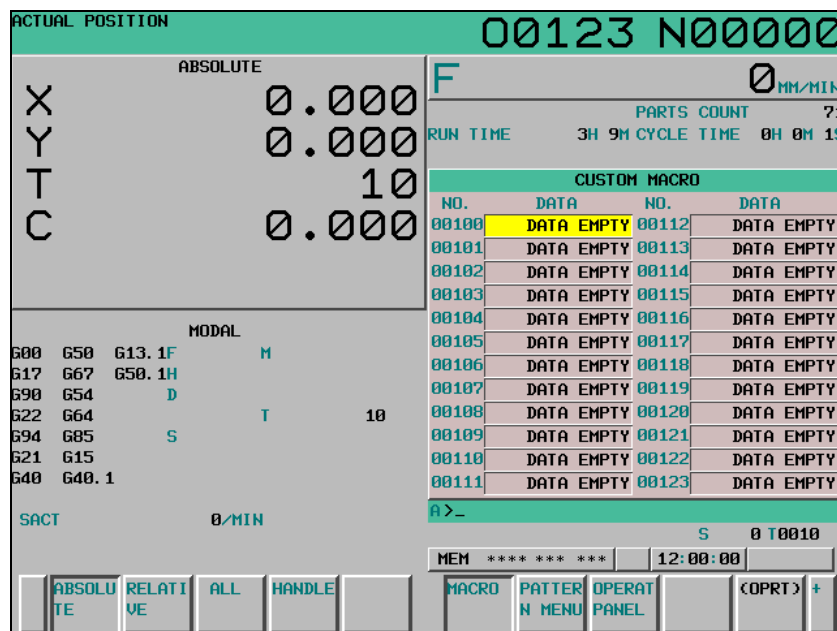


Fig. 12.3.7 (a) CUSTOM MACRO screen (10.4-inch display unit)

- 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]. The value field for the variable becomes blank.

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 (F16) of parameter No. 6008 set to 0):

Variable value range	Variable value indication
$0 < \text{Variable value} < +0.000000000001$	+Underflow
$0 > \text{Variable value} > -0.000000000001$	-Underflow
Variable value > 999999999999	+Overflow
Variable value < -999999999999	-Overflow

When the significant number of digits is 8 (with bit 0 (F16) of parameter No. 6008 set to 1):

Variable value range	Variable value indication
$0 < \text{Variable value} < +0.0000001$	+Underflow
$0 > \text{Variable value} > -0.0000001$	-Underflow
Variable value > 99999999	+Overflow
Variable value < -99999999	-Overflow

12.3.8 Displaying and Setting the Software Operator's Panel

Operations on the MDI unit 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 unit, 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 [OPR].
- 3 The screen consists of several pages. Press page key  or  until the desired screen is displayed.

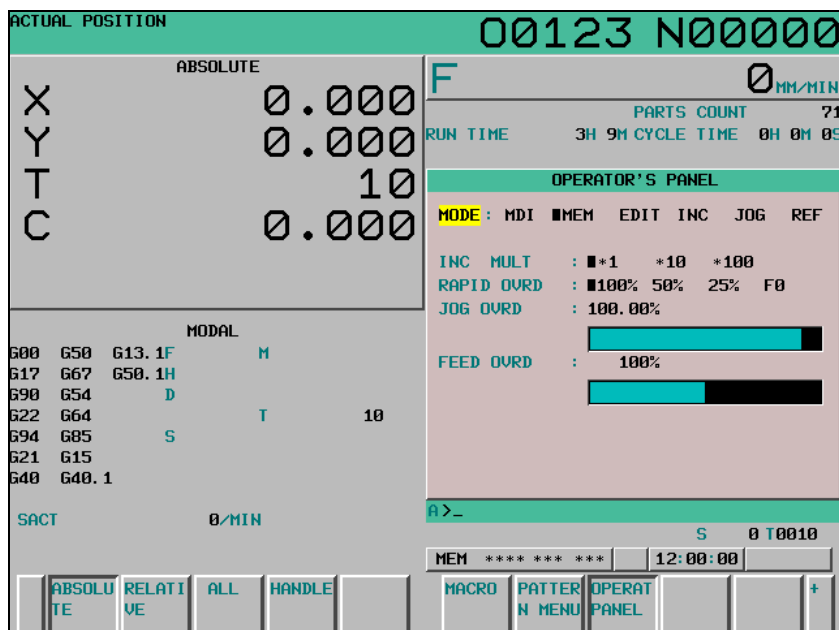


Fig. 12.3.8 (a) Page 1 of Software Operator's Panel screen (without the manual handle feed function)
(10.4-inch display unit)

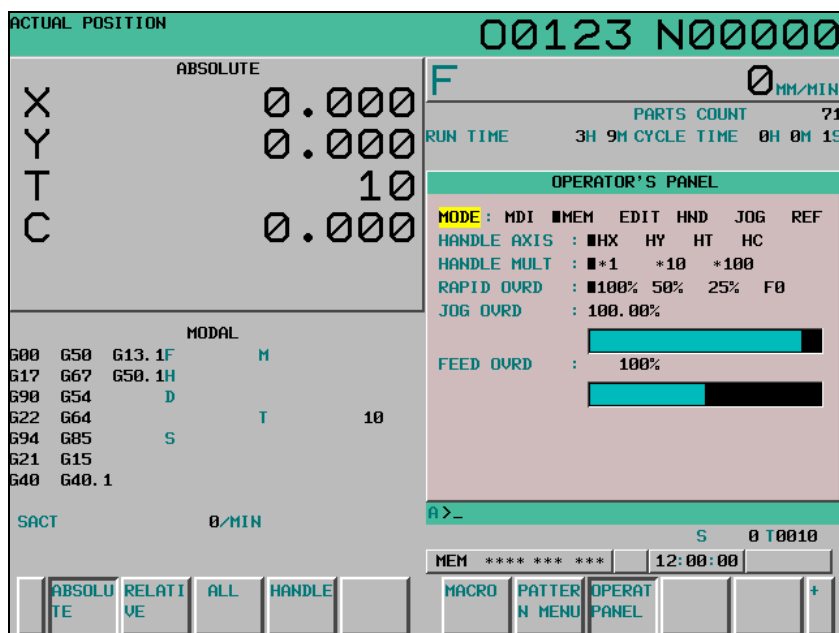


Fig. 12.3.8 (b) Page 1 of Software Operator's Panel screen (with the manual handle feed function)
(10.4-inch display unit)

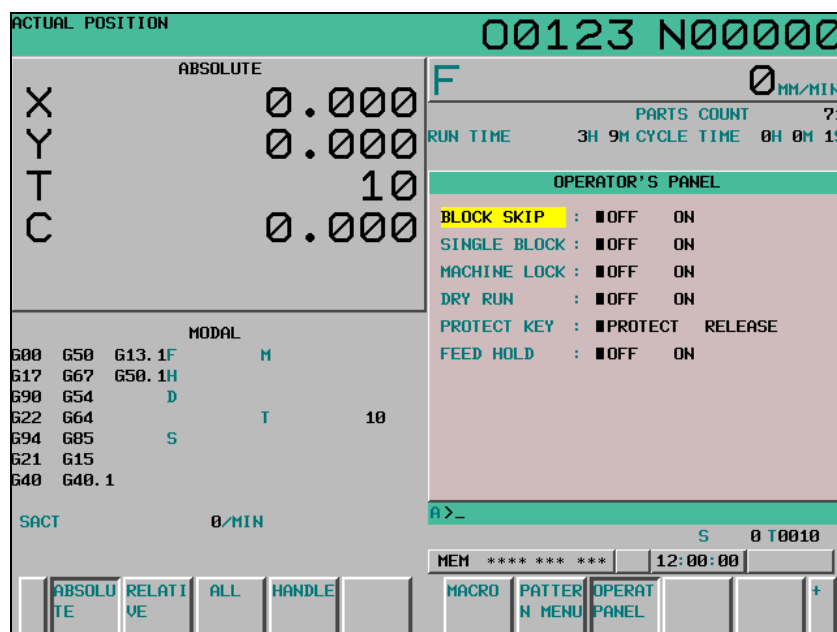


Fig. 12.3.8 (c) Page 2 of Software Operator's Panel screen (10.4-inch display unit)

- 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.

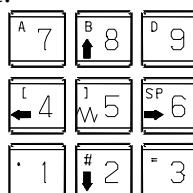


Fig. 12.3.8 (d) MDI arrow keys

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

The feed axis and direction corresponding to the arrow keys can be set with parameters Nos. 7210 to 7217.

- 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.

12.3.9 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 .
- 2 Press the continuous menu key  several times.
- 3 Press soft key [LANGUAGE] to display the language screen.

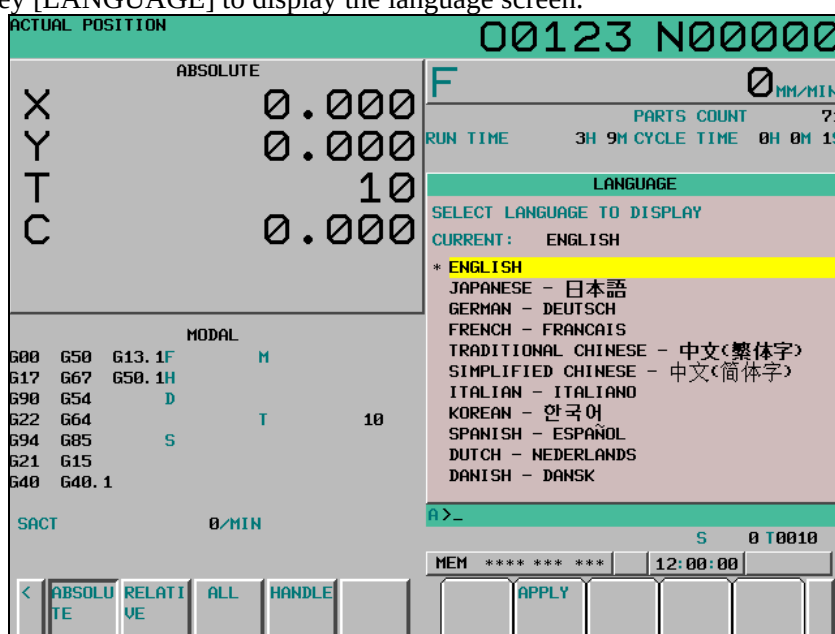


Fig. 12.3.9 (a) LANGUAGE screen (10.4-inch display unit)

- 4 Press page key  or , 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
20. Rumanian

Among the languages listed above, English and other usable languages are displayed on the screen as a list of switchable languages.

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.

12.3.10 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.

12.3.10.1 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.

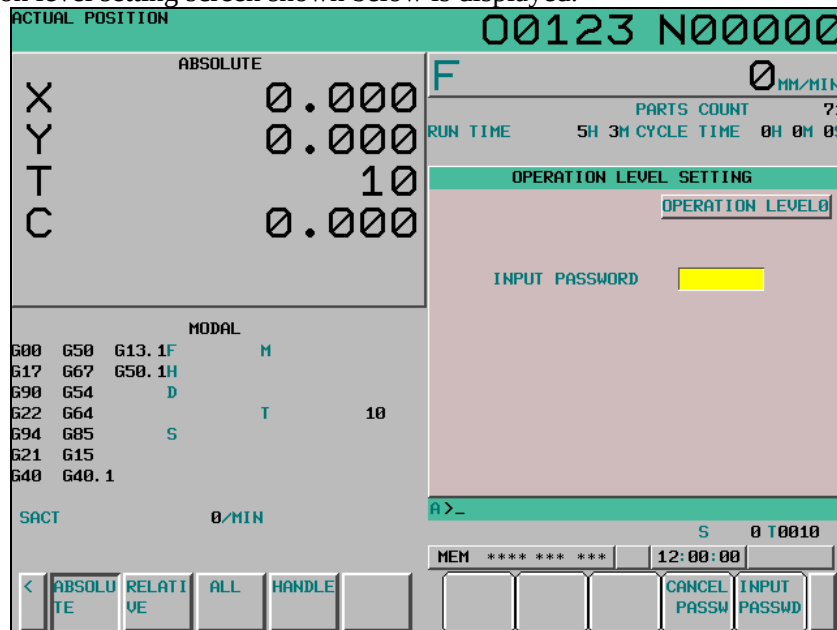


Fig. 12.3.10.1 (a) Operation level setting screen (10.4-inch display unit)

- 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 12.3.10.1 (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.

12.3.10.2 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.

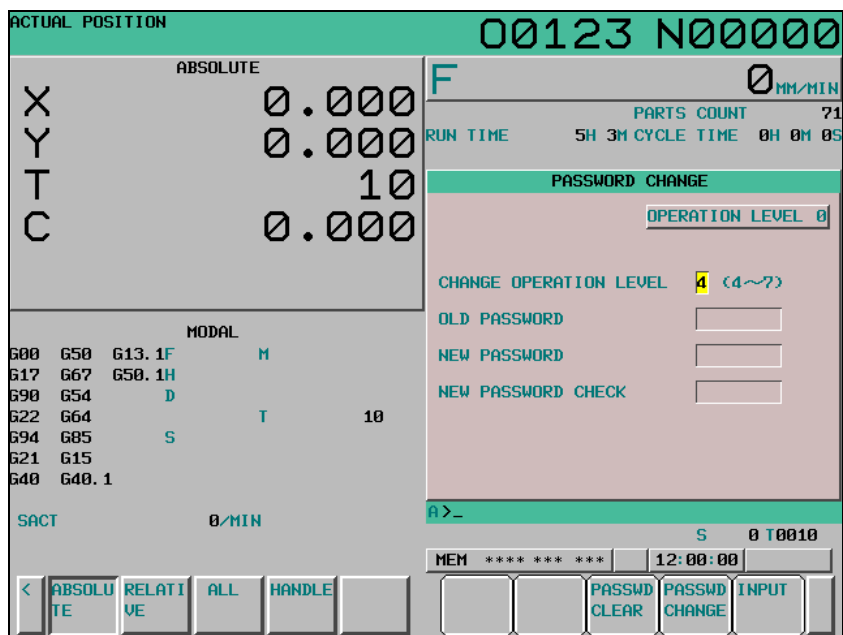


Fig. 12.3.10.2 (a) PASSWORD CHANGE screen (10.4-inch display unit)

- 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 [PASSWDCHANGE] ([CHANGE] for the 8.4-inch display unit).
- 10 To clear the password, press soft key [PASSWDCLEAR] ([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.

12.3.10.3 Protection level setting

The current operation 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.

The screenshot displays the 'PROTECT LEVEL' screen. At the top, 'ACTUAL POSITION' is shown as '00123 N00000'. Below this, 'ABSOLUTE' coordinates for X, Y, T, and C are listed. To the right, 'F' (Feed rate) is set to '0 MM/MIN'. Further right, 'PARTS COUNT' is '71' and 'RUN TIME' is '5H 3M CYCLE TIME 0H 0M 0S'. The 'PROTECT LEVEL' section shows 'OPERATION LEVEL 0'. A table lists 'DATA TYPE' and 'CHANOUTP' for various data items. At the bottom, there are buttons for '< ABSOLUTE RELATIVE ALL HANDLE' and a status bar with 'MEM **** * 12:00:00' and 'INPUT'.

DATA TYPE	CHANOUTP
PARAMETER DATA	4 0
SETTING DATA	0 0
PITCH ERROR DATA	0 0
TIME	0 0
CUSTOM MACRO	0 0
PERIODICAL MAINTENANCE DATA	0 0
WORK ZERO OFFSET	0 0
CHANGE PROGRAMS FOR EACH PAR	0 0
PRESET OF ABSOLUTE AXIS DATA	0 0
TOOL OFFSET DATA	0 0

Fig. 12.3.10.3 (a) Protection level change screen (10.4-inch display unit)

- 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 12.3.10.3 (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>	0	0
Clock data <TIME>	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
Absolute coordinate preset operation <PRESET OF ABSOLUTE AXIS DATA>	0	0

Table 12.3.10.3 (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

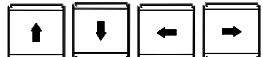
- 1 For some types of data, the output function is not provided.
- 2 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
- 3 The protection level of data cannot be changed to a level higher than the current operation level.
- 4 Settable 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 To change the protection level for each part program, do so on the PROGRAM FOLDER screen rather than on the PROTECT LEVEL screen.
- 7 Part program editing includes program editing for the MDI mode.
- 8 Presetting absolute coordinates causes preset workpiece coordinate system values to be protected.

12.3.11 Displaying and Setting Items on the Tool Registration Screens

Items concerning tools, such as the number of a tool to be used in machining, the position at which the turret is indexed for a tool, and tool position compensation, can be displayed or specified on the tool registration screens. Refer to the manual prepared by the machine tool builder for details, as the builder sets these items first.

12.3.11.1 Displaying and setting items on the initial tool registration screen

Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key  until soft key [TOOL] appears.
- 4 Press the [TOOL] soft key to display the initial tool registration screen.
- 5 Move the cursor to the item to be changed by pressing cursor keys .

- 6 Type the number of the tool management data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].

The screen displays tool management data. On the left, 'ACTUAL POSITION' shows X, Y, T, and C coordinates. Below this, 'MODAL' data lists parameters G00, G17, G90, G22, G94, G21, G40, G50, G67, G54, G64, G85, G15, and G40.1. The right side shows 'F' (feed rate) at 0 MM/MIN, 'PARTS COUNT' at 9, 'RUN TIME' at 0H 1M, and 'CYCLE TIME' at 0H 0M 1S. The 'INITIAL TOOL' section includes 'REGISTERED' (136), 'TOOL INDEX' (136), 'ZERO POINT' (10), 'MOVEMENT' (360.000), and 'PUNCH COUNT' (12345). At the bottom, there are buttons for '< ABSOLUTE', 'RELATIVE', 'ALL', 'HANDLE', 'TOOL', and '<OPRT>'. The status bar at the bottom shows 'MDI *****', '12:00:00', and 'S 0 T0010'.

Fig.12.3.11 (a) Initial Tool Registration Screen

Explanation

- Number of tools to be used

Specify the total number of tools to be used.

This data corresponds to parameter No. 16265.

- Number of tools for which the turret is indexed

Number of tools for which the turret is indexed (parameter No. 16266). When T-axis control is specified (TCL, bit 4 of parameter No. 16260, is set to 1), specify the total number of tools for which the turret is indexed. The setting must not be greater than the number of tools to be used.

This data corresponds to parameter No. 16266.

- Number of a tool to be initially selected

Specify the number of the tool to be selected when reference position return is completed.

This data corresponds to parameter No. No. 16267.

- Number of pulses sent to the turret per rotation

When T-axis is set as a rotation axis and T-axis control is specified (TCL, bit 4 of parameter No. 16260, is set to 1), specify the number of pulses sent to the turret per rotation.

This data corresponds to parameter No.1260.

- Number of punch operations

Specify the number of punch operations for the tool used.

Display and preset the total number of punch operation of all tool.

This data corresponds to parameter No.16269 or No.16270.

NOTE

If the memory protect signal KEY1<Gn046.3> is set to “1”, the number of punch operations is writable although parameter writing is disabled (bit 0 (PWE) of parameter No.8900 is set to 0).

12.3.11.2 Displaying and setting items on the tool number registration screen

Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key  until soft key [TOOL] appears.
- 4 Press the [TOOL] soft key to display the initial tool registration screen the initial tool registration screen.
- 5 Press the menu key . Then press the [TOOL NUMBER] soft key to display the tool number registration screen.
- 6 Move the cursor to the tool number to be changed.
 - Enter the registration number of the tool for which data is to be changed, then press the [NO.SRH] soft key.
 - Move the cursor to the tool number to be changed with the page keys   and cursor keys    .
- 7 To move the cursor to the tool number to be changed, press the cursor keys    .
- 8 Type the data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].

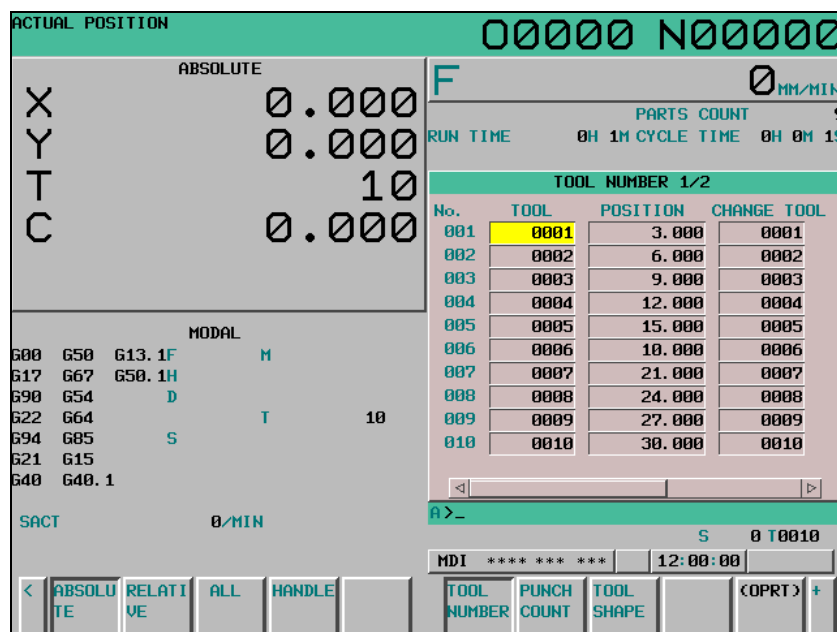


Fig.12.3.11 (b) Tool number registration screen 1/2 (10.4-inch display unit)

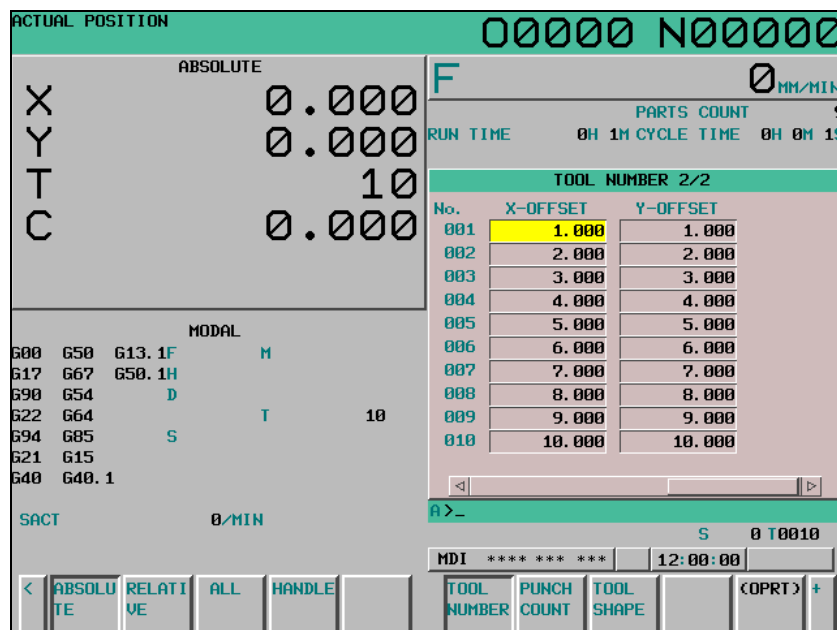


Fig.12.3.11 (c) Tool number registration screen 2/2 (10.4-inch display unit)

Explanation

- Number

Tool registration number. This number is a serial number corresponding to the registered tool.

- Tool number

Specify the numbers of tools to be used.

- Turret position

When T-axis control is specified (TCL, bit 4 of parameter No. 16260, is set to 1), specify the positions at which the turret is indexed for tools.

- Change tool Number

Specify the numbers of tools for exchange.

These numbers are limited to the tool numbers registered on the tool number registration screen

- X-axis offset

When the tool position compensation function is used (TOF, bit 2 of parameter No. 16263 is set to 1), specify X-axis offset for tool positions.

- Y-axis offset

When the tool position compensation function is used (TOF, bit 2 of parameter No. 16263 is set to 1), specify Y-axis offset for tool positions in the machine coordinate system.

12.3.11.3 Displaying and setting items on the screen for the number of press operations

Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key  until soft key [TOOL] appears.

- 4 Press the [TOOL] soft key to display the initial tool registration screen the initial tool registration screen.
- 5 Press the menu key . Then press the [PUNCH COUNT] soft key to display the screen for the number of press operations.
- 6 Move the cursor to the tool number to be changed.
 - Enter the registration number of the tool for which data is to be changed, then press the [NO.SRH] soft key.
 - Move the cursor to the tool number to be changed with the page keys   and cursor keys    .
- 7 To move the cursor to the tool number to be changed, press the cursor keys    .
- 8 Type the data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].

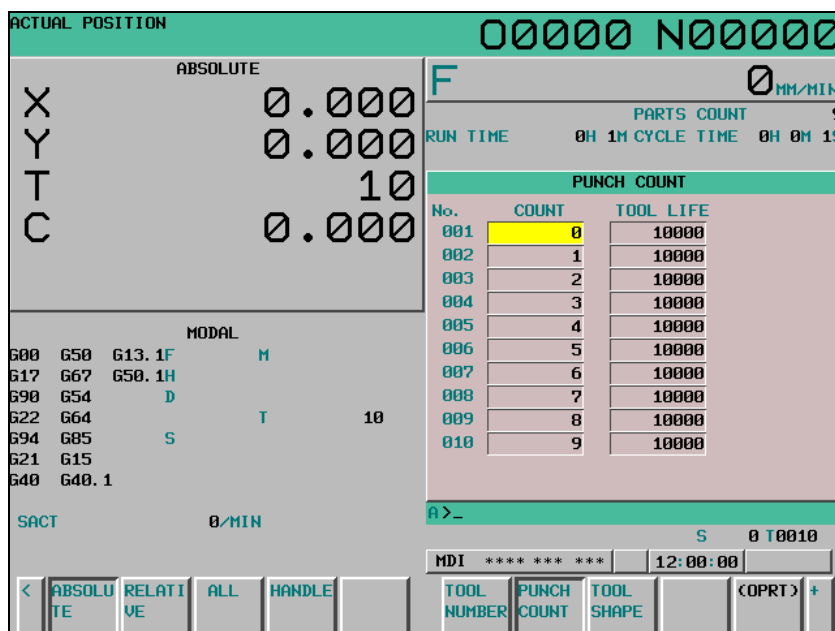


Fig.12.3.11 (d) Screen for the Number of Punch Operations(10.4-inch display unit)

Explanation

- Number

Tool registration number. This number is a serial number corresponding to the registered tool.

- Punch count number

Display and preset the total number of punch operation by each tool registered on the tool registration screens.

- Tool life

Specify the maximum number of punch operation by each tool registered on the tool registration screens.

NOTE

If the memory protect signal KEY1<Gn046.3> is set to "1", the punch count number and the tool life are writable although parameter writing is disabled (bit 0 (PWE) of parameter No.8900 is set to 0).

12.3.11.4 Displaying and setting items on the tool figure registration screen (for drawing figures)

Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key  until soft key [TOOL] appears.
- 4 Press the [TOOL] soft key to display the initial tool registration screen the initial tool registration screen.
- 5 Press the menu key . Then press the [TOOL SHAPE] soft key to display the screen for the number of press operations.
- 6 Move the cursor to the tool number to be changed.
 - Enter the registration number of the tool for which data is to be changed, then press the [NO.SRH] soft key.
 - Move the cursor to the tool number to be changed with the page keys   and cursor keys    .
- 7 To move the cursor to the tool number to be changed, press the cursor keys    .
- 8 Type the data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].

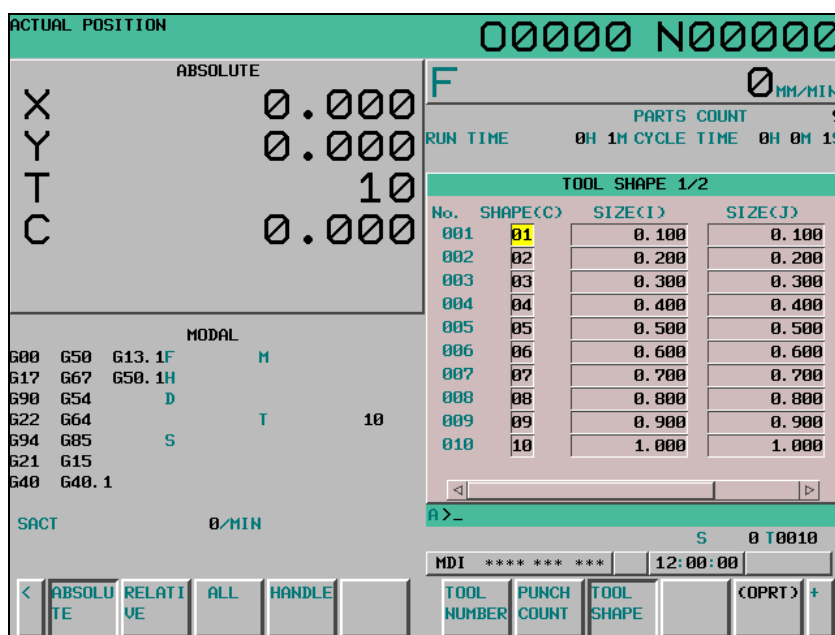


Fig.12.3.11 (e) Tool figure registration screen 1/2 (10.4-inch display unit)

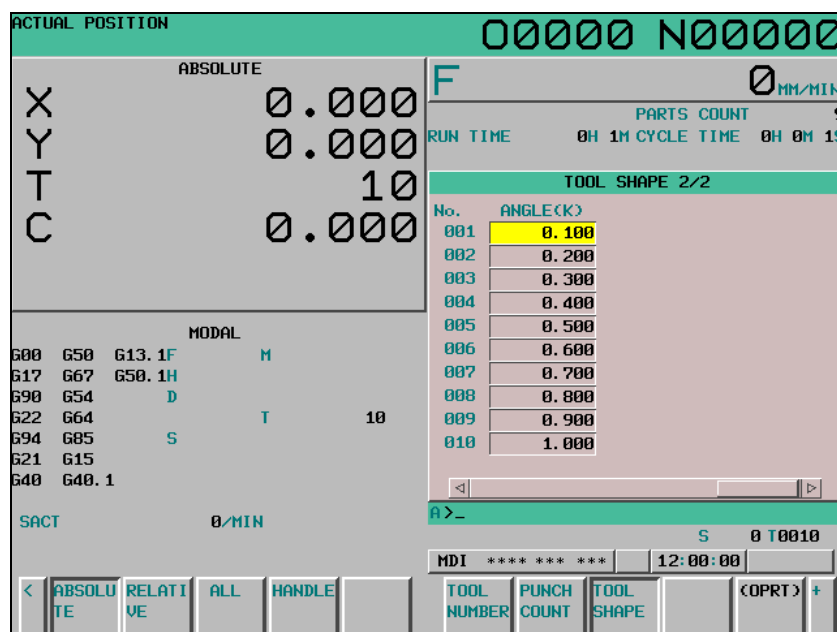


Fig.12.3.11 (f) Tool figure registration screen 2/2 (10.4-inch display unit)

Explanation

- Number

Tool registration number. This number is a serial number corresponding to the registered tool.

- Tool shape

Specify the tool shape for graphic operation.

The upper digit of the tool figure specifies the color of a line drawing and the lower digit specifies the figure.

Upper digit	Lower digit
0: Green	0: Asterisk
1: Red	1: Circle (line drawing)
2: Green	2: Quadrangle (line drawing)
3: Yellow	3: ellipse (line drawing)
4: Blue	4: Circle (filled-in drawing)
5: Pink	5: Quadrangle (filled-in drawing)
6: Sky blue	6: ellipse (filled-in drawing)
7: White	
8: Gray	

For details, refer to Table 12.3.11 (a).

- Tool size (I)

Specify the tool size for graphic operation.

For details, refer to Table 12.3.11 (a).

- Tool size (J)

Specify the tool size for graphic operation.

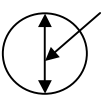
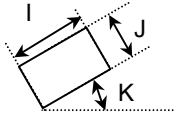
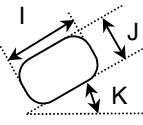
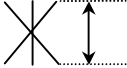
For details, refer to Table 12.3.11 (a).

- Tool angle (K)

Specify the tool size for graphic operation.

For details, refer to Table 12.3.11 (a).

Table12.3.11 (a) tool shape data

	Shape	I (Size)	J(Size)	K (Angle)
Circle	01 04		I : 0~999999.999 (Input unit : When the increment system is IS-B)	
Square, rectangle	02 05		I : 0~999999.999 J : 0~999999.999 K : 0~399.999 (Input unit : When the increment system is IS-B)	
Elongated hole	03 06		I : 0~999999.999 J : 0~999999.999 K : 0~399.999 (Input unit : When the increment system is IS-B)	
Others	00		I : 25.4mm(1inch) Fixed value	

12.3.12 Displaying and Setting Items on the Tool Registration Screen for Multiple Tools

Items concerning multiple tools, such as the number of a sub-tool for multiple tools, the position at which the turret is indexed for a sub-tool, and sub-tool position compensation, can be displayed or specified on the tool registration screens for multiple tools. Refer to the manual prepared by the machine tool builder for details, as the builder sets these items first.

12.3.12.1 Displaying and setting items on the tool number registration screen for multiple tools

Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key  until soft key [MULTI TOOL] appears.
- 4 Press the [MULTI TOOL] soft key to display the initial tool registration screen the tool number registration screen for multiple tools
- 5 Move the cursor to the tool number to be changed.
 - Enter the registration number of the tool for which data is to be changed, then press the [NO.SRH] soft key.
 - Move the cursor to the registration number of the tool to be changed with the page keys   and cursor keys    .
- 6 To move the cursor to the tool data to be changed, press the cursor keys    .
- 7 Type the data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].

ACTUAL POSITION 00000 N00000

ABSOLUTE

X 0.000
Y 0.000
T 10
C 0.000

MODAL

G00 G50 G13.1F M
G17 G67 G50.1H
G90 G54 D
G22 G64 T 10
G94 G85 S
G21 G15
G40 G40.1

MULTI NUMBER

No.	M. TOOL	ANGLE	Y-OFFSET
001	01	1.000	1.000
002	02	2.000	2.000
003	03	3.000	3.000
004	04	4.000	4.000
005	05	5.000	5.000
006	06	6.000	6.000
007	07	7.000	7.000
008	08	8.000	8.000
009	09	9.000	9.000
010	10	10.000	10.000

SACT 0/MIN

MDI **** * 12:00:00

MULTI SHAPE **MULTI NUMBER** **OPRT**

Fig.12.3.12 (a) Tool Number Registration Screen for Multiple Tools (10.4-inch display unit)

Explanation

- Number

Sub-tool registration number for multiple tools. This number is a serial number corresponding to the registered sub-tool.

- Sub-tool number

Specify sub-tool numbers used for controlling multiple tools.

- Angle

Specify angles used for indexing the turret for sub-tools.

- Y-axis offset

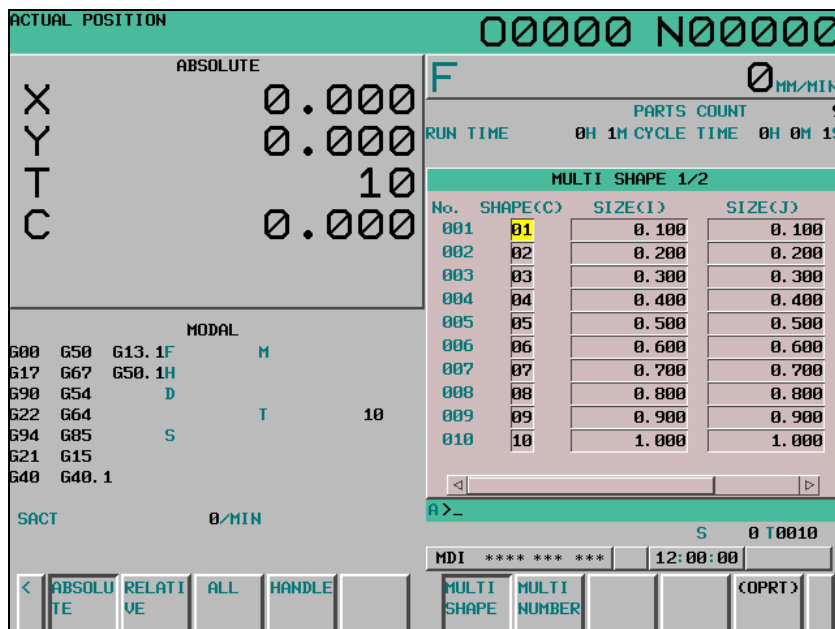
When the tool position compensation function is used (TOF, bit 2 of parameter No. 16263 is set to 1), specify tool position compensation for sub-tools along the Y-axis in the machine coordinate system.

12.3.12.2 Displaying and setting items on the tool figure registration screen for multiple tools (for drawing figures)

Procedure

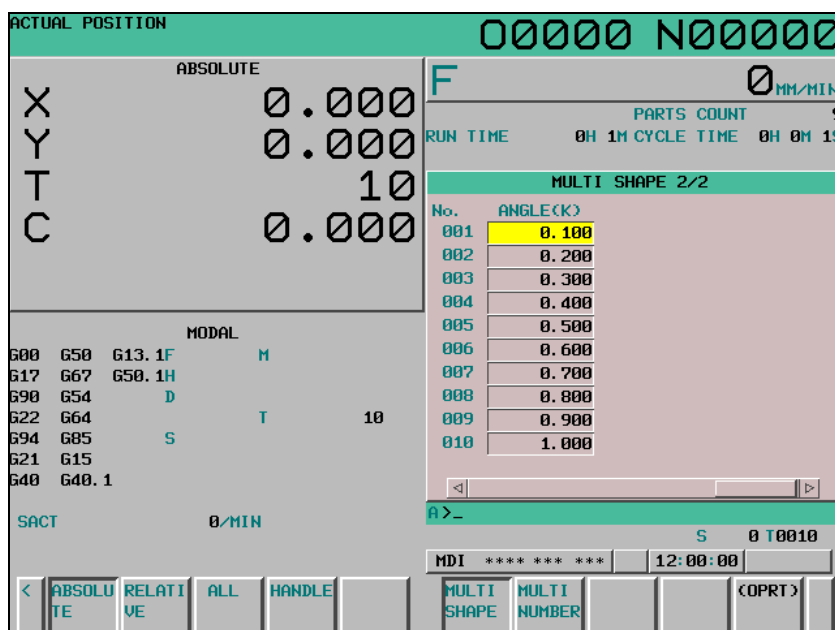
- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key until soft key [MULTI TOOL] appears.
- 4 Press the [MULTI TOOL] soft key to display the initial tool registration screen for multiple tools.
- 5 Press the menu key . Then press the [MULTI SHAPE] soft key to display the tool figure registration screen for multiple tools.
- 6 Move the cursor to the tool number to be changed.
 - Enter the registration number of the tool for which data is to be changed, then press the [NO.SRH] soft key.
 - Move the cursor to the tool number to be changed with the page keys and cursor keys .

- 7 To move the cursor to the tool data to be changed, press the cursor keys    .
- 8 Type the data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].



No.	SHAPE(C)	SIZE(I)	SIZE(J)
001	01	0.100	0.100
002	02	0.200	0.200
003	03	0.300	0.300
004	04	0.400	0.400
005	05	0.500	0.500
006	06	0.600	0.600
007	07	0.700	0.700
008	08	0.800	0.800
009	09	0.900	0.900
010	10	1.000	1.000

Fig.12.3.12 (b) Tool figure registration screen for multiple tools 1/2 (10.4-inch display unit)



No.	ANGLE(K)
001	0.100
002	0.200
003	0.300
004	0.400
005	0.500
006	0.600
007	0.700
008	0.800
009	0.900
010	1.000

Fig.12.3.12 (c) Tool figure registration screen for multiple tools 2/2 (10.4-inch display unit)

Explanation

- Number

Sub-tool registration number for multiple tools. This number is a serial number corresponding to the registered sub-tool.

- Tool shape

Specify the sub-tool shape for graphic operation.

The upper digit of the tool shape specifies the color of a line drawing and the lower digit specifies the shape.

Upper digit	Lower digit
0: Green	0: Asterisk
1: Red	1: Circle (line drawing)
2: Green	2: Quadrangle (line drawing)
3: Yellow	3: ellipse (line drawing)
4: Blue	4: Circle (filled-in drawing)
5: Pink	5: Quadrangle (filled-in drawing)
6: Sky blue	6: ellipse (filled-in drawing)
7: White	
8: Gray	

For details, refer to Table12.3.12 (a).

- Tool size (I)

Specify the sub-tool size for graphic operation.

For details, refer to Table12.3.12 (a).

- Tool size(J)

Specify the sub-tool size for graphic operation.

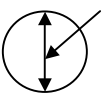
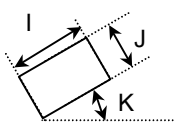
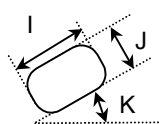
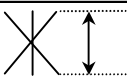
For details, refer to Table12.3.12 (a).

- Tool angle(K)

Specify the sub-tool size for graphic operation.

For details, refer to Table12.3.12 (a).

Table12.3.12 (a) Tool shape data

	Shape	I (Size)	J(Size)	K (Angle)
Circle	01 04		I : 0~999999.999 (Input unit : When the increment system is IS-B)	
Square, rectangle	02 05		I : 0~999999.999 J : 0~999999.999 K : 0~399.999 (Input unit : When the increment system is IS-B)	
Elongated hole	03 06		I : 0~999999.999 J : 0~999999.999 K : 0~399.999 (Input unit : When the increment system is IS-B)	
Others	00		I : 25.4mm(1inch) Fixed value	

12.3.13 Displaying and Setting Items on the Safety Zone Setting Screen

The safety zone can be displayed and changed.

By setting SF0, bit 0 of parameter No. 16500, the layout of the safety zone setting screen can be changed to type A or type B.

Displaying and Setting Items on the Safety Zone Setting Screen (Type A)

Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key  until soft key [SAFTY ZONE] appears.
- 4 Press the [SAFTY ZONE] soft key to display the safety zone setting screen.
- 5 The safety zone is displayed in the absolute coordinate system. By setting MDP, bit 0 of parameter No. 16502, the zone can be displayed in the machine coordinate system.
- 6 Usually, setting data is not enabled on the safety zone setting screen. However, Setting bit 4 (SZI) of parameter No.16502 allows setting data.
- 7 To move the cursor to the data of safety zone to be changed, press the page keys   and the cursor keys    .
- 8 Type the data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].

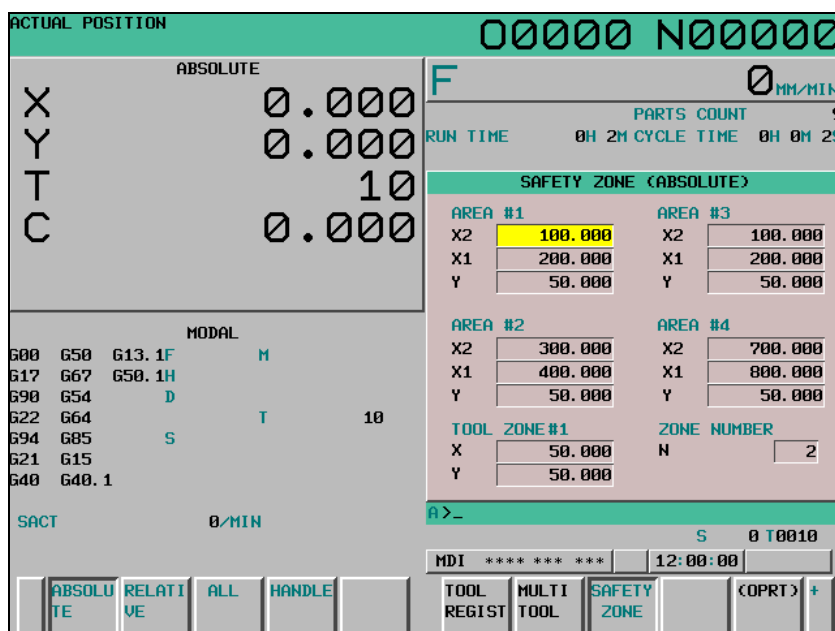


Fig.12.3.13 (a) Safety Zone Setting Screen (Type A), Absolute position display (10.4-inch display unit)

Fig.12.3.13 (b) Safety Zone Setting Screen (Type A), Machine position display (10.4-inch display unit)

NOTE

- 1 The tool position that is selected by selection signals for tool area of safety zone STZS0 - STZS3<Gn232.0 to .3>, is displayed.
- 2 Please set unnecessary safety zone to the tool coordinates used for reference position return.
 - At displaying in absolute coordinate :
Setting value = Coordinate by automatic coordinate system setting (parameter No.1250) — workpiece coordinate system offset amount
 - At displaying in machine coordinate : Setting 0.

Explanation

- **Safety zone #n (n: 1 to 4)**
 - X1**
Specify the coordinate of safety zone along the positive X direction.
 - X2**
Specify the coordinate of safety zone along the minus X direction
 - Y**
Specify the coordinate of safety zone along the Y direction
- **Tool position**
 - X**
Specify the dimensions of a tool along the X-axes. If the circle tool area is set, please set radius of the tool area.
 - Y**
Specify the dimensions of a tool along the Y-axes. If the circle tool area is set, please set 0.
- **Zone number**
Specified the number of safety zone detected when automatic safety-zone setting by external signal.

Displaying and Setting Items on the Safety Zone Setting Screen (Type B)

Procedure

- 1 Select the MDI mode.
- 2 Press function key .
- 3 Press the continuous menu key  until soft key [SAFTY ZONE] appears.
- 4 Press the [SAFTY ZONE] soft key to display the safety zone setting screen.
- 5 The safety zone is displayed in the absolute coordinate system. By setting MDP, bit 0 of parameter No. 16502, the zone can be displayed in the machine coordinate system.
- 6 Usually, setting data is not enabled on the safety zone setting screen. However, Setting bit 4 (SZI) of parameter No.16502 allows setting data.
- 7 To move the cursor to the data of safety zone to be changed, press the page keys   and the cursor keys    .
- 8 Type the data you want to set or modify, and press soft key [INPUT].
To modify the data, type the data to increase or decrease, then press soft key [+INPUT].
Alternatively, type the data to modify, then press soft key [+INPUT].

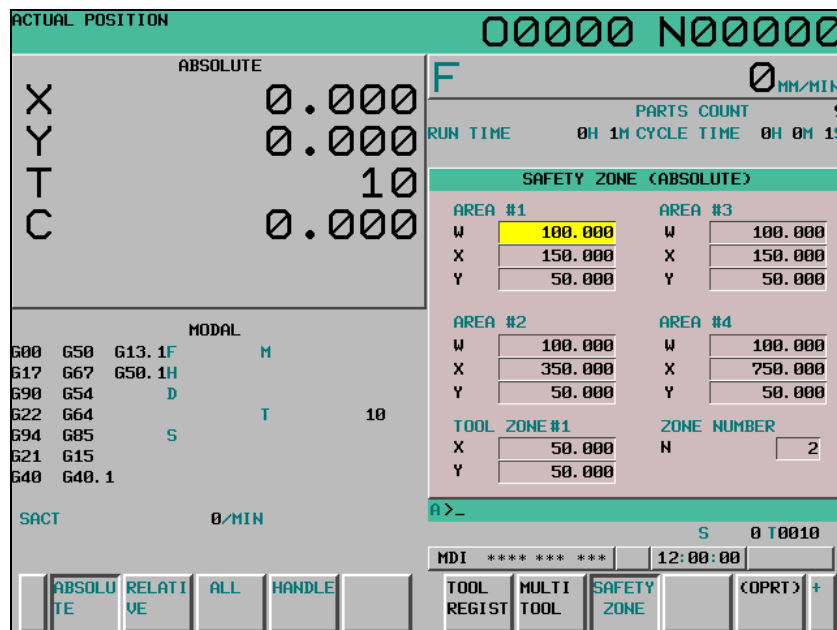


Fig.12.3.13 (c) Safety Zone Setting Screen (Type B), Machine position display (10.4-inch display unit)

Fig.12.3.13 (d) Safety Zone Setting Screen (Type B), Machine position display (10.4-inch display unit)

NOTE

- 1 The tool position that is selected by selection signals for tool area of safety zone STZS0 - STZS3<Gn232.0 to .3>, is displayed.
- 2 Please set unnecessary safety zone to the tool coordinates used for reference position return.
 - At displaying in absolute coordinate :
W is set to 0.
Setting value for X, Y = Coordinate by automatic coordinate system setting (parameter No.1250) — workpiece coordinate system offset amount
 - At displaying in machine coordinate : Setting 0.

Explanation

- **Safety zone (#1 - #4)**
 - W**
Specify the width of the safety zone along the X-axes.
 - X**
Specify the center coordinate of the safety zone along the X-axes.
 - Y**
Specify the coordinate of the safety zone along the Y-axes.
- **Tool position**
 - X**
Specify the dimensions of a tool along the X-axes. If the circle tool area is set, please set radius of the tool area.
 - Y**
Specify the dimensions of a tool along the Y-axes. If the circle tool area is set, please set 0.
- **Zone number**
Specified the number of safety zone detected when automatic safety-zone setting by external signal.

12.4 SCREENS DISPLAYED BY FUNCTION KEY

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 unit. Parameters can also be set with external input/output devices such as the memory card (see Chapter III-8, “DATA INPUT/OUTPUT”).

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 .

See Chapter III-7, “ALARM AND SELF-DIAGNOSIS FUNCTIONS” for the diagnosis screens displayed by pressing function key .

Section 12.4, “SCREENS DISPLAYED BY FUNCTION KEY ”, consists of the following subsections:

12.4.1	Displaying and Setting Parameters	582
12.4.2	Displaying and Setting Pitch Error Compensation Data.....	584
12.4.3	Servo Setting.....	587
12.4.4	Servo Tuning.....	590
12.4.5	Color Setting Screen	591
12.4.6	Parameter Setting Support Screen.....	593
12.4.7	Periodic Maintenance Screen.....	615
12.4.8	System Configuration Screen	622
12.4.9	Overview of the History Function	624

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/disabling parameter writing described below.
- 2 Press function key .
- 3 Press chapter selection soft key [PARAM] to display the parameter screen.

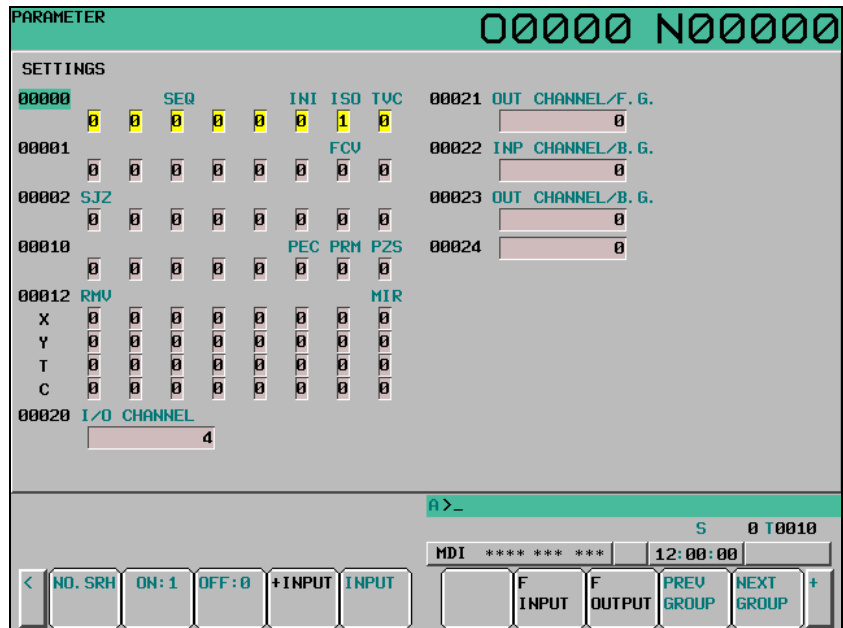


Fig. 12.4.1 (a) PARAMETER screen (10.4-inch display unit)

- 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]. 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.

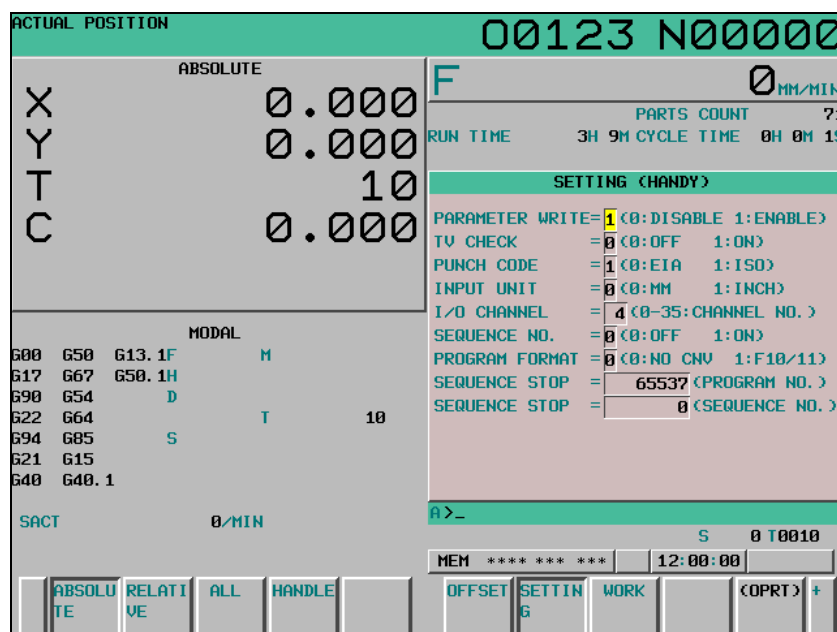


Fig. 12.4.1 (b) SETTING screen (10.4-inch display unit)

- 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 Chapter, "DATA INPUT/OUTPUT" 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, B-64560EN) 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.

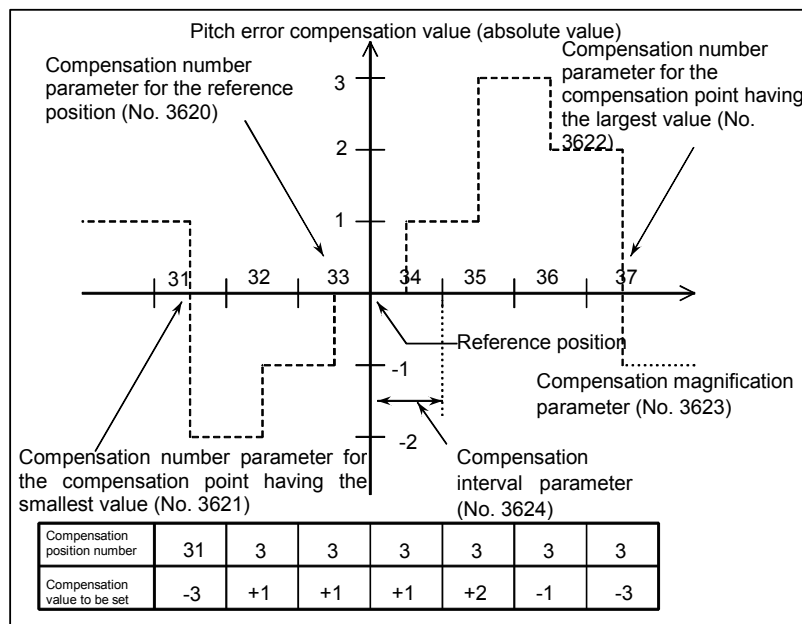
Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.

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, "DATA INPUT/OUTPUT"). Compensation data can also be written directly with the MDI unit.

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.



- 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 distance (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

- 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 (BDP_x) 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
- Press function key .
 - Press the continuous menu key , then press chapter selection soft key [PITCH ERROR]. The following screen is displayed:

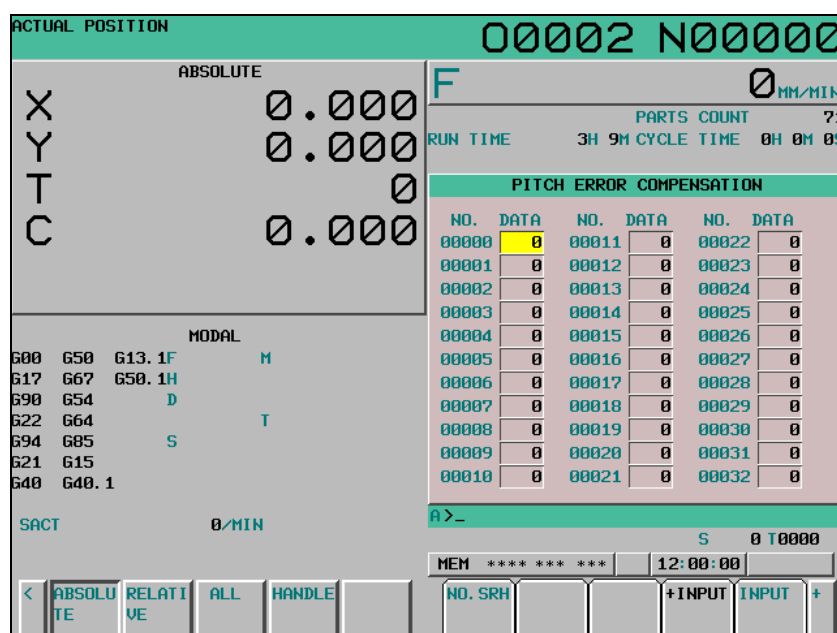


Fig. 12.4.2 (a) PITCH ERROR COMPENSATION screen (10.4-inch display unit)

- 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].

If bit 5 (PAD) of parameter No. 11350 is 1, an axis name is displayed next to the compensation point number set in parameter No. 3621 for determining the most negative pitch error compensation point number.

If the bi-directional pitch error compensation function is enabled, "+ axis name" is displayed next to the compensation point number set in parameter No. 3621 for setting the compensation point during movement in the positive direction. And, "- axis name" is displayed next to the compensation point number set in parameter No. 3626 for setting the compensation point during movement in the negative direction.

NOTE

- 1 If the setting of the pitch error compensation parameter is not correct, the axis name of that axis is not displayed.
- 2 For a rotation axis, an axis name is displayed next to the pitch error compensation point number of the reference position set in parameter No. 3620.

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)

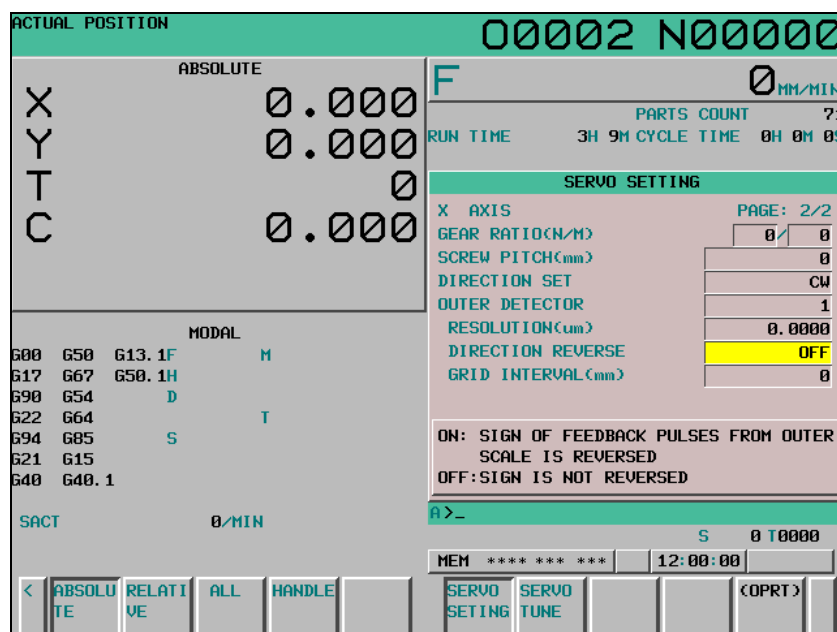


Fig. 12.4.3 (c) Servo setting screen (DIRECTION REVERSE) (10.4-inch)

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.

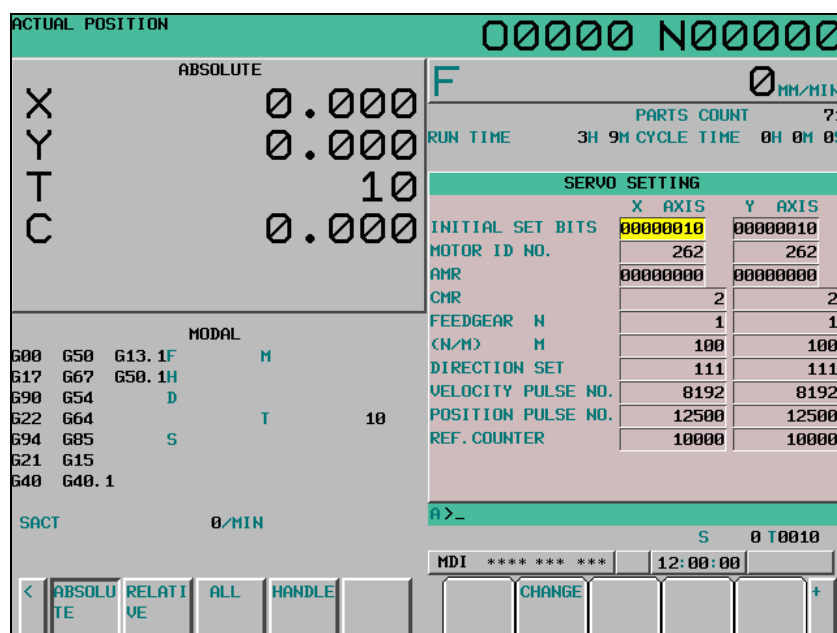


Fig. 12.4.3 (e) Servo setting screen for inputting parameters (10.4-inch)

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 Turn on the power in the emergency stop state.
- 2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
- 3 Press function key , continuous menu key , then soft key [SERVO SETTING] ([SV.SET] for the 8.4-inch display unit).
- 4 Press soft key [SERVO TUNE] ([SV.TUN] for the 8.4-inch display unit) to select the servo tuning screen.

The following screen (Fig. 12.4.4 (a)) appears:

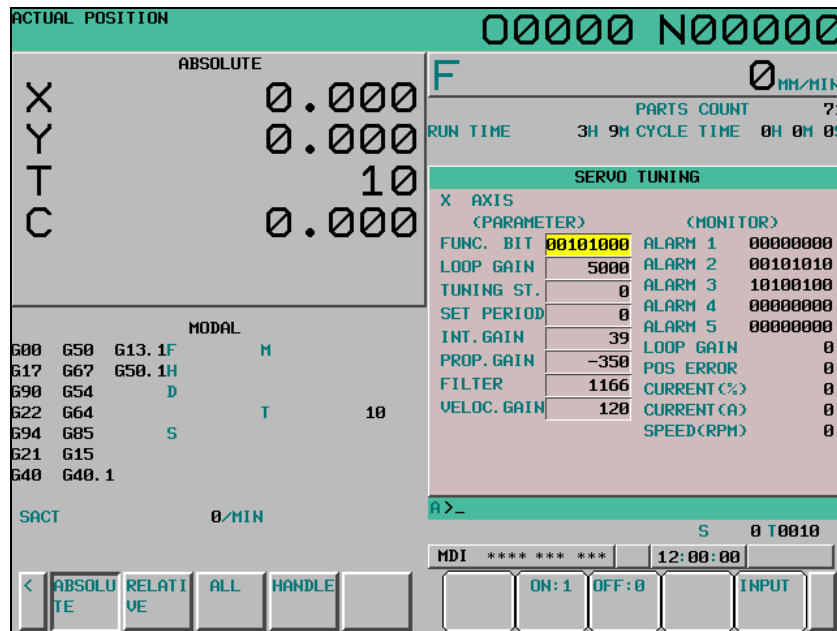


Fig. 12.4.4 (a) Servo tuning screen (10.4-inch display unit)

- 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].

12.4.5 Color Setting Screen

Screen colors can be set on the color setting screen.

Displaying the color setting screen

Procedure

- 1 Press function key .
- 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. 12.4.5 (a) Color setting screen (10.4-inch display unit)

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.
- 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.

12.4.6 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, 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.

12.4.6.1 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
- MISCELLANY

[TUNING]

- SERVO 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 .
- 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.

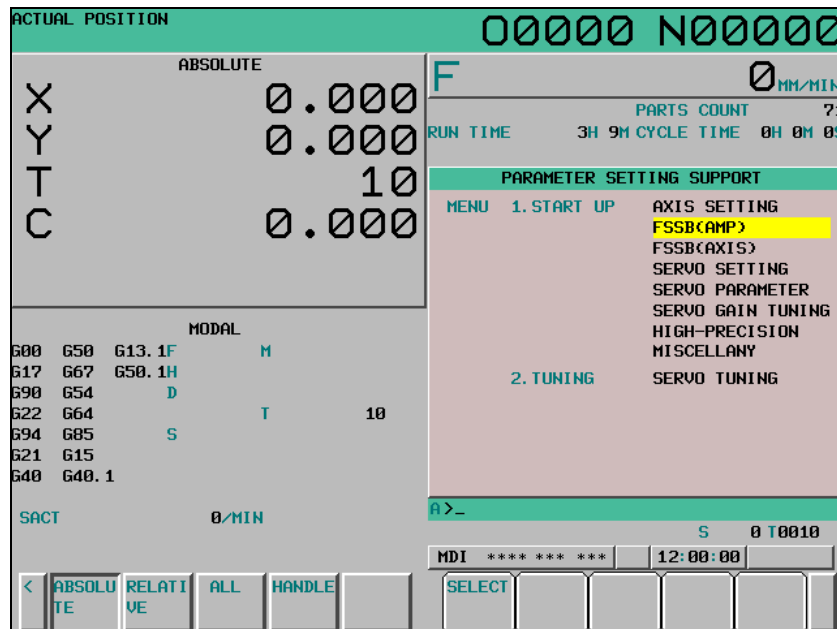


Fig. 12.4.6.1 (a) Menu screen for parameter setting support (10.4-inch)

- 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.)

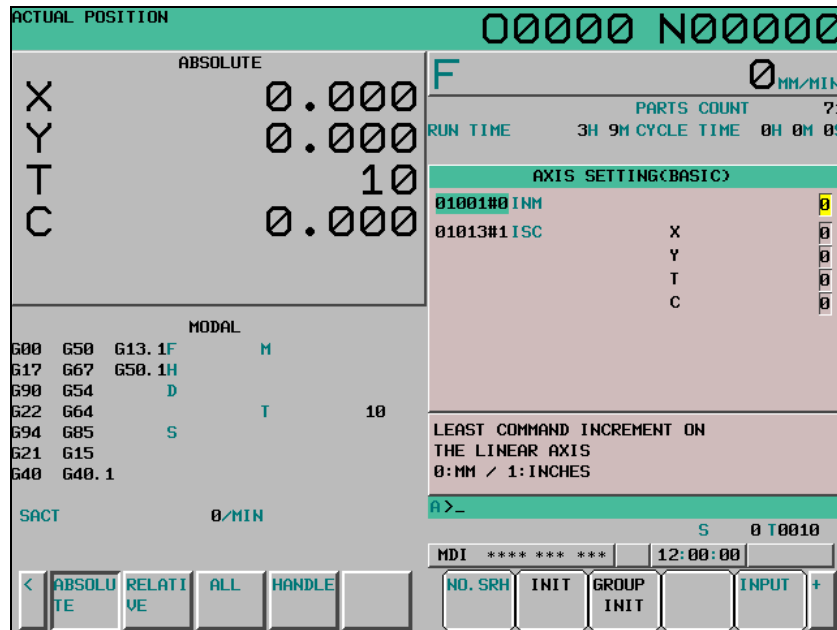


Fig. 12.4.6.1 (b) Axis setting screen (10.4-inch)

- 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 12.4.6.1 (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.
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. 12.4.6.1 (b) Items displayed with [TUNING]

Display item	Description
SERVO TUNING	Servo tuning screen

NOTE

Some items may not be displayed, depending on the system configuration.

12.4.6.2 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.

(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.

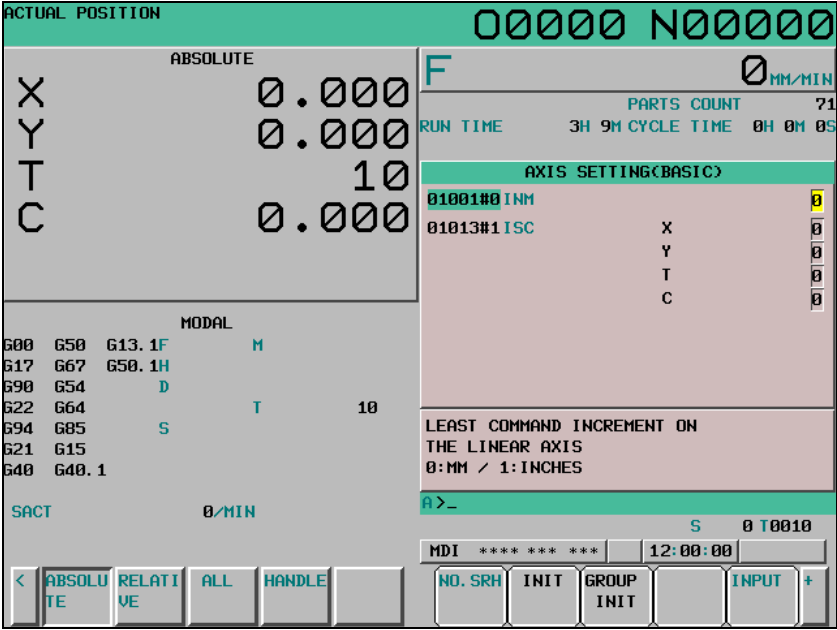


Fig. 12.4.6.2 (a) Parameter setting support screen (axis setting) (10.4-inch)

- 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.

12.4.6.3 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).

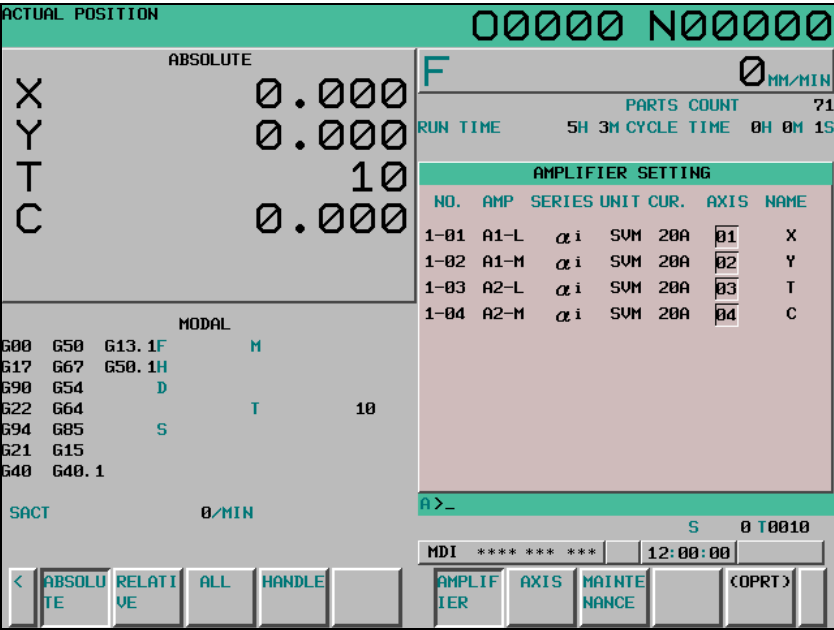


Fig. 12.4.6.3 (a) FSSB amplifier setting screen (10.4-inch)

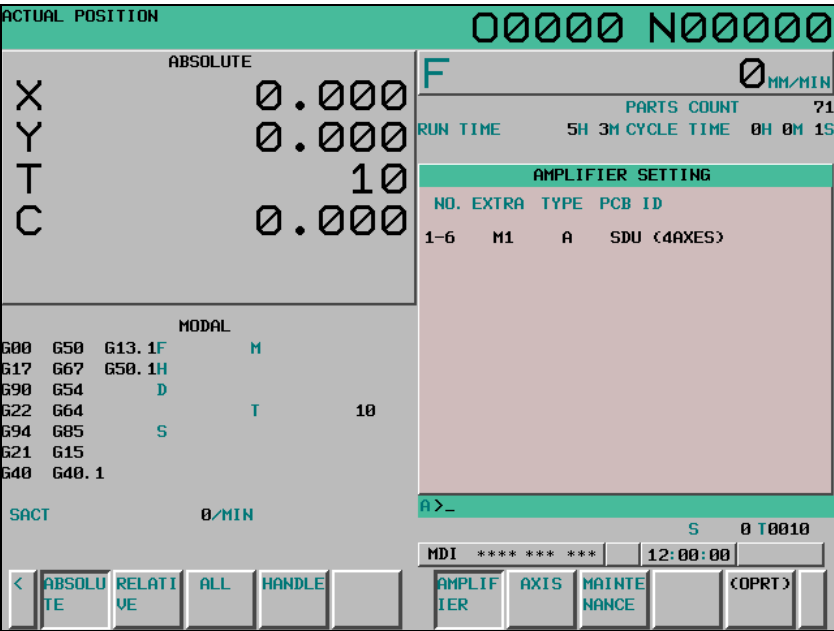


Fig. 12.4.6.3 (b) FSSB amplifier setting screen 2 (10.4-inch)

12.4.6.4 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).

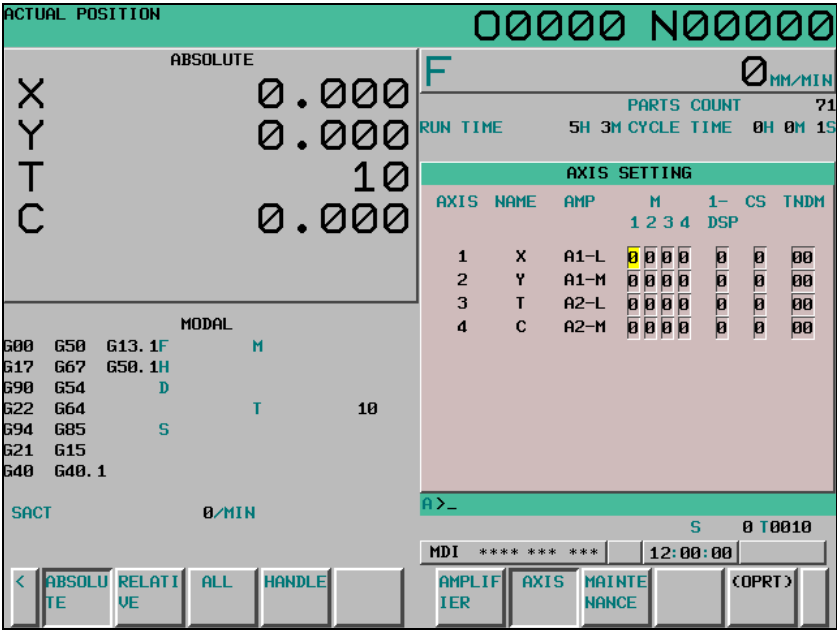


Fig. 12.4.6.4 (a) FSSB axis setting screen (10.4-inch)

12.4.6.5 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.

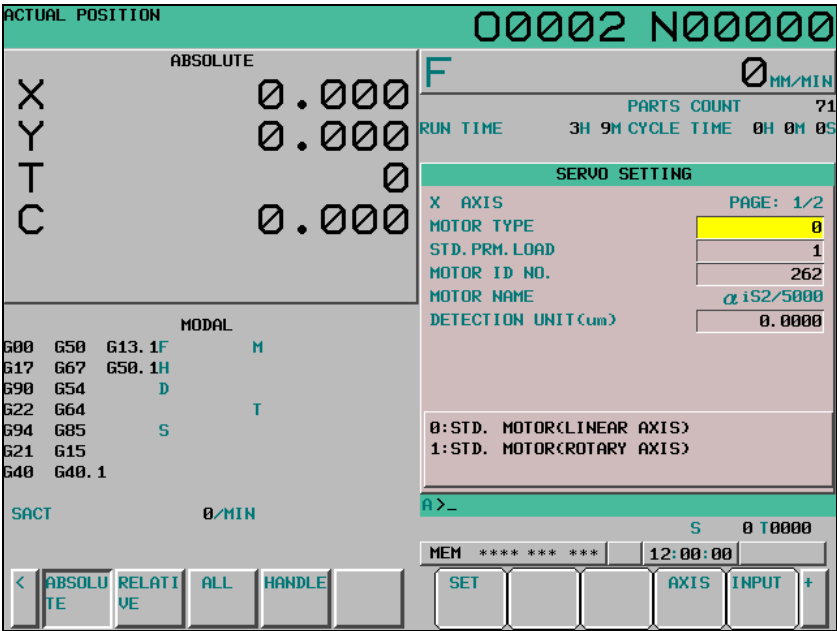


Fig. 12.4.6.5 (a) Servo setting screen (10.4-inch)

12.4.6.6 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. 12.4.6.6 (a) Servo parameter screen (10.4-inch)

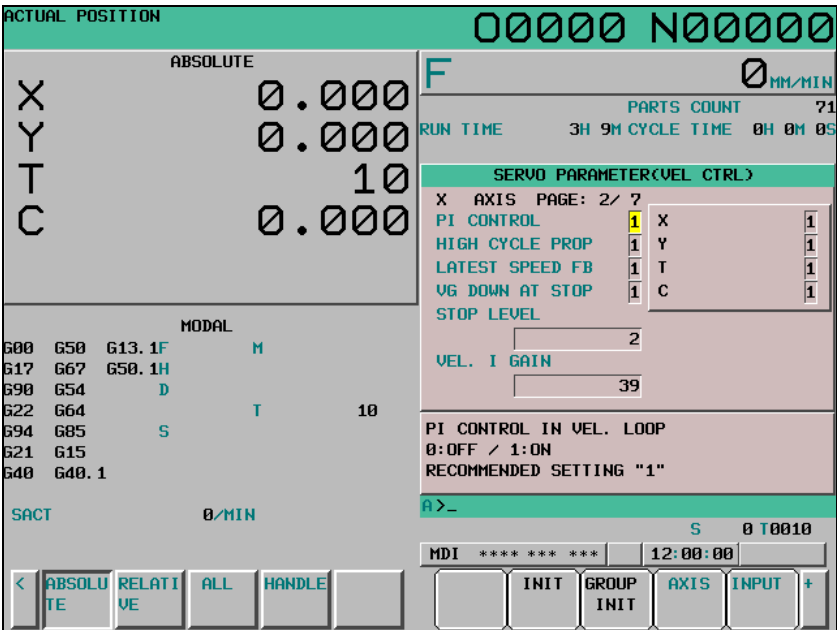


Fig. 12.4.6.6 (b) Servo parameter screen (display for each axis) (10.4-inch)

ACTUAL POSITION 00000 N00000

ABSOLUTE

X 0.000

Y 0.000

T 10

C 0.000

MODAL

G00 G50 G13.1F M

G17 G67 G50.1H

G90 G54 D T 10

G22 G64

G94 G85 S

G21 G15

G40 G40.1

SERVO PARAMETER(VEL CTRL)

02003#3 PI CONTROL X 1

Y 1

T 1

C 1

02017#7 HIGH CYCLE PROP X 1

Y 1

T 1

C 1

PI CONTROL IN VEL. LOOP

0:OFF / 1:ON

RECOMMENDED SETTING "1"

A>_

S 0 T0010

MDI ***** 12:00:00

< ABSOLUTE RELATIVE ALL HANDLE

INIT GROUP INIT AXIS INPUT +

Fig. 12.4.6.6 (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)".)

12.4.6.7 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.

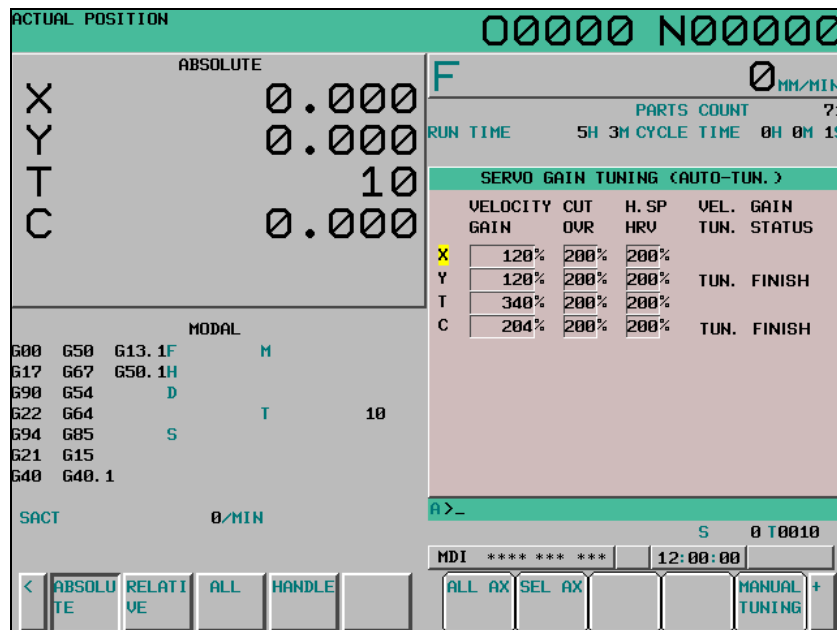


Fig. 12.4.6.7 (a) Servo gain tuning screen (automatic tuning screen) (10.4-inch)

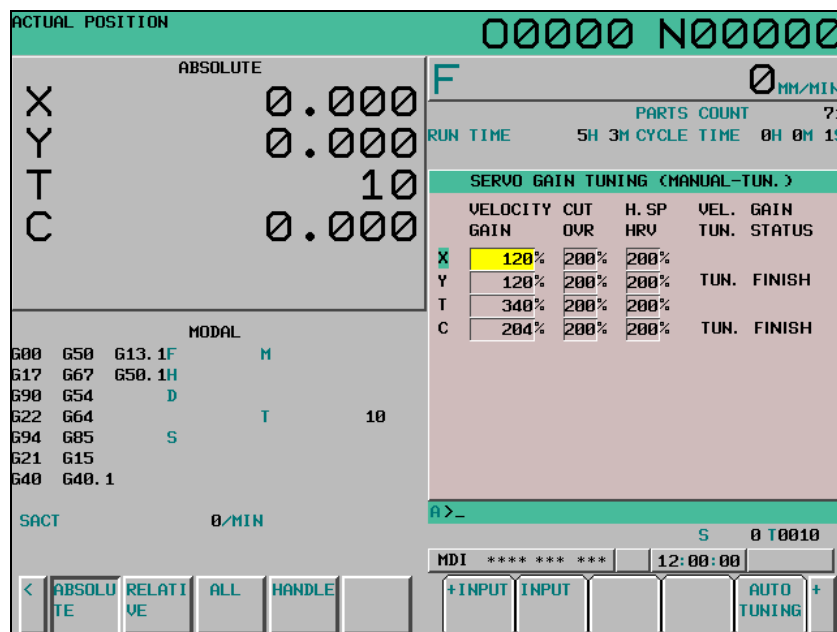


Fig. 12.4.6.7 (b) Servo gain tuning screen (manual tuning screen) (10.4-inch)

Display item

VELOCITY GAIN

The value calculated by the CNC with the following expression on the basis of parameter No. 2021 is displayed.

(Expression):

$$\text{VELOCITY GAIN} = (256 + \text{No. 2021}) / 256 \times 100$$

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.

ACTUAL POSITION 00000 N00000

ABSOLUTE

X	0.000
Y	0.000
T	10
C	0.000

MODAL

G00	G50	G13.1F	M
G17	G67	G50.1H	
G90	G54	D	
G22	G64		T 10
G94	G85	S	
G21	G15		
G40	G40.1		

SERV GAIN TUNING (AUTO-TUN.)

	VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. GAIN TUN. STATUS
X	120%	200%	200%	
Y	120%	200%	200%	TUN. FINISH
T	340%	200%	200%	
C	204%	200%	200%	TUN. FINISH

MDI ***** 12:00:00

ABSOLUTE **RELATIVE** **ALL** **HANDLE** **ALL AX** **SEL AX** **MANUAL TUNING**

Fig. 12.4.6.7 (c) Servo gain automatic tuning screen (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.

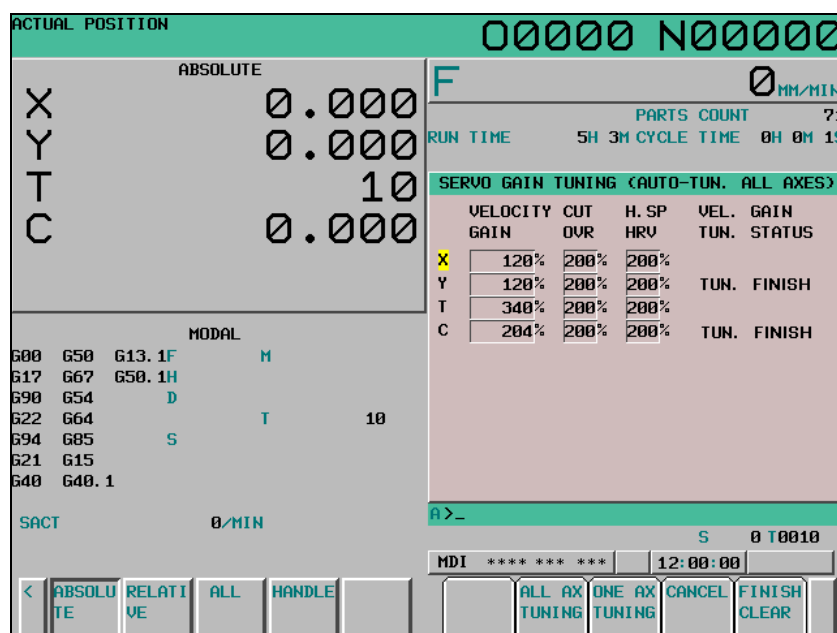


Fig. 12.4.6.7 (d) Automatic tuning screen (all axis tuning) (10.4-inch)

- 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 T-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 T-axis; then automatic tuning stops. If soft key [ONE AX TUNING] is pressed again at this time, the T-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.

ACTUAL POSITION				00000 N00000																												
ABSOLUTE				F 0 MM/MIN																												
X	0.000			PARTS COUNT 71																												
Y	0.000			RUN TIME 5H 3M CYCLE TIME 0H 0M 1S																												
T	10			SERVO GAIN TUNING (AUTO-TUN. ALL AXES)																												
C	0.000			<table border="1"> <thead> <tr> <th>VELOCITY GAIN</th> <th>CUT OVR</th> <th>H. SP HRV</th> <th>VEL. TUN.</th> <th>GAIN STATUS</th> </tr> </thead> <tbody> <tr> <td>X 95%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> <tr> <td>Y 120%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> <tr> <td>T 50%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> <tr> <td>C 204%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> </tbody> </table>				VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN.	GAIN STATUS	X 95%	200%	200%	TUN.	FINISH	Y 120%	200%	200%	TUN.	FINISH	T 50%	200%	200%	TUN.	FINISH	C 204%	200%	200%	TUN.	FINISH
VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN.	GAIN STATUS																												
X 95%	200%	200%	TUN.	FINISH																												
Y 120%	200%	200%	TUN.	FINISH																												
T 50%	200%	200%	TUN.	FINISH																												
C 204%	200%	200%	TUN.	FINISH																												
MODAL				A>_ S 0 T0010																												
G00 G50 G13.1F	M			MDI **** * 12:00:00																												
G17 G67 G50.1H				ALL AX TUNING ONE AX TUNING CANCEL FINISH CLEAR																												
G90 G54 D	T 10																															
G22 G64																																
G94 G85 S																																
G21 G15																																
G40 G40.1																																
SACT 0/MIN																																

↓ Cleared

ACTUAL POSITION				00000 N00000																												
ABSOLUTE				F 0 MM/MIN																												
X	0.000			PARTS COUNT 71																												
Y	0.000			RUN TIME 5H 3M CYCLE TIME 0H 0M 1S																												
T	10			SERVO GAIN TUNING (AUTO-TUN. ALL AXES)																												
C	0.000			<table border="1"> <thead> <tr> <th>VELOCITY GAIN</th> <th>CUT OVR</th> <th>H. SP HRV</th> <th>VEL. TUN.</th> <th>GAIN STATUS</th> </tr> </thead> <tbody> <tr> <td>X 95%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> <tr> <td>Y 120%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> <tr> <td>T 50%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> <tr> <td>C 204%</td> <td>200%</td> <td>200%</td> <td>TUN.</td> <td>FINISH</td> </tr> </tbody> </table>				VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN.	GAIN STATUS	X 95%	200%	200%	TUN.	FINISH	Y 120%	200%	200%	TUN.	FINISH	T 50%	200%	200%	TUN.	FINISH	C 204%	200%	200%	TUN.	FINISH
VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN.	GAIN STATUS																												
X 95%	200%	200%	TUN.	FINISH																												
Y 120%	200%	200%	TUN.	FINISH																												
T 50%	200%	200%	TUN.	FINISH																												
C 204%	200%	200%	TUN.	FINISH																												
MODAL				A>_ S 0 T0010																												
G00 G50 G13.1F	M			MDI **** * 12:00:00																												
G17 G67 G50.1H				ALL AX TUNING ONE AX TUNING CANCEL FINISH CLEAR																												
G90 G54 D	T 10																															
G22 G64																																
G94 G85 S																																
G21 G15																																
G40 G40.1																																
SACT 0/MIN																																

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.

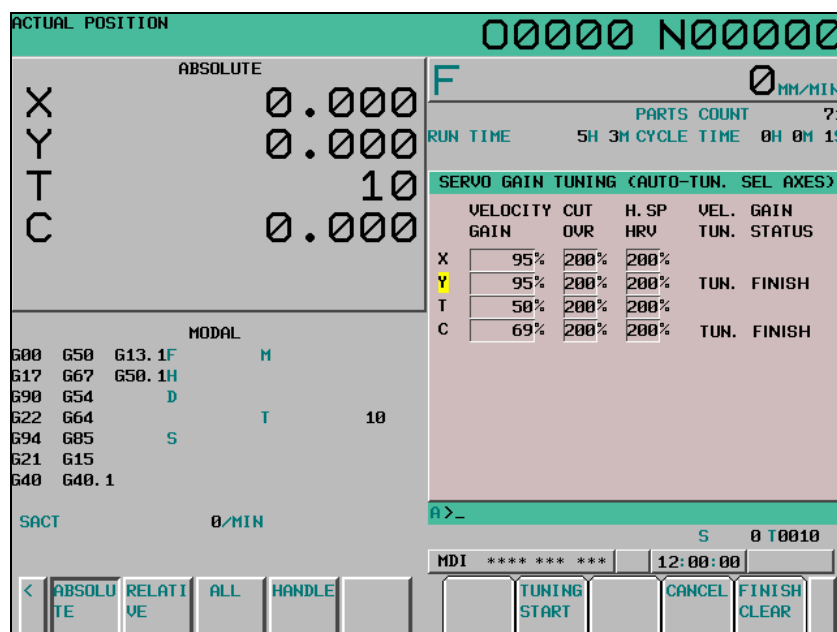


Fig. 12.4.6.7 (e) Automatic tuning screen (select axis tuning) (10.4-inch)

- 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.

ACTUAL POSITION				00000 N00000																							
ABSOLUTE				F 0 MM/MIN																							
X	0.000			PARTS COUNT 71																							
Y	0.000			RUN TIME 5H 3M CYCLE TIME 0H 0M 1S																							
T	10			SERVO GAIN TUNING (AUTO-TUN. SEL AXES)																							
C	0.000			<table border="1"> <thead> <tr> <th>VELOCITY GAIN</th> <th>CUT OVR</th> <th>H. SP HRV</th> <th>VEL. TUN. STATUS</th> </tr> </thead> <tbody> <tr> <td>X 95%</td> <td>200%</td> <td>200%</td> <td></td> </tr> <tr> <td>Y 95%</td> <td>200%</td> <td>200%</td> <td>TUN. FINISH</td> </tr> <tr> <td>T 50%</td> <td>200%</td> <td>200%</td> <td></td> </tr> <tr> <td>C 69%</td> <td>200%</td> <td>200%</td> <td>TUN. FINISH</td> </tr> </tbody> </table>				VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN. STATUS	X 95%	200%	200%		Y 95%	200%	200%	TUN. FINISH	T 50%	200%	200%		C 69%	200%	200%	TUN. FINISH
VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN. STATUS																								
X 95%	200%	200%																									
Y 95%	200%	200%	TUN. FINISH																								
T 50%	200%	200%																									
C 69%	200%	200%	TUN. FINISH																								
MODAL				A>_ S 0 T0010																							
G00 G50 G13.1F	M			MDI **** * 12:00:00																							
G17 G67 G50.1H				TUNING START CANCEL FINISH CLEAR																							
G90 G54 D	T 10																										
G22 G64																											
G94 G85 S																											
G21 G15																											
G40 G40.1																											
SACT 0/MIN																											

Cleared

ACTUAL POSITION				00000 N00000																							
ABSOLUTE				F 0 MM/MIN																							
X	0.000			PARTS COUNT 71																							
Y	0.000			RUN TIME 5H 3M CYCLE TIME 0H 0M 1S																							
T	10			SERVO GAIN TUNING (AUTO-TUN. SEL AXES)																							
C	0.000			<table border="1"> <thead> <tr> <th>VELOCITY GAIN</th> <th>CUT OVR</th> <th>H. SP HRV</th> <th>VEL. TUN. STATUS</th> </tr> </thead> <tbody> <tr> <td>X 95%</td> <td>200%</td> <td>200%</td> <td></td> </tr> <tr> <td>Y 95%</td> <td>200%</td> <td>200%</td> <td>TUN. FINISH</td> </tr> <tr> <td>T 50%</td> <td>200%</td> <td>200%</td> <td></td> </tr> <tr> <td>C 69%</td> <td>200%</td> <td>200%</td> <td>TUN. FINISH</td> </tr> </tbody> </table>				VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN. STATUS	X 95%	200%	200%		Y 95%	200%	200%	TUN. FINISH	T 50%	200%	200%		C 69%	200%	200%	TUN. FINISH
VELOCITY GAIN	CUT OVR	H. SP HRV	VEL. TUN. STATUS																								
X 95%	200%	200%																									
Y 95%	200%	200%	TUN. FINISH																								
T 50%	200%	200%																									
C 69%	200%	200%	TUN. FINISH																								
MODAL				A>_ S 0 T0010																							
G00 G50 G13.1F	M			MDI **** * 12:00:00																							
G17 G67 G50.1H				TUNING START CANCEL FINISH CLEAR																							
G90 G54 D	T 10																										
G22 G64																											
G94 G85 S																											
G21 G15																											
G40 G40.1																											
SACT 0/MIN																											

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.

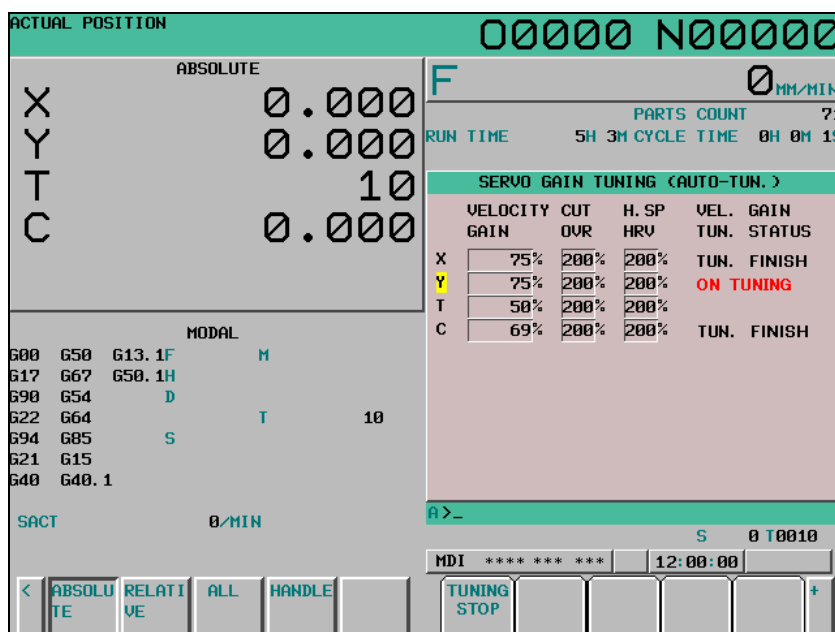


Fig. 12.4.6.7 (f) All axes being tuned (10.4-inch)

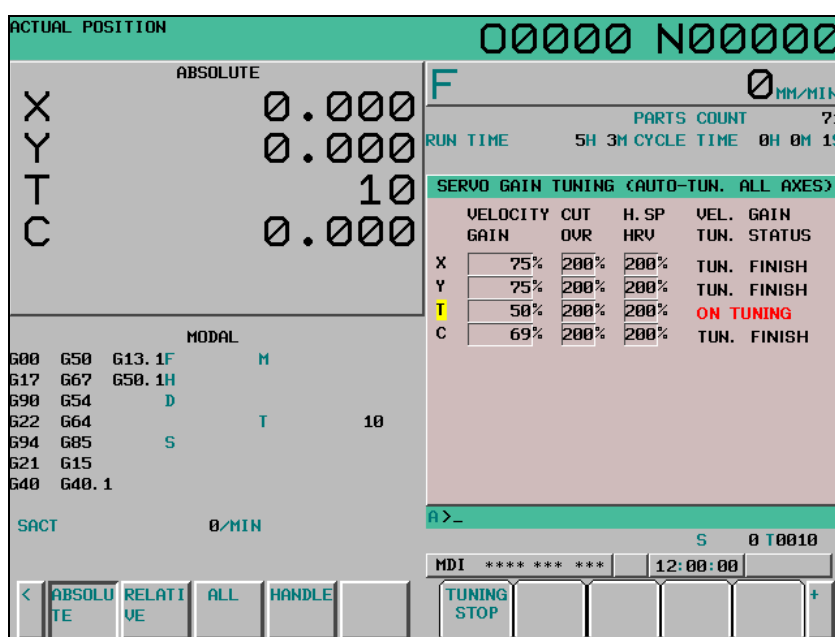


Fig. 12.4.6.7 (g) Selected axis being tuned (10.4-inch)

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.

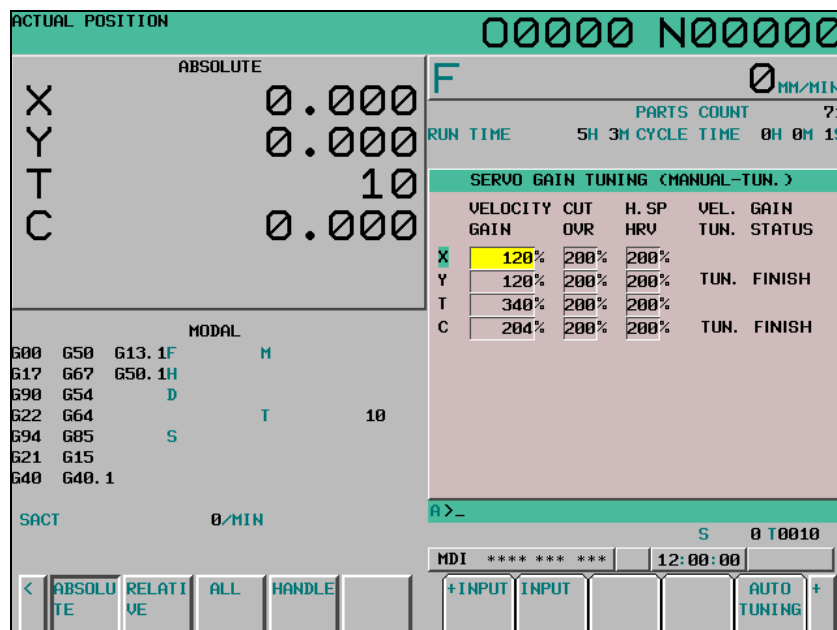


Fig. 12.4.6.7 (h) Servo gain tuning screen (manual tuning screen) (10.4-inch)

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.

12.4.6.8 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.)

The screen displays servo parameters for each axis (X, Y, T, C) under the 'ABSOLUTE' mode. The 'MODAL' section shows parameters G00, G50, G13.1F, M, G17, G67, G50.1H, G90, G54, D, G22, G64, G94, G85, S, T, 10, G21, G15, G40, G40.1. The 'HIGH-PRECISION (TIME CONST)' section shows parameters for X, Y, T, C axes, including RAPID LINEAR TC, RAPID BELL TC, ACC./DEC. TYPE, and AFT. TC. The 'TIME CONSTANT FOR LINEAR ACC./DEC. IN RAPID TRAVERSE' is also displayed. The bottom status bar shows MDI, INIT, GROUP INIT, AXIS, and INPUT keys.

Fig. 12.4.6.8 (a) High-precision setting screen (display for each axis) (10.4-inch)

The screen displays servo parameters for each item (01620, 01621) under the 'ABSOLUTE' mode. The 'MODAL' section shows parameters G00, G50, G13.1F, M, G17, G67, G50.1H, G90, G54, D, G22, G64, G94, G85, S, T, 10, G21, G15, G40, G40.1. The 'HIGH-PRECISION (TIME CONST)' section shows parameters for X, Y, T, C axes, including RAPID LINEAR TC, RAPID BELL TC, ACC./DEC. TYPE, and AFT. TC. The 'TIME CONSTANT FOR LINEAR ACC./DEC. IN RAPID TRAVERSE' is also displayed. The bottom status bar shows MDI, MENU, CHANGE, and other keys.

Fig. 12.4.6.8 (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.

12.4.6.9 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.

The screenshot shows the 'MISCELLANY (MISC)' screen. At the top, 'ACTUAL POSITION' is 00000 N00000. The 'ABSOLUTE' section shows X, Y, T, and C coordinates. The 'MODAL' section shows G00, G50, G13, 1F, M, G17, G67, G50, 1H, G90, G54, D, G22, G64, T, 10, G94, G85, S, G21, G15, G40, G40.1. The 'MISCELLANY (MISC)' section shows parameters 03017 (RST OUTPUT TIME), 03030 (M-CODE DIGITS), 03716#0 (A/S), and 03717 (SPDL INDEX NO.). The 'OUTPUT TIME OF RESET SIGNAL RST.' is 0. The 'SACT' section shows 0/MIN. The 'MDI' section shows 12:00:00. The 'NO. SRH' section shows 12:00:00. The 'INIT' section shows 12:00:00. The 'GROUP INIT' section shows 12:00:00. The 'INPUT' section shows 12:00:00.

Fig. 12.4.6.9 (a) Parameter setting support screen (miscellaneous) (10.4-inch)

12.4.6.10 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.

The screenshot shows the 'SERVO TUNING' screen. At the top, 'ACTUAL POSITION' is 00000 N00000. The 'ABSOLUTE' section shows X, Y, T, and C coordinates. The 'MODAL' section shows G00, G50, G13, 1F, M, G17, G67, G50, 1H, G90, G54, D, G22, G64, T, 10, G94, G85, S, G21, G15, G40, G40.1. The 'SERVO TUNING' section shows parameters for X AXIS (PARAMETER) and (MONITOR). The 'X AXIS (PARAMETER)' section shows FUNC. BIT (00101000), LOOP GAIN (5000), TUNING ST. (0), SET PERIOD (0), INT. GAIN (39), PROP. GAIN (-350), FILTER (1166), and VELOC. GAIN (120). The '(MONITOR)' section shows ALARM 1 (00000000), ALARM 2 (00101010), ALARM 3 (10100100), ALARM 4 (00000000), ALARM 5 (00000000), LOOP GAIN (0), POS ERROR (0), CURRENT (%) (0), CURRENT (A) (0), and SPEED (RPM) (0). The 'SACT' section shows 0/MIN. The 'MDI' section shows 12:00:00. The 'ON:1' section shows 12:00:00. The 'OFF:0' section shows 12:00:00. The 'INPUT' section shows 12:00:00.

Fig. 12.4.6.10 (a) Servo tuning screen (10.4-inch)

Explanation**- Parameters displayed for parameter setting support****Table 12.4.6.10 (a) Parameters displayed for parameter setting support (1)**

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	ZMlx	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 12.4.6.10 (b) Parameters displayed for parameter setting support (2)

Menu	Group	Parameter No.	Name	Brief description
AXIS SETTING	SPINDLE	3716#0	A/S	(Not used)
		3717		
	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 0.
		982		Sets 0.
		3017		Output time of reset signal
		3030		Allowable number of digits of an M code
		3716#0	A/Ss	(Not used)
		3717		

12.4.7 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 Item, “**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 Item, “**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 Item, “**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, “**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 Item, “**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 MAINT].
- 3 Press soft key [PERIOD MAINT] 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. 12.4.7 (a) Status screen

- 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 2-byte character codes must conform to FANUC codes. (See Appendix, "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 Item, "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.

The screenshot displays the machine's setting screen with the following sections:

- ACTUAL POSITION:** 00000 N00000
- ABSOLUTE:** X 0.000, Y 0.000, T 10, C 0.000
- MODAL:** G00 G50 G13.1F M, G17 G67 G50.1H, G90 G54 D, G22 G64 T 10, G94 G85 S, G21 G15, G40 G40.1
- PERIODICAL MAINTENANCE (SETTING):**

	LIFE	REMAIN	COUNT TYPE
01	100 H	50 H	ALL
02	1000 H	1000 H	POWER ON
03	720 H	720 H	RUNNING
04	1000 H	0 H	CUTTING
05			
06			
07			
08			
09			
10			
- MDI:** 12:00:00
- Buttons:** < ABSOLUTE RELATIVE ALL HANDLE CHANGE TYPE ERASE +INPUT INPUT +

Fig. 12.4.7 (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 horizontal 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.

The screenshot displays the Machine menu screen with the following sections:

- ACTUAL POSITION:** 00000 N00000
- ABSOLUTE:** X 0.000, Y 0.000, T 10, C 0.000
- MODAL:** G00 G50 G13.1F M, G17 G67 G50.1H, G90 G54 D, G22 G64 T 10, G94 G85 S, G21 G15, G40 G40.1
- PERIODICAL MAINTENANCE (MACHIN):**
 - ITEM NAME
 - 01 TOUCH PANEL
 - 02
 - 03
 - 04
 - 05
 - 06
 - 07
 - 08
 - 09
 - 10
- Bottom Status Bar:** SACT 0/MIN, MDI ****, 12:00:00, STATUS, MACHIN, NC, <OPRT>

Fig. 12.4.7 (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

An item name can be registered on the menu screen of the machine by typing the item name in the format shown below on the machine menu screen, then pressing soft key [INPUT] (or the  key).

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

Two-byte character codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)

When typing 2-byte characters, type an asterisk "*" before and after the character codes. 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.

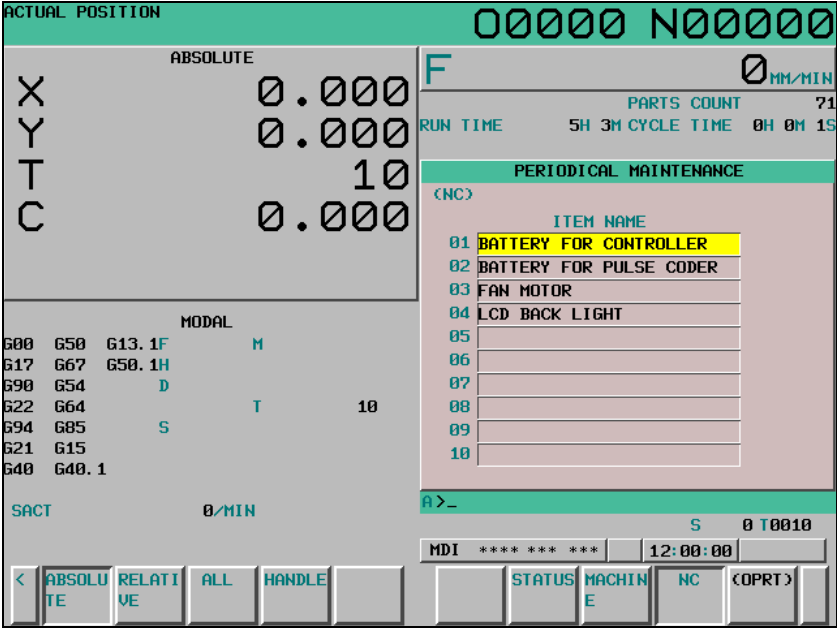


Fig. 12.4.7 (d) NC menu screen

- **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.

12.4.8 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 function 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.

ACTUAL POSITION		00000 N00000																																																									
ABSOLUTE		F 0 MM/MIN																																																									
X	0.000	PARTS COUNT 71																																																									
Y	0.000	RUN TIME 5H 3M CYCLE TIME 0H 0M 1S																																																									
T	10	SYSTEM CONFIG/HARDWARE																																																									
C	0.000	<table border="1"> <thead> <tr> <th>NAME</th> <th>ID-1</th> <th>ID-2</th> <th>SLOT</th> </tr> </thead> <tbody> <tr> <td colspan="4">MAIN BOARD</td> </tr> <tr> <td>MAIN BOARD</td> <td>0042A 80 0</td> <td>70000203</td> <td></td> </tr> <tr> <td>SERVO CARD</td> <td>0014F 30 0</td> <td></td> <td></td> </tr> <tr> <td>FROM/GRAM</td> <td>C3/04</td> <td></td> <td></td> </tr> <tr> <td colspan="4">DISPLAY</td> </tr> <tr> <td>DISP ID</td> <td>1010</td> <td></td> <td></td> </tr> <tr> <td colspan="4">OTHERS</td> </tr> <tr> <td>MDI ID</td> <td>18</td> <td></td> <td></td> </tr> <tr> <td>POWER SUPPLY</td> <td>01</td> <td></td> <td></td> </tr> <tr> <td colspan="4">CERTIFY ID</td> </tr> <tr> <td>ID DATA-1</td> <td>0216067F</td> <td></td> <td></td> </tr> <tr> <td>ID DATA-2</td> <td>E2A40F4D</td> <td></td> <td></td> </tr> <tr> <td>ID DATA-3</td> <td>4E60604E</td> <td></td> <td></td> </tr> </tbody> </table>		NAME	ID-1	ID-2	SLOT	MAIN BOARD				MAIN BOARD	0042A 80 0	70000203		SERVO CARD	0014F 30 0			FROM/GRAM	C3/04			DISPLAY				DISP ID	1010			OTHERS				MDI ID	18			POWER SUPPLY	01			CERTIFY ID				ID DATA-1	0216067F			ID DATA-2	E2A40F4D			ID DATA-3	4E60604E		
NAME	ID-1	ID-2	SLOT																																																								
MAIN BOARD																																																											
MAIN BOARD	0042A 80 0	70000203																																																									
SERVO CARD	0014F 30 0																																																										
FROM/GRAM	C3/04																																																										
DISPLAY																																																											
DISP ID	1010																																																										
OTHERS																																																											
MDI ID	18																																																										
POWER SUPPLY	01																																																										
CERTIFY ID																																																											
ID DATA-1	0216067F																																																										
ID DATA-2	E2A40F4D																																																										
ID DATA-3	4E60604E																																																										
MODAL		A>_ S 0 T0010																																																									
G00 G50 G13.1F	M	MDI ***** 12:00:00																																																									
G17 G67 G50.1H		SYSTEM SERVO SPINDL																																																									
G90 G54 D	T 10	INFO INFO																																																									
G22 G64		OPRT																																																									
G94 G85 S																																																											
G21 G15																																																											
G40 G40.1																																																											
SACT 0/MIN																																																											

Fig. 12.4.8 (a) System configuration screen

Hardware configuration screen

This screen shows the names and IDs of the hardware used by the NC.

SYSTEM CONFIG/HARDWARE			
NAME	ID-1	ID-2	SLOT
MAIN BOARD			
MAIN BOARD	0042B 80 0	70000203	
SERVO CARD	0014A 10 0		
FROM/GRAM	C3/04		
OPTION BOARD			
HSSB 1CH	00611 20 1		2
DISPLAY			
DISP ID	1010		
OTHERS			
MDI ID	F2		
POWER SUPPLY	10		
CERTIFY ID			
ID DATA-1	01718EA6		

Fig. 12.4.8 (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. 12.4.8 (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 amplifiers) 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. 12.4.8 (d) Servo information screen

12.4.9 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.

12.4.9.1 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 (HMOV) 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.

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 HISTORY] ([OPEHIS] for the 8.4-inch display unit) is displayed.
- 3 Press soft key [OPERAT HISTORY], then press newly displayed soft key [OPERAT HISTORY]. 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.

OPERATION HISTORY			
00000 N00000			
No. 0050 / 0127			
No. DATA	No. DATA	No. DATA	No. DATA
50 8	2008/06/18	2008/06/18	88 <CUR ↓>
51 1	13:58:20	13:58:53	89 0
52 3	66 0	74 <CANCEL>	90 <INPUT>
53 0	67 <INPUT>	75 <CANCEL>	91 PW0000
54 [SOFT 11	68 2008/06/18	76 <CANCEL>	2008/06/18
55 <CUR ↓>	13:58:23	77 <CANCEL>	14:00:23
56 <CUR →>	69 2008/06/18	78 <RESET>	92 <PROG>
57 <CUR →>	13:58:40	79 <SYSTEM>	93 2008/06/18
58 <CUR →>	70 SW0100	80 [RIGHT F]	14:00:27
59 <CUR →>	2008/06/18	81 [RIGHT F]	94 2008/06/18
60 <CUR →>	13:58:46	82 [SOFT 9]	14:00:44
61 <CUR →>	71 SW0100	83 8	95 SW0100
62 <CUR →>	2008/06/18	84 1	2008/06/18
63 1	13:58:47	85 3	14:00:56
64 <INPUT>	72 <RESET>	86 0	
65 PW0000	73 SW0100	87 [SOFT 1]	

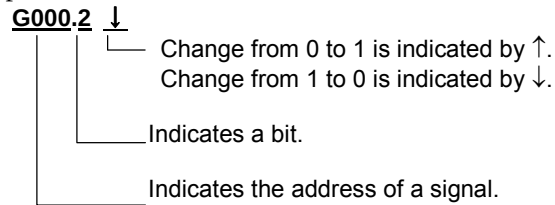
Fig. 12.4.9.1 (a) Operation history screen

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

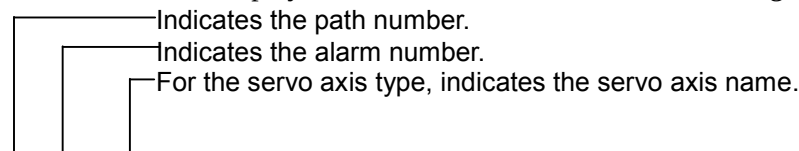
When bit 6 (HDE) of parameter No. 3195 is set to 0, I/O signals specified on the operation history signal selection screen are recorded. Recorded signals are indicated on a bit-by-bit basis with information about the signal address and a change in bit. These signals are displayed with purple characters.

**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.



1_OT0506 (XA1)

2008/01/11
11:22:33

Indicates the date and time of issuance in two lines.

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 (HMOV) 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.

12.4.9.2 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 .
- 2 Press continuous menu key  several times until soft key [OPERAT HISTORY] is displayed.
- 3 Press soft key [OPERAT HISTORY].
- 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 .

Example:

When G0004 is entered then  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].

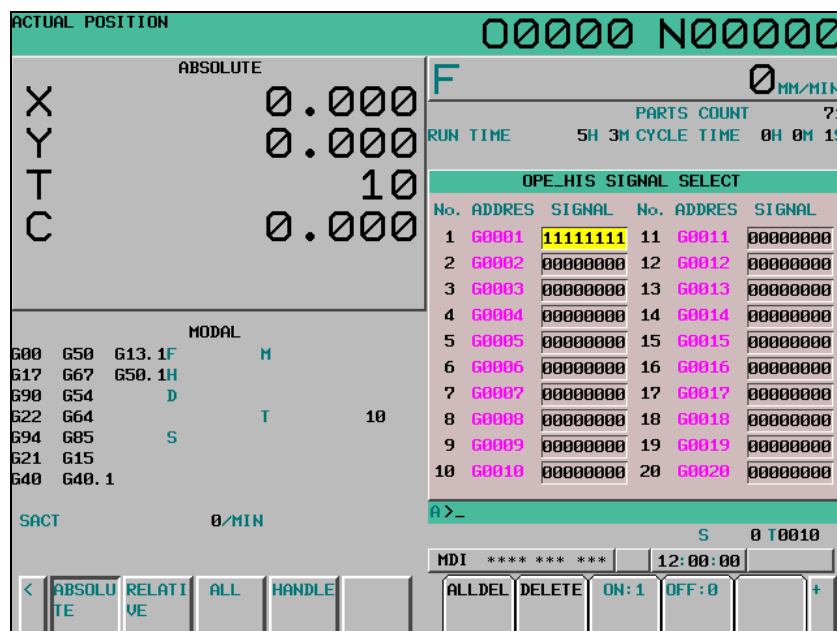


Fig. 12.4.9.2 (a) Operation history signal selection screen

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.

12.4.9.3 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 HISTORY] ([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 G17 G90 G22 G94 G21 G40 G50 G67 G54 B0. D0. E0. *F100. H0. M10. *N123. O123 S0. T10. X ABS 197.999 MCN 197.999 Y ABS -199806.00 MCN -199806.00 C ABS 297.009 MCN 0.123 at 2007/09/01 19:03:28
 - Alarm 02_OT00506(Y)
*G0 G17 G90 G22 G94 G21 G40 G50 G67 G54 B0. *D12. E0. *F100. H0. M0. *N123. O123 S0. T0. X ABS 123.999 MCN 234.000 Y ABS -123.00 MCN -234.00 C 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(X) 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 G17 G90 G22 G94 G21 G40 G50 G67 G54 B0. D0. E0. F0. H0. M0. N0. O9999 S0. T0.

- X ABS 10.000 MCN 0.000 Y ABS 123.000 MCN 0.000 C 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_", "offset-number", "offset-data-before-modification", "offset-data-after-modification", and "time-of-modification" are output in this order.
<Example>
Tool Offset 01_0032 12.340 → 12.569 at 11:39:42
 - 6 Modification of workpiece offset/workpiece shift (T series) data
After "Work Offset", "path-number_", "type(axis-name)", "offset-number", "offset-data-before-modification", "offset-data-after-modification", and "time-of-modification" are output in this order.
<Example>
Work Offset 01_G55(X) 15.000 → 0.007 at 09:23:03
Work Offset 01_EXT(Y) 0.000 → 300.003 at 09:22:50
 - 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.
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 L01 0.000 → 1000.000 at 18:58:48
Parameter Axis type N01423 A04(B) 0.000 → 10000.000 at 18:58:48
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 $M \times (10^{**}(-E))$.
<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)

12.5 SCREENS DISPLAYED BY FUNCTION KEY

By pressing the function key , data such as alarms, alarm history, and operator messages can be displayed.

For details, see Sections, “ALARM DISPLAY” and “ALARM HISTORY DISPLAY”.

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.

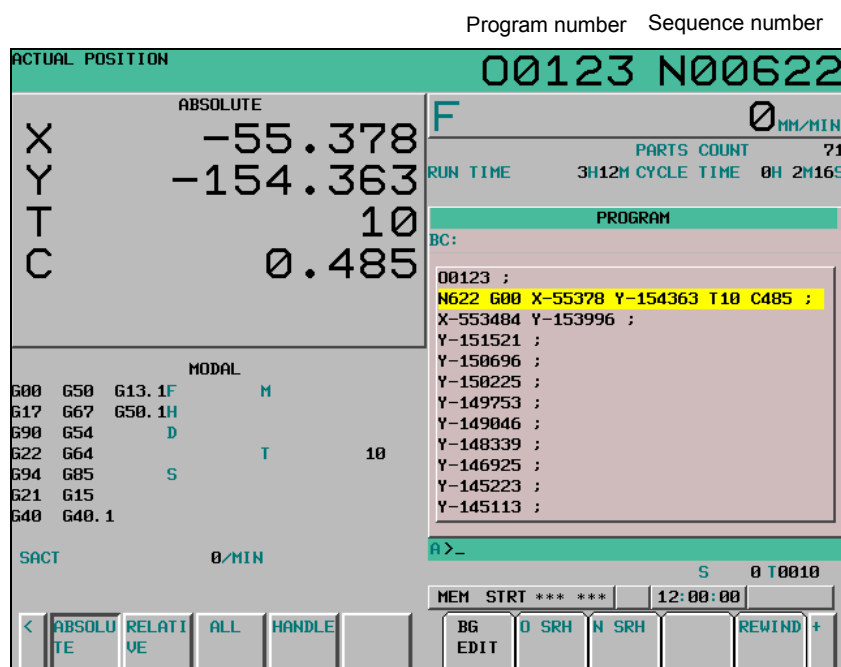


Fig. 12.6.1 (a) Program number and sequence number (10.4-inch display unit)

In the EDIT mode, the number of the program currently edited in the foreground is 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 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 second line from the bottom of the screen. This prevents invalid data setting and input/output errors.

Explanation

- Description of each display

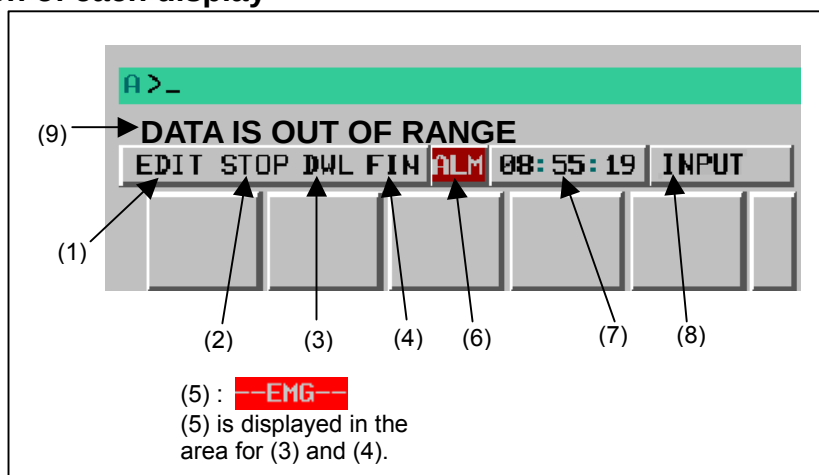


Fig. 12.6.2 (a) Positions of status indications

(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.)
- Space : Indicates a state other than the above.

NOTE

When the FAN in the αi servo amplifier or the αi 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 αi 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 αi 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 αi 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)".

(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.
- 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



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  key and any function key (such as  or ) 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 $\neq$ 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 key + function key

When a non-zero value is set in parameter No. 3123, the screen is not erased with the  key and a function key.

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.

13 GRAPHIC FUNCTION

Chapter 13, "GRAPHIC FUNCTION", consists of the following sections:

13.1 OPERATION	639
13.2 REGISTERING THE TOOL FIGURE.....	639
13.3 SPECIFYING DRAWING PARAMETERS	640
13.4 GRAPHIC DISPLAY SCREEN AND DRAWING	644
13.5 EXAMPLE.....	647

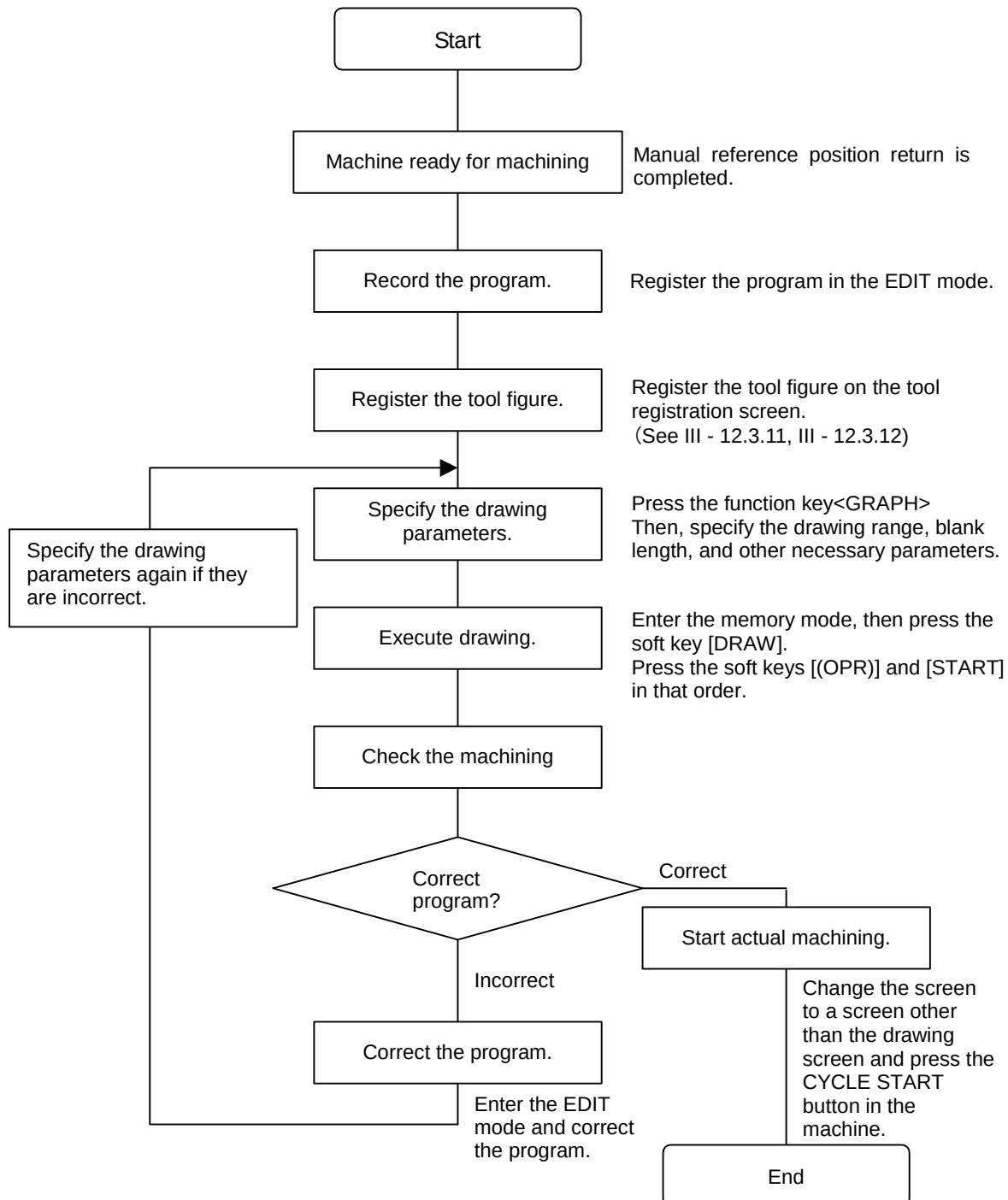
When programming is completed, the graphic function can be used to check whether machining will be performed as desired without operating the machine by drawing the programmed tool path and machining profile on the graphic display screen. After checking the profile, perform machining.

The function has the following features to simplify locating an error in the machining program:

- (i) Can draw a blank figure (workpiece).
- (ii) Can draw the figure of a tool used at the position where punching is performed.
A circle, a rectangle, and a capsule can be used for drawing the tool figure. The figures of other tools are drawn using asterisks.
- (iii) Can draw a workpiece holder that holds a blank.
The operator can check whether punching is performed at the position of the workpiece holder. This is used to ensure that the workpiece holder is not punched during machining.
- (iv) Can draw a tool path.
The tool path is drawn using a dashed line for rapid traverse and a solid line for cutting feed.
- (v) Can specify a drawing plane.
A drawing plane can be selected from four types. Drawing can be performed as if a blank is set in the machine.
- (vi) Checks the format of the program.
The format of the program, stored stroke limit 1, and the safety zone are checked during drawing.

13.1 OPERATION

The following flowchart shows an example of drawing a programmed figure on the screen. Refer to the flowchart if you forget the procedure.



13.2 REGISTERING THE TOOL FIGURE

To draw a machining profile, register the dimensions of the tool on the tool figure registration screen.
(See III-12.3.11.4, III-12.3.12.2.)

13.3 SPECIFYING DRAWING PARAMETERS

Specify the parameters for graphic drawing.

Explanation

Procedure

- 1 Press the function key . The graphic parameter setting screen appears. If it does not appear, press the soft key [PARAMETER].

Screen

ACTUAL POSITION				00123 N00000			
ABSOLUTE				F 0 MM/MIN			
X		0.000		PARTS COUNT 9			
Y		0.000		RUN TIME 0H 0M CYCLE TIME 0H 0M 1S			
T		10		GRAPHIC PARAMETER			
C		0.000		GRAPHIC COORDINATES P 0			
				RANGE (MAX)			
				X 1150.000			
				Y 700.000			
				RANGE (MIN)			
				X -100.000			
				Y -100.000			
				SCALE			
				K 0.14			
				GRAPHIC CENTER			
				X 525.000			
				Y 300.000			
				PROGRAM STOP			
				N 0			
				AUTO ERASE			
				A 1			
MODAL				A>_			
G00 G50 G13.1F 750M				S 0 T0010			
G17 G67 G50.1H				MEM **** * 12:00:00			
G90 G54 D				PARAMETER GRAPHIC			
G22 G64 T 10				COPRT			
G94 G85 S							
G21 G15							
G40 G40.1							
SACT 0/MIN							
ABSOLUTE							
RELATIVE							
ALL							
HANDLE							

ACTUAL POSITION				00123 N00000			
ABSOLUTE				F 0 MM/MIN			
X		0.000		PARTS COUNT 9			
Y		0.000		RUN TIME 0H 0M CYCLE TIME 0H 0M 1S			
T		10		GRAPHIC PARAMETER			
C		0.000		START POINT			
				X 0.000			
				Y 0.000			
				WORK LENGTH			
				X 1100.000			
				Y 650.000			
				RAPID PATH			
				P 1			
				(1:ON 0:OFF)			
MODAL				A>_			
G00 G50 G13.1F 750M				S 0 T0010			
G17 G67 G50.1H				MEM **** * 12:00:00			
G90 G54 D				PARAMETER GRAPHIC			
G22 G64 T 10				COPRT			
G94 G85 S							
G21 G15							
G40 G40.1							
SACT 0/MIN							
ABSOLUTE							
RELATIVE							
ALL							
HANDLE							

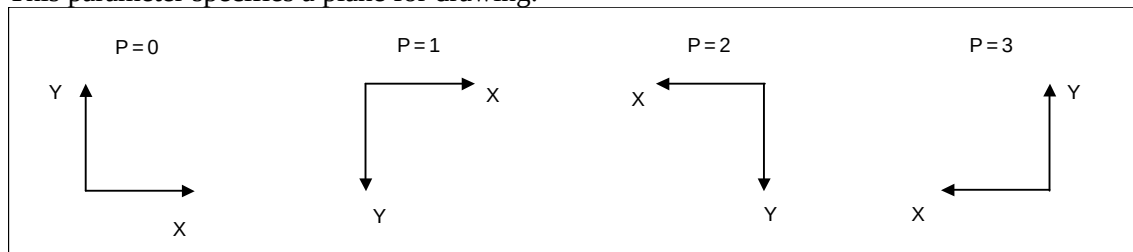
ACTUAL POSITION		00123 N00000	
ABSOLUTE		F 0 MM/MIN	
X	0.000	PARTS COUNT 9	
Y	0.000	RUN TIME 0H 0M CYCLE TIME 0H 0M 1S	
T	10	GRAPHIC PARAMETER	
C	0.000	HOLDER 1 (POSITION) X1 300.000	
		(SIZE) X 40.000	
		Y 10.000	
		HOLDER 2 (POSITION) X2 700.000	
		(SIZE) X 40.000	
		Y 10.000	
		HOLDER 3 (POSITION) X3 0.000	
		(SIZE) X 0.000	
		Y 0.000	
		HOLDER 4 (POSITION) X4 0.000	
		(SIZE) X 0.000	
		Y 0.000	
MODAL		A>_ S 0 T0010	
G00 G50 G13.1F 750M		MEM **** * 12:00:00	
G17 G67 G50.1H		PARAMETER GRAPHIC	
G90 G54 D		COPRT	
G22 G64 T 10			
G94 G85 S			
G21 G15			
G40 G40.1			
SACT 0/MIN			
ABSOLUTE RELATIVE ALL HANDLE			

2 Move the cursor to the parameter to be changed by pressing the cursor keys.

3 Enter new data and press the  key.

(a) Drawing plane

This parameter specifies a plane for drawing.



(b) Drawing range (maximum and minimum values)

This parameter specifies the desired drawing range for each axis using the maximum and minimum values. Specifying these values automatically determines the magnification for a screen so that the drawing can be made using the whole drawing range. The center of the drawing range matches that of the screen during drawing.

(c) Magnification

This parameter specifies the screen magnification. The magnification refers to the magnification to the center of the screen.

Specifying the drawing range in (b) automatically determines the effective magnification. To magnify part of a figure, specify another magnification based on the automatically specified value.

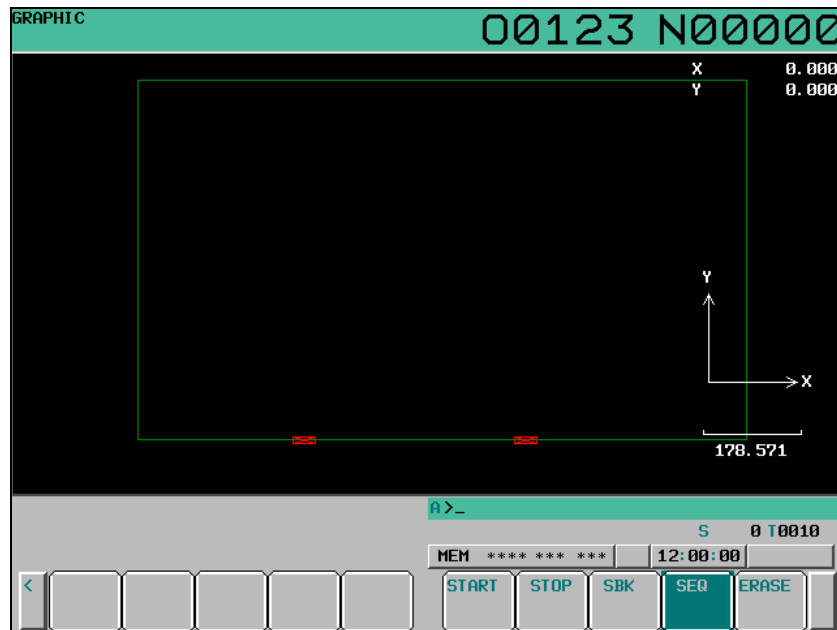
(d) Coordinates based on the center of a screen

This parameter specifies the coordinates in the workpiece coordinate system whose origin matches the center of the screen.

Specifying the drawing range in (b) automatically determines the coordinates. To magnify part of the figure, specify other coordinates based on the automatically specified coordinates.

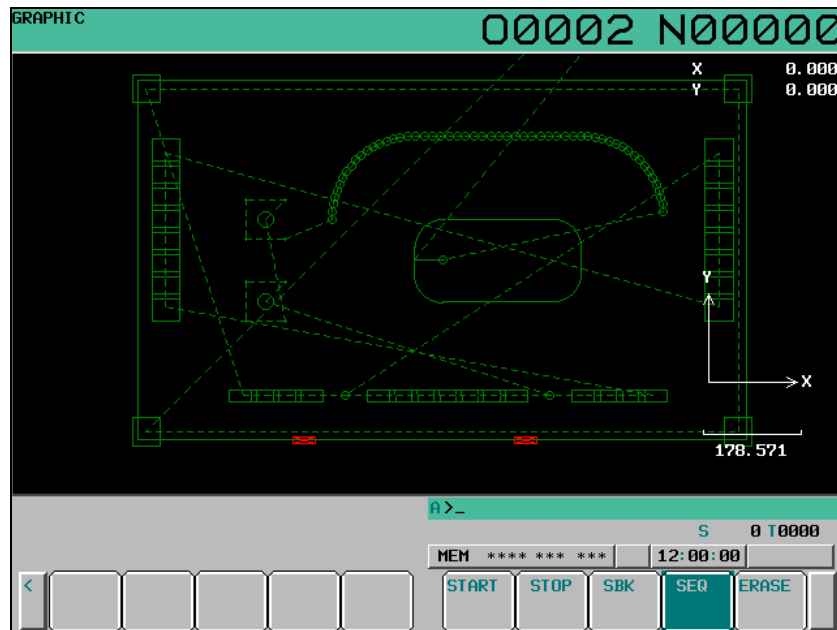
- (e) Drawing end block
To display part of the programmed figure, specify the sequence number of the end block. Once the figure is drawn, the sequence number is automatically canceled.
- (f) Automatic deletion
1 : Previously drawn figures are automatically deleted when automatic operation is started in the reset state.
0 : Previously drawn figures are not automatically deleted.
- (g) Drawing start position
When a coordinate system command, G92, is not specified in a drawing program, this parameter specifies the drawing start position using the coordinates in the workpiece coordinate system.
- (h) Blank length
This parameter specifies the blank length along the X-axis and Y-axis.
Setting this parameter draws a blank figure when the drawing screen is selected or when the drawing soft key [ERASE] is pressed.
The end of the blank figure in the drawing plane described in (a) is used as the origin of the workpiece coordinate system.

Screen



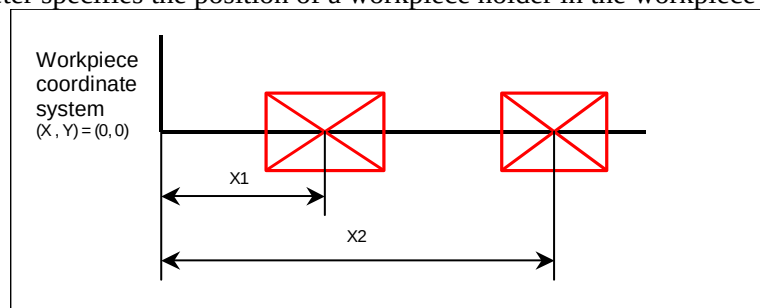
- (i) Rapid traverse
1 : A tool path for rapid traverse is drawn as a dotted line.
0 : No tool path for a rapid traverse is drawn.

Screen



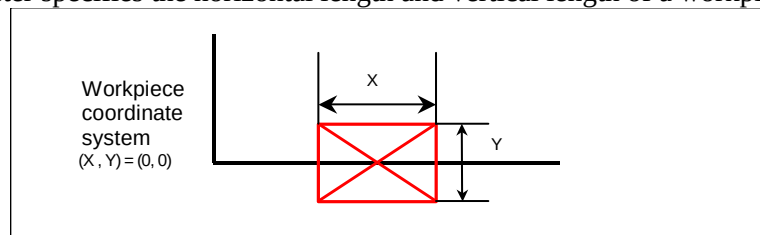
(j) Position of a workpiece holder

This parameter specifies the position of a workpiece holder in the workpiece coordinate system.



(k) Length of a workpiece holder

This parameter specifies the horizontal length and vertical length of a workpiece holder.

**NOTE**

When a safety zone check function is provided, data items (j) and (k) for the workpiece holder cannot be specified. Specifying the safety zone displays the figure of the workpiece holder (That is set for #1 to #4.).

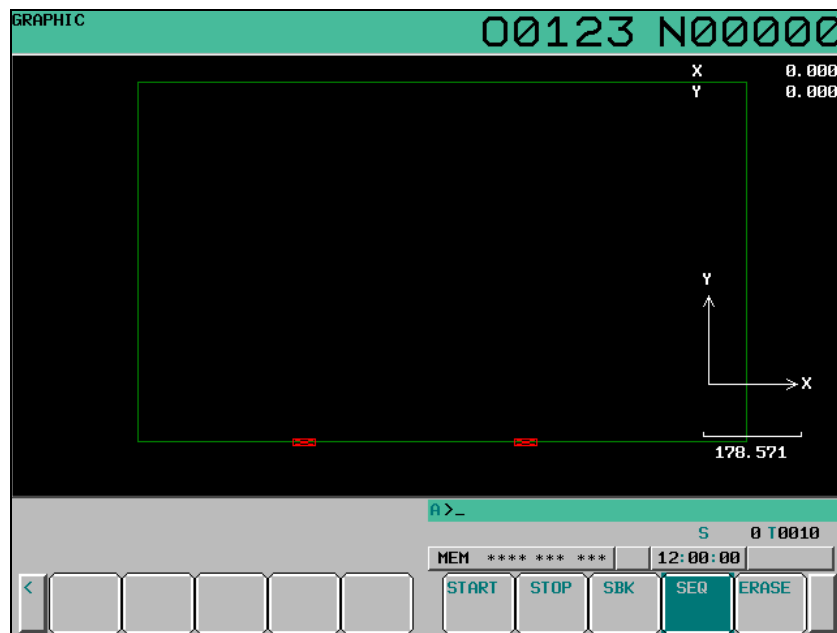
13.4 GRAPHIC DISPLAY SCREEN AND DRAWING

Explanation

Drawing screen

Press [GRAPH] key after pressing  key, the following graphic display screen appears.

Screen

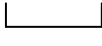


By selecting this screen, tool paths and holes figures are drawable for checking NC programs.

Software keys, status display for drawing, and ruler

- (1) Soft keys are used for drawing.
 - [START]key : Drawing start key
 - [STOP]key : Drawing stop key
 - [ERASE]key : Graphic display erasing key
 - [SEQ]key : Continuous drawing mode key
 - [SBK] key: Single block drawing mode key

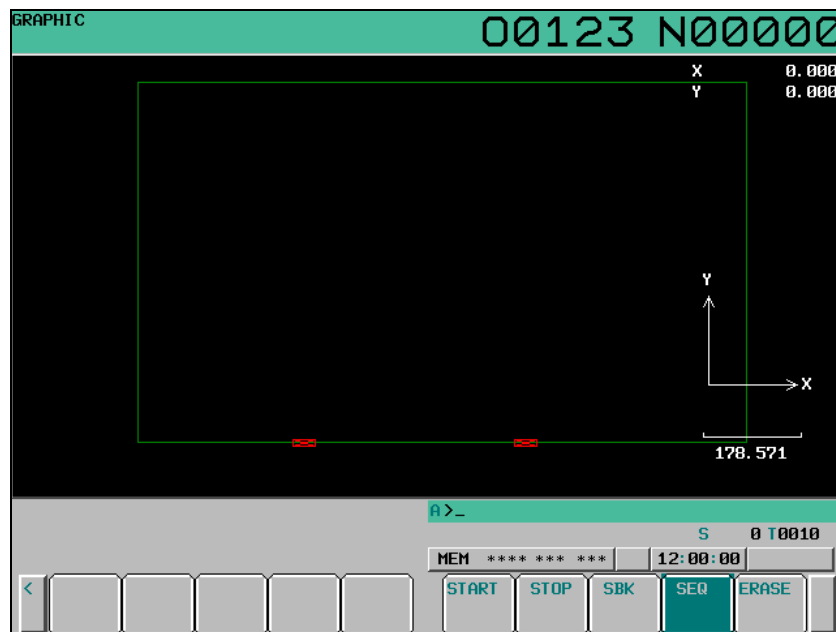
- (2) Ruler

The ruler is displayed at the lower right part of the screen. The value below  shows the length to the line (Input unit).

Drawing operation

- (1) Select either MEMORY mode.
(This is the preparation for automatic operation, such as heading of a program, etc.)
- (2) Depress [ERASE] key.
Depress this key when the previous graphic display is not required.
After erasing graphic display, the work sheet figure and work holder mark are drawn as shown in the left figure.

Screen

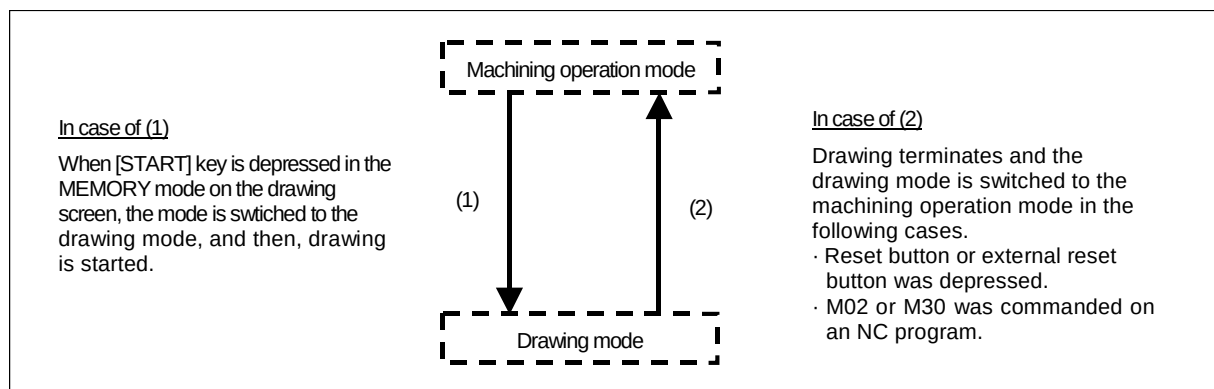


- (3) Depress [SEQ] and [START] keys (continuous drawing).
Drawing is started and continued up to the end of the NC program.
- (4) Depress [SBK] and [*SEQ] keys (Single block drawing).
Operation stops after drawing one block.
Drawing is done every block each time [START] key is depressed hereafter.
- (5) Depress [STOP] key
The system enters the feed hold state.
Pressing the [STOP] key during continuous drawing changes the operation to single-block drawing, causing the system to enter the feed hold state.
To perform continuous drawing again, press the [SEQ] key, then press the [START] key.

Relation between drawing mode and machining operation mode

The drawing mode means a condition which is not the drawing end condition (that is, drawing in progress or single block drawing stop condition), while the machining operation mode means the drawing end condition.

The relation between the drawing mode and the machining operation mode is illustrated below.



Since drawing is done under such a condition as MACHINE LOCK, the modal information, absolute coordinate value, etc. are updated.

When the mode is switched from the machining operation mode to the drawing mode, the following information is stored.

- 1) Relative coordinate value (RELATIVE)
- 2) Work coordinate system value (ABSOLUTE)
- 3) Machine coordinate system value (MACHINE)
- 4) T code
- 5) Tool position offset
- 6) Repositioning coordinate

After drawing, the stored information is restored as before, and the mode is switched to the machining operation mode after heading of the drawn program.

NOTE

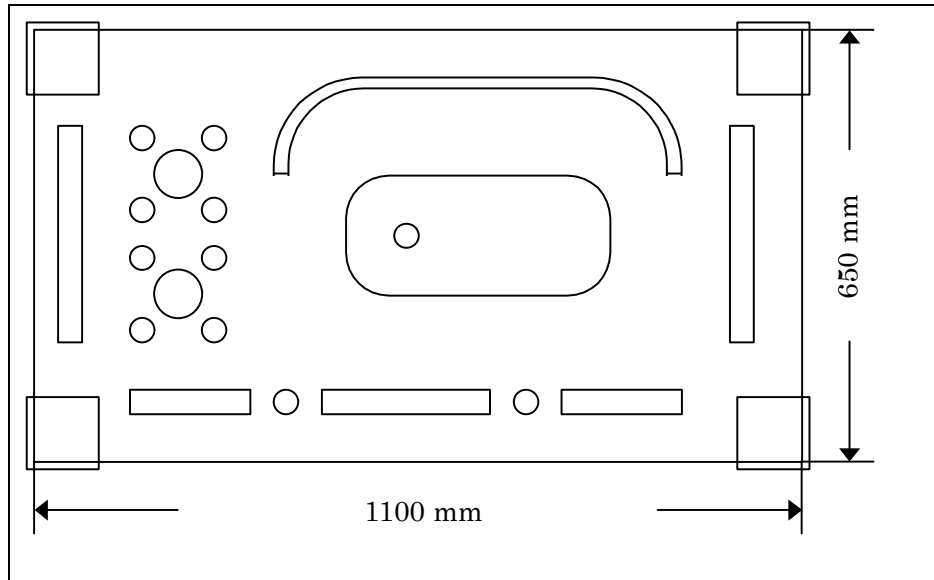
- 1 The dwell (G04) is not executed.
- 2 M, S, T codes signals and the press start signal are not output to the machine.

NOTE

- 1 Output signals OP, STL, SPL, AL are sent to the machine tool even during drawing in the same way as in normal machining operation.
- 2 Input signals ERS, *SP, *ESP, ST, etc. of the machine tool are effective even during drawing. Don't operate these switches, accordingly. (The single block switch SBK is ineffective.)
- 3 If manual reference point return function is provided, the stored stroke limit 1 is checked even during drawing. Thus, the machine tool position must coincide with actual machining start position before drawing start.
- 4 The tool profile is drawn at the end point in a block where positioning is done even in the punch off mode using M function or under the turned-on condition of the punch off switch on the machine tool side.
- 5 If the tool profile code of the designated T code is 00, mark (*) is drawn.
- 6 Machining operation is not executed in the drawing mode. Drawing is not attempted halfway during machining operation.
- 7 It is impossible to draw a helically interpolated figure.

13.5 EXAMPLE

Set drawing parameters for drawing the following NC program as follows.



:O00002:(NC program)

G92X1270.Y1270.;	G76I40.J90.K8T02 ;	G72X800.Y400. ;
G90G00X15.Y15.T02;	G72X1050.Y200. ;	G77I150.J90.P-5.K18 ;
X1085.Y15. ;	G76I40.J90.K8 ;	G90X550.Y325.T04 ;
X1085.Y635. ;	G90X375.Y80.T04 ;	M24 ;
X15.Y635. ;	X745.Y80.;	G01X500.Y325.F3000 ;
G72X150.Y80.;	X230.Y250.T05 ;	X500.Y300. ;
G76I40.J0K4T03 ;	G26I50.J45.K4T06 ;	G03X550.Y250.I50. ;
G72X400.Y80. ;	X230.Y400.T05 ;	G01X750.Y250. ;
G76I40.J0K7;	G26I50.J45.K4T06 ;	G03X800.Y300.J50. ;
G72X770.Y80. ;	G72X500.Y400. ;	G01X800.Y350. ;
G76I40.J0K4 ;	G77I150.J180.P-5.K18T04 ;	G03X750.Y400.I-50. ;
G72X50.Y200. ; ↗	G76I15.J0K20 ; ↗	G01X550.Y400. ;
		G03X500.Y350.J-50. ;
		G01X500.Y325. ;
		M25;
		G00G90X1270.Y1270.M02 ;

Screen

ACTUAL POSITION				00123 N00000			
ABSOLUTE X 0.000 Y 0.000 T 10 C 0.000				F 0 MM/MIN PARTS COUNT 9 RUN TIME 0H 0M CYCLE TIME 0H 0M 1S			
MODAL G00 G50 G13.1F 750M G17 G67 G50.1H G90 G54 D G22 G64 T 10 G94 G05 S G21 G15 G40 G40.1				GRAPHIC PARAMETER GRAPHIC COORDINATES P 0 RANGE (MAX) X 1150.000 Y 700.000 RANGE (MIN) X -100.000 Y -100.000 SCALE K 0.14 GRAPHIC CENTER X 525.000 Y 300.000 PROGRAM STOP N 0 AUTO ERASE A 1			
SACT 0/MIN				A>_ S 0 T0010			
MEM **** *				12:00:00			
ABSOLUTE RELATIVE ALL HANDLE				PARAMETER GRAPHIC (OPRT)			

ACTUAL POSITION				00123 N00000			
ABSOLUTE X 0.000 Y 0.000 T 10 C 0.000				F 0 MM/MIN PARTS COUNT 9 RUN TIME 0H 0M CYCLE TIME 0H 0M 1S			
MODAL G00 G50 G13.1F 750M G17 G67 G50.1H G90 G54 D G22 G64 T 10 G94 G05 S G21 G15 G40 G40.1				GRAPHIC PARAMETER START POINT X 0.000 Y 0.000 WORK LENGTH X 1100.000 Y 650.000 RAPID PATH (1:ON 0:OFF) P 1			
SACT 0/MIN				A>_ S 0 T0010			
MEM **** *				12:00:00			
ABSOLUTE RELATIVE ALL HANDLE				PARAMETER GRAPHIC (OPRT)			

ACTUAL POSITION				00123 N00000	
ABSOLUTE				F	0 MM/MIN
X	0.000			PARTS COUNT 9	
Y	0.000			RUN TIME 0H 0M CYCLE TIME 0H 0M 1S	
T	10			GRAPHIC PARAMETER	
C	0.000			HOLDER 1 (POSITION) X1 300.000	
				(SIZE) X 40.000	
				Y 10.000	
				HOLDER 2 (POSITION) X2 700.000	
				(SIZE) X 40.000	
				Y 10.000	
				HOLDER 3 (POSITION) X3 0.000	
				(SIZE) X 0.000	
				Y 0.000	
				HOLDER 4 (POSITION) X4 0.000	
				(SIZE) X 0.000	
				Y 0.000	
MODAL				A>_	
G00	G50	G13.1F	750M	S 0 T0010	
G17	G67	G50.1H		MEM **** * 12:00:00	
G90	G54	D		PARAMETER GRAPHIC	
G22	G64		T 10	COPT	
G94	G85	S			
G21	G15				
G40	G40.1				
SACT 0/MIN					

Tool figure data

T02 : Rectangle, X dimension = 50 mm, Y dimension = 50 mm

T03 : Rectangle, X dimension = 50 mm, Y dimension = 20 mm

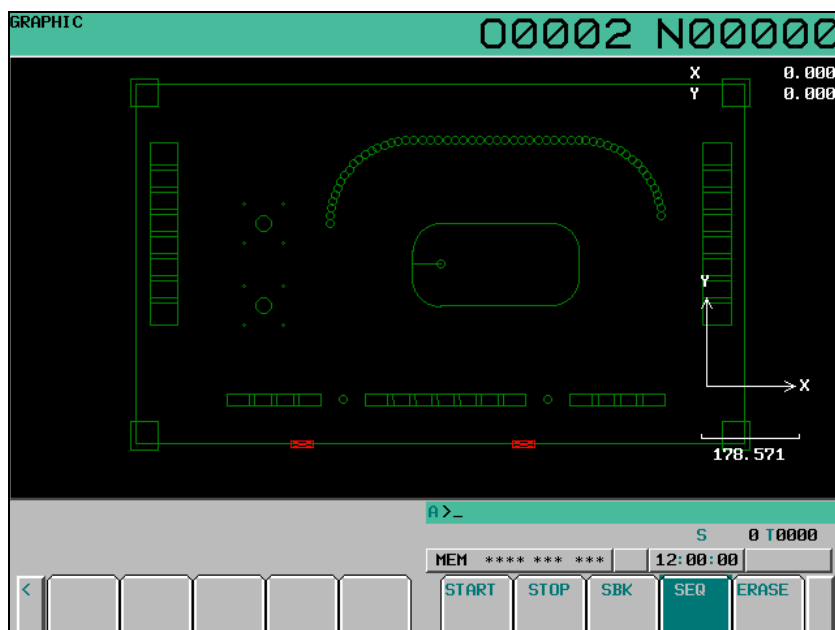
T04 : Circle, diameter = 15.5 mm

T05 : Circle, diameter = 30 mm

T06 : Circle, diameter = 5 mm

The following figures show the programmed figure.

Screen



14 VIRTUAL MDI KEY FUNCTION

Chapter 14, "VIRTUAL MDI KEY FUNCTION", consists of the following sections:

14.1 VIRTUAL MDI KEY	650

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 display unit is displayed in the upper left area of the screen, and the keyboard is displayed in the remaining area.

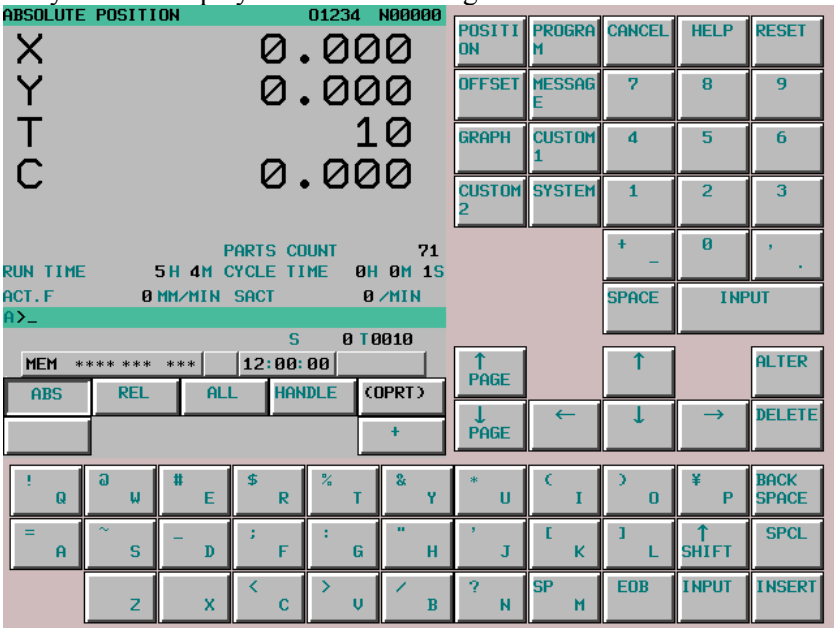


Fig. 14.1 (a) Screen on which a CNC screen is displayed in the upper left area

Operation

- **Input key**
The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.
- **Cancel key**
The displays "BACK SPACE" and "CAN" 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 "CAN" and "RESET" keys to erase alarm PS100, 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" → "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.

Screen on which a CNC screen is displayed in the entire area

Explanation

A CNC screen equivalent to that displayed on 8.4-inch display unit is displayed in the entire area. Function keys and the ON/OFF button for virtual keys are displayed at the bottom of the screen.

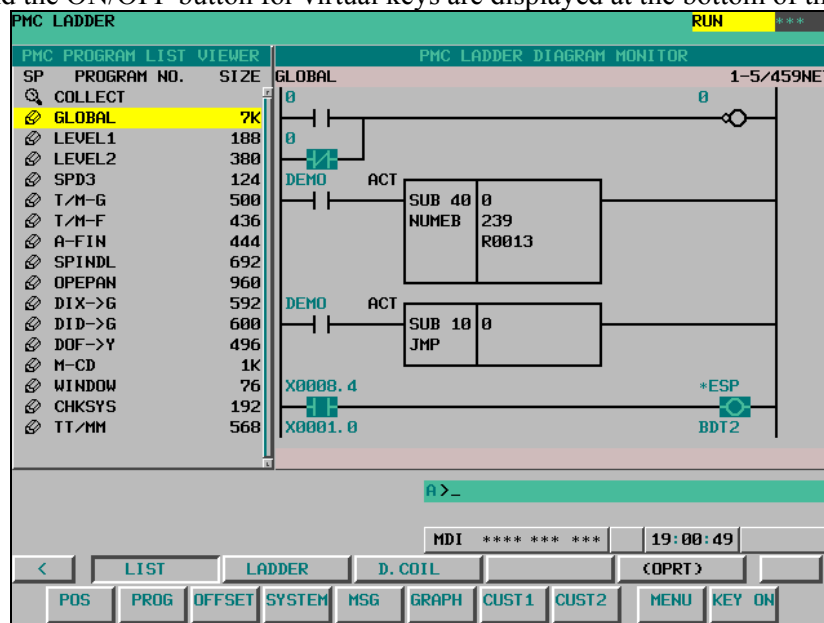


Fig. 14.1 (b) Screen on which a CNC screen is displayed in the entire area

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.

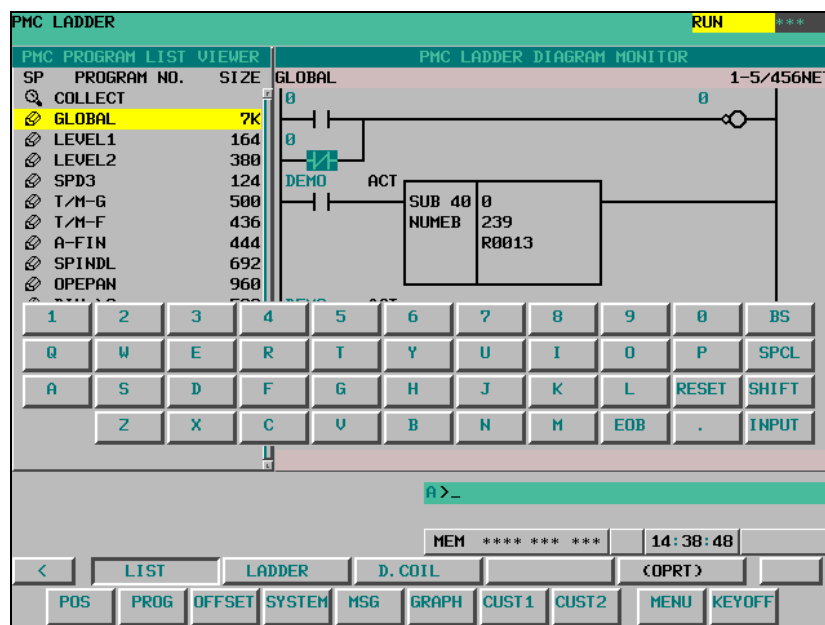


Fig. 14.1 (c) State in which virtual keys are ON

- Input key

The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.

- Cancel key

The displays "BS" and "CAN" 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.

The characters on the key tops change to those that can be entered in the shift state.

1	2	3	4	5	6	7	8	9	0	BS
!	@	#	\$	%	&	*	<	>	¥	SPCL
=	~	-	;	:	"	'	[	]	RESET	SHIFT
	-	+	<	>	/	?	SP	EOB	,	INPUT

Fig. 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 PS100, 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.

- Display unit

Display unit on which this function can be used are 10.4 display unit 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.

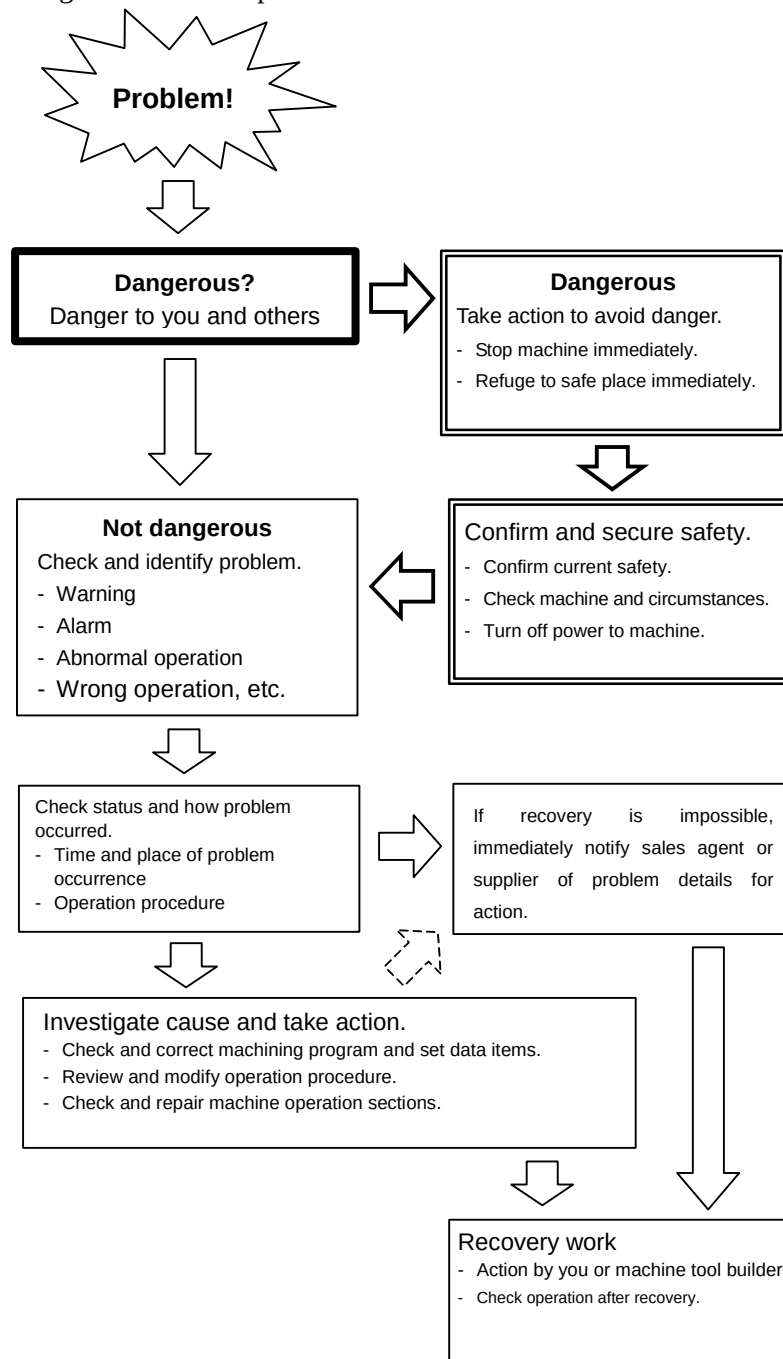
Chapter 1, "ROUTINE MAINTENANCE", consists of the following sections:

1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED	658
1.2 BACKING UP VARIOUS DATA ITEMS.....	659
1.3 METHOD OF REPLACING BATTERY	660

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" 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 Subsection III-8.2.1, "Inputting and Outputting a Program".
- <2> System parameters
 - See Subsection III-8.2.2, "Inputting and Outputting Parameters".
- <3> Tool offset data
 - See Subsection III-8.2.3, "Inputting and Outputting Offset Data".
- <4> PMC data
 - See PMC PROGRAMMING MANUAL (B-64393EN).
- <5> Pitch error compensation data (when the pitch error compensation function is applied.)
 - See Subsection III-8.2.4, "Inputting and Outputting Pitch Error Compensation Data".
- <6> Custom macro variables (when the custom macro function is applied.)
 - See Subsection III-8.2.5, "Inputting and Outputting Custom Macro Common Variables".
- <7> Workpiece coordinate system setting data (when the workpiece coordinate system function is applied.)
 - See Subsection III-8.2.6, "Inputting and Outputting Workpiece Coordinates System Data".
- <8> Operation history data
 - See Subsection III-8.2.7, "Inputting and Outputting Operation History Data".
- <9> Tool management data (when the tool management function is applied.)
 - See Subsection III-8.2.8, "Inputting and Outputting Tool Management Data".

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 control unit backup battery and absolute Pulsecoder battery. This section consists of the following subsections:

1.3.1 Replacing Battery for CNC Control Unit.....	660
1.3.2 Battery in the PANEL i (3 VDC).....	664
1.3.3 Replacing Battery for Absolute Pulsecoders.....	665

1.3.1 Replacing Battery for CNC Control Unit

Battery for memory backup

Such as 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. Therefore, the above data is not lost even if the main power is turned off.

When the battery voltage falls, alarm message "BAT" blinks on the LCD screen 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 one or two weeks of the alarm first being issued. This, however, depends on the system configuration.

If the battery voltage subsequently drops further, backup of memory can no longer be provided. Turning on the power to the control unit in this state causes system alarm to be issued because the contents of memory are lost. Replace the battery, clear the entire memory, then reenter the data.

For this reason, FANUC recommends that the battery be replaced once per 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

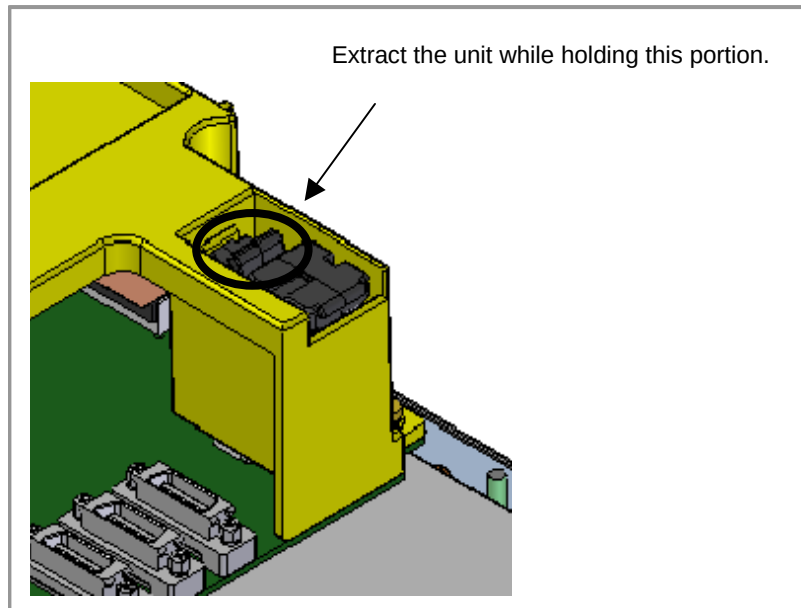
A lithium battery is installed as standard at the factory.

When a lithium battery is used (LCD mounted type)

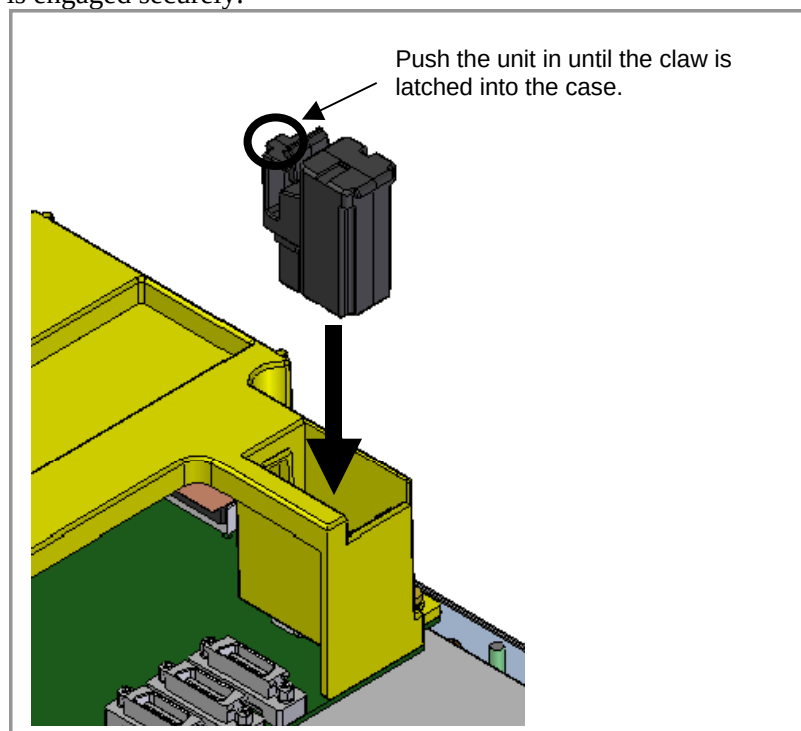
- Replacement procedure

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.

NOTE

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.

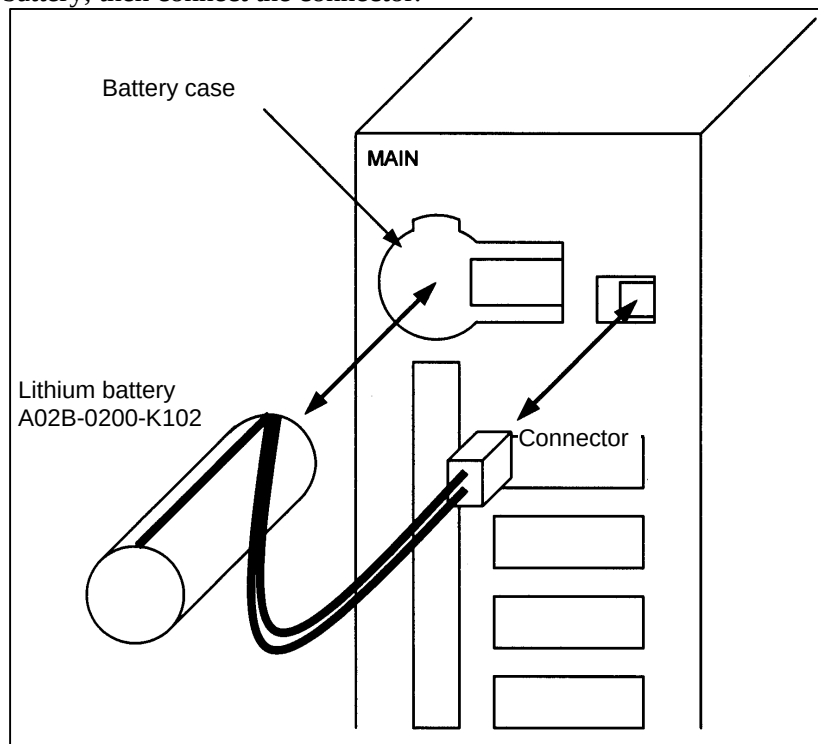
(1) Turn the CNC on. About 30 seconds later, turn the CNC off.

(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 F, "MAINTENANCE OF OPEN CNC (BOOT-UP AND IPL)" of maintenance manual (B-64305EN) for explanations about how to save the contents of the SRAM memory.

NOTE

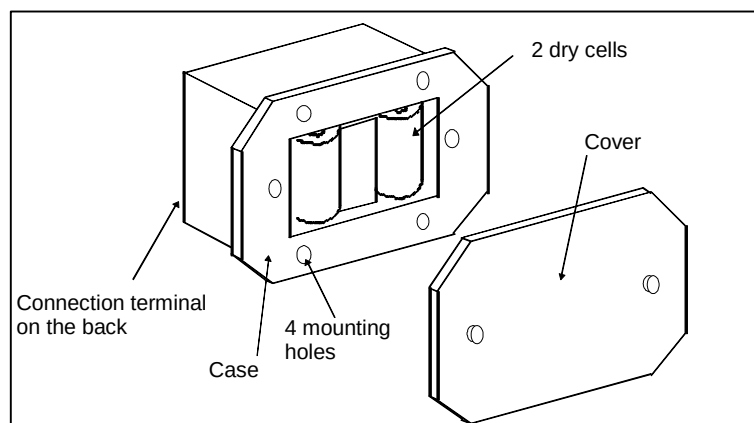
When discarding a battery, observe the applicable ordinances or other rules of your local government. In addition, cover the exposed pins with tape or other insulation materials to prevent a short circuit before discarding the battery.

When alkaline dry 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.

**NOTE**

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.

1.3.2 Battery in the PANEL *i* (3 VDC)

A lithium battery is used to back up BIOS data in the PANEL *i*. This battery is factory-set in the PANEL *i*. This battery has sufficient capacity to retain BIOS data for one year.

When the battery voltage becomes low, the following message appears on the self test screen displayed at turn-on, and the self test pauses.

CMOS Battery Failure

If this message appears, replace the battery as soon as possible (within one week). FANUC recommends that the battery be replaced once per year regardless of whether a battery alarm is issued.

- Replacing the battery

- (1) To guard against the possible loss or destruction of BIOS parameters, write down the BIOS parameter values.
- (2) Obtain a new lithium battery (A02B-0200-K102).
- (3) After power has been supplied for at least five seconds, turn off the power to PANEL *i*. Remove the intelligent terminal from the panel so that replacement work can be done from the rear of the intelligent terminal.
- (4) Detach the connector of the lithium battery, and remove the battery from the battery holder.
- (5) Attach the connector, and place the battery in the battery holder.
- (6) Install PANEL *i* again.
- (7) Turn on the power, and check that the BIOS parameters are maintained (BIOS setup is not activated forcibly).

Between removing an old battery and inserting new battery, no more than five minutes must be allowed to elapse.

NOTE

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.

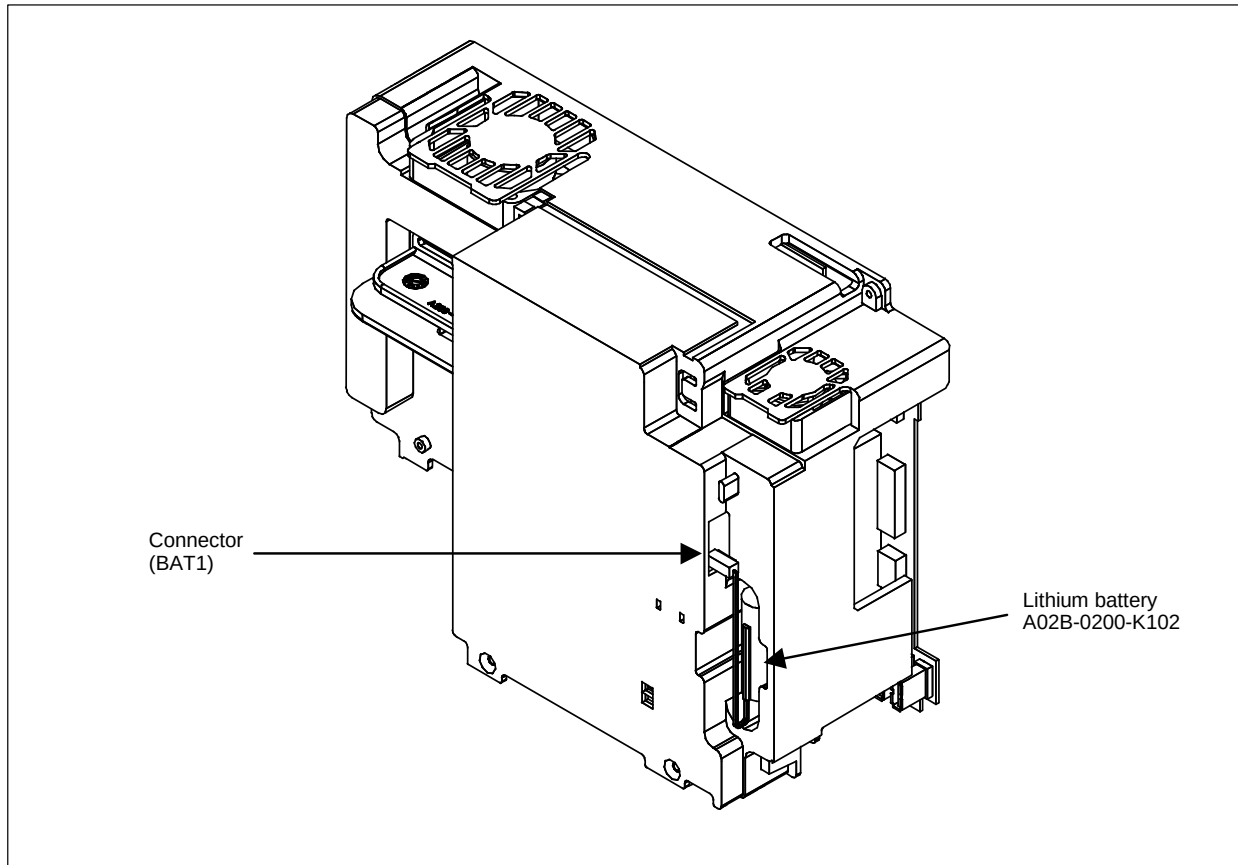


Fig. 1.3.2 (a) Lithium battery connection for PANEL *i*

1.3.3 Replacing Battery for Absolute Pulsecoders

1.3.3.1 Overview

- 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.

1.3.3.2 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 $\alpha i/\alpha i$ 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.

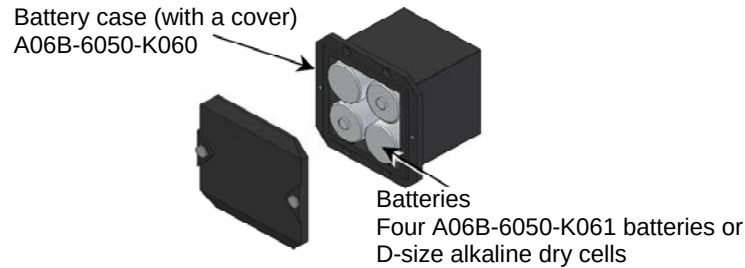
NOTE

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.

1.3.3.3 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.

1.3.3.4 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.

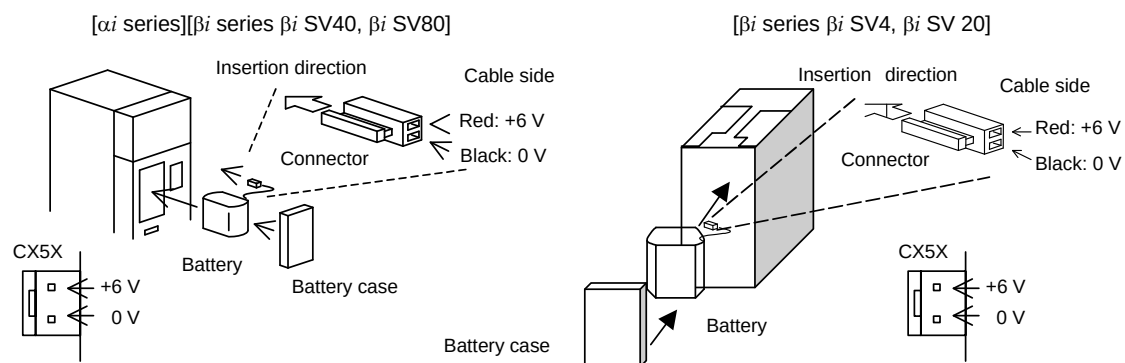
**CAUTION**

- 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 βi SV4 and βi SV20 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.

**[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)	αi series 60/90 mm width	A06B-6114-K505	
		αi series 150/300 mm width	A06B-6114-K506	
		βi series βi SV (two-axis model)	A06B-6114-K505	
A06B-6093-K001	BR-AGCF2W (Panasonic)	βi series βi SV4, βi SV20	A06B-6093-K002	
		βi series βi SV40, βi SV80	A06B-6093-K002	

NOTE

When an old type BR-CCF2TH battery is used, order a battery case that accommodates battery A06B-6114-K504.

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.

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 a data server is made by bit 0 (ISO) of parameter No. 0908.

	#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 Series 0i-PD, 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

Set the path to which each axis belongs.
For the Series 0i-PD, be sure to set this parameter to 1.

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.

0983

Path control type of each path

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 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

NOTE

In the Series 0i-PD, this parameter is automatically set to 1.

1001

#7	#6	#5	#4	#3	#2	#1	#0
							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 IS COMMANDED WITHOUT ZERO RETURN" 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 bit 1 (LRP) 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 bit 0 (IDGx) of parameter No. 1012 to prevent the reference position from being set again is:

0: Not performed.

1: Performed.

NOTE

When this parameter is set to 0, bit 0 (IDGx) of parameter No. 1012 is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
1004	IPR							

[Input type] Parameter input

[Data type] Bit path

#7 IPR When a number with no decimal point is specified, in the metric input with increment systems of IS-B, the least input increment of each axis is:

- 0: Not 10 times greater than the least command increment
1: 10 times greater than the least command increment

NOTE

When the increment system is IS-A, and bit 0 (DPI) of parameter No. 3401 is set to 1 (pocket calculator type decimal point programming), the least input increment cannot be 10 times greater than the least command increment.

	#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.)

#4 EDPx In cutting feed, an external deceleration signal in the + direction for each axis is:

- 0: Invalid
1: Valid

#5 EDMx In cutting feed, an external deceleration signal in the - direction for each axis is:

- 0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMlx				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 rotary axis.

#1 **ROSx** Setting linear or rotary 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	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

#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 rotary axis roll-over is

0: Invalid

1: Valid

NOTE

ROAx specifies the function only for a rotary axis (for which bit 0 (ROTx) of parameter No. 1006 is set to 1)

#1 **RABx** In the absolute programming, 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								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** Increment system of each axis

Increment system	Bit 0 (ISA)
IS-A	1
IS-B	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, 84 to 90

An axis name (parameter No. 1020) can be arbitrarily selected from A, B, C, T, U, V, W, X, Y, and Z.

(Tip) ASCII code

Axis name	X	Y	Z	U	V	W	A	B	C	T
Setting	88	89	90	85	86	87	65	66	67	84

NOTE

- 1 If the system supports the function for machining multiple workpieces, addresses U, V, and W cannot be used for the names of the axes controlled by the CNC.
- 2 If macro functions A, B, U, V, and W for a punch press are used, addresses A, B, U, V, and W cannot be used for the names of the axes controlled by the CNC.
- 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.

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, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane and 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	Rotary 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 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

- 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.

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 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, "IMPROPER G-CODE" 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 performing automatic coordinate system setting.

1260	The shift amount per one rotation of a rotary 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 rotary axis.

	#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:

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 G33 (skip) 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

The area outside the area set by parameters Nos. 1320 and 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

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

The area inside the area set by parameters Nos. 1324 and 1325 is a prohibited area.

1326	Coordinate value II of stored stroke check 1 in the positive 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 The area outside the area set by parameter No. 1326 and No. 1327 is a prohibited area.
- 2 The EXLM signal is valid only when bit 2 (LMS) of parameter No. 1300 is set to 1.
- 3 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				

[Input type] Parameter input

[Data type] Bit path

#4 JRV Jog feed or incremental feed is

0: Performed at feed per minute.

1: Performed at feed per rotation.

NOTE

- 1 Specify a feedrate in parameter No. 1423.
- 2 This bit must be set 0 with FANUC Series 0i-PD.

	#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		FC0						

[Input type] Parameter input

[Data type] Bit path

- #7 FC0** Specifies the behavior of the machine tool when a block (G01, G02, G03, etc.) containing a feedrate command (F command) that is 0 is issued during automatic operation, as follows:
 0: An alarm PS0011, "FEED ZERO (COMMAND)" occurs.
 1: An alarm PS0011 does not occur, and the block is executed.

NOTE

This parameter is set from 1 to 0, if a bit 6 (CLR) of parameter No. 3402 is 1, reset the CNC. Or if CLR is 0, turn off and on the CNC.

	#7	#6	#5	#4	#3	#2	#1	#0
1405			EDR					

[Input type] Parameter input

[Data type] Bit path

- #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 rotary axis is exercised using:
 0: Conventional method
 1: Method that specifies a feedrate on the virtual circle of the rotary 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.

1411	Cutting feedrate

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[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)
 A cutting feedrate can be specified with this parameter for a machine which does not have to change the cutting feedrate frequently during machining. This eliminates the need to specify a cutting feedrate (F code) in the NC program.

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)
 Specify a jog feedrate (feed per minute) 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 rapid traverse override is 100% for each axis.

NOTE

- 1 If 0 is set, the rate set in parameter No. 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.

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

No. 1420: Rapid traverse rate

No. 1423: Jog feedrate

No. 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 AI advanced preview control.
When the acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No.1430 is used.

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.

1465**Radius of a virtual circle when a feedrate is specified on the virtual circle of a rotary 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 rotary axis is specified.
 If 0 is set for a rotary 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.

	#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								CTLx

[Input type] Parameter input

[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run during cutting feed

- 0: Exponential acceleration/deceleration is applied.
 1: Linear acceleration/deceleration after interpolation is applied.

1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis							

[Input type] Parameter input

[Data type] Word axis

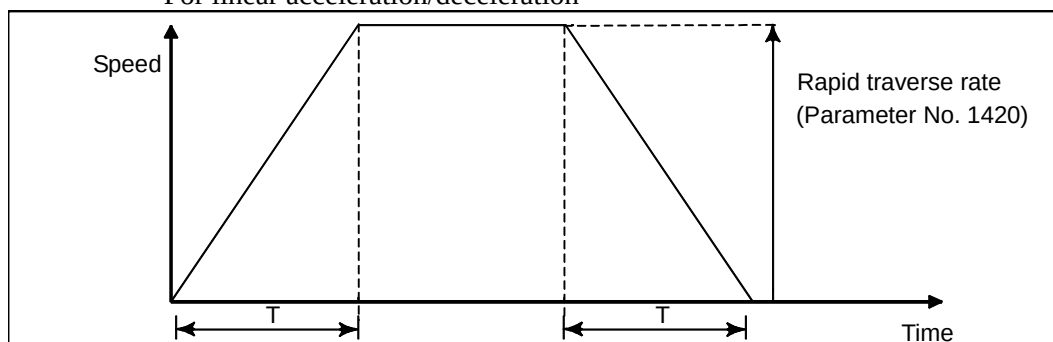
[Unit of data] msec

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.

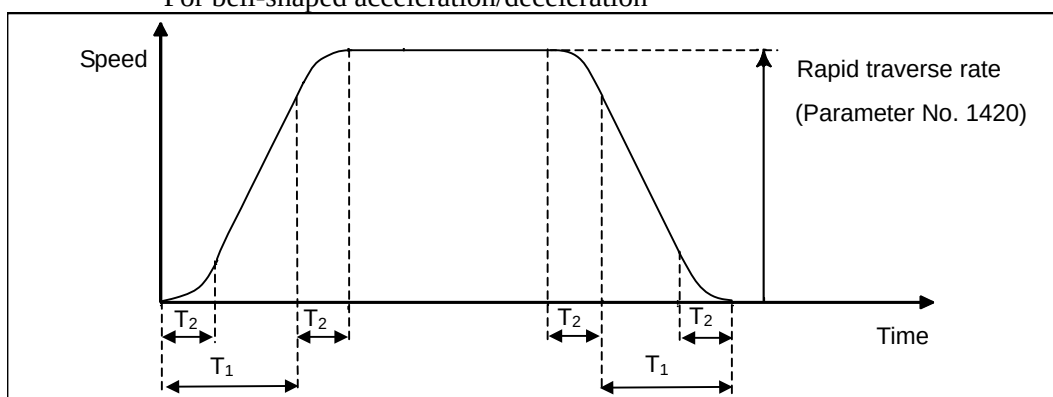
[Example]

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, or linear acceleration/deceleration after interpolation in cutting feed for each axis. Which type to use is selected with bit 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.

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] %

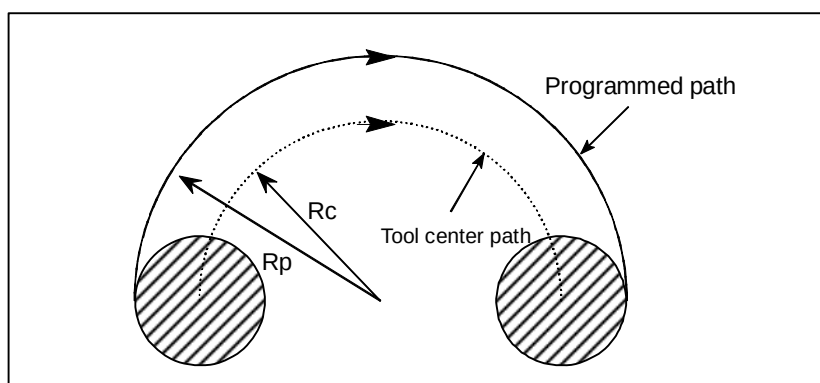
[Valid data range] 0 to 100

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:

$$F \times \frac{R_c}{R_p} \quad \left(\begin{array}{l} R_c: \text{Radius of tool center path} \\ R_p: \text{Programmed radius} \end{array} \right)$$

Thus, the feedrate along the programmed path satisfies the specified value of F.



However, if R_c is too small when compared with R_p , $R_c/R_p \rightarrow 0$ results to stop the tool. So, a minimum deceleration ratio (MDR) is set, and the feedrate of the tool is set to $F \times (\text{MDR})$ when $R_c/R_p \leq \text{MDR}$.

When this parameter is 0, the minimum deceleration ratio (MDR) is 100%.

1711

Inner determination angle (θ_p) 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**

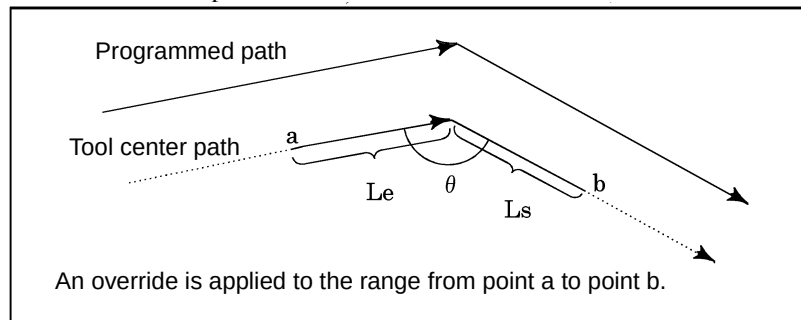
- [Input type] Setting input
 [Data type] Real path
 [Unit of data] mm, inch (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
 (When the increment system is IS-B, -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 θ_p .)

When a corner is determined to be an inner corner, an override is applied to the feedrate in the range of L_e in the previous block from the intersection of the corner and in the range of L_s in the next block from the intersection of the corner.

Distances L_e and L_s represent linear distances from the intersection of a corner to points on the tool center path.

L_e and L_s are set in parameters Nos. 1713 and 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/sec², inch/sec², degree/sec² (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.

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.

1802

#7	#6	#5	#4	#3	#2	#1	#0
					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

- When this parameter is set to 1, specify the direction of the scale zero point by setting bit 4 (SCP) of parameter No. 1817.
- 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.

1815

#7	#6	#5	#4	#3	#2	#1	#0
		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 Pulsecoder is not used.
1: A separate Pulsecoder 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 Pulsecoder)

	#7	#6	#5	#4	#3	#2	#1	#0
1817		TANx		SCPx				

[Input type] Parameter input

[Data type] Bit axis

NOTE

When this parameter is set, the power must be turned off before operation is continued.

#4 SCPx For two-point measurement (when bit 2 (DC2) of parameter No. 1802 is set to 1), the scale zero point direction is:

- 0: On the minus side. (The reference position is located in the plus direction when viewed from the scale zero point.)
1: On the plus side. (The reference position is located in the minus direction when viewed from the scale zero point.)

NOTE

- 1 This parameter is valid when bit 2 (DC2) of parameter No. 1802 is set to 1.
- 2 If this parameter is set to an incorrect value, an incorrect coordinate system is established. In such a case, reverse the setting then perform reference position establishment operation again.

- #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					SDC		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 <Fn120>= "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 not established (reference position establishment signal 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.
 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 parameters Nos. 1883 and 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 parameters Nos. 1815, 1821, and 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, parameters Nos. 1883 and 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 :

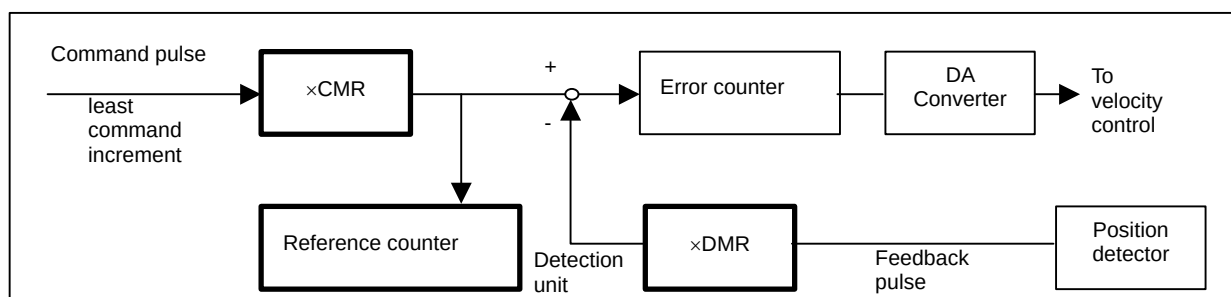
Set a command multiplier indicating the ratio of the least command increment to the detection unit for each axis.

Least command increment = detection unit × command multiplier

Relationship between the increment system and the least command increment

Increment system	Least input increment and least command increment		
	IS-A	IS-B	Unit
Millimeter machine	0.01	0.001	mm
Millimeter input	0.001	0.0001	inch
Rotary axis	0.01	0.001	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).

$$\begin{aligned} \text{[Least command increment]}/\text{CMR} &= \text{[Detection unit]} \\ &= \text{[Feedback pulse unit]}/\text{DMR} \end{aligned}$$

[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 Pulsecoder]/

[Number of pulses per rotation of the Pulsecoder]

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 Pulsecoder]

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

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:

$$F_{\max}[\text{mm/min}] = 196602 \times 10^4 \times \text{least command increment} / \text{CMR}$$

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, "EXCESS ERROR (MOVING)" 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, "EXCESS ERROR (STOP)" is generated, and operation is stopped immediately (as in emergency stop).

1838

Position deviation limit for each axis in moving during safety check

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 99999999

Position deviation limit for each axis during moving for safety check of Dual Check Safety function is specified.

If position deviation of a moving axis exceeds position deviation limit while Safety Check is carried out (Safety Monitoring Request "*VLDVx" =0), a servo alarm SV0475, "EXCESS ERROR(MOVE:SV)" or SV1071, "EXCESS ERROR(MOVE:CNC)" is generated and axes are stopped immediately like emergency stop state..

In Dual Check Safety function, position deviation is always checked by CNC and Servo. In case that Safety Check is carried out (Safety Monitoring Request "*VLDVx" =0), the servo alarm SV0475 or SV1071 is generated when each CPU finds out that the deviation exceeds position deviation limit in moving state.

1841

Position deviation limit of each axis in moving state during other than Dual Check Safety monitoring
(for Dual Check Safety Function)

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 99999999

Set the positioning deviation limit in moving state for each axis for Dual Check Safety function, in case that Safety Check is not carried out (Safety Monitoring Request “*VLDVx”=1).

In case that Safety Check is not carried out (Safety Monitoring Request “*VLDVx” =1), servo alarm SV0475, “EXCESS ERROR(MOVE:SV)” or SV1071, “EXCESS ERROR(MOVE:CNC)” is generated and operation is stopped immediately (as in emergency stop), when each CPU finds out that the deviation exceeds position deviation limit in moving state.

If the value of this parameter is “0”, the parameter No.1828 is used for the value of deviation limit in moving state.

In case that Safety Check is carried out (Safety Monitoring Request “*VLDVx” =0), the parameter No.1838 is used for the value of deviation limit in moving state.

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

$$= \text{No. 1884} \times 1,000,000,000 + \text{No. 1883}$$

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 Fig. A.1 (a).)

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.

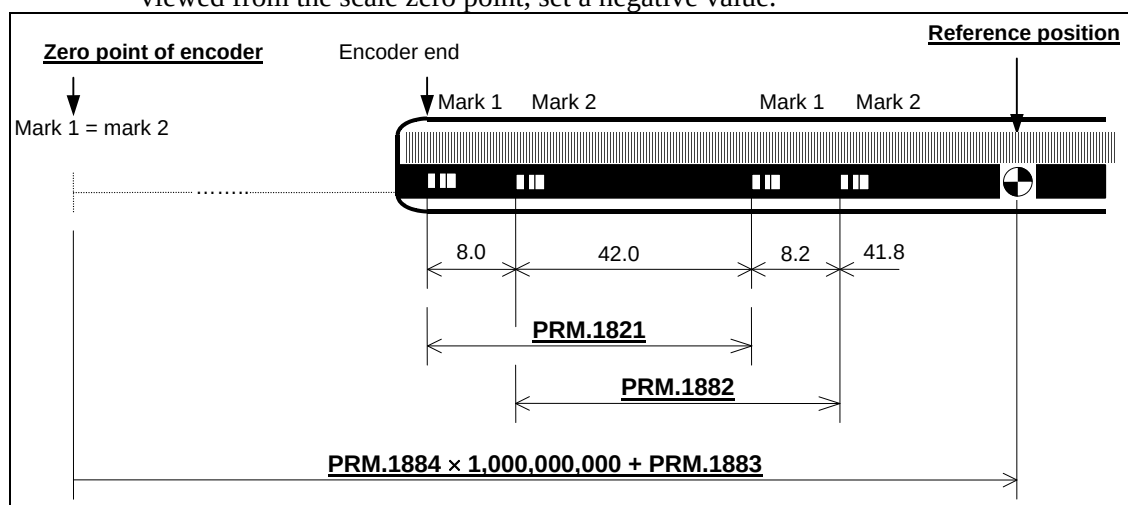


Fig. A.1 (a)

[Example of parameter settings]

When an encoder as shown Fig. A.1 (b) is used with an IS-B, millimeter machine:

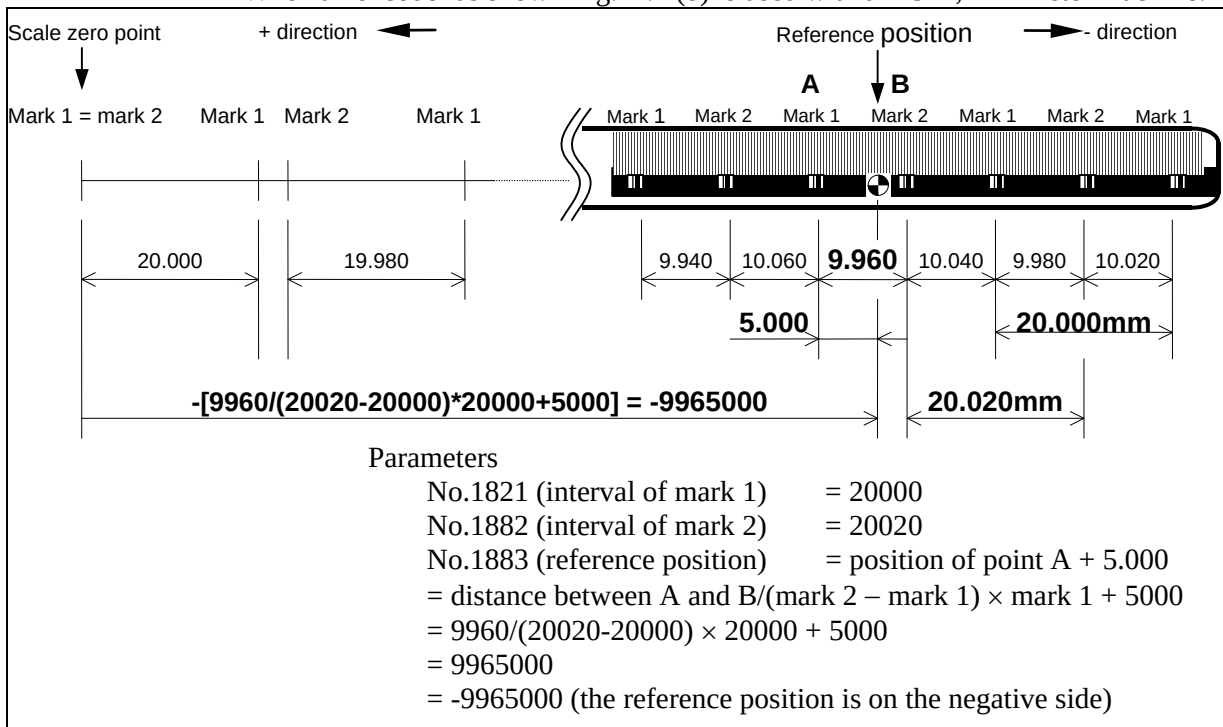


Fig. A.1 (b)

[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 Fig. A.1 (c).

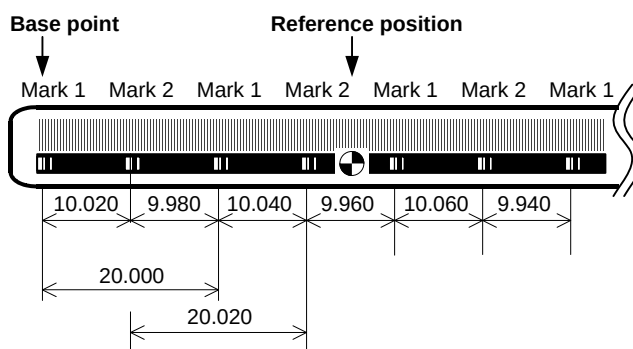


Fig. A.1 (c)

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 Nos. 1883 and 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 $\times$ 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 DS0016 or DS1448 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 bit 0 (FMD) 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 Nos. 2000 to 2999 are for digital servo, The following parameters are not explained in this manual. Refer to “FANUC AC SERVO MOTOR α i series, FANUC AC SERVO MOTOR β i series, FANUC LINEAR MOTOR LiS series, FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR DiS series PARAMETER MANUAL (B-65270EN)”.

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
3008						XSG		

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

- #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, and the PMC axis control skip 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.
 Depending on the option configuration of the I/O Link, the actually usable X addresses are:
 <X0000 to X0127>, <X0200 to X0327>

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.

Depending on the option configuration of the I/O Link, the actually usable X addresses are:

<X0000 to X0127>, <X0200 to X0327>

3019

Address to which the PMC axis control skip signal 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 path

[Valid data range] 0 to 327

Set an X address to which the PMC axis control skip signal ESKIP is to be assigned.

Example 1. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 6

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal is allocated to X0006 and the skip signal is allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0
X0005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7
	#7	#6	#5	#4	#3	#2	#1	#0
X0006		ESKIP						

Example 2. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 5

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, and skip signal are allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0
X0005	SKIP	ESKIP SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7

NOTE

This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.

Depending on the option configuration of the I/O Link, the actually usable X addresses are:

<X0000 to X0127>, <X0200 to X0327>

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, "TOO MANY DIGIT" is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC		DRC		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

(3) Workpiece coordinate system presetting based on G92.1

#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.

#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							

[Input type] Parameter input

[Data type] Bit path

#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			OPM					SVS

[Input type] Setting input

[Data type] Bit path

#0 SVS Servo setting screen and servo tuning screen

0: Not displayed

1: Displayed

#5 OPM Operating monitor

0: Not displayed

1: Displayed

	#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
3113	MS1	MS0						HMC

[Input type] Parameter input

[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

- 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.
- 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.
- 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.
- 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.

	#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.

#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.

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.

3131	Subscript of axis name

[Input type] Parameter input

[Data type] Byte axis

[Valid data range] 0 to 9, 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 parallel axis, 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

	#7	#6	#5	#4	#3	#2	#1	#0
3195	EKE	HDE	HKE					

[Input type] Parameter input

[Data type] Common to the bit system

#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

[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 set to 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 (PSW)							

[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 $\neq$ 0 and password $\neq$ 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, "ILLEGAL FORMAT IN G10 L52" 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 (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.

3280

#7

#6

#5

#4

#3

#2

#1

#0

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 19

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
- 19 : Rumanian

If a number not indicated above is set, English is selected.

	#7	#6	#5	#4	#3	#2	#1	#0
3401			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 programming 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 lathe system is used, this parameter is invalid.

- #5 ABS** Program command in MDI operation
- 0: Assumed as an incremental programming
 - 1: Assumed as an absolute programming

NOTE

ABS is valid when bit 4 (MAB) of parameter No. 3401 is set to 1.
When G code system A of the lathe system is used, this parameter is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
3402	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)

#3 G91 When the power is turned on or when the control is cleared
 0: G90 mode (absolute programming)
 1: G91 mode (incremental programming)

#6 CLR  key 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
3403		ADB						

[Input type] Parameter input

[Data type] Bit path

#6 ADB When the same address two or more times are specified in one block:
 0: The address specified last is valid.
 1: It is treated as a program error and the alarm PS5074, "ADDRESS DUPLICATION ERROR" is issued.

NOTE

The following notes apply when this parameter is set to 1:

- 1 When two or more M codes are acceptable to one block, up to three M codes can be specified in the same block. Specifying more than three results in the alarm PS5074.
- 2 You can specify any number of G codes in the same block as long as they belong to different groups. Specifying G codes belonging to the same group causes the alarm PS5074. You can however specify any number of G90 and G91 codes in the same block as they cause no alarm.
- 3 The alarm is not caused by blocks which call a custom macro or execution macro.

	#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, 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								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
Inch input system	IS-A for reference axis	100 times	1000 times
	IS-B for reference axis	1000 times	10000 times

	#7	#6	#5	#4	#3	#2	#1	#0
3406	C07	C06	C05	C04	C03	C02	C01	
3407	C15	C14	C13	C12	C11	C10	C09	C08
3408	C23	C22		C20	C19	C18	C17	C16
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 and 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.

3411	M code preventing buffering 1
3412	M code preventing buffering 2
to	to
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 parameters 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.
- 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, "ILLEGAL USE OF DECIMAL POINT" is issued.
3. When a negative value is specified
The alarm PS0006, "ILLEGAL USE OF MINUS SIGN" 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.

Bit 0 (DPI) of parameter No. 3401	Bit 0 (AUP) of parameter No. 3450	BDX=0		BDX=1
		Without the second auxiliary function option	With the second auxiliary function option	
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
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, degrees, 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.

	#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.

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 : mm, inch, deg or 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
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 tool 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. Tool center path Programmed path
0	1	Type B	A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output. Intersection point Tool center path Programmed path
1	0	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. Intersection point Tool center path Programmed path Shift 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.

	#7	#6	#5	#4	#3	#2	#1	#0
5004						ODI		

[Input type] Parameter input

[Data type] Bit path

#2 **ODI** The setting of a tool radius 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 (tool radius - tool nose radius compensation) is specified in the MDI mode, an alarm is:

0: Not raised.

1: Raised. (alarm PS5257, "G41/G42 NOT ALLOWED IN MDI MODE")

	#7	#6	#5	#4	#3	#2	#1	#0
5042					OFE	OFD	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 These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFA	Unit	Valid data range
1	0.01mm	±9999.99mm
0	0.001mm	±9999.999mm

For inch input

OFA	Unit	Valid data range
1	0.001inch	±999.999inch
0	0.0001inch	±999.9999inch

	#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 bit 7 (SCR) 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 bit 7 (SCR) 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 -99999999 to -1 and 1 to 99999999.

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**

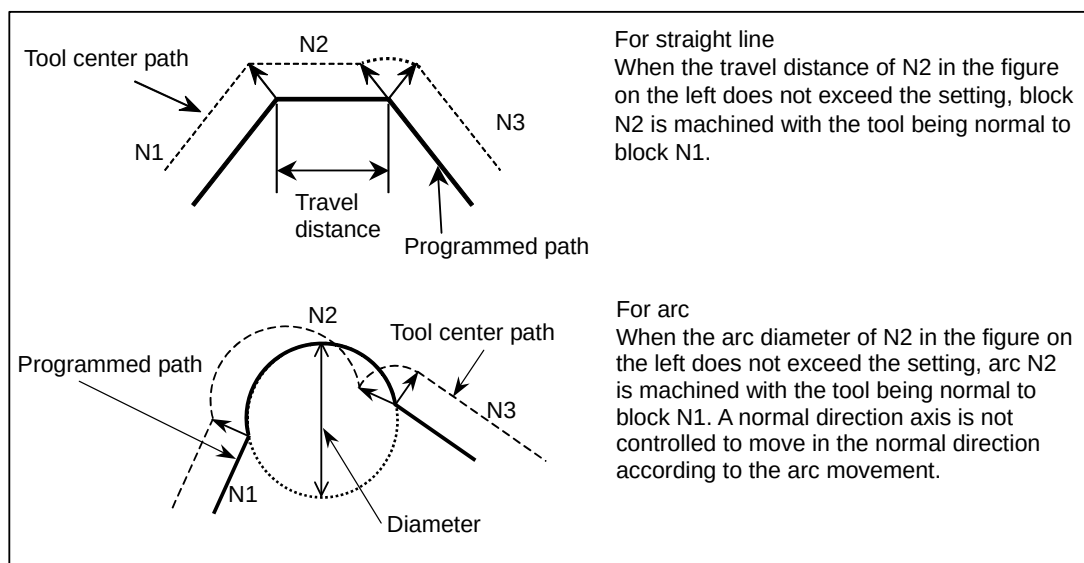
[Input type] Parameter input

[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))



	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM	HGO			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/G66.1) is not set:

0: Alarm PS1100, "CANCEL WITHOUT MODAL CALL" 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.

#4 HGO When a GOTO statement for specifying custom macro control is executed, a branch to 30 sequence numbers just before the GOTO statement or to up to 10 sequence numbers saved by a sequence number search previously made with a GOTO statement is:

0: Not made at high speed.

1: Made at high speed.

#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								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.

	#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	F16

[Input type] Parameter input

[Data type] Bit path

#0 F16 The precision of operation is based on:

0: New specification.

1: FS0i-PC compatible specification.

#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 patterns of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameter No. 6010 to No. 6018.
- #6 GMP** The calling of M, S, T, a second auxiliary function code, or a particular code during the calling of a G code, and the calling of a G code during the calling of M, S, T, a second auxiliary function code, or particular code are:
 - 0: Not allowed. (They are executed as an ordinary G, M, S, T, second auxiliary function code, 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 to 1:
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
	#7	#6	#5	#4	#3	#2	#1	#0
6011	=7	=6	=5	=4	=3	=2	=1	=0
	#7	#6	#5	#4	#3	#2	#1	#0
6012	#7	#6	#5	#4	#3	#2	#1	#0
	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0
	#7	#6	#5	#4	#3	#2	#1	#0
6014	]7	]6	]5	]4	]3	]2	]1	]0
	#7	#6	#5	#4	#3	#2	#1	#0
6015	?7	?6	?5	?4	?3	?2	?1	?0
	#7	#6	#5	#4	#3	#2	#1	#0
6016	@7	@6	@5	@4	@3	@2	@1	@0
	#7	#6	#5	#4	#3	#2	#1	#0
6017	&7	&6	&5	&4	&3	&2	&1	&0
	#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] 03 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.

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 parameters Nos. 6050 to 6059, the calls specified by parameters Nos. 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 parameters Nos. 6071 to 6079 used to call subprograms and in a parameters Nos. 6080 to 6089 used to call custom macros, a custom macro is called preferentially. For example, if 300 is set in parameters Nos. 6071 and 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	30i/31i-PB
A	65	O
B	66	O
D	68	O
F	70	O
H	72	O
I	73	O
J	74	O
K	75	O
L	76	O
M	77	O
P	80	O
Q	81	O
R	82	O
S	83	O
T	84	O
V	86	O
X	88	O
Y	89	O
Z	90	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.

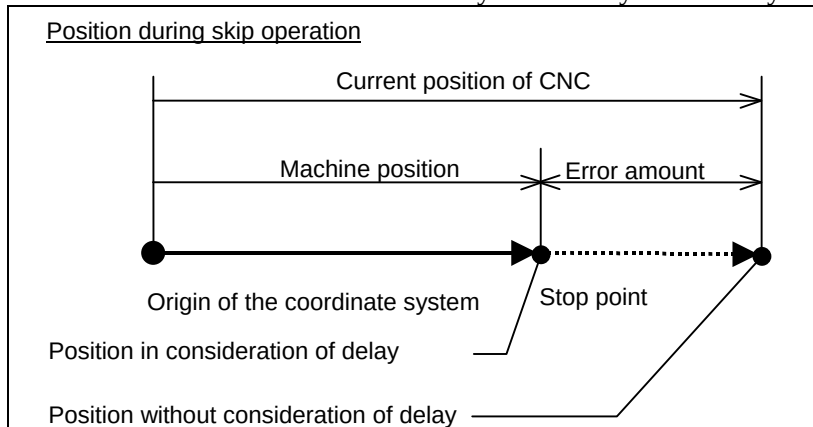
- #6 SRE** When a high-speed skip signal or high-speed measurement position arrival signal is used:
 0: The signal is assumed to be input on the rising edge (contact open → close).
 1: The signal is assumed to be input on the falling edge (contact close → open).
- #7 SKF** Dry run, override, and automatic acceleration/deceleration for G33 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 goes on while the skip function 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 goes on are considered to obtain the position at which the signal is input.
- #2 TSE** When the torque limit skip function (G33 P98/99) is used, the skip position held in a system variable (#5061 to #5080 (#100151 to #100182 when 21-axes-or-more-system is used)) is:
 0: Position that is offset considering the delay (positional deviation) incurred by the servo system.
 1: Position that does not reflect the delay incurred by the servo system.



- #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 (G33), the skip signal SKIP is:

0: Enabled.

1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SPE) of parameter No. 6201	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 G33 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	Parameter	High-speed skip signals
1S1	HDI0	1S5	HDI4
1S2	HDI1	1S6	HDI5
1S3	HDI2	1S7	HDI6
1S4	HDI3	1S8	HDI7

	#7	#6	#5	#4	#3	#2	#1	#0
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1
	#7	#6	#5	#4	#3	#2	#1	#0
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1
	#7	#6	#5	#4	#3	#2	#1	#0
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1
	#7	#6	#5	#4	#3	#2	#1	#0
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 (G33, or G33P1 to G33P4) 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	G33 G34P1 G04Q1	G33P2 G04Q2	G33P3 G04Q3	G33P4 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/HDI4	1S5	2S5	3S5	4S5	DS5
SKIP6/HDI5	1S6	2S6	3S6	4S6	DS6
SKIP7/HDI6	1S7	2S7	3S7	4S7	DS7
SKIP8/HDI7	1S8	2S8	3S8	4S8	DS8

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	G33P1,G04Q1
When bit 0 (2S1) of parameter No. 6203 is set to 1	G33P2,G04Q2
When bit 0 (3S1) of parameter No. 6204 is set to 1	G33P3,G04Q3
When bit 0 (4S1) of parameter No. 6205 is set to 1	G33P4,G04Q4
When bit 0 (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 (G33) 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] 2msec

[Valid data range] 0 to 65535

The torque limit skip arrival signal is ignored for a set period of time.

If G33P98 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 G33P99 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.

6281**Feedrate for the skip function (G33)**

[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 (G33, G33 P1)****6283****Feedrate for the skip function (G33 P2)****6284****Feedrate for the skip function (G33 P3)****6285****Feedrate for the skip function (G33 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, "EXCESS ERROR (G31)" is issued and an immediate stop takes place.

6581	RGB value of color palette 1
to	to
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 999999999

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, M30 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 parameters Nos. 6751 and 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 parameters Nos. 6753 and 6754.

	#7	#6	#5	#4	#3	#2	#1	#0
7100					HCL			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 is:
 0: Invalid.
 1: Valid.

#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 <Gn019.4, Gn019.5>), 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 <Gn019.4, Gn019.5>), 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

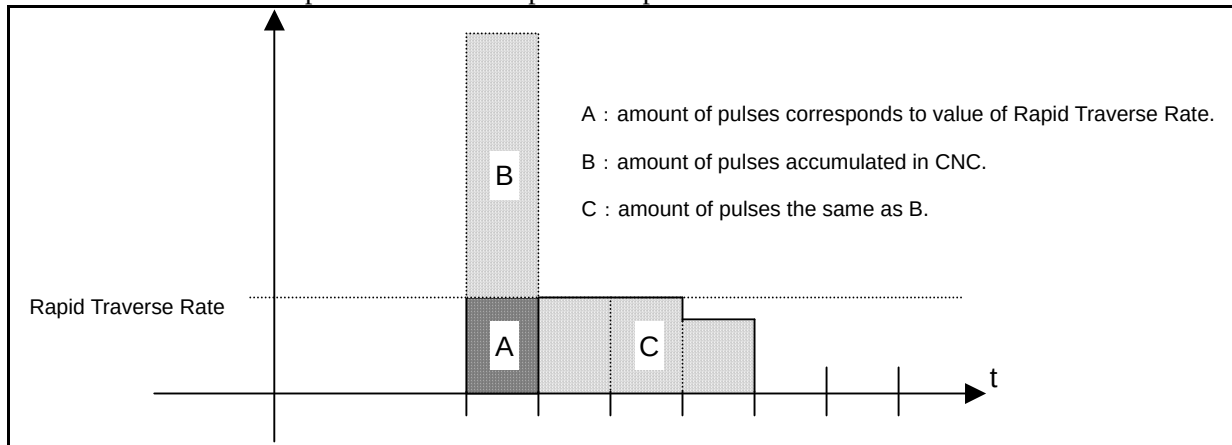
[Data type] 2-word path

[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

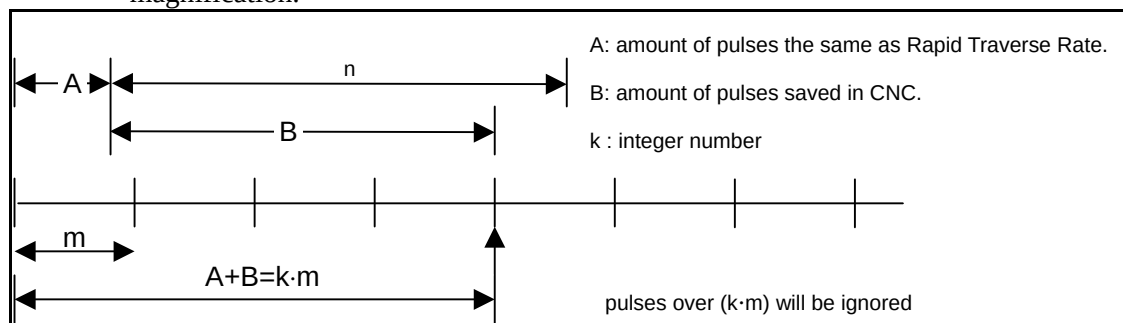
- 2) Parameter No.7117 > 0

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.

$n \geq m$: 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 ($n \geq m$)

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 " 

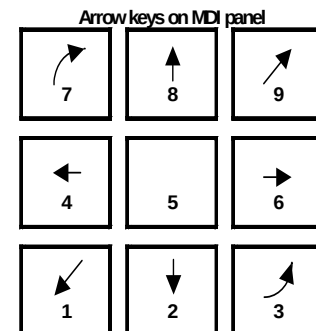
[Input type] Parameter input

[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 unit 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



[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
8001					RDE	OVE		MLE

[Input type] Parameter input

[Data type] Bit path

#0 MLE Whether all axis machine lock signal MLK <Gn108> 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	Bit 2 (OVE) of parameter No. 8001 = 0 (same signals as those used for the CNC)	Bit 2 (OVE) of parameter No. 8001 = 1 (signals specific to the PMC)
Feedrate override signals	*FV0 to *FV7 <G012>	*EFOV0 to *EFOV7 <G151>
Override cancellation signal	OVC <G006.4>	EOVC <G150.5>
Rapid traverse override signals	ROV1,2 <G014.0,1>	EROV1,2 <G150.0,1>
Dry run signal	DRN <G046.7>	EDRN <G150.7>
Rapid traverse selection signal	RT <G019.7>	ERT <G150.6>

(The listed signal addresses when PMC signals are selected are for the A-group. Actual addresses differ depending on the used group.)

#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			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
Millimeter input (mm/min)	0	10	1
	1	100	10
Inch input (inch/min)	0	0.1	0.01
	1	1	0.1

#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

	#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			MLS	

[Input type] Parameter input

[Data type] Bit path

#1 MLS When bit 0 (MLE) of parameter No. 8001 is set to 1 (to disable the all axis machine lock signal) in PMC axis control, axis-by-axis machine lock is:

- 0: Disabled.
1: Enabled.

#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, “ZERO RETURN NOT FINISHED” 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 MI8 <G106.0 to .7> 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] 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)

NOTE

When a value other than the above is set, the axis is not controlled by the PMC.

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.

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.

	#7	#6	#5	#4	#3	#2	#1	#0
8131					AOV	EDC		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.

#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			SCL			BCD		

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

#2 BCD Second auxiliary function is:

0: Not Used.

1: Used.

#5 SCL Scaling is:

0: Not Used.

1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8134	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 Stored stroke limits 2 and 3 are:

0: Used.

1: Not 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		NCV	NMC				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.

#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	#6	#5	#4	#3	#2	#1	#0
8136				NOW	NOP		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.

#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.

	#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 0, 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.

	#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 <G0140> or the axis synchronous control selection signal SYNC <G0138> 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 three sets of axis synchronous control is used:

When the master axes are the first axis, second axis, and third axis, and the slave axes are the sixth axis, fifth axis, and fourth 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) = 0

Parameter No. 8311 A (fourth axis) = 3

Parameter No. 8311 B (fifth axis) = 2

Parameter No. 8311 C (sixth 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, "SYNC EXCESS ERROR (MCN)".

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 9999999999
	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, "SYNC EXCESS ERROR (POS DEV)" 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, "SYNC ALIGNMENT ERROR" is issued, and the synchronization establishment is not performed. Specify a slave axis for this parameter. To enable this parameter, set the bit 7 (SOF) 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 9999999999
	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 999999999

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 999999999

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	Maximum allowable feedrate for AI advanced preview 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 maximum allowable feedrate for AI advanced preview control.

If a feedrate higher than the setting of this parameter is specified in the AI advanced preview control mode, the feedrate is clamped to that set in this parameter.

If this parameter is set to 0, no clamping is performed.

When bit 7 (NOF) of parameter No. 8451 is set to 1, the tool moves, assuming that the feedrate set in this parameter is specified. If 0 is set in this parameter at this time, a movement is made at the specified feedrate.

	#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

to

to

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
11222						IMG	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, "ILLEGAL INCH/METRIC CONVERSION" 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.:

- 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

#2 IMG Inch/metric changeover is:

0: Performed with the G20/G21.

1: Not performed with the G20/G21.

NOTE

If bit 2 of parameter No. 11222 is 1 (inch/metric changeover with G20/G21 is disabled), only bit 2 of parameter No. 0 can be used to perform inch/metric changeover. If bit 2 of parameter No.0 = 0, the metric system is used. If bit 2 of parameter No.0 = 1, the inch system is used.

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 unit
- Total position display screen

The display sequence of coordinates corresponds to the parameter setting as follows:

Display sequence of coordinates Setting	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
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.

	#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)

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 01 to 10 is recorded.

	#7	#6	#5	#4	#3	#2	#1	#0
14000						IRFx		

[Input type] Parameter input

[Data type] Bit axis

#2 IIRFx 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 rate 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, "SERIAL DCL:REF-POS ESTABLISH ERR" is issued. When this parameter is set to 0, the maximum allowable travel distance is not checked.

NOTE

To establish the reference position with axis synchronous control, set the parameter for both master and slave axes.

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.

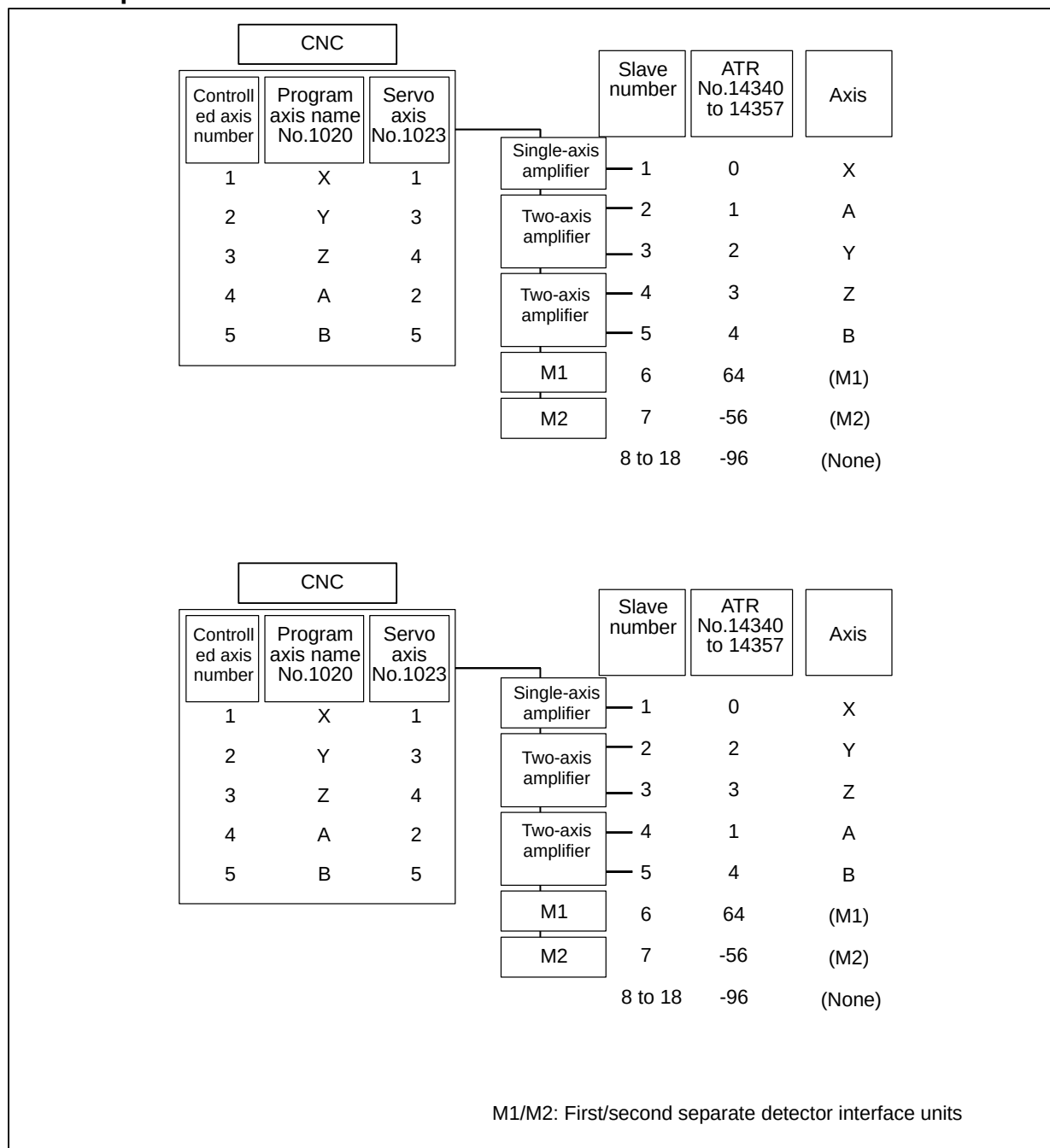
- When the slave is an amplifier:
Set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis to which the amplifier is assigned.
- When the slave is a separate detector interface unit:
Set 64 for the first separate detector interface unit (connected near the CNC), and set -56 for the second separate detector interface unit (connected far from the CNC).
- When the slave is nonexistent:
Set -96.

NOTE

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



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 FS0i-D-specific mode.

1: The FS0i-C-compatible mode.

	#7	#6	#5	#4	#3	#2	#1	#0
16001	CPF							

[Input type] Parameter input

[Data type] Bit path

#7 CPF At the end of the 01 group containing the G01, G02, or G03 code, the PF signal to start pressing is:

0: Not set to "1".

1: Set to "1".

	#7	#6	#5	#4	#3	#2	#1	#0
16003			TCF					

[Input type] Parameter input

[Data type] Bit path

#5 TCF After the OP signal indicating that automatic operation is in progress is set from "0" to "1", the PF signal to start pressing is set to "1":

0: Only when a T command is found.

This status is the same as the status in which the PFW signal to wait for the start of pressing is set to “1”.

1: Even if no T commands are found.

16010**Upper limit of the position deviation at which PF is set to “1”**

[Input type] Parameter input

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

For each axis, parameter No. 16010 sets the upper limit of the positional deviation at which the PF signal to start pressing is set to “1”. When the absolute value of the positional deviation does not exceed this highest limit, PF is set to “1”.

Parameter No.16010 is validated when parameter PFE (No. 16001, #1) is set to 1.

NOTE

The parameter can only be set for the X, Y, and C axes.

16032**Time interval by which setting of PF to “1” follows positioning in the forming mode**

[Input type] Parameter input

[Data type] Word path

[Unit of data] msec

[Valid data range] 0 to 248

Parameter No. 16032 sets the time interval by which setting the PF signal to start pressing to “1” follows positioning in the forming mode (except for nibbling).

16033**Time interval by which the start of the next block follows setting of *PFIN to “0” in the forming mode**

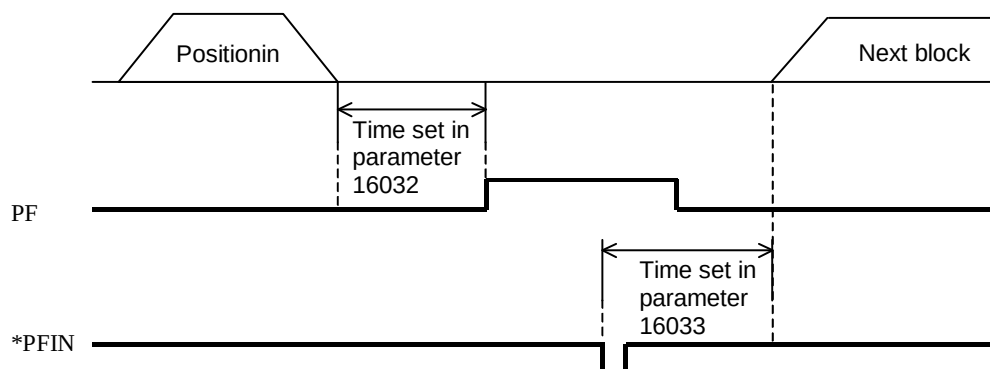
[Input type] Parameter input

[Data type] Word path

[Unit of data] msec

[Valid data range] 0 to 248

Parameter No. 16033 sets the time interval by which the start of the next block follows setting the contact of the *PFIN signal to complete punching for single-cycle pressing to “0” in the forming mode.

**16050**

#7	#6	#5	#4	#3	#2	#1	#0
							G0F

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

#0 G0F For a rapid traverse command (G00), the X-axis or Y-axis rapid traverse feedrate is set to the value:

0: Specified in the parameter.

1: Specified by the F code.

The maximum feedrate of the F command is limited to the rapid traverse feedrate in the parameter. OADx(No.6131, #0) and ILG(No.6132, #0) are valid.

	#7	#6	#5	#4	#3	#2	#1	#0
16181						NSP		NMG

[Input type] Parameter input

[Data type] Bit path

#0 NMG When the M code for canceling the nibbling mode (No. 16184) is specified, the G code in the 01 group is:

0: Not changed.

1: Changed to G00 (rapid traverse).

#2 NSP When the *SP signal to halt automatic operation is set to "0" in nibbling, automatic operation is:

0: Decelerated and halted immediately.

1: Halted after positioning for a nibbling pitch completes.

16183	M code for setting the nibbling mode

[Input type] Parameter input

[Data type] Word path

[Valid data range] 1 to 255

Parameter No.16183 sets the M code for setting the nibbling mode.

16184	M code for canceling the nibbling mode

[Input type] Parameter input

[Data type] Word path

[Valid data range] 1 to 255

Parameter 16183 sets the M code for canceling the nibbling mode.

16186	Maximum pitch that can be specified with G01, G02, or G03 for nibbling by G68 or G69 or by an M code

[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] Positive 9 digit of minimum unit of data

(When the increment system is IS-B, 0.001 to +999999.999)

Parameter No. 16186 specifies the maximum pitch that can be specified with G01, G02, or G03 for nibbling by G68 or G69 or by an M code.

16188	Maximum pitch of the G00 command for nibbling by the M code

[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] Positive 9 digit of minimum unit of data
(When the increment system is IS-B, 0.001 to +999999.999)
Parameter No. 16188 specifies the maximum pitch of the G00 command for nibbling by the M code.

16194	Maximum distance traveled along C-axis in nibbling

[Input type] Parameter input

[Data type] Real path

[Unit of data] degree (input unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Positive 9 digit of minimum unit of data
(When the increment system is IS-B, 0.001 to +999999.999)
The parameter sets the maximum distance traveled along the C-axis for the nibbling mode.

	#7	#6	#5	#4	#3	#2	#1	#0
16200								UVC

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

#0 UVC In the reset status, the macro stored under a U or V macro number is:

0: Deleted.

1: Not deleted.

	#7	#6	#5	#4	#3	#2	#1	#0
16202								AIP

[Input type] Parameter input

[Data type] Bit path

#0 AIP Specifies the condition for the share-proof command (G86).

0: Alarm (PS4506) is issued when $I < 1.5P$.

1: Alarm (PS4506) is issued when $I < P$.

	#7	#6	#5	#4	#3	#2	#1	#0
16205								SZR

[Input type] Parameter input

[Data type] Bit path

#0 SZR Automatic reference position return (G28) commands:

- 0: The axis returns directly to the reference position without passing through any intermediate points.
 1: The axis returns to the reference position via the specified intermediate point.

16209**Clearance and amount of return for the Y axis in automatic repositioning**

[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))

(When the increment system is IS-B, 0.0 to +999999.999)

Positive 9 digit of minimum unit of data

(When the increment system is IS-B, 0.001 to +999999.999)

Each of the parameters sets the clearance and amount of return for the Y-axis in automatic repositioning (G75).

16240

#7	#6	#5	#4	#3	#2	#1	#0
		RLM					

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

#5 RLM When the power is turned on or in the clear status, the machine is set in the:

0: Punch mode.

1: Laser mode.

16242

#7	#6	#5	#4	#3	#2	#1	#0
							ALA

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit

#0 ALA Switching between the punch mode and laser mode is:

0: Invalidated.

1: Validated.

16244**M code for setting the punch mode****16245****M code for setting te laser mode**

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 255

Each of the parameters set the M code for setting the punch mode or laser mode.

The parameters are validated when parameter ALA (No. 16242, #0) is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
16260			TNM	TCL				

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

- #4 TCL** The T axis is:
 0: Not controlled by the CNC machine.
 1: Controlled the CNC machine.

NOTE

- When this parameter is set to 1, the T axis should be set as a rotation axis. (bit 0 (ROTx) of parameter No. 1006 is set to 1)
- When this parameter is set to 1, setting the following parameters allows the CNC to be compatible with FANUC Series 0i-PD, as follows:

Setting	Meaning
Bit 0 (ROTx) of parameter No.1006=1	Rotary axis (A type)
Bit 1 (ROSx) of parameter No.1006=0	
Bit 0 (ROAx) of parameter No.1008=1	The rotary axis roll-over is valid.
Bit 1 (RABx) of parameter No.1008=0	The axis rotates in the direction in which the distance to the target is shorter.
Bit 2 (RRLx) of parameter No.1008=1	Relative coordinates are rounded by the amount of the shift per one rotation

- #5 TNM** When machine lock signal MLK and the TNG signal for ignoring a T command are on, whether the number following address T is cataloged as a tool number is:
 0: Not checked.
 1: Checked.

NOTE

Generally, the tool number is not checked when the TNG signal is set to "1".

	#7	#6	#5	#4	#3	#2	#1	#0
16263	NDA					TOF		

[Input type] Parameter input

[Data type] Bit path

- #2 TOF** The function for compensating the tool position is:
 0: Invalidated.
 1: Validated.
 Specify a tool position compensation value on the tool input screen.

#7 **NDA** If a T command is specified in normal direction mode:

0: An alarm is issued (alarm PS 4606).

1: An alarm is not issued.

If a multi-tool command is specified, however, an alarm is issued.

16265

Total number of tools to be used

[Input type] Parameter input

[Data type] Word path

[Valid data range] 0 to 136

This parameter specifies the total number of tools to be used by the tool function. If T-axis control is selected (TCL bit (bit 4 of parameter No. 16260) is set to 1), the total number should include the number of tools for which T-axis control (turret index) is not executed. This parameter can be specified on the tool input screen. The tool numbers to be used should be specified on the tool input screen.

16266

Number of tools for which T-axis control is executed

[Input type] Parameter input

[Data type] Word path

[Valid data range] 0 to 136

This parameter specifies the number of tools for which T-axis control (turret index) is executed. The parameter can be specified on the tool input screen.

NOTE

This parameter is valid when the TCL bit (bit 4 of parameter No. 16260) is set to 1.

16267

Reference-position tool number under T-axis control

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 99999999

This parameter specifies the tool number to be selected upon reference position return for the T-axis. The parameter can be specified on the tool input screen.

NOTE

This parameter is valid when the TCL bit (bit 4 of parameter No. 16260) is set to 1.

16269

Punching count for all tools (low-order)

16270

Punching count for all tools (high-order)

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 99999999

These parameters preset the punching count for all tools to be used. Parameter No. 16269 and No. 16270 can be preset on the tool input screen.

16360

#7	#6	#5	#4	#3	#2	#1	#0
		CIP					

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

#5 CIP In G01, G02, and G03 modes, a C-axis command is:

0: Invalidated.

1: Validated.

	#7	#6	#5	#4	#3	#2	#1	#0
16362			CRM					

[Input type] Parameter input

[Data type] Bit path

#5 CRM According to a T command, the machine is:

0: Moved along the C-axis to the reference point.

1: Not moved along the C-axis to the reference point.

	#7	#6	#5	#4	#3	#2	#1	#0
16363								G91

[Input type] Parameter input

[Data type] Bit path

#0 G91 For C-axis control, a G91 command is:

0: Disabled

1: Enabled

16370	Number of tool 1 for which C-axis control can be executed
16371	Number of tool 2 for which C-axis control can be executed
16372	Number of tool 3 for which C-axis control can be executed
16373	Number of tool 4 for which C-axis control can be executed
16374	Number of tool 5 for which C-axis control can be executed
16375	Number of tool 6 for which C-axis control can be executed
16376	Number of tool 7 for which C-axis control can be executed
16377	Number of tool 8 for which C-axis control can be executed
16378	Number of tool 9 for which C-axis control can be executed
16379	Number of tool 10 for which C-axis control can be executed
16380	Number of tool 11 for which C-axis control can be executed

16381	Number of tool 12 for which C-axis control can be executed
16382	Number of tool 13 for which C-axis control can be executed
16383	Number of tool 14 for which C-axis control can be executed
16384	Number of tool 15 for which C-axis control can be executed
16385	Number of tool 16 for which C-axis control can be executed
16386	Number of tool 17 for which C-axis control can be executed
16387	Number of tool 18 for which C-axis control can be executed
16388	Number of tool 19 for which C-axis control can be executed
16389	Number of tool 20 for which C-axis control can be executed

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 99999999

Each of the parameters set the number of a tool for which C-axis control can be executed. C-axis control can be effective for the tool set to this parameter.

16390	C-axis backlash 1
16391	C-axis backlash 2
16392	C-axis backlash 3
16393	C-axis backlash 4
16394	C-axis backlash 5
16395	C-axis backlash 6
16396	C-axis backlash 7
16397	C-axis backlash 8
16398	C-axis backlash 9
16399	C-axis backlash 10
16400	C-axis backlash 11
16401	C-axis backlash 12
16402	C-axis backlash 13
16403	C-axis backlash 14
16404	C-axis backlash 15

16405	C-axis backlash 16
16406	C-axis backlash 17
16407	C-axis backlash 18
16408	C-axis backlash 19
16409	C-axis backlash 20

[Input type] Parameter input

[Data type] Word path

[Unit of data] Detection unit

[Valid data range] -9999 to 9999

Each of these parameters specifies a C-axis backlash for each tool for which C-axis control can be executed (C1-axis backlash under C-axis synchronous control).

The parameter values correspond to the tool numbers specified in parameter Nos. 16370 to 16389, respectively.

NOTE

The parameters are valid when the CBR bit (bit 7 of parameter 16360) is set to 1. When these parameters are valid, the C-axis backlash specified in parameter 1852 is invalid.

16430	C-axis position compensation 1
16431	C-axis position compensation 2
16432	C-axis position compensation 3
16433	C-axis position compensation 4
16434	C-axis position compensation 5
16435	C-axis position compensation 6
16436	C-axis position compensation 7
16437	C-axis position compensation 8
16438	C-axis position compensation 9
16439	C-axis position compensation 10
16440	C-axis position compensation 11
16441	C-axis position compensation 12
16442	C-axis position compensation 13
16443	C-axis position compensation 14
16444	C-axis position compensation 15

16445	C-axis position compensation 16
16446	C-axis position compensation 17
16447	C-axis position compensation 18
16448	C-axis position compensation 19
16449	C-axis position compensation 20

[Input type] Parameter input

[Data type] Real path

[Unit of data] 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)

Each of these parameters specifies a C-axis position compensation value (C1-axis backlash under C-axis synchronous control).

These compensated values correspond to the tool numbers set in parameter Nos. 16370 to 16389.

NOTE

The values validated when parameter MAI (No. 16360, #2) is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
16500								SF0

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

#0 SF0 The safety zone of type:

0: A is used.

1: B is used.

NOTE

When type B is used, punching is inhibited in punch mode and entry is inhibited in laser mode.

	#7	#6	#5	#4	#3	#2	#1	#0
16501					SZ4	SZ3	SZ2	SZ1

[Input type] Parameter input

[Data type] Bit path

#0 to 3 SFj When a safety zone check is executed, in the #j (j=1 to 4) area,

0: An entry is inhibited.

1: Punching is inhibited.

NOTE

When type B of safety zone check is used, this parameter is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
16502				SZI				MDP

[Input type] Parameter input

[Data type] Bit path

- #0 MDP** On the safety zone setting display,
 0: The workpiece coordinate system is indicated.
 1: The machine coordinate system is indicated.

- #4 SZI** Data set on the safety zone setting display is:
 0: Invalidated.
 1: Validated.

	#7	#6	#5	#4	#3	#2	#1	#0
16504							RTZ	

[Input type] Parameter input

[Data type] Bit path

- #1 RTZ** In the safety zone check, setting area of the circle tool figure is:
 0: Disabled.
 1: Enabled.

16505	Positive X coordinate for safety zone 1
16506	Negative X coordinate for safety zone 1
16507	Y coordinate for safety zone 1
16508	Positive X coordinate for safety zone 2
16509	Negative X coordinate for safety zone 2
16510	Y coordinate for safety zone 2
16511	Positive X coordinate for safety zone 3
16512	Negative X coordinate for safety zone 3
16513	Y coordinate for safety zone 3
16514	Positive X coordinate for safety zone 4
16515	Negative X coordinate for safety zone 4
16516	Y coordinate for safety zone 4

[Input type] Parameter input

- [Data type] Real path
 [Unit of data] mm, inch (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)
 Each set of the parameters specify safety zone 1, 2, 3, or 4.

NOTE

- 1 Set four safety zones to be arranged sequentially in the order of #1, #2, #3 and #4 from the origin to positive of the X-axis.
- 2 Zeros must be specified for zones which need not be specified.

16517	Size of tool area 1 in the X direction
16518	Size of tool area 1 in the Y direction
16519	Size of tool area 2 in the X direction
16520	Size of tool area 2 in the Y direction
16521	Size of tool area 3 in the X direction
16522	Size of tool area 3 in the Y direction
16523	Size of tool area 4 in the X direction
16524	Size of tool area 4 in the Y direction
16525	Size of tool area 5 in the X direction
16526	Size of tool area 5 in the Y direction
16527	Size of tool area 6 in the X direction
16528	Size of tool area 6 in the Y direction
16529	Size of tool area 7 in the X direction
16530	Size of tool area 7 in the Y direction
16531	Size of tool area 8 in the X direction
16532	Size of tool area 8 in the Y direction

- [Input type] Parameter input
 [Data type] Real path
 [Unit of data] mm, inch (machine 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))
 (When the increment system is IS-B, 0.0 to +999999.999)

In the safety zone check, each set of the parameters specify tool area 1, 2, 3, 4, 5, 6, 7, or 8.

Tool area 9, 10, 11, or 12 are specified by setting each set of the parameters among Nos.16551 to 16558.

NOTE

A tool area is selected by signals SZTS0 to SZTS3 input from a PMC machine.

16540	Width of workpiece holder 1 along the X-axis for automatic setting of a safety zone
16541	Width of workpiece holder 2 along the X-axis for automatic setting of a safety zone
16542	Width of workpiece holder 3 along the X-axis for automatic setting of a safety zone
16543	Width of workpiece holder 4 along the X-axis for automatic setting of a safety zone

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (machine 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))
(When the increment system is IS-B, 0.0 to +999999.999)

Each of the parameters specifies the width of a workpiece holder along the X-axis for the automatic setting of a safety zone by an external signal.

The parameter values correspond to safety zones 1 to 4, specified in parameter Nos. 16505 to 16516, respectively.

NOTE

When automatic setting is executed, these parameters must be set.

16551	X dimension of tool area 9
16552	Y dimension of tool area 9
16553	X dimension of tool area 10
16554	Y dimension of tool area 10
16555	X dimension of tool area 11
16556	Y dimension of tool area 11
16557	X dimension of tool area 12
16558	Y dimension of tool area 12

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (machine 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))
(When the increment system is IS-B, 0.0 to +999999.999)

In the safety zone check, each set of the parameters specifies tool area 9, 10, 11, or 12. Tool area 1, 2, 3, 4, 5, 6, 7, or 8 are specified by setting each set of the parameters among Nos.16517 to 16532.

NOTE

A tool area is selected by signals SZTS0 to SZTS3 input from a PMC machine.

16610	M code 1 for canceling the Y-axis crack
16611	M code 2 for canceling the Y-axis crack
16612	M code 3 for canceling the Y-axis crack
16613	M code 4 for canceling the Y-axis crack
16614	M code 5 for canceling the Y-axis crack

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 65535

Set an M code for Y-axis crack cancel.

The cancel M-code must be specified in these parameters by different from M-code used for another function. But M02 and M30 are possible.

	#7	#6	#5	#4	#3	#2	#1	#0
16844							OVE	

[Input type] Parameter input

[Data type] Bit axis

#1 OVE Rapid traverse override is:

0: Enabled.

1: Disabled. (The override is fixed to 100% at all times.)

	#7	#6	#5	#4	#3	#2	#1	#0
19500		FNW						

[Input type] Parameter input

[Data type] Bit path

#6 FNW When the feedrate is determined according to the feedrate difference and acceleration in AI advanced preview control:

0: The maximum feedrate at which the allowable feedrate difference and acceleration for each axis are not exceeded is used.

1: A feedrate is determined to satisfy the condition that the allowable feedrate difference and allowable acceleration rate of each axis are not exceeded, and also to ensure that a constant deceleration rate is applied to the same figure regardless of the direction of movement.

	#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 AI advance preview 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 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 AI advance preview 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	0 or 1	
Bit machine group		
Bit path		
Bit axis		
Byte	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Byte machine group		
Byte path		
Byte axis		
Word	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
Word machine group		
Word path		
Word axis		

Data type	Valid data range	Remarks
2-word	0 to ± 999999999	Some parameters handle these types of data as unsigned data.
2-word machine group		
2-word path		
2-word axis		
Real	See the standard parameter setting tables.	
Real machine group		
Real path		
Real axis		

NOTE

- 1 Each of the parameters of the bit, bit machine group, bit path, and bit axis 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.
In the FANUC Series 0i-PD, the maximum number of machine group is 1.
- 3 For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
In the FANUC Series 0i-PD, the maximum number of controlled path is 1.
- 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 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, and real axis 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
inch	IS-A	0.001	-99999.999	to +99999.999
	IS-B	0.0001	-99999.9999	to +99999.9999

(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
inch	IS-A	0.001	0.000	to +99999.999
	IS-B	0.0001	0.0000	to +99999.9999

(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	0.00	to +999000.00
	IS-B	0.001	0.000	to +999000.000
inch/min	IS-A	0.001	0.000	to +96000.000
	IS-B	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
inch/sec ²	IS-A	0.001	0.000	to +99999.999
	IS-B	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

- 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.
- Codes not in this table are ignored if their parity is correct.
- 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

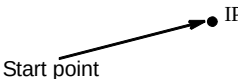
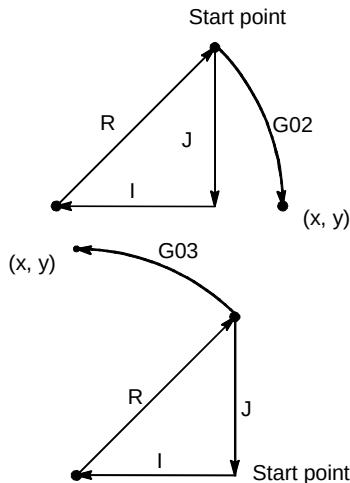
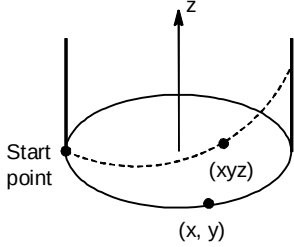
Some functions cannot be used depending on the machine.
For details of command formats, see the relevant sections or subsections.

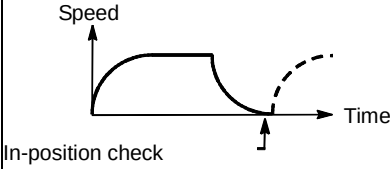
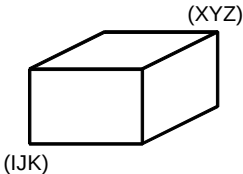
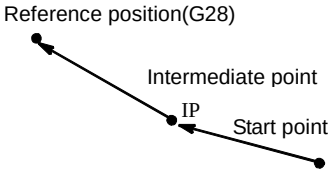
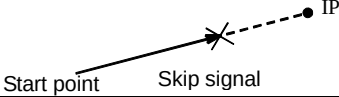
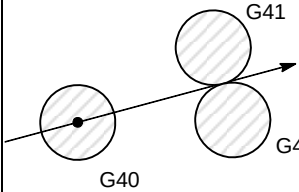
In the list, the following symbols are used:

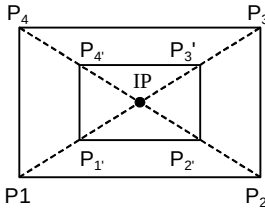
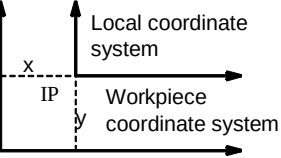
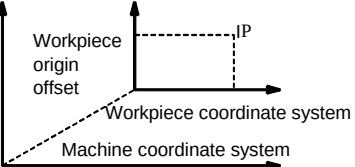
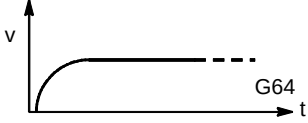
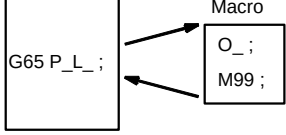
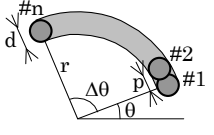
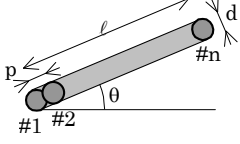
x : 1st basic axis (X),
y : 2nd basic axis (Y),

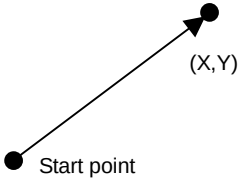
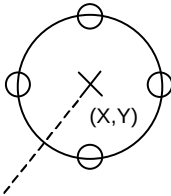
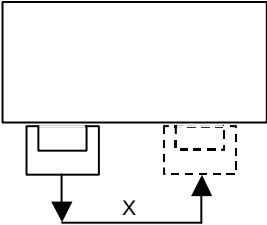
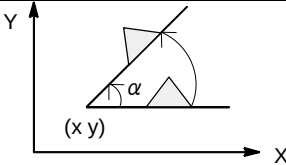
coded using G code system A

- IP_ : presents a combination of arbitrary axis addresses (such as X_ Y_ Z_).

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 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y\_ Z\_ \left\{ \begin{matrix} R\_ \\ J\_ K\_ \end{matrix} \right\} F\_ ;$
Helical interpolation (G02, G03)	 In case of G03 on X-Y plane	G17 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X\_ Y\_ \left\{ \begin{matrix} R\_ \\ I\_ J\_ \end{matrix} \right\} \alpha\_ F\_ ;$ G18 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X\_ Z\_ \left\{ \begin{matrix} R\_ \\ I\_ K\_ \end{matrix} \right\} \alpha\_ F\_ ;$ G19 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y\_ Z\_ \left\{ \begin{matrix} R\_ \\ J\_ K\_ \end{matrix} \right\} \alpha\_ F\_ ;$ α: Arbitrary address except the circular interpolation axis
Dwell (G04)		G04 $\left\{ \begin{matrix} X\_ \\ P\_ \end{matrix} \right\} ;$
AI contour control (G05.1)		G05.1 Q1 ; AI advanced preview control mode on G05.1 Q0 ; AI advanced preview control mode off

Functions	Illustration	Program format
High-speed HRV current control (G05.1)		G05.4 Q1 ; High-speed HRV current control mode on G05.4 Q0 ; High-speed HRV current control mode off
Exact stop (G09)		G09 { G01 G02 G03 } IP_ ;
Programmable data input (G10, G11)		Tool compensation memory A G10 L01 P_ R_ ; Tool data input G10L30 ; Tool data entry mode setting N_P_R_ ; Tool data entry G11 ; : Tool data entry mode 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_ ; G23 ; Cancel
Reference position return (G28)		G28 ; : Return to the reference position without passing through the intermediate point G28 IP_ ; : Return to the reference position via the intermediate point
Automatic setting of the safety zone (G32)	Please refer to the chapter "Automatic Setting of the Safety Zone".	Workpiece holder detection command: G32 X_ F_ P_ Q_ ;
Skip function (G33)		G33 IP_ F_ ;
Bending compensation (G38, G39)	Please refer to the chapter "BENDING COMPENSATION (G38, G39)".	G38 I_ J_ K_ P_ Q_ R_ ; : Bending compensation for X axis direction : G39 I_ J_ K_ P_ Q_ R_ ; : Bending compensation for Y axis direction
Tool radius compensation (G40 to G42)		{ G17 G18 G19 } { G41 G42 } D_ ; D : Tool compensation number G40 : Cancel
Normal direction control (G40.1, G41.1, G42.1) (G150, G151, G152)		G41.1(G151); Normal direction control on: left G42.1(G152); Normal direction control on: right G40.1(G150); Normal direction control cancel
Linear punching command (G45)	Please refer to the chapter "LINEAR PUNCHING COMMAND (G45)".	G45 X_ Y_ P_ Q_ R_ D_ ;

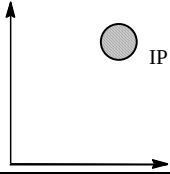
Functions	Illustration	Program format
Circular punching commands (G46,47)	Please refer to the chapter "CIRCULAR PUNCHING COMMANDS (G46, G47)".	G46 X_ Y_ R_ P_ Q_ ; G47 X_ Y_ R_ P_ Q_ ;
Scaling (G50, G51)		G51 X_ Y_ Z_ $\left\{ \begin{matrix} P_ \\ I_ J_ K_ \end{matrix} \right\}$; P, I, J, K : Scaling magnification X, Y, Z : Control position of scaling G50 : Cancel
Local coordinate system setting (G52)		G52 IP_ ;
Command in machine coordinate system (G53)		G53 IP_ ;
Selection of workpiece coordinate system (G54 to G59)		$\left\{ \begin{matrix} G54 \\ G59 \end{matrix} \right\}$ IP_ ;
Cutting mode (G64) Exact stop mode (G61)		G64_ ; Cutting mode G61_ ; Exact stop mode
Automatic corner override (G62)		G62_ ; Automatic corner override
Custom macro (G65, G66, G67)		• One-shot call G65 P_ L_ <Argument assignment> ; P : Program number L : Number of repetition • Modal call G66 P_ L_ <Argument assignment> ; Call after the move command G67 ; Cancel
Circular nibbling (G68)		G68 I_r_ J_θ_ K_Δθ_ P_d_ Q_p_ ;
Linear nibbling (G69)		G69 I_l_ J_θ_ P_d_ Q_p_ ;

Functions	Illustration	Program format
Nibbling by M function		M## ←M code of nibbling mode } } The route and the pitch are } specified according to these } command M\$\$ ←M code of nibbling mode cancel ## : Set value of parameter No.16183 \$\$: Set value of parameter No.16184
Positioning and no press start signal (G70)		G70 X_ Y_ C_ ;
Base point command (G72)		G72 X_ Y_ ;
Multiple-piece machining (G73, G74, G98)	Please refer to the chapter "MULTIPLE-PIECE MACHINING"	G73 } G74 } W_ Q_ ; W : Macro number G98 X_ Y_ I_ P_ J_ K_ ;
Automatic repositioning (G75)		G75 X_ ;
Pattern function (G26, G76, G77, G78, G79, G86, G87, G88, G89)	Please refer to the chapter "PATTERN FUNCTION"	G26_ ; G76_ ; G77_ ; G78_ ; G79_ ; G86_ ; G87_ ; G88_ ; G89_ ;
Coordinate system rotation (G84, G85)	 In case of X-Y plane	G84 { G17 X_ Y_ G18 Z_ X_ G19 Y_ Z_ } R α ; G85 ; : cancel
Absolute/incremental programming (G90/G91)		G90_ ; Absolute programming G91_ ; Incremental programming : G90_ G91_ ; Programming in both modes

C. LIST OF FUNCTIONS AND PROGRAM FORMAT

APPENDIX

B-64554EN/01

Functions	Illustration	Program format
Maximum incremental command value check (G91.1)		G91.1 IP_ ; IP_ ; Maximum incremental value Set 0 to cancel maximum incremental value check.
Change of workpiece coordinate system (G92) Maximum spindle speed clamp (G92)		G92 IP_ ; Change of workpiece coordinate system
Workpiece coordinate system preset (G92.1)		G92.1 IP 0 ;
Feed per minute,	mm/min inch/min mm/rev inch/rev	F_ ; Feed per minute

D RANGE OF COMMAND VALUE

Linear axis

- In case of millimeter input, feed screw is millimeter

	Increment system	
	IS-A	IS-B
Least input increment (mm)	0.01	0.001
Least command increment (mm)	0.01	0.001
Max. programmable dimension (mm)	±999,999.99	±999,999.999
Max. rapid traverse (mm/min) ^{*1}	999,000	999,000
Feedrate range (mm/min) ^{*1}	0.01 to 999,000	0.01 to 999,000
Incremental feed (mm/step)	0.01	0.001
	0.1	0.01
	1.0	0.1
	10.0	1.0
Tool compensation amount (mm) ^{*2}	0 to ±9,999.99	0 to ±9,999.999
Backlash compensation amount (pulses) ^{*3}	0 to ±9,999	0 to ±9,999
Dwell (sec) ^{*4}	0 to 999,999.99	0 to 999,999.999

- In case of inch input, feed screw is millimeter

	Increment system	
	IS-A	IS-B
Least input increment (inch)	0.001	0.0001
Least command increment (inch)	0.001	0.0001
Max. programmable dimension (inch)	±39,370.078	±39,370.0787
Max. rapid traverse (mm/min) ^{*1}	999,000	999,000
Feedrate range (inch/min) ^{*1}	0.001 to 96,000	0.0001 to 9,600
Incremental feed (inch/step)	0.001	0.0001
	0.01	0.001
	0.1	0.01
	1.0	0.1
Tool compensation amount (inch) ^{*2}	0 to ±999.999	0 to ±999.9999
Backlash compensation amount (pulses) ^{*3}	0 to ±9,999	0 to ±9,999
Dwell (sec) ^{*4}	0 to 999,999.99	0 to 999,999.999

- In case of inch input, feed screw is inch

	Increment system	
	IS-A	IS-B
Least input increment (inch)	0.001	0.0001
Least command increment (inch)	0.001	0.0001
Max. programmable dimension (inch)	±99,999.999	±99,999.9999
Max. rapid traverse (inch/min) ^{*1}	96,000	9,600
Feedrate range (inch/min) ^{*1}	0.001 to 96,000	0.0001 to 9,600
Incremental feed (inch/step)	0.001	0.0001
	0.01	0.001
	0.1	0.01
	1.0	0.1
Tool compensation amount (inch) ^{*4}	0 to ±999.999	0 to ±999.9999
Backlash compensation amount (pulses) ^{*3}	0 to ±9,999	0 to ±9,999
Dwell (sec) ^{*4}	0 to 999,999.99	0 to 999,999.999

- In case of millimeter input, feed screw is inch

	Increment system	
	IS-A	IS-B
Least input increment (mm)	0.01	0.001
Least command increment (mm)	0.01	0.001
Max. programmable dimension (mm)	±999,999.99	±999,999.999
Max. rapid traverse (inch/min) ^{*1}	96,000	9,600
Feedrate range (mm/min) ^{*1}	0.01 to 999,000	0.001 to 999,000
Incremental feed (mm/step)	0.01	0.001
	0.1	0.01
	1.0	0.1
	10.0	1.0
Tool compensation amount (mm) ^{*2}	0 to ±9,999.99	0 to ±9,999.999
Backlash compensation amount (pulses) ^{*3}	0 to ±9,999	0 to ±9,999
Dwell (sec) ^{*4}	0 to 999,999.99	0 to 999,999.999

- Rotary axis

	Increment system	
	IS-A	IS-B
Least input increment (deg)	0.01	0.001
Least command increment (deg)	0.01	0.001
Max. programmable dimension (deg)	±999,999.99	±999,999.999
Max. rapid traverse (deg/min) ^{*1}	999,000	999,000
Feedrate range (deg/min) ^{*1}	0.01 to 999,000	0.001 to 999,000
Incremental feed (deg/step)	0.01	0.001
	0.1	0.01
	1.0	0.1
	10.0	1.0
Tool compensation amount (deg) ^{*2}	0 to ±9,999.99	0 to ±9,999.999
Backlash compensation amount (pulses) ^{*3}	0 to ±9,999	0 to ±9,999
Dwell (sec) ^{*4}	0 to 999,999.99	0 to 999,999.999

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 TOOL PATH AT CORNER	793
E.2 RADIUS DIRECTION ERROR AT CIRCLE CUTTING	796

E.1 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.1 (a).

Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

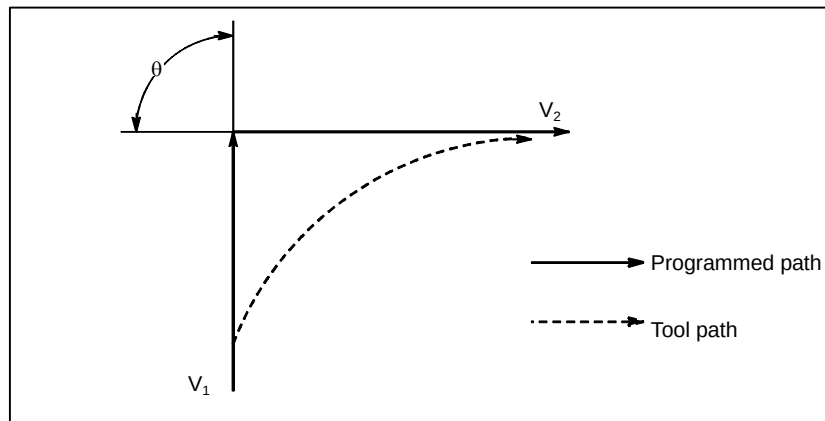


Fig. E.1 (a) Slight deviation between the tool path and the programmed path

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.1 (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.)

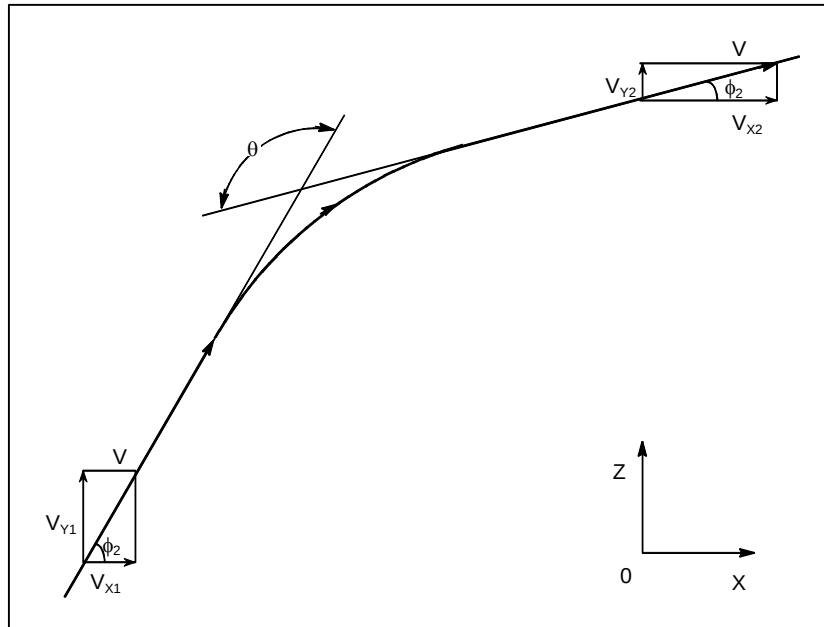


Fig. E.1 (b) Example of tool path

- 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

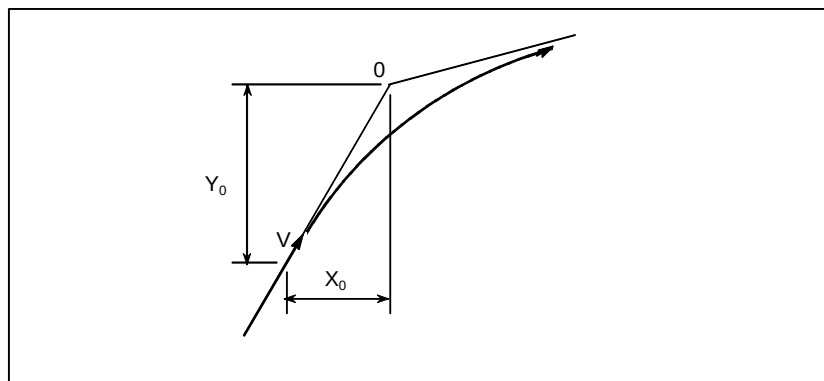


Fig. E.1 (c) Initial value

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.

$$V_x^{(t)} = (V_{x2} - V_{x1}) \left[1 - \frac{V_{x1}}{T_1 - T_2} \left\{ T_1 \exp\left(-\frac{t}{T_1}\right) - T_2 \exp\left(-\frac{t}{T_2}\right) \right\} + V_{x1} \right]$$

$$= V_{x2} \left[1 - \frac{V_{x1}}{T_1 - T_2} \left\{ T_1 \exp\left(-\frac{t}{T_1}\right) - T_2 \exp\left(-\frac{t}{T_2}\right) \right\} \right]$$

$$V_y^{(t)} = \frac{V_{y1} - V_{y2}}{T_1 - T_2} \left\{ T_1 \exp\left(-\frac{t}{T_1}\right) - T_2 \exp\left(-\frac{t}{T_2}\right) \right\} + V_{y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^t V_x(t) dt - X_0$$

$$= \frac{V_{x2} - V_{x1}}{T_1 - T_2} \left\{ T_1^2 \exp\left(-\frac{t}{T_1}\right) - T_2^2 \exp\left(-\frac{t}{T_2}\right) \right\} - V_{x2}(T_1 + T_2 - t)$$

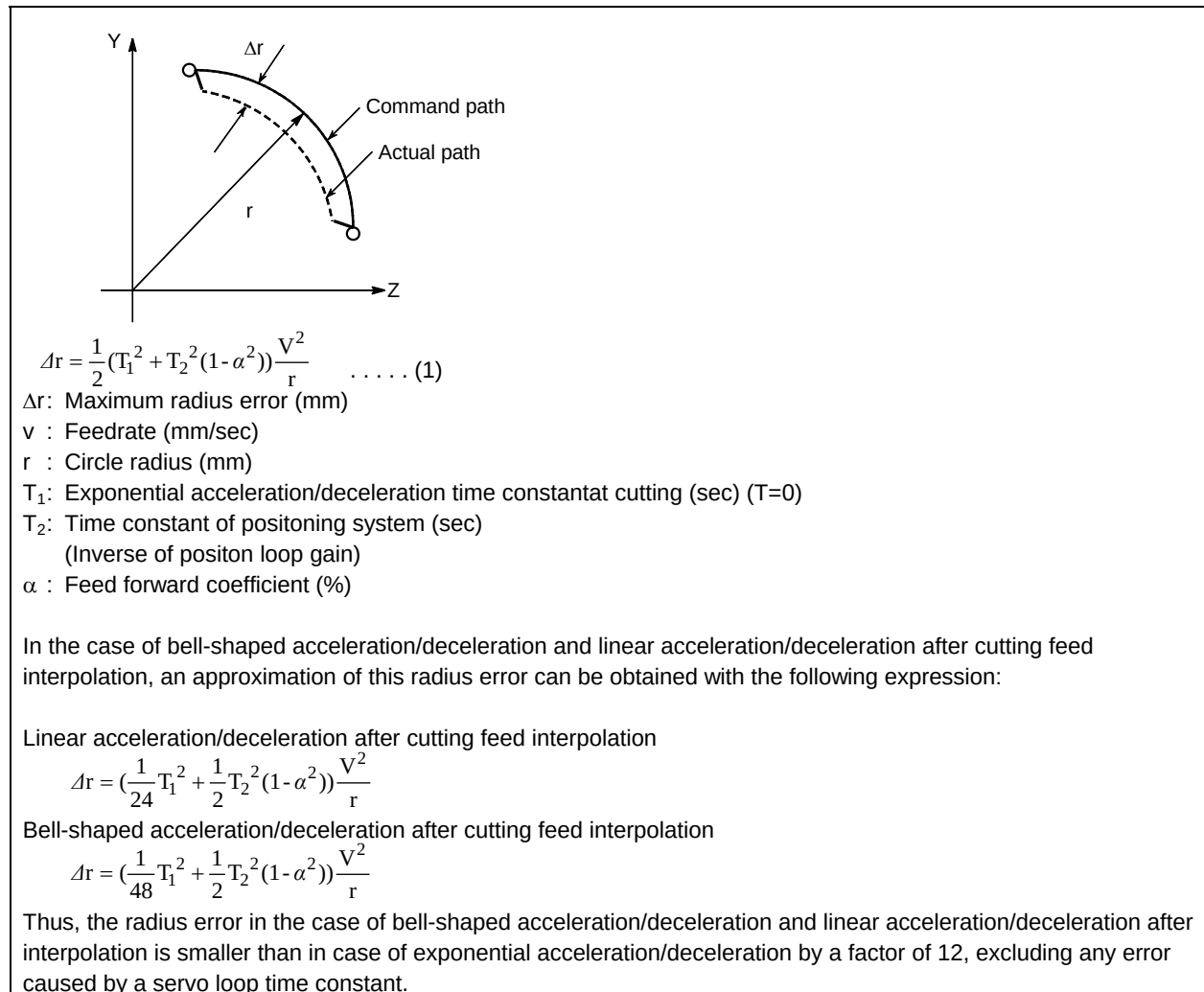
$$Y(t) = \int_0^t V_y(t) dt - Y_0$$

$$= \frac{V_{y2} - V_{y1}}{T_1 - T_2} \left\{ T_1^2 \exp\left(-\frac{t}{T_1}\right) - T_2^2 \exp\left(-\frac{t}{T_2}\right) \right\} - V_{y2}(T_1 + T_2 - t)$$

E.2 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:



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 6)	× (Notes 1, 2, 6)	○ (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)
	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 radius compensation	×	×	×
	Storing a called subprogram	×	× (Note 5)	×
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	×	×	×
	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	×	×	×

NOTE

- 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-64310EN, B-64560EN).
 - Bit 0 (G01) of parameter No. 3402
 - Bit 1 (G18) of parameter No. 3402
 - Bit 2 (G19) of parameter No. 3402
 - Bit 3 (G91) of parameter No. 3402
 - Bit 7 (G23) of parameter No. 3402
 - Parameters No. 3406 to 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 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	799
G.2 FANUC DOUBLE-BYTE CHARACTER CODE TABLE.....	800

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	あ	あ	い	い	う	う	え	え	お	お	か	が	き	ぎ	く	ぐ
0220	け	げ	こ	ご	さ	ざ	し	じ	す	ず	せ	げ	そ	ぞ	た	だ
0240	ち	ぢ	っ	つ	づ	て	で	と	ど	な	に	ぬ	ね	の	は	ば
0260	ば	び	び	び	ふ	ぶ	ふ	へ	べ	な	ほ	ぼ	ぼ	ま	み	む
0280	め	も	ゃ	や	ゆ	ゆ	ょ	よ	り	り	る	れ	ろ	わ	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	拭
03C0	無	視	器	原	登	録	再	処	理	描	画	過	容	編	集	未
03E0	対	相	座	標	示	名	齒	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	鼎	書	個	桁	稼	由	兩	半	逃	底	逆	下	空	四
0420	觸	平	代	辺	格	子	周	心	本	群	停	止	巾	微	狀	路
0440	範	圀	倍	率	注	側	特	殊	距	離	連	続	增	隔	件	初
0460	期	条	經	握	庄	扱	陰	隱	右	押	橫	黃	億	屋	化	何
0480	繪	階	概	該	卷	換	氣	起	軌	技	疑	供	共	境	強	教
04A0	掘	縹	係	傾	型	檢	權	研	肩	見	驗	元	弦	減	孔	巧
04C0	控	更	校	構	根	左	差	雜	參	散	産	算	治	耳	式	失
04E0	修	十	從	勝	商	少	尚	昇	植	色	食	伸	信	侵	振	浸
0500	真	暗	以	意	異	影	銳	越	価	可	科	果	箇	課	各	抃
0520	核	学	掛	漢	簡	觀	関	含	却	客	休	急	業	曲	均	筋
0540	繼	計	輕	言	限	互	降	採	濟	細	姿	思	写	射	斜	者
0560	車	借	縱	重	出	述	術	涉	照	省	章	証	象	身	進	人
0580	図	違	印	沿	遠	央	奧	往	応	会	解	改	割	活	願	基
05A0	奇	寄	岐	既	近	区	矩	驅	偶	旧	求	球	究	級	欠	結
05C0	口	語	誤	交	厚	項	刻	告	黑	財	策	糸	試	資	事	持
05E0	似	釈	弱	受	収	純	順	所	序	剩	場	常	飾	水	錐	据
0600	制	整	製	前	全	然	則	属	即	他	多	存	谷	探	短	徵
0620	鎖	調	頂	鉄	添	頭	同	導	道	熱	年	濃	箱	発	拔	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	搜	太	打	体	待	態
0680	替	段	知	地	致	遲	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	搖	樣	溶	要	抑	良	輪	和	話	梓	節	說	絶
06E0	千	專	淺	旋	総	走	退	台	第	題	卓	室	着	柱	鑄	丁
0700	低	訂	肉	日	白	薄	比	皮	被	非	美	普	伏	步	包	門
0720	問	絡	列	万	利	訳	礼	乱	放	枚	約	練	油	劣	例	郭
0740	戾	冷	垂	緑	紫	許	測	精	効	→	↗	↑	↖	←	↙	↓
0760	↘	Q	○	○	○	■	板	予	〃	家	装	管	粉	等	▽	▽
0780	▽	義	貫	安	α	β	程	抗	張	任	破	損	御	足	守	般
07A0	納	混	丸	汎	固	毎	当	樹	詳	鳥	適	論	額	緣	温	給
07C0	界	績	監	締	護	己	称	壁	脂	料	落	確	認	報	排	性
07E0			判	搬	砥	θ	島		◇	◇	◇	◇	□	〃	〃	■

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0800	阿慰院衛憶菓効乾	哀易羽液臆貨害寒	愛為雨益牡我慨刊	挨維渦馱乙牙街勸	逢緯咀園恩芽垣喚	惡胃浦延穩賀殼官	旭衣瓜援音雅獲寬	宛遺噂演佳介覺干	案医雲炎夏壞較幹	闇井營煙暇廻革感	鞍育映鉛架快樂慣	伊一榮塩歌怪笠敢	依稻永汚河海括飲	偉員泳王火灰滑汗	委因洩岡花皆株環	威飲英冲華貝刈甘
0820																
0840																
0860																
0880																
08A0																
08C0																
08E0																
0900	看喜議拳鏡屈劇犬	緩希菊虚響熊激猷	缶幾詰魚驚君隙絹	肝揮脚亨凝訓潔臬	還机丘享局軍血謙	鑑旗久京極郡月軒	閑棄及競玉刑儉鍵	陷毅吸協勤兄健險	韓祈宮叫錦契兼幻	館季弓挟琴揭券古	岸稀救橋銀敬劍庫	眼徽泣況九景圈戸	岩輝牛狹句莖堅故	顏騎居胸苦警建湖	企擬巨興馳芸憲狐	危犧拒鄉屑迎拳誇
0920																
0940																
0960																
0980																
09A0																
09C0																
09E0																
0A00	顧江国災冊士湿舟	五港穀碎刷姉芝週	午甲酷祭察市縞住	侯稿腰菜撮師捨柔	候絞骨裁擦支煮宿	光綱此載札枝社祝	公考頃際殺死謝縮	勾肯今剂皿私尺熟	喉衡困罪三紙若春	好講婚坂撒詞酒瞬	孝購查阪讚詩首盾	幸郊砂咲賛字授巡	康鉞債昨酸寺需暑	弘香妻索伺磁秋女	拘剛彩錯刺辞習傷	攻克才桜史七臭唱
0A20																
0A40																
0A60																
0A80																
0AA0																
0AC0																
0AE0																
0B00	獎淨陣清占訴東隊	將蒸須盛宣倉俗淹	床錠酢聖尖層卒宅	承職吹声川掃其拓	招唇粹西戰巢揃濯	昭寢遂誓扇争尊託	燒審杉請栓窓村濁	焦森裾静泉草詫奪	笑申澄稅洗騷墮脫	紹神世昔染像妥棚	衝芯是析潜臟耐誰	賞親勢籍船蔵帶嘆	障辛征責銑贈怠担	乘針政跡鮮造滯淡	城震星雪善促袋団	情尽晴舌組息貸彈
0B20																
0B40																
0B60																
0B80																
0BA0																
0BC0																
0BE0																

- 803 -

	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								ガ	ギ	グ	ゲ	ゴ	ザ	ジ	ズ
10A0	ハ	ビ	ブ	ヘ	ホ	パ	ピ	プ	ペ	ポ	ヴ	カ	ケ	ク	グ	ア
10C0	オ	カ	キ	ク	ケ	コ	タ	チ	テ	ト	ツ	ツ	テ	デ	ト	
10E0	ス	ズ	セ	ゼ	ソ	ゾ	ダ	ヂ	ヅ	ヅ	ツ	ツ	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	バ	ビ	フ	ブ	プ	ヘ
1140	ラ	リ	ル	レ	ロ	ワ	ワ	ヲ	メン	ヴ	カ	ケ	ヅ	ヅ	ヅ	ヅ
1160	—	、	。	、	、	、	、	、	、	、	、	、	、	、	、	、
1180	—	、	、	、	、	、	、	、	、	、	、	、	、	、	、	、
11A0	～	∥		...	..	、	、	、	、	、	、	、	、	、	、	、
11C0	}	<	>	《	》	「	」	『	』	【	】	+	—	±	×	÷
11E0	=	≠	<	>	≤	≥	∞	∴	♂	♀	°	′	″	℃	¥	\$
1200	¢	£	%	井	&	*	@	§	☆	★	○	●	◎	◇	◆	□
1220	■	△	▲	▽	▼	※	〒	→	←	↑	↓	=	□	△	◇	□
1240	1/1	2/2	3/3	4/4	5/5	6/6	□	□	mm	cm	km	cm ²	m ²	km ²	cm ³	m ³
1260	mg	kg	cc	dl	ℓ	kl	ms	μs	ns	HP	ps	Hz	ℓ	℃	℃	℃
1280	亜	芦	尉	壺	逸	芋	姻	韻	詠	疫	悦	調	猿	毆	翁	虞
12A0	卸	嫁	禍	彦	蚊	餓	悔	塊	戒	嚇	岳	檜	喝	褐	轄	且
12C0	勘	堪	棺	款	憾	艦	頑	忌	紀	飢	棋	宜	儀	吉	虐	朽
12E0	凶	峽	恭	矯	曉	斤	桐	菌	謹	襟	吟	隈	勲	薰	惠	溪
1300	蚩	鷄	鯨	遣	賢	繭	顯	玄	孤	枯	鼓	吳	悟	碁	后	恒
1320	皇	慌	醇	拷	豪	獄	昆	恨	紺	魂	墾	懇	佐	峻	詐	宰
1340	齋	歲	搾	棧	蚕	慘	暫	氏	司	祉	肢	嗣	飼	雌	賜	諮
1360	児	侍	滋	慈	蠶	疾	執	漆	舍	赦	遮	邪	蛇	勺	酌	爵
1380	寂	朱	珠	儒	蠶	州	宗	拾	愁	酬	醜	汁	銃	獸	叔	爵
13A0	肅	塾	俊	旬	囚	殉	循	潤	遵	庶	如	徐	升	召	匠	肖
13C0	償	抄	昌	晶	准	招	宵	症	祥	底	粧	詔	彰	礁	文	冗
13E0	償	嬢	讓	釀	殖	囑	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	穗	鍾	隨	髓	樞	崇	菅	畝	姓	齊	牲	逝	婿
1420	脆	夕	斥	隻	惜	拙	竊	撰	仙	踐	錢	遷	薦	禪	祖	租
1440	措	疎	壯	莊	桑	曹	喪	葬	僧	遭	槽	燥	藻	霜	憎	賊
1460	孫	馱	舵	胎	泰	逮	瀧	汎	但	丹	胆	鍛	壇	痴	稚	畜
1480	逐	窒	嫡	衷	著	弔	眺	潮	聰	勅	朕	陳	鎮	津	漬	坪
14A0	呈	弟	邸	亭	貞	帝	艇	通	偵	提	迭	哲	徹	澱	斗	渡
14C0	奴	桃	悼	棟	痘	唐	塔	搭	騰	豆	騰	洞	童	匿	篤	屯
14E0	豚	尼	忒	妊	忍	寧	婆	俳	輩	梅	培	陪	賠	伯	迫	漠
1500	鉢	伐	帆	畔	煩	頒	藩	晚	蛩	妃	披	卑	碑	罷	姬	漂
1520	苗	猫	賓	頻	瓶	扶	赴	膚	賦	附	譜	侮	覆	霧	墳	陞
1540	塤	弊	遍	舖	暮	邦	奉	峰	抱	俸	砲	褒	坊	某	冒	貿
1560	謀	朴	牧	墨	掘	奔	凡	盆	麻	妹	埋	又	抹	慢	漫	岬
1580	眠	娘	銘	妄	猛	匆	厄	愉	癰	唯	幽	悠	猶	裕	雄	憂
15A0	羊	庸	窰	擁	謠	翌	羅	齡	吏	吏	痢	履	柳	竜	疏	涼
15C0	獵	陵	僚	糧	厘	倫	零	麗	廉	廉	鍊	垆	露	朗	廊	樓
15E0	賄	國	搖	條	櫻	澤	瀘	碌	鐵	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)	806
(2) Background edit alarms (BG alarm)	806
(3) Communication alarms (SR alarm)	806
(4) Parameter writing alarm (SW alarm)	819
(5) Servo alarms (SV alarm)	819
(6) Overtravel alarms (OT alarm)	824
(7) Memory file alarms (IO alarm)	824
(8) Alarms requiring power to be turned off (PW alarm)	824
(9) Overheat alarms (OH alarm)	825
(10) Other alarms (DS alarm)	825
(11) Malfunction prevention function alarms (IE alarm)	830

(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 bit 0 (TVC) of parameter No. 0000 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	INVALID BREAK POINT OF WORDS	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.
0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0.

Number	Message	Description
0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. 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 or involute interpolation.
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.
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.
0029	ILLEGAL OFFSET VALUE	Illegal offset No.
0030	ILLEGAL OFFSET NUMBER	An illegal offset No. was specified.
0031	ILLEGAL P COMMAND IN G10	Data input for the L No. of G10 or the corresponding function is not enabled. No data setting address such as P or R was 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 compensation. Modify the program.
0034	NO CIRC ALLOWED IN STUP/EXT BLK	In tool radius compensation, a startup or cancellation is performed in the G02 or G03 mode. Modify the program.
0035	CAN NOT COMMANDED G31	- Skip function(G33) cannot be specified. This alarm is generated when a G code (such as for tool radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G33P98 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 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.
0041	INTERFERENCE IN G41/G42	In tool radius compensation, excessive cutting may occur. Modify the program.
0043	ILLEGAL T-CODE COMMAND	On a system with a DRILL-MATE ATC installed, M06 is not specified in a block that specifies a T code. Alternatively, a T code beyond the allowable range is specified.

Number	Message	Description
0060	SEQUENCE NUMBER NOT FOUND	The specified number could not be found for program number and sequence number searches. [External workpiece number search] The program corresponding to the specified workpiece number could not be found.
0070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
0071	DATA NOT FOUND	- The address to be searched was not found. - The program with specified program number was not found in program number search. Check the data.
0072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 400. 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	ILLEGAL PROGRAM NUMBER	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. Not only a P instruction in M98, G65, G66 or interrupt type custom macro, but also the M/G/T codes or a specific code call the subprogram or macro program. 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 bit 6(NPE) of parameter No. 3201 is set to "1". Set bit 6(NPE) of parameter No. 3201 to "0" before executing a match.
0085	OVERRUN ERROR	The next character was received from the I/O device connected to Reader/Puncher interface 1 before it could read a previously received character.
0086	DR OFF	During I/O process by Reader/Puncher 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.

Number	Message	Description
0087	BUFFER OVERFLOW	During a read by Reader/Puncher 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.
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.
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.
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 specified 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.

Number	Message	Description
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	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 and coordinate rotation overflows the data storage. It also is generated during input of the manual intervention amount.
0144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or tool radius-tool nose radius compensation plane must be the same. Modify the program.
0148	SETTING ERROR	Automatic corner override deceleration rate is out of the settable range of judgment angle. Modify the parameters (No.1710 to No.1714).
0199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
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. 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 synchronization 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 (jog feed or incremental feed) 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.
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	A reference 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.
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.
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.
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.
0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program.
0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although bit 0 (IDGx) of parameter No. 1012 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.

Number	Message	Description
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. - Since the one-rotation signal sent from the position detector is not detected, the grid for manual reference position return is not established. (Bit 6 of diagnostic data No. 0201 must be 1.)
0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
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.
0334	OFFSET IS OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (malfunction prevention function)
0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (malfunction prevention function)
0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The maximum control axis number which could be used within a path was exceeded. Check parameter No. 0981 and No. 0982. If this alarm is generated, the emergency stop state cannot be released.
0369	G31 FORMAT ERROR	- No axis is specified or two or more axes are specified in the torque limit switch instruction (G33P98/P99). - The high-speed continuous skip function (G33P90) is not present.
0370	G31P/G04Q ERROR	1) The specified address P value for skip instruction (G33) 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 G33, or Q1-4 for G04 was commanded without a multistage skip function option.
0376	SERIAL DCL: ILLEGAL PARAMETER	1. When bit 1 of parameter No. 1815 is set to "1", parameter No.2002#3 is set to "0" 2. The absolute-position detection function is enabled. (Bit 5 of parameter No. 1815 is set to "1".)
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 "(".
1001	AXIS CONTROL MODE ILLEGAL	Axis control mode is illegal.
1013	ILLEGAL POS. OF PROGRAM NO.	Address O or N is specified in an illegal location (e.g. after a macro statement).
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)

Number	Message	Description
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.
1090	PROGRAM FORMAT ERROR	A lowercase alphabetic character is found in other than an NC program statement comment section, program name, or folder name.
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.
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.
1099	ILLEGAL 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.
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 99999999).
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 contains a character that cannot be used in a variable name.

Number	Message	Description
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	The G10 L No. contains no relevant data input or corresponding option. 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.
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.
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.
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.

Number	Message	Description
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	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 (bit 0 (ROTx) of parameter No. 1006) = 1 and No.1022=0).
1471	ILLEGAL COMMAND IN G40.1 –G42.1	An invalid G code was specified in the normal direction control mode.
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 bit 0 (TVC) of parameter No. 0000 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.
1805	ILLEGAL COMMAND	[I/O Device] An attempt was made to specify an illegal command during I/O processing on an I/O device.
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 were not stopped or in which a command was in execution. 2. When an M code for performing preset with an each axis workpiece coordinate system preset signal was specified, the each axis workpiece coordinate system preset signal was not turned "1". 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/Puncher interface 1 was not detected.
1830	DR OFF(2)	The data set ready input signal DR of the I/O device connected to Reader/Puncher interface 2 turned OFF.
1832	OVERRUN ERROR(2)	The next character was received from the I/O device connected to Reader/Puncher 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/Puncher 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/Puncher interface 2 even though the NC sent a stop code (DC3) during data reception.

Number	Message	Description
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.
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.
4500	REPOSITIONING COMMAND ERROR	(1) M code for clamping a workpiece or M code for releasing the workpiece is not set. (2) A repositioning command(G75) and M/T code is commanded in the same block. (3) A repositioning command(direct command of M code for clamping a workpiece) was specified in the circular interpolation(G02, G03) mode. (4) A repositioning command(direct command of M code for clamping a workpiece) was specified in the coordinate system rotation(G84) mode. (5) A repositioning command(direct command of M code for clamping a workpiece) was specified in the offset(G41, G42) mode.
4502	ILLEGAL COMMAND IN BOLT HOLE	The bolt hole circle (G26) was executed in the following state, • I, J or K was not specified. • $I \leq 0$ • $K = 0$

Number	Message	Description
4503	ILLEGAL COMMAND IN LINE AT ANGLE	The line at angle (G76) was executed in the following state, • I, J or K was not specified. • $K \leq 0$
4504	ILLEGAL COMMAND IN ARC	The arc (G77) was executed in the following state, • I, J, P or K was not specified. • $I \leq 0$ • $K \leq 0$
4505	ILLEGAL COMMAND IN GRID	The grid (G78, G79) was executed in the following state, • I, J, P or K was not specified. • $P \leq 0$ • $K \leq 0$
4506	ILLEGAL COMMAND IN SHARE PROOFS	The shear proof (G86) was executed in the following state, • I, J or P was not specified. • In case of parameter AIP (No.16202#0) = 0 $I < 1.5 \times P$ • In case of parameter AIP (No.16202#0) = 1 $I < P$ • $P = 0$
4507	ILLEGAL COMMAND IN SQUARE	The square (G87) was executed in the following state, • I, J or P was not specified. • $P \leq 0$ • $Q \leq 0$ • $I < 3 \times P$ • $J < 3 \times Q$
4508	ILLEGAL COMMAND IN RADIUS	The radius (G88) was executed in the following state, • I, J, K, P or Q was not specified. • $I \leq 0$ • $Q \leq 0$ • In case of specifying negative value for P $P > 2.0 \times I$ • Commanded arc length $< Q$
4509	ILLEGAL COMMAND IN CUT AT ANGLE	The cut at angle (G89) was executed in the following state, • I, J, P or Q was not specified. • $Q \leq 0$ • $I < Q$
4510	ILLEGAL COMMAND IN LINE-PUNCH	The linear punching (G45) was executed in the following state, • X, Y or P was not specified. • Line length $< 1.5 \times P$ • $P \leq 0$ • $Q \leq 0$
4511	ILLEGAL COMMAND IN CIRCLE-PUNCH	The circular punching (G46,G47) was executed in the following state, • R or Q was not specified. • $R = 0$ • $Q \leq 0$ • Start point = End point • Arc length $< Q$
4520	INHIBITED IN NIBBLING-MODE	T code, M code, G04, G05.1, G28, G70 or G75 was specified in the nibbling mode.
4521	EXCESS NIBBLING MOVEMENT (X,Y)	In the nibbling mode, the X-axis or Y-axis traveling distance was larger than or equal to the limit (No.16188 to 16192)
4522	EXCESS NIBBLING MOVEMENT (C)	In the nibbling mode, the C-axis traveling distance was larger than or equal to the limit (No.16194)

Number	Message	Description
4523	ILLEGAL COMMAND IN CIRCLE-NIBBL	The circular nibbling (G68) was executed in the following state, • I, J, K, P or Q was not specified. • I, J, K, P or Q was out of range. • In case of specifying negative value for P $P \geq 2I$ • $Q > \text{Maximum pitch (parameter No.16186)}$
4524	ILLEGAL COMMAND IN LINE-NIBBL	The linear nibbling (G69) was executed in the following state, • I, J, P or Q was not specified. • I, J, P or Q was out of range. • $Q > \text{Maximum pitch (parameter No.16186)}$
4530	A/B MACRO COMMAND ERROR	(1) M code for clamping a workpiece or M code for releasing the workpiece is not set. (2) A repositioning command(G75) and M/T code is commanded in the same block. (3) A repositioning command(direct command of M code for clamping a workpiece) was specified in the circular interpolation(G02, G03) mode. (4) A repositioning command(direct command of M code for clamping a workpiece) was specified in the coordinate system rotation(G68) mode. (5) A repositioning command(direct command of M code for clamping a workpiece) was specified in the offset(G41, G42) mode.
4531	U/V MACRO FORMAT ERROR	An attempt was made to store a macro while storing another macro using a U or V macro. A V macro was specified although the processing to store a macro was not in progress. A U macro number and V macro number do not correspond with each other.
4532	IMPROPER U/V MACRO NUMBER	The number of an inhibited macro was specified in a U or V macro command.
4533	U/V MACRO MEMORY OVERFLOW	An attempt was made to store too many macros with a U or V macro command.
4534	W MACRO NUMBER NOT FOUND	Macro number W specified in a U or V macro command is not stored.
4536	NO W,Q COMMAND IN MULTI-PIECE	W or Q was not specified in the command for taking multiple workpieces.
4537	ILLEGAL Q VALUE IN MULTI-PIECE	In the command for taking multiple workpieces, Q is set to a value beyond the range from 1 to 4.
4538	W NO. NOT FOUND IN MULTI-PIECE	Macro number W specified in the command for taking multiple workpieces is not stored.
4539	MULTI-PIECE SETTING IS ZERO	The command for taking multiple workpieces was specified although zero is specified for the function to take multiple workpieces (No. 15649 or signals MLP1 and MLP2 (PMC address G231, #0 and #1)).
4540	MULTI-PIECE COMMAND WITHIN MACRO	The command for taking multiple workpieces was specified when a U or V macro was being stored.
4542	MULTI-PIECE COMMAND ERROR	Although G98 P0 was specified, the G73 command was issued. Although G98 K0 was specified, the G74 command was issued.
4543	MULTI-PIECE Q COMMAND ERROR	Although G98 P0 was specified, the Q value for the G74 command was not 1 or 3. Although G98 P0 was specified, the Q value for the G73 command was not 1 or 2.

Number	Message	Description
4544	MULTI-PIECE P/R COMMAND ERROR	(1) In the command for resuming taking multiple workpieces, the resume position (P) is set to a value beyond the range from 1 to total number of workpieces to be machined. (2) In the command for end position of taking multiple workpieces, the end position (R) is set to a value beyond the range from 1 to total number of workpieces to be machined. (3) In the command for end position of taking multiple workpieces, the end position (R) is set to a smaller value than the resume position (P).
4600	T,C COMMAND IN INTERPOLATION	In the linear interpolation(G01) mode or circular interpolation (G02,G03) mode, a T command or C-axis command was specified.
4601	INHIBITED T,M COMMAND	In the block of G10, G22, G23, G52, G53, G72, G73, G74, G75, G92, G98 a T or M command was specified.
4602	ILLEGAL T-CODE	The specified T command is not cataloged on the tool register screen.
4604	ILLEGAL AXIS OPERATION	A C-axis command was specified in the block containing a T command for multiple tools.
4606	INHIBITED T COMMAND	A T command was specified during normal line control.
4610	RAM AXIS POSITION IS WRONG	Ram-axis is not located on the press start position.
4630	ILLEGAL COMMAND IN LASER MODE	In the laser mode, a nibbling command or pattern command was specified. In gap control mode, an attempt was made to make a switch to the punch mode.
4650	IMPROPER G-CODE IN OFFSET MODE	In the cutter compensation mode, an inhibited G code was specified.
4751	CAN NOT COMMAND C AXIS	The C axis was commanded at the C axis command invalid block.
4872	AUTO SETTING COMMAND ERROR	M code, S code, T code is specified with safety zone auto setting command(G32). G32 is specified in the nibbling mode, in the cutter compensation, in the rotation mode or the scaling mode.
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 system. 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. 1422 for each axis has been set to 0. The parameter for the maximum cutting feedrate for each axis is No. 1432 if acceleration/deceleration before interpolation is enabled and No. 1430 otherwise.
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. The parameter is No. 1432 if acceleration/deceleration before interpolation is enabled and No. 1430 otherwise.
5064	DIFFERENT AXIS UNIT	Circular interpolation has been specified on a plane consisting of axes having different increment systems.

Number	Message	Description
5065	DIFFERENT 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	An unusable G code was specified in the AI advanced preview control mode.
5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting bit 2 of parameter No. 1819 is set to "1". Move the machine to reference position by manual operation and execute manual reference return.
5257	G41/G42 NOT ALLOWED IN MDI MODE	Tool radius compensation was specified in MDI mode. (Depending on the setting of the bit 4 (MCR) of parameter No. 5008)
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.
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.
5379	WRITE PROTECTED TO SLAVE AXIS	A parameter cannot be set directly for a slave axis of axis synchronous control.
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 (bit (PWE) of parameter No. 8000 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 synchronization control, the amount of compensation for synchronization exceeded the parameter No. 8325 setting value. This alarm occurs for a master or slave axis.
SV0004	EXCESS ERROR (G31)	The amount of positional deviation during torque limit skip (G33P98, G33P99) command operation exceeded the limit value of the parameter No.6287.
SV0005	SYNC EXCESS ERROR (MCN)	In feed axis synchronization 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.

Number	Message	Description
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).
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.
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.

Number	Message	Description
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. - The correct axis control card is not attached. - 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.
SV0409	DETECT ABNORMAL TORQUE	An abnormal load was detected on the servo motor, or during Cs axis or spindle positioning. 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.
SV0417	ILL DGTL SERVO PARAMETER	A digital serve parameter setting is incorrect. When bit 4 of diagnosis data No. 203 is 1, an illegal parameter was detected by the servo software. Identify the cause with reference to diagnosis data No. 352. When bit 4 of diagnosis data No. 203 is 0, the CNC software detected an illegal parameter. Probable causes are given below (see diagnosis data 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 synchronization 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. 2118.
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.

Number	Message	Description
SV0431	CNV. OVERLOAD	Power Supply (PS) : Overheat Servo Amplifier (SV) : Overheat
SV0432	CNV. LOW VOLT. CONTROL	Power Supply (PS) : The control power supply voltage has dropped. Servo Amplifier (SV) : The control power supply voltage has dropped.
SV0433	CNV. LOW VOLT. DC LINK	Power Supply (PS) : Low DC link voltage Servo Amplifier (SV) : Low DC link voltage
SV0434	INV. LOW VOLT CONTROL	Servo Amplifier (SV) : Low control power voltage
SV0435	INV. LOW VOLT DC LINK	Servo Amplifier (SV) : Low DC link voltage
SV0436	SOFTTHERMAL(OVC)	The digital servo software detected a software thermal (OVC).
SV0437	CNV. OVERCURRENT	Power Supply (PS) : Overcurrent on input circuit section.
SV0438	INV. ABNORMAL CURRENT	Servo Amplifier (SV) : Motor overcurrent
SV0439	CNV. OVER VOLT. DC LINK	Power Supply (PS) : The DC link voltage is too high. Servo Amplifier (SV) : The DC link voltage is too high.
SV0440	CNV. EXCESS-REGENERATION2	Power Supply (PS) : Excessive generative discharge Servo Amplifier (SV) : 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. PRE-CHARGE FAILURE	Power Supply (PS) : The spare charge circuit for the DC link is abnormal.
SV0443	CNV. INTERNAL FAN FAILURE	Power Supply (PS) : Internal cooling fan failure. Servo Amplifier (SV) : Internal cooling fan failure.
SV0444	INV. INTERNAL FAN FAILURE	Servo Amplifier (SV) : 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 (SV) : The IPM (Intelligent Power Module) detected an alarm.
SV0453	SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the built-in 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.

Number	Message	Description
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 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	Servo Amplifier (SV) : DC link overcurrent.
SV0601	INV. EXTERNAL FAN FAILURE	Servo Amplifier (SV) : External radiator cooling fan failure.
SV0602	INV. OVERHEAT	Servo Amplifier (SV) : The servo motor has overheated.
SV0603	INV. IPM ALARM(OH)	Servo Amplifier (SV) : The IPM (Intelligent Power Module) detected an overheat alarm.
SV0604	AMP COMMUNICATION ERROR	Servo Amplifier (SV) : The communication between Servo amplifier (SV) and Power Supply (PS) is in error.
SV0605	CNV. EXCESS-REGENERATION1	Power Supply (PS) : The motor regenerative power is too much.
SV0606	CNV. EXTERNAL FAN FAILURE	Power Supply (PS) : External radiator cooling fan failure.
SV0607	CNV. IMPROPER INPUT POWER	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. - Parameter No. 1023 (servo axis number of each axis) is set to a negative value or a duplicate value. - The settings for parameter No. 1023 (servo axis number of each axis) were made with a certain setting skipped among 1 to 6, 9 to 14, or 17 to 22. - A setting of a multiple of 8 or a multiple of 8 minus 1 was made.
SV1055	ILLEGAL TANDEM AXIS	In tandem control, the setting of the parameter No. 1023 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.
SV5134	FSSB:OPEN READY TIME OUT	In the initialization, the FSSB could not be in an open ready state. 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.

Number	Message	Description
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 FSSB could not be opened although the CNC permitted the opening of the FSSB. Check the connection between the CNC and the amplifier.

(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.
OT0503	- OVERTRAVEL (SOFT 2)	Exceeded the negative side stored stroke check 2.
OT0504	+ OVERTRAVEL (SOFT 3)	Exceeded the positive side stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	Exceeded the negative side stored stroke check 3.
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.
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.

(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.

Number	Message	Description
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: - 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: - The parameter No.3021 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: - During the set of parameter No. 3012, the skip signal of the X address was not assigned correctly. - 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: - When the number of pitch error compensation points on the axis on which slope compensation is executed exceeds 128 between the most negative side and most positive side - When the size relationship between the slope compensation point Nos. is incorrect - When the slope compensation point is not located between the most negative side and most positive side of pitch error compensation - When the compensation per compensation point is too small or too great.
PW4549	ILLEGAL TOOL DATA FORMAT	The size of the registered tool data patterns has exceeded the upper limit of 32KB.

(9) Overheat alarms (OH alarm)

Number	Message	Description
OH0700	LOCKER OVERHEAT	CNC cabinet overheat
OH0701	FAN MOTOR STOP	PCB cooling fan motor abnormality

(10) Other alarms (DS alarm)

Number	Message	Description
DS0001	SYNC EXCESS ERROR (POS DEV)	In feed axis synchronization 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 for the master or 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.
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.

Number	Message	Description
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.
DS0016	SERIAL DCL:FOLLOW-UP ERROR	(1) The settings of parameters Nos.1883 and 1884 fall outside the range. (2) The current position at establishment of the origin 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 origin exceeded the setting of parameter No. 14010.
DS0018	SERIAL DCL:MISMATCH(SSYNC CTRL)	Of the master and slave axes for feed axis synchronous control, one axis is a linear scale with the origin and the other is not a linear scale with the origin. In such a configuration, the feed axis synchronous control selection signal (SYNC<Gn138> or SYNCJ <Gn140>) needs to be set to 0 to establish the origin.
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.
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.
DS0050	TOO MANY SIMULTANEOUS AXES	A movement was performed along more axes than can be controlled by simultaneous axis control. Check whether a command in the program is specified for more axes than can be controlled by simultaneous axis control.
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 a pot No. or 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.

Number	Message	Description
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. The battery or cable is thought to 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.
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.
DS0608	INV. COOLING FAN FAILURE	Servo Amplifier (SV) : Internal agitating fan failure.
DS0609	INV. RADIATOR FAN FAILURE	Servo Amplifier (SV) : Radiator cooling fan failure.
DS0610	CNV. INTERNAL FAN FAILURE	Power Supply (PS) : Internal agitating fan failure. Servo Amplifier (SV) : Internal agitating fan failure.
DS0611	CNV. EXTERNAL FAN FAILURE	Power Supply (PS) : External radiator cooling fan failure.
DS0612	CNV. OVERLOAD	Power Supply (PS) : Overheat Servo Amplifier (SV) : Overheat
DS0613	CNV. IMPROPER INPUT POWER	Power Supply (PS) : Input power supply fault
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.

Number	Message	Description
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. - The absolute-position detection function is enabled. - Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0. - Parameters 1821 and 1882 have identical settings. - The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting. - 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 parameters Nos.1821 and 1882 setting value.
DS1450	ZERO RETURN NOT FINISHED	1st reference position return (CDxX7 to CDxX0: 17h (Hex)) was specified when the manual reference position return was not executed with the reference position return function enabled (bit 0 (ZRN) of parameter No. 1005 set to "0").
DS1451	IMPROPER PMC AXIS COMMAND	The PMC axes cannot be controlled in this state.
DS4603	C AXIS SYNCHRONOUS ERROR	The difference between the position deviation value of C1 axis and C2 axis exceeds the parameter value (No. 16364, 16365) with the C axis synchronous control function.
DS4605	NEED C AXIS REFERENCE RETURN	The C axis synchronization is not done normally.
DS4750	TOOL OFFSET CANNOT BE EXECUTED	The state of mirror image is different between the time when look-ahead of a block for tool offset was performed and the time when execution of the block was started, so tool offset cannot be executed.
DS4752	C AXIS OFFSET CANNOT BE EXECUTED	The state of mirror image is different between the time when look-ahead of a block for C axis offset was performed and the time when execution of the block was started, so C axis offset cannot be executed.
DS4800	ZONE : PUNCHING INHIBITED 1	When a safety zone check was executed, a punch command was specified in area 1 where punching is inhibited.
DS4801	ZONE : PUNCHING INHIBITED 2	When a safety zone check was executed, a punch command was specified in area 2 where punching is inhibited.
DS4802	ZONE : PUNCHING INHIBITED 3	When a safety zone check was executed, a punch command was specified in area 3 where punching is inhibited.
DS4803	ZONE : PUNCHING INHIBITED 4	When a safety zone check was executed, a punch command was specified in area 4 where punching is inhibited.
DS4810	ZONE : ENTERING INHIBITED 1 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 1 into which entry is inhibited.

Number	Message	Description
DS4811	ZONE : ENTERING INHIBITED 1 -X	When a safety zone check was executed, the machine moving in the negative X direction entered area 1 into which entry is inhibited.
DS4812	ZONE : ENTERING INHIBITED 2 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 2 into which entry is inhibited.
DS4813	ZONE : ENTERING INHIBITED 2 -X	When a safety zone check was executed, the machine moving in the negative X direction entered area 2 into which entry is inhibited.
DS4814	ZONE : ENTERING INHIBITED 3 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 3 into which entry is inhibited.
DS4815	ZONE : ENTERING INHIBITED 3 -X	When a safety zone check was executed, the machine moving in the negative X direction entered area 3 into which entry is inhibited.
DS4816	ZONE : ENTERING INHIBITED 4 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 4 into which entry is inhibited.
DS4817	ZONE : ENTERING INHIBITED 4 -X	When a safety zone check was executed, the machine moving in the negative X direction entered area 4 into which entry is inhibited.
DS4830	ZONE : ENTERING INHIBITED 1 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 1 into which entry is inhibited.
DS4831	ZONE : ENTERING INHIBITED 1 -Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 1 into which entry is inhibited.
DS4832	ZONE : ENTERING INHIBITED 2 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 2 into which entry is inhibited.
DS4833	ZONE : ENTERING INHIBITED 2 -Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 2 into which entry is inhibited.
DS4834	ZONE : ENTERING INHIBITED 3 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 3 into which entry is inhibited.
DS4835	ZONE : ENTERING INHIBITED 3 -Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 3 into which entry is inhibited.
DS4836	ZONE : ENTERING INHIBITED 4 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 4 into which entry is inhibited.
DS4837	ZONE : ENTERING INHIBITED 4 -Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 4 into which entry is inhibited.
DS4870	AUTO SETTING FEED ERROR	The feed rate of safety zone auto setting is other than the parameter value (No. 16538, No. 16539).
DS4871	AUTO SETTING PIECES ERROR	In safety zone auto setting, the safety zone pieces are not correct. Or the position detector has gone wrong, please tell your machine tool builder.
DS5340	PARAMETER CHECK SUM ERROR	Since a parameter was changed, the parameter checksum did not match the reference checksum. Set the original value to the parameter or set the reference checksum again.

(11) 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.
IE0008	ILLEGAL ACC/DEC	The malfunction prevention function detected the acceleration/deceleration error.

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	831
I.2	NAMING RULES.....	840
I.3	RULES OF CHARACTERS IN PROGRAM FILE	841
I.4	ERROR MESSAGE AND NOTE.....	843

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.

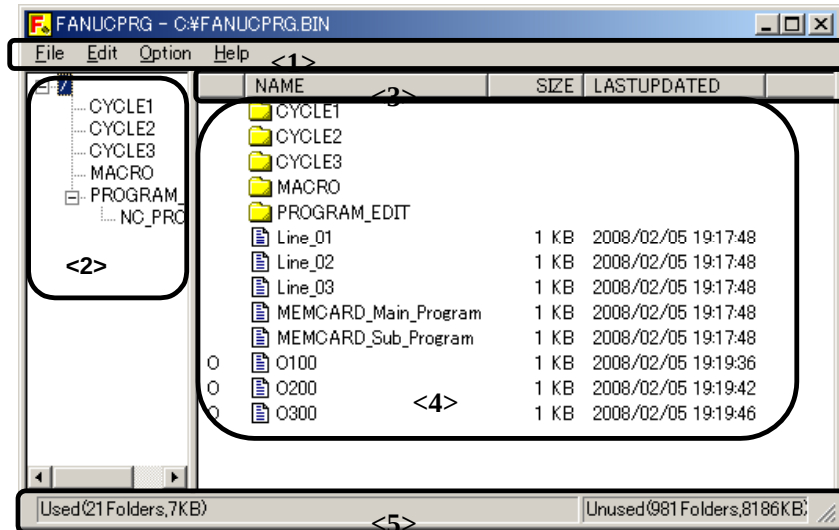
I.1.2 List of Functions of PC Tool

- 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

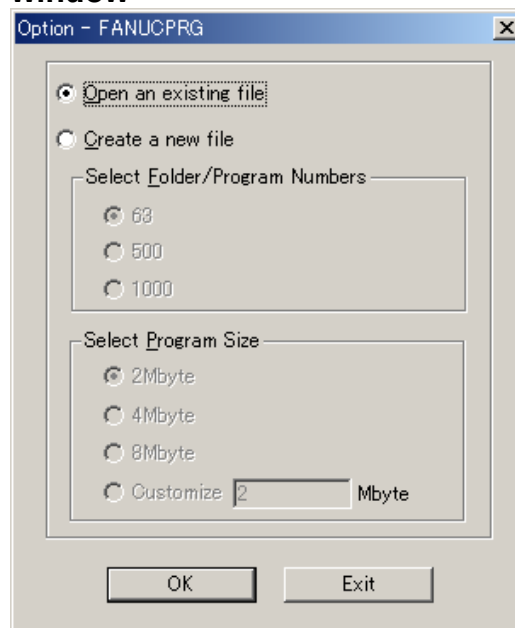
I.1.3 Explanation of Operations

- 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

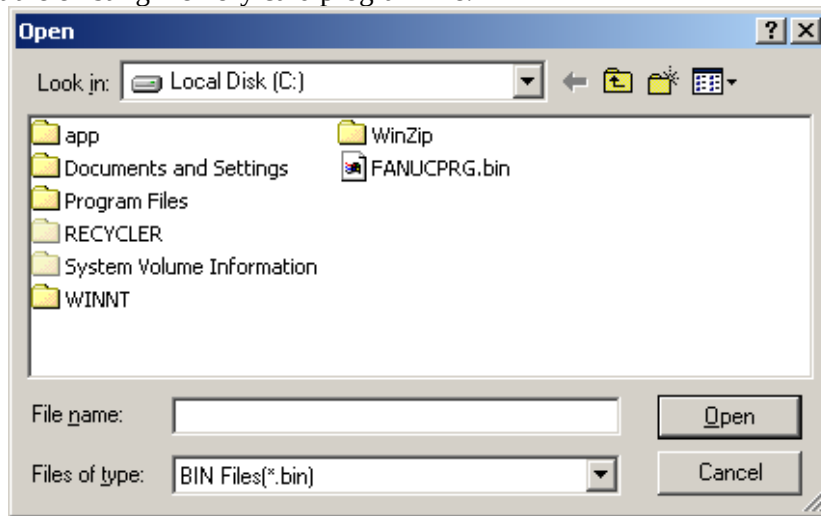


When this PC tool starts up, the Option dialogue window is displayed. Please select "Open an existing file" or "Create a new file".

- **When "Open an existing file" is selected**

After OK button pushed, "Open" dialogue window is displayed.

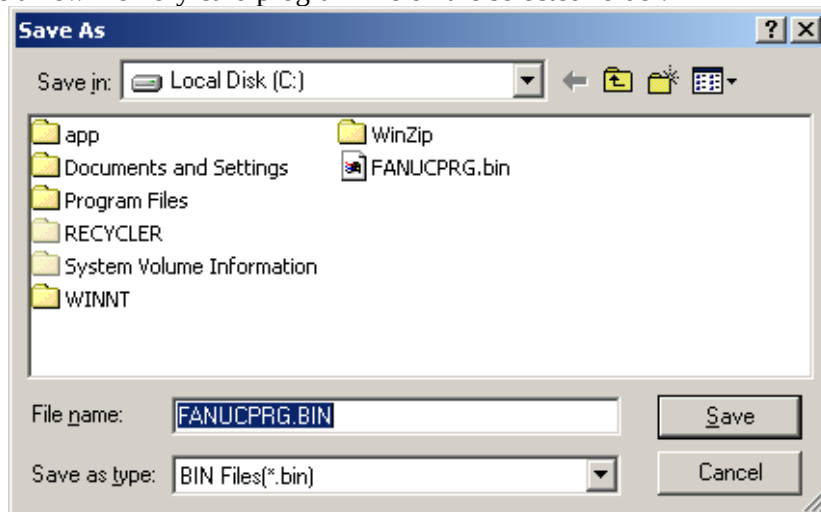
Please select the existing memory card program file.



- **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 Series 0i-PD, select 63.

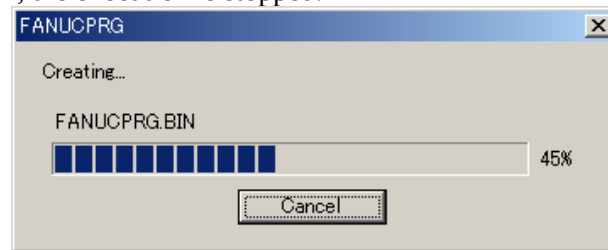
"Program Size" can be selected among 2Mbyte, 4Mbyte, 8Mbyte, and Customize. The default value is 2Mbyte.

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.

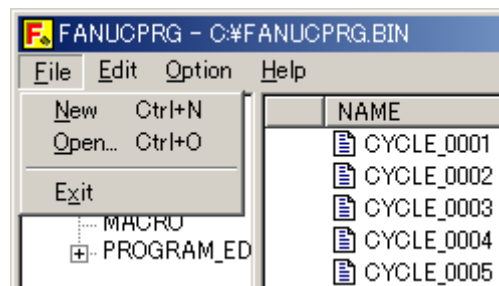
During creating of the memory card program file, the progress bar is being displayed. The progress bar is also displayed during drag-in or drag-out.

If you push [Cancel] button, the execution is stopped.



- Menu

File menu



[New]

Create a new memory card program file.

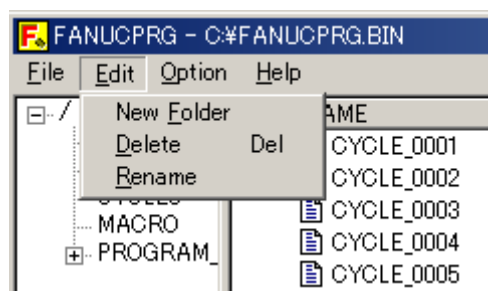
[Open...]

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)

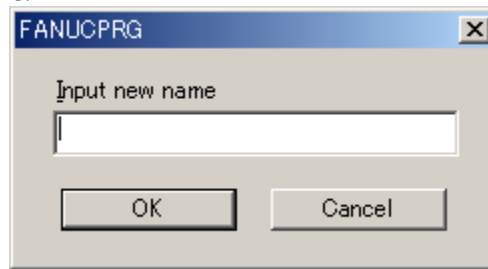
[Delete]

Delete program files or folders.

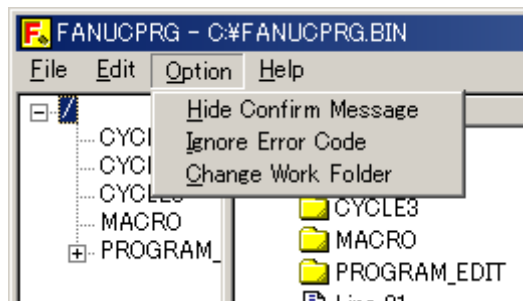
If you delete a folder, all folders and program files in the folder will be deleted.

[Rename]

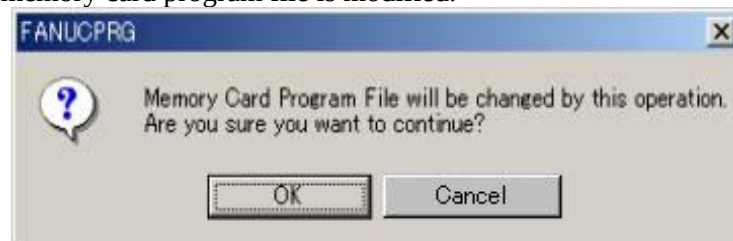
Rename a folder or file.

**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.

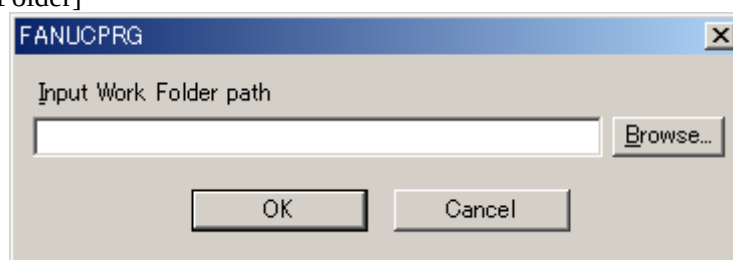
If the [Cancel] button is pushed, the operation is abandoned.

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.

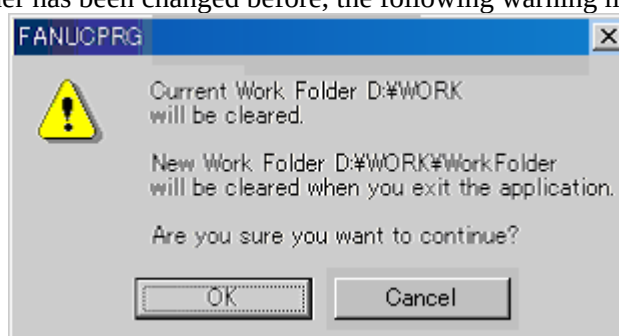
When you start this PC tool, and then try to specify a new work folder for the first time, the work folder provided by the system has been set. The text box in the dialogue is therefore blank. When you change the work folder, the following warning message is popped up.



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.

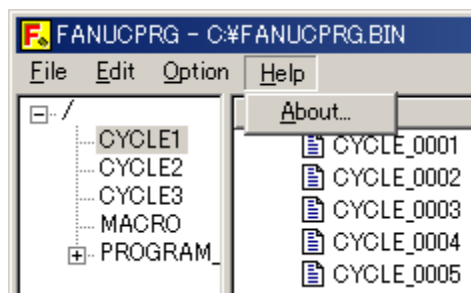


If the [OK] button is pushed, the operation advances.

If the [Cancel] button is pushed, the operation is abandoned.

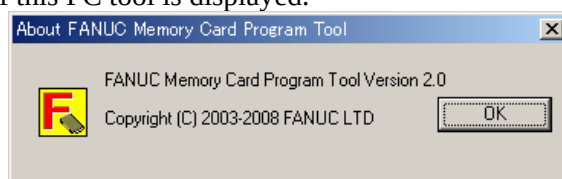
 WARNING

- 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.



[About...]

Version number of this PC tool is displayed.



- Mouse Operation

[Drag-in and Drag-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 "Oxxxxxxx" or "<xxxx>" exists at the top of files, These "Oxxxxxxx" and "<xxxx>" become NC program name. If not exist, each file name becomes NC program name.

Examples			
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

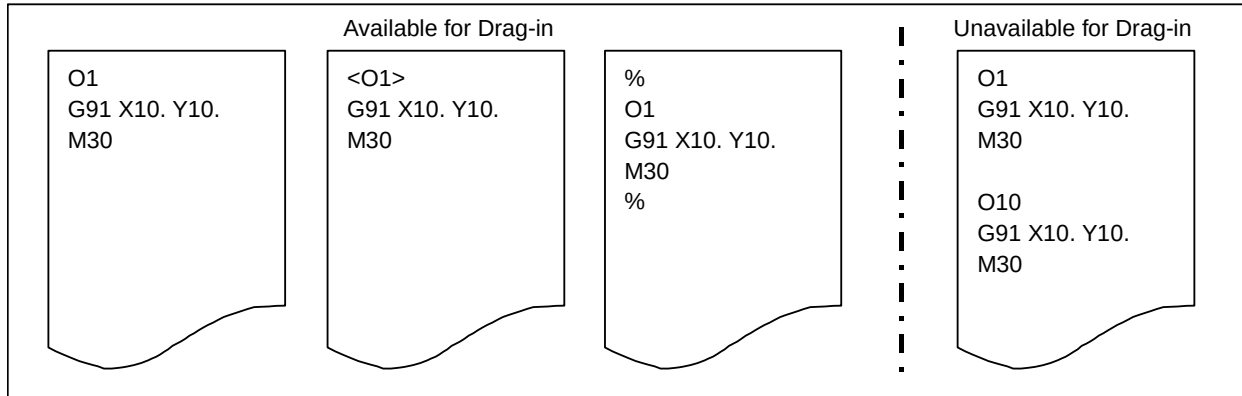
NOTE

- 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 Series 0i-PD are 1 to 9999. The Series 0i-PD 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.



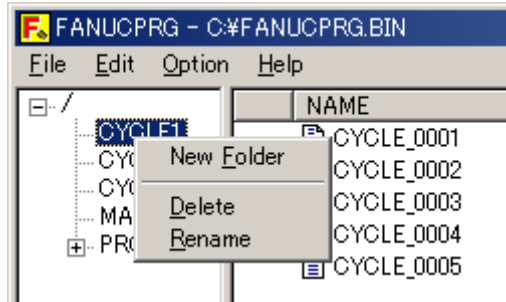
CAUTION

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



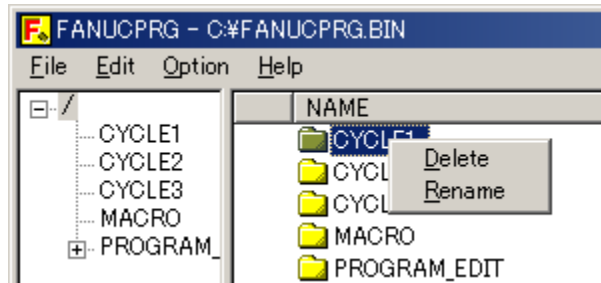
Clicking "New Folder", a new folder is created on selected folder.

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.

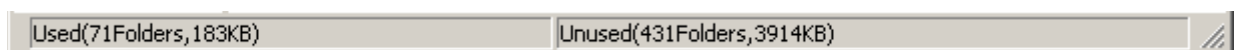
- Focus on List view



- 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.



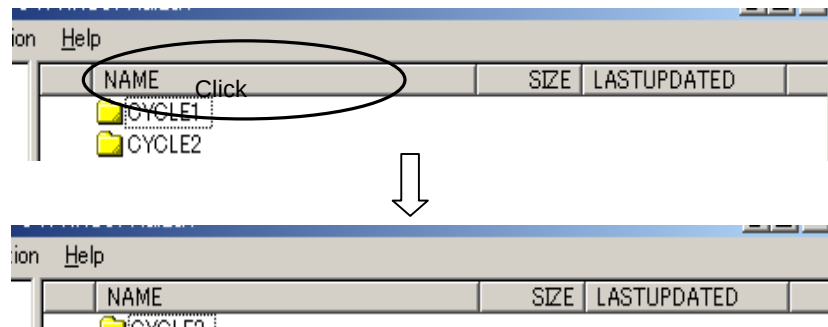
When a new memory card program file is created, two reserved folder are also created. Therefore, the number of used folder is two. However it does not show that number of unused folder is reduced.

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 99999999

(The program numbers allowed in the Series Oi-PD 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" Characters following the "O" contains alphabet.

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
2c	,	46	F	62	b		
2d	-	47	G	63	c		
2e	.	48	H	64	d		

Code	Character	Code	Character	Code	Character	Code	Character
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 Series 0i-PD 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 Series 0i-PD cannot use that memory card program file.

J ISO/ASCII CODE CONVERSION TOOL

Overview

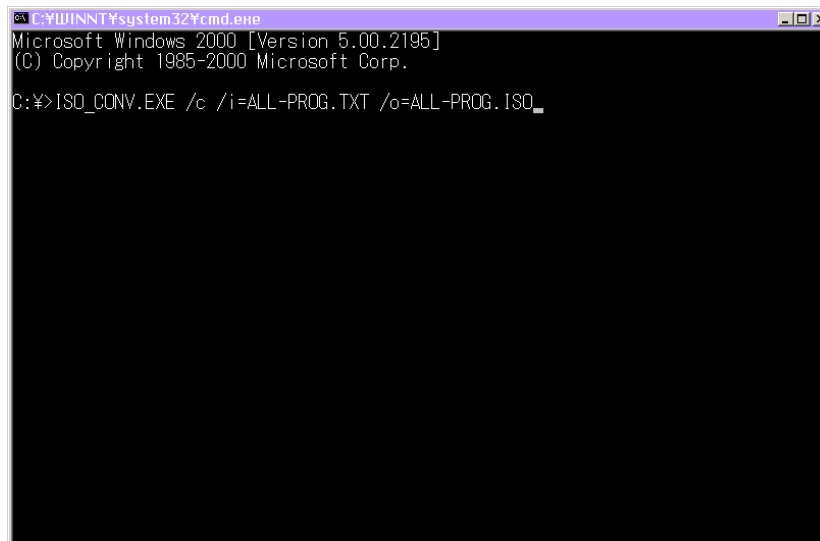
FANUC ISO Converter is a tool that converts a file created or externally output with ASCII code to ISO code format.

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.

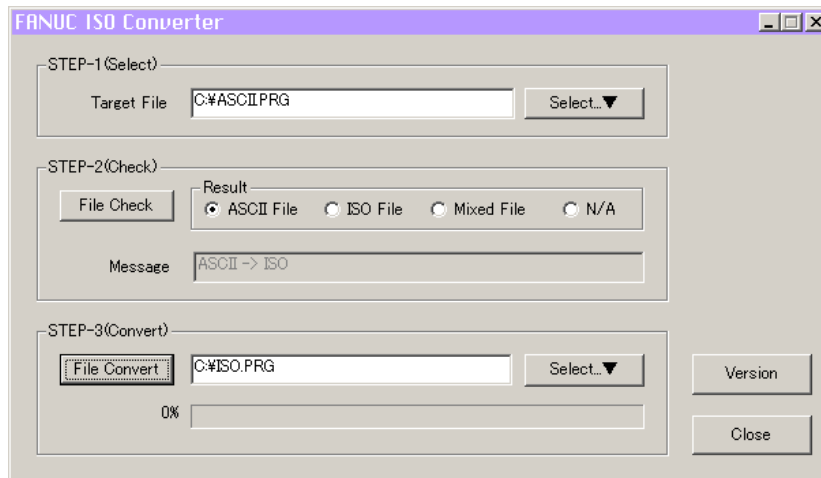
Example

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.

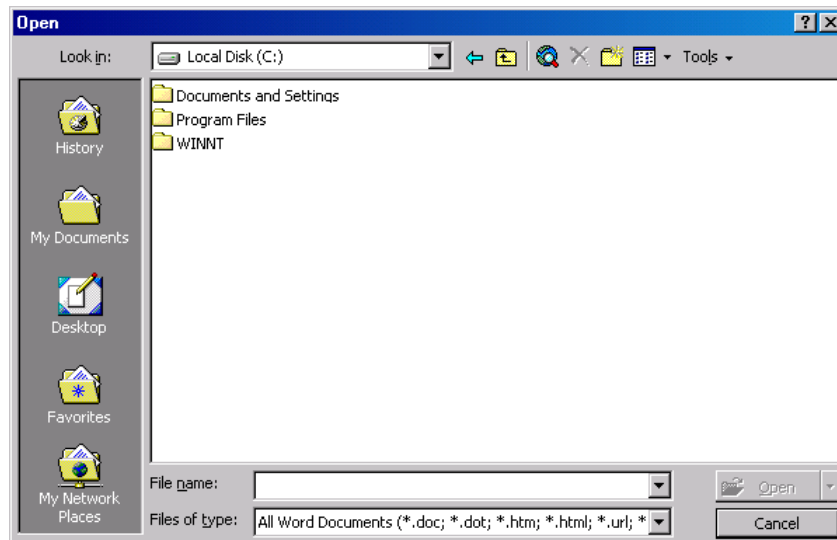


Conversion procedure

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.



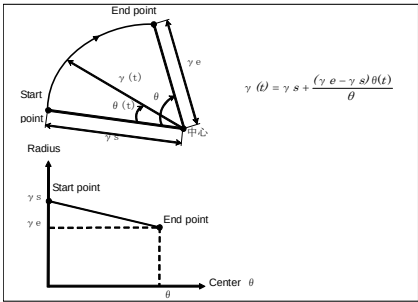
K DIFFERENCES FROM SERIES 0i-PC

Appendix K, "DIFFERENCES FROM SERIES 0i-PC", consists of the following sections:

K.1	CIRCULAR INTERPOLATION	849
K.2	HELICAL INTERPOLATION	850
K.3	SKIP FUNCTION	851
K.4	MANUAL REFERENCE POSITION RETURN	853
K.5	WORKPIECE COORDINATE SYSTEM	855
K.6	LOCAL COORDINATE SYSTEM	856
K.7	TOOL COMPENSATION MEMORY	857
K.8	CUSTOM MACRO	858
K.9	INTERRUPTION TYPE CUSTOM MACRO	860
K.10	PROGRAMMABLE PARAMETER INPUT (G10)	860
K.11	AI ADVANCED PREVIEW CONTROL	861
K.12	AXIS SYNCHRONOUS CONTROL	862
K.13	RUN HOUR AND PARTS COUNT DISPLAY	867
K.14	MANUAL HANDLE FEED	867
K.15	PMC AXIS CONTROL	868
K.16	EXTERNAL SUBPROGRAM CALL (M198)	872
K.17	SEQUENCE NUMBER SEARCH	873
K.18	STORED STROKE CHECK	874
K.19	STORED PITCH ERROR COMPENSATION	876
K.20	SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION	877
K.21	RESET AND REWIND	878
K.22	MANUAL ABSOLUTE ON	878
K.23	EXTERNAL DATA INPUT	879
K.24	DATA SERVER FUNCTION	881
K.25	POWER MATE CNC MANAGER	882
K.26	CUTTER COMPENSATION	882
K.27	RAPID TRAVERSE OVERRIDE	887
K.28	POSITIONING BY OPTIMUM ACCELERATIONS	887
K.29	CONSTANT CONTROL OF POSITIONING TIME	888

K.1 CIRCULAR INTERPOLATION

K.1.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Interpolation method when the arc end point is not on the arc	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. - Circular interpolation is performed using the radius value of the start point and, when an axis reaches the end point, it is moved linearly. Parameter No. 3410 In a circular interpolation command, set the limit allowed for the difference between the radius values of the start point and end point.	- 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 $\theta(t)$. 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.

K.1.2 Differences in Diagnosis Display

None.

K.2 HELICAL INTERPOLATION

K.2.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Specification of the feedrate	- Specify the feedrate along a circular arc. Therefore, the feedrate of the linear axis is as follows: $F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$	- 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: $F \times \frac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$ The velocity along the linear axis is expressed as follows: $F \times \frac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$ 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.2.2 Differences in Diagnosis Display

None.

K.3 SKIP FUNCTION

K.3.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD																	
Setting to enable the high-speed skip signal for normal skip (G33) when the multi-stage skip function is enabled	- Set 1 in bit 5 (SLS) of parameter No. 6200.	- Set 1 in bit 4 (HSS) of parameter No. 6200.																	
	<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-PC</th><th>FS0i-PD</th></tr><tr><td>Disabled</td><td>G33 (normal skip)</td><td>HSS</td><td>HSS</td></tr><tr><td rowspan="2">Enabled</td><td>G33 (normal skip)</td><td><u>SLS</u></td><td><u>HSS</u></td></tr><tr><td>G33P1 to G33P4 (multi-stage skip)</td><td>SLS</td><td>SLS</td></tr></table>		Multi-stage skip function	Command	Parameter to decide on use of the high-speed skip signal		FS0i-PC	FS0i-PD	Disabled	G33 (normal skip)	HSS	HSS	Enabled	G33 (normal skip)	<u>SLS</u>	<u>HSS</u>	G33P1 to G33P4 (multi-stage skip)	SLS	SLS
	Multi-stage skip function	Command			Parameter to decide on use of the high-speed skip signal														
			FS0i-PC	FS0i-PD															
	Disabled	G33 (normal skip)	HSS	HSS															
	Enabled	G33 (normal skip)	<u>SLS</u>	<u>HSS</u>															
G33P1 to G33P4 (multi-stage skip)		SLS	SLS																
Target of acceleration / deceleration and servo system delay compensation	- Compensation is performed for the skip coordinates obtained when the high-speed skip signal is set to "1".	- Compensation is performed for the skip coordinates obtained when the skip or high-speed skip signal is set to "1".																	
Method of acceleration / deceleration and servo system delay compensation	- 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.	- 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.																	
Skip cutting feedrate (normal skip)	- Feedrate specified by the F code in the program	- Depends on bit 1 (SFP) of parameter No. 6207. When 0 is set, the processing is the same as Series 0i-PC. <u>Bit 1 (SFP) of parameter No. 6207</u> The feedrate during the skip function (G33) is: 0: Feedrate specified by the F code in the program. 1: Feedrate specified in parameter No. 6281.																	

Function	Series 0i-PC	Series 0i-PD
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 0i-PC. <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 (G33 P99/98) monitors: 0: All axes. 1: Only the axis specified in the same block as G33 P99/98.	- Bit 3 (TSA) of parameter No. 6201 is not available. Only the axis specified in the same block as G33 P99/98 is monitored.
High-speed skip signal input for the G33 P99 command (torque limit skip)	As the skip signal for the G33 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 G33 P99/98 is specified without a torque limit being specified in advance (torque limit skip)	- The G33 P99/98 command is executed as is. (No alarm is issued.)	- Alarm PS0035 is issued.

K.3.2 Differences in Diagnosis Display

None.

K.4 MANUAL REFERENCE POSITION RETURN

K.4.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Conditions for performing manual reference position return during feed hold	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.	
	- 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.	- Bit 2 (OZR) of parameter No. 1800 is not available. Alarm PS0091 occurs, and manual reference position return is not performed.
When inch/metric switch is done	- The reference position is lost. (The reference position is not established.)	- The reference position is not lost. (The reference position remains established.)
Reference position setting without dogs for all axes	- Set 1 in bit 1 (DLZ) of parameter No. 1002.	- 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.
Function that performs reference position setting without dogs two or more times when the reference position is not established in absolute position detection	- Not available.	- 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.

Function	Series 0i-PC	Series 0i-PD
Behavior when manual reference position return is started on a rotation axis with the deceleration dog pressed before a reference position is established	- 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 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 (HZIP) of parameter No. 1206. <u>Bit 1 (HZIP) of parameter No.1206</u> Upon high-speed manual reference position return, the coordinate system is: 0: Preset. 1: Not preset (FS0i-PC compatible specification).

Function	Series 0i-PC	Series 0i-PD
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. [Conditions required when specifying the command] (3) In an incremental command, the travel distance of the middle point is 0. [Conditions required after specifying the command] (4) The first move command specified for the target axis of coordinate system rotation, scaling, or programmable mirror image is an absolute command.

K.4.2 Differences in Diagnosis Display

None.

K.5 WORKPIECE COORDINATE SYSTEM

K.5.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
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.5.2 Differences in Diagnosis Display

None.

K.6 LOCAL COORDINATE SYSTEM

K.6.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
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. <u>Bit 3 (RLC) of parameter No. 1202</u> Upon reset, the local coordinate system is: 0: Not canceled. 1: Canceled. <u>Bit 5 (SNC) of parameter No. 1202</u> 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. <u>Bit 7 (WZR) of parameter No. 1201</u> 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. <u>Bit 3 (RLC) of parameter No. 1202</u> 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. <u>Bit 6 (CLR) of parameter No. 3402</u> 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. <u>Bit 6 (C14) of parameter No. 3407</u> 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.

Function	Series 0i-C	Series 0i-D
Operation with the local coordinate system setting (G52)	- Make a selection using bit 4 (G52) of parameter No. 1202. <u>Bit 4 (G52) of parameter No. 1202</u> 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.6.2 Differences in Diagnosis Display

None.

K.7 TOOL COMPENSATION MEMORY

K.7.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD																		
Unit and range of tool compensation values	- The unit and range of tool compensation values are determined by the setting unit.	- Set the unit and range using bit 0 (OFA) of parameter No. 5042. <u>Bit 0 (OFA) of parameter No. 5042</u> Select the setting unit and range of tool offset values. Metric input <table border="1"> <thead> <tr> <th>OFA</th><th>Unit</th><th>Range</th></tr> </thead> <tbody> <tr> <td>1</td><td>0.01mm</td><td>±9999.99mm</td></tr> <tr> <td>0</td><td>0.001mm</td><td>±9999.999mm</td></tr> </tbody> </table> Inch input <table border="1"> <thead> <tr> <th>OFA</th><th>Unit</th><th>Range</th></tr> </thead> <tbody> <tr> <td>1</td><td>0.001inch</td><td>±999.999inch</td></tr> <tr> <td>0</td><td>0.0001inch</td><td>±999.9999inch</td></tr> </tbody> </table>	OFA	Unit	Range	1	0.01mm	±9999.99mm	0	0.001mm	±9999.999mm	OFA	Unit	Range	1	0.001inch	±999.999inch	0	0.0001inch	±999.9999inch
OFA	Unit	Range																		
1	0.01mm	±9999.99mm																		
0	0.001mm	±9999.999mm																		
OFA	Unit	Range																		
1	0.001inch	±999.999inch																		
0	0.0001inch	±999.9999inch																		

Function	Series 0i-PC	Series 0i-PD
Automatic conversion of tool compensation values upon inch/metric switch	- 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.	- Bit 0 (OIM) of parameter No. 5006 is not available. Tool compensation values are always converted automatically.

K.7.2 Differences in Diagnosis Display

None.

K.8 CUSTOM MACRO

K.8.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Keep-type common variable (#500 to #999)	- The default value is <null>. - The Series 0i-PD function (described at right) is not available.	- The default value is 0. - The range specified by parameter Nos. 6031 and 6032 can be made write-protected (read-only).
System variable to read machine coordinates #5021 to #502n (n: number of axes)	- Machine coordinates are always read in machine units (output units).	- 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.
Logical operations in an if statement	- 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.	- Bit 0 (MLG) of parameter No. 6006 is not available. Logical operations can always be used.

Function	Series 0i-PC	Series 0i-PD
Behavior of the GOTO statement when a sequence number is not found at the start of the block	The command after the sequence number of the block (to the right of the sequence number) is executed.	- 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.
	* Use a sequence number at the start of a block.	
Behavior of "GOTO 0" when there is a sequence number	The program jumps to the block containing the sequence number.	- No jump occurs. - Alarm PS1128 is issued.
	* Do not use a sequence number.	
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 ;	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.	- 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).
Behavior when subprogram call using an M code and subprogram call using an T code are done	- 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] <pre> O0001; T100; (1) M06 T200; (2) T300 M06; (3) M30; %</pre>	
	In FS0i-PC, 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.	In FS0i-PD, 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).
Block containing "M98 Pxxxx" or "M99" without any addresses other than O, N, P, and L	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.)	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.)
	* For details about macro and NC statements, refer to Section 17.5, "MACRO STATEMENTS AND NC STATEMENTS", in "OPERATOR'S MANUAL" (B-64554EN).	

Function	Series 0i-PC	Series 0i-PD																	
Subprogram and macro calls	- The call nesting level differs as follows.																		
	<table><tr><th rowspan="2">Model Call method</th><th colspan="2">Series 0i-PC</th><th colspan="2">Series 0i-PD</th></tr><tr><th>Independent nesting level</th><th>Total</th><th>Independent nesting level</th><th>Total</th></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>		Model Call method	Series 0i-PC		Series 0i-PD		Independent nesting level	Total	Independent nesting level	Total	Macro call (G65/G66)	4 in all	(G65/G66/M98) 8 in all	5 in all	(G65/G66/M98) 15 in all	Subprogram call (M98)	4	10
	Model Call method	Series 0i-PC		Series 0i-PD															
		Independent nesting level	Total	Independent nesting level	Total														
	Macro call (G65/G66)	4 in all	(G65/G66/M98) 8 in all	5 in all	(G65/G66/M98) 15 in all														
Subprogram call (M98)	4	10																	
Local variable clear operation by reset	- 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.	- Bit 7 (CLV) of parameter No. 6001 is not available. Local variables are always cleared to <null> when reset.																	

K.8.2 Differences in Diagnosis Display

None.

K.8.3 Miscellaneous

Series 0i-PD 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 0i-PC. For details, refer to Chapter II-17, "CUSTOM MACRO", in "OPERATOR'S MANUAL" (B-64554EN).

K.9 INTERRUPTION TYPE CUSTOM MACRO

K.9.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Interruption type custom macro in DNC operation	- Not available.	- Available.

K.9.2 Differences in Diagnosis Display

None.

K.10 PROGRAMMABLE PARAMETER INPUT (G10)

K.10.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Parameter input mode setting	- Specify G10 L50.	- Specify G10 L52.

K.10.2 Differences in Diagnosis Display

None.

K.11 AI ADVANCED PREVIEW CONTROL

K.11.1 Differences in Specifications

Differences regarding AI advanced preview control

Function	Series 0i-PC	Series 0i-PD
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
	- Acceleration-based feedrate clamp	- Speed control with the acceleration on each axis
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).
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.

Function	Series 0i-PC	Series 0i-PD
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 AI advanced preview control mode.	- Enabled only when acceleration/deceleration after interpolation is used in the AI advanced preview control mode.
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.

K.11.2 Differences in Diagnosis Display

None.

K.12 AXIS SYNCHRONOUS CONTROL

K.12.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Function name	- Quick synchronous control	- Axis synchronous control

Function	Series 0i-PC	Series 0i-PD
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 0i-PC. <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)	- 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.
Synchronization error check based on positional difference	- 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

Function	Series 0i-PC	Series 0i-PD
Synchronization error check based on machine coordinates	- 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	- 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	- 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.)
Maximum compensation for synchronization	- 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)

Function	Series 0i-PC	Series 0i-PD
Automatic setting for grid position matching	- 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	- 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	- 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.)
Setting to use the external machine coordinate system shift function for the slave axis	- 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	- 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.

Function	Series 0i-PC	Series 0i-PD
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. <u>Parameter No. 8337</u> Specify an M code that changes synchronous operation to normal operation. <u>Parameter No. 8338</u> Specify an M code that changes normal operation to synchronous operation.
Automatic slave axis parameter setting	- 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.
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.12.2 Differences in Diagnosis Display

Item	Series 0i-PC	Series 0i-PD
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.13 RUN HOUR AND PARTS COUNT DISPLAY

K.13.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Data range of the M code that counts the number of machined parts	Parameter No. 6710 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	Parameter No. 6713 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	Parameter No. 6711 Number of parts machined The data range is as follows.	Parameter No. 6712 Total number of parts machined The data range is as follows.
	- 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	Parameter No. 6750 Integrated value of power-on period Parameter No. 6756 Integrated value of time when input signal TMRON (G053.0) is on The data range is as follows.	Parameter No. 6752 Integrated value of time during automatic operation Parameter No. 6754 Integrated value of cutting time Parameter No. 6758 Integrated value of one automatic operation time
	- 0 to 99999999 (8 digits)	- 0 to 999999999 (9 digits)

K.13.2 Differences in Diagnosis Display

None.

K.14 MANUAL HANDLE FEED

K.14.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Handle pulses exceeding the rapid traverse rate	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.	
	- Depends on bit 4 (HPF) of parameter No. 7100. The amount of pulses to be accumulated is set in parameter No. 7117.	- 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.
Permissible amount of pulses for manual handle feed	- The value range of parameter No. 7117 is 0 to 99999999 (8 digits).	- The value range of parameter No. 7117 is 0 to 999999999 (9 digits).

Function	Series 0i-PC	Series 0i-PD
Value range of the magnification parameter for manual handle feed	- 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.	For parameter No. 7113, 7114, 7131, 7132, 7133, 7134, 12350, and 12351, magnification ranges from 1 to 2000.
	<u>Parameter No. 7113</u> Magnification when manual handle feed amount selection signals MP1 = 0 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	<u>Parameter No. 7114</u> Magnification when manual handle feed amount selection signals MP1 = 1 and MP2 = 1
	<u>Parameter No. 7131</u> Magnification when manual handle feed amount selection signals MP21 = 0 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.	<u>Parameter No. 7132</u> Magnification when manual handle feed amount selection signals MP21 = 1 and MP22 = 1
	<u>Parameter No. 7133</u> Magnification when manual handle feed amount selection signals MP31 = 0 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.	<u>Parameter No. 7134</u> Magnification when manual handle feed amount selection signals MP31 = 1 and MP32 = 1
	<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
Number of manual pulse generators used	Set the value in parameter No. 7110.	Parameter No. 7110 is not available. Up to three generators can be used without setting the parameter.

K.14.2 Differences in Diagnosis Display

None.

K.15 PMC AXIS CONTROL

K.15.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Relationship with the feed-forward and advanced preview feed-forward functions	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.	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.

Function	Series 0i-PC	Series 0i-PD																																																																																						
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)	- The data range is as follows. <table><tr><th rowspan="3"></th><th rowspan="3"></th><th colspan="2">Valid data range</th><th rowspan="3">Unit of data</th></tr><tr><th>IS-A, IS-B</th><th>IS-C</th></tr><tr><td></td><td></td></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>			Valid data range		Unit of data	IS-A, IS-B	IS-C			Linear axis	Millimeter machine	30 to 15000	30 to 12000	mm/min	Inch machine	30 to 6000	30 to 4800	inch/min	Rotation axis		30 to 15000	30 to 12000	deg/min	- 1 to 65535 The data unit is as follows. <table><tr><th colspan="2">Data unit</th><th rowspan="2">Unit</th></tr><tr><th colspan="2">IS-A to IS-C</th></tr><tr><td rowspan="2">Linear axis</td><td>Metric machine</td><td>mm/min</td></tr><tr><td>Inch machine</td><td>inch/min</td></tr><tr><td>Rotation axis</td><td></td><td>deg/min</td></tr></table>	Data unit		Unit	IS-A to IS-C		Linear axis	Metric machine	mm/min	Inch machine	inch/min	Rotation axis		deg/min																																																		
				Valid data range			Unit of data																																																																																	
				IS-A, IS-B	IS-C																																																																																			
Linear axis	Millimeter machine	30 to 15000	30 to 12000	mm/min																																																																																				
	Inch machine	30 to 6000	30 to 4800	inch/min																																																																																				
Rotation axis		30 to 15000	30 to 12000	deg/min																																																																																				
Data unit		Unit																																																																																						
IS-A to IS-C																																																																																								
Linear axis	Metric machine	mm/min																																																																																						
	Inch machine	inch/min																																																																																						
Rotation axis		deg/min																																																																																						
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)	- 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</td><td>±99999.999</td><td>±9999.9999</td><td>mm</td></tr><tr><td>deg unit input</td><td></td><td></td><td>deg</td></tr><tr><td>inch unit input</td><td>±9999.9999</td><td>±999.99999</td><td>inch</td></tr></table>	Input increment	IS-B	IS-C	Unit	mm unit input	±99999.999	±9999.9999	mm	deg unit input			deg	inch unit input	±9999.9999	±999.99999	inch	- 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>	IS-A	IS-B,IS-C	-99999999 to 99999999 (8 digits)	-999999999 to 999999999 (9 digits)	Setting unit	Minimum data unit	IS-A	0.01	IS-B	0.001	IS-C	0.0001																																																										
Input increment	IS-B	IS-C	Unit																																																																																					
mm unit input	±99999.999	±9999.9999	mm																																																																																					
deg unit input			deg																																																																																					
inch unit input	±9999.9999	±999.99999	inch																																																																																					
IS-A	IS-B,IS-C																																																																																							
-99999999 to 99999999 (8 digits)	-999999999 to 999999999 (9 digits)																																																																																							
Setting unit	Minimum data unit																																																																																							
IS-A	0.01																																																																																							
IS-B	0.001																																																																																							
IS-C	0.0001																																																																																							
Data range of cutting feedrate for rapid traverse (01h) and skip - feed per minute (03h)	- 1 to 65535 The specified feedrate must be within the range shown in the table below. <table><tr><th rowspan="3"></th><th rowspan="3"></th><th colspan="2">Valid data range</th><th rowspan="3">Unit of data</th></tr><tr><th>IS-B</th><th>IS-C</th></tr><tr><td></td><td></td></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>			Valid data range		Unit of data	IS-B	IS-C			Linear axis	Millimeter machine	1 to 100000	0.1 to 12000.0	mm/min	Inch machine	0.01 to 4000.00	0.01 to 480.000	inch/min	Rotation axis		1 to 100000	0.1 to 12000.0	deg/min	- 1 to 65535																																																															
				Valid data range			Unit of data																																																																																	
				IS-B	IS-C																																																																																			
Linear axis	Millimeter machine	1 to 100000	0.1 to 12000.0	mm/min																																																																																				
	Inch machine	0.01 to 4000.00	0.01 to 480.000	inch/min																																																																																				
Rotation axis		1 to 100000	0.1 to 12000.0	deg/min																																																																																				
Function to increase the specification unit by a factor of 200 for continuous feed (06h)	- Not available.	- By setting 1 in bit 2 (JFM) of parameter No. 8004, it is possible to increase the specification unit by a factor of 200. Bit 2 (JFM) of parameter No. 8004 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>	Increment system	Bit 2 (JFM) of No. 8004	Millimeter input (mm/min)	Inch input (inch/min)	Rotation axis (min ⁻¹)	IS-B	0	1	0.01	0.00023	1	200	2.00	0.046	IS-C	0	0.1	0.001	0.000023	1	20	0.200	0.0046																																																															
Increment system	Bit 2 (JFM) of No. 8004	Millimeter input (mm/min)	Inch input (inch/min)	Rotation axis (min ⁻¹)																																																																																				
IS-B	0	1	0.01	0.00023																																																																																				
	1	200	2.00	0.046																																																																																				
IS-C	0	0.1	0.001	0.000023																																																																																				
	1	20	0.200	0.0046																																																																																				
Maximum feedrate for continuous feed (06h)	- When an override of 254% is applied <table><tr><th rowspan="2"></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr><tr><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr><tr><td>1 time</td><td>166458 mm/min</td><td>1664.58 inch/min</td><td>16645 mm/min</td><td>166.45 inch/min</td></tr><tr><td>10 times</td><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 rowspan="2"></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr><tr><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr><tr><td>1 time</td><td>65535 mm/min</td><td>655.35 inch/min</td><td>6553 mm/min</td><td>65.53 inch/min</td></tr><tr><td>10 times</td><td>655350 mm/min</td><td>6553.50 inch/min</td><td>65535 mm/min</td><td>655.35 inch/min</td></tr></table>		IS-B		IS-C		Metric input	Inch input	Metric input	Inch input	1 time	166458 mm/min	1664.58 inch/min	16645 mm/min	166.45 inch/min	10 times	1664589 mm/min	16645.89 inch/min	166458 0mm/min	1664.58 inch/min		IS-B		IS-C		Metric input	Inch input	Metric input	Inch input	1 time	65535 mm/min	655.35 inch/min	6553 mm/min	65.53 inch/min	10 times	655350 mm/min	6553.50 inch/min	65535 mm/min	655.35 inch/min	- When an override of 254% is applied <table><tr><th rowspan="2"></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr><tr><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><td>1 time</td><td>166458</td><td>1664.58</td><td>16645</td><td>166.46</td></tr><tr><td>10 times</td><td>999000</td><td>16645.89</td><td>99900</td><td>1664.58</td></tr><tr><td>200 times</td><td>999000</td><td>39330.0</td><td>99900</td><td>3933.0</td></tr></table> - When override is canceled <table><tr><th rowspan="2"></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr><tr><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><td>1 time</td><td>65535</td><td>655.35</td><td>6553</td><td>65.53</td></tr><tr><td>10 times</td><td>655350</td><td>6553.5</td><td>65535</td><td>655.35</td></tr><tr><td>200 times</td><td>999000</td><td>39330.0</td><td>999000</td><td>3933.0</td></tr></table>		IS-B		IS-C		Metric input (mm/min)	Inch input (inch/min)	Metric input (mm/min)	Inch input (inch/min)	1 time	166458	1664.58	16645	166.46	10 times	999000	16645.89	99900	1664.58	200 times	999000	39330.0	99900	3933.0		IS-B		IS-C		Metric input (mm/min)	Inch input (inch/min)	Metric input (mm/min)	Inch input (inch/min)	1 time	65535	655.35	6553	65.53	10 times	655350	6553.5	65535	655.35	200 times	999000	39330.0	999000	3933.0
	IS-B		IS-C																																																																																					
	Metric input	Inch input	Metric input	Inch input																																																																																				
1 time	166458 mm/min	1664.58 inch/min	16645 mm/min	166.45 inch/min																																																																																				
10 times	1664589 mm/min	16645.89 inch/min	166458 0mm/min	1664.58 inch/min																																																																																				
	IS-B		IS-C																																																																																					
	Metric input	Inch input	Metric input	Inch input																																																																																				
1 time	65535 mm/min	655.35 inch/min	6553 mm/min	65.53 inch/min																																																																																				
10 times	655350 mm/min	6553.50 inch/min	65535 mm/min	655.35 inch/min																																																																																				
	IS-B		IS-C																																																																																					
	Metric input (mm/min)	Inch input (inch/min)	Metric input (mm/min)	Inch input (inch/min)																																																																																				
1 time	166458	1664.58	16645	166.46																																																																																				
10 times	999000	16645.89	99900	1664.58																																																																																				
200 times	999000	39330.0	99900	3933.0																																																																																				
	IS-B		IS-C																																																																																					
	Metric input (mm/min)	Inch input (inch/min)	Metric input (mm/min)	Inch input (inch/min)																																																																																				
1 time	65535	655.35	6553	65.53																																																																																				
10 times	655350	6553.5	65535	655.35																																																																																				
200 times	999000	39330.0	999000	3933.0																																																																																				
Minimum unit of feedrate for the speed command (10h)	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																																																																																							
	- Fmin = P ÷ 7500 (mm/min)	- Fmin = P ÷ 1000 (mm/min)																																																																																						
Speed specification in the speed command (10h)	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																																																																																							

Function	Series 0i-PC	Series 0i-PD								
	- $F = N \times P \div 7500$ (mm/min)	- $F = N \times P \div 1000$ (mm/min)								
Setting range of torque data for torque control (11h)	- 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>	Valid data range	Unit	-99999999 to +99999999	0.0000 1 Nm	- 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>	Valid data range	Unit	-999999999 to +999999999 (9 digits)	0.0000 1 Nm
Valid data range	Unit									
-99999999 to +99999999	0.0000 1 Nm									
Valid data range	Unit									
-999999999 to +999999999 (9 digits)	0.0000 1 Nm									
Note on executing an absolute command from the program for an axis subject to PMC axis control during automatic operation	- [For Series 0i-PD] 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-PC] Control does not need to be executed using a non-buffering M code.									
Acceleration/deceleration control for an axis synchronized with external pulses using external pulse synchronization (0Bh, 0Dh to 0Fh)	- Depends on bit 2 (SUE) of parameter No. 8002. Bit 2 (SUE) of parameter No. 8002 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.	- 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).								
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.								

Function	Series 0i-PC	Series 0i-PD
Setting to change all axes to CNC axes or PMC axes	- Depends on bit 1 (PAX) of parameter No. 8003. <u>Bit 1 (PAX) of parameter No. 8003</u> 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. <u>Bit 0 (CMV) of parameter No. 8004</u> 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. <u>Bit 1 (NMT) of parameter No. 8004</u> 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.)
Individual output of the auxiliary function	- Depends on bit 7 (MFD) of parameter No. 8005. <u>Bit 7 (MFD) of parameter No. 8005</u> 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. <u>Bit 4 (EVP) of parameter No. 8005</u> 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. <u>Bit 2 (VCP) of parameter No. 8007</u> The speed command in PMC axis control is: 0: FS10/11 type. 1: FS0 type.

Function	Series 0i-PC	Series 0i-PD													
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.													
Operation when instructing in machine coordinate system selection (20h) to the axis to which roll-over is effective	- Depends on bit 1 (RAB) of parameter No. 1008. Bit 1 (RAB) of parameter No. 1008 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.	- Depends on bit 1 (RAB) of parameter No. 1008 and bit 4 (R20) of parameter No. 8013. <table><tr><th colspan="2" rowspan="2"></th><th colspan="2">Bit 4 (R20) of parameter No.8013</th></tr><tr><th>0</th><th>1</th></tr><tr><th rowspan="2">Bit 1 (RAB) of parameter No.1008</th><th>0</th><td>Direction of the shortest path</td><td>Direction of the shortest path</td></tr><tr><th>1</th><td>Direction of sign of the amount of the movement to be made</td><td>Direction of sign of command value</td></tr></table>			Bit 4 (R20) of parameter No.8013		0	1	Bit 1 (RAB) of parameter No.1008	0	Direction of the shortest path	Direction of the shortest path	1	Direction of sign of the amount of the movement to be made	Direction of sign of command value
		Bit 4 (R20) of parameter No.8013													
		0	1												
Bit 1 (RAB) of parameter No.1008	0	Direction of the shortest path	Direction of the shortest path												
	1	Direction of sign of the amount of the movement to be made	Direction of sign of command value												

K.15.2 Differences in Diagnosis Display

None.

K.16 EXTERNAL SUBPROGRAM CALL (M198)

K.16.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Address P format when calling a subprogram on the memory card (file number)	Depends on bit 2 (SBP) of parameter No. 3404. Bit 2 (SBP) of parameter No. 3404 In the external device subprogram call	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

Function	Series 0i-PC	Series 0i-PD
specification/program number specification)	M198, address P is specified using: 0: File number. 1: Program number.	dependent on the setting of bit 2 (SBP) of parameter No. 3404.
Multiple call alarm	If a subprogram called by an external subprogram call specifies a further external subprogram call, the following alarms are issued, respectively:	
	- Alarm PS0210	- Alarm PS1080
External subprogram call in MDI mode	- Enabled.	- Depends on bit 1 (MDE) of parameter No. 11630. <u>Bit 1 (MDE) of parameter No. 11630</u> In MDI mode, an external device subprogram call (M198 command) is: 0: Disabled. (Alarm PS1081 is issued.) 1: Enabled.

K.16.2 Differences in Diagnosis Display

None.

K.17 SEQUENCE NUMBER SEARCH

K.17.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Return from a subprogram to the calling program's block that has a specified sequence number Sequence number search when (M99 Pxxxx) is executed	The calling program is searched from the beginning, and control is returned to the first block found to have sequence number Nxxxxx.	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.
	Example) Main program O0001 ; N100 ; (1) N100 ; (2) M98 P9001 ; N100 ; (3) N100 ; (4) M30 ; [For Series 0i-PC] Control is returned to block (1).	Sub program O9001 ; M99 P100 ; [For Series 0i-PD] 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.	

K.17.2 Differences in Diagnosis Display

None.

K.18 STORED STROKE CHECK

K.18.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Stored stroke check immediately following powering on	- This function is always enabled for all axes.	- 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.
	- Machine coordinates are set upon powering on. Absolute and relative coordinates are not set. (They are set when the absolute position detector is provided.)	- Machine coordinates are set upon powering on. Absolute and relative coordinates are set based on these machine coordinates.
Overtravel alarm	- Stored stroke 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.	- Stored stroke 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 stroke check 1, 2, or 3 alarm occurs, the tool stops: 0: After entering the prohibited area. 1: Before entering the prohibited area.
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).

Function	Series 0i-PC	Series 0i-PD
Block that judges the distance to the stored stroke limit in AI advanced preview control	- A selection can be made using bit 5 (ODA) of parameter No. 7055. <u>Bit 5 (ODA) of parameter No. 7055</u> The distance to the stored stroke limit in AI advanced preview 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.
The method of stored stroke check	- If the end point of rapid traverse positioning is in the forbidden area, the alarm is output without moving axes. (Stroke check before movement is performed regardless of parameter setting.)	- For stroke check before movement, Either valid (1) or invalid (0) is selected by following parameter. Parameter PLC (No.1301#7)
Alarms for stroke check before movement	- Following alarms are output for X and Y axis. Alarm PS4700 : X axis +side Alarm PS4701 : X axis -side Alarm PS4702 : Y axis +side Alarm PS4703 : Y axis -side	- Following alarms are output for each axis. Alarm OT510 : +side Alarm OT511 : -side

K.18.2 Differences in Diagnosis Display

None.

K.19 STORED PITCH ERROR COMPENSATION

K.19.1 Differences in Specifications

Function	Explanation																					
Value of parameter No. 3621 for the setting of a rotary axis (type A)	Reference position Compensation values are output at the positions indicated by O. - 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-PC</th><th>Series 0i-PD</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-PC = Number of the compensation position of the reference position (parameter No. 3620) Series 0i-PD = Number of the compensation position of the reference position (parameter No. 3620) + 1	Parameter	Series 0i-PC	Series 0i-PD	No. 3620: Number of the compensation position of the reference position	60	60	No. 3621: Smallest compensation position number	60	61	No. 3622: Largest compensation position number	68	68	No. 3623: Compensation magnification	1	1	No. 3624: Interval between compensation positions	45000	45000	No. 3625: Amount of movement per rotation	360000	360000
Parameter	Series 0i-PC	Series 0i-PD																				
No. 3620: Number of the compensation position of the reference position	60	60																				
No. 3621: Smallest compensation position number	60	61																				
No. 3622: Largest compensation position number	68	68																				
No. 3623: Compensation magnification	1	1																				
No. 3624: Interval between compensation positions	45000	45000																				
No. 3625: Amount of movement per rotation	360000	360000																				

K.19.2 Differences in Diagnosis Display

None.

K.20 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION

K.20.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
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.20.2 Differences in Diagnosis Display

None.

K.21 RESET AND REWIND

K.21.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Modal data when reset during the execution of a block	- 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. Maintained.	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. <pre>N1 G00 X120. Z0. T0101 ; ; N2 G00 X180. Z20. T0202 ; ;</pre>
Information in a block that is pre-read when a reset is made during an automatic operation (contents of the buffer)	- 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.	- The information in the block is not held regardless whether MDI mode is in progress.

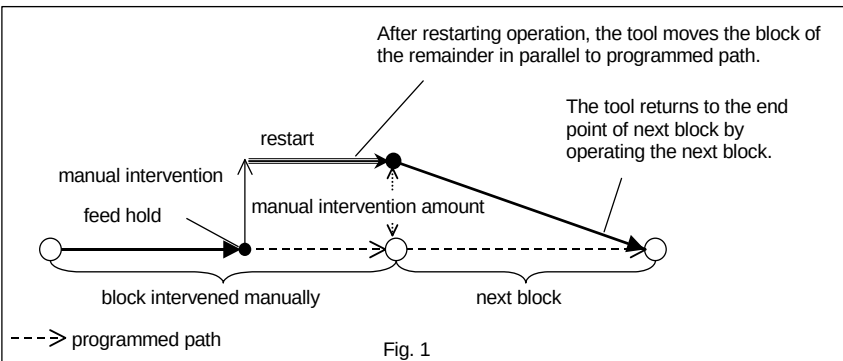
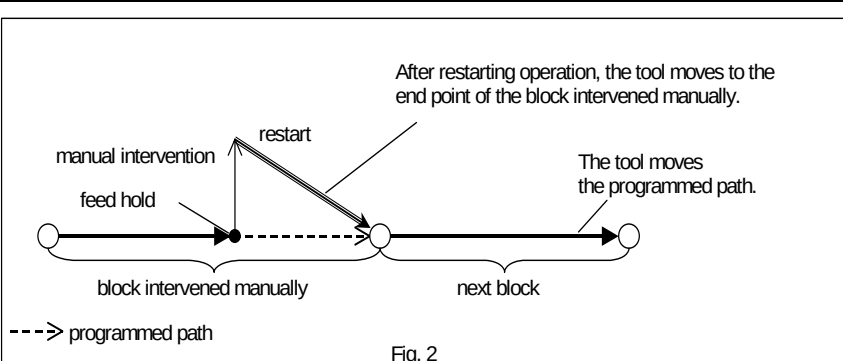
K.21.2 Differences in Diagnosis Display

None.

K.22 MANUAL ABSOLUTE ON

K.22.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
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)

Function	Series 0i-PC	Series 0i-PD
	 After restarting operation, the tool moves the block of the remainder in parallel to programmed path. The tool returns to the end point of next block by operating the next block. Fig. 1	
	 After restarting operation, the tool moves to the end point of the block intervened manually. The tool moves the programmed path. Fig. 2	

K.22.2 Differences in Diagnosis Display

None.

K.23 EXTERNAL DATA INPUT

K.23.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
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 0i-PC. 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

Function	Series 0i-PC	Series 0i-PD		
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. <u>Bit 0 (EXA) of parameter No. 6301</u> 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.		
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 0i-PC. <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. <table><tr><td> - 0 to 1000 </td><td> - 0 to 4096 </td></tr></table>		- 0 to 1000	- 0 to 4096
- 0 to 1000	- 0 to 4096			

Function	Series 0i-PC	Series 0i-PD
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.23.2 Differences in Diagnosis Display

None.

K.24 DATA SERVER FUNCTION

K.24.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
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.

K.24.2 Differences in Diagnosis Display

None.

K.25 POWER MATE CNC MANAGER

K.25.1 Differences in Specifications

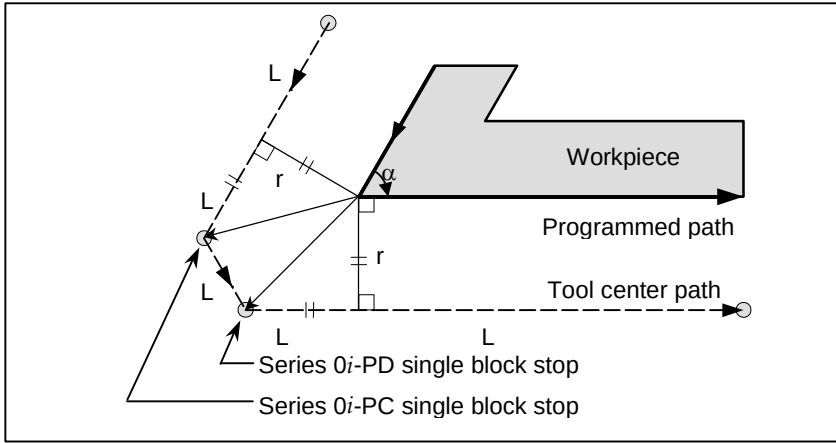
Function	Series 0i-PC	Series 0i-PD
4-slave display function	By setting 1 in bit 0 (SLV) of parameter No. 0960, it is possible to split the screen into four windows, enabling up to four slaves to be displayed. Bit 0 (SLV) of parameter No. 0960 When Power Mate CNC Manager is selected, the screen: 0: Displays 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.25.2 Differences in Diagnosis Display

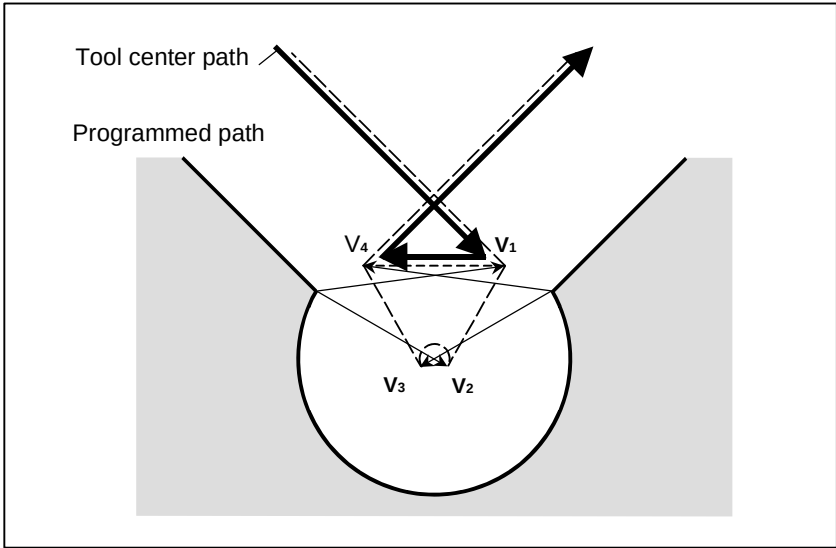
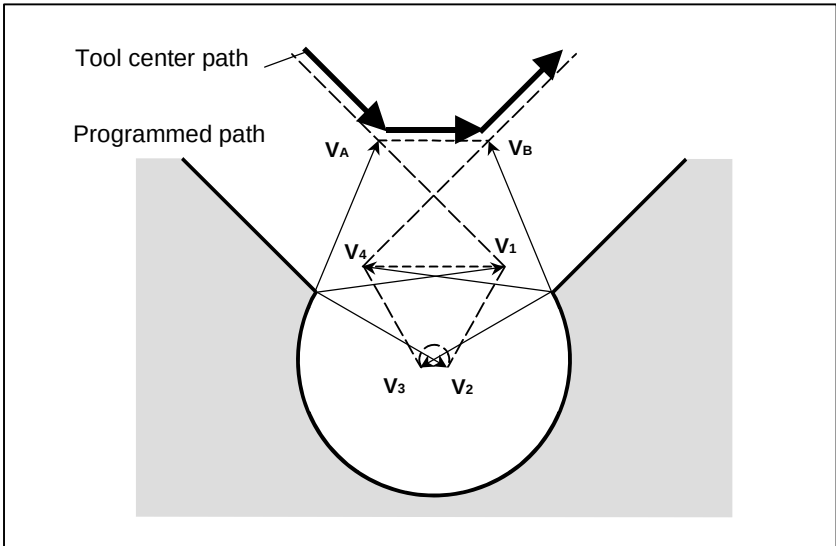
None.

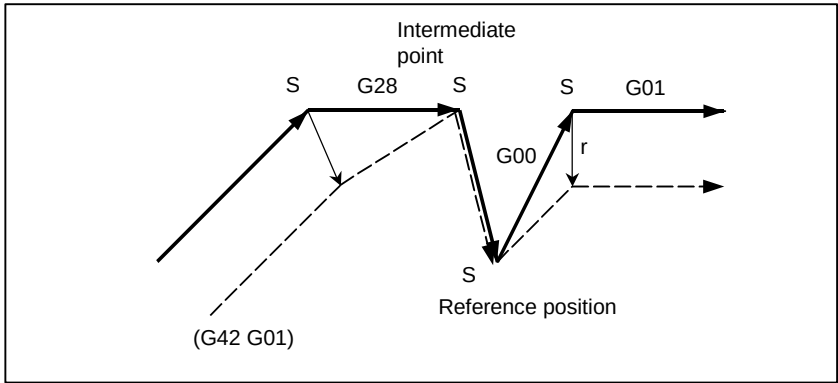
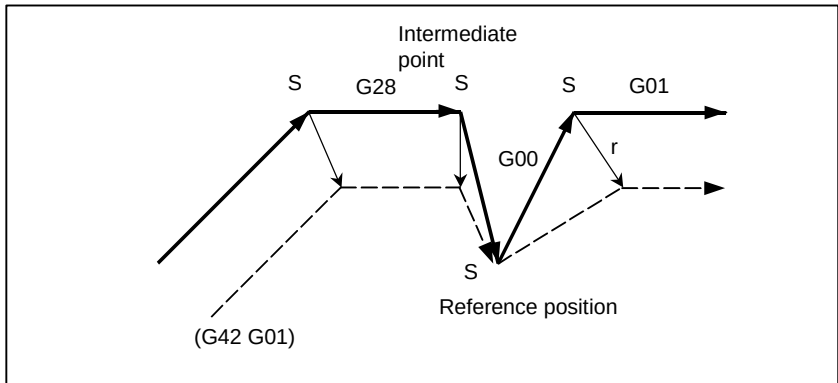
K.26 CUTTER COMPENSATION

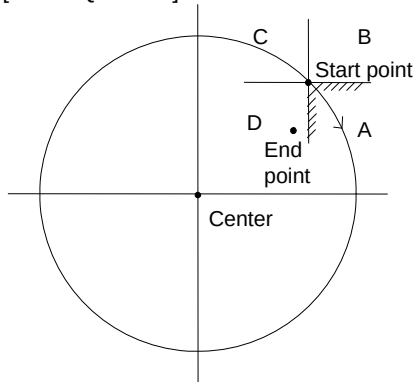
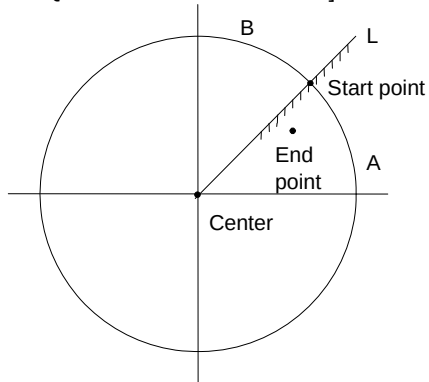
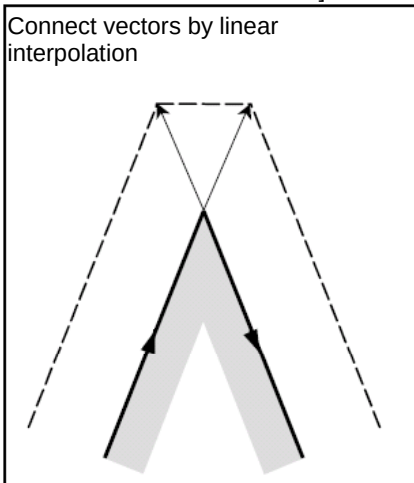
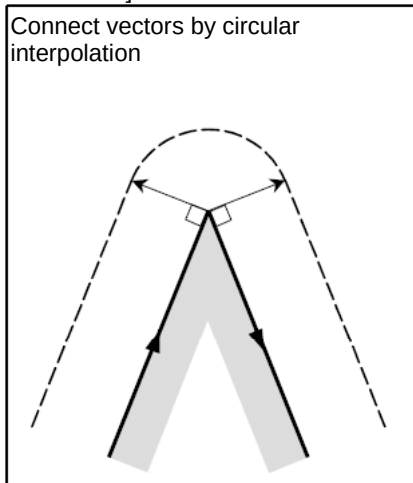
K.26.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
Cutter compensation	In Series 0i-PD, the cutter compensation C functions of Series 0i-PC are collectively referred to as cutter compensation.	
Corner circular interpolation (G39)	Enabled by setting 1 in bit 2 (G39) of parameter No. 5008.	Available. It is included in cutter compensation. Since corner circular interpolation (G39) is always enabled, bit 2 (G39) of parameter No. 5008 is not available.
Cutter compensation in MDI operation	Cutter compensation C is not available in MDI operation.	Cutter compensation is also available in MDI operation.
Single block stop position during the cutter compensation mode	The single block stop position differs as shown below.  Series 0i-PD single block stop Series 0i-PC single block stop	

Function	Series 0i-PC	Series 0i-PD
Function to change the compensation direction intentionally (IJ type vector, KI type vector, and JK type vector)	- Not available.	- At the start of or during the cutter 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.
Stop position upon an overcutting alarm	- If the specified radius value for circular interpolation is smaller than that for cutter compensation, as in the example below, performing compensation inwardly through cutter compensation causes overcutting, generating an alarm and stopping the tool. The stop position differs. [When single block stop occurs in the preceding block in Series 0i-PC] Since the tool moves until it reaches the end point of the block (P₃ in the figure), overcutting may result. [When single block stop does not occur in the preceding block in Series 0i-PC] The tool stops immediately after executing the block (P₂ in the figure). [In the case of Series 0i-PD] Since the tool stops at the start point of the block (P₁ in the figure), regardless of the single block state, overcutting can be prevented.	
Single block stop in a block created internally for cutter 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, single block stop is: 0: Not performed. 1: Performed. This parameter is used to check a program including cutter compensation.

Function	Series 0i-PC	Series 0i-PD
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 0i-PC]		
 The diagram shows a circular tool path within a V-shaped groove. The programmed path is indicated by a dashed line. The tool center path is shown as a solid line that follows the programmed path from point V_1 to point V_4. Interfering vectors V_2 and V_3 are shown but not followed by the tool center path.		
[In the case of Series 0i-PD]		
 The diagram shows a circular tool path within a V-shaped groove. The programmed path is indicated by a dashed line. The tool center path is shown as a solid line that follows the programmed path from point V_A to point V_B. Interfering vectors V_1, V_4, V_2, and V_3 are shown but not followed by the tool center path.		
Number of blocks to be read in the cutter 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 0i-PC (3 blocks) is assumed.

Function	Series 0i-PC	Series 0i-PD
When circular interpolation is specified that causes the center to coincide with the start or end point during the cutter compensation mode	- Alarm PS0038 is issued, and the tool stops at the end point of the block preceding the circular interpolation block.	- Alarm PS0041 is issued, and the tool stops at the start point of the block preceding the circular interpolation block.
Behavior when automatic reference position return is specified during the cutter compensation mode	- Depends on bit 2 (CCN) of parameter No. 5003. [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-PD] 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. 	- Bit 2 (CCN) of parameter No. 5003 is not available. The tool always behaves as when CCN is set to 1.

Function	Series 0i-PC	Series 0i-PD
Travel distance judgment method for circular interpolation in cutter 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 0i-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 mode	- Connected by linear interpolation. [When CCC = 0 or for Series 0i-C]  Connect vectors by linear interpolation	- Depends on bit 2 (CCC) of parameter No. 19607. [When CCC = 1]  Connect vectors by circular interpolation

K.26.2 Differences in Diagnosis Display

None.

K.27 RAPID TRAVERSE OVERRIDE

K.27.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
The parameter to validate rapid traverse override function for T or C axis	Either invalid (1) or valid (0) is selected by following parameter. Parameter TCO (No.16052#1)	Either invalid (1) or valid (0) is selected for each axis by following parameter. Parameter OVEx (No.16844#1)

K.27.2 Differences in Diagnosis Display

None.

K.28 POSITIONING BY OPTIMUM ACCELERATIONS

K.28.1 Differences in Specifications

The function of "changing rapid traverse rate, time constant and servo loop gain according to the positioning distance" for Series 0i-PC is integrated into the function of "positioning by optimum accelerations" in Series 0i-PD.

Function	Series 0i-PC	Series 0i-PD
Parameters to validate this function	Either valid (1) or invalid (0) is selected for each axis kind by following parameters. (For X or Y axis) Parameter KLV (No.16050#7) : Changing rapid traverse rate and time constant (7 levels) Parameter LPG (No.16051#4) : Changing servo loop gain (7 levels) (For T axis) Parameter KLT (No.16051#3) : Changing time constant and servo loop gain (3 levels) (For C axis) Parameter KLC (No.16050#4) : Changing rapid traverse rate and time constant (3 levels)	For Changing rapid traverse rate and time constant (7 levels), either valid (1) or invalid (0) is selected for each axis by following parameter. Parameter OADx (No.6131#0) For Changing servo loop gain (7 levels), either valid (1) or invalid (0) is selected by following parameter. (All axes in same path are affected by this parameter.) Parameter ILG (No.6132#0)
Parameters to set distance for changing	Following parameters are used for each axis kind. (For X or Y axis) Parameter No.16055 to 16060 : mm Parameter No.16061 to 16066 : inch (For T axis) Parameter No.16117 to 16118 (For C axis) Parameter No.16140 to 16141	Following parameters are used for each axis. Parameter No.6136 to 6138 Parameter No.11230 to 11232

Function	Series 0i-PC	Series 0i-PD
Parameters to set rapid traverse rate for each level	Following parameters are used for each axis kind. (For X axis) Parameter No.16067 to 16073 (For Y axis) Parameter No.16081 to 16087 (For C axis) Parameter No.16142 to 16144	Following parameters are used for each axis. Parameter No.6161 to 6167
Parameters to set time constant for each level	Following parameters are used for each axis kind. (For X axis) Parameter No.16074 to 16080 (For Y axis) Parameter No.16088 to 16094 (For T axis) Parameter No.16119 to 16121 (For C axis) Parameter No.16145 to 16147	Following parameters are used for each axis. Parameter No.6171 to 6177
Parameters to set servo loop gain for each level	Following parameters are used for each axis kind. (For X axis) Parameter No.16103 to 16109 (For Y axis) Parameter No.16110 to 16116 (For T axis) Parameter No.16122 to 16124	Following parameters are used for each axis. Parameter No.6181 to 6187 If changing servo loop gain is valid and 0 is set to these parameters for an axis, servo loop gain of this axis is not changed.
Parameters to set time constant of bell-shaped acceleration / deceleration for each level	None.	Following parameters are used for each axis. Parameter No.6191 to 6197
Parameters to set time during which PF is set to 1 prior to end of positioning for each level	Following parameters are used for each axis kind. (For X axis) Parameter No.16013 to 16019 (For Y axis) Parameter No.16020 to 16026 (For C axis) Parameter No.16027 to 16029	Following parameters are used for each axis. Parameter No.16857 to 16863

K.28.2 Differences in Diagnosis Display

None.

K.29 CONSTANT CONTROL OF POSITIONING TIME

K.29.1 Differences in Specifications

Function	Series 0i-PC	Series 0i-PD
The parameter to validate this function	Either valid (1) or invalid (0) is selected by following parameter. Parameter PCT (No.16050#6)	Either valid (1) or invalid (0) is selected for each axis by following parameter. Parameter PT2x (No.16844#6)

Function	Series 0i-PC	Series 0i-PD
Parameters to set positioning time of level 1 (For override of 100% or 75%)	Following parameters are used for each axis kind. (For X axis) Parameter No.16095 (For Y axis) Parameter No.16099	Following parameter is used for each axis. Parameter No.16878
Parameters to set positioning time of level 2 (For override of 100% or 75%)	Following parameters are used for each axis kind. (For X axis) Parameter No.16097 (For Y axis) Parameter No.16101	Following parameter is used for each axis. Parameter No.16879
Parameters to set positioning time of level 1 (For override of 50% or 25%)	Following parameters are used for each axis kind. (For X axis) Parameter No.16096 (For Y axis) Parameter No.16100	Following parameter is used for each axis. Parameter No.16880
Parameters to set positioning time of level 2 (For override of 50% or 25%)	Following parameters are used for each axis kind. (For X axis) Parameter No.16098 (For Y axis) Parameter No.16102	Following parameter is used for each axis. Parameter No.16881
The parameter to double setting value for positioning time	Either double (1) or not (0) is selected by following parameter. Parameter CT2 (No.16050#5)	Positioning time becomes double by doubling value of parameter No.16878 to 16881.

K.29.2 Differences in Diagnosis Display

None.

INDEX

<Number>

10.4" LCD	346
8.4" LCD/MDI	345

<A>

ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)	66
ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED	658
Actual Feedrate Display	531
AI ADVANCED PREVIEW CONTROL	861
AI ADVANCED PREVIEW CONTROL (G05.1)	300
ALARM AND SELF-DIAGNOSIS FUNCTIONS	434
Alarm Display	341, 434
ALARM HISTORY DISPLAY	436
ALARM LIST	806
Altering a Word	485
Arc (G77)	121
ARITHMETIC AND LOGIC OPERATION	256
Automatic Acceleration/Deceleration	322
Automatic Coordinate System Setting	63
Automatic Corner Override	45
AUTOMATIC INSERTION OF SEQUENCE NUMBERS	474
AUTOMATIC OPERATION	333, 380
Automatic override for inner corners (G62)	45
AUTOMATIC REPOSITIONING (G75)	135
Automatic Setting for Grid Position Matching	315
Automatic Setting of the Safety Zone	431
AUXILIARY FUNCTION	95
AUXILIARY FUNCTION (M FUNCTION)	95
Axis Configuration for Axis Synchronous Control	311
Axis Control by PMC	375
AXIS CONTROL FUNCTIONS	311
Axis status display	420
Axis Synchronization Control	374
AXIS SYNCHRONOUS CONTROL	311, 862
Axis Synchronous Control Torque Difference Alarm	318

Background Editing	543
BACKING UP VARIOUS DATA ITEMS	659
Base Point Command (G72)	117
Base Point Command of Multi-Piece Machining (G98)	145
Battery in the PANEL <i>i</i> (3 VDC)	664
BENDING COMPENSATION (G38, G39)	152
Block in which Punching is Made	71
Blocks Where C-axis Command is Possible	323
Bolt Hole Circle (G26)	118
BRANCH AND REPETITION	264

<C>

C AXIS CONTROL (DIE ANGLE INDEXING)	321
C-axis Command and its Operation	323

C-axis Command in Nibbling Mode	325
CHANGING PROGRAM ATTRIBUTES	513
Changing Workpiece Coordinate System	60
CHARACTER-TO-CODES CORRESPONDENCE TABLE	799
Check by Running the Machine	334
CHECKING BY DIAGNOSTIC DISPLAY	437
CIRCULAR INTERPOLATION	849
CIRCULAR INTERPOLATION (G02, G03)	28
Circular Nibbling (G68)	75
Circular Punch Commands (G46 and G47)	157
CODES AND RESERVED WORDS USED IN CUSTOM MACROS	283
Color Setting Screen	591
COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION	16
Command for Restarting Machining Multiple Products	150
COMPACT-TYPE MDI KEY INPUT	505
Compensating Backlash Along the C-axis for Each Tool Group	326
Compensating the Position of the C-axis	326
COMPENSATION FUNCTION	160
Conditional Branch (IF Statement)	265
Confirmation of a data update during the data setting process	418
Confirmation of incremental input	416
Confirmation of the deletion of all data	418
Confirmation of the deletion of the program	417
Confirmation of the start from a middle block	420
CONSTANT CONTROL OF POSITIONING TIME	888
CONTROLLED AXES	20
Controlling the C-axis	158
CONTROLLING THE TURRET-AXIS (T-AXIS)	90
CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION	475
COORDINATE SYSTEM	56
Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System	13
COORDINATE SYSTEM ROTATION (G84, G85)	224
COORDINATE VALUE AND DIMENSION	66
Copy Specifying a Program Number	501
Copying a Part of a Program	491
Copying an Entire Program	497
COPYING/MOVING PROGRAMS	491
Copying/Moving to the Key-in Buffer	502
CREATING PROGRAMS	473
CREATING PROGRAMS USING THE MDI PANEL	473
Current Block Display Screen (Only for the 8.4-Inch Display Unit)	541
Current Position Display	340
CUSTOM MACRO	231, 858
Cut at Angle (G89)	130
CUTTER COMPENSATION	882
CUTTING FEED	42

CUTTING FEEDRATE CONTROL	43	Displaying and setting the FSSB amplifier setting screen	597
<D>		Displaying and setting the FSSB axis setting screen.....	598
DATA INPUT/OUTPUT	440	Displaying and setting the high-precision setting screen	611
Data range check.....	421	Displaying and setting the miscellaneous setting screen	613
DATA SERVER FUNCTION	881	Displaying and setting the servo gain tuning screen	601
DATA TYPE.....	781	Displaying and setting the servo setting screen.....	599
DECIMAL POINT PROGRAMMING.....	69	Displaying and setting the servo tuning screen	613
DEFINITION OF WARNING, CAUTION, AND NOTE	s-1	Displaying and Setting the Software Operator's Panel	558
Deleting a Block.....	486	Displaying and Setting the Workpiece Origin Offset Value	555
DELETING A PROGRAM.....	513	Displaying and Switching the Display Language	561
Deleting a Word	485	Displaying the Floppy Cassette Screen	469
Deleting All Programs.....	490	Displaying the Memory Card Screen	461
DELETING BLOCKS	486	Displaying the menu screen and selecting a menu item	593
Deleting Multiple Blocks	486	Displaying the Program Number, and Sequence Number	633
Deleting One Program.....	490	DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION	633
DELETING PROGRAMS	490	Displaying the Safety Zones and Tool Zone	432
Deletion of Stored Macros	145	Displaying the Status and Warning for Data Setting or Input/Output Operation	634
DESCRIPTION OF PARAMETERS.....	671	DISTANCE CODED LINEAR SCALE INTERFACE	372
Details of Functions	290	Distance Coded Rotary Encoder	374
DETAILS OF TOOL RADIUS COMPENSATION	175	DNC OPERATION.....	385
DIFFERENCES FROM SERIES 0i-PC	848	DRY RUN.....	404
Direct Input of Workpiece Origin Offset Value Measured	555	DWELL	50
DISPLAY	339	<E>	
Display of Run Time and Parts Count.....	533	EDIT DISABLE ATTRIBUTE.....	480
Display of updated modal information.....	419	EDITING A PROGRAM	336,536
Displaying and Entering Setting Data	549	EDITING OF CUSTOM MACROS	503
Displaying and Operating the File List	462,469	EDITING PROGRAMS.....	480
Displaying and Setting Custom Macro Common Variables.....	557	EMBEDDED ETHERNET OPERATIONS	465
DISPLAYING AND SETTING DATA	336	EMERGENCY STOP	408
Displaying and setting items on the initial tool registration screen.....	567	ERROR MESSAGE AND NOTE.....	843
Displaying and Setting Items on the Safety Zone Setting Screen.....	578	Exact Stop (G09, G61), Cutting Mode (G64)	44
Displaying and setting items on the screen for the number of press operations.....	570	EXAMPLE.....	647
Displaying and setting items on the tool figure registration screen (for drawing figures)	572	Explanation of Operations.....	832
Displaying and setting items on the tool figure registration screen for multiple tools (for drawing figures)	575	EXPLANATION OF THE MDI UNIT.....	349
Displaying and setting items on the tool number registration screen.....	569	EXTERNAL DATA INPUT	879
Displaying and setting items on the tool number registration screen for multiple tools	574	EXTERNAL I/O DEVICES.....	360
Displaying and Setting Items on the Tool Registration Screen for Multiple Tools.....	574	EXTERNAL MOTION FUNCTION.....	86
Displaying and Setting Items on the Tool Registration Screens	567	EXTERNAL OUTPUT COMMANDS.....	284
Displaying and Setting Parameters.....	582	EXTERNAL SUBPROGRAM CALL (M198).....	391,872
Displaying and Setting Pitch Error Compensation Data	584	<F>	
Displaying and Setting Run Time, Parts Count, and Time	553	F1-digit (Programmable Rapid Traverse Override)	42
Displaying and setting the axis setting screen.....	596	FANUC DOUBLE-BYTE CHARACTER CODE TABLE.....	800
		FEED FUNCTIONS	40
		FEED-FEED FUNCTION	12
		FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS.....	47

- FEEDRATE OVERRIDE 402
- File Format.....461
- FLOPPY CASSETTE SCREEN 469
- FTP File Transfer Function.....465
- Function Keys 351
- FUNCTION KEYS AND SOFT KEYS.....350
- Functions that are Used When Data is Set 415
- Functions that are Used when the Program is Executed.....418
- FUNCTIONS TO SIMPLIFY PROGRAMMING 116
- <G>**
 - G CODE LIST 23
 - G00 Command in Nibbling Mode.....82
 - G01, G02, and G03 Commands in Nibbling Mode 83
 - GENERAL 3,9,331
 - General Screen Operations.....350
 - GENERAL WARNINGS AND CAUTIONS s-2
 - GOTO Statement Using Stored Sequence Numbers 264
 - GRAPHIC DISPLAY SCREEN AND DRAWING 644
 - GRAPHIC FUNCTION 638
 - Graphical Conversational Programming Screen 541
 - Grid (G78, G79).....122
- <H>**
 - Heading a Program.....483
 - HELICAL INTERPOLATION 850
 - HELICAL INTERPOLATION (G02, G03).....32
 - HIGH-SPEED CUTTING FUNCTIONS.....300
 - HIGH-SPEED SKIP SIGNAL (G33).....36
 - How to Indicate Command Dimensions for Moving the
Tool (Absolute and Incremental Programming) 15
 - How to View the Position Display Change without
Running the Machine 336
- <I>**
 - INCH/METRIC CONVERSION (G20, G21) 66
 - INCREMENT SYSTEM.....20,322
 - Incremental Command Just After Pattern Function 130
 - INCREMENTAL FEED 364
 - Input data range check 415
 - Input/output format of pitch error compensation data..448
 - INPUT/OUTPUT ON EACH SCREEN.....443
 - INPUT/OUTPUT ON THE ALL IO SCREEN.....454
 - Inputting a program.....443
 - Inputting and Outputting a Program.....443
 - Inputting and Outputting Custom Macro Common
Variables.....449
 - Inputting and Outputting Offset Data.....446,457
 - Inputting and Outputting Operation History Data.452,460
 - Inputting and Outputting Parameters.....445,456
 - Inputting and Outputting Pitch Error Compensation
Data 447
 - Inputting and Outputting Workpiece Coordinates
System Data.....451,460
 - Inputting custom macro common variables.....449
 - Inputting offset data 446
 - Inputting Outputting Tool Data.....453
 - Inputting parameters.....445
 - Inputting pitch error compensation data.....447
 - Inputting tool data.....453
 - Inputting workpiece coordinate system data 451
 - Inputting/Outputting a File.....463,470
 - Inputting/Outputting a Program 455
 - Inputting/Outputting Custom Macro Common Variables
.....459
 - Inputting/Outputting Pitch Error Compensation Data..458
 - Inserting a Word 484
 - INSERTING, ALTERING AND DELETING A
WORD.....481
 - Interference Check 211
 - Interference check alarm function.....215
 - Interference check avoidance function.....216
 - Internal circular cutting feedrate change.....47
 - INTERPOLATION FUNCTIONS.....25
 - INTERRUPTION TYPE CUSTOM MACRO.....288,860
 - ISO/ASCII CODE CONVERSION TOOL 845
- <J>**
 - JOG FEED (JOG) 363
- <K>**
 - Key Input and Input Buffer 359
- <L>**
 - Limitations 653
 - Line at Angle (G76) 120
 - LINEAR AND CIRCULAR PUNCH COMMAND 154
 - LINEAR INTERPOLATION (G01) 26
 - Linear Nibbling (G69) 79
 - Linear Punch Command (G45) 154
 - LINEAR SCALE WITH DISTANCE-CODED
REFERENCE MARKS (SERIAL).....376
 - List of Error Message.....843
 - LIST OF FUNCTIONS AND PROGRAM FORMAT 786
 - List of Functions of PC Tool.....831
 - LOCAL COORDINATE SYSTEM.....63,856
- <M>**
 - M COMMAND FOR SWITCHING THE PUNCH
AND LASER MODE 100
 - MACHINE COORDINATE SYSTEM.....56
 - MACHINE LOCK AND AUXILIARY FUNCTION
LOCK.....401
 - Macro Call 142,268
 - Macro Call Using a G Code 273
 - Macro Call Using a G Code (Specification of Multiple
Definitions) 275
 - Macro Call Using an M Code 275
 - Macro Call Using an M Code (Specification of Multiple
Definitions) 276
 - MACRO STATEMENTS AND NC STATEMENTS..263
 - Macro Storage Capacity 144
 - MAKING A PROGRAM COMPACT 514
 - MANUAL ABSOLUTE ON.....368,878
 - Manual Continuous Feed, Incremental Feed, Manual
Reference Point Return 322

MANUAL HANDLE FEED	365,867	Outputting a program	444
MANUAL HANDLE INTERRUPTION	393	Outputting all history data.....	630
MANUAL OPERATION.....	331,362	Outputting custom macro common variables.....	450
MANUAL PUNCH.....	407	Outputting offset data.....	447
MANUAL REFERENCE POSITION RETURN..	362,853	Outputting operation history data.....	452
Maximum incremental value check	421	Outputting parameters.....	445
Maximum Programmable Dimension	322	Outputting pitch error compensation data	448
MAXIMUM STROKE.....	21	Outputting tool data	453
MDI OPERATION	382	Outputting workpiece coordinate system data	452
MEMORY AND CALL BY A/B MACRO	134	Overall Position Display	529
MEMORY CARD SCREEN.....	461	OVERTRAVEL	408
MEMORY OPERATION	380	OVERVIEW	40,175,665
METHOD OF REPLACING BATTERY	660	Overview of the History Function.....	624
Methods of Alarm Recovery by Synchronous Error		OVERVIEW OF TOOL COMPENSATION (G40-G42)	
Check.....	317		170
MIRROR IMAGE	399	OVERWRITING FILES ON A MEMORY CARD.....	441
Miscellaneous.....	860		
Modal Call: Call After the Move Command (G66).....	272	<P>	
Moving a Part of a Program	494	Parameter Setting Support Screen.....	593
Moving an Entire Program	499	PARAMETERS	671
Multi-Piece Machining Commands (G73, G74).....	148	PART DRAWING AND TOOL MOVEMENT	12
MULTI-PIECE MACHINING FUNCTION	145	PART PROGRAM STORAGE SIZE / NUMBER OF	
MULTIPLE M COMMANDS IN A SINGLE BLOCK..	97	REGISTERABLE PROGRAMS.....	102
MULTIPLE TOOL CONTROL.....	90	Parts Count Display, Run Time Display	341
MULTI-STEP SKIP (G33)	35	PASSWORD FUNCTION.....	504
		Password modification	564
<N>		PATTERN FUNCTION.....	116
NAMES OF AXES	20	Pattern Function, Nibbling Function and C-axis	
NAMING RULES.....	840	Command	324
Naming Rules of Folder	841	PC TOOL FOR MEMORY CARD PROGRAM	
Naming Rules of Program File.....	840	OPERATION/EDITING	831
Nesting Call of Macros	143	Periodic Maintenance Screen.....	615
Next Block Display Screen	539	PLANE SELECTION	65
NIBBLING BY M FUNCTION.....	82	PMC AXIS CONTROL	868
NIBBLING FUNCTION.....	73	Position Display in the Relative Coordinate System....	527
NOMOGRAPHS.....	793	Position Display in the Workpiece Coordinate System	526
NORMAL DIRECTION CONTROL		POSITIONING & PRESSING OFF (G70).....	73
(G40.1,G41.1,G42.1, or G150,G151, G152)	166	POSITIONING (G00).....	25
Note.....	375,844	POSITIONING BY OPTIMUM ACCELERATIONS..	887
Notes on Circular Nibbling (G68) and Linear Nibbling		Positioning in Smaller Angle Rotating Direction.....	323
(G69)	80	Power Disconnection	344
Notes on Nibbling by M Function.....	86	POWER MATE CNC MANAGER	882
Notes on Pattern Functions	133	POWER ON/OFF	343
NOTES ON READING THIS MANUAL	5	PREPARATORY FUNCTION (G FUNCTION)	22
NOTES ON VARIOUS KINDS OF DATA.....	5	PRESS FUNCTION.....	71
NUMBER OF CONTROLLED AXES.....	20	Press Motion by RAM-axis Control.....	327
		Prevention of Overcutting Due to Tool Radius	
<O>		Compensation.....	208
Operating Monitor Display	534	Procedure for Reference Position Establishment	372
OPERATION	639	PROCESSING MACRO STATEMENTS	281
Operation confirmation function setting screen	423	PROGRAM ATTRIBUTES.....	101
Operation history.....	624	Program Check Screen.....	540
Operation level setting	562	PROGRAM CODE LIST.....	784
Operation to be performed if an interference is judged to		PROGRAM COMPONENTS OTHER THAN	
occur.....	214	PROGRAM SECTIONS	104
OPERATIONAL DEVICES	343	PROGRAM CONFIGURATION	17,103
Operational Notes.....	93	Program Contents Display	535
Output format of tool data.....	454	Program Display	339

Program List Screen	538	SAFETY ZONE CHECK	427
PROGRAM MANAGEMENT	101,508	SCALING (G50, G51)	160
Program Screen for MDI Operation	538	SCHEDULE OPERATION	387
PROGRAM SEARCH	487	SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION	636,877
PROGRAM SECTION CONFIGURATION	106	SCREEN HARD COPY FUNCTION	471
PROGRAMMABLE PARAMETER INPUT	296	SCREENS DISPLAYED BY FUNCTION KEY 	526
PROGRAMMABLE PARAMETER INPUT (G10)	296,860	SCREENS DISPLAYED BY FUNCTION KEY 	535
Prohibition of the absolute input by the soft key	417	SCREENS DISPLAYED BY FUNCTION KEY 	547
Protection level setting	565	SCREENS DISPLAYED BY FUNCTION KEY 	582
Protection of Data at Eight Levels	562	SCREENS DISPLAYED BY FUNCTION KEY 	633
PUNCH	406	SECOND AUXILIARY FUNCTIONS (B CODES)	98
Punch Forbidden Area and Approach Forbidden Area (Type A)	428	SELECTING A DEVICE	508
Punch Forbidden Area and Approach Forbidden Area (Type B)	429	Selecting a From Cassette as a Device	512
PUNCH FUNCTION (1-CYCLE PRESSING)	71	SELECTING A MAIN PROGRAM	514
<R>		Selecting a Memory Card Program as a Device	509
Radius (G88)	129	Selecting a Workpiece Coordinate System	59
RADIUS DIRECTION ERROR AT CIRCLE CUTTING	796	Selecting operation history signals	629
RAM-AXIS CONTROL	326	SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION	16
RAM-axis Up Motion	327	Sequence Number Comparison and Stop	551
RANGE OF COMMAND VALUE	791	SEQUENCE NUMBER SEARCH	488,873
RAPID TRAVERSE	41	Servo Setting	587
RAPID TRAVERSE OVERRIDE	403,887	Servo Tuning	590
Rapid Traverse Rate by F Command	41	Setting a Workpiece Coordinate System	58
READING PARAMETERS	262	SETTING AND DISPLAY UNITS	344
REFERENCE POSITION	52	SETTING AND DISPLAYING DATA	516
Reference Position (Machine-specific Position)	12	Setting and Displaying the Tool Compensation Value	547
REFERENCE POSITION RETURN	52,373	Setting of Machining Method for Multi-Piece Machining	149
Register and Change of Tool Life Management Data	93	Setting RAM-axis Motion Pattern	326
REGISTERING CUSTOM MACRO PROGRAMS	283	Setting Screen	422
REGISTERING THE TOOL FIGURE	639	Setting the Safety Zone	430
RELATED PARAMETERS	101	Setting the Tool Shape Area	430
Relationship between the Multiple-Tool System and the C Axis	92	SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE	797
Relationship with Absolute/Incremental Command (G90/G91)	323	Share Proofs (G86)	125
Repetition (WHILE Statement)	266	Simple Call (G65)	269
REPLACING	503	Simultaneously Controlled Axes	322
Replacing batteries	666	SINGLE BLOCK	405
Replacing Battery for Absolute Pulsecoders	665	SKIP FUNCTION	851
Replacing Battery for CNC Control Unit	660	SKIP FUNCTION (G33)	33
Replacing the batteries in a separate battery case	666	SKIP POSITION MACRO VARIABLE IMPROVEMENT	36
Replacing the battery built into the servo amplifier	667	Small MDI Unit (ONG Key)	347
RESET AND REWIND	878	Soft Keys	352
RESTRICTIONS	287	Specification Method	289
RETURN FROM THE ALARM SCREEN	438	SPECIFYING DRAWING PARAMETERS	640
ROTARY AXIS ROLL-OVER	320	SPECIFYING THE S CODE WITH A BINARY CODE	87
ROUTINE MAINTENANCE	657	Square (G87)	127
RULES OF CHARACTERS IN PROGRAM FILE	841	Standard MDI Unit (ONG Key)	347
RUN HOUR AND PARTS COUNT DISPLAY	867	STANDARD PARAMETER SETTING TABLES	783
<S>		Start check signal	419
S FUNCTION	87		
SAFETY FUNCTIONS	408		
SAFETY PRECAUTIONS	s-1		

Storage and Call of Multiple Macros (Macro Numbers 90 to 99)	144	<V>	VARIABLES	231
Storage of Macros	141		VIRTUAL MDI KEY	650
STORED PITCH ERROR COMPENSATION.....	876		VIRTUAL MDI KEY FUNCTION	650
STORED STROKE CHECK	409,874	<W>		
STROKE LIMIT CHECK BEFORE MOVE	412		Warning display during a reset in program operation ..	422
SUBPROGRAM (M98, M99).....	112		Warning Messages	360
Subprogram Call Using a Specific Address	279		WARNINGS AND CAUTIONS RELATED TO HANDLING.....	s-4
Subprogram Call Using an M Code	277		WARNINGS AND CAUTIONS RELATED TO PROGRAMMING.....	s-3
Subprogram Call Using an M Code (Specification of Multiple Definitions)	278		WARNINGS RELATED TO DAILY MAINTENANCE.....	s-6
Subprogram Calls Using a T Code	278		Word Search.....	482
Synchronous Error Check	316		WORKPIECE COORDINATE SYSTEM	58,855
Synchronous Establishment	314		Workpiece Coordinate System Preset	530
System Configuration Screen.....	622		Workpiece Coordinate System Preset (G92.1).....	61
SYSTEM VARIABLES.....	236		Workpiece origin offset range setting screen	426
<T>			WRONG OPERATION PREVENTION FUNCTIONS	414
T COMMAND NEGLECT	89	<Y>		
TANDEM CONTROL	321		Y-AXIS CRACK CANCEL	159
T-axis Command Ignore Signal TNG and C-axis Command	325			
TEST OPERATION.....	401			
TESTING A PROGRAM.....	334			
Tool Compensation	92			
TOOL COMPENSATION MEMORY	857			
TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	223			
TOOL DATA INPUT	298			
TOOL FUNCTION (T FUNCTION).....	88			
Tool Life	94			
Tool Life Management Data	93			
TOOL LIFE MANAGEMENT FUNCTION	93			
TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION.....	10			
TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION.....	332			
Tool Movement in Offset Mode.....	184			
Tool Movement in Offset Mode Cancel.....	202			
Tool Movement in Start-up.....	179			
TOOL MOVEMENT RANGE - STROKE.....	19			
Tool Number	91			
TOOL OFFSET.....	90			
Tool offset range setting screen	424			
TOOL PATH AT CORNER	793			
Tool Radius Compensation for Input from MDI	222			
TOOL SELECTION FUNCTION	88			
TORQUE LIMIT SKIP	36			
Turning on the Power.....	343			
<U>				
U/V/W MACRO FUNCTION	141			
Unconditional Branch (GOTO Statement).....	264			
Usable Characters in Program File.....	841			
Usage Notes	831			

REVISION RECORD

Edition	Date	Contents
01	Feb., 2011	

