



GE Fanuc Automation

Computer Numerical Control Products

*Series 0 / 00 / 0-Mate
for Machining Center*

Operator's Manual

GFZ-61404E/07

August 1997

Warnings, Cautions, and Notes as Used in this Publication

Warning

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

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

Caution

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

Note

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

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

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

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units. It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.

Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

Contents

1. DEFINITION OF WARNING, CAUTION, AND NOTE	s-2
2. GENERAL WARNINGS AND CAUTIONS	s-3
3. WARNINGS AND CAUTIONS RELATED TO PROGRAMMING	s-5
4. WARNINGS AND CAUTIONS RELATED TO HANDLING	s-7
5. WARNINGS RELATED TO DAILY MAINTENANCE	s-9

1

DEFINITION OF WARNING, CAUTION, AND NOTE

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

WARNING

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

CAUTION

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

NOTE

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

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

2 GENERAL WARNINGS AND CAUTIONS

WARNING

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
2. Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
4. When using a tool compensation function, thoroughly check the direction and amount of compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
5. The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.
7. This manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

WARNING

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

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.

To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

3

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

WARNING

1. Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.

Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming.

Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Function involving a rotation axis

When programming polar coordinate interpolation or normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely.

Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. Inch/metric conversion

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

5. Constant surface speed control

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

WARNING**6. Stroke check**

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

7. Tool post interference check

A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user.

After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8. Absolute/incremental mode

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

9. Plane selection

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

10. Torque limit skip

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

11. Programmable mirror image

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

12. Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.

Before issuing any of the above commands, therefore, always cancel compensation function mode.

4

WARNINGS AND CAUTIONS RELATED TO HANDLING

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

WARNING

1. Manual operation

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

2. Manual reference position return

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

3. Manual numeric command

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

4. Manual handle feed

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

5. Disabled override

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

6. Origin/preset operation

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

WARNING**7. Workpiece coordinate system shift**

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.

If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

8. Software operator's panel and menu switches

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

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

9. Manual intervention

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

10. Feed hold, override, and single block

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

11. Dry run

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

12. Cutter and tool nose radius compensation in MDI mode

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

13. Program editing

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


5

WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

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

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

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

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.


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

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.

Refer to the maintenance section of this 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 CRT screen.


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

Refer to the maintenance section of this 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.

SAFETY PRECAUTIONS	s-1
---------------------------------	------------

I. GENERAL

1. GENERAL	3
1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL	5
1.2 NOTES ON READING THIS MANUAL	7

II. PROGRAMMING

1. GENERAL	11
1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION	12
1.2 FEED-FEED FUNCTION	13
1.3 PART DRAWING AND TOOL MOVEMENT	14
1.3.1 Reference Position (Machine-Specific Position)	14
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System	15
1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands	18
1.4 CUTTING SPEED – SPINDLE SPEED FUNCTION	19
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION	20
1.6 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION	20
1.7 PROGRAM CONFIGURATION	21
1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM	24
1.9 TOOL MOVEMENT RANGE – STROKE	25
2. CONTROLLED AXES	26
2.1 CONTROLLED AXES	27
2.2 NAME OF AXES	27
2.3 INCREMENT SYSTEM	27
2.4 MAXIMUM STROKE	27
3. PREPARATORY FUNCTION (G FUNCTION)	28
4. INTERPOLATION FUNCTIONS	32
4.1 POSITIONING (G00)	33
4.2 SINGLE DIRECTION POSITIONING (G60)	34
4.3 LINEAR INTERPOLATION (G01)	35
4.4 CIRCULAR INTERPOLATION (G02, G03)	37
4.5 HELICAL CUTTING (G02, G03)	41
4.6 CYLINDRICAL INTERPOLATION (G107)	42
4.7 CONSTANT LEAD THREADING (G33)	45
4.8 SKIP FUNCTION (G31)	47

5. FEED FUNCTIONS	49
5.1 GENERAL	50
5.2 RAPID TRAVERSE	53
5.3 CUTTING FEED	54
5.4 CUTTING FEEDRATE CONTROL	58
5.4.1 Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)	59
5.4.2 Automatic Override for Inner Corners (G62)	60
5.4.3 Internal Circular Cutting Feedrate Change	62
5.5 DWELL (G04)	63
5.6 CUTTING FEED LINEAR ACCELERATION/DECELERATION BEFORE INTERPOLATION	64
5.7 CUTTING FEED BLOCK OVERLAP	67
5.8 AUTOMATIC CORNER DECELERATION	68
5.9 FEEDRATE CLAMP BASED ON ARC RADIUS	74
5.10 RAPID TRAVERSE BELL-SHAPED ACCELERATION/DECELERATION	76
6. REFERENCE POSITION	77
7. COORDINATE SYSTEM	81
7.1 MACHINE COORDINATE SYSTEM	82
7.2 WORKPIECE COORDINATE SYSTEM	83
7.2.1 Setting a Workpiece Coordinate System	83
7.2.2 Selecting a Workpiece Coordinate System	84
7.2.3 Changing Workpiece Coordinate System	85
7.2.4 Adding Workpiece Coordinate Systems	88
7.3 LOCAL COORDINATE SYSTEM	90
7.4 PLANE SELECTION	92
8. COORDINATE VALUE AND DIMENSION	93
8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)	94
8.2 POLAR COORDINATE COMMAND (G15, G16)	95
8.3 INCH/METRIC CONVERSION (G20, G21)	98
8.4 DECIMAL POINT PROGRAMMING	99
9. SPINDLE SPEED FUNCTION (S FUNCTION)	100
9.1 SPECIFYING THE SPINDLE SPEED WITH A BINARY CODE	101
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND)	101
9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)	102
10. TOOL FUNCTION (T FUNCTION)	106
10.1 TOOL SELECTION FUNCTION	107
10.2 TOOL LIFE MANAGEMENT FUNCTION	108
10.2.1 Tool Life Management Data	109
10.2.2 Register, Change and Delete of Tool Life Management Data	110
10.2.3 Tool Life Management Command in a Machining Program	111
10.2.4 Tool Life	114

11. AUXILIARY FUNCTION	115
11.1 AUXILIARY FUNCTION (M FUNCTION)	116
11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK	117
11.3 THE SECOND AUXILIARY FUNCTIONS (B CODES)	117
12. PROGRAM CONFIGURATION	118
12.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS	120
12.2 PROGRAM SECTION CONFIGURATION	123
12.3 SUBPROGRAM	129
13. FUNCTIONS TO SIMPLIFY PROGRAMMING	132
13.1 CANNED CYCLE	133
13.1.1 High-speed Peck Drilling Cycle (G73)	137
13.1.2 Left-handed Tapping Cycle (G74)	139
13.1.3 Fine Boring Cycle (G76)	141
13.1.4 Drilling Cycle, Spot Drilling (G81)	143
13.1.5 Drilling Cycle Counter Boring Cycle (G82)	145
13.1.6 Peck Drilling Cycle (G83)	147
13.1.7 Tapping Cycle (G84)	149
13.1.8 Boring Cycle (G85)	151
13.1.9 Boring Cycle (G86)	153
13.1.10 Boring Cycle Back Boring Cycle (G87)	155
13.1.11 Boring Cycle (G88)	157
13.1.12 Boring Cycle (G89)	159
13.1.13 Canned Cycle Cancel (G80)	161
13.2 RIGID TAPPING	164
13.2.1 Rigid Tapping (G84)	165
13.2.2 Left-handed Rigid Tapping Cycle (G74)	167
13.2.3 Peck Rigid Tapping Cycle (G84 or G74)	169
13.2.4 Canned Cycle Cancel (G80)	171
13.3 CANNED GRINDING CYCLE (0-GSC, 0-GSD/II)	172
13.3.1 Plunge Grinding Cycle (G75)	173
13.3.2 Direct Constant-dimension Plunge Grinding Cycle (G77)	175
13.3.3 Continuous-feed Surface Grinding Cycle (G78)	177
13.3.4 Intermittent-feed Surface Grinding Cycle (G79)	179
13.4 GRINDING-WHEEL WEAR COMPENSATION BY CONTINUOUS DRESSING (0-GSC, 0-GSD/II)	181
13.5 AUTOMATIC GRINDING WHEEL DIAMETER COMPENSATION AFTER DRESSING	182
13.5.1 Checking the Minimum Grinding Wheel Diameter (0-GSC, 0-GSD/II)	182
13.6 IN-FEED GRINDING ALONG THE Y AND Z AXES AT THE END OF TABLE SWING (0-GSC, 0-GSD/II)	183
13.7 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING	184
13.8 EXTERNAL MOTION FUNCTION (G81)	186
13.9 SMALL-DIAMETER PECK DRILLING CYCLE	187

14. COMPENSATION FUNCTION	191
14.1 TOOL LENGTH OFFSET (G43, G44, G49)	192
14.2 AUTOMATIC TOOL LENGTH MEASUREMENT (G37)	196
14.3 TOOL OFFSET (G45 TO G48)	200
14.4 CUTTER COMPENSATION B (G39 TO G42)	205
14.4.1 Cutter Compensation Left (G41)	208
14.4.2 Cutter Compensation Right (G42)	210
14.4.3 Corner Offset Circular Interpolation (G39)	212
14.4.4 Cutter Compensation Cancel (G40)	213
14.4.5 Switch between Cutter Compensation Left and Cutter Compensation Right	214
14.4.6 Change of the Cutter Compensation Value	215
14.4.7 Positive/Negative Cutter Compensation Value and Tool Center Path	216
14.5 OVERVIEW OF CUTTER COMPENSATION C (G40 TO G42)	218
14.6 DETAILS OF CUTTER COMPENSATION C	224
14.6.1 General	224
14.6.2 Tool Movement in Start-up	225
14.6.3 Tool Movement in Offset Mode	229
14.6.4 Tool Movement in Offset Mode Cancel	243
14.6.5 Interference Check	249
14.6.6 Overcutting by Cutter Compensation	254
14.6.7 Input Command from MDI	257
14.7 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	258
14.8 SCALING (G50, G51)	260
14.9 COORDINATE SYSTEM ROTATION (G68, G69)	266
14.10 INDEX TABLE INDEXING FUNCTION	272
14.11 NORMAL DIRECTION CONTROL (G150, G151, G152)	275
15. CUSTOM MACRO A	279
15.1 CUSTOM MACRO COMMAND	280
15.1.1 M98 (Single Call)	280
15.1.2 Subprogram Call Using M Code	280
15.1.3 Subprogram Call Using T Code	281
15.1.4 G66 (Modal Call)	282
15.1.5 Argument Specification	283
15.2 CUSTOM MACRO BODY	285
15.2.1 Variables	285
15.2.2 Kind of Variables	286
15.2.3 Operation Instruction and Branch Instruction (G65)	291
15.2.4 Notes on Custom Macro	296
16. CUSTOM MACRO B	297
16.1 VARIABLES	298
16.2 SYSTEM VARIABLES	301
16.3 ARITHMETIC AND LOGIC OPERATION	308
16.4 MACRO STATEMENTS AND NC STATEMENTS	312






16.5	BRANCH AND REPETITION	313
16.5.1	Unconditional Branch (GOTO Statement)	313
16.5.2	Conditional Branch (IF Statement)	313
16.5.3	Repetition (WHILE Statement)	314
16.6	MACRO CALL	317
16.6.1	Simple Call (G65)	317
16.6.2	Modal Call (G66)	322
16.6.3	Macro Call Using G Code	324
16.6.4	Macro Call Using an M Code	325
16.6.5	Subprogram Call Using an M Code	326
16.6.6	Subprogram Calls Using a T Code	327
16.6.7	Sample Program	328
16.7	PROCESSING MACRO STATEMENTS	330
16.8	REGISTERING CUSTOM MACRO PROGRAMS	331
16.9	LIMITATIONS	332
16.10	EXTERNAL OUTPUT COMMANDS	333
16.11	INTERRUPTION TYPE CUSTOM MACRO	337
16.11.1	Specification Method	338
16.11.2	Details of Functions	339
17.	PATTERN DATA INPUT FUNCTION	346
17.1	DISPLAYING THE PATTERN MENU	347
17.2	PATTERN DATA DISPLAY	351
17.3	CHARACTERS AND CODES TO BE USED FOR THE PATTERN DATA INPUT FUNCTION	355
18.	PROGRAMMABLE PARAMETER ENTRY (G10)	357
19.	MEMORY OPERATION USING Series 10/11 TAPE FORMAT	359
20.	HIGH SPEED CYCLE CUTTING	360
20.1	NUMBER OF CONTROL AXES	361
20.2	PULSE DISTRIBUTION	361
20.3	CONFIGURATION OF HIGH SPEED CYCLE CUTTING DATA	362
20.3.1	Number of Registered Cycles	362
20.3.2	Header	362
21.	SIMPLE SYNCHRONOUS CONTROL	365
22.	ROTARY AXIS ROLL-OVER	368
23.	ANGULAR AXIS CONTROL (0-GSC, 0-GSD/II)	369
24.	ADVANCED PREVIEW CONTROL	371

III. OPERATION

1. GENERAL	375
1.1	MANUAL OPERATION 376
1.2	TOOL MOVEMENT BY PROGRAMMING-AUTOMATIC OPERATION 378
1.3	AUTOMATIC OPERATION 380
1.4	TESTING A PROGRAM 382
1.4.1	Check by Running the Machine 382
1.4.2	How to View the Position Display Change without Running the Machine 383
1.5	EDITING A PART PROGRAM 384
1.6	DISPLAYING AND SETTING DATA 385
1.7	DISPLAY 388
1.7.1	Program Display (See Subsections III-11.2.1 and III-11.3.1) 388
1.7.2	Current Position Display (See Section III-11.1.1 to 11.1.3) 389
1.7.3	Alarm Display (See Section III-7.1) 389
1.7.4	Parts Count Display, Run Time Display (See Section III-11.1.5) 390
1.7.5	Graphic Display (See Section III-12) 390
1.8	DATA OUTPUT (SEE CHAPTER III-8) 391
2. OPERATIONAL DEVICES	392
2.1	CRT/MDI PANELS AND LCD/MDI PANELS 393
2.2	FUNCTION KEYS AND SOFT KEYS 398
2.2.1	General Screen Operations 398
2.2.2	Function Keys 399
2.2.3	Key Input and Input Buffer 400
2.2.4	14" CRT Soft Key Configuration 402
2.3	EXTERNAL I/O DEVICES 403
2.3.1	FANUC Handy File 405
2.3.2	FANUC Floppy Cassette 405
2.3.3	FANUC FA Card 406
2.3.4	FANUC PPR 406
2.3.5	Portable Tape Reader 407
2.4	POWER ON/OFF 408
2.4.1	Turning on the Power 408
2.4.2	Display of Software Configuration 409
2.4.3	Power Disconnection 409
3. MANUAL OPERATION	410
3.1	MANUAL REFERENCE POSITION RETURN 411
3.2	JOG FEED 413
3.3	INCREMENTAL FEED 415
3.4	MANUAL HANDLE FEED 416
3.5	MANUAL ABSOLUTE ON AND OFF 418

4. AUTOMATIC OPERATION	423
4.1 MEMORY OPERATION	424
4.2 MDI OPERATION	426
4.3 DNC OPERATION	431
4.4 PROGRAM RESTART	432
4.5 SCHEDULING FUNCTION	436
4.6 SUBPROGRAM CALL FUNCTION	441
4.7 MANUAL HANDLE INTERRUPTION	443
4.8 MIRROR IMAGE	446
4.9 SEQUENCE NUMBER SEARCH	448
5. TEST OPERATION	450
5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK	451
5.2 FEEDRATE OVERRIDE	452
5.3 RAPID TRAVERSE OVERRIDE	453
5.4 DRY RUN	454
5.5 SINGLE BLOCK	455
6. SAFETY FUNCTIONS	457
6.1 EMERGENCY STOP	458
6.2 OVERTRAVEL	459
6.3 STROKE CHECK	460
7. ALARM AND SELF-DIAGNOSIS FUNCTIONS	464
7.1 ALARM DISPLAY	465
7.2 CHECKING BY SELF-DIAGNOSTIC SCREEN	467
8. DATA INPUT/OUTPUT	469
8.1 FILES	470
8.2 FILE SEARCH	472
8.3 FILE DELETION	473
8.4 PROGRAM INPUT/OUTPUT	474
8.4.1 Inputting a Program	474
8.4.2 Outputting a Program	476
8.5 OFFSET DATA INPUT AND OUTPUT	479
8.5.1 Inputting Offset Data	479
8.5.2 Outputting Offset Data	480
8.6 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA	481
8.6.1 Inputting Parameters	481
8.6.2 Outputting Parameters	482
8.7 INPUTTING/OUTPUTTING CUSTOM MACRO B COMMON VARIABLES	483
8.7.1 Inputting Custom Macro B Common Variables	483
8.7.2 Outputting Custom Macro B Common Variable	484

8.8	DISPLAYING DIRECTORY OF FLOPPY DISK	485
8.8.1	Displaying the Directory	486
8.8.2	Reading Files	488
8.8.3	Outputting Programs	489
8.8.4	Deleting Files	490
8.8.5	Changing the File Name	492
9.	EDITING PROGRAMS	493
9.1	INSERTING, ALTERING AND DELETING A WORD	494
9.1.1	Word Search	496
9.1.2	Heading a Program	498
9.1.3	Inserting a Word	499
9.1.4	Altering a Word	500
9.1.5	Deleting a Word	501
9.2	DELETING BLOCKS	502
9.2.1	Deleting a Block	502
9.2.2	Deleting Multiple Blocks	503
9.3	PROGRAM NUMBER SEARCH	504
9.4	DELETING PROGRAMS	505
9.4.1	Deleting One Program	505
9.4.2	Deleting All Programs	505
9.4.3	Deleting More Than One Program by Specifying a Range	506
9.5	EXTENDED PART PROGRAM EDITING FUNCTION	507
9.5.1	Copying an Entire Program	508
9.5.2	Copying Part of a Program	509
9.5.3	Moving Part of a Program	510
9.5.4	Merging a Program	511
9.5.5	Supplementary Explanation for Copying, Moving and Merging	512
9.5.6	Replacement of Words and Addresses	513
9.6	EDITING OF CUSTOM MACRO B	515
9.7	BACKGROUND EDITING	516
9.8	REORGANIZING MEMORY	518
10.	CREATING PROGRAMS	519
10.1	CREATING PROGRAMS USING THE MDI PANEL	520
10.2	AUTOMATIC INSERTION OF SEQUENCE NUMBERS	521
10.3	CREATING PROGRAMS IN TEACH IN MODE	523
10.4	MENU PROGRAMMING	526
10.5	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION	528
11.	SETTING AND DISPLAYING DATA	532
11.1	SCREENS DISPLAYED BY FUNCTION KEY POS	540
11.1.1	Position Display in the Work Coordinate System	541
11.1.2	Position Display in the Relative Coordinate System	542
11.1.3	Overall Position Display	544
11.1.4	Actual Feedrate Display	545
11.1.5	Display of Run Time and Parts Count	546
11.1.6	Operating Monitor Display	547

11.2	SCREENS DISPLAYED BY FUNCTION KEY  (IN AUTO MODE OR MDI MODE)	548
11.2.1	Program Contents Display	549
11.2.2	Current Block Display Screen	550
11.2.3	Next Block Display Screen	551
11.2.4	Program Check Screen	552
11.2.5	Program Screen for MDI Operation	554
11.3	SCREENS DISPLAYED BY FUNCTION KEY  (IN THE EDIT MODE)	556
11.3.1	Displaying Memory Used and a List of Programs	557
11.4	SCREENS DISPLAYED BY FUNCTION KEY 	559
11.4.1	Setting and Displaying the Tool Offset Value	560
11.4.2	Tool Length Measurement	562
11.4.3	Displaying and Setting the Workpiece Origin Offset Value	564
11.4.4	Displaying and Setting Custom Macro Common Variables	566
11.4.5	Displaying Pattern Data and Pattern Menu	567
11.4.6	Displaying and Setting Tool Life Management Data	569
11.5	SCREENS DISPLAYED BY FUNCTION KEY 	571
11.5.1	Displaying and Setting Parameters	572
11.5.2	Displaying and Setting Pitch Error Compensation Data	574
11.5.3	Displaying and Entering Setting Data	579
11.5.4	Sequence Number Comparison and Stop	581
11.5.5	Displaying and Setting Run Time, Parts Count, and Time	583
11.6	SCREENS DISPLAYED BY FUNCTION KEY 	586
11.6.1	Displaying Operator Message	586
11.6.2	Displaying and Setting the Software Operator's Panel	587
11.7	DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING	589
11.7.1	Displaying the Program Number and Sequence Number	589
11.7.2	Displaying the Status and Warning for Data Setting	590
12.	GRAPHICS FUNCTION	591
12.1	GRAPHICS DISPLAY	592
12.2	DYNAMIC GRAPHIC DISPLAY	598
12.2.1	Path Drawing	598
12.2.2	Solid Graphics	607
13.	DISPLAY AND OPERATION OF 00-MC	620
13.1	DISPLAY	621
13.2	OPERATION	622
 IV. MAINTENANCE		
1.	METHOD OF REPLACING BATTERY	625
1.1	REPLACING CNC BATTERY FOR MEMORY BACK-UP	626
1.2	REPLACING BATTERIES FOR ABSOLUTE PULSE CODER	627

APPENDIX

A. TAPE CODE LIST	631
B. LIST OF FUNCTIONS AND TAPE FORMAT	634
C. RANGE OF COMMAND VALUE	639
D. NOMOGRAPHS	642
D.1 INCORRECT THREADED LENGTH	643
D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH	645
D.3 TOOL PATH AT CORNER	647
D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING	650
E. STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET	651
F. CHARACTER-TO CODES CORRESPONDENCE TABLE	653
G. ALARM LIST	654
H. OPERATION OF PORTABLE TAPE READER	673
I. Series 0-D SPECIFICATIONS	677
J. CORRESPONDENCE BETWEEN ENGLISH KEY AND SYMBOLIC KEY	685

I. GENERAL

1 GENERAL

About this manual

This manual consists of the following parts:

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, characteristics, and restrictions. When a program is created through conversational automatic programming function, refer to the manual for the conversational automatic programming function (Table 1).

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 procedure for replacing batteries.

APPENDIX

Lists tape codes, valid data ranges, and error codes.

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to the manual for parameters (B-61410E).

This manual describes all optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

The models covered by this manual, and their abbreviations are:

Product name	Abbreviations		
FANUC Series 0-MC	0-MC	Series 0	
FANUC Series 0-MF	0-MF		
FANUC Series 0-GSC	0-GSC		
FANUC Series 00-MC	00-MC	Series 00	
FANUC Series 0-Mate MC	0-Mate MC	Series 0-Mate	
FANUC Series 0-Mate MF	0-Mate MF		
FANUC Series 0-MD	0-MD	Series 0-D	Series 0-D
FANUC Series 0-GSD	0-GSD		
FANUC Series 0-MD II	0-MD II	Series 0-D II	
FANUC Series 0-GSD II	0-GSD II		

Special symbols

This manual uses the following symbols:

IP_: Indicates a combination of axes such as X__ Y__ Z (used in PROGRAMMING.).

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

Related manuals

The table below lists manuals related to the FANUC Series 0/00/0-Mate. In the table, this manual is marked with an asterisk (*).

- **Series 0/00/0-Mate C**

List of related manuals

Manual name	Specification number	
FANUC Series 0/00 DESCRIPTIONS	B-61392E	
FANUC Series 0/00/0-Mate DESCRIPTIONS (Supplement for Remote buffer)	B-61392EN-1	
FANUC Series 0/00/0-Mate CONNECTION MANUAL (HARDWARE)	B-61393E	
FANUC Series 0/00/0-Mate CONNECTION MANUAL (FUNCTION)	B-61393E-2	
FANUC Series 0/00/0-Mate FOR LATHE OPERATOR'S MANUAL	B-61394E	
FANUC Series 0/00/0-Mate FOR MACHINING CENTER OPERATOR'S MANUAL	B-61404E	*
FANUC Series 0/00/0-Mate MAINTENANCE MANUAL	B-61395E	
FANUC Series 0/00/0-Mate OPERATION AND MAINTENANCE HANDBOOK	B-61397E	
FANUC Series 0/00/0-Mate FOR LATHE PARAMETER MANUAL	B-61400E	
FANUC Series 0/00/0-Mate FOR MACHINING CENTER PARAMETER MANUAL	B-61410E	
GRAPHIC CONVERSATION FOR MACHINING CENTER (Series 0-MC, Series 0-MF, Series 0-Mate MF) OPERATOR'S MANUAL	B-61434E	
FANUC PMC-MODEL K/L/M PROGRAMMING MANUAL (LADDER LANGUAGE)	B-55193E	
FANUC Series 0/0-Mate PROGRAMMING MANUAL (Macro Compiler / Macro Executer)	B-61393E-1	

- **Series 0-D**

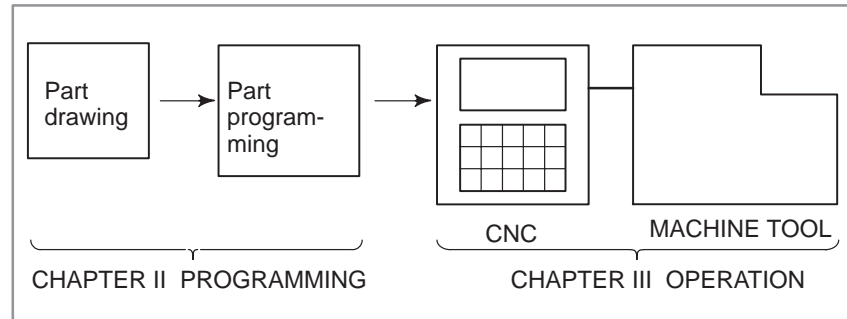
List of related manuals

Manual name	Specification number	
FANUC Series 0-TD/MD DESCRIPTIONS	B-62542EN	
FANUC Series 0-TD/MD/PD/GCD/GSD CONNECTION MANUAL (HARDWARE)	B-62543EN	
FANUC Series 0-TD/MD/GCD/GSD CONNECTION MANUAL (FUNCTION)	B-62543EN-1	
FANUC Series 0-PD CONNECTION MANUAL (FUNCTION)	B-62973EN	
FANUC Series 0/00/0-Mate FOR LATHE OPERATOR'S MANUAL	B-61394E	
FANUC Series 0/00/0-Mate FOR MACHINING CENTER OPERATOR'S MANUAL	B-61404E	*
FANUC Series 0-PD OPERATOR'S MANUAL	B-62974EN	
FANUC Series 0/00/0-Mate MAINTENANCE MANUAL	B-61395E	
FANUC Series 0-PD MAINTENANCE MANUAL	B-62975EN	
FANUC Series 0-TD/GCD PARAMETER MANUAL	B-62550EN	
FANUC Series 0-MD/GSD PARAMETER MANUAL	B-62580EN	

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL

When machining the part using the CNC machine tool, first prepare the program, then operate the CNC machine by using the program.

- 1) First, prepare the program from a part drawing to operate the CNC machine tool. Store the program to a media appropriate for the CNC. How to prepare the program is described in the Chapter II. PROGRAMMING.
- 2) The program is to be read into the CNC system. Then, mount the workpieces and tools on the machine, and operate the tools according to the programming. Finally, execute the machining actually. How to operate the CNC system is described in the Chapter III. OPERATION.



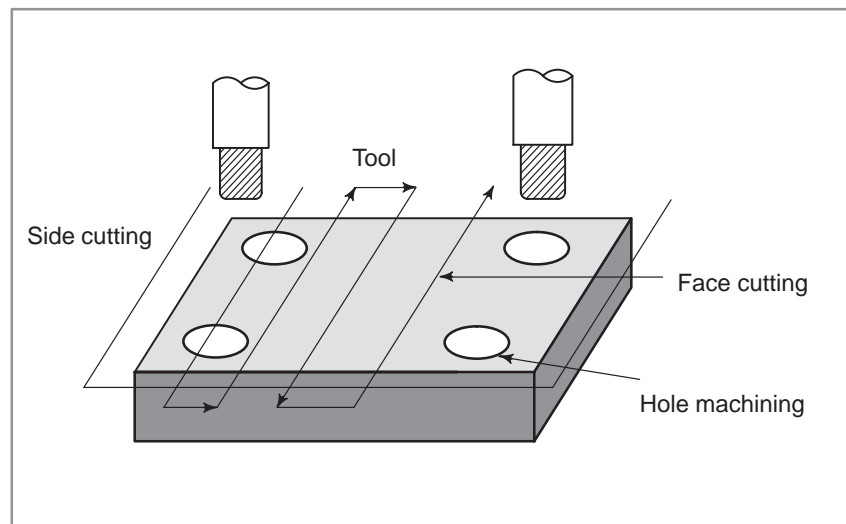
Before the actual programming, make the machining plan for how to machine the part.

Machining plan

1. Determination of workpieces machining range
2. Method of mounting workpieces on the machine tool
3. Machining sequence in every cutting process
4. Cutting tools and cutting conditions

Decide the cutting method in every cutting process.

Cutting process	1	2	3
Cutting procedure	Feed cutting	Side cutting	Hole machining
1. Cutting method: Rough Semi Finish			
2. Cutting tools			
3. Cutting conditions: Feedrate Cutting depth			
4. Tool path			



Prepare the program of the tool path and machining condition according to the workpiece figure, for each machining.

1.2

NOTES ON READING THIS MANUAL

NOTE

- 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 Headings are placed in the left margin so that the reader can easily access necessary information. When locating the necessary information, the reader can save time by searching through these headings.
- 3 Machining programs, parameters, variables, 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.
- 4 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.
- 5 This manual describes the functions supported by the software of the following series and editions of FANUC Series 0/00/0-Mate. These functions may not be supported by other series or editions of the software.

Product name	Series	Editions
FANUC Series 0-MC	0469	01 or more
FANUC Series 0-MF	0467	14 or more
FANUC Series 0-GSC	0866	09 or more
FANUC Series 00-MC	0468	05 or more
FANUC Series 0-Mate MC	0455	06 or more
FANUC Series 0-Mate MF	0454	03 or more
FANUC Series 0-MD	0472	01 or more
FANUC Series 0-GSD	0891	01 or more
FANUC Series 0-MD II	0473	01 or more
FANUC Series 0-GSD II	0892	01 or more

For the functions supported by the Series 0-D, see "Specifications" in Appendix I.

II. PROGRAMMING

1

GENERAL



1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE– INTERPOLATION

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

Explanations

- Tool movement along a straight line

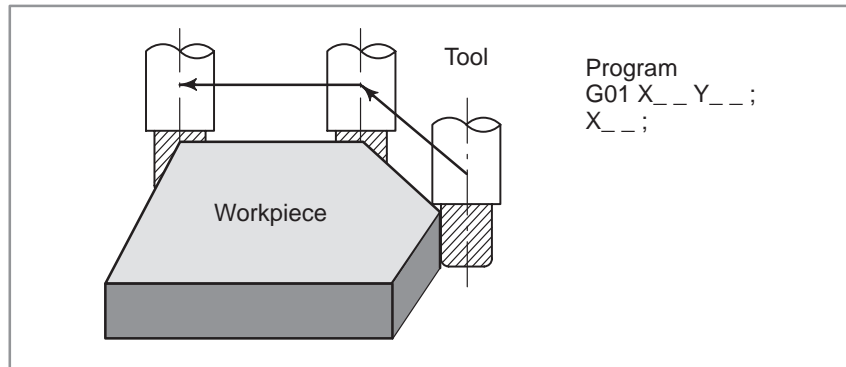


Fig.1.1 (a) Tool movement along a straight line

- Tool movement along an arc

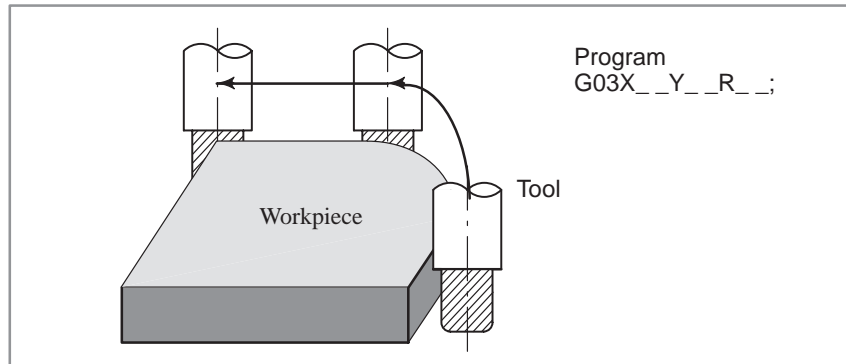


Fig.1.1 (b) Tool movement along an arc

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

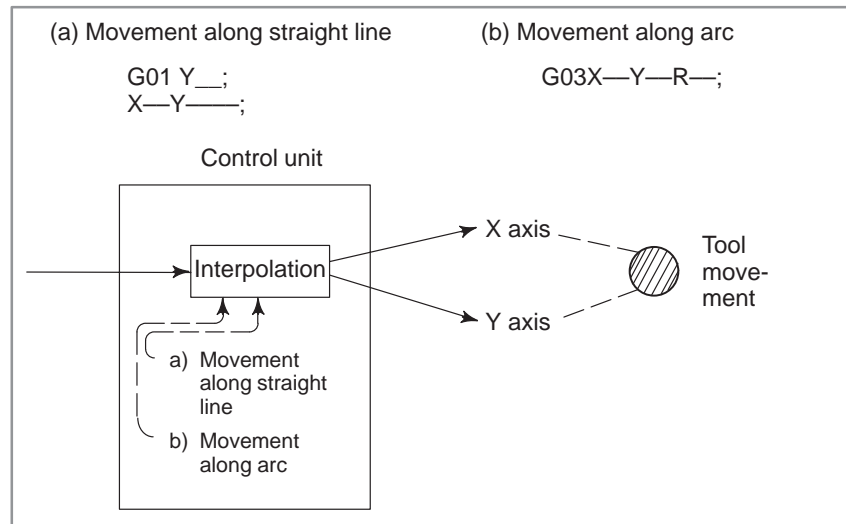


Fig.1.1 (c) Interpolation function

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 cutting a workpiece is called the feed.

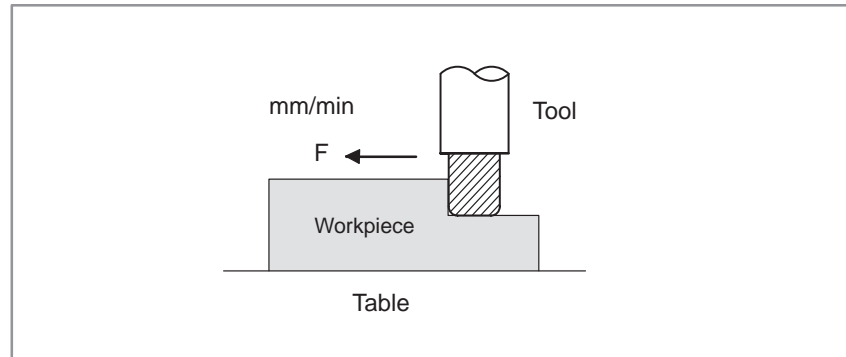


Fig.1.2 Feed function

Feedrates can be specified by using actual numerics. For example, to feed the tool at a rate of 150 mm/min, specify the following in the program:
 F150.0

The function of deciding the feed rate is called the feed function (See II-5).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine-Specific Position)

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

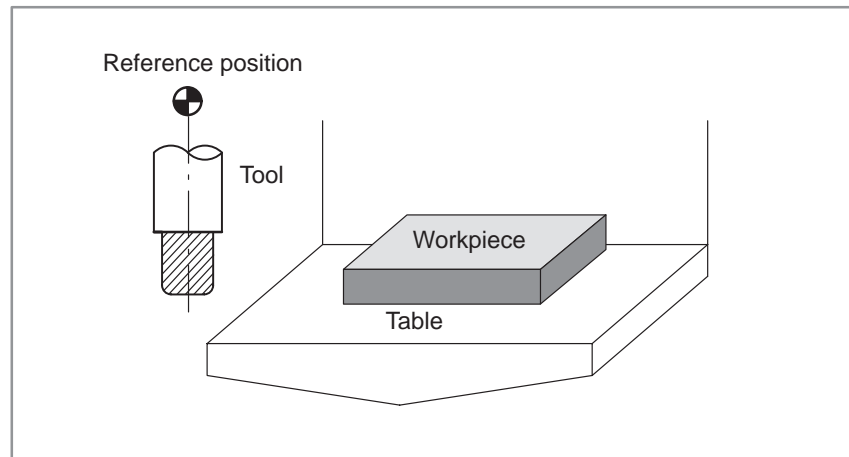


Fig.1.3.1 Reference position

Explanations

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

- (1) Manual reference position return (See III-3.1)

Reference position return is performed by manual button operation.

- (2) Automatic reference position return (See II-6)

In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

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

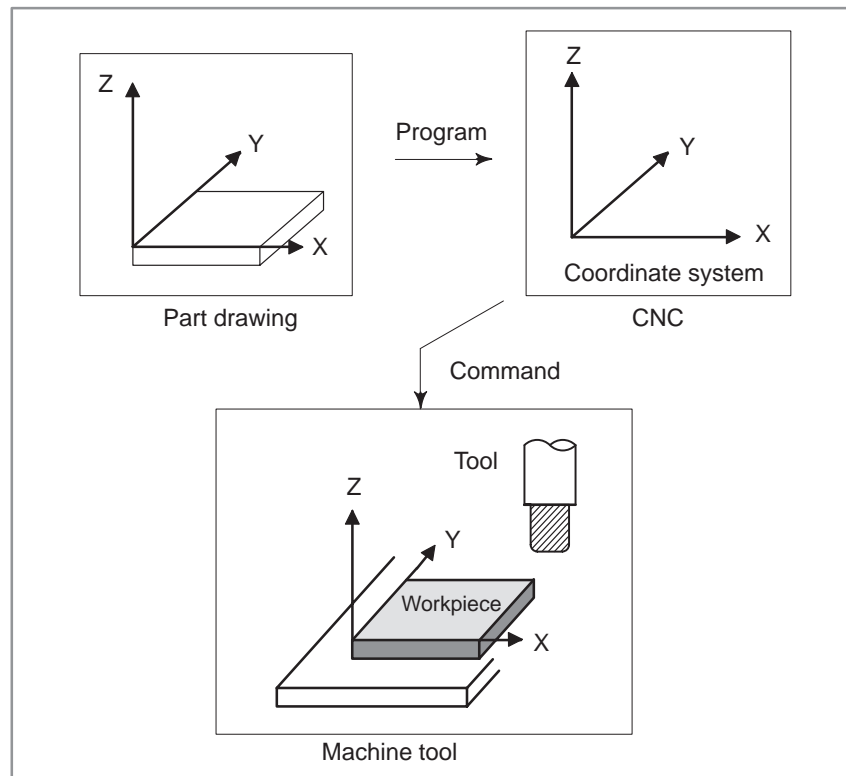


Fig.1.3.2 (a) Coordinate system

Explanations

- **Coordinate system**

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

(1) Coordinate system on part drawing

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

(2) Coordinate system specified by the CNC

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

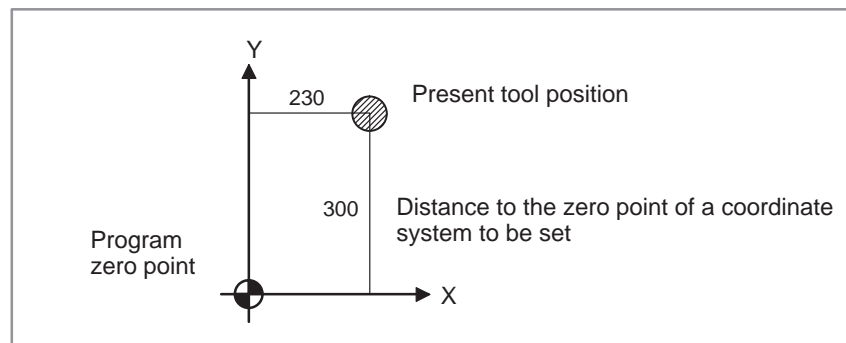


Fig.1.3.2 (b) Coordinate system specified by the CNC

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

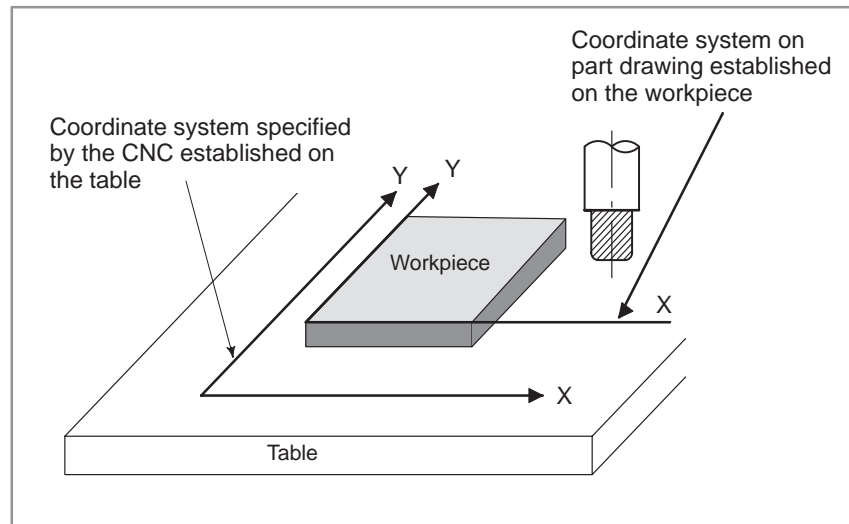


Fig.1.3.2 (c) Coordinate system specified by CNC and coordinate system on part drawing

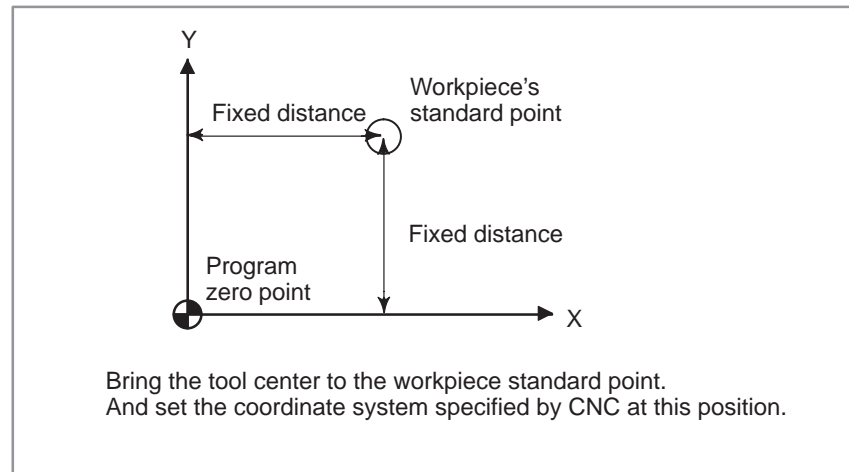
The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.

Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

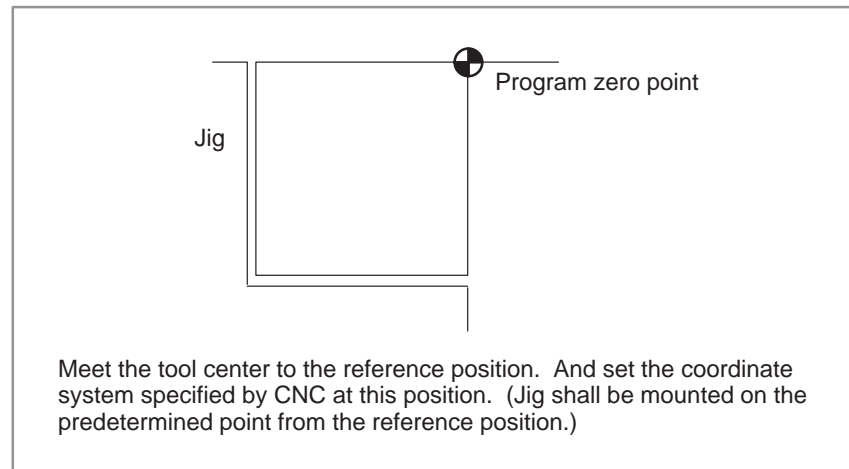
- **Methods of setting the two coordinate systems in the same position**

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

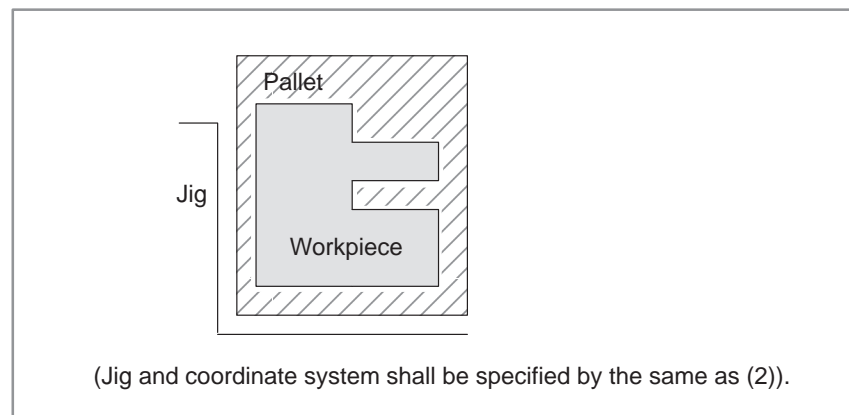
(1) Using a standard plane and point of the workpiece.



(2) Mounting a workpiece directly against the jig



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



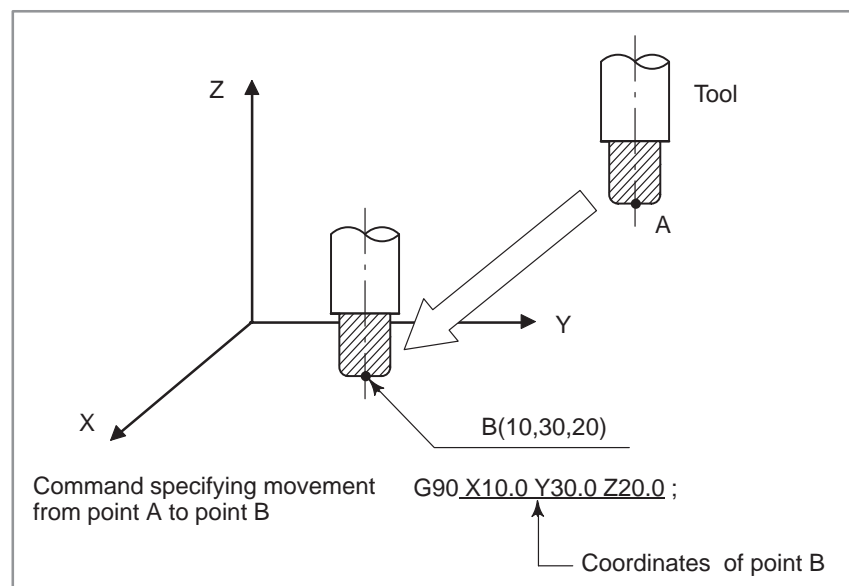
1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands

Explanations

- **Absolute coordinates**

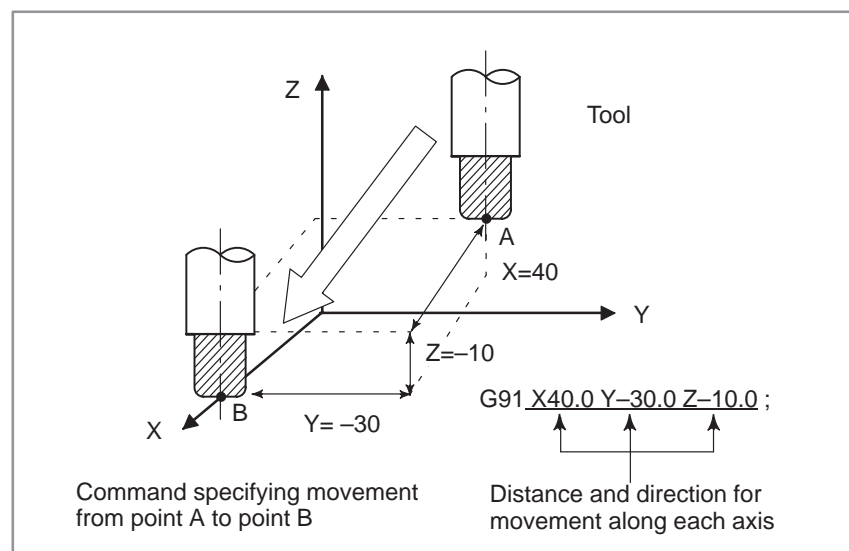
Coordinate values of command for moving the tool can be indicated by absolute or incremental designation (See II-8.1).

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 coordinates**

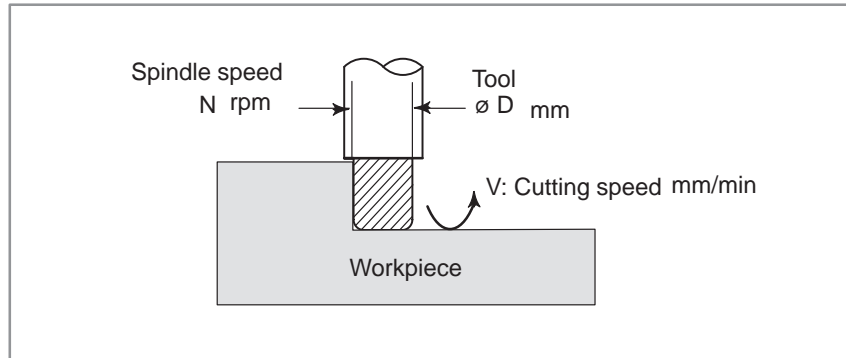
Specify the distance from the previous tool position to the next tool position.



1.4 CUTTING SPEED – SPINDLE SPEED FUNCTION

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed.

As for the CNC, the cutting speed can be specified by the spindle speed in rpm unit.



Examples

<When a workpiece should be machined with a tool 100 mm in diameter at a cutting speed of 80 mm/min. >

The spindle speed is approximately 250 rpm, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

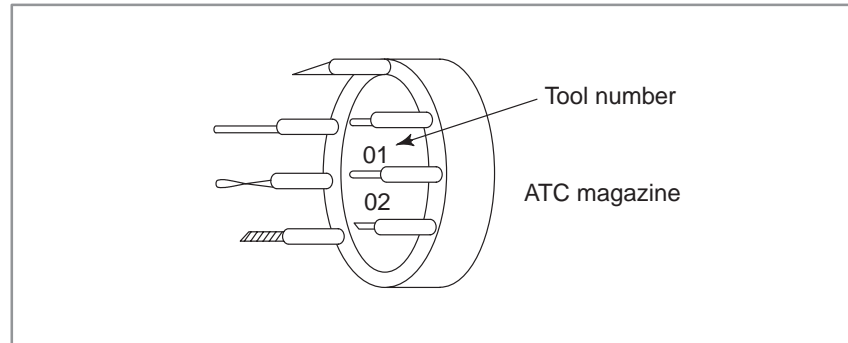
S250 ;

Commands related to the spindle speed are called the spindle speed function

(See II-9).

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION

When drilling, tapping, boring, milling 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.



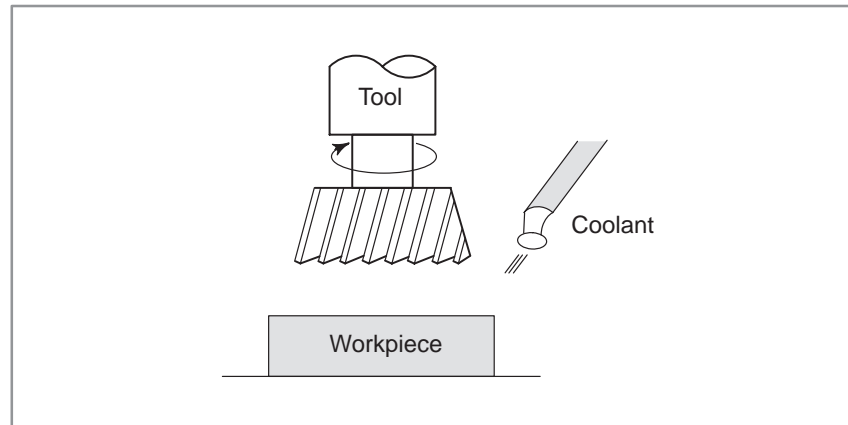
Examples

<When No.01 is assigned to drilling tool>

When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function (See II-10).

1.6 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION

When machining is actually started, it is necessary to rotate the spindle, and feed coolant. For this purpose, on-off operations of spindle motor and coolant valve should be controlled (See II-11).



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 an M code.

For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

1.7 PROGRAM CONFIGURATION

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

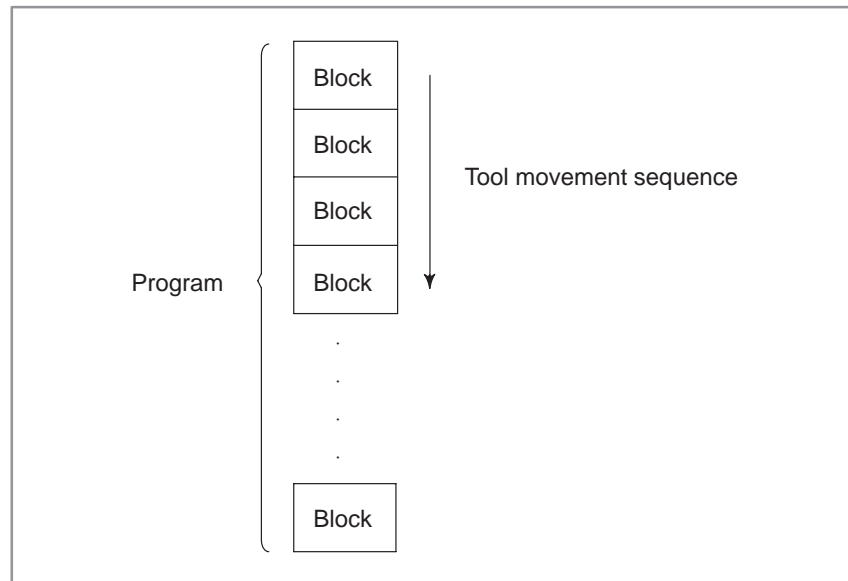


Fig.1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See II-12).

Explanations

The block and the program have the following configurations.

• Block

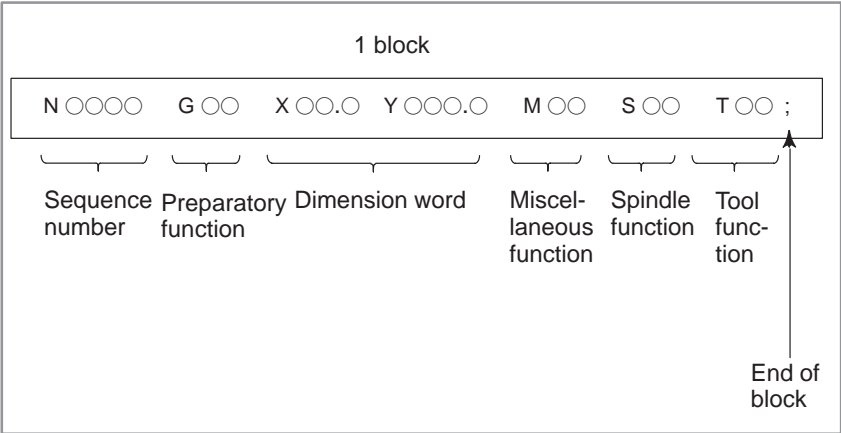


Fig.1.7 (b) Block configuration

A block has a sequence number at its head, which identifies the block, and an end-of-block code at the end, indicating the end of the block. This manual indicates the end-of-block code by ; (LF in the ISO code and CR in the EIA code).

• Program

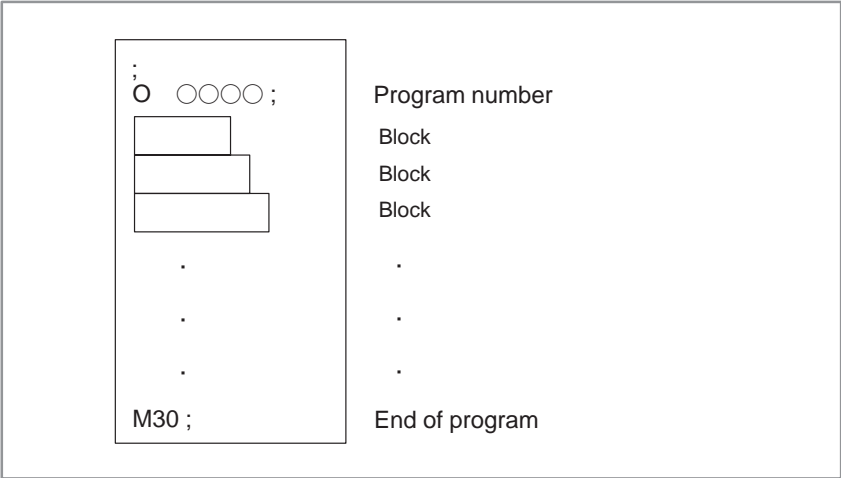
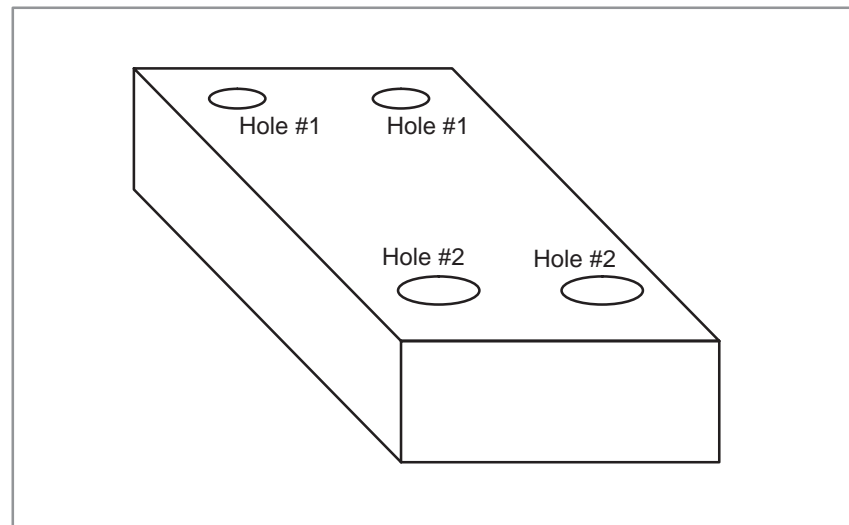
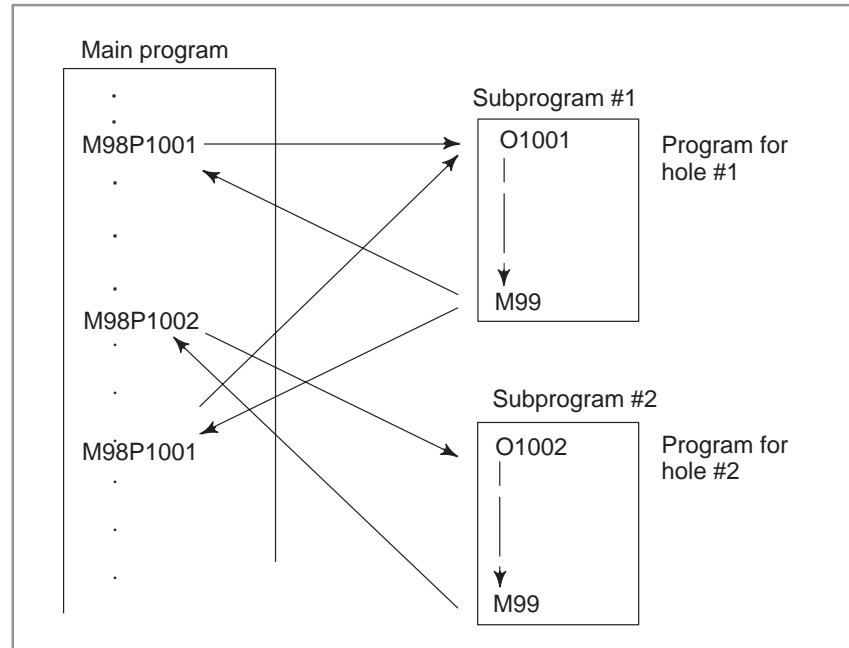


Fig.1.7 (c) Program configuration

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

- **Main program and subprogram**

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



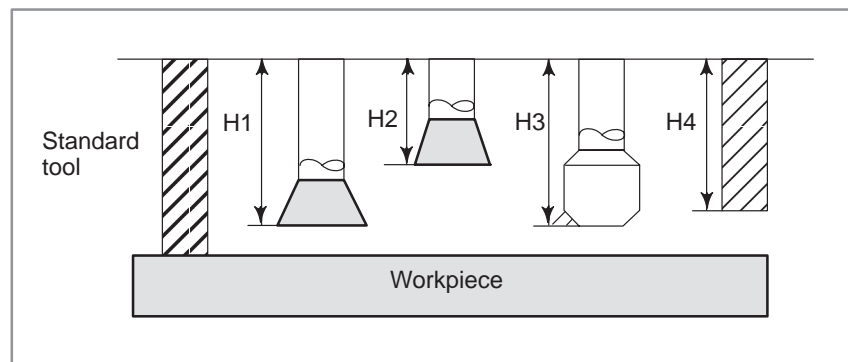
1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM

Explanations

- **Machining using the end of cutter – Tool length compensation function (See II-14.1)**

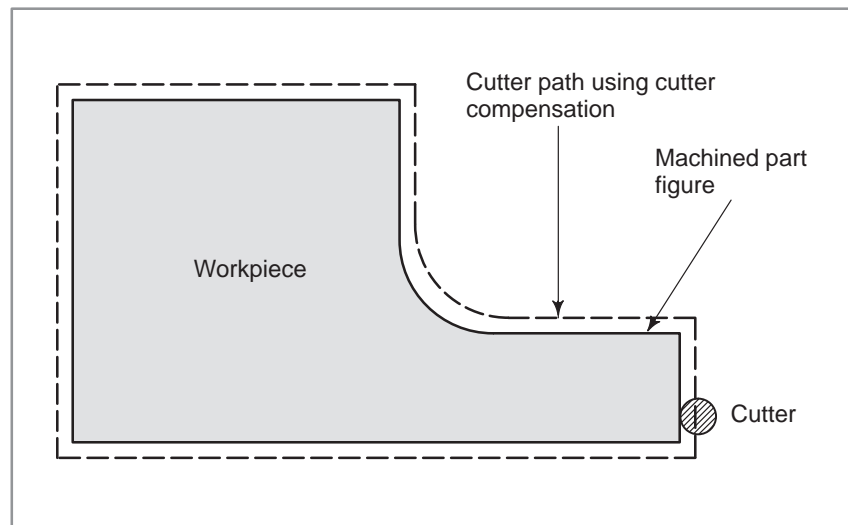
Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.

Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (data display and setting : see III-11), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation.



- **Machining using the side of cutter – Cutter compensation function (See II-14.4 to 14.6)**

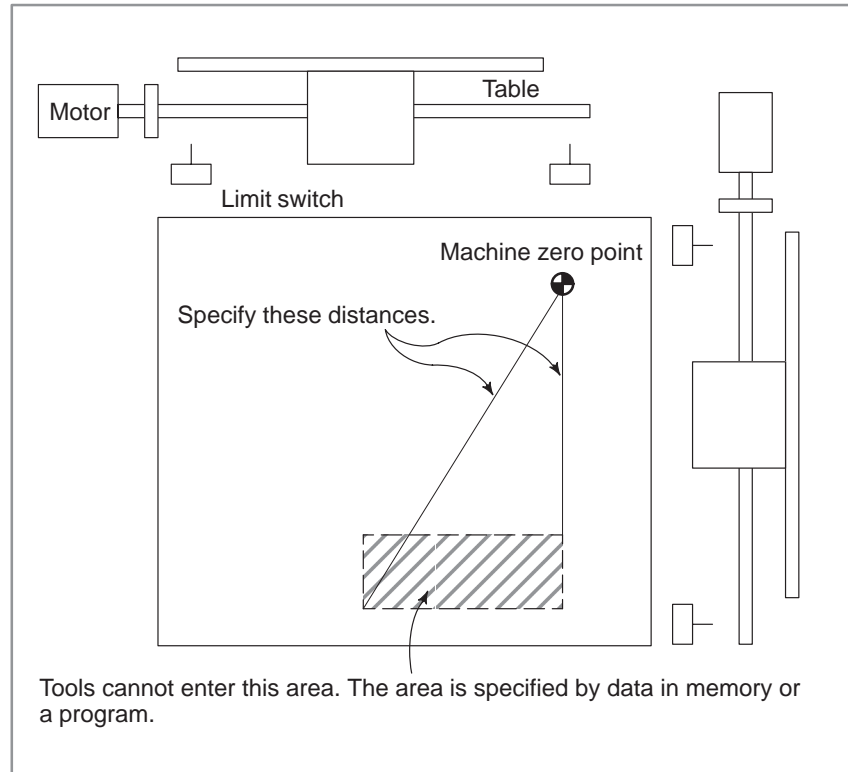
Because a cutter has a radius, the center of the cutter path goes around the workpiece with the cutter radius deviated.



If radius of cutters are stored in the CNC (Data Display and Setting : see III-11), the tool can be moved by cutter radius apart from the machining part figure. This function is called cutter compensation.

1.9 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 (see Section III-11). This function is called stroke check.

2

CONTROLLED AXES



2.1 CONTROLLED AXES

	Series 0/00–C	Series 0–Mate C	Series 0–D
No. of basic controlled axes	3 axes	3 axes	3 axes
Controlled axes expansion (PMC axis is not included.)	Up to 4 axes	Up to 4 axes	Up to 4 axes
Basic simultaneously controlled axes	2 axes	2 axes	2 axes
Simultaneously controlled axes expansion	Up to 4 axes	Up to 3 axes	Up to 4 axes
Axes controlled by PMC	Up to 2 axes (Note)	—	—

NOTE

The number of PMC-controlled axes. The PMC can control up to four axes, including PMC-controlled axes and those axes that are usually controlled by the CNC.

2.2 NAME OF AXES

Names of the three basic axes are fixed as X, Y, and Z. Names of additional axes can be optionally selected from A, B, C, U, V, and W. They can be set by parameter No. 008 #2, #3, #4.

2.3 INCREMENT SYSTEM

Name of increment system	Least input increment	Least command increment	Maximum stroke
IS-B	0.001mm 0.0001inch 0.001deg	0.001mm 0.0001inch 0.001deg	99999.999mm 9999.9999inch 99999.999deg

Name of increment system	Least input increment	Least command increment	Maximum stroke
IS-C	0.0001mm 0.00001inch 0.00001deg	0.0001mm 0.00001inch 0.00001deg	9999.9999mm 999.99999inch 9999.9999deg

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

Maximum stroke = Least command increment × 99999999
See 2.3 Increment System.

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.

Explanations

G01 and G00 are modal G codes in group 01.

```

G01X— ;
      Z— ; } G01 is effective in this range.
      X— ; }
G00Z— ;

```

Explanations


- Modal G codes have the following initial conditions when the power is turned on or the system is reset to the clear state (bit 6 of parameter No. 045).
 - Those G codes marked  in Table 3 are specified automatically.
 - G20 and G21 retain their original conditions.
 - When the power is turned on, G22 is specified automatically. When the system is reset, G22 and G23 retain their original conditions.
 - G00 or G01 is automatically selected depending on the setting of bit 6 of parameter No. 011.
 - G90 or G91 is automatically selected depending on the setting of bit 7 of parameter No. 030.
 - G17, G18, or G19 is automatically selected depending on the setting of parameter No. 212.
- The G codes of group 00, except G10 and G11, are one-shot G codes.
- If a G code that does not appear in the G code list is specified, or a G code whose options are not supported is specified, alarm No. 010 is displayed.
- Multiple G codes of different groups can be specified in a single block. When multiple G codes of one group are specified in a block, the G code specified last is effective.
- If any G code of group 01 is specified in a canned cycle mode, the canned cycle is automatically cancelled and the G80 condition is entered. However, a G code of group 01 is not affected by any of the canned cycle G codes.
- A G code is displayed from each group.

Table 3 G code list (1/3)

G code	Group	Function	
G00	01	Positioning	
G01		Linear interpolation	
G02		Circular interpolation/Helical interpolation CW	
G03		Circular interpolation/Helical interpolation CCW	
G04	00	Dwell, Exact stop	
G05		High speed cycle machining	
G08		Advanced preview control	
G09		Exact stop	
G10		Data setting	
G11		Data setting mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G27	00	Reference position return check	
G28		Return to reference position	
G29		Return from reference position	
G30		2nd, 3rd and 4th reference position return	
G31		Skip function	
G33	01	Thread cutting	
G37	00	Automatic tool length measurement	
G39		Corner offset circular interpolation	
G40	07	Cutter compensation cancel	
G41		Cutter compensation left	
G42		Cutter compensation right	
G43	08	Tool length compensation + direction	
G44		Tool length compensation – direction	
G45	00	Tool offset increase	
G46		Tool offset decrease	
G47		Tool offset double increase	
G48		Tool offset double decrease	
G49	08	Tool length compensation cancel	
G50	11	Scaling cancel	
G51		Scaling	

Table 3 G code list (2/3)

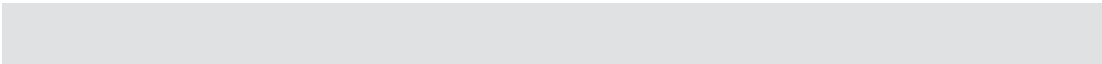
G code	Group	Function
G52	00	Local coordinate system setting
G53		Machine coordinate system selection
G54	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call
G67		Macro modal call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74		Counter tapping cycle
G75	01	Plunge grinding cycle (0-GSC, 0-GSD/II)
G76	09	Fine boring cycle
G77	01	Direct constant-dimension plunge grinding cycle (0-GSC, 0-GSD/II)
G78		Continuous-feed surface grinding cycle (0-GSC, 0-GSD/II)
G79		Intermittent-feed surface grinding cycle (0-GSC, 0-GSD/II)
G80	09	Canned cycle cancel/external operation function cancel
G81		Drilling cycle, spot boring cycle or external operation function
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle

Table 3 G code list (3/3)

G code	Group	Function
G90	03	Absolute command
G91		Increment command
G92	00	Setting for work coordinate system or clamp at maximum spindle speed
G94	05	Feed per minute
G95		Feed per rotation
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle
G107	00	Cylindrical interpolation
G150	19	Normal direction control cancel mode
G151		Normal direction control left side on
G152		Normal direction control right side on
G160	20	In-feed control function cancel (0-GSC, 0-GSD/II)
G161		In-feed control function (0-GSC, 0-GSD/II)

4

INTERPOLATION FUNCTIONS



4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.

In the absolute command, coordinate value of the end point is programmed.

In the incremental command the distance the tool moves is programmed.

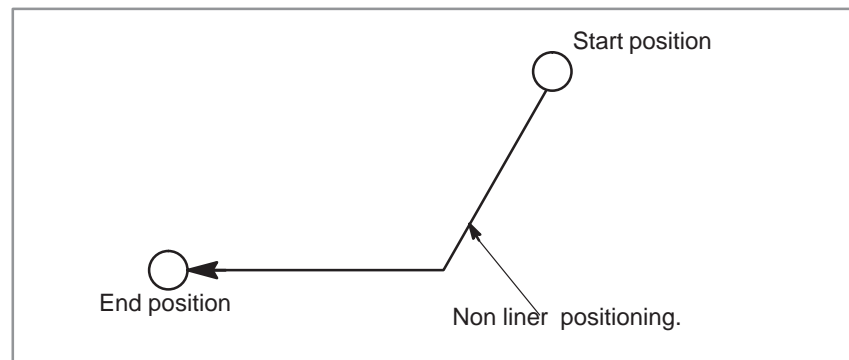
Format

G00 IP _ ;

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

Explanations

Tool path generally does not become a straight line.



The rapid traverse rate in the G00 command is set to the parameter No. 518 to 521 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.

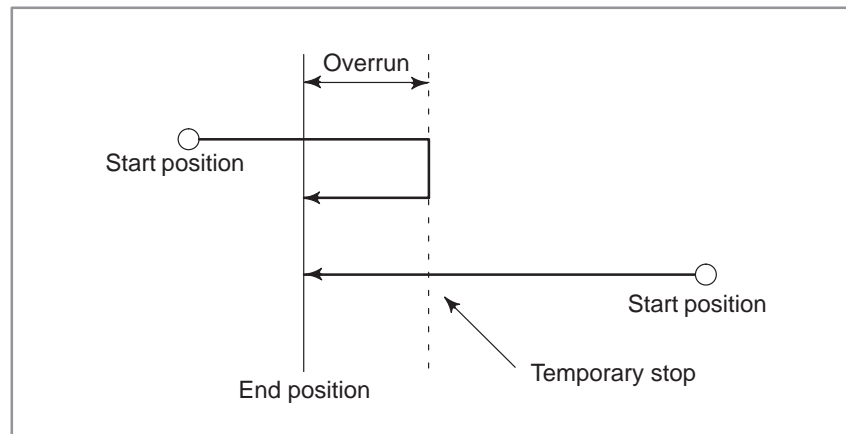
This range is determined by the machine tool builder by setting to parameter No. 500 to 503.

Restrictions

The rapid traverse rate cannot be specified in the address F.

4.2 SINGLE DIRECTION POSITIONING (G60)

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



Format

G60 IP _ ;

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

Explanations

An overrun and a positioning direction are set by the parameter (No. 029 #0 to #3, No. 204 to 207). Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.

Restrictions

- During drilling canned cycle, no single direction positioning is effected in Z axis.
- No single direction positioning is effected in an axis for which no overrun has been set by the parameter.
- When the move distance 0 is commanded, the single direction positioning is not performed.
- The direction set to the parameter is not effected by mirror image.
- The single direction positioning does not apply to the shift motion in the canned cycles of G76 and G87.

4.3 LINEAR INTERPOLATION (G01)

Tools can move along a line

Format

G01 IP _ F_ ;

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

F_: Speed of tool feed (Feedrate)

Explanations

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 α β γ ζ F_ ;

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

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

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

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

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

The feed rate 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 feed rate is that in which the tangential feed rate 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 1min.

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:

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

The feed rate for the C axis is

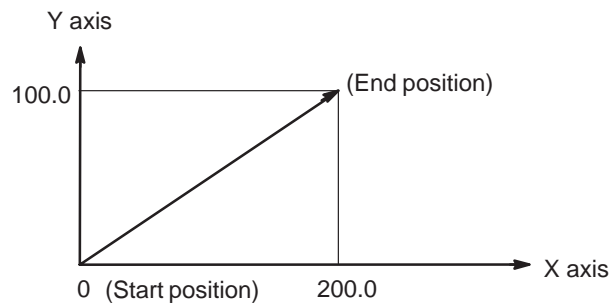
$$\frac{40}{0.14907} \doteq 268.3 \text{ deg/min}$$

In simultaneous 3 or 4 axes control, the feed rate is calculated the same way as in 2 axes control.

Examples

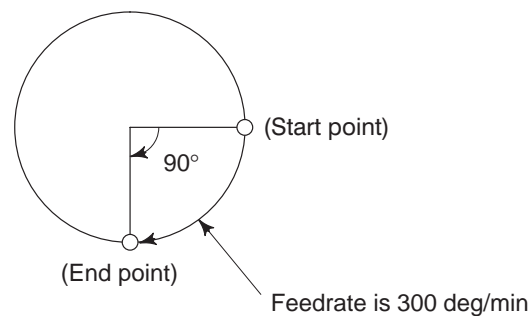
- Linear interpolation

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



- Feedrate for the rotation axis

G91G01C-90.0 G300.0 ;Feed rate of 300deg/min



4.4 CIRCULAR INTERPOLATION (G02, G03)

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

Format

Arc in the XpYp plane

$$G17 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Xp_Yp_ \left\{ \begin{array}{c} I_J_ \\ R_ \end{array} \right\} F_;$$

Arc in the ZpXp plane

$$G18 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Xp_Zp_ \left\{ \begin{array}{c} I_K_ \\ R_ \end{array} \right\} F_;$$

Arc in the YpZp plane

$$G19 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Yp_Zp_ \left\{ \begin{array}{c} J_K_ \\ R_ \end{array} \right\} F_;$$

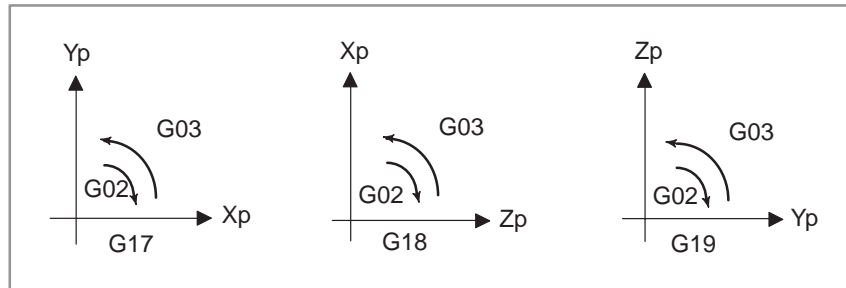
Table 4.4 Description of the command format

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
Yp_	Command values of Y axis or its parallel axis
Zp_	Command values of Z axis or its parallel axis
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 fixed to radius designation.
F_	Feedrate along the arc

Explanations

- **Direction of the circular interpolation**

“Clockwise”(G02) and “counterclockwise”(G03) on the X_pY_p plane (Z_pX_p plane or Y_pZ_p plane) are defined when the X_pY_p plane is viewed in the positive-to-negative direction of the Z_p axis (Y_p axis or X_p axis, respectively) in the Cartesian coordinate system. See the figure below.



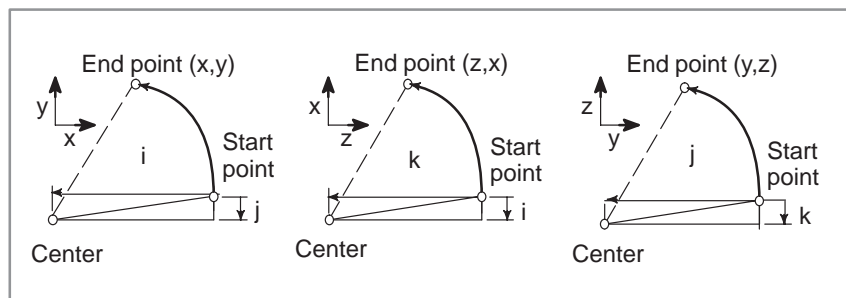
- **Distance moved on an arc**

The end point of an arc is specified by address 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.

- **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.

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



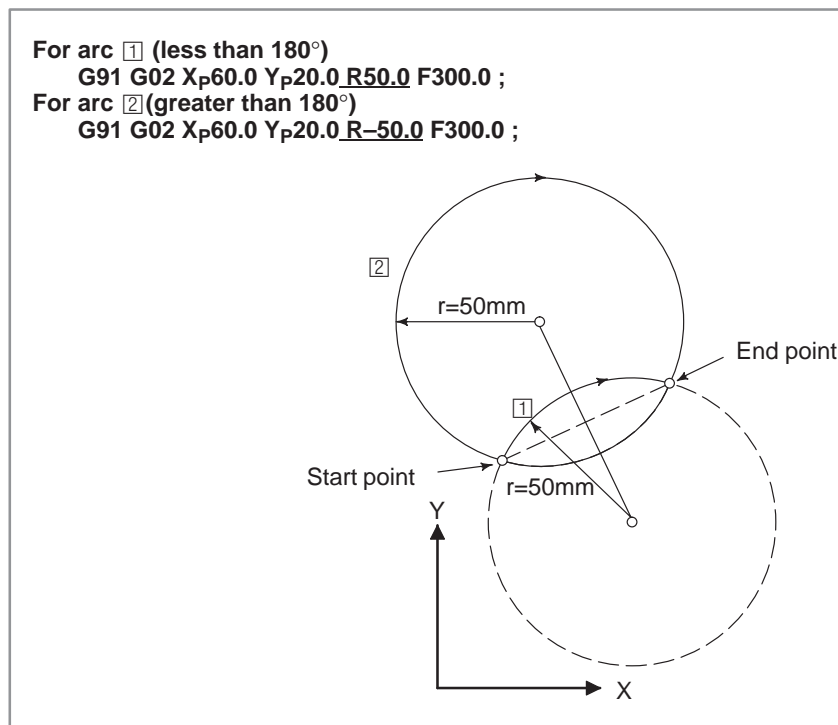
$I0$, $J0$, and $K0$ can be omitted. When X_p , Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.

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.876), an alarm (No.020) occurs. (Valid only when bit 6 of parameter No. 393 is set to 1.)

• 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 Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed G02R ; (The cutter does not move.)



• Feedrate

The feedrate in circular interpolation is equal to the feed rate 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 feed rate is measured along the arc after the cutter compensation is applied.

Restrictions

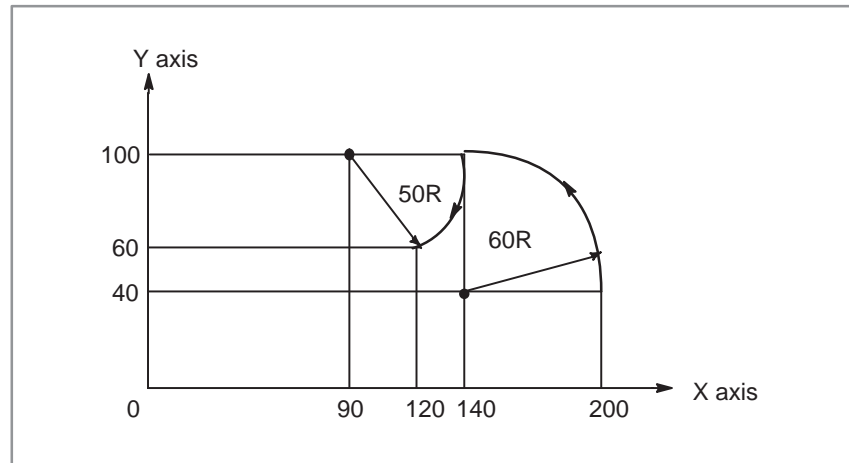
If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

If an axis not comprising the specified plane is commanded, an alarm is displayed.

For example, if axis U is specified as a parallel axis to X axis when plane XY is specified, an alarm (No.028) is displayed.

When an arc having a center angle close to 180° is specified using its radius R, the system may fail to calculate the center of the arc correctly. Therefore, specify the arc with I, J, and K.

Examples



The above tool path can be programmed as follows ;

(1) In absolute programming

```
G92X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0R60.0 F300.;
G02 X120.0 Y60.0R50.0 ;
```

or

```
G92X200.0 Y40.0Z0 ;
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 ;
```

4.5 HELICAL CUTTING (G02, G03) Format

Helical interpolation is possible, by which the tool is moved along a helix, by specifying circular interpolation together with movement along an axis in a plane other than that specified for circular interpolation.

Synchronously with arc of XpYp plane

$$G17 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Xp_Yp_ \left\{ \begin{array}{c} I_J_ \\ R_ \end{array} \right\} \alpha_ F_;$$

Synchronously with arc of ZpXp plane

$$G18 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Xp_Zp_ \left\{ \begin{array}{c} I_K_ \\ R_ \end{array} \right\} \alpha_ F_;$$

Synchronously with arc of YpZp plane

$$G19 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Yp_Zp_ \left\{ \begin{array}{c} J_K_ \\ R_ \end{array} \right\} \alpha_ F_;$$

α : Any one axis where circular interpolation is not applied.

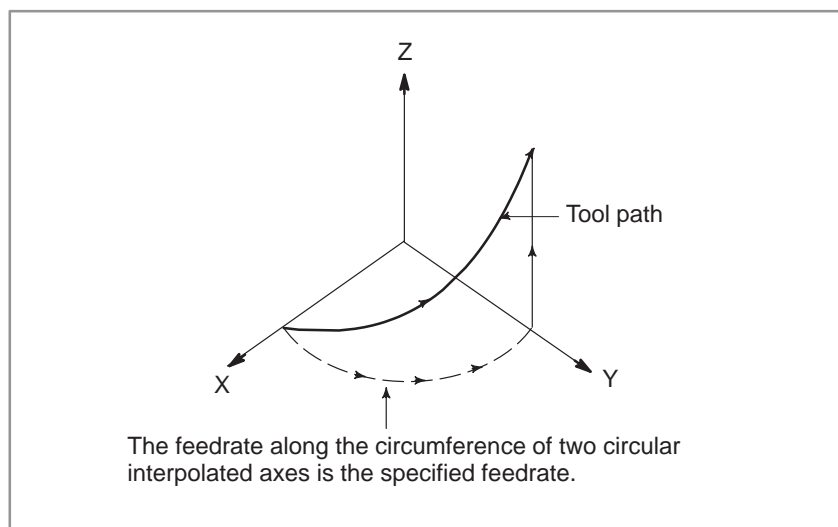
Explanations

The command method is to simply or secondary add one or two move command axes which is not circular interpolation axes. An F command specifies a feed rate along a circular arc. Therefore, the feed rate of the linear axis is as follows:

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

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

Setting bit 4 of parameter No. 0393 clamps the linear axis speed for helical cutting to the setting in parameter No. 527.



Restrictions

Cutter compensation is applied only for a circular arc.

Tool offset and tool length compensation cannot be used in a block in which a helical cutting is commanded.

4.6 CYLINDRICAL INTERPOLATION (G107)

The amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to the amount of travel of the rotary axis.

The cylindrical interpolation function allows the side of a cylinder to be developed for programming. So programs such as a program for cylindrical cam grooving can be created very easily.

Format

**G107 IP r ; Starts the cylindrical interpolation mode
(enables cylindrical interpolation).**
:
:
:

G107 IP 0 ; The cylindrical interpolation mode is cancelled.

IP : An address for the rotation axis
r : The radius of the cylinder

Specify G107 IP r ; and G107 IP 0 ; in separate blocks.

Explanations

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

Use parameter No. 279 to specify whether the rotation axis is the X-, Y-, or Z-axis, or an axis parallel to one of these axes. Specify the G code to select a plane for which the rotation axis is the specified linear axis. For example, when the rotation axis is an axis parallel to the X-axis, G17 must specify an Xp-Yp plane, which is a plane defined by the rotation axis and the Y-axis or an axis parallel to the Y-axis.

- **Feedrate**

A feedrate specified in the cylindrical interpolation mode is a speed on the developed cylindrical surface.

- **Circular interpolation
(G02,G03)**

In the cylindrical interpolation mode, circular interpolation is possible with the rotation axis and another linear axis. Radius R is used in commands in the same way as described in Section 4.4.

The unit for a radius is not degrees but millimeters (for metric input) or inches (for inch input).

<Example Circular interpolation between the Z axis and C axis>

For the parameter No. 279, 5 (axis parallel with the X axis) is to be set.

In this case, the command for circular interpolation is

G18 Z__C__;

G02 (G03) Z__C__R__;

For the parameter No. 279, 6 (axis parallel with the Y axis) may be specified instead. In this case, however, the command for circular interpolation is

G19 C__Z__;

G02 (G03) Z__C__R__;

- **Cutter compensation**

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

- **Cylindrical interpolation accuracy**

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment.

So when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative.

If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

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

MOTION REV : The amount of travel per rotation of the rotation axis
(Setting value of parameter No. 1260)

R : Workpiece radius

$\left[\right]$: Rounded to the least input increment

Restrictions

- **Arc radius command in the cylindrical interpolation mode**

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

- **Circular interpolation and cutter compensation**

If the cylindrical interpolation mode is started when cutter compensation is already applied, circular interpolation is not correctly performed in the cylindrical interpolation mode.

- **Positioning**

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

- **Coordinate system setting**

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

- **Cylindrical interpolation mode setting**

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

- **Tool offset**

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

- **Index table indexing function**

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

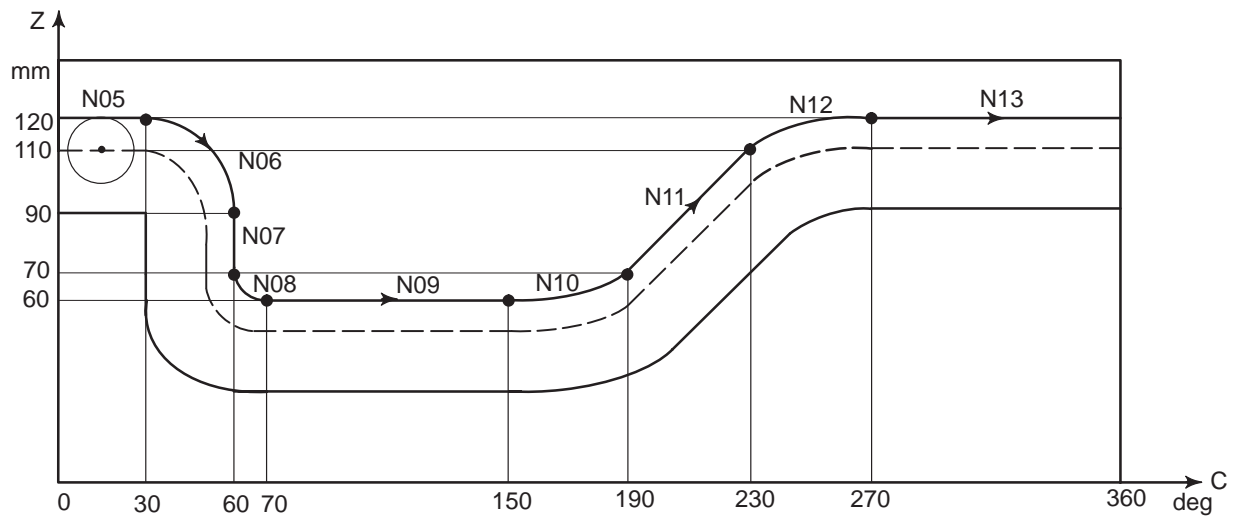
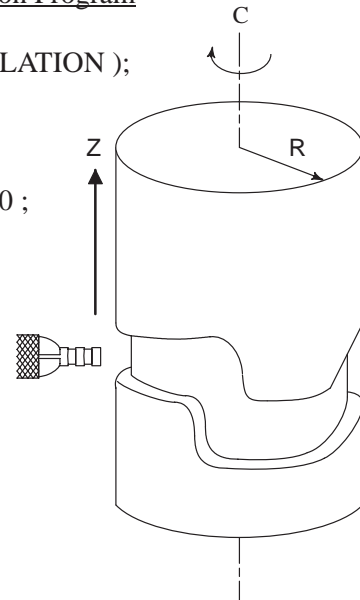
Examples

Example of a Cylindrical Interpolation Program

```

O0001 (CYLINDRICAL INTERPOLATION);
N01 G00 G90 Z100.0 C0;
N02 G01 G91 G18 Z0 C0;
N03 G107 C57299;
N04 G90 G01 G42 Z120.0 D01 F250;
N05 C30.0;
N06 G02 Z90.0 C60.0 R30.0;
N07 G01 Z70.0;
N08 G03 Z60.0 C70.0 R10.0;
N09 G01 C150.0;
N10 G03 Z70.0 C190.0 R75.0;
N11 G01 Z110.0 C230.0;
N12 G02 Z120.0 C270.0 R75.0;
N13 G01 C360.0;
N14 G40 Z100.0;
N15 G107 C0;
N16 M30;

```



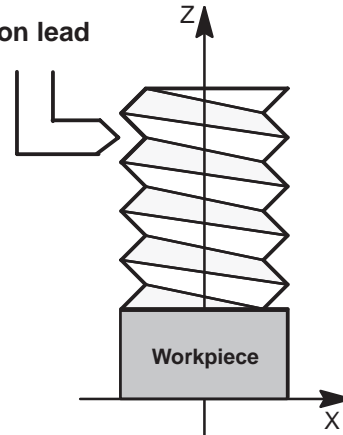
4.7 CONSTANT LEAD THREADING (G33)

Straight threads with a constant lead can be cut. The position coder mounted on the spindle reads the spindle speed in real-time. The read spindle speed is converted to the feedrate per minute to feed the tool.

Format

G33IP _ F_ ;

F : Long axis direction lead



Explanations

In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1-turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.

Table 4.7 lists the ranges for specifying the thread lead.

Table 4.7 Ranges of lead sizes that can be specified

	Increment system	
	IS-B	IS-C
mm input	0.01 to 500.00 mm	0.01 to 500.00 mm
Inch input	0.0001 to 9.9999 inch	0.0001 to 9.9999 inch

NOTE

- 1 The spindle speed is limited as follows :

$$1 \leq \text{spindle speed} \leq \frac{\text{Maximum feedrate}}{\text{Thread lead}}$$

Spindle speed : rpm

Thread lead : mm or inch

Maximum feedrate : mm/min or inch/min ; maximum command-specified feedrate for feed-per-minute mode or maximum feedrate that is determined based on mechanical restrictions including those related to motors, whichever is smaller

- 2 Cutting feedrate override is not applied to the converted feedrate in all machining process from rough cutting to finish cutting. The feedrate is fixed at 100%
- 3 The converted feedrate is limited by the upper feedrate specified.
- 4 Feed hold is disabled during threading. Pressing the feed hold key during threading causes the machine to stop at the end point of the next block after threading (that is, after the G33 mode is terminated)

Examples

Threading at a lead of 1.5mm
G33 Z10. F1.5;

4.8 SKIP FUNCTION (G31)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.

The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format

G31 IP _ ;

G31_: One-shot G code (If is effective only in the block in which it is specified)

Explanations

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5064, as follows:

#5061 X axis coordinate value

#5062 Y axis coordinate value

#5063 Z axis coordinate value

#5064 4th axis coordinate value

WARNING

Disable feedrate override, dry run, and automatic acceleration/deceleration (These operations can be validated by setting bit 3 of parameter No. 015 to 1.) when the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

NOTE

If G31 command is issued while cutter compensation C is applied, an P/S alarm of No.035 is displayed. Cancel the cutter compensation with the G40 command before the G31 command is specified.

Examples

- The next block to G31 is an incremental command

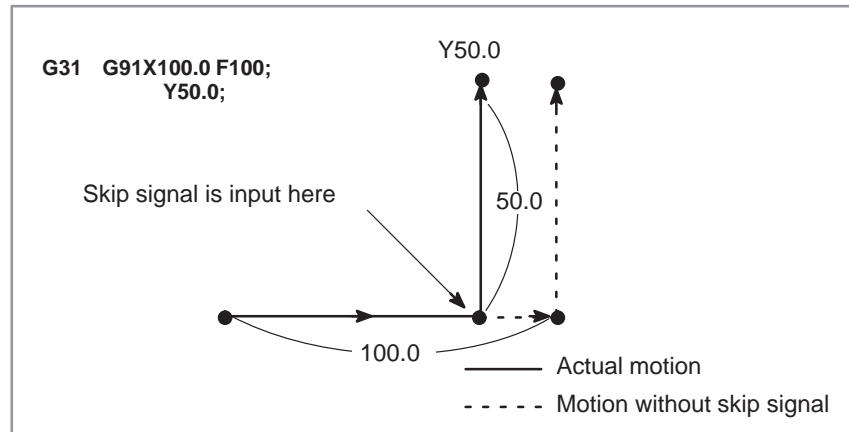


Fig.4.8 (a) The next block is an incremental command

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

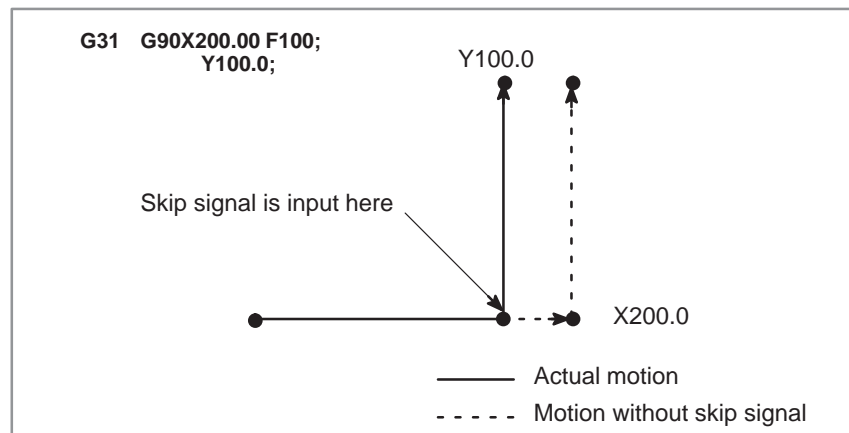


Fig.4.8 (b) The next block is an absolute command for 1 axis

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

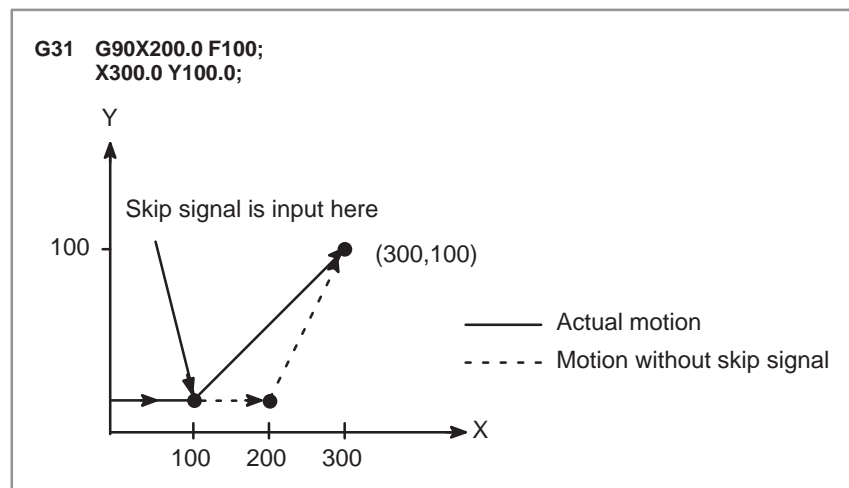


Fig.4.8 (c) The next block is an absolute command for 2 axes

5

FEED FUNCTIONS



5.1 GENERAL

- **Feed functions**

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

1. **Rapid traverse**
When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameter No. 518 to 521).
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)).

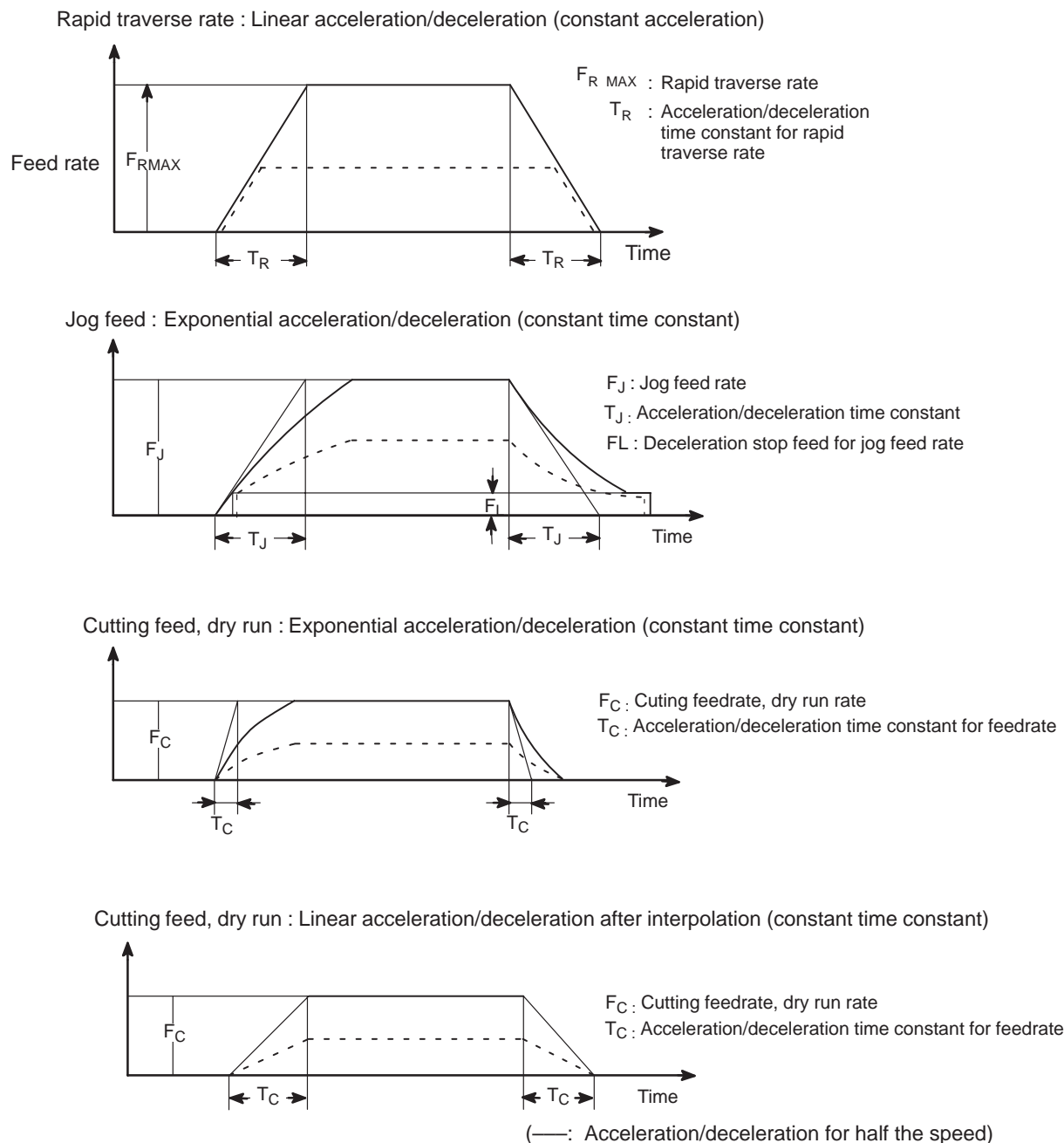


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- **Tool path in a cutting feed**

If the direction of movement changes between specified blocks during cutting feed, a rounded-corner path may result (Fig. 5.1 (b)).

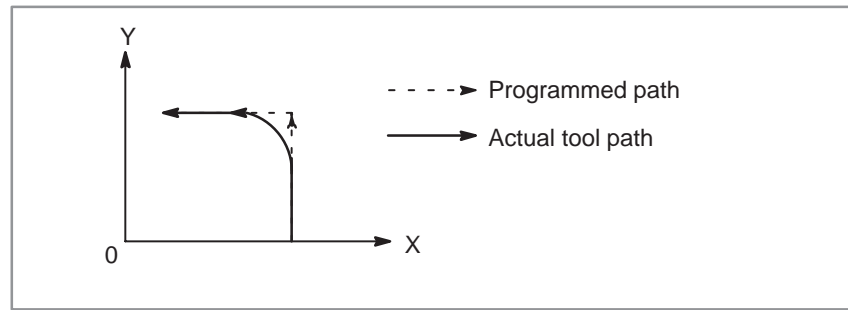


Fig. 5.1 (b) Example of tool path between two blocks

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

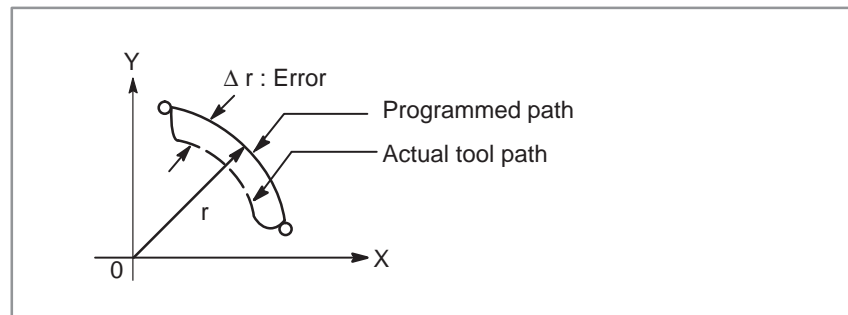


Fig. 5.1 (c) Example of radial error in circular interpolation

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

G00IP _ ;

G00 : G code (group 01) for positioning (rapid traverse)

IP _ ; Dimension word for the end point

Explanations

The positioning command (G00) positions the tool by rapid traverse. In rapid traverse, the next block is executed after the specified feedrate becomes 0 and the servo motor reaches a certain range set by the machine tool builder (in-position check). (In-position check can be disabled for each block by setting bit 5 of parameter No. 020 to 1.)

A rapid traverse rate is set for each axis by parameter No. 518 to 521, so no rapid traverse feedrate need be programmed.

The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel: F0, 25, 50, 100%

F0: Allows a fixed feedrate to be set for each axis by parameter No. 533. For detailed information, refer to the appropriate manual of the machine tool builder.

- Command value range of rapid traverse rate

	Increment system	
	IS-B	IS-C
Metric output	30 to 100,000 mm/min 30 to 100,000 deg/min	30 to 24,000 mm/min 30 to 24,000 deg/min
Inch output	3.0 to 4,000.0 inch/min 3.0 to 100,000 deg/min	3.0 to 960.0 inch/min 3.0 to 24,000 deg/min

5.3 CUTTING FEED

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.

Three modes of specification are available:

1. Feed per minute (G94)
After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G95)
After F, specify the amount of feed of the tool per spindle revolution.
3. F1-digit feed
Specify a desired one-digit number after F. Then, the feedrate set with the CNC for that number is set.

Format

Feed per minute

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

Feed per revolution

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

F1-digit feed

Fn ;
n : Number from 1 to 9

Explanations

- Tangential speed constant control

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

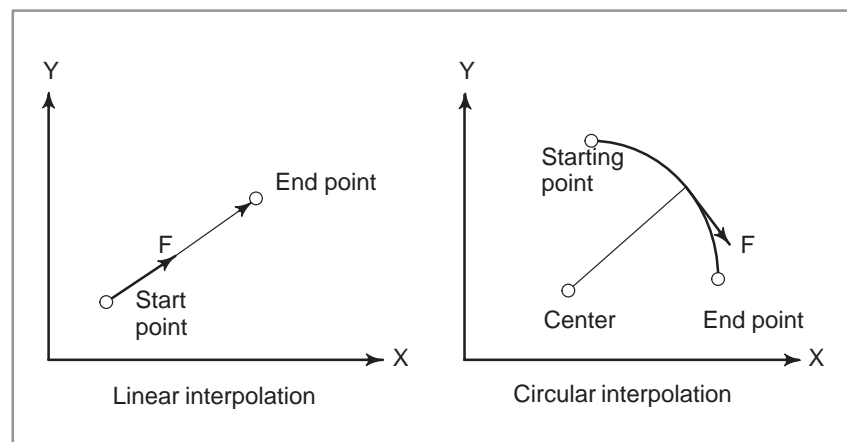


Fig. 5.3 (a) Tangential feedrate (F)

• Feed per minute (G94)

After specifying G94 (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G94 is a modal code. Once a G94 is specified, it is valid until G95 (feed per revolution) is specified. At power-on, the feed per minute mode is set.

An override from 0% to 150% (in 10% 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.

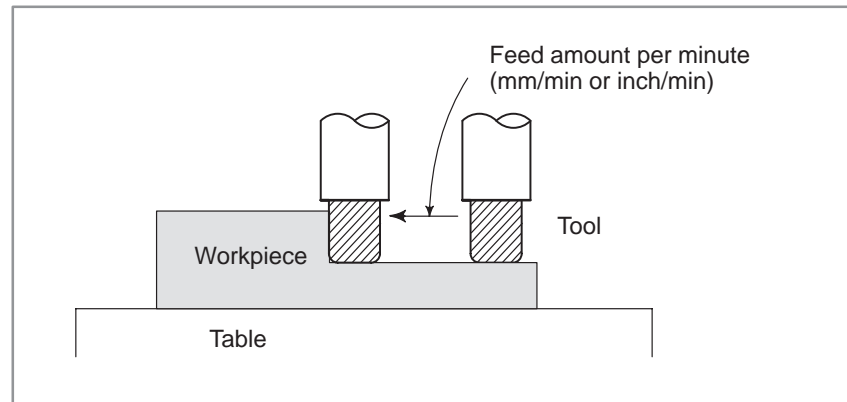


Fig. 5.3 (b) Feed per minute

NOTE

No override can be used for some commands such as for threading.

<Command value range of feed per minute>

	Increment system	
	IS-B	IS-C
Metric input	1 to 100,000 mm/min 1 to 100,000 deg/min	1 to 12,000 mm/min 1 to 12,000 deg/min
Inch input	0.01 to 4,000.0 inch/min 0.01 to 6,000.0 deg/min	0.01 to 480.0 inch/min 0.01 to 600.0 deg/min

<Feedrate command value with a decimal fraction for feed per minute>

When a value with a decimal fraction is specified in a feedrate command for feed per minute, the value is valid within the following range. To specify a value that falls outside this range, specify an integer.

	Increment system	
	IS-B	IS-C
Metric input	0.001 to 99,999.999 mm/min 0.001 to 99,999.999 deg/min	0.001 to 12,000.000 mm/min 0.001 to 12,000.000 deg/min
Inch input	0.00001 to 999.99999 inch/min 0.00001 to 999.99999 deg/min	0.00001 to 480.00000 inch/min 0.00001 to 600.00000 deg/min

- **Feed per revolution (G95)**

After specifying G95 (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G95 is a modal code. Once a G95 is specified, it is valid until G94 (feed per minute) is specified.

An override from 0% to 150% (in 10% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

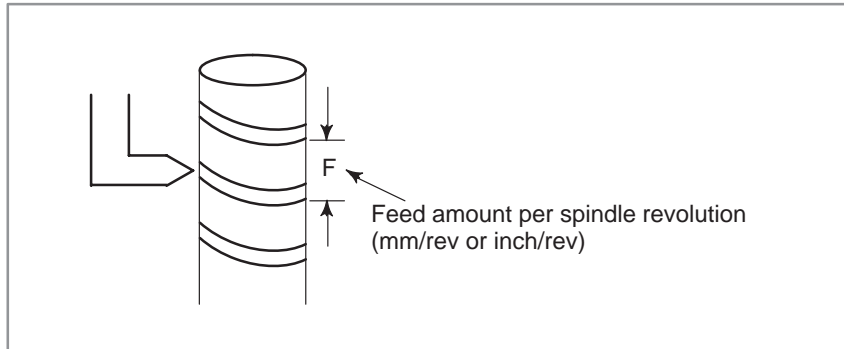


Fig. 5.3 (c) Feed per revolution

<Command value range of feed per revolution>

	Increment system	
	IS-B	IS-C
Metric input	0.01 to 500.00 mm/rev	0.01 to 500.00 mm/rev
Inch input	0.0001 to 9.9999 inch/rev	0.0001 to 9.9999 inch/rev

Although a value in the above range can be specified in a feedrate command for feed per rotation, the feedrate is actually clamped at the maximum cutting feedrate.

NOTE

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

- **One-digit F code feed**

When a one-digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameter (Nos. 788 to 796) is used. When F0 is specified, the rapid traverse rate is applied.

The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing F1-digit feedrate on the machine operator's panel, then by rotating the manual pulse generator.

The increment/decrement, ΔF , in feedrate per scale of the manual pulse generator is as follows:

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

F_{\max} : feedrate upper limit for F1–F4 set by parameter 583, or
feedrate upper limit for F5–F9 set by parameter 584

X : any value of 1–127 set by parameter 216

The feedrate set or altered is kept even while the power is off. The current feed rate is displayed on the CRT screen.

- **Cutting feedrate clamp**

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 527. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit.

NOTE

An upper limit is set in mm/min or inch/min. CNC calculation may involve a feedrate error of $\pm 2\%$ with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more during the steady state:

5.4 CUTTING FEEDRATE CONTROL

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

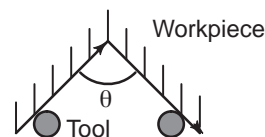
Table 5.4 Cutting feedrate control

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		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.
Tapping mode		G63	Once specified, this function is valid until G61, G62, or G64 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.
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 cutter 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 change	—	This function is valid in the cutter 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 can be disabled by setting bit 5 of parameter No. 020 to 1.
- 2 Inner corner angle θ : $2^\circ < \theta \leq \alpha \leq 178^\circ$

(α is a set value)



Format

Exact stop	G09IP _ ; G61 ;
Cutting mode	G64 ;
Tapping mode	G63 ;
Automatic corner override	G62 ;

5.4.1

Exact Stop (G09, G61)

Cutting Mode (G64)

Tapping Mode (G63)

Explanations

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

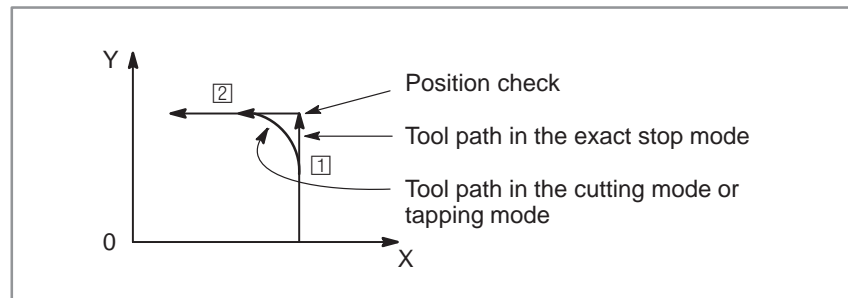


Fig. 5.4.1 Example of tool paths from block 1 to block 2

CAUTION

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

5.4.2 Automatic Override for Inner Corners (G62)

Explanations

- **Override condition**

When G62 is specified, and the tool path with cutter 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)

θ_p is a value set with parameter No. 215. When θ is approximately equal to θ_p , the inner corner is determined with an error of 0.001° or less.

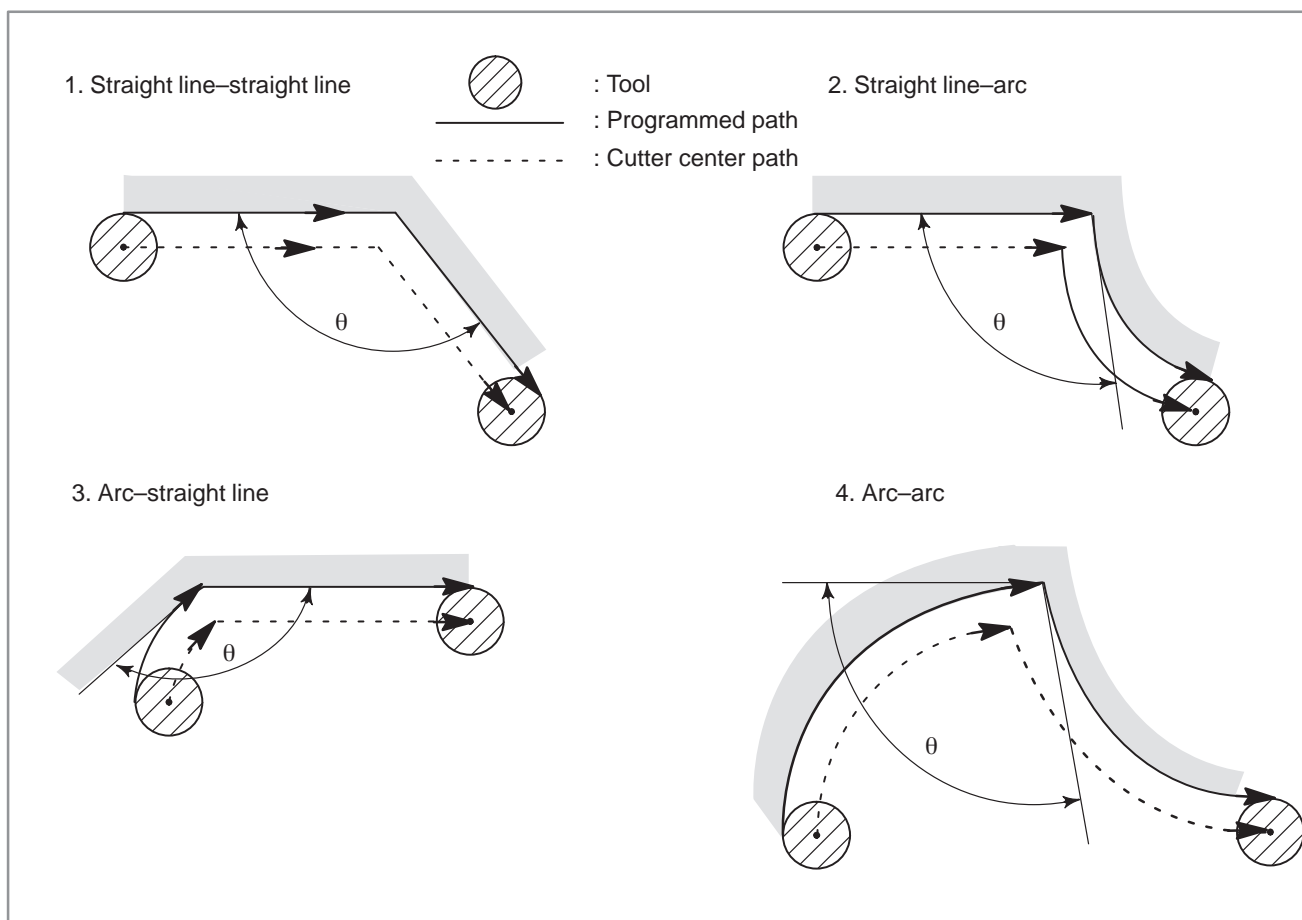


Fig. 5.4.2 (a) Inner corner

WARNING

When the block before a corner is a start-up block, or the block after a corner includes G41 or G42, the inner corner is not overridden. The inner corner override function is disabled when the offset value is 0.

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 inner corner is overridden, are distances from points on the cutter 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 parameter Nos. 580 and 581.

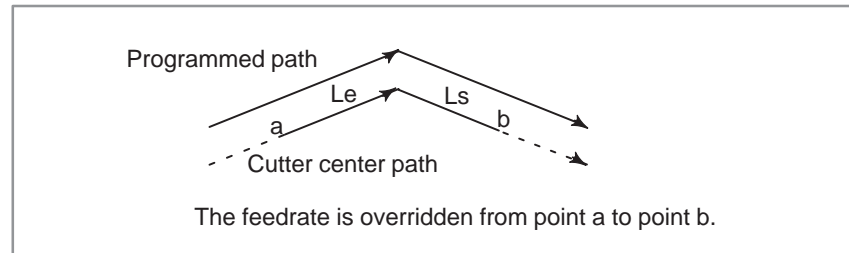


Fig. 5.4.2 (b) Override range (Straight line to straight line)

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)).

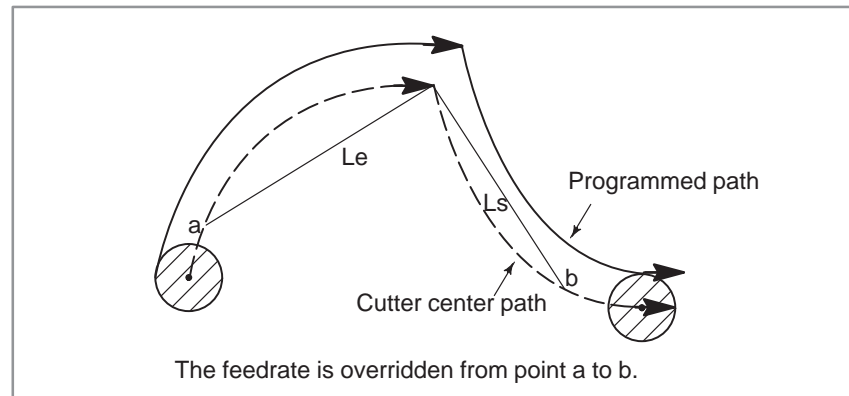


Fig. 5.4.2 (c) Override range (Arc to arc)

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)).

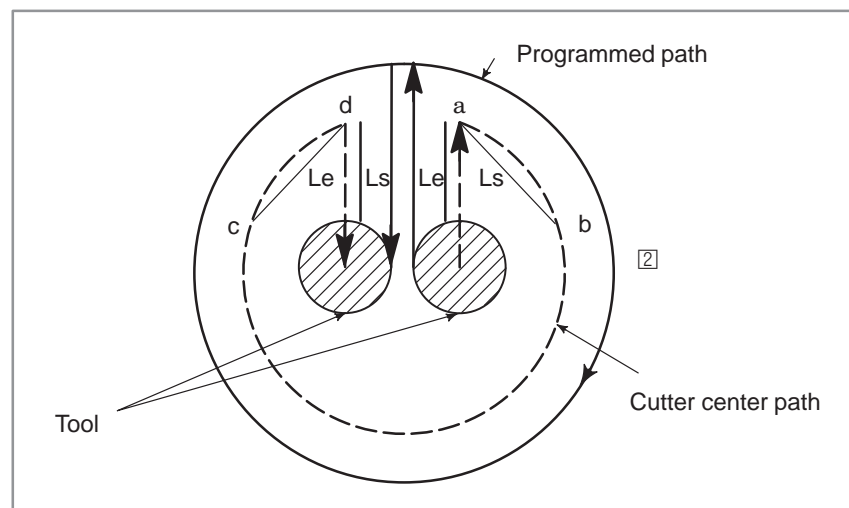


Fig. 5.4.2 (d) Override range (Straight line to arc, arc to straight line)

Override value

An override value is set with parameter No. 214. An override value is valid even for dry run and F1-digit specification.

In the feed per minute mode, the actual feedrate is as follows:

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

5.4.3 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. 5.4.3). This function is valid in the cutter compensation mode, regardless of the G62 code.

$$F \times \frac{R_c}{R_p}$$

R_c : Cutter center path radius

R_p : Programmed radius

It is also valid for the dry run and the one-digit F command.

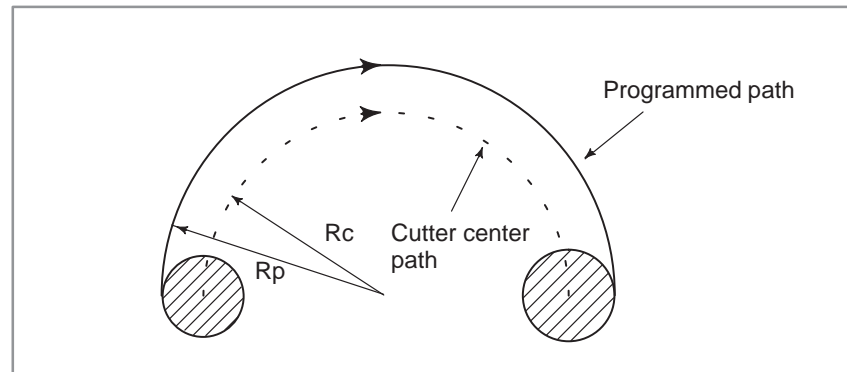


Fig. 5.4.3 Internal circular cutting feedrate change

If R_c is much smaller than R_p , $R_c/R_p \approx 0$; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 213. When $R_c/R_p \leq \text{MDR}$, the feedrate of the tool is $(F \times \text{MDR})$.

NOTE

When internal circular cutting must be performed together with automatic override for inner corners, the feedrate of the tool is as follows:

$$F \times \frac{R_c}{R_p} \times (\text{automatic override for the inner corners}) \times (\text{feedrate override})$$

5.5 DWELL (G04)

Format

Dwell **G04 X_ ;** or **G04 P_ ;**
 X_ : Specify a time (decimal point permitted)
 P_ : Specify a time (decimal point not permitted)

Explanations

By specifying a dwell, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact check in the cutting mode (G62 mode).

When neither P nor X is specified, exact stop is performed.

**Table 5.5 (a) Command value range of the dwell time
(Command by X)**

Increment system	Command value range	Dwell time unit
IS-B	0.001 to 99999.999	s
IS-C	0.0001 to 9999.9999	

**Table 5.5 (b) Command value range of the dwell time
(Command by P)**

Increment system	Command value range	Dwell time unit
IS-B	1 to 99999999	0.001 s
IS-C	1 to 99999999	0.0001 s

5.6 CUTTING FEED LINEAR ACCELERATION/ DECELERATION BEFORE INTERPOLATION

By using linear acceleration/deceleration before interpolation, linear acceleration/deceleration can be applied to a specified cutting feedrate before interpolation. With this function, the machining figure error resulting from delay in the acceleration/deceleration can be reduced.

This function is optional. It should be used in combination with two other functions: the block overlap function that suppresses feedrate variations between adjacent blocks, and the corner deceleration function that suppresses abrupt feedrate variations at a corner.

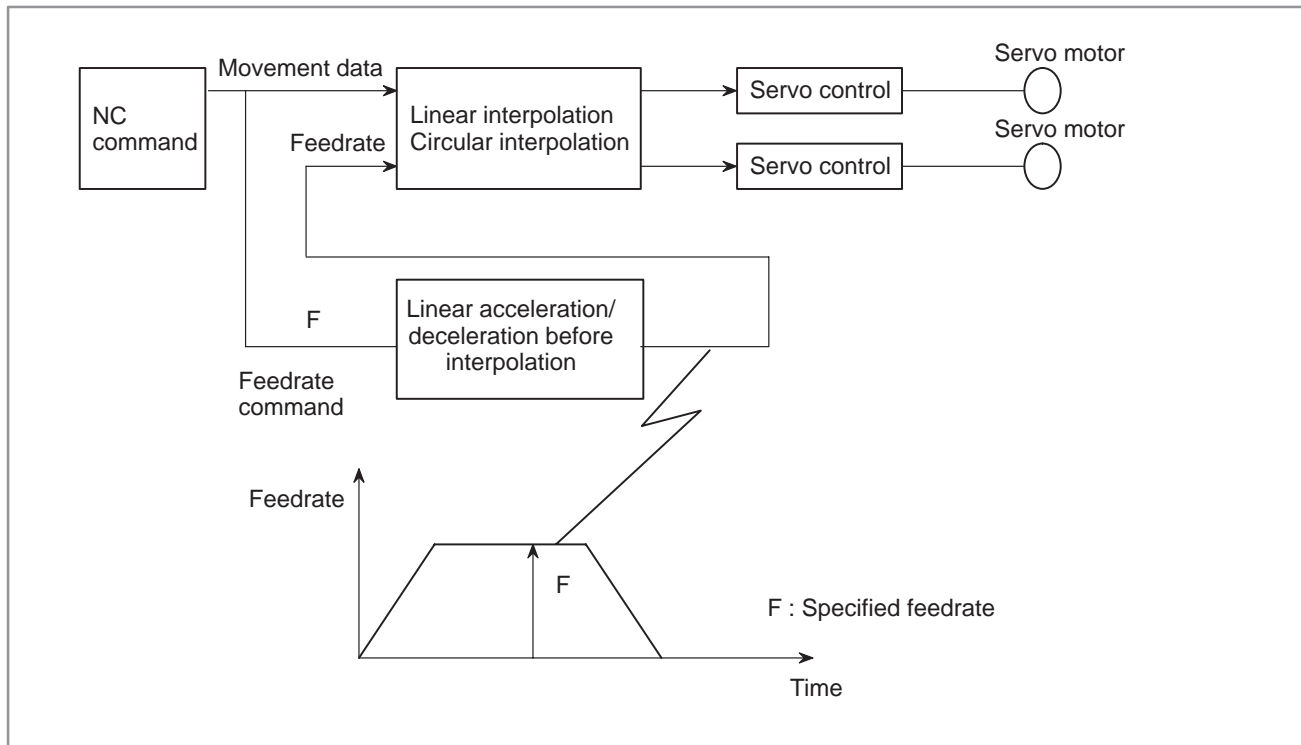


Fig. 5.6 (a)

Explanations

With linear acceleration/deceleration before interpolation, a specified feedrate that takes a feed-per-minute command (G94) override for linear or circular interpolation (G01, G02, G03) into consideration can be controlled such that the feedrate changes linearly. This means that acceleration/deceleration control may not be applied normally if the feedrate is changed during machining by other than programmed commands.

To control variations in the programmed feedrate, the user can select either of two control methods by setting parameters.

① Type A

This type of control starts acceleration/deceleration at that block where the specified feedrate is changed.

② Type B

For deceleration: Deceleration starts at the previous block, such that the deceleration ends at the start point of the block where a feedrate command change is made.

For acceleration: Acceleration starts at the start point of the block where a feedrate command change is made.

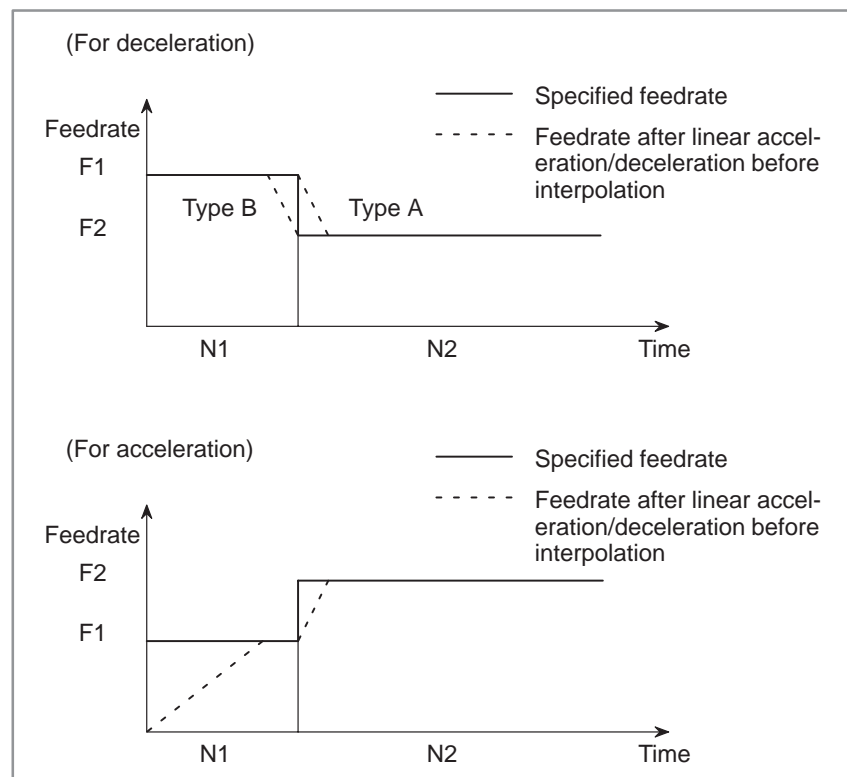


Fig. 5.6 (b)

WARNING

- 1 If an overtravel alarm is issued during linear acceleration/deceleration before interpolation, the tool decelerates and stops after the issue of the alarm, thus resulting in an overrun by an amount equal to the deceleration distance.
- 2 In inch input mode, simultaneous movement on two or more axes, including a rotation axis, cannot be specified during linear acceleration/deceleration before interpolation.

NOTE

- 1 The following cannot be specified during linear acceleration/deceleration before interpolation:
 - Control along the C-axis normal
 - Cylindrical interpolation
 - Polar coordinate specification
 - F 1-digit feed/threading/synchronous feed
 - Index table indexing
 - Rigid tapping
 - High-speed cycle machining/high-speed remote buffer
 - Handle interrupt
 - Program restart
 - Simple synchronous control
 - Interrupt-type custom macro
 - Automatic tool length measurement
 - Small-diameter peck drilling cycle
- 2 If any of the operations listed below is performed during operation, acceleration/deceleration will not be performed correctly. This means that the operation of a block may end before the completion of deceleration, or that the movement of a block may not have been completed when deceleration is completed.
 - ① Override change
 - ② Turning the dry run function on or off
 - ③ Enabling external deceleration
 - ④ Operation in a feed mode in which no feedrate is specified by the CNC program
- 3 Linear acceleration/deceleration before interpolation is valid for the X-axis, Y-axis, Z-axis, and 4th axis. If there is a block containing no move command along the X-axis, Y-axis, Z-axis, and 4th axis during linear acceleration/deceleration before interpolation, the tool decelerates and temporarily stops in the previous block.
- 4 If a one-shot G code is specified during linear acceleration/deceleration before interpolation, the tool decelerates and temporarily stops in the previous block.
- 5 If a block containing a move command also specifies an M/S/T code during linear acceleration/deceleration before interpolation, the tool decelerates and temporarily stops in the block.
- 6 Even during linear acceleration/deceleration before interpolation, acceleration/deceleration is not applied to a G31 (skip function) block.
- 7 Even if the Z-axis command cancel signal (ZNG) is turned on or off during linear acceleration/deceleration before interpolation, Z-axis acceleration/deceleration control is not applied.
- 8 During linear acceleration/deceleration before interpolation, the automatic corner override function can be used only to make an inner circular cutting feedrate change.
- 9 Even during linear acceleration/deceleration before interpolation, acceleration/deceleration after interpolation is enabled. So, to use acceleration/deceleration before interpolation only, set the time constant for acceleration/deceleration after interpolation to 0.

5.7 CUTTING FEED BLOCK OVERLAP

With this function, the remaining pulses output upon the completion of interpolation of a block are output together with the interpolation pulses of the next block to suppress the feedrate variations that can occur between the two blocks. This function is optional.

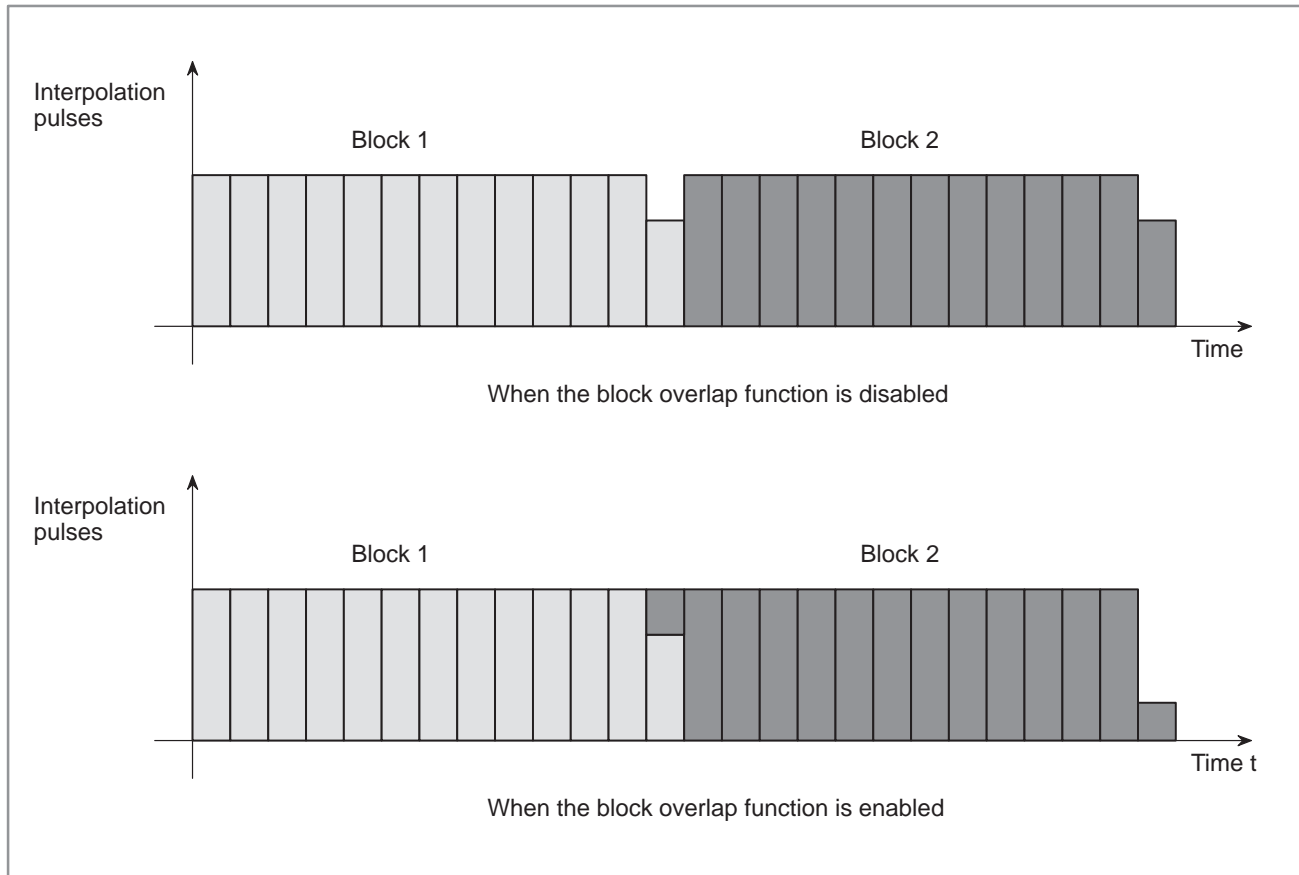


Fig. 5.7

Explanations

This function is enabled when G01, G02, and G03 blocks occur successively in G64 mode.

NOTE

- 1 If blocks specifying small amounts of movement occur in succession, the overlap function may not be enabled.
- 2 If the radius of the start point of a circular cutting command does not match the radius of the end point of the command, the overlap function will not be applied normally.

5.8 AUTOMATIC CORNER DECELERATION

In corner machining, this function controls the feedrate by decelerating the tool at a corner according to the corner angle between the blocks or according to the feedrate difference along each axis to improve the machining precision (acceleration/deceleration after interpolation) or to reduce the mechanical stress and strain (acceleration/deceleration before interpolation) at the corner. This function is optional.

This function is enabled when G64 mode (machining mode) specifying a cutting feed time constant is set, a cutting-feed block (block A) is followed by another cutting-feed block (block B), and deceleration is performed in block A. Two corner deceleration control methods are supported. One method is based on the corner angle between two blocks, while the other is based on the feedrate difference along each axis. The user can select either of the two methods by specifying bit 3 of parameter No. 395.

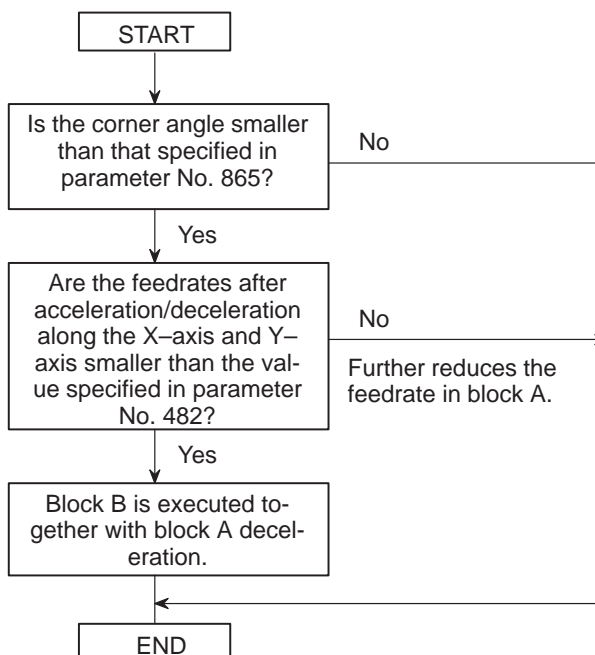
Explanations

- **Corner deceleration based on the corner angle (with bit 3 of parameter No. 395 set to 0)**

(1) Specification

When the angle between block A and block B (XY plane) is smaller than that specified in parameter No. 865, this function reduces the feedrate at the corner. This means that the execution of block B is suspended until the feedrates after acceleration/deceleration along the X-axis and Y-axis become smaller than the value specified in parameter No. 482.

The flowchart for this control mode is shown below.



When the corner angle is smaller than that specified in the parameter, the relationship between the feedrate and time will be as shown below. The movement of block A equivalent to the hatched area remains at time t . The execution of block B is started, however, because the feedrate after acceleration/deceleration is now smaller than the parameter-set value.

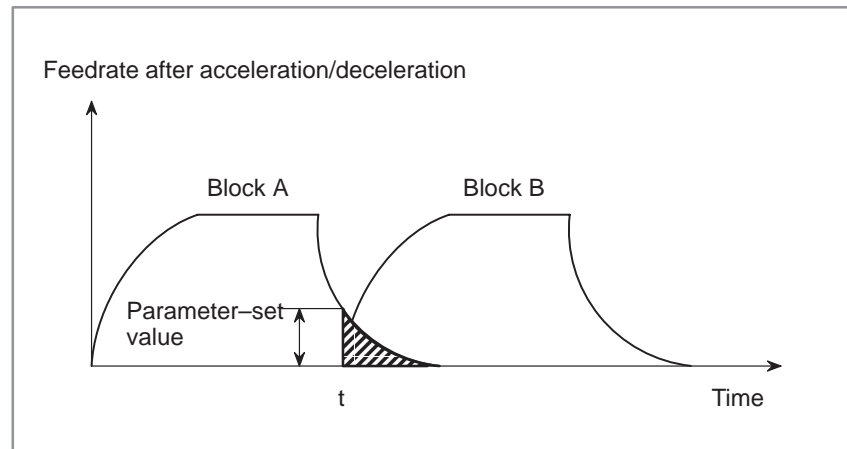


Fig. 5.8 (a)

(2) When linear acceleration/deceleration before interpolation is enabled

When the angle between block A and block B (XY plane) is smaller than that specified in parameter No. 865, and the feedrates specified for block A and block B are larger than the value specified in parameter No. 482, this function reduces the feedrate at the corner to the parameter-set feedrate at the end point of block A, then increases the feedrate to the specified feedrate when block B starts. In this case, acceleration/deceleration is performed according to the parameter setting for linear acceleration/deceleration before interpolation.

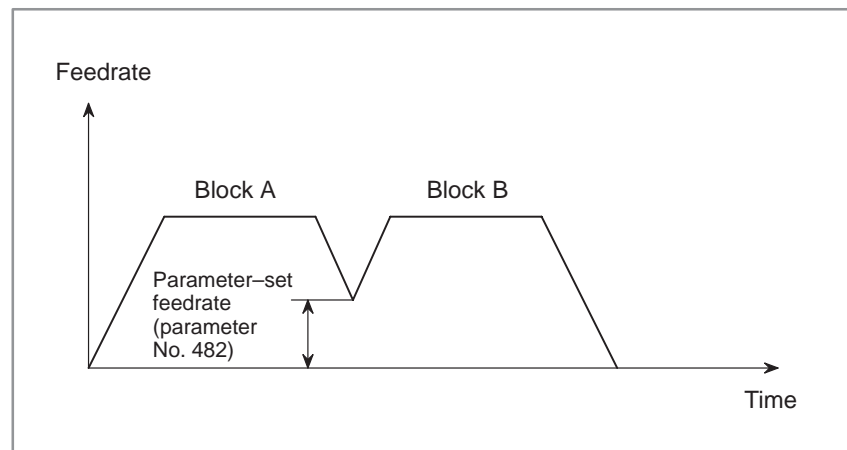
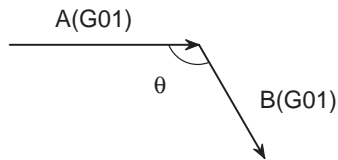


Fig. 5.8 (b)

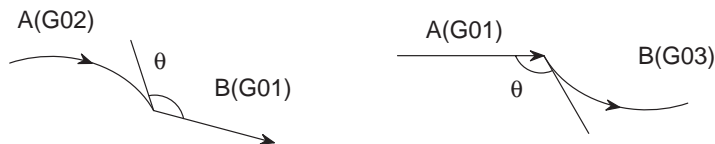
NOTE

- 1 Comparison between a machining angle and parameter-set angle (parameter No. 865) is made only for the XY plane. Comparison between a machining feedrate and parameter-set feedrate (parameter No. 482) is made only for the X-axis and Y-axis. So, when movement is made along three or more axes at the same time, only the X-axis and Y-axis are considered.
- 2 Corner roundness is determined by the angle and feedrate set in parameter Nos. 865 and 482. To ensure that a sharp corner is obtained at all times, set a feedrate of 0 and an angle of 180 degrees in the respective parameters.
- 3 When G09 (exact stop) is specified, exact stop operation is performed, regardless of the angle and feedrate set in the parameters.
- 4 This function cannot be used with the single step and dry run functions.
- 5 The angle (θ) between block A and block B is defined as shown below.

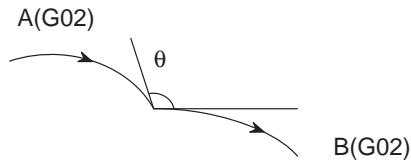
① Between two straight lines



② Between a straight line and an arc: Angle subtended by a straight line and the tangent to an arc



③ Between two arcs; Angle subtended by the tangent to one arc and the tangent to the other arc

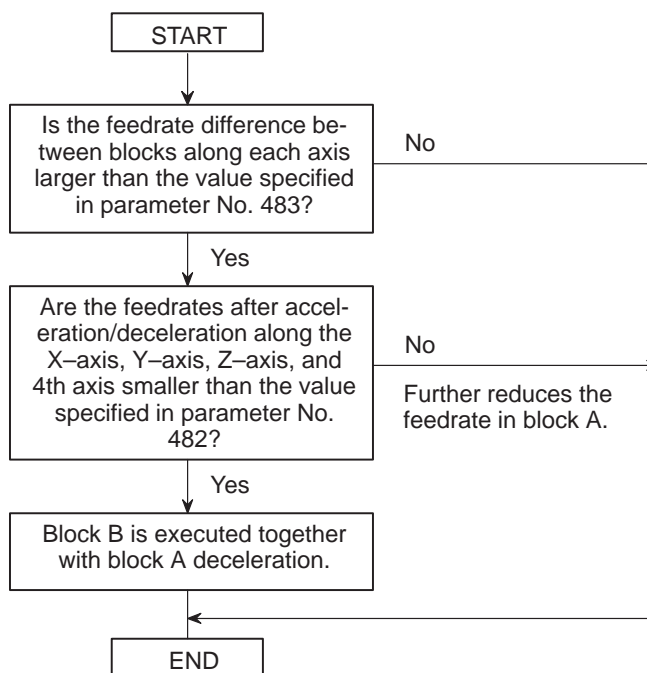


- **Corner deceleration based on feedrate difference between blocks along each axis (with bit 3 of parameter No. 395 set to 1)**

(1) Specification

When the difference between the specified feedrate at the end point of block A and the specified feedrate at the start point of block B along each axis is larger than the value specified in parameter No. 483, this function reduces the feedrate at the corner. This means that the execution of block B is suspended until the feedrates after acceleration/deceleration along the X-axis, Y-axis, Z-axis, and 4th axis become smaller than the value specified in parameter No. 482.

The flowchart for this control mode is shown below.



When the feedrate difference between blocks along each axis is larger than the value specified in parameter No. 483, the relationship between the feedrate and time will be as shown below. The movement of block A indicated by the hatched area remains at time t . However, the execution of block B is started because the feedrate after acceleration/deceleration has become smaller than the value set in parameter No. 482.

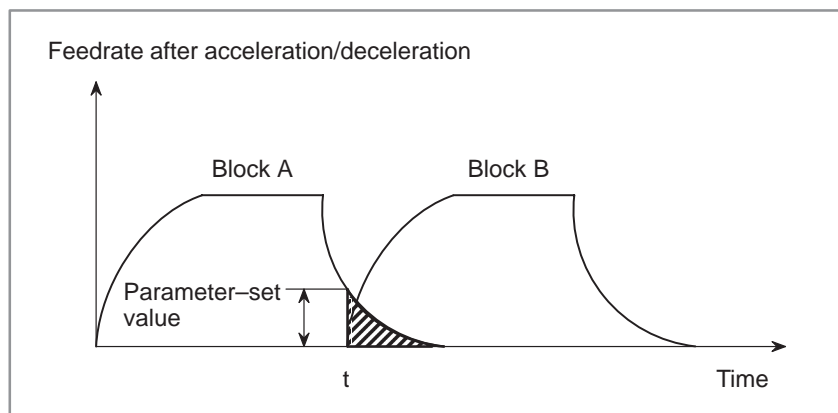


Fig. 5.8 (c)

(2) When linear acceleration/deceleration before interpolation is enabled

When the feedrate difference between block A and block B along each axis is larger than the value set in parameter No. 483, the feedrate is reduced in block A to the corner feedrate calculated from the feedrate difference along each axis.

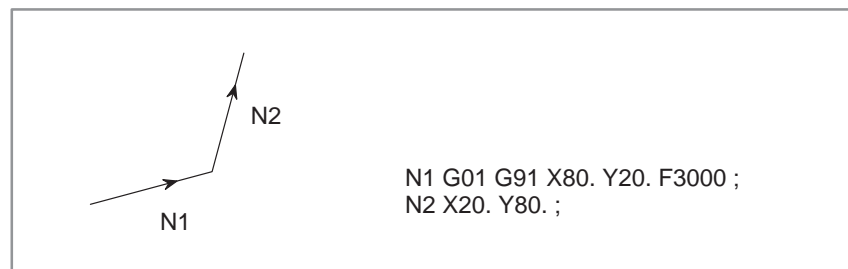
Let the feedrate be F. Compare the feedrate change along each axis ($V_c[X]$, $V_c[Y]$, ...) with the value, V_{max} , specified in parameter No. 483. When the change in the feedrate along at least one axis exceeds V_{max} , calculate R as follows:

$$R = \frac{V_c}{V_{max}}$$

Find the maximum value for R among the calculated values for the axes. Let this value be R_{max} . Then, the corner feedrate can be obtained as described below, and the feedrate reduced to the obtained corner feedrate.

$$F_c = F \times \frac{1}{R_{max}}$$

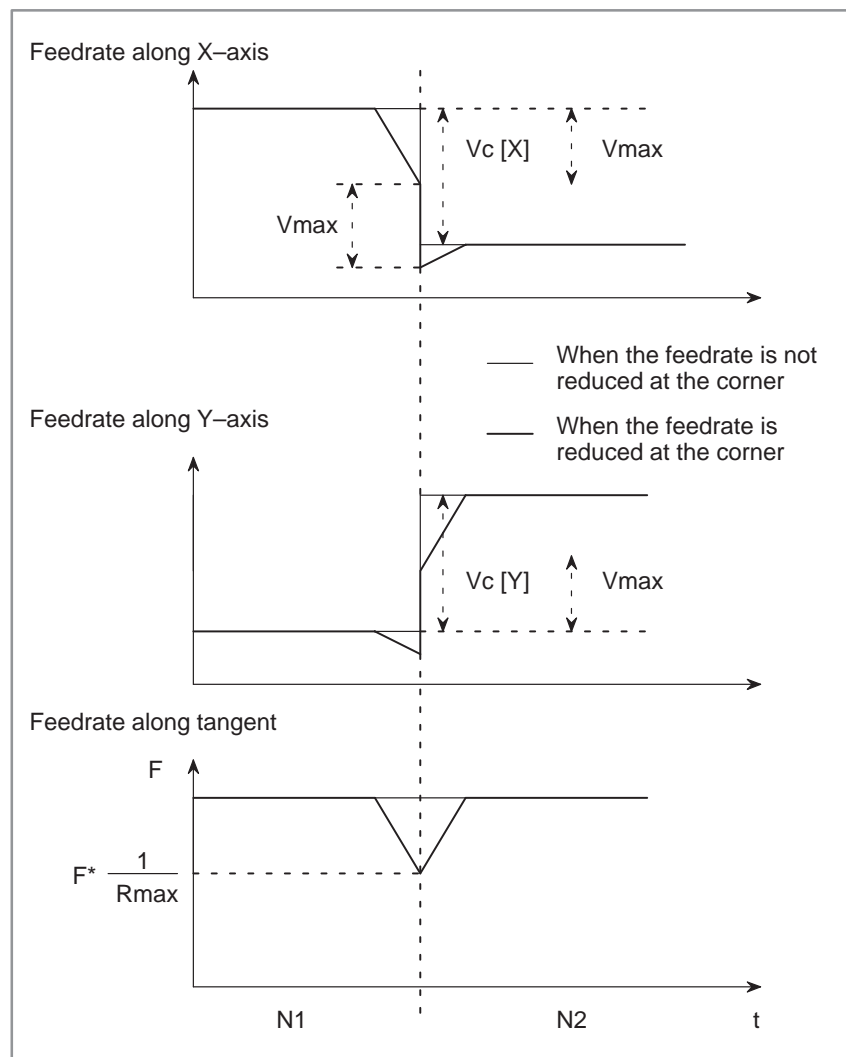
Example)



When this movement is specified, the feedrate along each axis will be as shown in the next figure. From the figure, it can be seen that the changes in the feedrate along the X-axis and Y-axis ($V_c[X]$ and $V_c[Y]$) exceed V_{max} . So, the feedrate change along each axis can be reduced to V_{max} or less by using the following values:

$$R_{max} = \frac{V_c [X(Y)]}{V_{max}}$$

$$\text{Corner feedrate} = F \times \frac{1}{R_{max}}$$

**NOTE**

- 1 A feedrate difference comparison between machining blocks is made for each of the X-axis, Y-axis, Z-axis, and 4th axis. Even during dry run operation or external deceleration, a feedrate difference check is made by coding an F command in the program.
- 2 When G09 (exact stop) is specified, exact stop operation is performed, regardless of the parameter settings.
- 3 This function cannot be used with a command based on feed per rotation, the F 1-digit command, rigid tapping command, or single block operation.
- 4 If an override change is applied during operation, the feedrate difference cannot be checked correctly.

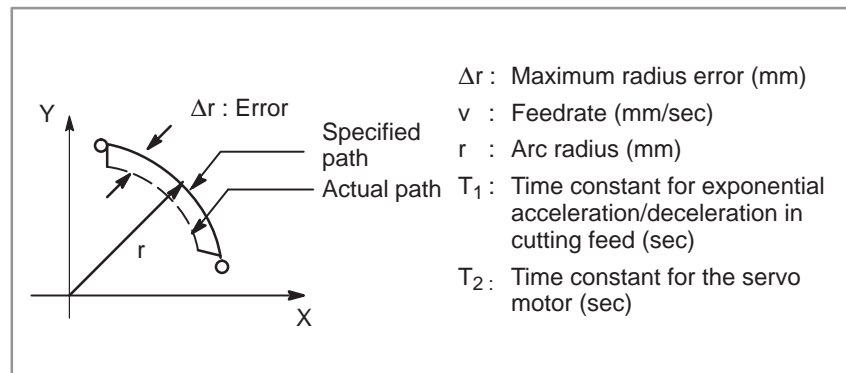
5.9 FEEDRATE CLAMP BASED ON ARC RADIUS

Particularly when high-speed circular cutting is performed using circular interpolation, the actual tool path incurs a radial error with respect to the specified arc. When a feedrate specified for an arc with a programmed radius can cause a radial error that exceeds the allowable error, this function automatically clamps the circular cutting feedrate.

This function is optional.

Explanations

An approximate arc radius error can be obtained using the following expression:



$$\Delta r = \frac{1}{2} (T_1^2 + T_2^2) \frac{v^2}{r} \quad \dots\dots\dots \text{(Expression 1)}$$

Let V be a maximum allowable feedrate for the arc radius R . Then, the arc radius error ΔR is given by:

$$\Delta R = \frac{1}{2} (T_1^2 + T_2^2) \frac{V^2}{R} \quad \dots\dots\dots \text{(Expression 2)}$$

On the other hand, from Expressions 1 and 2, the maximum allowable feedrate v for maintaining the allowable error ΔR for the arc radius r is found as follows:

$$\frac{1}{2} (T_1^2 + T_2^2) \frac{v^2}{r} = \frac{1}{2} (T_1^2 + T_2^2) \frac{V^2}{R}$$

Accordingly,

$$v = \sqrt{\frac{r}{R}} V \quad \dots\dots\dots \text{(Expression 3)}$$

By setting an arc radius (R) and the corresponding maximum allowable feedrate (V) in the parameters, the maximum allowable feedrate (v) for a programmed radius (r) can be found from Expression 3. If the specified feedrate exceeds the feedrate v , this function automatically clamps the feedrate to v . The maximum allowable feedrate (v), found from Expression 3, decreases as a specified arc radius decreases. When an excessively low feedrate is undesirable, parameter No. 496 can be used to set a lower limit on the circular cutting feedrate clamping.

If a specified feedrate is within the maximum allowable feedrate (v) found using Expression 3, circular cutting is performed at the specified feedrate.

NOTE

- 1 This function is also usable with linear acceleration/deceleration before interpolation. In this case, no arc radius error is caused by acceleration/deceleration before interpolation. When the time constant for acceleration/deceleration after interpolation is set to 0, an approximate arc radius error can be obtained using Expression 4:

$$\Delta r = \frac{1}{2} T_2^2 \times \frac{V^2}{r} \dots\dots\dots \text{(Expression 4)}$$

- 2 This function is also usable with linear acceleration/deceleration after interpolation for cutting feed. In this case, an approximate circular cutting error can be obtained from Expression 5:

$$\Delta r = \left(\frac{1}{24} T_1^2 + \frac{1}{2} T_2^2 \right) \frac{V^2}{r} \dots\dots\dots \text{(Expression 5)}$$

where

- Δr : Maximum radius error (mm)
 v : Feedrate (mm/sec)
 r : Arc radius (mm)
 T_1 : Time constant for linear acceleration/deceleration in cutting feed (sec)
 T_2 : Time constant for the servo motor (sec)

- 3 Expressions 1, 4, and 5 can be used only to find approximate values based on the NC theory. They do not determine actual machining errors. Let Δr_{all} be the actual machining error. Then, the following can be written:

$$\Delta r_{\text{all}} = \Delta r_{\text{NC}} + \Delta r_{\text{machine}} \dots\dots\dots \text{(Expression 6)}$$

This function controls or suppresses only those errors caused by the first term on the right-hand side of Expression 6, that is, errors caused by the time constant for acceleration/deceleration (T_1) and that for the servo motor (T_2). This function has no effect on errors caused by the machine. Note also that Expressions 1, 4, and 5 can determine only approximate values. As the arc radius decreases, the approximation precision degrades; in this case, even if the feedrate is clamped to the maximum allowable feedrate (v) obtained using Expression 3, the error may not be suppressed to within the allowable range.

5.10 RAPID TRAVERSE BELL-SHAPED ACCELERATION/ DECELERATION

This function can apply smooth acceleration/deceleration to a rapid traverse rate to reduce the mechanical stress and strain caused by acceleration/deceleration variations when the feedrate changes. Thus, a smaller time constant can be set, compared with linear acceleration/deceleration, to reduce the time required for acceleration/deceleration. Feed-forward control can be enabled for a rapid traverse command.

This function is optional.

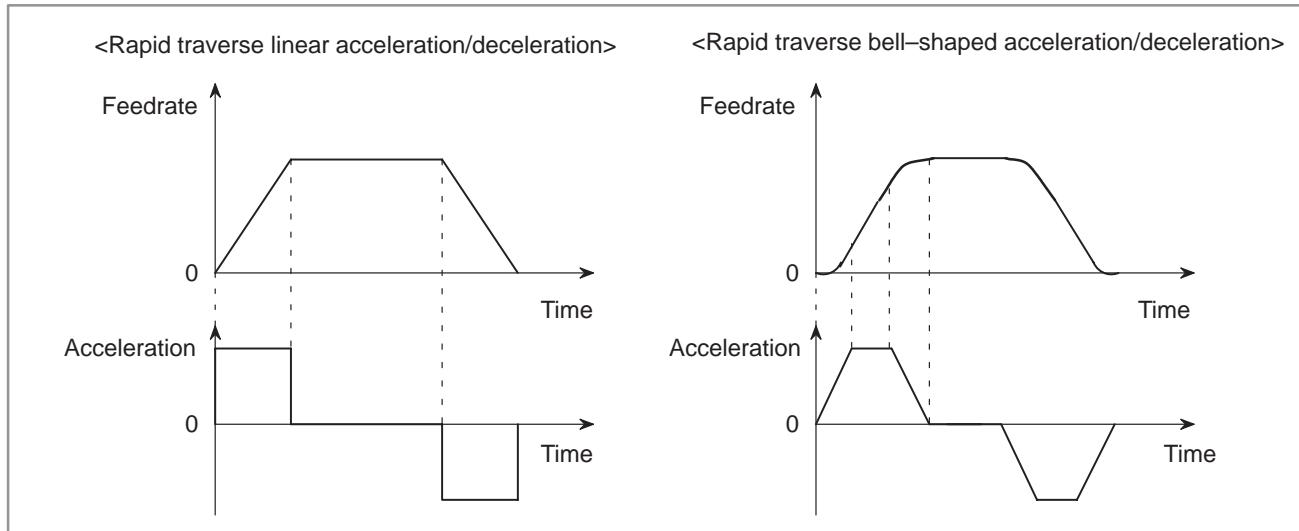


Fig. 5.10 (a)

Explanations

Rapid traverse bell-shaped acceleration/deceleration is enabled by setting two time constants (T_1 , T_2) (parameter No. 522 to No. 525, and parameter No. 952 to No. 955). In this case, the acceleration/deceleration time will be as shown below.

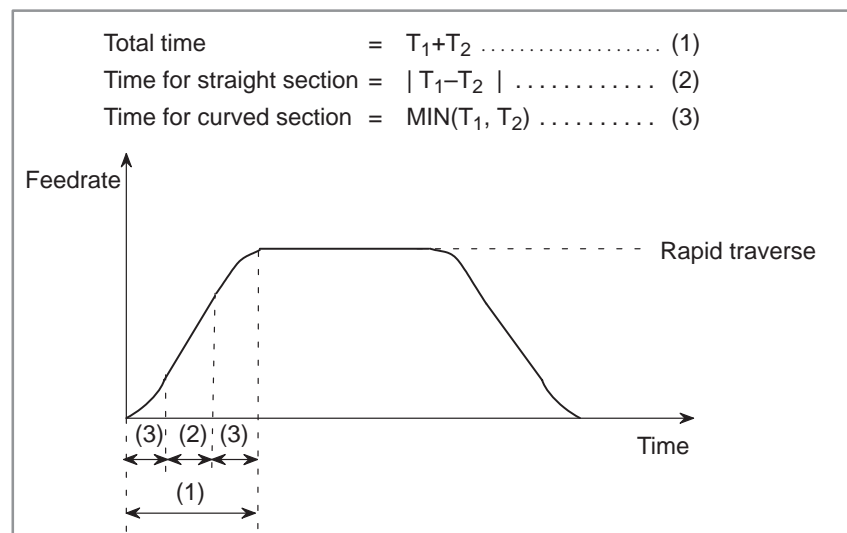


Fig. 5.10 (b)

NOTE

Rapid traverse bell-shaped acceleration/deceleration cannot be used with the 5th/6th axis and 7th/8th axis.

6 REFERENCE POSITION

General

- **Reference position**

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function.

For example, the reference position is used as a position at which tools are automatically changed. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters. The first reference position must be the machine zero point.

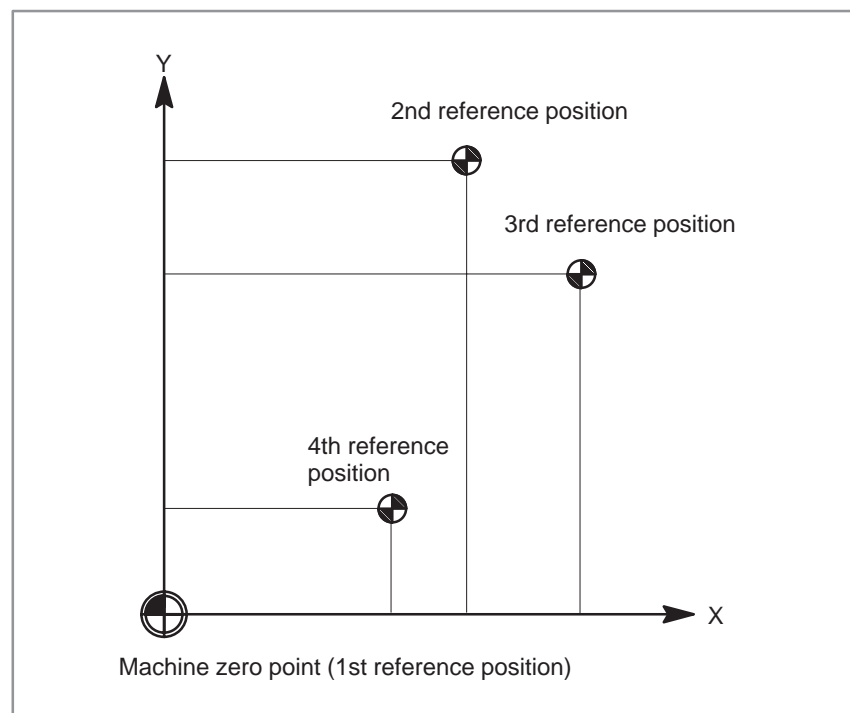


Fig. 6 (a) Machine zero point and reference positions

- **Reference position return and movement from the reference position**

Tools are automatically moved to the reference position via an intermediate position along a specified axis. Or, tools are automatically moved from the reference position to a specified position via an intermediate position along a specified axis. When reference position return is completed, the lamp for indicating the completion of return goes on.

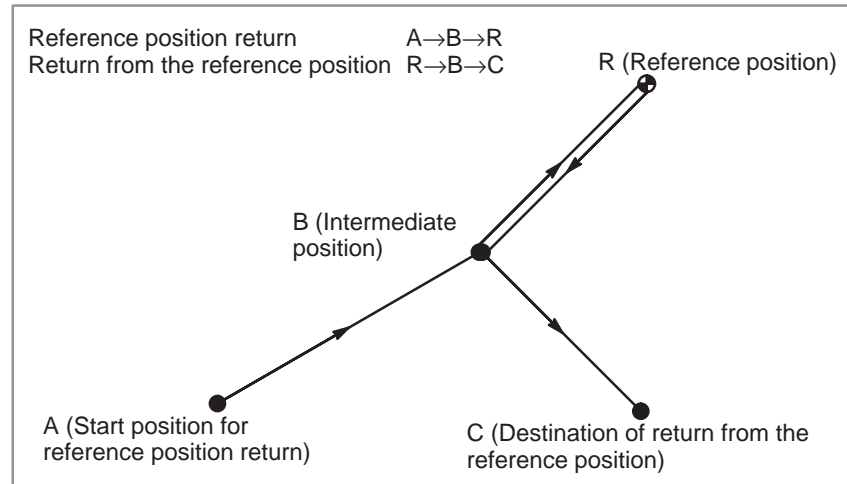


Fig. 6 (b) Reference position return and return from the reference position

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

- **Reference position return check**

Format

- **Reference position return**

G28 IP _ ; Reference position return

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

G30 P3 IP _ ; 3rd reference position return

G30 P4 IP _ ; 4th reference position return

IP_ : Command specifying the intermediate position
 (Absolute/incremental command)

- **Return from reference position**

G29 IP _ ;

IP_ : Command specifying the destination of return from reference position (Absolute/incremental command)

- **Reference position return check**

G27 IP _ ;

IP_ : Command specifying the reference position
 (Absolute/incremental command)

Explanations

- **Reference position return (G28)**

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis.
Therefore, for safety, the cutter compensation, and tool length compensation should be cancelled before executing this command.
The coordinates for the intermediate position are stored in the CNC only for the axes for which a value is specified in a G28 block. For the other axes, the previously specified coordinates are used.
Example N1 G28 X40.0 ; Intermediate position (X40.0)
N2 G28 Y60.0 ; Intermediate position (X40.0, Y60.0)
- **2nd, 3rd, and 4th reference position return (G30)**

In a system without an absolute-position detector, the first, third, and fourth reference position return functions can be used only after the reference position return (G28) or manual reference position return (see III-3.1) is made. The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.
- **Return from the reference position (G29)**

In general, it is commanded immediately following the G28 command or G30. For incremental programming, the command value specifies the incremental value from the intermediate point.
Positioning to the intermediate or reference points are performed at the rapid traverse rate of each axis.
When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to the commanded position through the intermediate point which has been shifted to the new coordinate system.
The same operations are performed also for G30 commands.
- **Reference position return check (G27)**

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the reference position return lamp lights up. However, if the position reached by the tool is not the reference position, an alarm (No. 092) is displayed.

Restrictions

- **Status the machine lock being turned on**

The lamp for indicating the completion of return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a G27 command is specified.
- **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 (No. 003 #0 to #3). Therefore the specified intermediate position must be a position to which reference position return is possible.

- **Reference position return check in an offset mode**

In an offset mode, the position to be reached by the tool with the G27 command is the position obtained by adding the offset value. Therefore, if the position with the offset value added is not the reference position, the lamp does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

- **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 occurs because the least input increment of the machine tool system is smaller than its least command increment.

Reference

- **Manual reference position return**

See III-3.1.

Examples

G28G90X1000.0Y500.0 ; (Programs movement from A to B)
 T1111 ; (Changing the tool at the reference position)
 G29X1300.0Y200.0 ; (Programs movement from B to C)

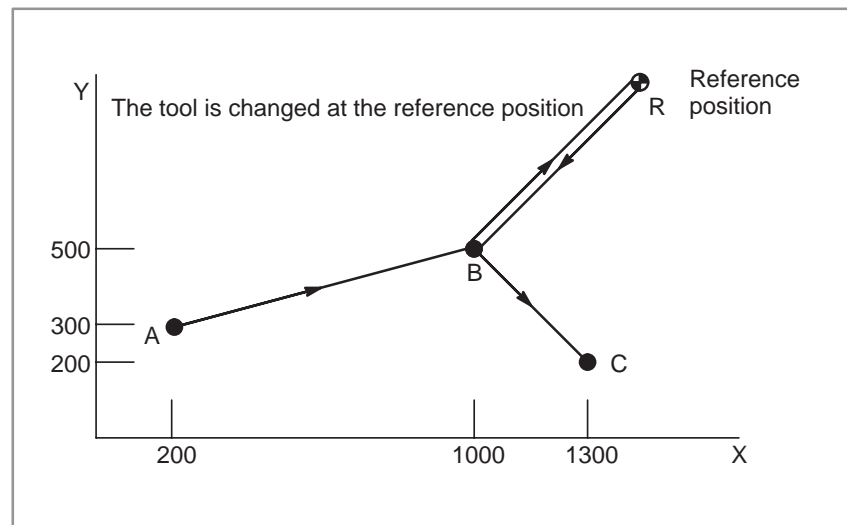


Fig. 6 (c) Reference position return and return from the reference position

7 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes. When three program axes, the X-axis, Y-axis, and Z-axis, are used, coordinates are specified as follows:

X_Y_Z_

This command is referred to as a dimension word.

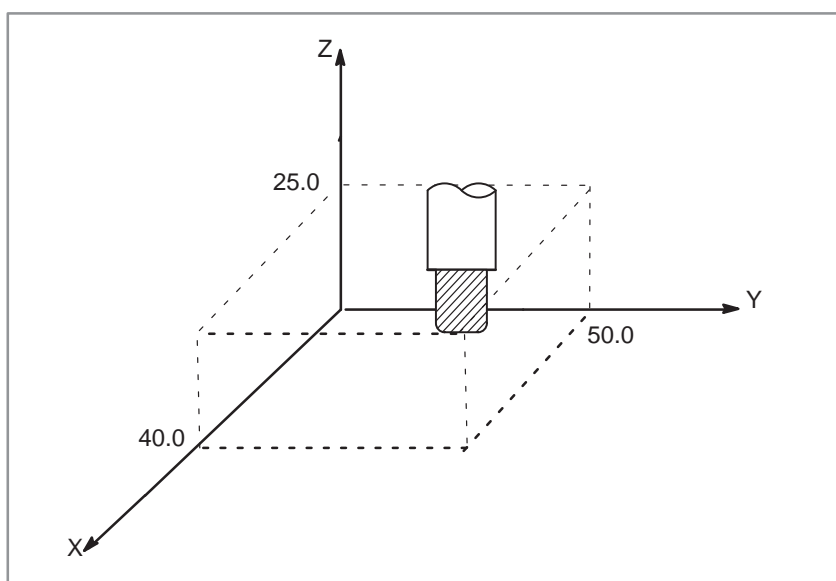


Fig. 7 Tool position specified by X40.0Y50.0Z25.0

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. The machine zero point matches the first reference position.

A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.

A machine coordinate system is set by performing manual reference position return after power-on (see III-3.1). A machine coordinate system, once set, remains unchanged until the power is turned off.

Format

```
G53 IP _ ;  
IP _ ; Absolute dimension word
```

Explanations

- **Selecting a machine coordinate system (G53)**

When a command is specified based on a machine coordinate system, the tool moves 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. The absolute command G90 is valid, but the incremental command G91 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.

Restrictions

- **Cancel of the compensation function**
- **G53 specification immediately after power-on**

When the G53 command is specified, cancel the cutter compensation, tool length offset, and tool offset.

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

The option for the workpiece coordinate system (G54 to G59) is required.

7.2 WORKPIECE COORDINATE SYSTEM

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 NC 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 G92

A workpiece coordinate system is set by specifying a value after G92 in the program.

(2) Automatic setting

If bit 0 of parameter No. 010 #7 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed (see Part III.).

(3) Input using the CRT/MDI panel

Six workpiece coordinate systems can be set beforehand using the CRT/MDI panel (see Part III-3.1.).

To use absolute programming, establish a workpiece coordinate system by applying one of the methods described above.

Format

Setting a workpiece
coordinate system by G92

G92IP _ ;

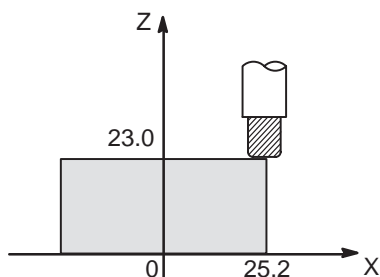
Explanations

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set. Cutter compensation is cancelled temporarily with G92.

Examples

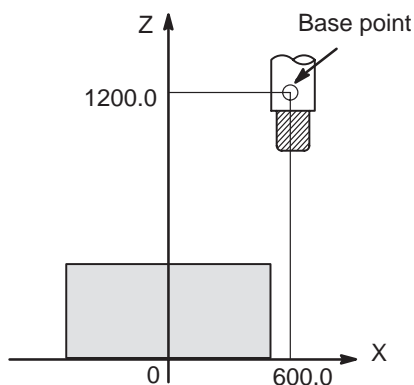
Example 1

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



Example 2

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



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

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 Section 7.2.1.)

(1) Selecting a workpiece coordinate system set by G92 or automatic workpiece coordinate system setting

Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

(2) Choosing from six workpiece coordinate systems set using the CRT/MDI panel

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

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

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

Examples

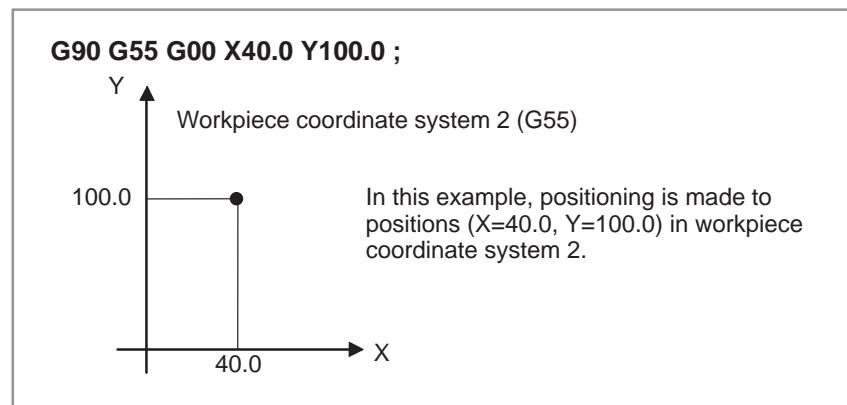


Fig. 7.2.2

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 zero point offset value or workpiece zero point offset value.

Four methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

- (1) Inputting from the CRT/MDI panel (see III-11.4.3)
- (2) Programming by G10 or G92
- (3) Changing an external workpiece zero point offset value (refer to machine tool builder's manual)
- (4) System variables by Custom Macro B

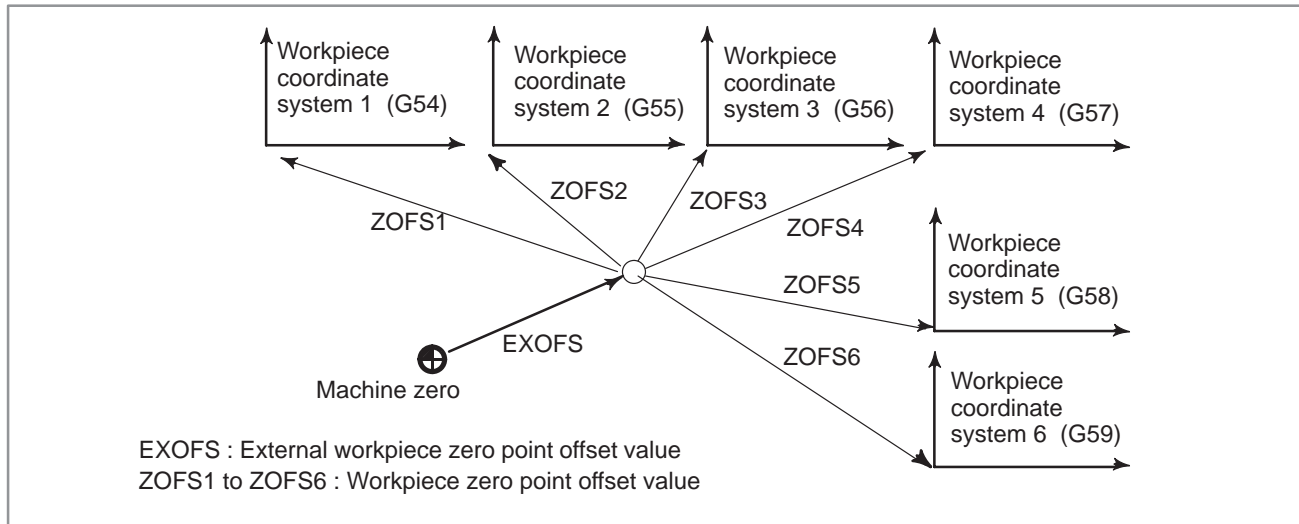


Fig. 7.2.3 Changing an external workpiece zero point offset value or workpiece zero point offset value

Format

• Changing by G10

G10 L2 P_p IP _ ;

p=0 : External workpiece zero point offset value

p=1 to 6 : Workpiece zero point offset value correspond to workpiece coordinate system 1 to 6

IP : Workpiece zero point offset value of each axis

• Changing by G92

G92 IP _ ;

Explanations

- **Changing by G10**

With the G10 command, each workpiece coordinate system can be changed separately.

When an absolute workpiece zero point offset value is specified, the specified value becomes a new offset value. When an incremental workpiece zero point offset value is specified, the specified value is added to the current offset value to produce a new offset value.

- **Changing by G92**

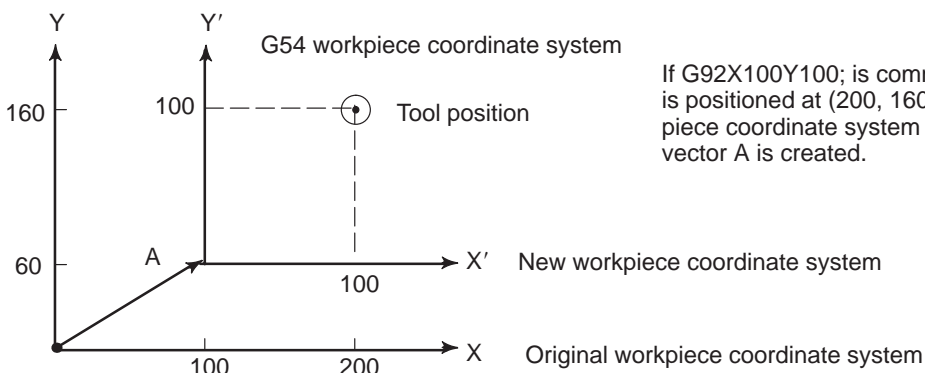
By specifying G92IP_;, a workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).

Then, the amount of coordinate system shift is added to all the workpiece zero point offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

WARNING

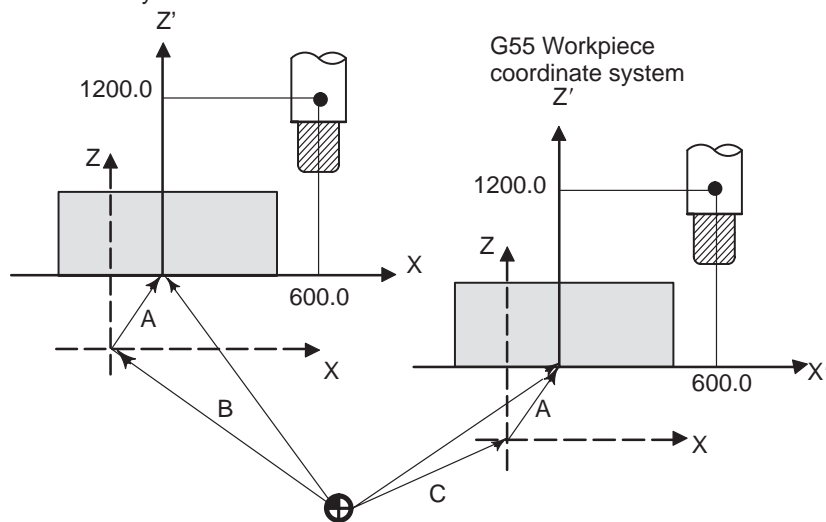
When a coordinate system is set with G92 after an external workpiece zero point offset value is set, the coordinate system is not affected by the external workpiece zero point offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position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

Examples



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

G54 Workpiece coordinate system



X' - Z' -- New workpiece coordinate system
 X - Z -- Original workpiece coordinate system
 A : Offset value created by G92
 B : G54 workpiece zero point offset value
 C : G55 workpiece zero point offset value

Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black circle on the tool (figure at the left) is at (600.0, 12000.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly: G92X600.0Z1200.0; Also, suppose that pallets are loaded at two different positions. If the relative relationship of the coordinate systems of the pallets at the two positions is correctly set by handling the coordinate systems as the G54 workpiece coordinate system and G55 workpiece coordinate system, a coordinate system shift with G92 in one pallet causes the same coordinate system shift in the other pallet. This means that workpieces on two pallets can be machined with the same program just by specifying G54 or G55.

7.2.4

Adding Workpiece Coordinate Systems

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

Format

- **Selecting the additional workpiece coordinate systems**

G54Pn ;
Pn : Codes specifying the additional workpiece coordinate systems
n : 1 to 48

- **Setting the workpiece zero point offset value in the additional workpiece coordinate systems**

G10L20 Pn IP _ ;
Pn : Codes specifying the workpiece coordinate system for setting the workpiece zero point offset value
n : 1 to 48
IP_: Axis addresses and a value set as the workpiece zero point offset

Explanations

- **Selecting the additional workpiece coordinate systems**


When a P code is specified together with G54, the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48).

A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power-on.

G54 P1 Additional workpiece coordinate system 1
 G54 P2 Additional workpiece coordinate system 2
 ⋮
 G54 P48 Additional workpiece coordinate system 48

- **Setting the workpiece zero point offset value in the additional workpiece coordinate systems**

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

- (1) The  function key can be used to display and set a workpiece zero point offset value.
- (2) The G10 function enables a workpiece zero point offset value to be set by programming.
- (3) A custom macro allows a workpiece zero point offset value to be handled as a system variable.
- (4) Workpiece zero point offset data can be entered or output as external data.

Restrictions

- **Specifying P codes**

A P code must be specified after G54.

If a value not within the specifiable range is specified in a P code, an alarm (No. 030) is issued.

7.3 LOCAL COORDINATE SYSTEM

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

Format

G52IP _ ; Setting the local coordinate system

.....

G52IP 0 ; Canceling of the local coordinate system

IP_ : Origin of the local coordinate system

Explanations

By specifying G52IP_;, 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.

When a local coordinate system is set, the move commands in absolute mode (G90), which is subsequently commanded, are the coordinate values in the local coordinate system. The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system.

To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

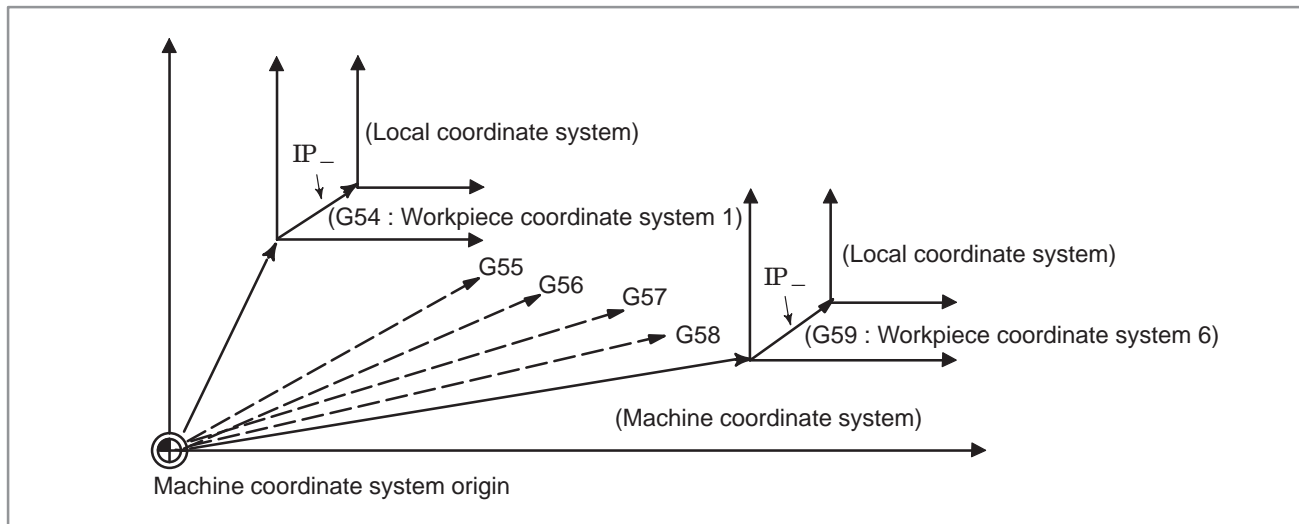


Fig. 7.3 Setting the local coordinate system

WARNING

- 1 When an axis returns to the reference point by the manual reference point return function, the zero point of the local coordinate system of the axis matches that of the work coordinate system. The same is true when the following command is issued:

G52 α 0;

α :Axis which returns to the reference point

- 2 The local coordinate system setting does not change the workpiece and machine coordinate systems.
- 3 The local coordinate system is cancelled when the reset operation is performed.
- 4 When setting a workpiece coordinate system with the G92 command, the local coordinate systems are cancelled.
- 5 G52 cancels the offset temporarily in cutter compensation.
- 6 Command a move command immediately after the G52 block in the absolute mode.
- 7 The option for the workpiece coordinate system (G54 to G59) is required.

7.4 PLANE SELECTION

Select the planes for circular interpolation, cutter compensation, and drilling by G-code.

The following table lists G-codes and the planes selected by them.

Explanations

Table 7.4 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. 279 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.

Examples

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

G17X_Y_ XY plane,

G17U_Y_ UY plane

G18X_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.

NOTE

When the system is turned on or placed in the clear state by a reset (bit 6 of parameter No. 045), G17, G18, or G19 is selected according to the setting of parameter No. 212.

8

COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

8.2 POLAR COORDINATE COMMAND (G15, G16)

8.3 INCH/METRIC CONVERSION (G20, G21)

8.4 DECIMAL POINT PROGRAMMING

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

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

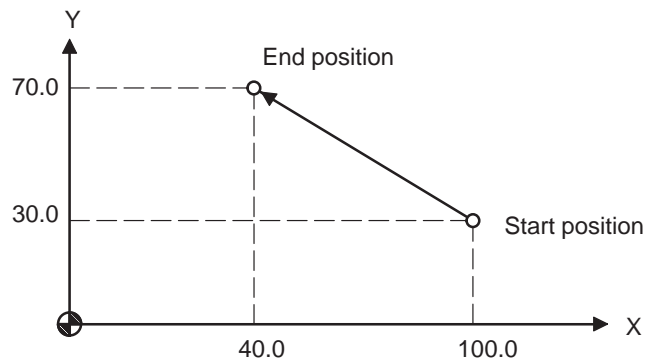
Format

Absolute command **G90 IP _ ;**

Incremental command **G91 IP _ ;**

Examples

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



8.2

POLAR COORDINATE COMMAND (G15, G16)

The end point coordinate value can be input in polar coordinates (radius and angle).

The plus direction of the angle is counterclockwise of the selected plane first axis + direction, and the minus direction is clockwise.

Both radius and angle can be commanded in either absolute or incremental command (G90, G91).

Format

G□□ G○○ G16 ; Starting the polar coordinate command
(polar coordinate mode)

G△△ IP _ ;
⋮
⋮ } Polar coordinate command

G15 ; Canceling the polar coordinate command
(polar coordinate mode)

G16 : Polar coordinate command

G15 : Polar coordinate command cancel

G□□ : Plane selection of the polar coordinate command
(G17, G18 or G19)

G○○ : G90 specifies the zero point of the Iworkpiece coordinate
system as the origin of the polar coordinate system, from
which a radius is measured.
G91 specifies the current position as the origin of the polar
coordinate system, from which a radius is measured.

G△△ : Angle selection of the polar coordinate command

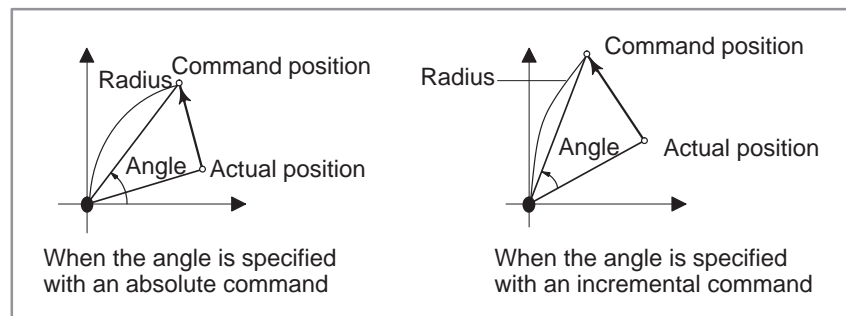
G90 : Angle is specified with absolute command

G91 : Angle is specified with incremental command

IP _ ; Specifying the addresses of axes constituting the plane se-
lected for the polar coordinate system, and their values
First axis : radius of polar coordinate
Second axis : radius of polar coordinate

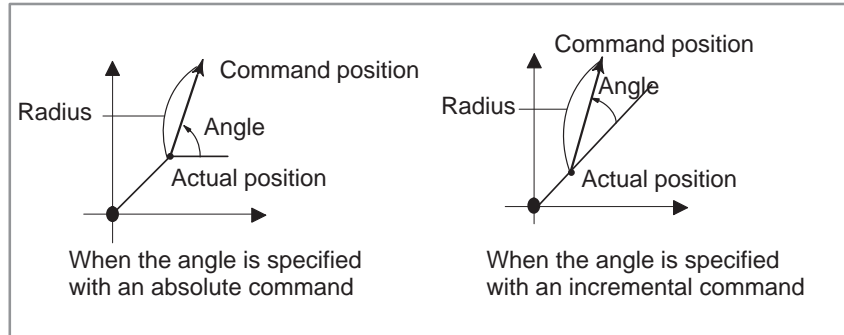
- **Setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system**

Specify the radius (the distance between the zero point and the point) to be programmed with an absolute command. The zero point of the workpiece coordinate system is set as the origin of the polar coordinate system.



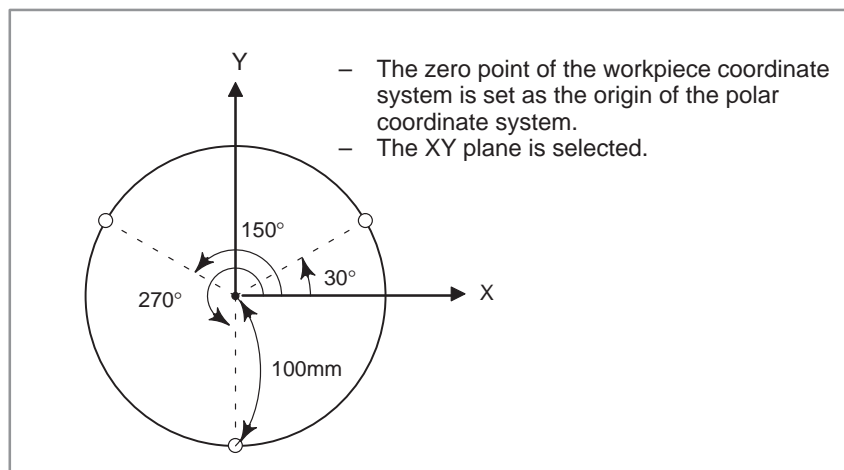
- **Setting the current position as the origin of the polar coordinate system**

Specify the radius (the distance between the current position and the point) to be programmed with an incremental command. The current position is set as the origin of the polar coordinate system.



Examples

Bolt hole circle



- **Specifying angles and a radius with absolute commands**

N1 G17 G90 G16 ;

Specifying the polar coordinate command and selecting the XY plane
Setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system

N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;

Specifying a distance of 100 mm and an angle of 30 degrees

N3 Y150.0 ;

Specifying a distance of 100 mm and an angle of 150 degrees

N4 Y270.0 ;

Specifying a distance of 100 mm and an angle of 270 degrees

N5 G15 G80 ;

Canceling the polar coordinate command

- **Specifying angles with incremental commands and a radius with absolute commands**

N1 G17 G90 G16;

Specifying the polar coordinate command and selecting the XY plane
Setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system

N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;

Specifying a distance of 100 mm and an angle of 30 degrees

N3 G91 Y120.0 ;

Specifying a distance of 100 mm and an angle of +120 degrees

N4 Y120.0 ;

Specifying a distance of 100 mm and an angle of +120 degrees

N5 G15 G80 ;

Canceling the polar coordinate command

Limitations

- **Specifying a radius in the polar coordinate mode**

In the polar coordinate mode, specify a radius for circular interpolation or helical cutting (G02, G03) with R.

- **Axes that are not considered part of a polar coordinate command in the polar coordinate mode**

Axes specified for the following commands are not considered part of the polar coordinate command:

- Dwell (G04)
- Changing an offset value with a program (G10)
- Setting the local coordinate system (G52)
- Converting the workpiece coordinate system (G92)
- Selecting the machine coordinate system (G53)
- Stored stroke check (G22)
- Coordinate system rotation (G68)
- Scaling (G51)

8.3

INCH/METRIC CONVERSION (G20, G21)

Format

Either inch or metric input can be selected by G code.

G20 ; Inch input

G21 ; mm 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 IS-B or IS-C (Section 2.3). The unit of data input for degrees remains unchanged. The unit systems for the following values are changed after inch/metric conversion:

- Feedrate commanded by F code
- Positional command
- Work zero point 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

- 1 G20 and G21 must not be switched during a program.
- 2 When switching inch input (G20) to metric input (G21) and vice versa, the tool compensation value must be re-set according to the least input increment.

CAUTION

Reference position return is performed at a low speed for the first G28 command after the inch input is switched to the metric input or vice versa.

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.

8.4 DECIMAL POINT PROGRAMMING

Explanations

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, and F.

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 mm, 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 parameter No. 051#7. Values can be specified both with and without decimal point in a single program.

Examples

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value with- out decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

WARNING

In a single block, specify a G code before entering a value. The position of decimal point may depend on the command.

Examples:

G20; Input in inches

X1.0 G04; X1.0 is considered to be a distance and processed as X10000. This command is equivalent to G04 X10000. The tool dwells for 10 seconds.

G04 X1.0; Equivalent to G04 X1000. The tool dwells for one second.

NOTE

- 1 Fractions less than the least input increment are truncated.

Examples:

X1.2345; Truncated to X1.234 when the least input increment is 0.001 mm.
Processed as X1.2345 when the least input increment is 0.0001 inch.

- 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 003 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

SPINDLE SPEED FUNCTION (S FUNCTION)

The spindle speed can be controlled by specifying a value following address S.

This chapter contains the following topics.

- 9.1 SPECIFYING THE SPINDLE SPEED WITH A BINARY CODE**
- 9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY
(S5-DIGIT COMMAND)**
- 9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)**

9.1 SPECIFYING THE SPINDLE SPEED WITH A BINARY CODE

A 2-digit S code can be specified in a block. For a description of the use of S codes, such as their execution sequence in a block in which a spindle speed, move command, and S code are specified, see the manual provided by the machine tool builder.

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

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

9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

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

Format

- Constant surface speed control command

G96 P α S○○○○○ ;

↑Surface speed (m/min or feet/min)

P0 : Axis set in the parameter (No. 041#4, #5)

P1 : X axis, P2 : Y axis, P3 : Z axis, P4 : 4th axis

Note : This surface speed unit may change according to machine tool builder's specification.

If a surface speed is omitted, the S value specified in the previous G96 is used.

- Constant surface speed control cancel command

G97 S○○○○○ ;

↑Spindle speed (rpm)

Note : This surface speed unit may change according to machine tool builder's specification.

If a spindle speed is omitted, the last speed specified in G96 remains valid.

- Clamp of maximum spindle speed

G92 S○○○○○ ;

↑Maximum spindle speed (rpm) used during constant surface speed control

Note : The measurement unit of this maximum spindle speed may vary from one machine builder to another.

Explanations

- **Constant surface speed control command (G96)**

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G96 command must specify the axis along which constant surface speed control is applied. If address P is omitted, P0 is assumed. A G97 command cancels the G96 mode. If an S value is omitted in a G97 block, the last speed specified in the G96 mode remains valid. When constant surface speed control is applied, a spindle speed higher than the value specified in G92S_; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped. S (surface speed) commands in the G96 mode are assumed as $S = 0$ (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

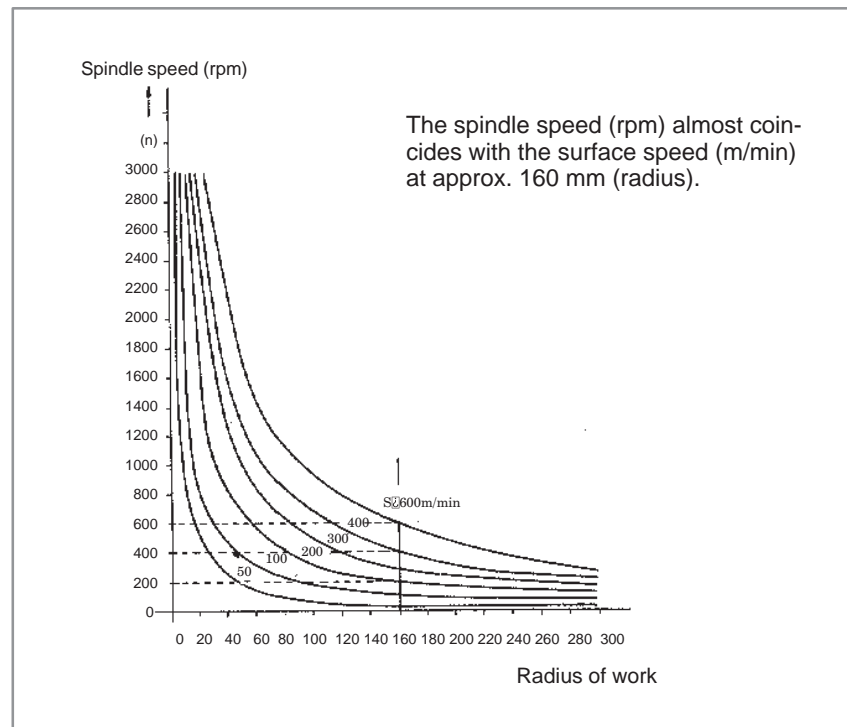


Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

- **Setting the workpiece coordinate system for constant surface speed control**

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

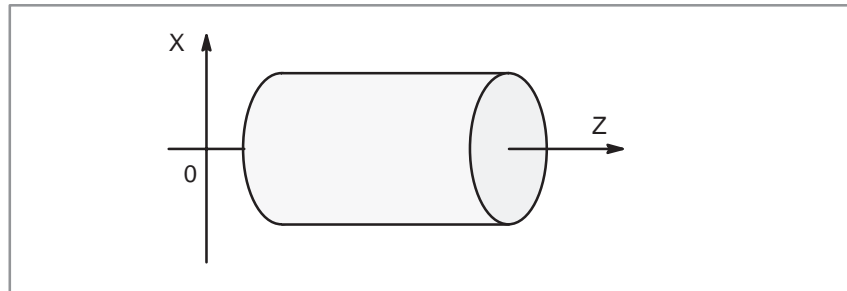
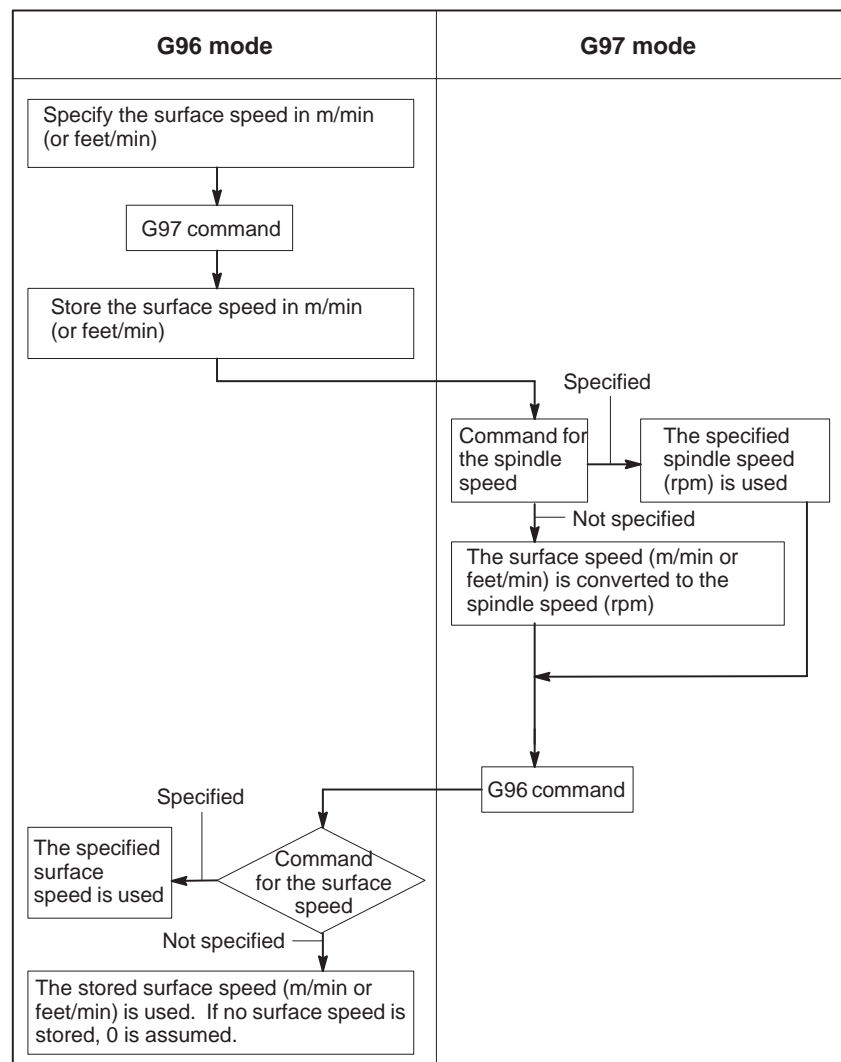


Fig. 9.3 (b) Example of the workpiece coordinate system for constant surface speed control

- **Surface speed specified in the G96 mode**



Restrictions

- **Constant surface speed control for threading**

The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.
- **Constant surface speed control for rapid traverse (G00)**

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

10

TOOL FUNCTION (T FUNCTION)



General

Two tool functions are available. One is the tool selection function, and the other is the tool life management function.

10.1 TOOL SELECTION FUNCTION

By specifying two or four-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:

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

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

10.2

TOOL LIFE
MANAGEMENT
FUNCTION

Tools are classified into various groups, with the tool life (time or frequency of use) for each group being specified. The function of accumulating the tool life of each group in use and selecting and using the next tool previously sequenced in the same group, is called the tool life management function.

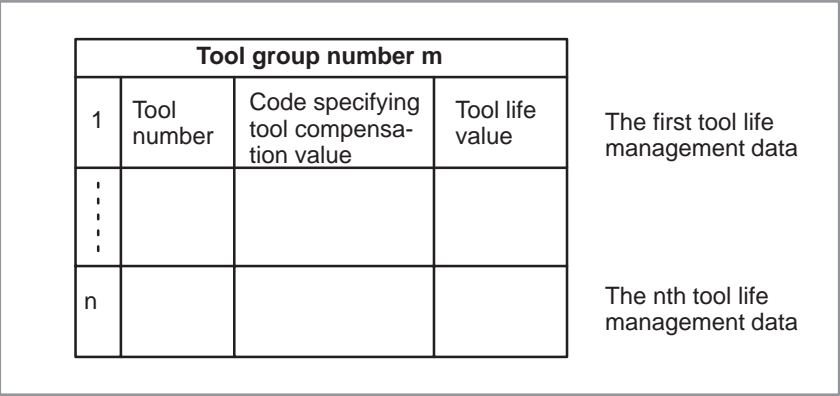


Fig. 10.2(a) Tool life management data (number of n tools)

By choosing a tool from a tool group specified by a machining program, the tool life can be managed.

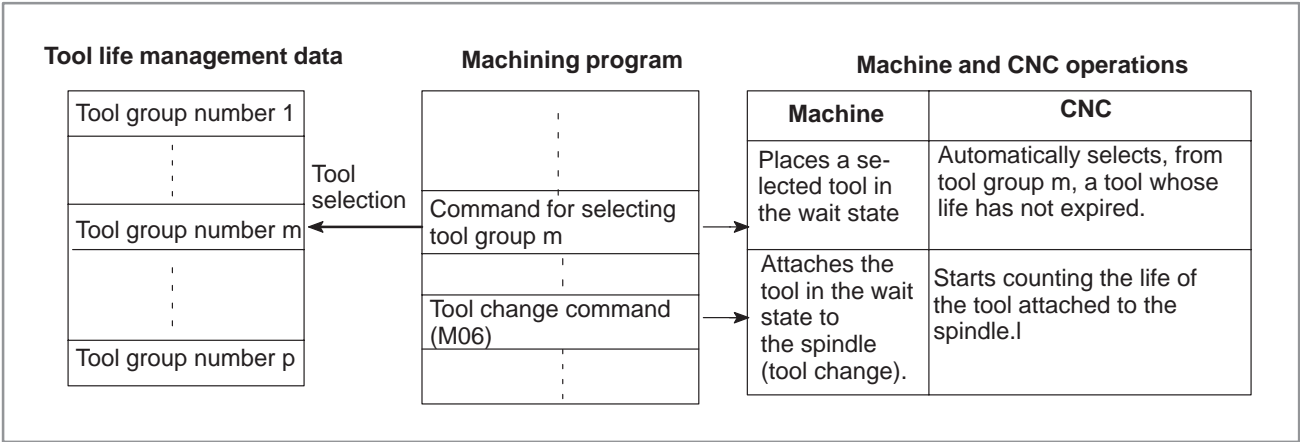


Fig. 10.2(b) Tool Selection by machining program

10.2.1 Tool Life Management Data

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

Explanations

- **Tool group number**

The Max. number of groups and the number of tools per group that can be registered are set by parameter (No. 039#0, #1) (Table 10.2.1).

Table 10.2.1 The max. number of groups and tools that can be registered

The Max. number of groups and tools without optional function of 512 tool pairs		The Max. number of groups and tools with optional function of 512 tool pairs	
Number of group	Number of tool	Number of group	Number of tool
16	16	64	32
32	8	128	16
64	4	256	8
128	2	512	4

NOTE

When bits 0 or 1 of parameter 039 is changed, re-register tool life management data with the G10L3 command (for registering and deleting data for all groups). Otherwise, new data pairs cannot be set.

- **Tool number**

Specify a two or four-digit number after T.

- **Code specifying tool compensation value**

Codes specifying tool offset values are classified into H codes (for tool length offset) and D codes (cutter compensation). The maximum code numbers that can be registered to specify a tool offset value are 200. Code number 99, however, cannot be used.

NOTE

When codes specifying tool offset values are not used, registration can be omitted.

- **Tool life value**

Refer to subsections 10.2.2 and 10.2.4.

10.2.2

Register, Change and Delete 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 or deleted.

Explanations

- **Register with deleting all groups**
After all registered tool life management data is deleted, programmed tool life management data is registered.
- **Deletion of tool life management data**
Programmed tool life management data for a group can be deleted. (MDI operation)
- **Register of tool life count type**
Tool life counters can be preset for each group. (MDI operation)
- **Life value**
Whether tool life is to be indicated by time (minutes) or by frequency, it is set by a parameter (No. 039 #2) .
Maximum value of tool life is as follows.
In case of minute :4300(minutes)
In case of frequency :9999(times)

Format

- **Register with deleting all groups**

Format	Meaning of command
G10L3 ; P_L_ ; T_H_D_ ; T_H_D_ ; P_L_ ; T_H_D_ ; T_H_D_ ; G11 ; M02 (M30) ;	G10L3 : Register with deleting all groups P_ : Group number L_ : Life value T_ : Tool number H_ : Code specifying tool offset value (H code) D_ : Code specifying tool offset value (D code) G11 : End of registration

10.2.3

Tool Life Management Command in a Machining Program

Explanations

- **Command**

The following command is used for tool life management:

T▽▽▽▽ ;————— Specifies a tool group number.

The tool life management function selects, from a specified group, a tool whose life has not expired, and outputs its T code. In ▽▽▽▽, specify a number calculated by adding the tool life management cancel number specified in parameter 599 to a group number. For example, to set tool group 1 when the tool life management cancel number is 100, specify T101;.

NOTE

When ▽▽▽▽ is less than a tool life management cancel number, the T code is treated as an ordinary T code.

M06;————— Terminates life management for the previously used tools, and begins counting the life of the new tools selected with the T code.

WARNING

When an option for specifying multiple M codes is selected, specify this code by itself or as the first M code.

H99;————— Selects the H code of tool life management data for the tool currently being used.

H00;————— Cancels tool length offset

D99;————— Selects the D code of tool life management data for the tool currently being used.

D00;————— Cancels cutter compensation

WARNING

H99 or D99 must be specified after the M06 command. When a code other than H99 or D99 is specified after the M06 command, the H code and D code of tool life management data are not selected.

• Types

For tool life management, the four tool change types indicated below are available. The type used varies from one machine to another. For details, refer to the appropriate manual of each machine tool builder.

Table 10.2.3 Tool change type

Tool change type	A	B	C	D
Tool group number specified in the same block as the tool change command (M06)	Previously used tools	Tools to be used next		
Tool life count timing	Life counting is performed for a tool in the specified tool group when M06 is specified next.			Life counting is performed when a tool in the tool group specified in the same block as M06 is specified.
Remarks		Normally, when a tool group number is specified by itself, type B is used. However, no alarm is raised even if the tool group number is specified by itself as type C.		
Parameter	039#7=0 041#7=0	039#7=1 041#7=0		041#7=1

NOTE

When a tool group number is specified and a new tool is selected, the new tool selection signal is output.

Examples

• Tool change type A

Suppose that the tool life management cancel number is 100.

```

T101;  A tool whose life has not expired is selected from group 1.
      |  (Suppose that tool number 010 is selected.)
M06;   Tool life counting is performed for the tool in group 1.
      |  (The life of tool number 010 is counted.)
T102;  A tool whose life has not expired is selected from group 2.
      |  (Suppose that tool number 100 is selected.)
M06T101; Tool life counting is performed for the tool in group 2.
        |  (The life of tool number 100 is counted.)
        |  The number of the tool currently used (in group 1) is
        |  output with a T code signal. (Tool number 010 is output.)

```

- **Tool change type B and C**

Suppose that the tool life management ignore number is 100.

T1Q1; A tool whose life has not expired is selected from group 1.
 ' (Suppose that tool number 010 is selected.) Note)
 M06T102; Tool life counting is performed for the tool in group 1.
 ' (The life of tool number 010 is counted.)
 ' A tool whose life has not expired is selected from group 2.
 ' (Suppose that toolnumber 100 is selected.)
 M06T103; Tool life counting is lperformed for the tool in group 2.
 ' (The life of tool number 100 is counted.)
 ' A tool whose life has not expired is selected from group 3.
 ' (Suppose that tool number 200 is selected.)

NOTE

Specify M06T101; for tool change method C.

- **Tool change type D**

Suppose that the tool life management ignore number is 100.

T101M06; A tool whose life has not expired is selected from group 1.
 ' (Suppose that tool number 010 is selected.)
 ' Tool life counting is performed for the tool in group 1.
 '
 T102M06; A tool whose life has not expired is selected from group 2.
 ' (Suppose that tool number 100 is selected.)
 ' Tool life counting is performed for the tool in group 2.
 ' (The life of tool number 100 is counted.)

10.2.4 Tool Life

The life of a tool is specified by a usage frequency (count) or usage time (in minutes).

Explanations

- **Usage count**

The usage count is incremented by 1 for each tool used in a program. In other words, the usage count is incremented by 1 only when the first tool group number and tool change command are specified after the CNC unit enters the automatic operation state from the reset state.

CAUTION

Even if the same tool group number is specified more than once in a program, the usage count is only incremented by 1 and no new tools are selected.

- **Usage time**

When a tool change is specified (M06), tool life management is started for the tools specified by the tool group number. In tool life management, the time during which a tool is used in the cutting mode is counted in four second increments. If the tool group is changed before the incremental time of four seconds elapses, the time is not counted. The time a tool is used for single block stop, feed hold, rapid traverse, dwell, machine lock, and interlock is not counted.

NOTE

- 1 When a tool is selected from available tools, tools are searched starting from the current tool towards the lasttool to find a tool whose life has not expired.
- 2 When tool life counting indicates that the life of the last tool in a group has expired, the tool change signal is output. When tool life is managed by time, the signal is output when the life of the last tool in the group has expired. When tool life is managed by usage frequency (count), the signal is output when the CNC unit is reset or the tool life count restart M code is specified.

11

AUXILIARY FUNCTION

There are two types of auxiliary functions ; miscellaneous function (M code) for specifying spindle start, spindle stop program end, and so on, and secondary auxiliary function (B code) for specifying index table positioning.

When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:

- i) Simultaneous execution of the move command and miscellaneous function commands.
- ii) Executing miscellaneous 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.

11.1 AUXILIARY FUNCTION (M FUNCTION)

When a three-digit 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. In some cases, however, up to three M codes can be specified for some types of machine tools.

Which M code corresponds to which machine function is determined by the machine tool builder.

The machine processes all operations specified by M codes except those specified by M98 or M99. Refer to the machine tool builder's instruction manual for details.

Explanations

- **M02,M03**
(End of program)

The following M codes have special meanings.

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.

Bit 5 of parameter 019 (M02) can be used to disable M02 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
sub-program)

This code is used to call a subprogram. The code and strobe signals are not sent. See the subprogram **section 13.3** for details .

- **M99**
(End of subprogram)

This code indicates the end of a subprogram.

M99 execution returns control to the main program. See the subprogram **section 12.3** for details.

- **M198**
(subprogram calling)

This code is used to call a file on the floppy cassette as a subprogram.

NOTE

The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, two M codes which do not buffer can be set by parameters (Nos. 111 to 112). Refer to the machine tool builder's instruction manual for these M codes.

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

So far, one block has been able to contain only one M code. However, this function allows up to three M codes to be contained in one block.

Up to three M codes specified in a block are simultaneously output to the machine. This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining. To use this function, set bit 7 of parameter No. 065 to 1.

Explanations

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.

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.

Examples

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28G91X0Y0Z0 ; : : :	M40M50M60 ; G28G91X0Y0Z0 ; : : : :

11.3 THE SECOND AUXILIARY FUNCTIONS (B CODES)

Indexing of the table is performed by address B and a following 3 or 6-digit number. The relationship between B codes and the corresponding indexing differs between machine tool builders.

Refer to the manual issued by the machine tool builder for details.

Restrictions

When this functions is used, the B address specifying an axis movement disabled.

12 PROGRAM CONFIGURATION

General

- **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.

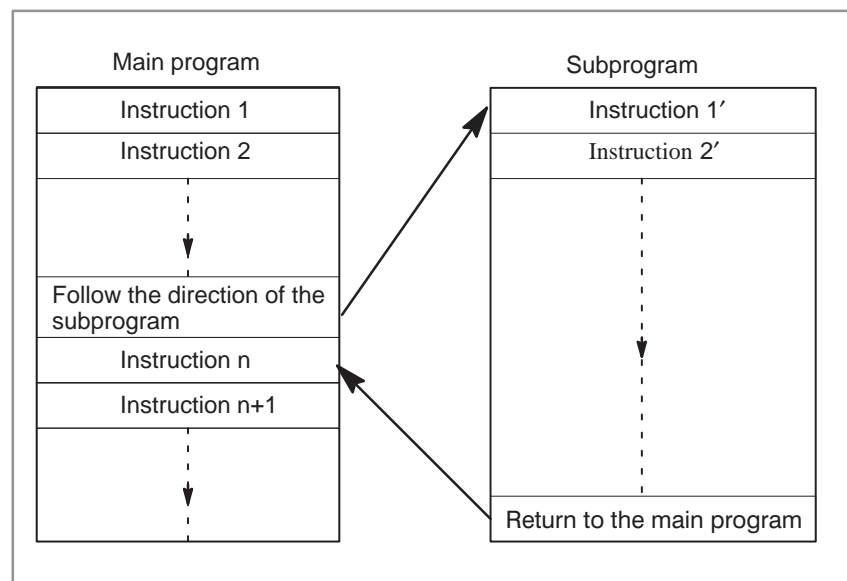


Fig. 12 (a) Main program and subprogram

The CNC memory can hold up to 200 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. See Chapter 10 in OPERATION for the methods of registering and selecting programs.

- **Program components**

A program consists of the following components:

Table 12 Program components

Components	Descriptions
Tape 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
Tape end	Symbol indicating the end of a program file

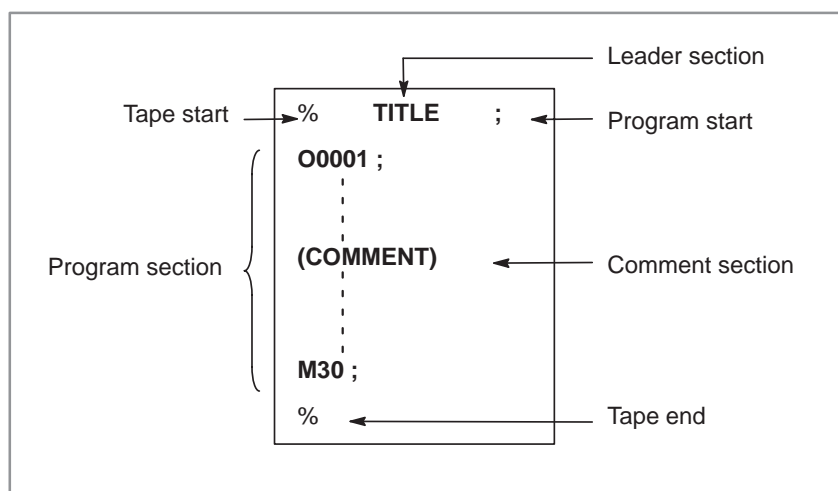


Fig. 12 Program configuration

- **Program section configuration**

A program section consists of several blocks. A program section starts with a program number and ends with a program end code.

Program section configuration

Program number
Block 1
Block 2
:
Block n
Program end

Program section

```
O0001 ;
N1 G91 G00 X120.0 Y80.0 ;
N2 G43 Z-32.0 H01 ;
:
Nn M20 ;
M30 ;
```

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a value following a slash (/) at the start of a block disables the execution of some blocks (see “optional block skip” in Section 12.2).

12.1

PROGRAM
COMPONENTS
OTHER THAN
PROGRAM
SECTIONS

This section describes program components other than program sections. See Section 12.2 for a program section.

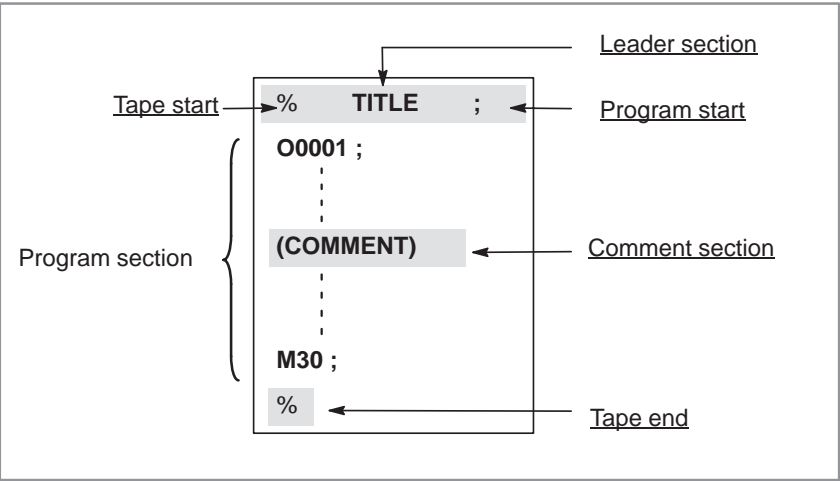


Fig. 12.1 Program configuration

Explanations

• Tape start

The tape start indicates the start of a file that contains NC programs. The mark is not required when programs are entered using SYSTEM P or ordinary personal computers. The mark is not displayed on the CRT display screen. However, if the file is output,the mark is automatically output at the start of the file.

Table 12.1(a) Code of a tape start

Name	ISO code	EIA code	Notation in this manual
Tape 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 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.

Table 12.1(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. However, an program start is required at the start of a program if the preceding program ends with %.

- **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. There is no limit on the length of a comment section.

Table 12.1 (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 command tape 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 9 are ignored, and thus are not read into memory. When data in memory is punched out on paper tape with the punch function, the comment sections are also punched out.

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 punched out or displayed.

During memory operation, all comment sections are ignored.

The TV check function can be used for a comment section by setting parameter (No. 018 #6).

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

If only a control-in code is read with no matching control-out code, the read control-in code is ignored.

- **Tape end**

A tape 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. 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 M03 is not placed at the end of the program, the alarm (No. 008) is occurred.

Table 12.1 (d) Code of a tape end

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

12.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section 12.1 for program components other than program sections.

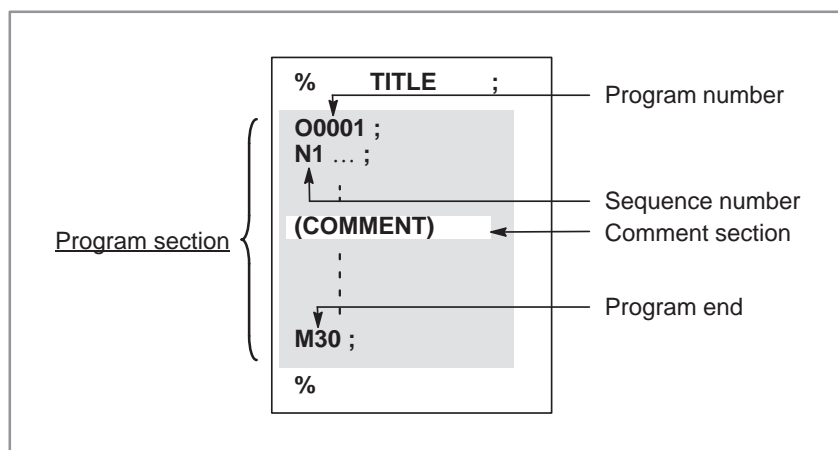


Fig. 12.2 (a) Program configuration

• 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.

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. 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 CRT/MDI panel when the program is stored in memory (See Section 9.3 in Part III.).

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 12.2 (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 four digits (1 to 9999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

N300 X200.0 Z300.0 ; A sequence number is underlined.

Fig. 12.2 (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 along tape)**

A parity check is made for a block on input tape horizontally. 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 (No.002) is output. No TV check is made only for those parts that are skipped by the label skip function. A comment section enclosed in parentheses is also subject to TV check to count the number of characters. The TV check function can be enabled or disabled by setting on the MDI unit (See Subsec. 11.5.3 in Part III.).

- **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.)

Word = Address + number (Example : X–1000)

For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address. Table 12.2 (b) indicates the usable addresses and their meanings.

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

Table 12.2(b) Major functions and addresses

Function	Address	Meaning
Program number	O ⁽¹⁾	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Y, Z, U, V, W, A, B, C	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
	B	Table indexing, etc.
Offset number	D, H	Offset number
Dwell	P, X	Dwell time
Program number designation	P	Subprogram number
Number of repetitions	P	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

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	Spindle speed function	Tool function	Miscellaneous function

Fig. 12.2 (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 federate of up to 100 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 12.2(c) Major addresses and ranges of command values

Function		Address	Input in mm	Input in inch
Program number		O (1)	1–9999	1–9999
Sequence number		N	1–9999	1–9999
Preparatory function		G	0–255	0–255
Dimension word	Increment system IS–B	X, Y, Z, U, V, W, A, B, C, I, J, K, R,	±99999.999mm	±9999.9999inch
	Increment system IS–C		±9999.9999mm	±999.99999inch
Feed per minute	Increment system IS–B	F	1–100000mm/min	0.01–4000 inch/min
	Increment system IS–C		1–24000mm/min	0.01–480.00 inch/min
Feed per revolution		F	0.01–500.00 mm/rev	0.0001–9.9999 inch/rev
Spindle speed function		S	0–20000	0–20000
Tool function		T	0–9999	0–9999
Auxiliary function		M	0–999	0–9999
		B	0–999999	0–999999
Offset number		H, D	0–200	0–200
Dwell	Increment system IS–B	X, P	0–99999.999s	0–99999.999s
	Increment system IS–C		0–9999.9999s	0–9999.9999s
Designation of a program number		P	1–9999	1–9999
Number of repetitions		P	1–999	1–999

- **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 switch n on the machine operator panel is set to on, the information contained in the block for which /n corresponding to switch number n is specified is ignored in tape operation or memory operation.

When optional block skip switch n is set to off, the information contained in the block for which /n is specified is valid. This means that the operator can determine whether to skip the block containing /n.

Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used for one block, number 1 for /1 cannot be omitted.

Example)

(Incorrect)	(Correct)
//3 G00X10.0;	/1/3 G00X10.0;

This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip switch is set.

Programs held in memory can be output, regardless of how the optional block skip switches are set.

Optional block skip is effective even during sequence number search operation.

Depending on the machine tool, all optional block skip switches (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which switches are usable.

WARNING

1 Position of a slash

A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

2 Disabling an optional block skip switch

Optional block skip operation is processed when blocks are read from memory or tape into a buffer. Even if a switch is set to on after blocks are read into a buffer, the blocks already read are not ignored.

NOTE

TV and TH check

When an optional block skip switch is on. TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is off.

- **Program end**

The end of a program is indicated by commanding one of the following codes at the end of the program:

Table 12.2 (d) Code of a program end

Code	Meaning usage
M02	For main program
M30	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

NOTE

A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program, if the optional block skip switch on the machine operator's panel is set to on.
(See item “● Optional block skip”.)

12.3 SUBPROGRAM

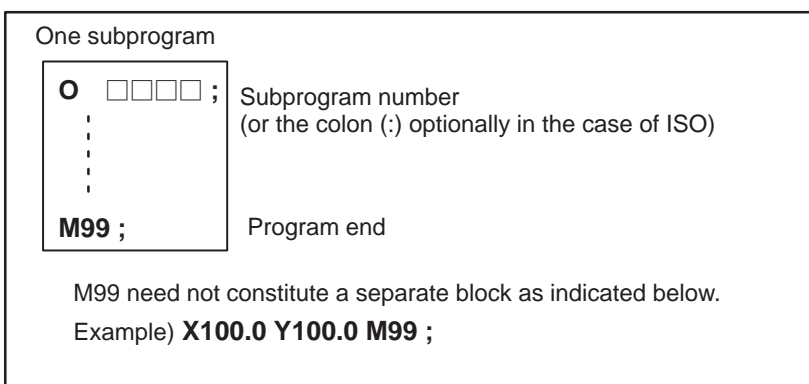
If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.

A subprogram can be called from the main program.

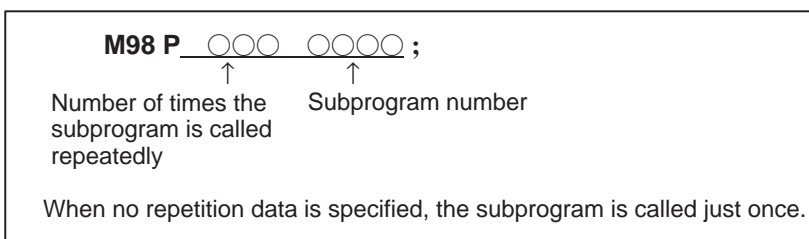
A called subprogram can also call another subprogram.

Format

- Subprogram configuration

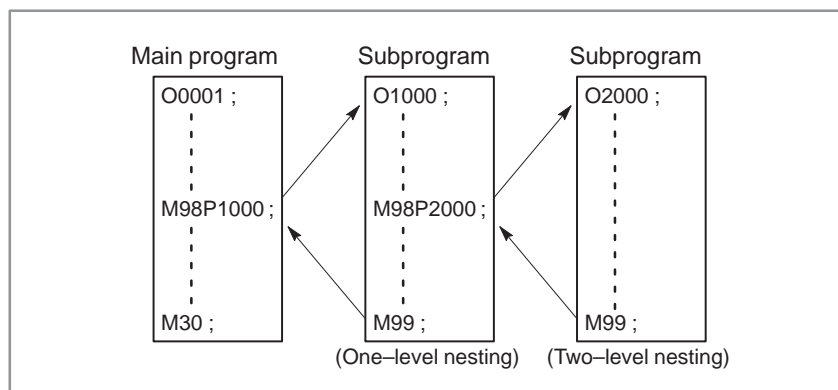


- Subprogram call



Explanations

When the main program calls a subprogram, it is regarded as a one-level subprogram call. Thus, subprogram calls can be nested up to two levels as shown below.



A single call command can repeatedly call a subprogram up to 999 times. For compatibility with automatic programming systems, in the first block, Nxxxx can be used instead of a subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

Reference

See Chapter 10 in Part III for the method of registering a subprogram.

- NOTE
- 1

The M98 and M99 signals are not output to the machine tool.
- 2

If the subprogram number specified by address P cannot be found, an alarm (No. 078) is output.

Examples

☆ **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

Main program

N0010 0 ;

N0020 0 ;

N0030 M98 P21010 ;

N0040 0 ;

N0050 M98 P1010 ;

N0060 0 ;

1 2 3

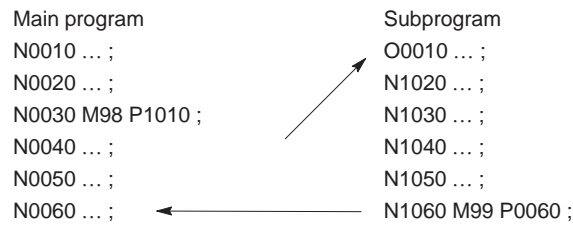
```
graph TD
    subgraph MainProgram [Main program]
        direction TB
        M1[N0010 0 ;]
        M2[N0020 0 ;]
        M3[N0030 M98 P21010 ;]
        M4[N0040 0 ;]
        M5[N0050 M98 P1010 ;]
        M6[N0060 0 ;]
    end
    subgraph Subprograms [Subprogram]
        direction TB
        S1[N0100 0 ;]
        S2[N1020 0 ;]
        S3[N1030 0 ;]
        S4[N1040 0 ;]
        S5[N1050 0 ;]
        S6[N1060 0 M99 ;]
    end
    M3 --> S1
    S1 --> S2
    S2 --> S3
    S3 --> M6
    S2 --> M6
    S1 --> M4
    M5 --> S4
    S4 --> S5
    S5 --> S6
    S6 --> M6
```

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. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode. This method consumes a longer time than the normal return method to return to the main program.

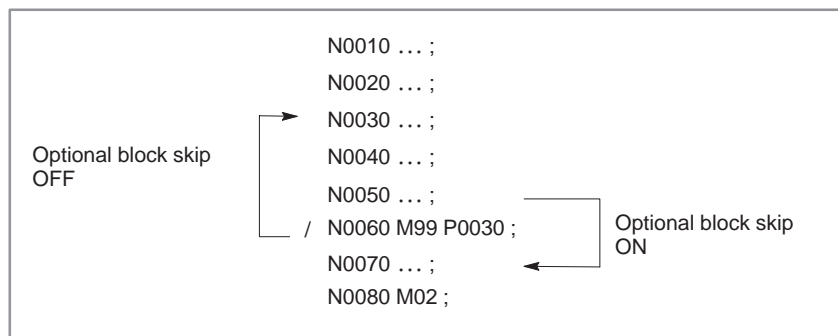


- **Using M99 in the main program**

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.

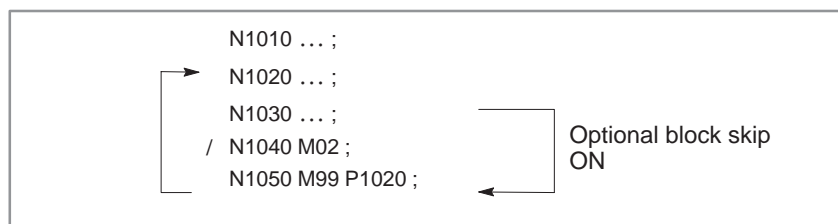


- **Using a subprogram only**

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

(See Section 9.4 in Part III 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 skip must be set to off ; this switch is to be set to on first.



13

FUNCTIONS TO SIMPLIFY PROGRAMMING

General

This chapter explains the following items:

- 13.1 CANNED CYCLE
- 13.2 RIGID TAPPING
- 13.3 CANNED GRINDING CYCLE (0-GSC, 0-GSD/II)
- 13.4 GRINDING-WHEEL WEAR COMPENSATION BY
CONTINUOUS DRESSING (0-GSC, 0-GSD/II)
- 13.5 AUTOMATIC GRINDING WHEEL DIAMETER
COMPENSATION AFTER DRESSING
- 13.6 IN-FEED GRINDING ALONG THE Y AND Z AXES AT THE
END OF TABLE SWING (0-GSC, 0-GSD/II)
- 13.7 OPTIONAL ANGLE CHAMFERING AND CORNER
ROUNDING
- 13.8 EXTERNAL MOTION FUNCTION (G81)
- 13.9 SMALL-DIAMETER PECK DRILLING CYCLE

13.1 CANNED CYCLE

Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently-used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.

Table 13.1 (a) lists canned cycles.

Table 13.1 (a) Canned cycles

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

Explanations

A canned cycle consists of a sequence of six operations (Fig. 13.1 (a))

- Operation 1 -----Positioning of axes X and Y
(including also another axis)
- Operation 2 -----Rapid traverse up to point R level
- Operation 3 -----Hole machining
- Operation 4 -----Operation at the bottom of a hole
- Operation 5 -----Retraction to point R level
- Operation 6 -----Rapid traverse up to the initial point

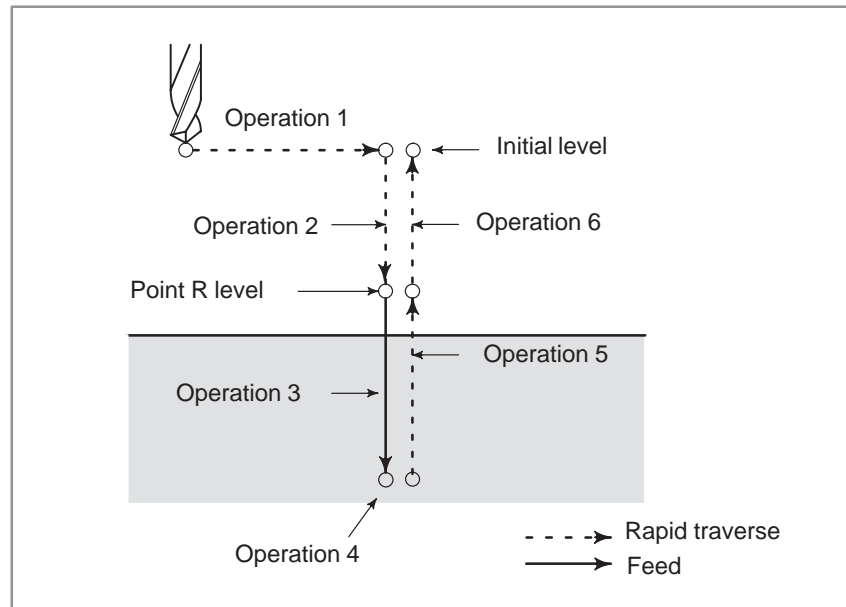


Fig. 13.1 Canned cycle operation sequence

- **Positioning plane**

The positioning plane is determined by plane selection code G17, G18, or G19.

The positioning axis is an axis other than the drilling axis.

- **Drilling axis**

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.

The drilling axis is a basic axis (X, Y, or Z) not used to define the positioning plane, or any axis parallel to that basic axis.

Table 13.1 (b) Positioning plane and drilling axis

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

G17 to G19 may be specified in a block in which any of G73 to G89 is not specified.

CAUTION

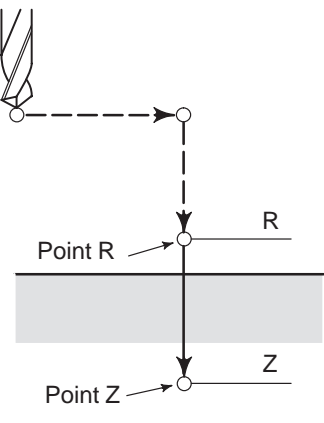
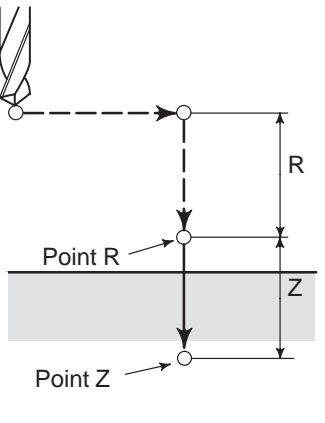
Switch the drilling axis after canceling a canned cycle.

NOTE

A parameter (No. 057 #6) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

● **Travel distance along the drilling axis G90/G91**

The travel distance along the drilling axis varies for G90 and G91 as follows:

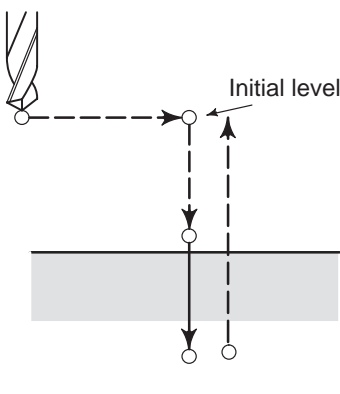
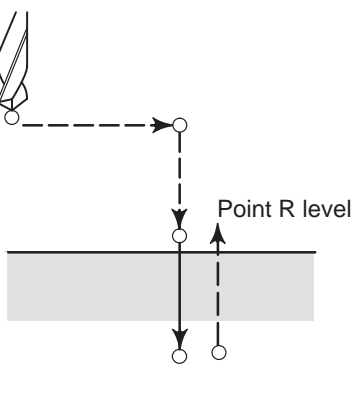
G90 (Absolute Command)	G91 (Incremental Command)
 The diagram shows a drill bit moving from an initial level to Point R, then down to Point Z. The distance from the initial level to Point R is labeled 'R', and the distance from Point R to Point Z is labeled 'Z'. The workpiece is represented by a shaded gray area below Point R.	 The diagram shows a drill bit moving from an initial level to Point R, then down to Point Z. The distance from the initial level to Point R is labeled 'R', and the distance from Point R to Point Z is labeled 'Z'. The workpiece is represented by a shaded gray area below Point R.

● **Drilling mode**

G73, G74, G76, and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode. Once drilling data is specified in the drilling mode, the data is retained until modified or canceled. Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

● **Return point level G98/G99**

When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation. The initial level does not change even when drilling is performed in the G99 mode.

G98(Return to initial level)	G99(Return to point R level)
 The diagram shows a drill bit moving from an initial level to Point R, then down to Point Z. After drilling, the tool returns to the initial level. The distance from the initial level to Point R is labeled 'Initial level'. The workpiece is represented by a shaded gray area below Point R.	 The diagram shows a drill bit moving from an initial level to Point R, then down to Point Z. After drilling, the tool returns to Point R level. The distance from the initial level to Point R is labeled 'Point R level'. The workpiece is represented by a shaded gray area below Point R.

- **Repeat**

To repeat drilling for equally-spaced holes, specify the number of repeats in K_.

K is effective only within the block where it is specified.

Specify the first hole position in incremental mode (G91).

If it is specified in absolute mode (G90), drilling is repeated at the same position.

Number of repeats K The maximum command value = 9999

If K0 is specified, drilling data is stored, but drilling is not performed.

- **Cancel**

To cancel a canned cycle, use G80 or a group 01 G code.

Group 01 G codes

G00 : Positioning (rapid traverse)






G01 : Linear interpolation

G02 : Circular interpolation or helical interpolation (CW)

G03 : Circular interpolation or helical interpolation (CCW)

- **Symbols in figures**

Subsequent sections explain the individual canned cycles. Figures in these explanations use the following symbols:

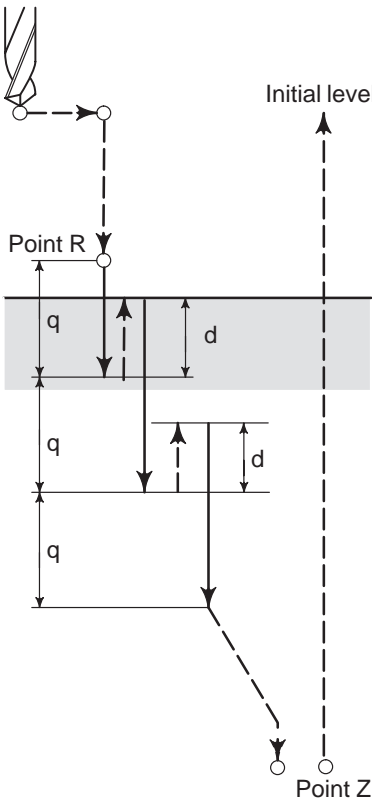
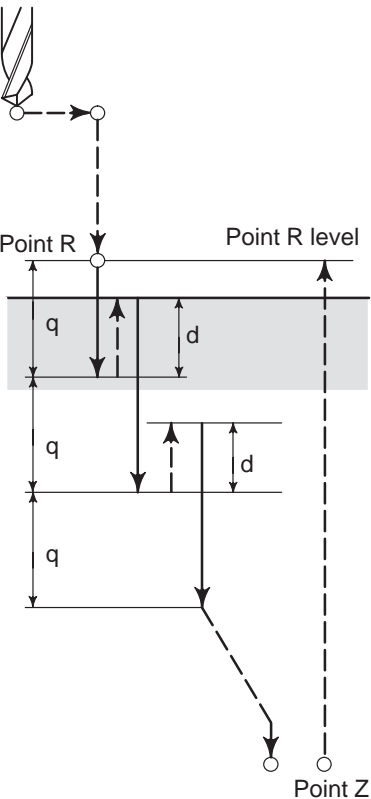
	Positioning (rapid traverse G00)
	Cutting feed (linear interpolation G01)
	Manual feed
	Spindle orientation (The spindle stops at a fixed rotation position)
	Shift (rapid traverse G00)
P	Dwell

13.1.1

High-speed Peck Drilling Cycle (G73)

This cycle performs high-speed peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

Format

<div>G73 X_ Y_ Z_ R_ Q_ F_ K_ ; <div>X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level Q_ : Depth of cut for each cutting feed F_ : Cutting feedrate K_ : Number of repeats</div></div>	
G73 (G98)	G73 (G99)
 <p>The diagram illustrates the G73 (G98) cycle. A drill bit starts at an 'Initial level' (indicated by a dashed line). It moves down to 'Point R' (indicated by a solid line). From Point R, it performs a series of peck drilling operations, each with a depth 'q' and a retract distance 'd'. The cycle repeats until it reaches 'Point Z' (indicated by a dashed line). The final position is 'Point Z'.</p>	 <p>The diagram illustrates the G73 (G99) cycle. A drill bit starts at an 'Initial level' (indicated by a dashed line). It moves down to 'Point R' (indicated by a solid line). From Point R, it performs a series of peck drilling operations, each with a depth 'q' and a retract distance 'd'. The cycle repeats until it reaches 'Point Z' (indicated by a dashed line). The final position is 'Point Z'.</p>

Explanations

The high-speed peck drilling cycle performs intermittent feeding along the Z-axis. When this cycle is used, chips can be removed from the hole easily, and a smaller value can be set for retraction. This allows, drilling to be performed efficiently. Set the clearance, d, in parameter 531.

The tool is retracted in rapid traverse.

Before specifying G73, rotate the spindle using a miscellaneous function (M code).

When the G73 code and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **Q/R** Specify Q and R in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

```

M3 S2000 ;   Cause the spindle to start rotating.
G90 G99 G73 X300. Y-250. Z-150. R-100. Q15. F120. ;
                                     Position, drill hole 1, then return to point R.
Y-550. ;      Position, drill hole 2, then return to point R.
Y-750. ;      Position, drill hole 3, then return to point R.
X1000. ;      Position, drill hole 4, then return to point R.
Y-550. ;      Position, drill hole 5, then return to point R.
G98 Y-750. ;   Position, drill hole 6, then return to the ini-tial level.
G80 G28 G91 X0 Y0 Z0 ; Return to the reference position return
M5 ;          Cause the spindle to stop rotating.

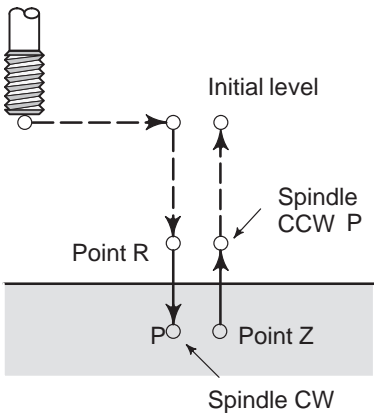
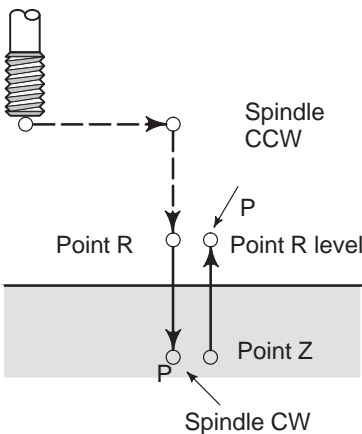
```

13.1.2

Left-handed Tapping Cycle (G74)

This cycle performs left-handed tapping. In the left-handed tapping cycle, when the bottom of the hole has been reached, the spindle rotates clockwise.

Format

<div>G74 X_ Y_ Z_ R_ P_ F_ K_ ; <div>X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level P_ : Dwell time F_ : Cutting feedrate K_ : Number of repeats</div></div>	
G74 (G98)	G74 (G99)
	

Explanations

Tapping is performed by turning the spindle counterclockwise. When the bottom of the hole has been reached, the spindle is rotated clockwise for retraction. This creates a reverse thread.

Feedrate overrides are ignored during left-handed tapping. A feed hold does not stop the machine until the return operation is completed.

Before specifying G74, use a miscellaneous function (M code) to rotate the spindle counterclockwise.

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G74 in the same block. If they are specified together, G74 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

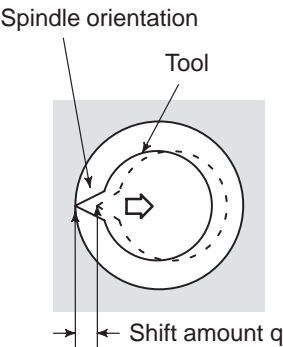
Examples

M4 S100 ;	Cause the spindle to start rotating.
G90 G99 G74 X300. Y-250. Z-150. R-120. F120. ;	
Y-550. ;	Position, tapping hole 1, then return to point R.
Y-750. ;	Position, tapping hole 2, then return to point R.
X1000. ;	Position, tapping hole 3, then return to point R.
Y-550. ;	Position, tapping hole 4, then return to point R.
G98 Y-750. ;	Position, tapping hole 5, then return to point R.
	initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.3
Fine Boring Cycle
(G76)

Format

The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.



G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;	
X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level Q_ : Shift amount at the bottom of a hole P_ : Dwell time at the bottom of a hole F_ : Cutting feedrate K_ : Number of repeats	
G76 (G98)	G76 (G99)

WARNING
Q (shift at the bottom of a hole) is a modal value retained within canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

Explanations

When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring to be performed.

Before specifying G76, use a miscellaneous function (M code) to rotate the spindle.

When the G76 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled.

- **Boring**

In a block that does not contain X, Y, Z, R, or any additional axes, boring is not performed.

- **Q/R**

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in bits 4 and 5 of parameter 002. Specify Q and R in a block that performs boring. If they are specified in a block that does not perform boring, they are not stored as modal data.

- **Cancel**

Do not specify a group 01 G code (G00 to G03) and G76 in the same block. If they are specified together, G76 is canceled.

- **Tool offset**

In the canned cycle mode, tool offsets are ignored.

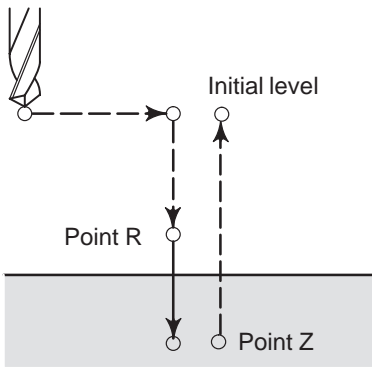
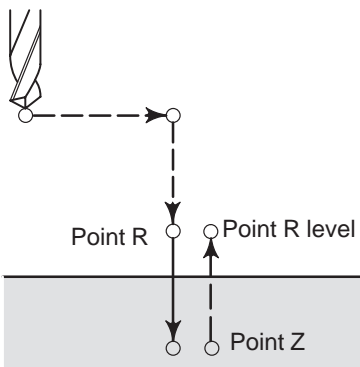
Examples

M3 S500 ;	Cause the spindle to start rotating.
G90 G99 G76 X300. Y-250. Z-150. R-120. Q5. P1000 F120. ;	Position, bore hole 1, then return to point R. Orient at the bottom of the hole, then shift by 5 mm. Stop at the bottom of the hole for 1 s.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.4
Drilling Cycle, Spot
Drilling (G81)

This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

<div>G81 X_ Y_ Z_ R_ F_ K_ ; X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level F_ : Cutting feedrate K_ : Number of repeats</div>	
G81 (G98)	G81 (G99)
	

Explanations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
The tool is then retracted in rapid traverse.
Before specifying G81, use a miscellaneous function (M code) to rotate the spindle.
When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G81 in the same block. If they are specified together, G81 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

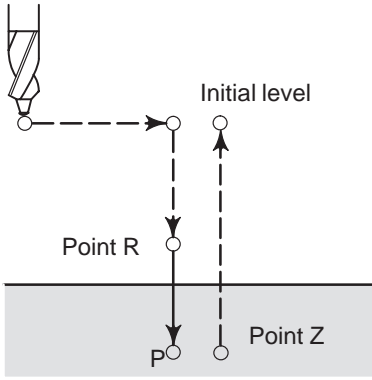
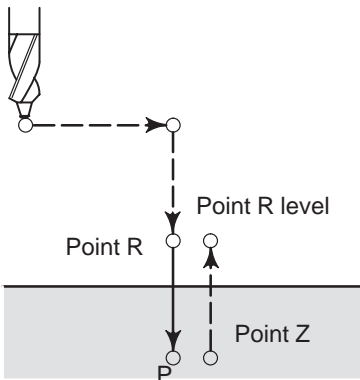
M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G81 X300. Y-250. Z-150. R-100. F120. ;	
Y-550. ;	Position, drill hole 1, then return to point R.
Y-750. ;	Position, drill hole 2, then return to point R.
X1000. ;	Position, drill hole 3, then return to point R.
Y-550. ;	Position, drill hole 4, then return to point R.
G98 Y-750. ;	Position, drill hole 5, then return to point R.
	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.5

Drilling Cycle Counter Boring Cycle (G82)

This cycle is used for normal drilling.
Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse.
This cycle is used to drill holes more accurately with respect to depth.

Format

<div>G82 X_ Y_ Z_ R_ P_ F_ K_ ;</div> <div>X_ Y_ : Hole position data</div> <div>Z_ : The distance from point R to the bottom of the hole</div> <div>R_ : The distance from the initial level to point R level</div> <div>P_ : Dwell time at the bottom of a hole</div> <div>F_ : Cutting feed rate</div> <div>K_ : Number of repeats</div>	
G82 (G98)	G82 (G99)
	

Explanations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is then performed from point R to point Z.
When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.
Before specifying G82, use a miscellaneous function (M code) to rotate the spindle.
When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G82 in the same block. If they are specified together, G82 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

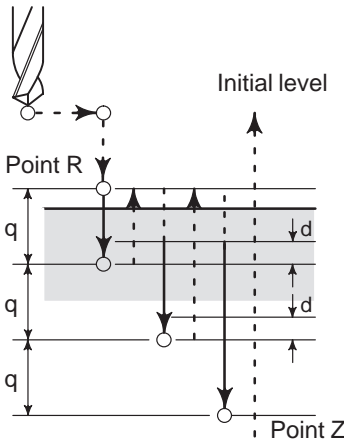
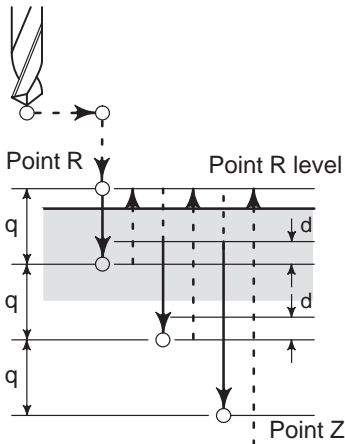
Examples

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120. ;	
	Position, drill hole 2, and dwell for 1 s at the bottom of the hole, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.6
Peck Drilling Cycle
(G83)

This cycle performs peck drilling.
It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Format

<div>G83 X_ Y_ Z_ R_ Q_ F_ K_ ; X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level Q_ : Depth of cut for each cutting feed F_ : Cutting feedrate K_ : Number of repeats</div>	
G83 (G98)	G83 (G99)
	

Explanations

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value.

In the second and subsequent cutting feeds, rapid traverse is performed up to a point just before where the last drilling ended, and cutting feed is performed again. Specify the amount of retraction in parameter No. 532. Be sure to specify a positive value in Q. Negative values are ignored.

Before specifying G83, use a miscellaneous function (M code) to rotate the spindle.

When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **Q/R** Specify Q and R in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G83 in the same block. If they are specified together, G83 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G83 X300. Y-250. Z-150. R-100. Q15. F120. ;	
Y-550. ;	Position, drill hole 1, then return to point R.
Y-750. ;	Position, drill hole 2, then return to point R.
X1000. ;	Position, drill hole 3, then return to point R.
Y-550. ;	Position, drill hole 4, then return to point R.
G98 Y-750. ;	Position, drill hole 5, then return to point R.
	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.7

Tapping Cycle (G84)

This cycle performs tapping.

In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format

<div>G84 X_ Y_ Z_ R_ P_ F_ K_ ;</div> <div>X_ Y_ : Hole position data</div> <div>Z_ : The distance from point R to the bottom of the hole</div> <div>R_ : The distance from the initial level to point R level</div> <div>P_ : Dwell time</div> <div>F_ : Cutting feedrate</div> <div>K_ : Number of repeats</div>	
G84 (G98)	G84 (G99)

Explanations

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is completed.

Before specifying G84, use a miscellaneous function (M code) to rotate the spindle.

When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When the K is used to specify number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G84 in the same block. If they are specified together, G84 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

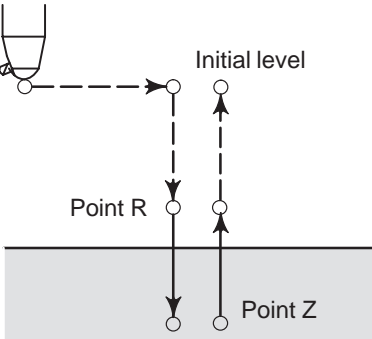
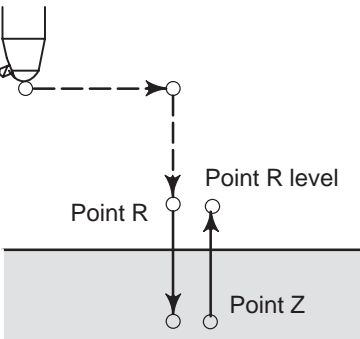
M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G84 X300. Y-250. Z-150. R-120. P300 F120. ;	
Y-550. ;	Position, drill hole 1, then return to point R.
Y-750. ;	Position, drill hole 2, then return to point R.
X1000. ;	Position, drill hole 3, then return to point R.
Y-550. ;	Position, drill hole 4, then return to point R.
G98 Y-750. ;	Position, drill hole 5, then return to point R.
	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.8

Boring Cycle (G85)

This cycle is used to bore a hole.

Format

<div>G85 X_ Y_ Z_ R_ F_ K_ ; X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level F_ : Cutting feed rate K_ : Number of repeats</div>	
G85 (G98)	G85 (G99)
	

Explanations

After positioning along the X- and Y- axes, rapid traverse is performed to point R.

Drilling is performed from point R to point Z.

When point Z has been reached, cutting feed is performed to return to point R.

Before specifying G85, use a miscellaneous function (M code) to rotate the spindle.

When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G85 in the same block. If they are specified together, G85 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G85 X300. Y-250. Z-150. R-120. F120. ;	
Y-550. ;	Position, drill hole 1, then return to point R.
Y-750. ;	Position, drill hole 2, then return to point R.
X1000. ;	Position, drill hole 3, then return to point R.
Y-550. ;	Position, drill hole 4, then return to point R.
G98 Y-750. ;	Position, drill hole 5, then return to point R.
	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.9

Boring Cycle (G86)

This cycle is used to bore a hole.

Format

<div>G86 X_ Y_ Z_ R_ F_ K_ ; <div>X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level F_ : Cutting feed rate K_ : Number of repeats</div></div>	
G86 (G98)	G86 (G99)

Explanations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.

Drilling is performed from point R to point Z.

When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.

Before specifying G86, use a miscellaneous function (M code) to rotate the spindle.

When the G86 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation.

The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G86 in the same block. If they are specified together, G86 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G86 X300. Y-250. Z-150. R-100. F120. ;	
Y-550. ;	Position, drill hole 1, then return to point R.
Y-750. ;	Position, drill hole 2, then return to point R.
X1000. ;	Position, drill hole 3, then return to point R.
Y-550. ;	Position, drill hole 4, then return to point R.
G98 Y-750. ;	Position, drill hole 5, then return to point R.
	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.10 Boring Cycle Back Boring Cycle (G87)

This cycle performs accurate boring.

Format

G87 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_ Y_ : Hole position data

Z_ : The distance from the bottom of the hole to point Z

R_ : The distance from the initial level to point R
(the bottom of the hole) level

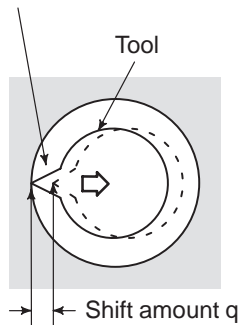
Q_ : Tool shift amount

P_ : Dwell time

F_ : Cutting feed rate

K_ : Number of repeats

Spindle orientation



G87 (G98)	G87 (G99)
	Not used

WARNING

Q (shift at the bottom of a hole) is a modal value retained in canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

Explanations

After positioning along the X- and Y-axes, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tool tip, positioning (rapid traverse) is performed to the bottom of the hole (point R).

The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise. Boring is performed in the positive direction along the Z-axis until point Z is reached.

At point Z, the spindle is stopped at the fixed rotation position again, the tool is shifted in the direction opposite to the tool tip, then the tool is returned to the initial level. The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise to proceed to the next block operation.

Before specifying G87, use a miscellaneous function (M code) to rotate the spindle.

When the G87 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation.

The system then proceeds to the next drilling operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Boring** In a block that does not contain X, Y, Z, R, or any additional axes, boring is not performed.
- **Q/R** Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in bits 4 and 5 of parameter 002. Specify Q and R in a block that performs boring. If they are specified in a block that does not perform boring, they are not stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G76 in the same block. If they are specified together, G76 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

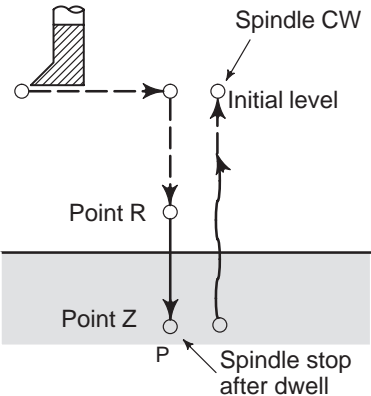
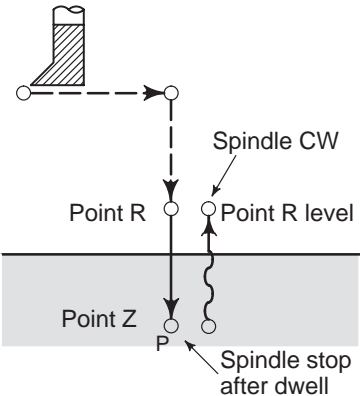
M3 S500 ;	Cause the spindle to start rotating.
G90 G87 X300. Y-250.	Position, bore hole 1.
Z120. R-150. Q5.	Orient at the initial level, then shift by 5 mm.
P1000 F120. ;	Stop at point Z for 1 s.
Y-550. ;	Position, drill hole 2.
Y-750. ;	Position, drill hole 3.
X1000. ;	Position, drill hole 4.
Y-550. ;	Position, drill hole 5.
Y-750. ;	Position, drill hole 6
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.11

Boring Cycle (G88)

This cycle is used to bore a hole.

Format

<div>G88 X_ Y_ Z_ R_ P_ F_ K_ ;</div> <div>X_ Y_ : Hole position data</div> <div>Z_ : The distance from point R to the bottom of the hole</div> <div>R_ : The distance from the initial level to point R level</div> <div>P_ : Dwell time at the bottom of a hole</div> <div>F_ : Cutting feed rate</div> <div>K_ : Number of repeats</div>	
G88 (G98)	G88 (G99)
	

Explanations

After positioning along the X- and Y-axes, rapid traverse is performed to point R. Boring is performed from point R to point Z. When boring is completed, a dwell is performed, then the spindle is stopped. The tool is manually retracted from the bottom of the hole (point Z) to point R. At point R, the spindle is rotated clockwise, and rapid traverse is performed to the initial level.

Before specifying G88, use a miscellaneous function (M code) to rotate the spindle.

When the G88 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G88 in the same block. If they are specified together, G88 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

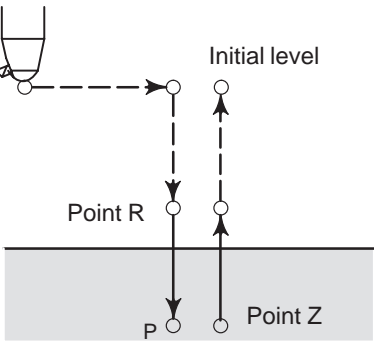
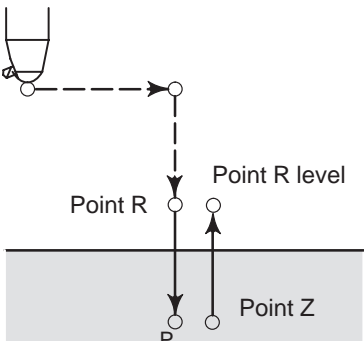
M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G88 X300. Y-250. Z-150. R-100. P1000 F120. ;	Position, drill hole 1, return to point R then stop at the bottom of the hole for 1 s.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.12

Boring Cycle (G89)

This cycle is used to bore a hole.

Format

<div>G89 X_ Y_ Z_ R_ P_ F_ K_ ;</div> <div>X_ Y_ : Hole position data</div> <div>Z_ : The distance from point R to the bottom of the hole</div> <div>R_ : The distance from the initial level to point R level</div> <div>P_ : Dwell time at the bottom of a hole</div> <div>F_ : Cutting feed rate</div> <div>K_ : Number of repeats</div>	
G89 (G98)	G89 (G99)
	

Explanations

This cycle is almost the same as G85. The difference is that this cycle performs a dwell at the bottom of the hole.

Before specifying G89, use a miscellaneous function (M code) to rotate the spindle.

When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching** Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling** In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R** Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel** Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.
- **Tool offset** In the canned cycle mode, tool offsets are ignored.

Examples

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G89 X300. Y-250. Z-150. R-120. P1000 F120. ;	Position, drill hole 1, return to point R then stop at the bottom of the hole for 1 s.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.13 Canned Cycle Cancel (G80)

G80 cancels canned cycles.

Format

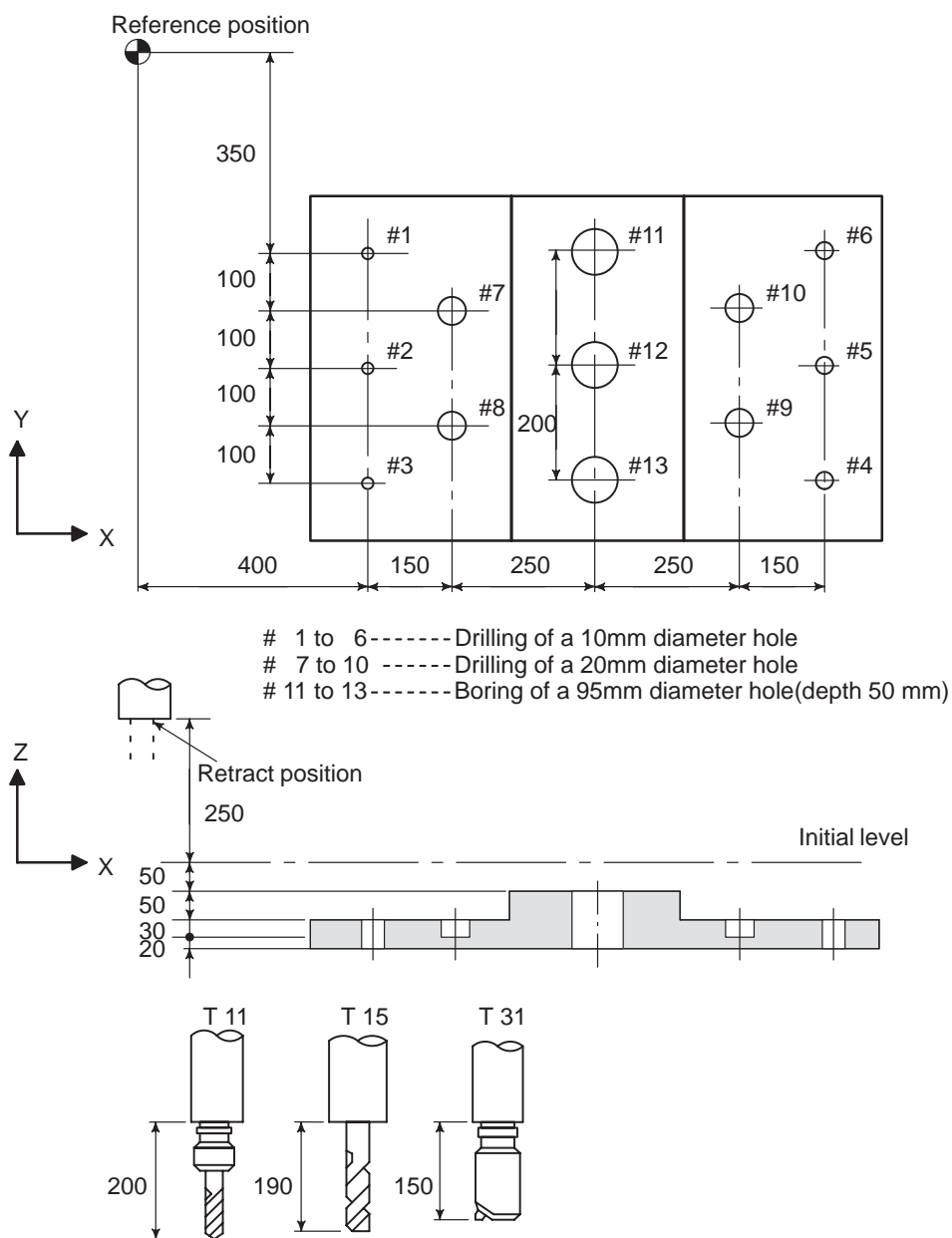
G80 ;

Explanations

All canned cycles are canceled to perform normal operation. Point R and point Z are cleared. This means that R = 0 and Z = 0 in incremental mode. Other drilling data is also canceled (cleared).

Examples

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G88 X300. Y-250. Z-150. R-120. F120. ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return, canned cycle cancel
M5 ;	Cause the spindle to stop rotating.

Program example using tool length offset and canned cycles

Offset value +200.0 is set in offset No.11, +190.0 is set in offset No.15, and +150.0 is set in offset No.31

Program example

;	
N001 G92X0Y0Z500.0;	Coordinate setting at reference position
N002 G90 G00 Z250.0 T11 M6;	Tool change
N003 G43 Z0 H11;	Initial level, tool length offset
N004 S30 M3	Spindle start
N005 G99 G81X400.0 R Y-350.0 Z-153.0R-97.0 F120;	Positioning, then #1 drilling
N006 Y-550.0;	Positioning, then #2 drilling and point R level return
N007 G98Y-750.0;	Positioning, then #3 drilling and initial level return
N008 G99X1200.0;	Positioning, then #4 drilling and point R level return
N009 Y-550.0;	Positioning, then #5 drilling and point R level return
N010 G98Y-350.0;	Positioning, then #6 drilling and initial level return
N011 G00X0Y0M5;	Reference position return, spindle stop
N012 G49Z250.0T15M6;	Tool length offset cancel, tool change
N013 G43Z0H15;	Initial level, tool length offset
N014 S20M3;	Spindle start
N015 G99G82X550.0Y-450.0 Z-130.0R-97.0P300F70;	Positioning, then #7 drilling, point R level return
N016 G98Y-650.0;	Positioning, then #8 drilling, initial level return
N017 G99X1050.0;	Positioning, then #9 drilling, point R level return
N018 G98Y-450.0;	Positioning, then #10 drilling, initial level return
N019 G00X0Y0M5;	Reference position return, spindle stop
N020 G49Z250.0T31M6;	Tool length offset cancel, tool change
N021 G43Z0H31;	Initial level, tool length offset
N022 S10M3;	Spindle start
N023 G85G99X800.0Y-350.0 Z-153.0R47.0F50;	Positioning, then #11 drilling, point R level return
N024 G91Y-200.0K2;	Positioning, then #12, 13 drilling. point R level return
N025 G28X0Y0M5;	Reference position return, spindle stop
N026 G49Z500.0;	Tool length offset cancel
N027 M0;	Program stop

13.2

RIGID TAPPING

The tapping cycle (G84) and left-handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode.

In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping. In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle.

When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.

Rigid mode eliminates the need to use a floating tap required in the standard tapping mode, thus allowing faster and more precise tapping.

13.2.1 Rigid Tapping (G84)

When the spindle motor is controlled in rigid mode as if it were a servo motor, a tapping cycle can be sped up.

Format

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_ : Hole position data

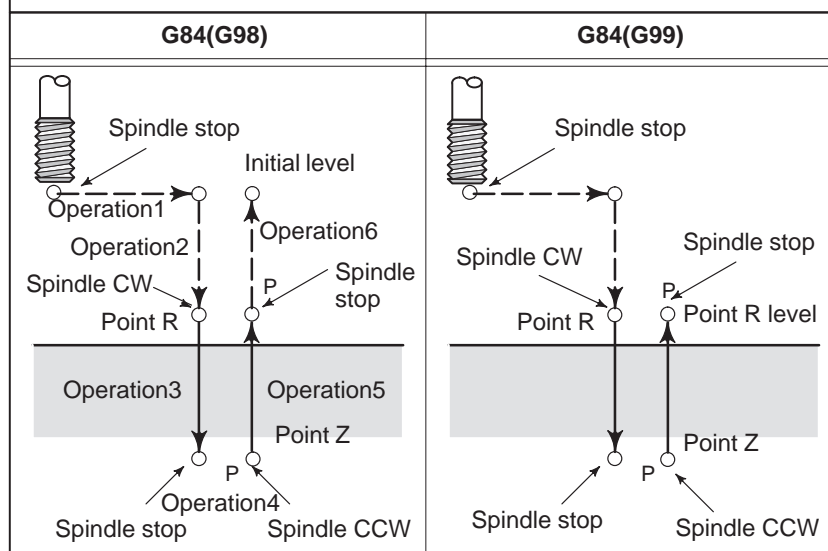
Z_ : The distance from point R to the bottom of the hole or the position of the bottom of the hole

R_ : The distance from the initial level to point R level

P_ : Dwell time at the bottom of the hole and at point R when a return is made

F_ : Cutting feedrate

K_ : Number of repeats



Explanations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.

Tapping is performed from point R to point Z. When tapping is completed, a dwell is performed and the spindle is stopped. The spindle is then rotated in the reverse direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed. While tapping is being performed, the feedrate override and spindle override are assumed to be 100%.

However, the speed for extraction (operation 5) can be overridden by up to 200% depending on the setting at bit 4 of parameter and parameter 063 and parameter 258.

• Rigid mode

Rigid mode can be specified using any of the following methods:

- **Specify M29 S ***** before a tapping command.**
- **Specify M29 S ***** in a block which contains a tapping command.**
- **Specify G84 for rigid tapping (parameter No. 076 #3 = 1).**

- **Thread lead**

In feed-per-minute mode, the thread lead is obtained from the expression, feedrate \times spindle speed. In feed-per-revolution mode, the thread lead equals the feedrate speed.

If a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm (No. 206) is issued.

- **S command**

If a speed higher than the maximum speed for the gear being used is specified, alarm (No. 200) is issued.

- **F command**

If a value exceeding the upper limit of cutting feedrate is specified, alarm (No. 011) is issued.

- **M29**

If an S command and axis movement are specified between M29 and G84, alarm (No. 203) is issued. If M29 is specified in a tapping cycle, alarm (No. 204) is issued.

- **R**

Specify R in a block that performs drilling. If R is specified in a non-drilling block, it is not stored as modal data.

- **Cancel**

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

- **Tool offset**

In the canned cycle mode, tool offsets are ignored.

- **Unit of F**

	Metric input	Inch input	Remarks
G94	1 mm/min	0.01 inch/min	Decimal point programming allowed
G95	0.01 mm/rev	0.0001 inch/rev	Decimal point programming allowed

Examples

Z-axis feedrate 1000 mm/min

Spindle speed 1000 rpm

Thread lead 1.0 mm

<Programming of feed per minute>

G94 ; Specify a feed-per-minute command.

G00 X120.0 Y100.0 ; Positioning

M29 S1000 ; Rigid mode specification

G84 Z-100.0 R-20.0 F1000 ; Rigid tapping

<Programming of feed per revolution>

G95 ; Specify a feed-per-revolution command.

G00 X120.0 Y100.0 ; Positioning

M29 S1000 ; Rigid mode specification

G84 Z-100.0 R-20.0 F1.0 ; Rigid tapping

13.2.2 Left-handed Rigid Tapping Cycle (G74)

Format

When the spindle motor is controlled in rigid mode as if it were a servo motor, tapping cycles can be sped up.

G74 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_ : Hole position data

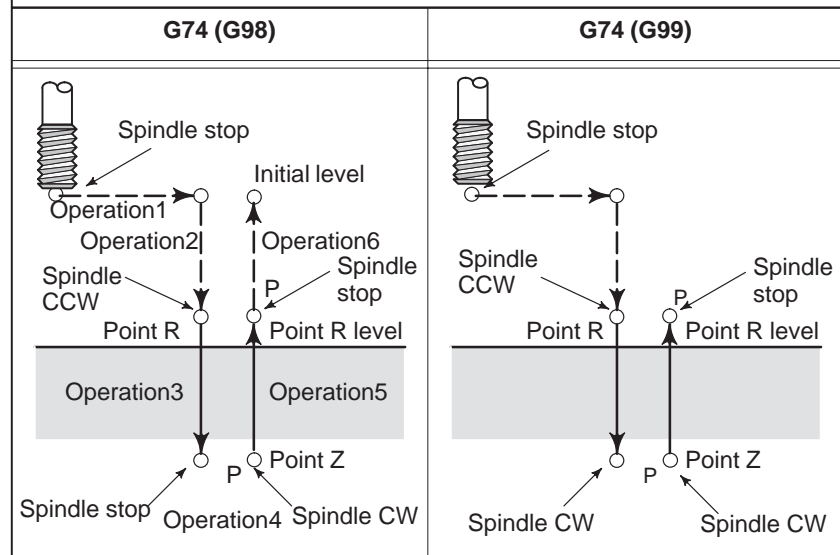
Z_ : The distance from point R to the bottom of the hole or the position of the bottom of the hole

R_ : The distance from the initial level to point R level

P_ : Dwell time at the bottom of the hole and at point R when return is made.

F_ : Cutting feedrate

K_ : Number of repeats



Explanations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.

Tapping is performed from point R to point Z. When tapping is completed, a dwell is performed and the spindle is stopped. The spindle is then rotated in the normal direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed. While tapping is being performed, the feedrate override and spindle override are assumed to be 100%.

However, the speed for extraction (operation 5) can be overridden by up to 200% depending on the setting at bit 4 of parameter 063 and parameter 258.

• Rigid mode

Rigid mode can be specified using any of the following methods:

- **Specify M29 S ***** before a tapping command.**
- **Specify M29 S ***** in a block which contains a tapping command.**
- **Specify G84 for rigid tapping. (parameter No. 076#3 = 1).**

- **Thread lead**

In feed-per-minute mode, the thread lead is obtained from the expression, feedrate \times spindle speed. In feed-per-revolution mode, the thread lead equals the feedrate.

If a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm (No. 206) is issued.

- **S command**

Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm (No. 200).

- **F command**

Specifying a value that exceeds the upper limit of cutting feedrate causes alarm (No. 011).

- **M29**

Specifying an S command or axis movement between M29 and G84 causes alarm (No. 203).

Then, specifying M29 in the tapping cycle causes alarm (No. 204).

- **R**

Specify R in a block that performs drilling. If R is specified in a non-drilling block, it is not stored as modal data.

- **Cancel**

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

- **Tool offset**

In the canned cycle mode, tool offsets are ignored.

- **Unit of F**

	Metric input	Inch input	Remarks
G94	1 mm/min	0.01 inch/min	Decimal point programming allowed
G95	0.01 mm/rev	0.0001 inch/rev	Decimal point programming allowed

Examples

Z-axis feedrate 1000 mm/min

Spindle speed 1000 rpm

Thread lead 1.0 mm

<Programming for feed per minute>

G94 ; Specify a feed-per-minute command.

G00 X120.0 Y100.0 ; Positioning

M29 S1000 ; Rigid mode specification

G84 Z-100.0 R-20.0 F1000 ; Rigid tapping

<Programming for feed per revolution>

G95 ; Specify a feed-per-revolution command.

G00 X120.0 Y100.0 ; Positioning

M29 S1000 ; Rigid mode specification

G74 Z-100.0 R-20.0 F1.0 ; Rigid tapping

13.2.3 Peck Rigid Tapping Cycle (G84 or G74)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.

In this cycle, cutting is performed several times until the bottom of the hole is reached. Two peck tapping cycles are available: High-speed peck tapping cycle and standard peck tapping cycle. These cycles are selected using the bit (bit 0) of parameter 388.

Format

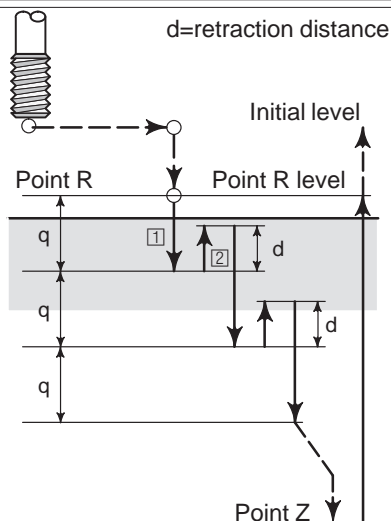
G84 (or G74) X_ Y_ Z_ R_ P_ Q_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole or the position of the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of the hole and at point R when a return is made
Q_ : Depth of cut for each cutting feed
F_ : The cutting feedrate
K_ : Number of repeats

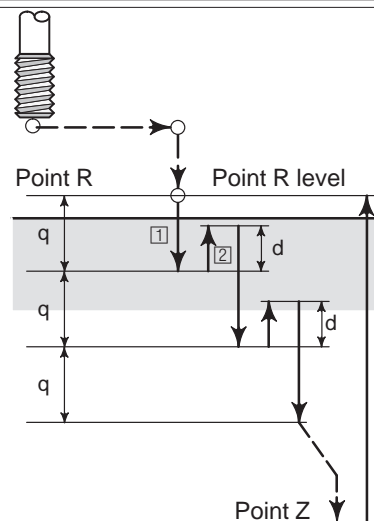
- High-speed peck tapping cycle
(Parameter No.388#0=0)

- 1 The tool operates at a normal cutting feedrate. The normal time constant is used.
- 2 Retraction can be overridden. The retraction time constant is used.

G84, G74 (G98)



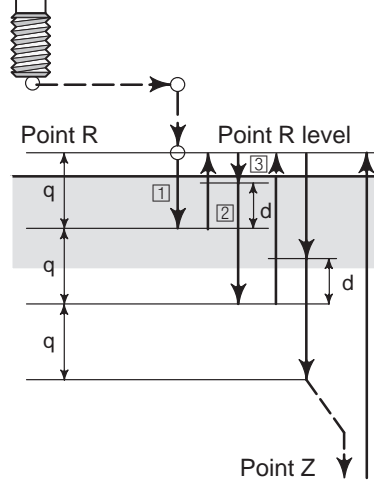
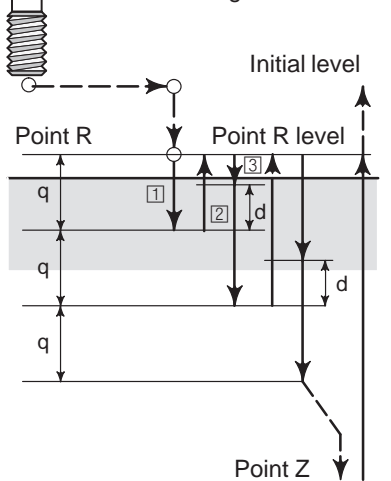
G84, G74 (G99)



- Peck tapping cycle
(Parameter No.388#0=1)

- 1 The tool operates at a normal cutting feedrate. The normal time constant is used.
- 2 Retraction can be overridden. The retraction time constant is used.
- 3 Retraction can be overridden. The normal time constant is used.

d=cutting start distance



During a rigid tapping cycle, CNC checks whether the pulse distribution is performed or not and start the next operation at the end of each operation of 1 and 2 in the peck tapping cycle.

Explanations

- **High-speed peck tapping cycle**

After positioning along the X- and Y-axes, rapid traverse is performed to point R. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d. The bit (bit 4) of parameter 063 specifies whether retraction can be overridden or not. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.

Set the retraction distance, d, in parameter 403.

- **Peck tapping cycle**

After positioning along the X- and Y-axes, rapid traverse is performed to point R level. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then a return is performed to point R. The bit (bit 4) of parameter 063 specifies whether the retraction can be overridden or not. Traverse is performed from point R to a position distance d from the end point of the last cutting, which is where cutting is restarted. For this traverse, the specification of the bit (bit 4) of parameter 063 is also valid. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.

Set d (distance to the point at which cutting is started) in parameter 403.

Restrictions

- **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm (No. 206) is issued.

- **S command**

Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm (No. 200).

- **F command**

Specifying a value that exceeds the upper limit of cutting feedrate causes alarm (No. 011).

- **M29**

Specifying an S command or axis movement between M29 and G84 causes alarm (No. 203).

Then, specifying M29 in the tapping cycle causes alarm (No. 204).

- **Q/R**

Specify Q and R in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data. When Q0 is specified, the peck rigid tapping cycle is not performed.

- **Cancel**

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

- **Tool offset**

In the canned cycle mode, tool offsets are ignored.

- **Unit of F**

	Metric input	Inch input	Remarks
G94	1 mm/min	0.01 inch/min	Decimal point programming allowed
G95	0.01 mm/rev	0.0001 inch/rev	Decimal point programming allowed

13.2.4

Canned Cycle Cancel (G80)

The rigid tapping canned cycle is canceled. For how to cancel this cycle, see Section 13.1.13.

13.3

CANNED GRINDING CYCLE

(0-GSC, 0-GSD/II)

Canned grinding cycles make it easier for the programmer to create programs that include grinding. With a canned grinding cycle, repetitive operation peculiar to grinding can be specified in a single block with a G function; without canned grinding cycles, normally more than one block is required. In addition, the use of canned grinding cycles shortens the program to save memory. The following four canned grinding cycles are available:

- Plunge grinding cycle (G75)
- Direct constant-dimension plunge grinding cycle (G77)
- Continuous-feed surface grinding cycle (G78)
- Intermittent-feed surface grinding cycle (G79)

13.3.1 Plunge Grinding Cycle (G75)

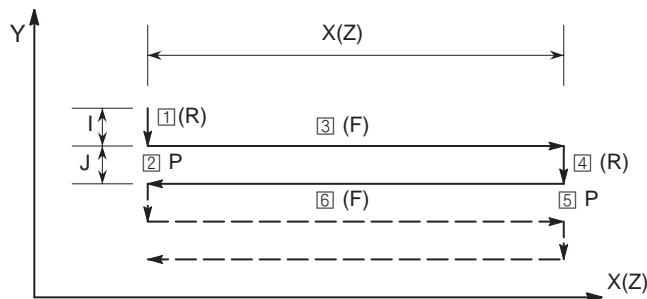
A plunge grinding cycle is performed.

Format

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

I_ : Depth-of-cut 1 (A sign in the command specifies the direction of cutting.)
 J_ : Depth-of-cut 2 (A sign in the command specifies the direction of cutting.)
 K_ : Total depth of cut
 X(Z)_ : Range of grinding (A sign in the command specifies the direction of grinding.)
 R_ : Feedrate for I and J
 F_ : Feedrate for X (Z)
 P_ : Dwell time
 L_ : Grinding-wheel wear compensation (Only for continuous dressing)

G75



Explanations

- Grinding wheel cutting
- Dwell
- Grinding
- Grinding wheel cutting
- Dwell
- Grinding (return direction)

The plunge grinding cycle consists of six operation sequences. Operations 1 to 6 are repeated until the depth reaches the total depth of cut specified at address K. In the single block stop mode, operations 1 to 6 are performed every cycle start.

- 1 Cutting is performed along the Y-axis in cutting feed mode for the amount specified by I (depth of cut 1). The feedrate is specified by R.
- 2 Dwell is performed for the time specified by P.
- 3 Cutting feed is performed for the amount specified by X (or Z). The feedrate is specified by F.
- 4 Cutting is performed along the Y-axis in cutting feed mode for the amount specified by J (depth of cut 2). The feedrate is specified by R.
- 5 Dwell is performed for the time specified by P.
- 6 Feeding is performed in the reverse direction for the amount specified by X (or Z) at a feedrate specified by F.

Restrictions

- **X(Z), I, J, K**

X, (Z), I, J, and K must all be specified in incremental mode.

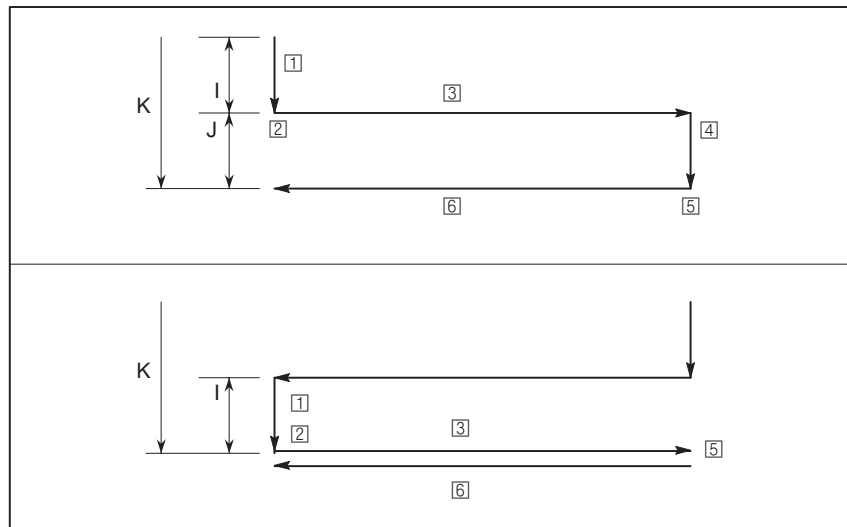
- **Clear**

I, J, X, and Z in canned cycles are modal data common to G75, G77, G78, and G79. They remain valid until new data is specified. They are cleared when a group 00 G code other than G04 or a group 01 G code other than G75, G77, G78, and G79 is specified.

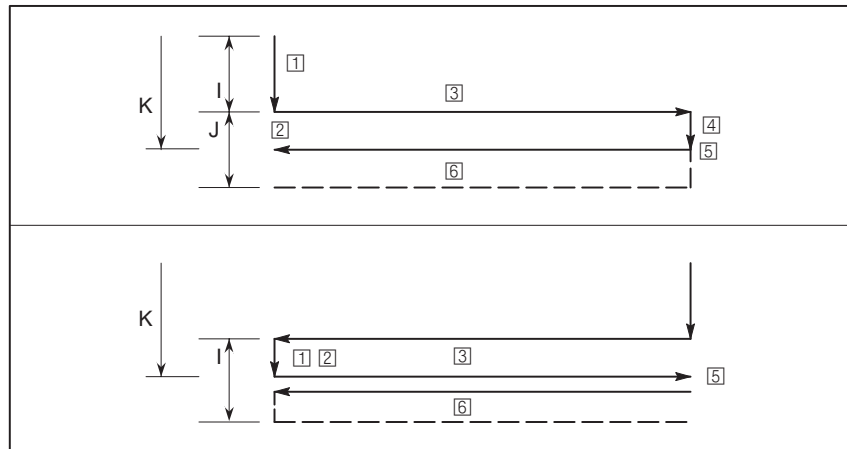
- **Operation performed when the total depth of cut is reached**

When the total depth of cut is reached during cutting using I or J, the subsequent operation sequences (up to 6) are executed, then the cycle terminates. In this case, no further cutting is performed after the total depth of cut is reached.

- Chart of operation in which the total depth of cut is reached by cutting specified by I and J:



- Chart of operation in which the total depth of cut is reached during cutting specified by I and J:



13.3.2

Direct Constant–dimension Plunge Grinding Cycle (G77)

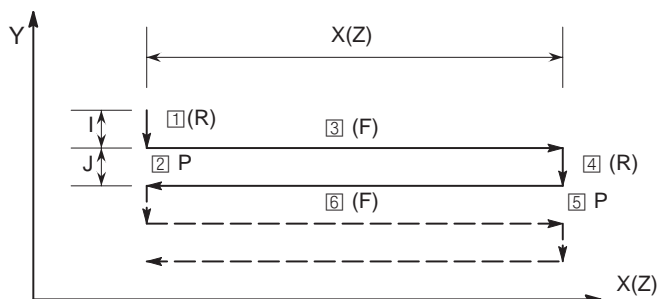
A direct constant–dimension plunge grinding cycle is performed.

Format

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

I_ : Depth–of–cut 1 (A sign in the command specifies the direction of cutting.)
 J_ : Depth–of–cut 2 (A sign in the command specifies the direction of cutting.)
 K_ : Total depth of cut
 X(Z)_ : Range of grinding (A sign in the command specifies the direction of grinding.)
 R_ : Feedrate for I and J
 F_ : Feedrate for X (Z)
 P_ : Dwell time
 L_ : Grinding–wheel wear compensation (Only for continuous dressing)

G77



Explanations

- Grinding wheel cutting
- Dwell
- Grinding
- Grinding wheel cutting
- Dwell
- Grinding (return direction)

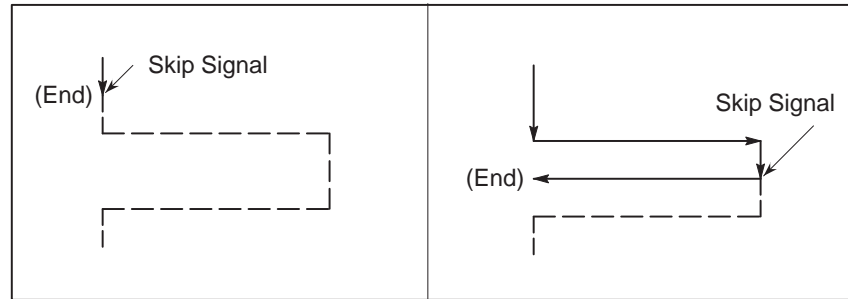
The constant–dimension plunge grinding cycle consists of six operation sequences. Operations ① to ⑥ are repeated until the depth reaches the total depth of cut specified at address K.

- ① Cutting is performed along the Y–axis in cutting feed mode for the amount specified by I (depth of cut 1). The feedrate is specified by R.
 ② Dwell is performed for the time specified by P.
 ③ Cutting feed is performed for the amount specified by X (or Z). The feedrate is specified by F.
 ④ Cutting is performed along the Y–axis in cutting feed mode for the amount specified by J (depth of cut 2). The feedrate is specified by R.
 ⑤ Dwell is performed for the time specified by P.
 ⑥ Feeding is performed in the reverse direction for the amount specified by X (or Z) at a feedrate specified by F.

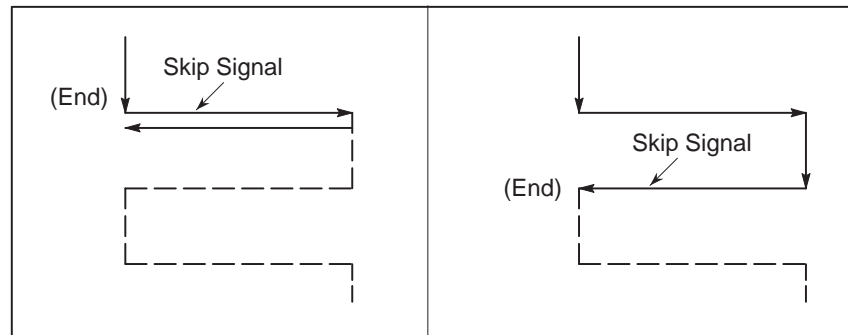
• Skip signal

When the cycle is performed using G77, a skip signal can be input to terminate the cycle. When a skip signal is input, the current operation sequence is interrupted or completed, then the cycle is terminated. The following shows how the system operates when the skip signal is input during each operation sequence.

- When the skip signal is input during operation sequence 1 or 4 (cutting feed specified by I or J), cutting is stopped immediately and the tool returns to the X (Z) coordinate at which the cycle started.



- When the skip signal is input during operation sequence 2 or 5 (dwell), dwell is stopped immediately and the tool returns to the X (Z) coordinate at which the cycle started.
- When the skip signal is input during operation sequence 3 or 6 (movement), the tool returns to the X (Z) coordinate at which the cycle started after the movement specified by X (Z) is completed.



Restrictions

- X(Z), I, J, K
- Clear

X, (Z), I, J, and K must all be specified in incremental mode.

I, J, X, and Z in canned cycles are modal data common to G75, G77, G78, and G79. They remain valid until new data is specified. They are cleared when a group 00 G code other than G04 or a group 01 G code other than G75, G77, G78, and G79 is specified.

13.3.3 Continuous-feed Surface Grinding Cycle (G78)

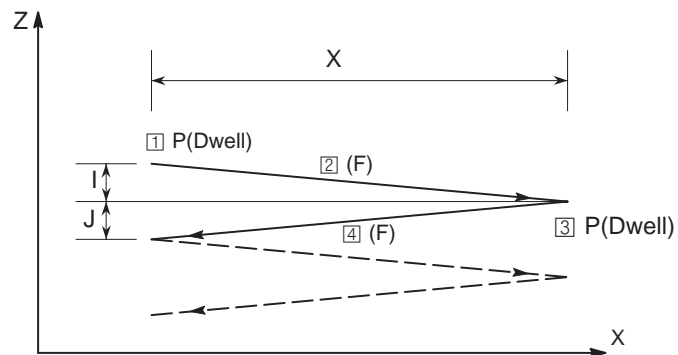
A continuous-feed surface grinding cycle is performed.

Format

G78 I_ (J_) K_ X_ F_ P_ L_ ;

I_ : Depth-of-cut 1 (A sign in the command specifies the direction of cutting.)
 J_ : Depth-of-cut 2 (A sign in the command specifies the direction of cutting.)
 K_ : Total depth of cut
 X(Z)_ : Range of grinding (A sign in the command specifies the direction of grinding.)
 R_ : Feedrate for I and J
 F_ : Feed rate
 P_ : Dwell time
 L_ : Grinding-wheel wear compensation (Only for continuous dressing)

G78



Explanations

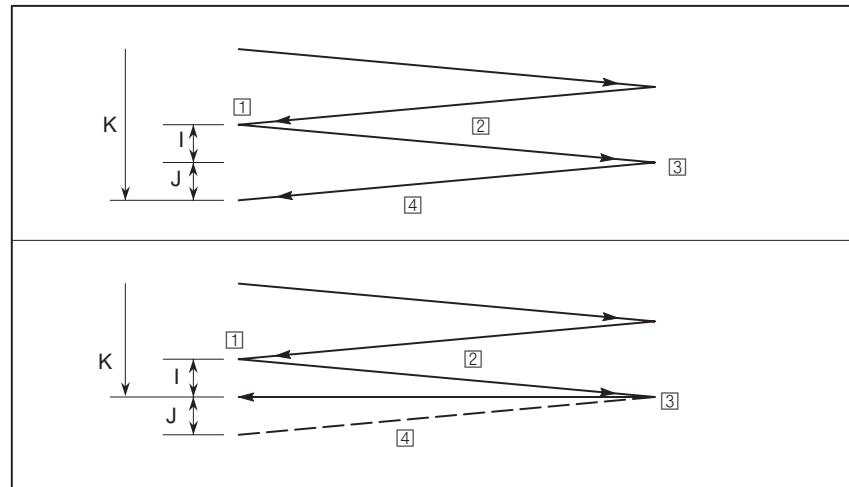
The continuous-feed surface grinding cycle consists of four operation sequences. Operations ① to ④ are repeated until the depth reaches the total depth of cut specified in address K. In the single block stop mode, operations ① to ④ are performed every cycle start.

- ① Dwell
- ② Grinding
- ③ Dwell
- ④ Grinding (in reverse direction)

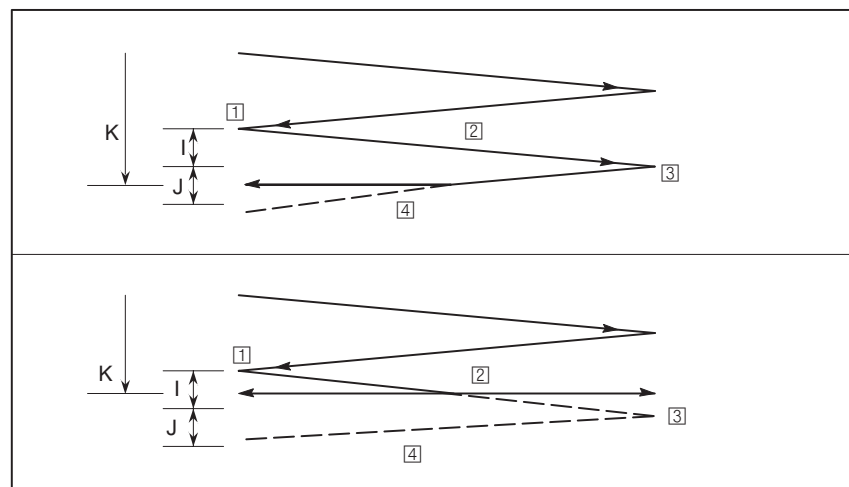
Restrictions

- **J** When J is omitted, it is assumed to be 1. J is valid only in the block where it is specified.
- **I, J, K, X** X, (Z), I, J, and K must all be specified in incremental mode.
- **Clear** I, J, X, and Z in canned cycles are modal data common to G75, G77, G78, and G79. They remain valid until new data is specified. They are cleared when a group 00 G code other than G04 or a group 01 G code other than G75, G77, G78, and G79 is specified.
- **Operation performed when the total depth of cut is reached** When the total depth of cut is reached during cutting using I or J, the subsequent operation sequences (up to 6) are executed, then the cycle terminates. In this case, no further cutting is performed after the total depth of cut is reached.

- Chart of operation in which the total depth of cut is reached by cutting specified by I and J:



- Chart of operation in which the total depth of cut is reached during cutting specified by I and J:



13.3.4 Intermittent-feed Surface Grinding Cycle (G79)

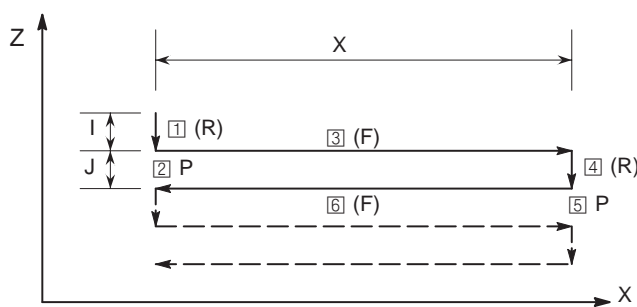
An intermittent-feed surface grinding cycle is performed.

Format

G79 I_ J_ K_ X_ R_ F_ P_ L_ ;

I_ : Depth-of-cut 1 (A sign in the command specifies the direction of cutting.)
 J_ : Depth-of-cut 2 (A sign in the command specifies the direction of cutting.)
 K_ : Total depth of cut
 X(Z)_ : Range of grinding (A sign in the command specifies the direction of grinding.)
 R_ : Feedrate for I and J
 F_ : Feedrate for X (Z)
 P_ : Dwell time
 L_ : Grinding-wheel wear compensation (Only for continuous dressing)

G79



Explanations

- Grinding wheel cutting
- Dwell
- Grinding
- Grinding wheel cutting
- Dwell
- Grinding (return direction)

The intermittent-feed surface grinding cycle consists of six operation sequences. Operations ① to ⑥ are repeated until the depth reaches the total depth of cut specified at address K. In the single block stop mode, operations ① to ⑥ are performed every cycle start.

- ① Cutting is performed along the Z-axis in cutting feed mode for the amount specified by I (depth of cut 1). The feedrate is specified by R.
- ② Dwell is performed for the time specified by P.
- ③ Cutting feed is performed for the amount specified by X (or Z). The feedrate is specified by F.
- ④ Cutting is performed along the Z-axis in cutting feed mode for the amount specified by J (depth of cut 2). The feedrate is specified by R.
- ⑤ Dwell is performed for the time specified by P.
- ⑥ Feeding is performed in the reverse direction for the amount specified by X at a feedrate specified by F.

Restrictions

- **X, I, J, K** X, (Z), I, J, and K must all be specified in incremental mode.
- **Clear** I, J, X, and Z in canned cycles are modal data common to G75, G77, G78, and G79. They remain valid until new data is specified. They are cleared when a group 00 G code other than G04 or a group 01 G code other than G75, G77, G78, and G79 is specified.

13.4 GRINDING—WHEEL WEAR COMPENSATION BY CONTINUOUS DRESSING (0—GSC, 0—GSD/II)

Explanations

• Specification

This function enables continuous dressing.

When G75, G77, G78, or G79 is specified, grinding wheel cutting and dresser cutting are compensated continuously according to the amount of continuous dressing during grinding.

Specify an offset number (grinding-wheel wear compensation number) at address L in the block containing G75, G77, G78, or G79. The compensation amount set in the offset memory area corresponding to the specified number is used as the amount of dressing.

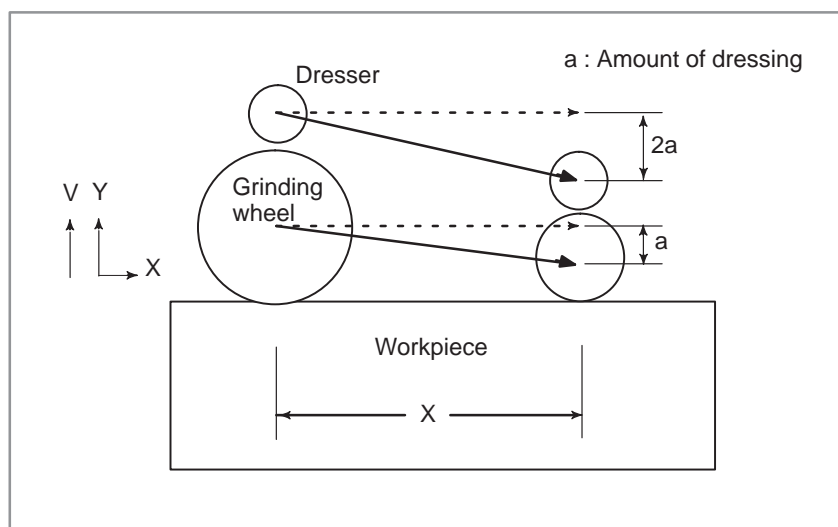
Up to 200 offset numbers (L1 to L200) can be specified. Compensation amounts must be set beforehand in offset memory corresponding to offset numbers from the CRT/MDI panel.

When L is omitted or L0 is specified in a surface grinding canned cycle block, compensation is not performed.

• Compensation

Compensation is performed for every grinding operation (every movement along the X-axis) in the operation sequences of a canned grinding cycle. While the tool moves along the X-axis, compensation is performed along the Y-axis (grinding wheel cutting) and the V-axis (dresser cutting) for simultaneous three-axis interpolation.

The travel distance (compensation amount) along the Y-axis is the same as specified dressing amount, and the travel distance along the V-axis is twice as long (diameter).



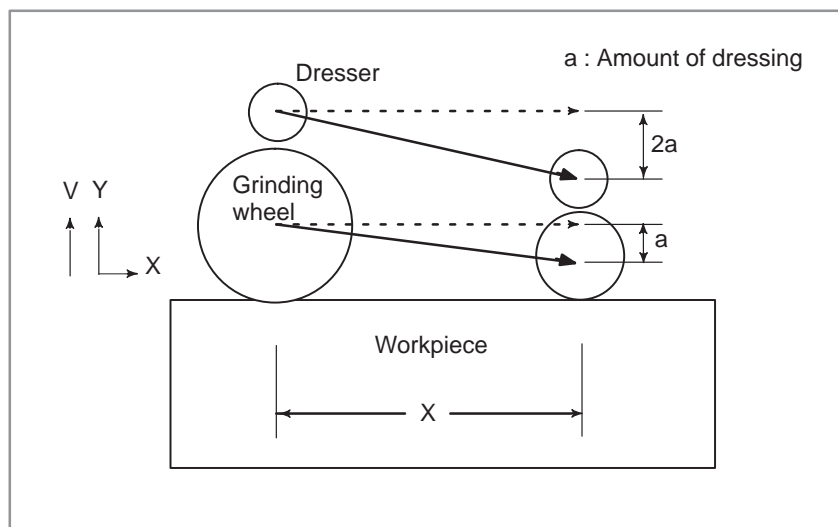
13.5 AUTOMATIC GRINDING WHEEL DIAMETER COMPENSATION AFTER DRESSING

13.5.1 Checking the Minimum Grinding Wheel Diameter (0-GSC, 0-GSD/II)

Compensation amounts set in offset memory can be modified by using the external tool compensation function or programming (by changing offsets using custom macro variables).

With these functions, the compensation amount for the diameter of the dressed grinding wheel can be changed.

If the compensation amount associated with the offset number specified in the H code is smaller than the minimum grinding wheel diameter specified in parameter 838 when programmed compensation (using G43 or G44) is performed, a signal is output to the PMC.



13.6 IN-FEED GRINDING ALONG THE Y AND Z AXES AT THE END OF TABLE SWING (0-GSC, 0-GSD/II)

Format

```
G161 R_ ;
  profile program
G160 ;
```

Explanations

- **G161 R_**
- **Profile program**
- **G160**

Specify the start of an operation mode and profile program. Also specify the depth of cut in R.

Program a workpiece figure in the Y-Z plane using linear interpolation (G01) and/or circular interpolation (G02 or G03). One or more blocks can be specified.

Cancel the operation mode (end of the profile program).

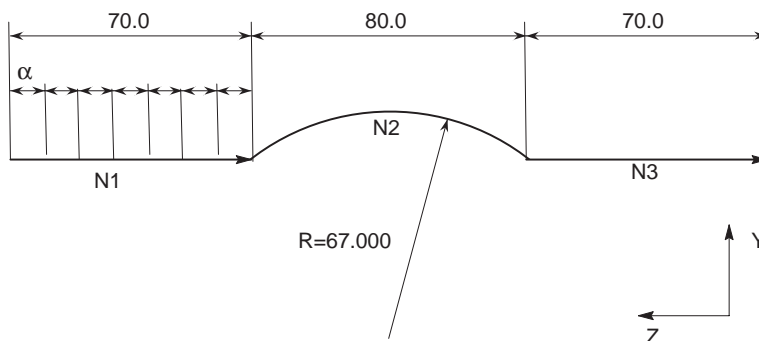
Restrictions

- **Profile program**

Do not specify codes other than G01, G02, and G03 within the profile program.

Examples

```
O0001 ;
:
N0 G161 R10.0 ;
N1 G91 G01 Z-70.0 F100 ;
N2 G19 G02 Z-80.0 R67.0 ;
N3 G01 Z-70.0 ;
N4 G160 ;
:
```



In the above program, every time the in-feed cutting start signal is input, the tool is moved by 10.000 along the machining profile shown above.

α = Travel distance for each in-feed control cutting start signal input
The feedrate is programmed with an F code.

13.7

OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING

Chamfering and corner rounding blocks can be inserted automatically between linear interpolation and linear interpolation blocks

Format

, C_	Chamfering
, R_	Corner R

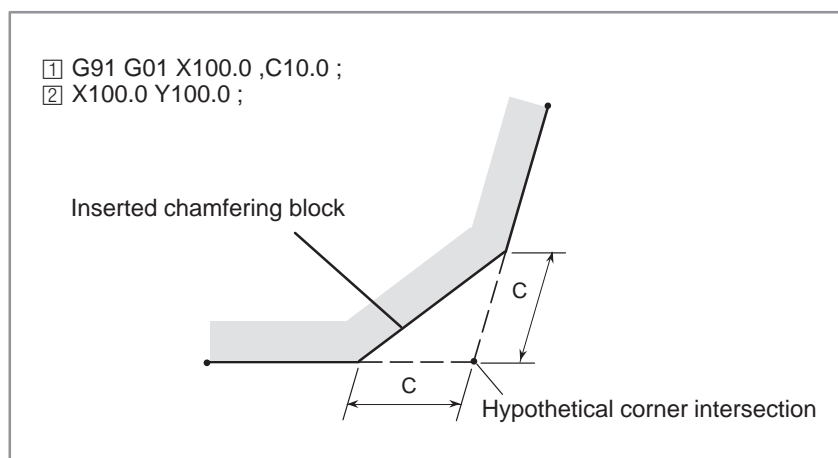
Explanations

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

Blocks specifying chamfering and corner rounding can be specified consecutively.

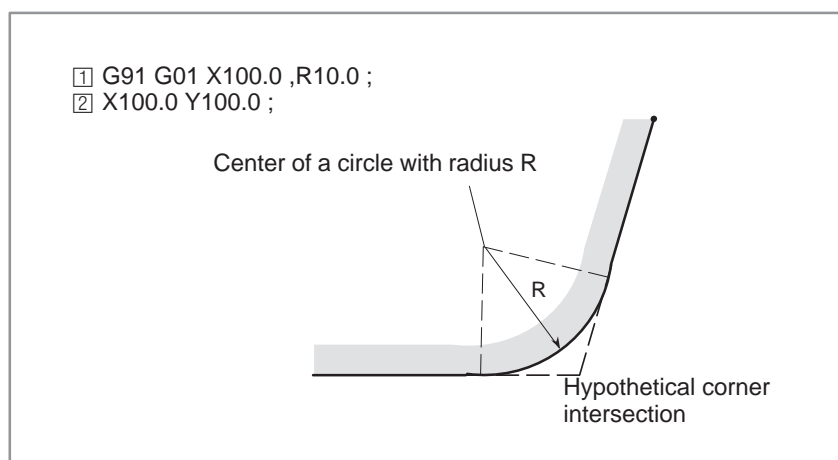
- **Chamfering**

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



- **Corner R**

After R, specify the radius for corner rounding.



Restrictions

- **Plane selection**

Chamfering and corner rounding can be performed only in the plane specified by plane selection (G17, G18, or G19). These functions cannot be performed for parallel axes.
- **Next block**

A block specifying chamfering or corner rounding must be followed by a block that specifies a move command using linear interpolation (G01). If the next block does not contain these specifications, alarm 052 is issued.
- **Plane switching**

A chamfering or corner rounding block can be inserted only for move commands which are performed in the same plane. In a block that comes immediately after plane switching (G17, G18, or G19 is specified), neither chamfering nor corner rounding can be specified.
- **Exceeding the move range**

If the inserted chamfering or corner rounding block causes the tool to go beyond the original interpolation move range, alarm 055 is issued.
- **Coordinate system**

In a block that comes immediately after the coordinate system is changed (G92, or G52 to G59) or a return to the reference position (G28 to G30) is specified, neither chamfering nor corner rounding can be specified.
- **Travel distance 0**

The chamfering or corner rounding block is regarded as having a travel distance of zero if the angle between the two straight lines is within $+1^\circ$.
- **Unavailable G codes**

The following G codes cannot be used in a block that specifies chamfering or corner rounding. They also cannot be used between chamfering and corner rounding blocks that define a continuous figure.

 - G codes of group 00 (except G04)
 - G68 of group 16
- **Threading**

Corner rounding cannot be specified in a threading block.

13.8

EXTERNAL MOTION FUNCTION (G81)

Upon completion of positioning in each block in the program, an external operation function signal can be output to allow the machine to perform specific operation.
Concerning this operation, refer to the manual supplied by the machine tool builder.

Format

G81 IP _ ; (IP_ Axis move command)

Explanations

Every time positioning for the IP_ move command is completed, the CNC sends a external operation function signal to the machine. An external operation signal is output for each positioning operation until canceled by G80 or a group 01 G code.

Restrictions

- **A block without X or Y axis**

No external operation signals are output during execution of a block that contains neither X nor Y.

13.9 SMALL-DIAMETER PECK DRILLING CYCLE

When drilling is performed using a tool with an overload torque detection function, this function allows the small-diameter peck drilling cycle to be performed by entering an overload torque detection signal as a skip signal to change the spindle speed and cutting feedrate. The machining sequence for small-diameter peck drilling is shown below.

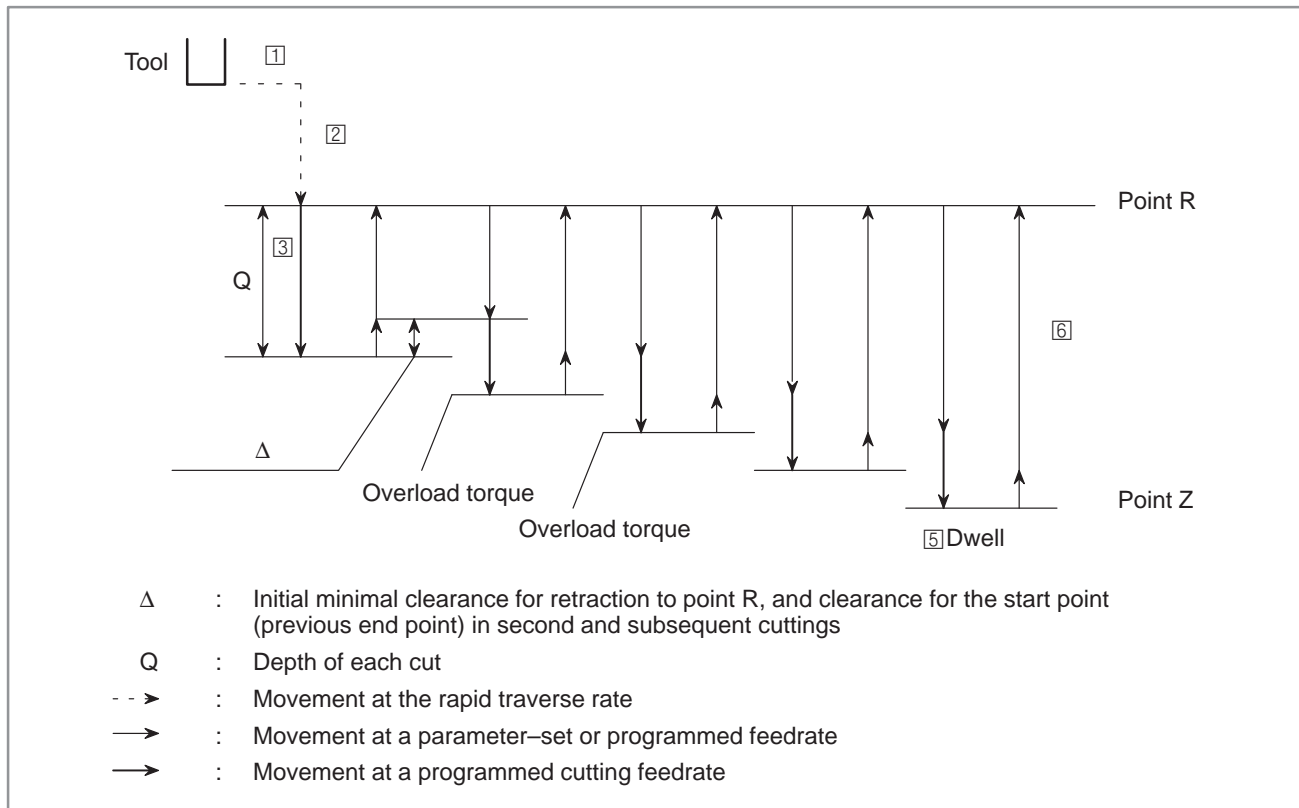


Fig. 13.9

The machining sequence of the small-diameter peck drilling cycle is as follows:

- ① Positioning along the X-axis and Y-axis
- ② Positioning at point R along the Z-axis
- ③ Drilling along the Z-axis (maximum depth of cut Q)
- ④ Repetition of the following operations along the Z-axis until the bottom is reached
 - Retract (→ Minimal clearance Δ)
 - Retract (→ Point R)
 - Advance (Point R → Start point + Clearance Δ)
 - Cut (Depth of cut Q + Δ)
- ⑤ Dwell at the hole bottom
- ⑥ Return to point R along the Z-axis (or to the initial point)

Format

```

M□□ ;
G83 X_ Y_ Z_ R_ Q_ F_ I_ K_ P_ ;
X_ Y_ ;
X_ Y_ ;

G80 ;

```

Specification	Code	Description
Specification of small-diameter peck drilling cycle	M□□	Specify an M code for the small-diameter peck drilling cycle set in parameter No. 304. G83, specified below, is regarded as being the small-diameter peck drilling cycle.
Specification of the peck drilling cycle	G83	G83 is modal. Once this code is specified, the code remains valid until another canned cycle is specified or a G code for canceling the canned cycle is specified.
Specification of a hole position	X_ Y_	Specify a hole position by using either an incremental or absolute value. The path and feedrate specification is the same as for positioning using G00.
Hole machining data	Z_	Specify the distance from point R to the hole bottom by using an incremental value, or specify the position of the hole bottom by using an absolute value.
	Q_	Depth of each cut
	R_	Specify the distance from the initial level to point R by using an incremental value, or specify the position of point R by using an absolute value.
	F_	Cutting feedrate
	I_	Advance or retract feedrate (The default is the parameter-set value.)
	K_	Number of repeats (The default is 1.)
	P_	Dwell time at the hole bottom (The default is 0 seconds.)
Cancellation of small-diameter peck drilling cycle	G80	This code cancels the small-diameter peck drilling cycle. This code also cancels the M code for small-diameter peck drilling cycle specification.

NOTE

The hole machining data is modal. When identical hole machining is repeated, the hole machining data need not be specified in each of the subsequent blocks.

Explanations

- (1) The small-diameter peck drilling cycle is executed by specifying G83 after specifying an M code for the small-diameter peck drilling cycle. During the execution of this cycle, the small-diameter peck drilling cycle in-progress signal is output during the period between point R positioning of the drilling axis and the return to point R/initial level.
- (2) As an overload torque detection signal, the skip signal (X008#7) is used. The skip signal is enabled only while the drilling axis is placed between point R and point Z, and an advance or cutting operation is being performed. When the skip signal is entered while an advance or cutting operation is being performed, a retract operation is started immediately.
- (3) In one G83 cycle, the cutting condition is changed for each peck drilling operation (advance → cut → retract).

- Cutting feedrate change

The cutting feedrate programmed using an F code is changed for each of the second and subsequent cutting operations. In parameter Nos. 307 and 308, specify the change ratios applicable when the skip signal is received and when the skip signal is not received, respectively, in the previous cutting operation.

$$\text{Cutting feedrate} = F \times \alpha$$

(First operation)

$$\alpha = 1.0$$

(Second and subsequent operations)

$$\alpha = \alpha \times \beta \div 100$$

β represents the change ratio for one operation. If the skip signal was entered during the previous cutting:

$$\beta = \text{Parameter No. 307}$$

When the skip signal was not entered during the previous cutting:

$$\beta = \text{Parameter No. 308}$$

If the cutting feedrate change ratio α becomes smaller than the value set in parameter No. 309, no further cutting feedrate changes are made.

- Spindle speed change

The spindle speed programmed using an S code is changed for each of second and subsequent advance operations. In parameter Nos. 305 and 306, specify the change ratios applicable when the skip signal is received and when the skip signal is not received, respectively, in the previous cutting operation.

$$\text{Spindle speed} = S \times \gamma$$

(First operation)

$$\gamma = 1.0$$

(Second and subsequent operations)

$$\gamma = \gamma \times \delta \div 100$$

δ represents the change ratio of one operation. If the skip signal was entered during the previous cutting:

$$\delta = \text{Parameter No. 305}$$

When the skip signal was not entered during the previous cutting:

$$\delta = \text{Parameter No. 306}$$

If the spindle speed is clamped to the lower limit, no spindle speed change is made.

- (4) Advance and retract operations during peck drilling are performed not as positioning by rapid traverse but as cutting feed.
- (5) When an advance/retract feedrate during peck drilling is to be specified using an I code, the same format as that for an F code is used. An I code specified in G83 remains valid until the specification of G80 or the input of a reset signal.

NOTE

- 1 An M code for the small-diameter peck drilling cycle must be specified before the specification of G83. Furthermore, the block containing the M code must not include any other codes. The M code is subjected to internal processing, but is not output.
- 2 Retract and advance operations are controlled by the cutting feedrate acceleration/deceleration function. An in-position check is made at point R during a retract operation.
- 3 When the skip signal is received during advance operation, a retract operation is started.
- 4 To change the spindle speed during peck drilling in the small-diameter peck drilling cycle, a 4- or 5-digit S command is required.
- 5 Point R must be positioned away from the workpiece surface by an amount equal to clearance Δ or more.

14

COMPENSATION FUNCTION

General

This chapter describes the following compensation functions:

TOOL LENGTH OFFSET (G43, G44, G49)	Sec.14.1
AUTOMATIC TOOL LENGTH MEASUREMENT (G37)	Sec.14.2
TOOL OFFSET (G45 TO G48)	Sec.14.3
CUTTER COMPENSATION B (G39 TO G42)	Sec.14.4
OVERVIEW OF CUTTER COMPENSATION C (G40 TO G42)	Sec.14.5
DETAILS OF CUTTER COMPENSATION C	Sec.14.6
TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	Sec.14.7
SCALING (G50, G51)	Sec.14.8
COORDINATE SYSTEM ROTATION (G68, G69)	Sec.14.9
INDEX TABLE INDEXING FUNCTION	Sec.14.10
NORMAL DIRECTION CONTROL (G150, G151, G152)	Sec.14.11

14.1

TOOL LENGTH OFFSET (G43, G44, G49)

This function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.

Specify the direction of offset with G43 or G44. Select a tool length offset value from the offset memory by entering the corresponding address and number (H code).

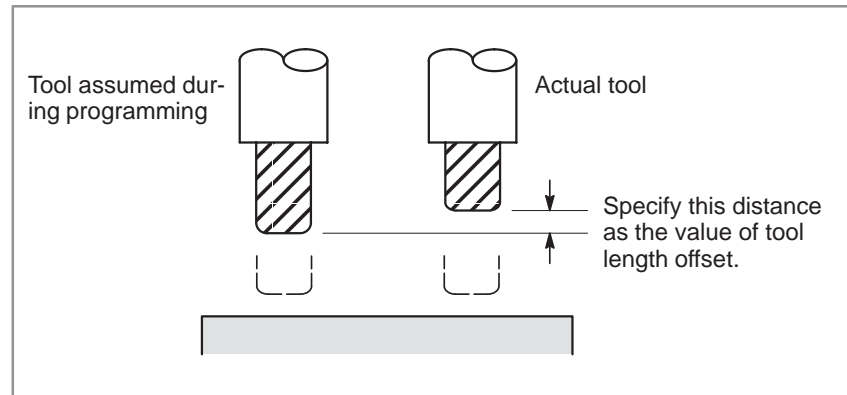


Fig14.1 Tool length offset

The following three methods of tool length offset can be used, depending on the axis along which tool length offset can be made.

· **Tool length offset A**

Compensates for the difference in tool length along the Z-axis.

· **Tool length offset B**

Compensates for the difference in tool length along the X-, Y-, or Z-axis.

· **Tool length offset C**

Compensates for the difference in tool length along a specified axis.

Format

Tool length offset A	G43 Z_ H_ ; G44 Z_ H_ ;	Explanation of each address G43 : Positive offset G44 : Negative offset G17 : XY plane selection G18 : ZX plane selection G19 : YZ plane selection α : Address of a specified axis H : Code for specifying the tool length offset value
Tool length offset B	G17 G43 Z_ H_ ; G17 G44 Z_ H_ ; G18 G43 Y_ H_ ; G18 G44 Y_ H_ ; G19 G43 X_ H_ ; G19 G44 X_ H_ ;	
Tool length offset C	G43 α_ H_ ; G44 α_ H_ ;	
Tool length offset cancel	G49 ; or H0 ;	

Explanations

- **Selection of tool length offset**

Select tool length offset A, B, or C, by setting bit 6 of parameter 003 and bit 3 of parameter No. 019.

- **Direction of the offset**

When G43 is specified, the tool length offset value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.

If movement along an axis is not specified, the system assumes that a move command that causes no movement is specified. When a positive value is specified for tool length offset with G43, the tool is moved accordingly in the positive direction. When a positive value is specified with G44, the tool is moved accordingly in the negative direction. When a negative value is specified, the tool is moved in the opposite direction. G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

- **Specification of the tool length offset value**

The tool length offset value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the commands specified by a command in the program. The tool length offset value may be set in the offset memory through the CRT/MDI panel.

The range of values that can be set as the tool length offset value is as follows.

Increment system	Metric input	Inch input
IS-B	0 to $\pm 999.999\text{mm}$	0 to $\pm 99.9999\text{inch}$
IS-C	0 to $\pm 999.9999\text{mm}$	0 to $\pm 99.99999\text{inch}$

WARNING

When the tool length offset value is changed due to a change of the offset number, the offset value changes to the new tool length offset value, the new tool length offset value is not added to the old tool length offset value.

H1 : tool length offset value 20.0

H2 : tool length offset value 30.0

G90 G43 Z100.0 H1 ; Z will move to 120.0

G90 G43 Z100.0 H2 ; Z will move to 130.0

NOTE

The tool length offset value corresponding to offset No. 0 , that is, H0 always means 0. It is impossible to set any other tool length offset value to H0.

- **Performing tool length offset along two or more axes**

Tool length offset B can be executed along two or more axes when the axes are specified in two or more blocks.

Offset in X and Y axes.

G19 G43 H _ ; Offset in X axis

G18 G43 H _ ; Offset in Y axis

(Offsets in X and Y axes are performed)

If the bit (bit 5 of parameter No. 036) is set to 1, an alarm will not occur even when tool length offset C is executed along two or more axes at the same time.

- **Tool length offset cancel**

To cancel tool length offset, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

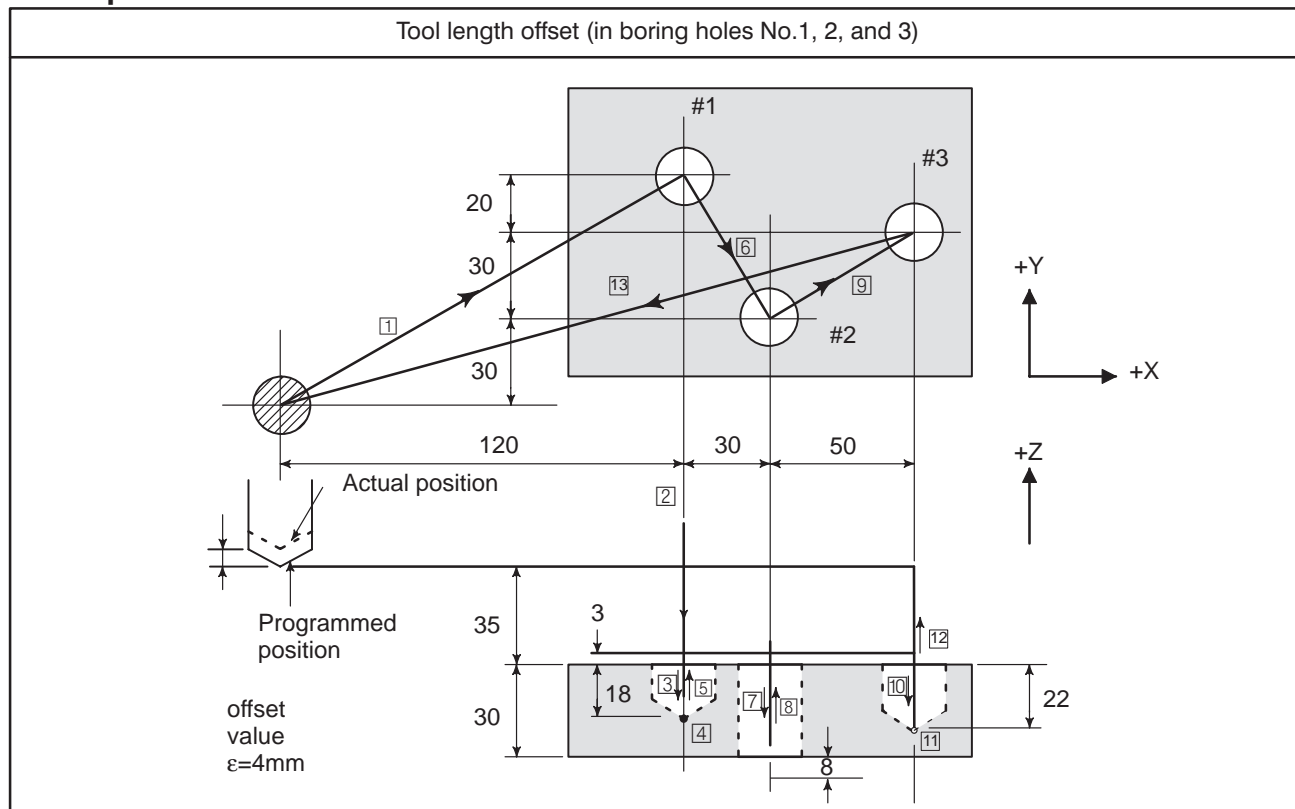
CAUTION

After tool length offset B is executed along two or more axes, offset along all the axes is canceled by specifying G49. If H0 is specified, only offset along an axis perpendicular to the specified plane is canceled.

NOTE

In the case of the offset in three axes or more, if the offset is canceled by G49 code, the P/S alarm 015 is generated. Cancel the offset by using G49 and H00.

Examples



· Program

H1=-4.0 (Tool length offset value)

```

N1  G91 G00 X120.0 Y80.0 ; ..... [1]
N2  G43 Z-32.0 H1 ; ..... [2]
N3  G01 Z-21.0 F1000 ; ..... [3]
N4  G04 P2000 ; ..... [4]
N5  G00 Z21.0 ; ..... [5]
N6  X30.0 Y-50.0 ; ..... [6]
N7  G01 Z-41.0 ; ..... [7]
N8  G00 Z41.0 ; ..... [8]
N9  X50.0 Y30.0 ; ..... [9]
N10 G01 Z-25.0 ; ..... [10]
N11 G04 P2000 ; ..... [11]
N12 G00 Z57.0 H0 ; ..... [12]
N13 X-200.0 Y-60.0 ; ..... [13]
N14 M2 ;

```

14.2

AUTOMATIC TOOL LENGTH MEASUREMENT (G37)

By issuing G37 the tool starts moving to the measurement position and keeps on moving till the approach end signal from the measurement device is output. Movement of the tool is stopped when the tool tip reaches the measurement position.

Difference between coordinate value when tool reaches the measurement position and coordinate value commanded by G37 is added to the tool length offset amount currently used.

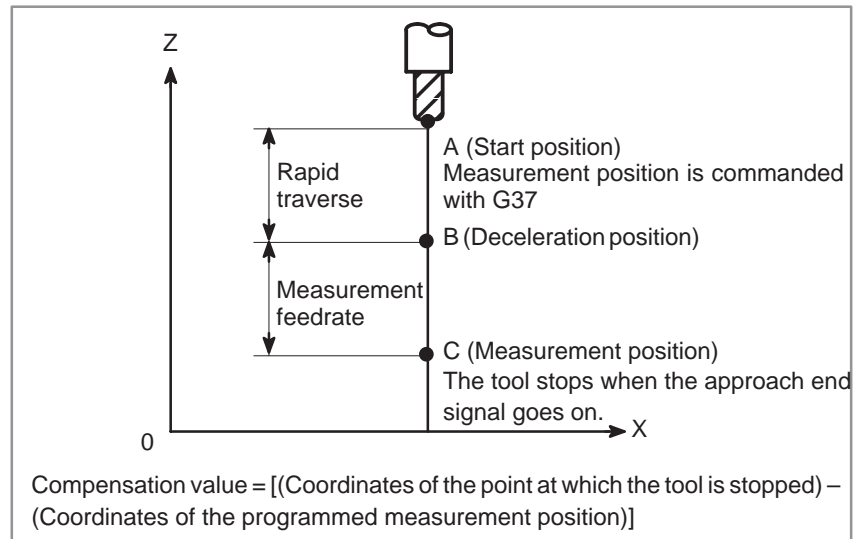


Fig. 14.2 (a) Automatic tool length measurement

Format

G92 IP_ ;	Sets the workpiece coordinate system. (It can be set with G54 to G59. See Chapter 7, "Coordinate System.")
H○○	Specifies an offset number for tool length offset.
G90 G37 IP_ ;	Absolute command
	• G37 is valid only in the block in which it is specified.
	IP_ indicates the X-, Y-, Z-, or fourth axis.

Explanations

- **Setting the workpiece coordinate system**

Set the workpiece coordinate system so that a measurement can be made after moving the tool to the measurement position. The coordinate system must be the same as the workpiece coordinate system for programming.

- **Specifying G37**

Specify the absolute coordinates of the correct measurement position. Execution of this command moves the tool at the rapid traverse rate toward the measurement position, reduces the feedrate halfway, then continues to move it until the approach end signal from the measuring instrument is issued. When the tool tip reaches the measurement position, the measuring instrument sends an approach end signal to the CNC which stops the tool.

- **Changing the offset value**

The difference between the coordinates of the position at which the tool reaches for measurement and the coordinates specified by G37 is added to the current tool length offset value.

Offset value =

[(Coordinates of the position at which the tool reaches for measurement) – (Coordinates specified by G37)]

These offset values can be manually changed from MDI.

- **Alarm**

When automatic tool length measurement is executed, the tool moves as shown in Fig. 15.2 (b). If the approach end signal goes on while the tool is traveling from point B to point C, an alarm occurs. Unless the approach end signal goes on before the tool reaches point F, the same alarm occurs. The alarm number is 080.

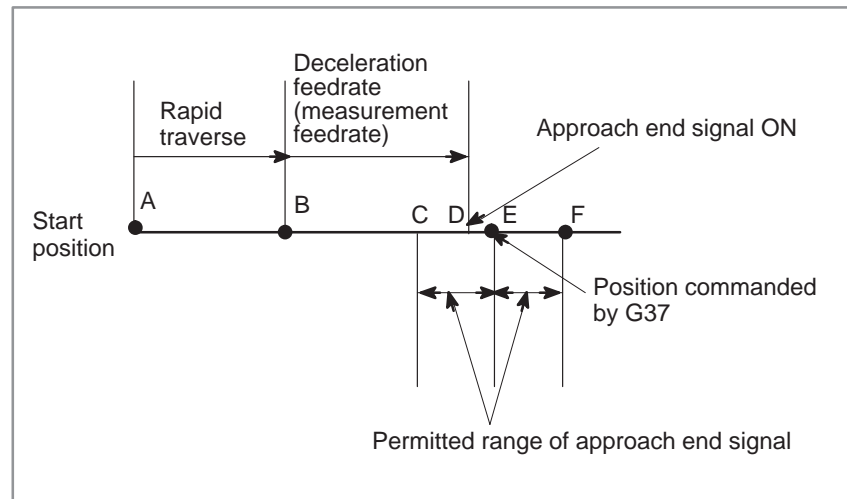


Fig. 14.2 (b) Tool movement to the measurement position

WARNING

When a manual movement is inserted into a movement at a measurement federate, return the tool to the position before the inserted manual movement for restart.

NOTE

- 1 When an H code is specified in the same block as G37, an alarm is generated. Specify H code before the block of G37.
- 2 The measurement speed (parameter No. 558), deceleration position (parameter No. 813), and permitted range of the approach end signal (parameter No. 814) are specified by the machine tool builder.
- 3 When offset memory A is used, the offset value is changed. When offset memory B is used, the tool wear compensation value is changed.

- 4 The approach end signal is monitored usually every 2 ms. The following measuring error is generated:

$$ERR_{\max} = F_m \times 1/60 \times 4/1000 \quad \text{where}$$

ERR_{\max} : maximum measuring error (mm)

F_m : measurement federate (mm/min.)

For example, when $F_m = 100$ mm/min., $ERR_{\max} = 0.007$ mm

- 5 The tool stops a maximum of 16 ms after the approach end signal is detected. But the value of the position at which the approach end signal was detected (note the value when the tool stopped) is used to determine the offset amount. The overrun for 16 ms is:

$$Q_{\max} = F_m \times 1/60 \times 20 + TS/1000$$

TS : Servo time constant

Q_{\max} : maximum overrun (mm)

F_m : measurement federate (mm/min.)

Examples

G92 Z760.0 X1100.0 ; Sets a workpiece coordinate system with respect to the programmed absolute zero point.

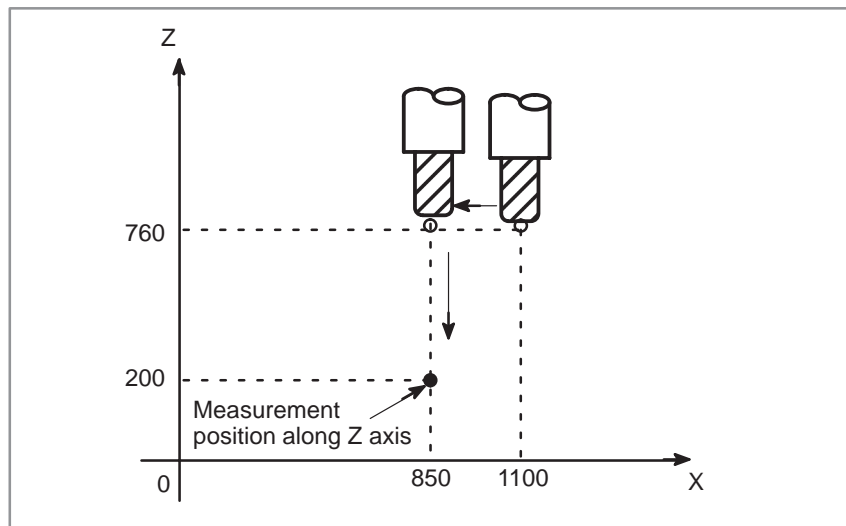
G00 G90 X850.0 ; Moves the tool to X850.0.
That is the tool is moved to a position that is a specified distance from the measurement position along the Z-axis.

G43 H01 ; Specifies offset number 1.

G37 Z200.0 ; Moves the tool to the measurement position.

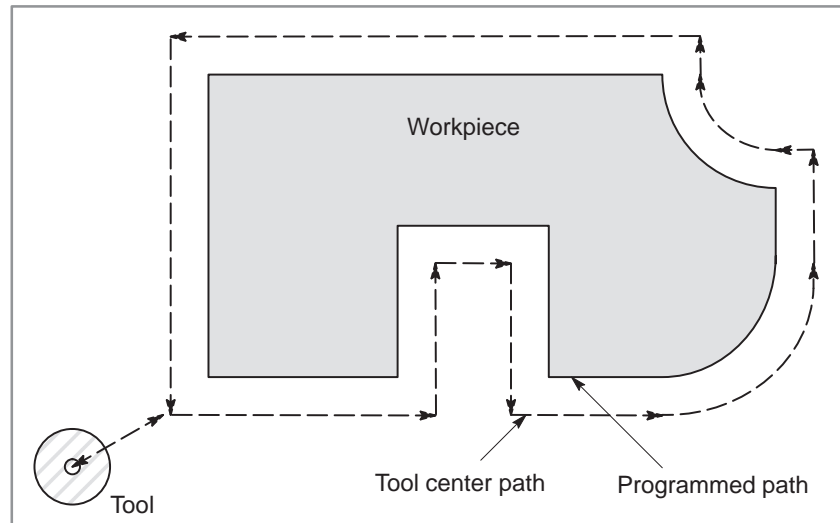
G00 Z204.0 ; Retracts the tool a small distance along the Z-axis.

For example, if the tool reaches the measurement position with Z198.0;, the compensation value must be corrected. Because the correct measurement position is at a distance of 200 mm, the compensation value is lessened by 2.0 mm ($198.0 - 200.0 = -2.0$).



14.3 TOOL OFFSET (G45 TO G48)

The programmed travel distance of the tool can be increased or decreased by a specified tool offset value or by twice the offset value. The tool offset function can also be applied to an additional axis.



Format

G45 IP_D_ ; Increase the travel distance by the tool offset value

G46 IP_D_ ; Decrease the travel distance by the tool offset value

G47 IP_D_ ; Increase the travel distance by twice the tool offset value

G48 IP_D_ ; Decrease the travel distance by twice the tool offset value

G45 to G48 : One-shot G code for increasing or decreasing the travel distance

IP : Command for moving the tool

D : Code for specifying the tool offset value

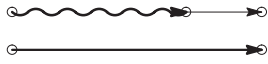
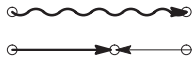


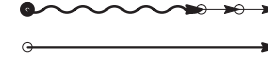

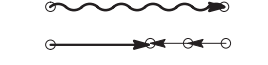

Explanations




• Increase and decrease

As shown in Table 14.3 (a), the travel distance of the tool is increased or decreased by the specified tool offset value.

In the absolute mode, the travel distance is increased or decreased as the tool is moved from the end position of the previous block to the position specified by the block containing G45 to G48.

Table14.3 (a) Increase and decrease of the tool travel distance

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G45	 Start position End position	 Start position End position
G46	 Start position End position	 Start position End position
G47	 Start position End position	 Start position End position
G48	 Start position End position	 Start position End position

 Programmed movement distance
 Tool offset value
 Actual movement position

If a move command with a travel distance of zero is specified in the incremental command (G91) mode, the tool is moved by the distance corresponding to the specified tool offset value.

If a move command with a travel distance of zero is specified in the absolute command (G90) mode, the tool is not moved.

• Tool offset value

Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.

Tool offset values can be set within the following range:

Table14.3 (b) Range of tool offset values

Increment system	Metric input	inch input
IS-B	0 to $\pm 999.999\text{mm}$	0 to $\pm 99.9999\text{inch}$
IS-C	0 to $\pm 999.9999\text{mm}$	0 to $\pm 99.99999\text{inch}$

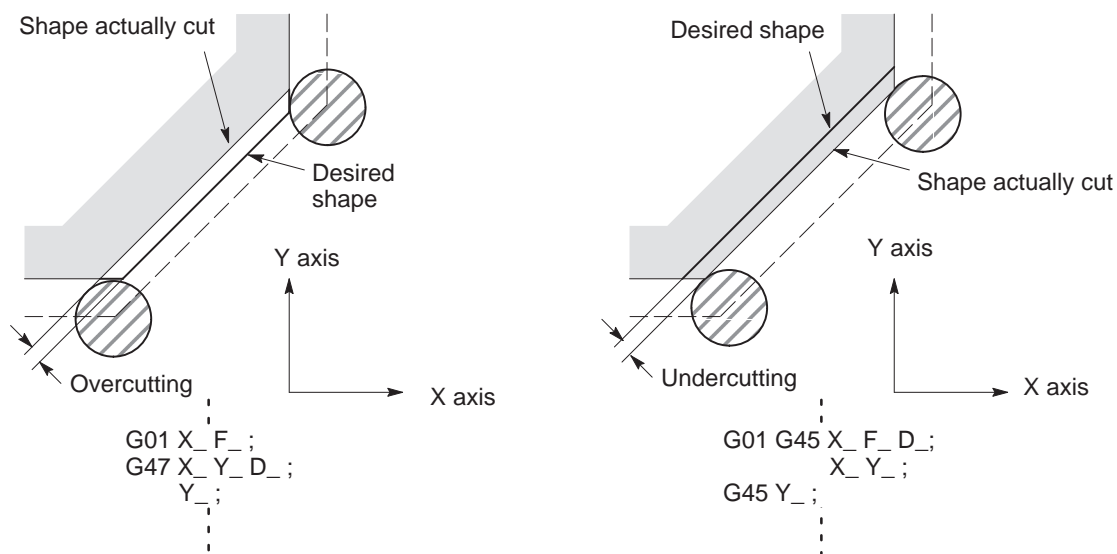
D0 always indicates a tool offset value of zero.

WARNING

When G45 to G48 is specified to n axes (n=1–4) simultaneously in a motion block, offset is applied to all n axes.

When the cutter is offset only for cutter radius or diameter in taper cutting, overcutting or undercutting occurs.

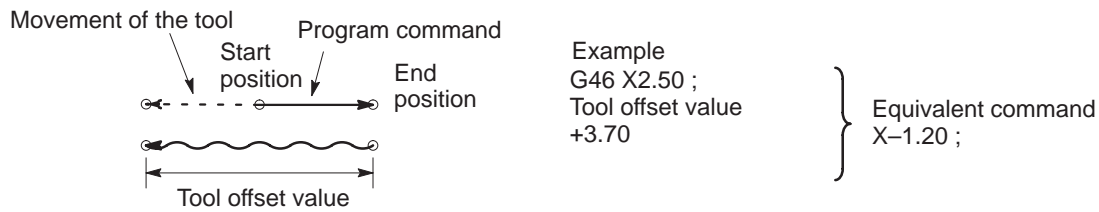
Therefore, use cutter compensation (G40 or G42) shown in section 14.4 or 14.5.

**CAUTION**

G45 to G48 (tool offset) must not be used in the G41 or G42 (cutter compensation) mode.

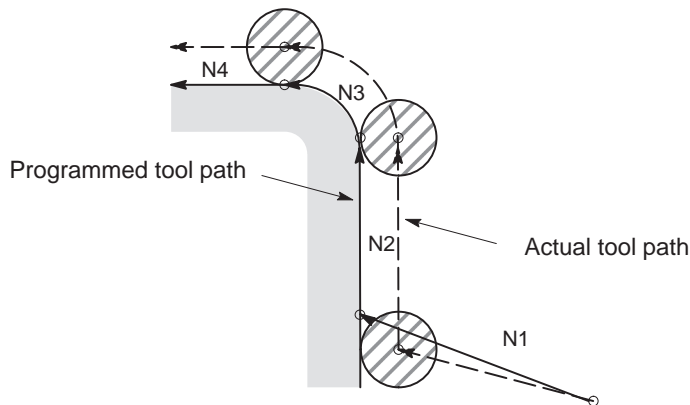
NOTE

- 1 When the specified direction is reversed by decrease as shown in the figure below, the tool moves in the opposite direction.



- 2 Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I, J and K, providing that the coordinate rotation be not specified at the same time. This function is provided for compatibility with the conventional CNC tape without any cutter compensation. The function should not be used when a new CNC program is prepared.

Tool offset for circular interpolation

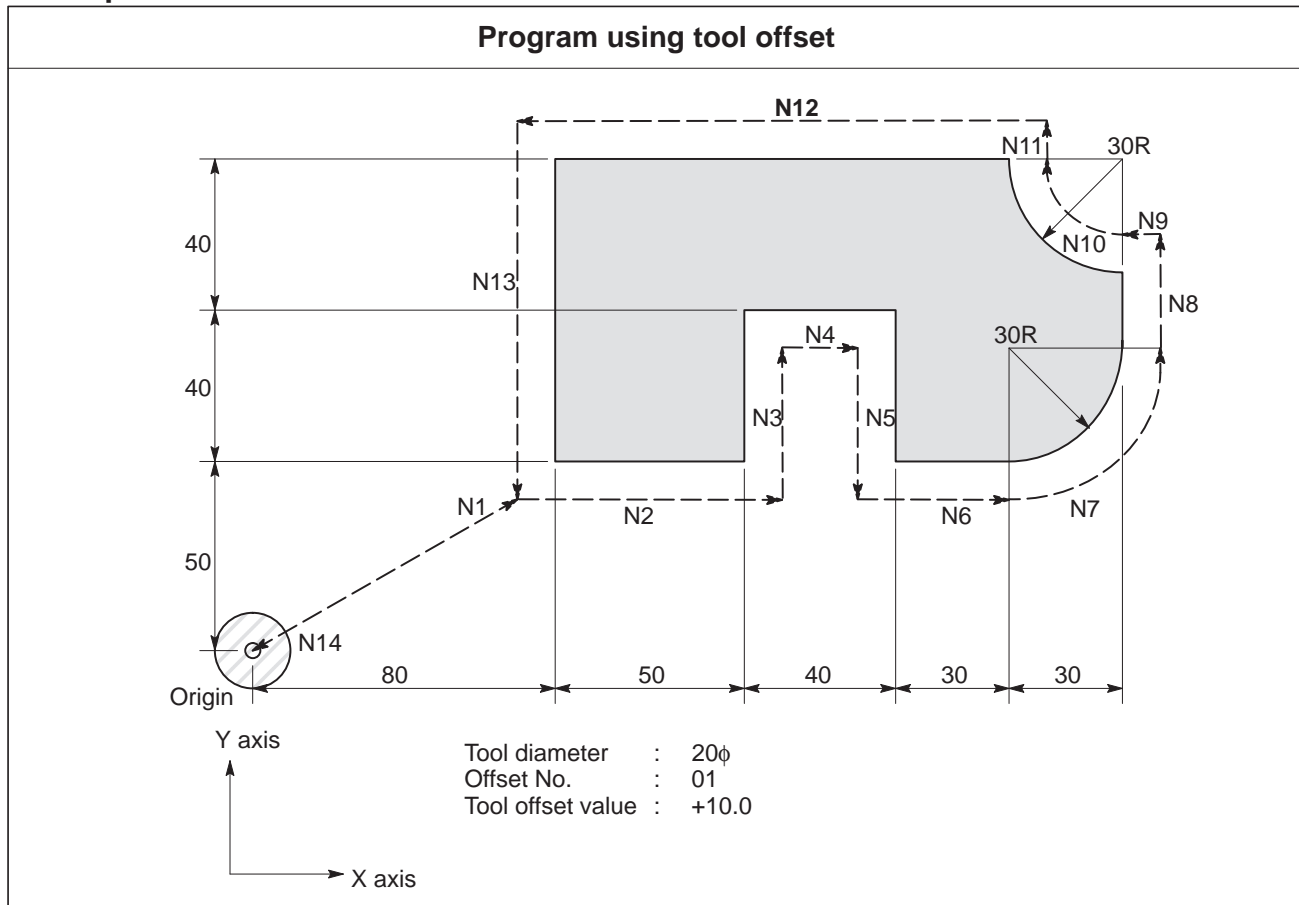


Program

```
N1 G46 G00 X_ Y_ D_ ;
N2 G45 G01 Y_ F_ ;
N3 G45 G03 X_ Y_ I_ ;
N4 G01 X_ ;
```

- 3 D code should be used in tool offset mode (G45 to G48). However, H code can be used because of compatibility with conventional CNC tape format. The H code must be used under tool length offset cancel (G49).
- 4 G45 to G48 are ignored in canned cycle mode. Perform tool offset by specifying G45 to G48 before entering canned cycle mode and cancel the offset after releasing the canned cycle mode.

Examples

**Program**

N1 G91 G46 G00 X80.0 Y50.0 D01 ;

N2 G47 G01 X50.0 F120.0 ;

N3 Y40.0 ;

N4 G48 X40.0 ;

N5 Y-40.0 ;

N6 G45 X30.0 ;

N7 G45 G03 X30.0 Y30.0 J30.0 ;

N8 G45 G01 Y20.0 ;

N9 G46 X0 ; Decreases toward the positive direction for movement amount "0". The tool moves in the -X direction by the offset value.

N10 G46 G02 X-30.0 Y30.0 J30.0 ;

N11 G45 G01 Y0 ; ... Increase toward the positive direction for movement amount "0". The tool moves in the +Y direction by the offset value.

N12 G47 X-120.0 ;

N13 G47 Y-80.0 ;

N14 G46 G00 X-80.0 Y-50.0 ;

14.4 CUTTER COMPENSATION B (G39 TO G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 14.4).

To make an offset as large as the radius of the tool, first create 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 points to the center of the tool.

If a linear interpolation, corner offset, 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 cutter compensation mode.

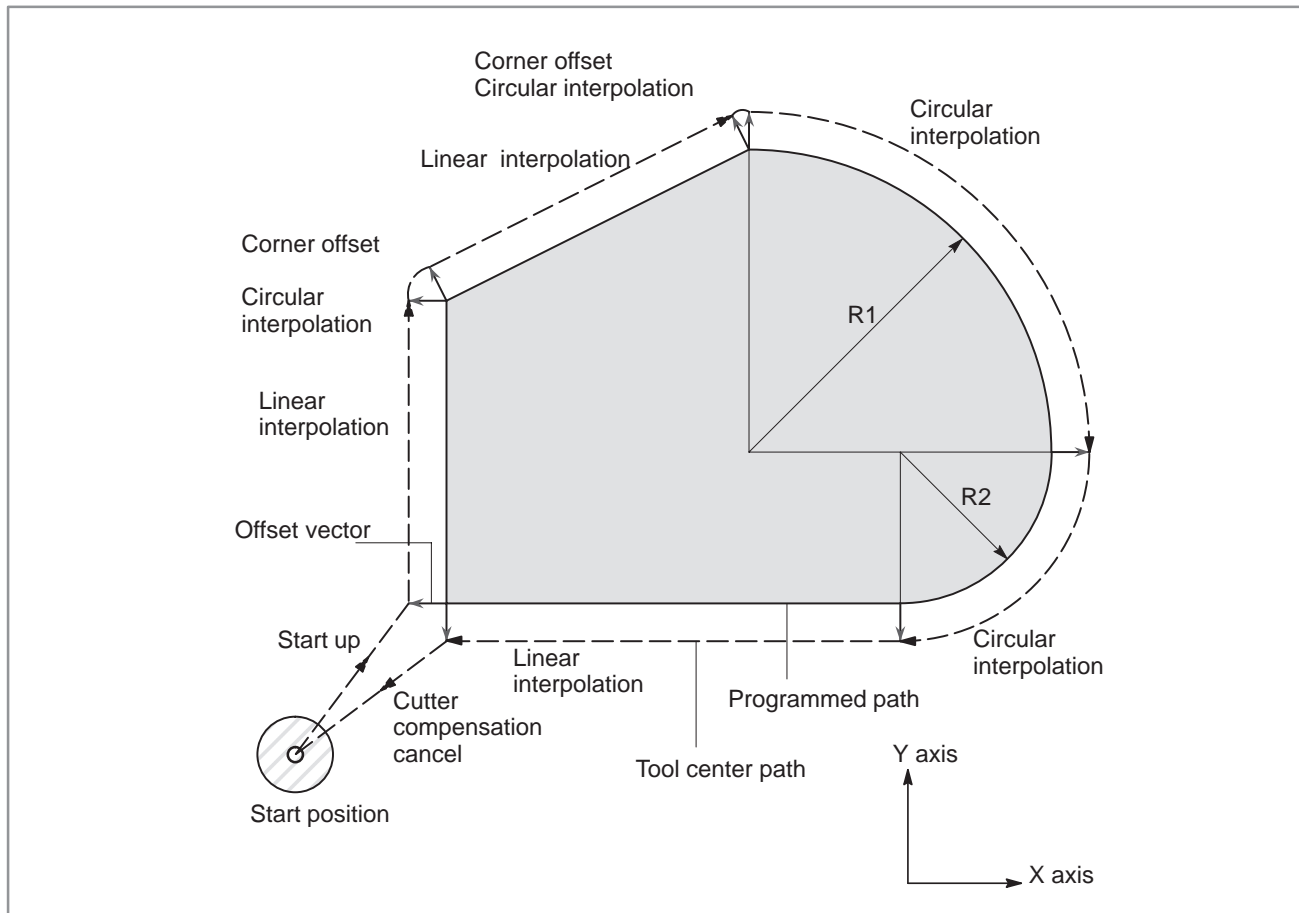


Fig.14.4 Outline of cutter compensation

To obtain the tool path appropriately offset by cutter compensation B, create the program as follows:

In an outer corner, corner offset circular interpolation is specified to direct the offset vector in the direction perpendicular to that of the tool movement. In an inner corner, an arc with a radius smaller than the offset radius is specified to prevent the tool from biting the outline of the workpiece.

Format

- **Start up
(Cutter compensation start)**

G00 (or G01) G41 (or G42) IP_ IR_ H_ ;

G41 : Cutter compensation left (Group 07)
G42 : Cutter compensation right (Group 07)
IP_ : Command for axis movement
IR_ : Incremental value from the end position. Perpendicular to the offset vector at the end position.
H_ : Code for specifying the cutter compensation value (1 to 3 digits)

- **Corner offset circular interpolation**

G39 IP_ (or IR_) ;

G39 : Corner offset circular interpolation (Group 00)
IP_ (or IR_) : Incremental value from the end position.
 Perpendicular to the offset vector at the end position.

- **Cutter compensation cancel**

G40 IP_ ;

G40 : Cutter compensation cancel (Group 07)
IP_ : Command for axis movement

- **Selection of the offset plane**

Offset plane	Command of the plane selection	IP_	IR_
XpYp	G17 ;	Xp_Yp_	I_J_
ZpXp	G18 ;	Xp_Zp_	I_K_
YpZp	G19 ;	Yp_Zp_	J_K_

Explanations

- **H code**

Specify the number assigned to a cutter compensation value with a 1– to 3–digit number after address H (H code) in the program. The H code can be specified in any position before the offset cancel mode is first switched to the cutter compensation mode. The H code need not be specified again unless the cutter compensation value needs to be changed.

Assign cutter compensation values to the H codes on the CRT/MDI panel. For the specification of the cutter compensation value, see Subsection 9.1.1 in the section on operation.

The table below shows the range in which the cutter compensation values can be specified.

Table 14.4 Valid range of cutter compensation values

Increment system	Metric input	inch input
IS-B	0 to $\pm 999.999\text{mm}$	0 to $\pm 99.9999\text{inch}$
IS-C	0 to $\pm 999.9999\text{mm}$	0 to $\pm 99.99999\text{inch}$

NOTE

The cutter compensation value corresponding to offset No.0, that is, H0 always gets 0. It is impossible to set H00 to any other cutter compensation value.

- **Offset plane selection and offset vector**

Cutter compensation is carried out in the plane determined by G17, G18 and G19 (G codes for plane selection.). This plane is called the offset plane. If the offset plane is not specified, G17 is assumed to be programmed.

Compensation is not executed for the coordinates of a position which is not in the specified plane. The programmed values are used as they are. In the sequel, what vector is created, what offset calculation is made, by an offset command, will be discussed on assumption that an XY plane is selected. This discussion applies also when another plane is selected.

The offset vector is cleared by reset.

After the power is turned on, the length of the offset vector is set to zero and the cutter compensation cancel mode is selected.

- **Transition from the offset cancel mode to the cutter compensation mode (Start up)**

NOTE

A move command mode at the time of change from the offset cancel mode to the cutter compensation mode, is positioning (G00) or linear interpolation (G01). The circular interpolation (G02, G03) cannot be used.

14.4.1 Cutter Compensation Left (G41)

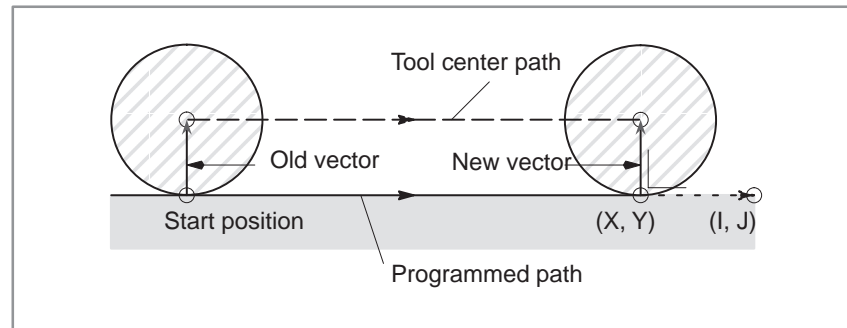
Explanations

- **G00 (positioning) or
G01 (linear interpolation)**

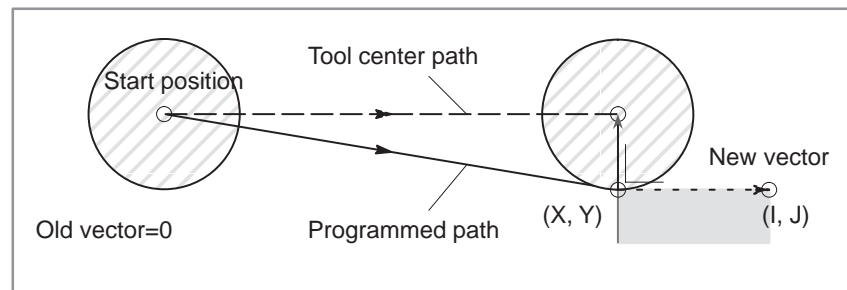
G41 offsets the tool towards the left of the workpiece as you see when you face in the same direction as the movement of the cutting tool.

G41 X_ Y_ I_ J_ H_ ;

Specifies a new vector to be created at right angles with the direction of (I, J) on the end point, and the tool center moves toward the point of the new vector from that of the old vector on the start point. (I, J) is expressed in an incremental value from the end point, and is significant only as a direction, and its amount is arbitrary.



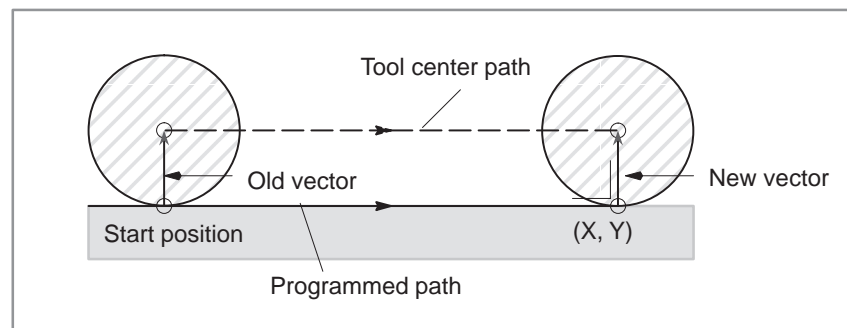
In case the old vector is 0, this command specifies the equipment to enter from the cancel mode into the cutter compensation mode. At this time, the offset number is specified by the H code.



Unless otherwise specified, (I, J) are assumed to be equal to (X, Y). When the following command is specified, a vector perpendicular to a line connecting the start position and position (X, Y) is created.

G41 X_ Y_ ;

If, however, G00 is specified, each axis moves independently at the rapid traverse rate.



- **G02, G03**
(Circular interpolation)

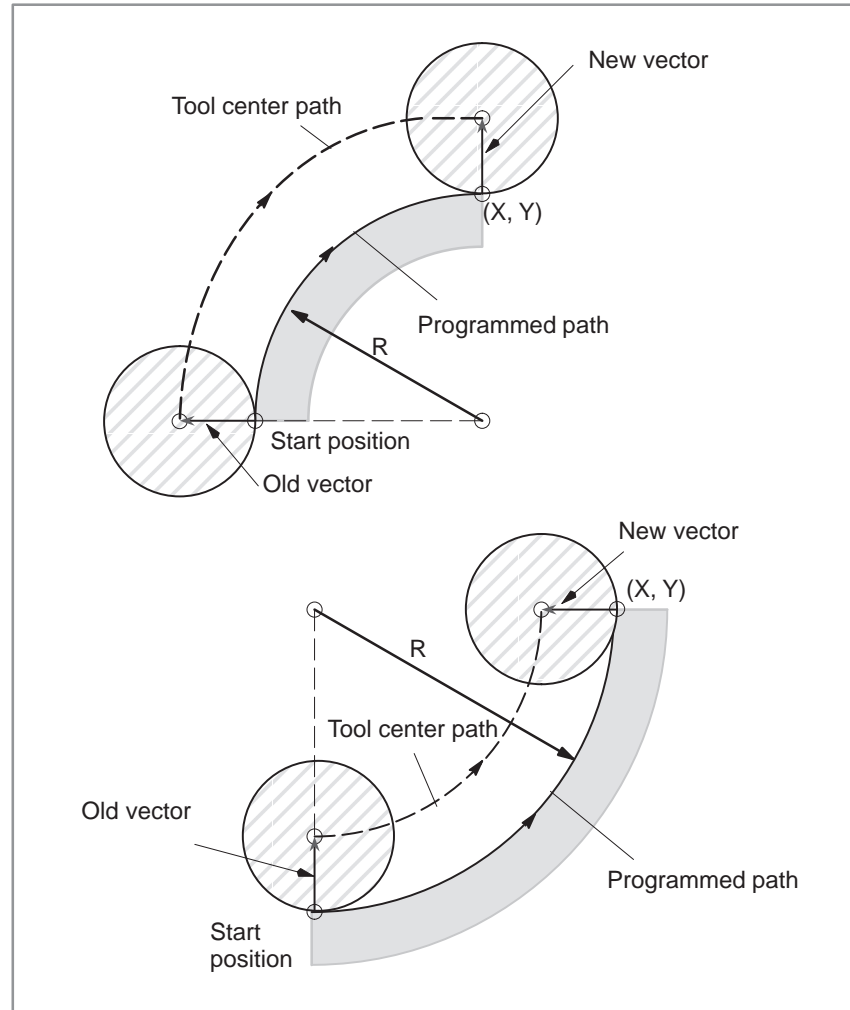
G41... ;

:

G02 (or G03) X_ Y_ R_ ;

Above command specifies a new vector to be created to the left looking toward the direction in which an arc advances on a line connecting the arc center and the arc end point, and the tool center to move along the arc advancing from the point of the old vector on the arc start point toward that of the new vector. This is, however, established on assumption the old vector is created correctly.

The offset vector is created toward the arc center or opposite direction against the arc center.



14.4.2 Cutter Compensation Right (G42)

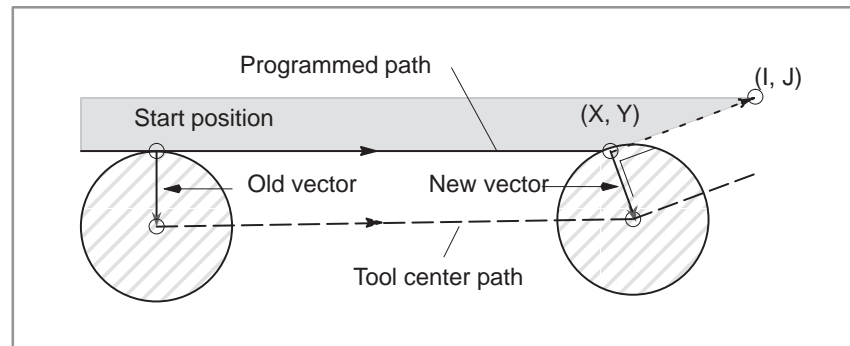
G42, contrary to G41, specifies a tool to be offset to the right of work piece looking toward the direction in which the tool advances.

G42 has the same function as G41, except that the directions of the vectors created by the commands are the opposite.

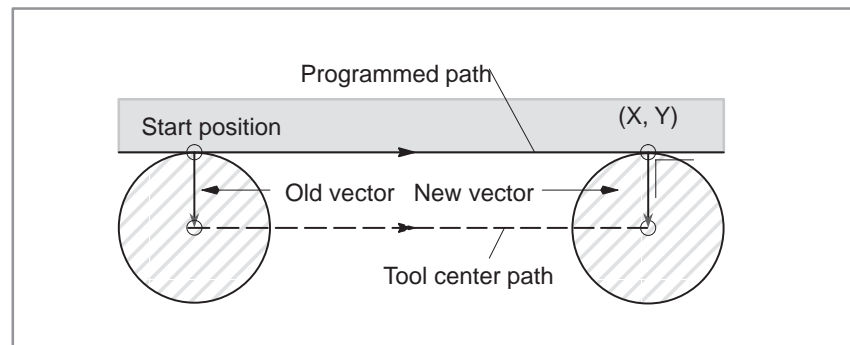
Explanations

- G00 (positioning) or G01 (linear interpolation)

G42 X_ Y_ I_ J_ H_ ;



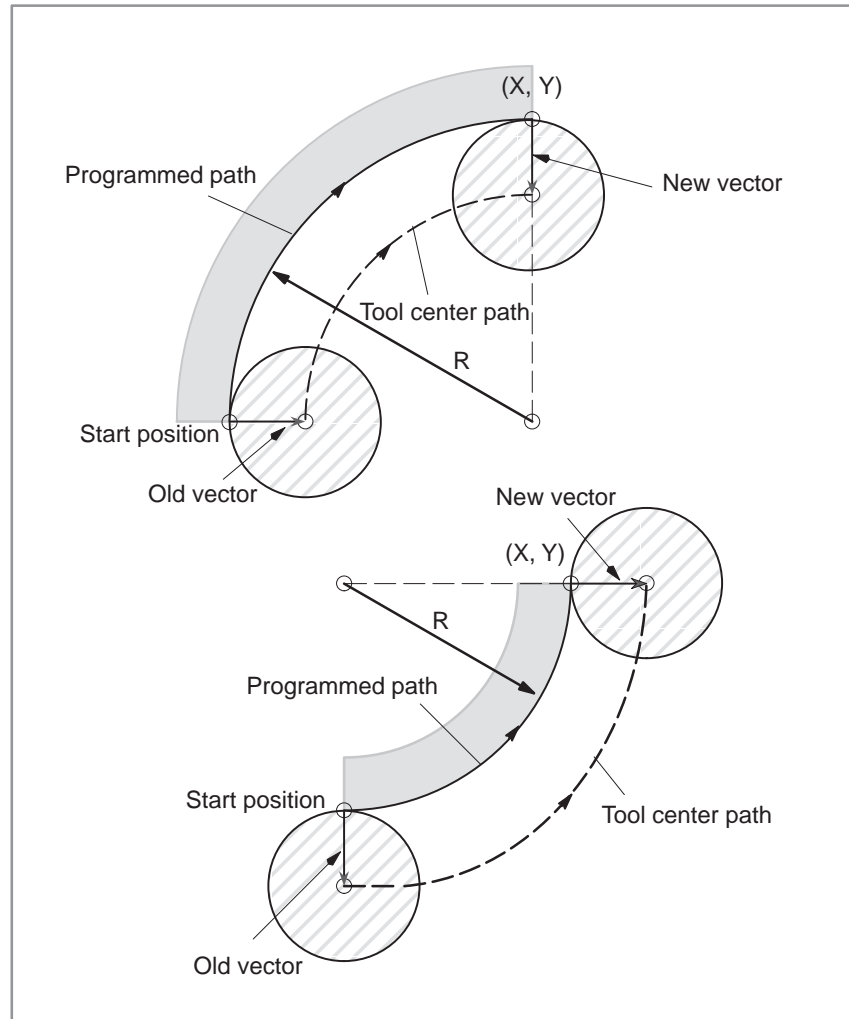
G42 X_ Y_ ;



In the case of G00, however, each axis moves independently at the rapid traverse rate.

- **G02 or G03**
(Circular interpolation)

G42... ;
:
G02 (or G03) X_ Y_ R_;

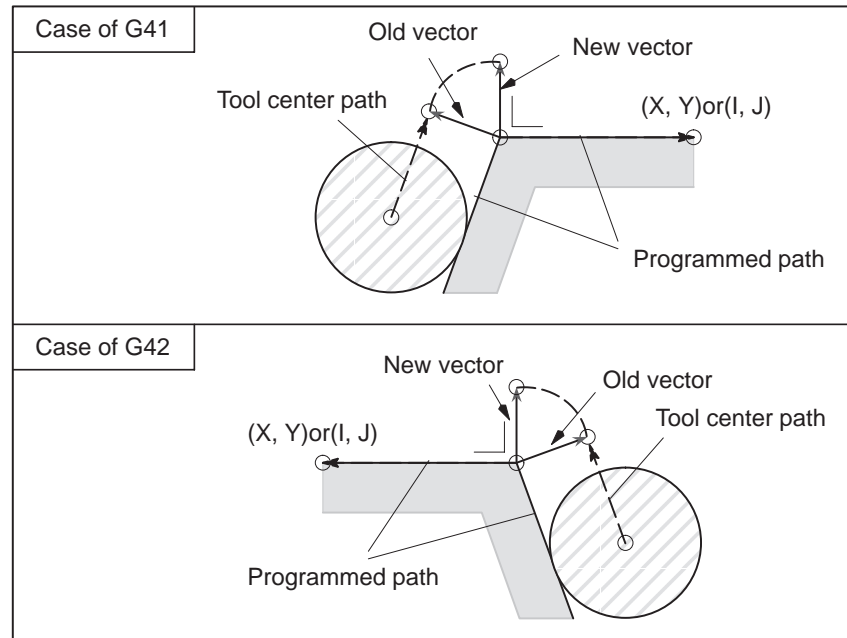


14.4.3 Corner Offset Circular Interpolation (G39)

When the following command is specified in the G01, G02, or G03 mode, corner offset circular interpolation can be executed with respect to the radius of the tool.

G39 X_ Y_ ; or G39 I_ J_ ;

A new vector is created to the left (G41) or to the right (G42) looking toward (X, Y) from the end point at right angles therewith, and the tool moves along the arc from the point of the old vector toward that of the new vector. (X, Y) is expressed in a value according to the G90/G91 respectively. (I, J) is expressed in an incremental value from the end point.

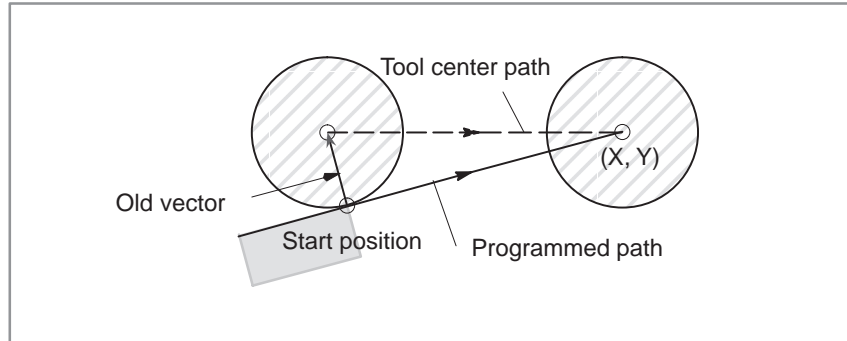


This command can be given in offset mode, that is, only when G41 or G42 has already been specified. Whether the arc is to turn clockwise or counterclockwise, is defined by G41 or G42, respectively. This command is not modal, and executes circular interpolation, whatever the G function of group 01 may be. The G function of group 01 remains even though this command is specified.

14.4.4 Cutter Compensation Cancel (G40)

When the following command is specified in the G00 or G01 mode, the tool moves from the head of the old vector at the start position to the end position (X, Y). In the G01 mode, the tool moves linearly. In the G00 mode, rapid traverse is carried out along each axis.

G40 X_ Y_ ;



This command changes the mode of the equipment from the cutter compensation mode to the cancel mode.

When only G40; is specified, and X_ Y_ is not specified, the tool moves by the old vector amount in the opposite direction.

NOTE

Cutter compensation cannot be canceled in the circular interpolation (G02, G03) mode.

14.4.5 Switch between Cutter Compensation Left and Cutter Compensation Right

The offset direction is switched from left to right, or from right to left generally through the offset cancel mode, but can be switched not through it only in positioning (G00) or linear interpolation (G01). In this case, the tool path is as shown in Fig 14.4.5.

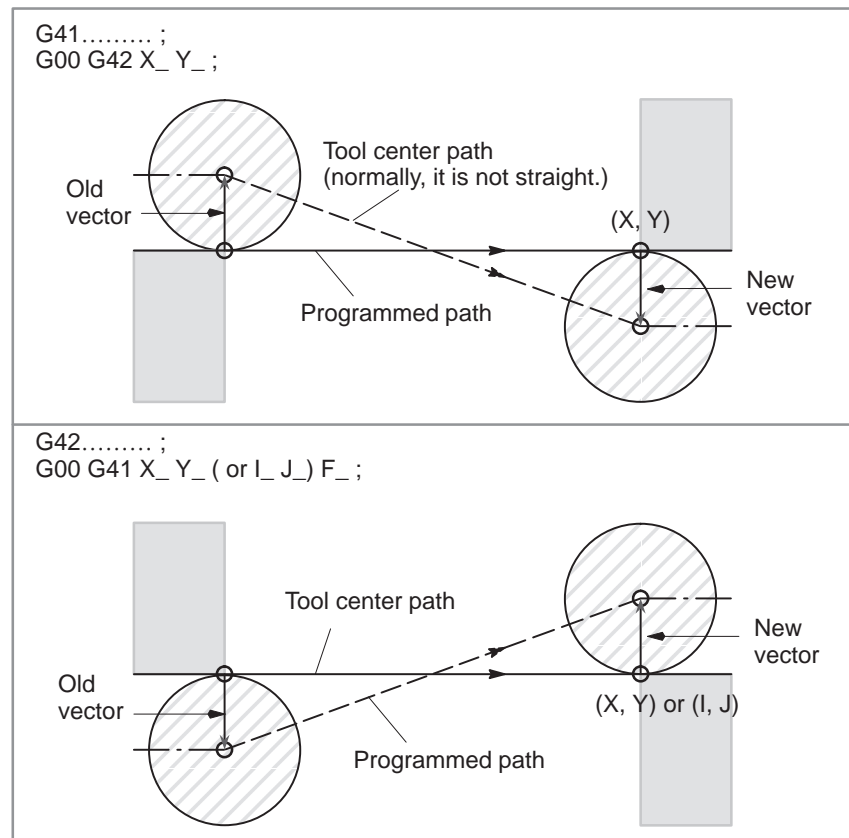


Fig.14.4.5 Switching the direction of cutter compensation

14.4.6 Change of the Cutter Compensation Value

The offset amount is changed generally when the tool is changed in the offset cancel mode, but can be changed in the offset mode only in positioning (G00) or linear interpolation (G01).

Program as described below:

G00 (or G01) X_ Y_ H_ ; (H_ indicates the number of a new cutter compensation value.)

Fig. 14.4.6 shows how the tool is moved when the change in compensation is specified.

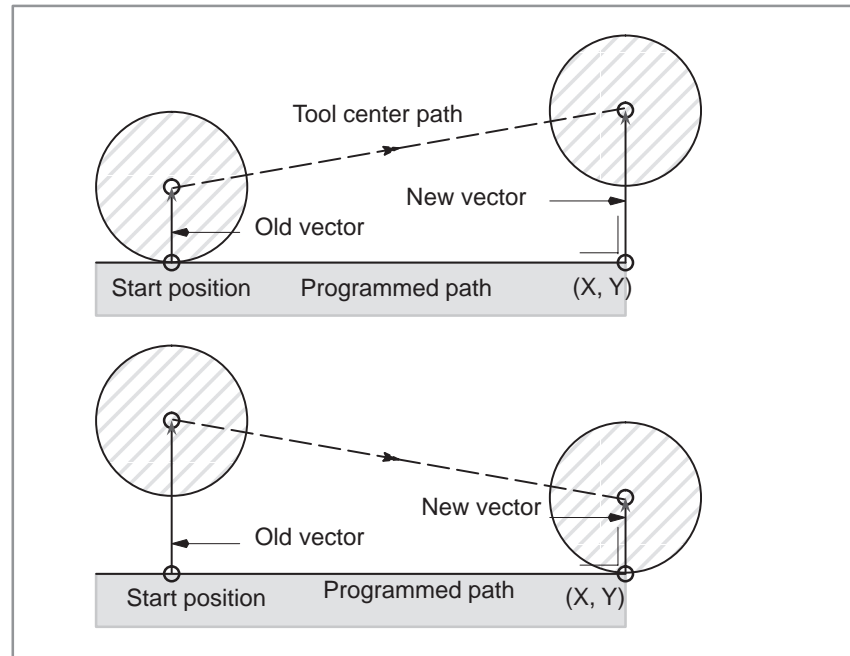


Fig.14.4.6 Change of the cutter compensation value in offset mode

14.4.7 Positive/Negative Cutter Compensation Value and Tool Center Path

If the tool compensation value is made negative ($-$), it is equal that G41 and G42 are replaced with each other in the process sheet. Consequently, if the tool center is passing around the outside of the workbench it will pass around the inside thereof, and vice versa.

Fig. 14.4.7 shows one example. Generally speaking, the cutter compensation value shall be programmed to be positive ($+$). When a tool path is programmed as shown in [1], if the cutter compensation value is made negative ($-$), the tool center moves as shown in [2].

If the cutter compensation value is changed to a negative value when tool path [2] shown in Fig. 14.4.7 is programmed, the tool follows tool path [1] shown in the same figure.

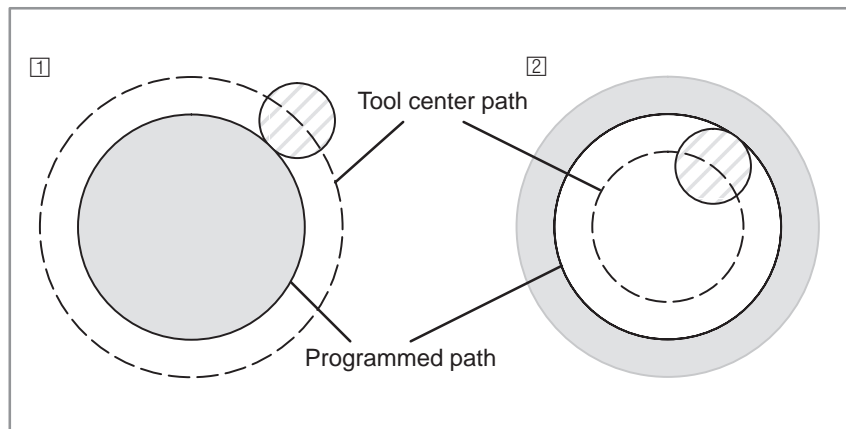


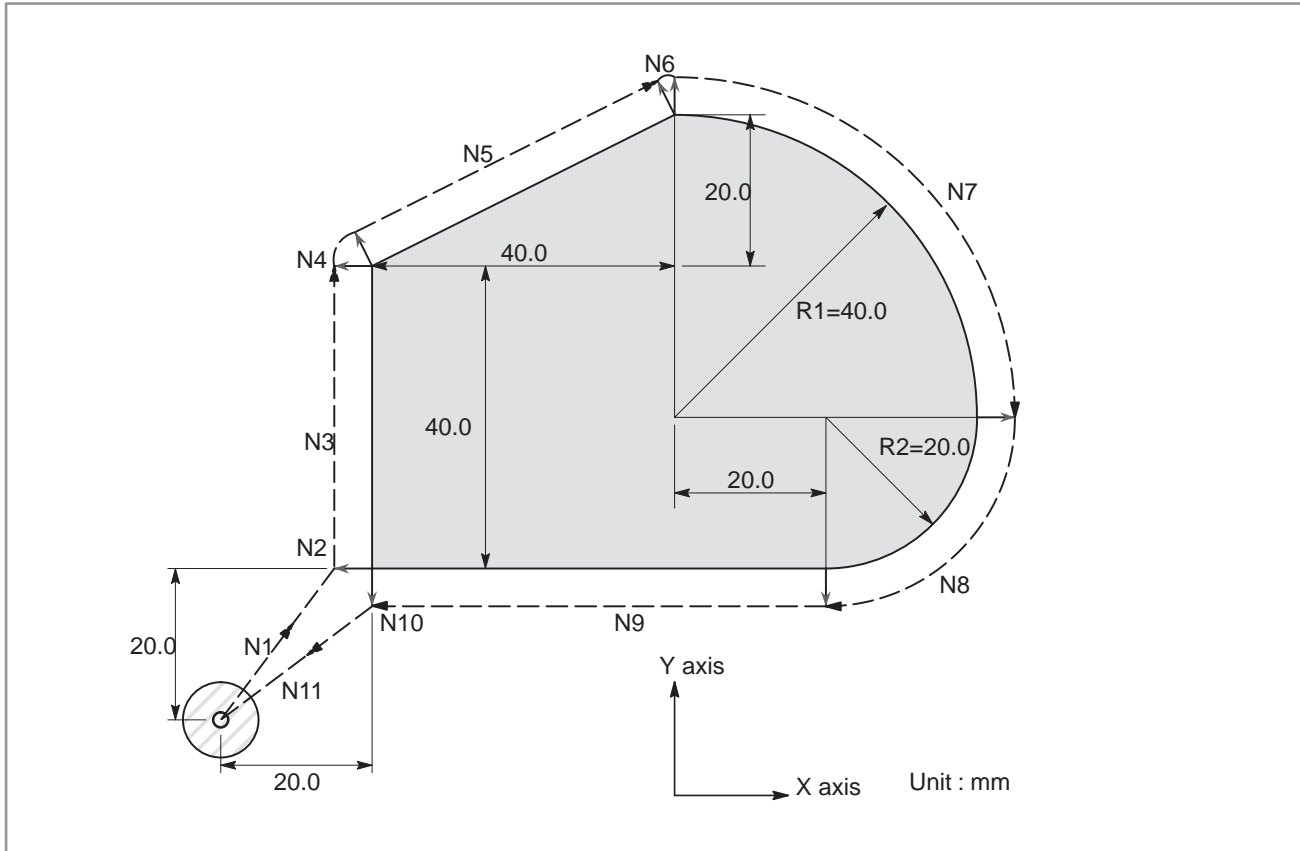
Fig.14.4.7 Tool center paths when positive and negative cutter compensation values are specified

For a cornered figure (involved in corner circular interpolation) in general, the cutter compensation value naturally cannot be made negative ($-$) to cut the inside. In order to cut the inside corner of a cornered figure, an arc with an appropriate radius must be inserted there to provide smooth cutting.

WARNING

If the tool length offset is commanded during cutter compensation, the offset amount of cutter compensation is also regarded to have been changed.

Examples



N1G91 G17 G00 G41 X20.0 Y20.0 J40.0 H08 ;

N2G01 Z-25.0 F100 ;

N3Y40.0 F250 ;

N4G39 I40.0 J20.0 ;

N5X40.0 Y20.0 ;

N6G39 I40.0 ;

N7G02 X40.0 Y-40.0 R40.0 ;

N8X-20.0 Y-20.0 R20.0 ;

N9G01 X-60.0 ;

N10 G00 Z25.0 ;

N11 G40 X-20.0 Y-20.0 M02 ;

(H08 is a tool offset number, and the cutter radius value should be stored in the memory corresponding to this number).

14.5 OVERVIEW OF CUTTER COMPENSATION C (G40 TO G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 14.5 (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 position at the end of machining, cancel the cutter compensation mode.

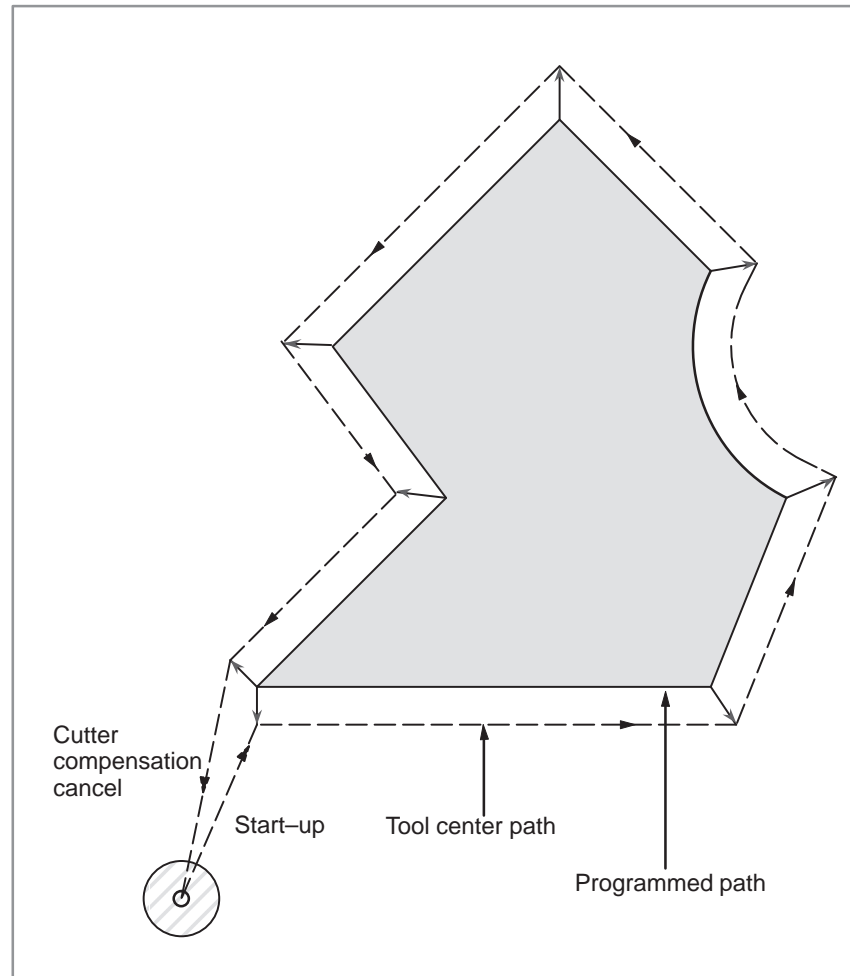


Fig.14.5 (a) Outline of cutter compensation C

In cutter compensation C, an appropriately offset tool path can be obtained for both the inner and outer corners simply by specifying the offset direction. Unlike cutter compensation B, programs can easily be created without having to consider the difference between the outer and inner corners.

Format

- **Start up
(Tool compensation start)**

G00 (or G01) G41 (or G42) IP_ H_ ;

G41 : Cutter compensation left (Group07)
G42 : Cutter compensation right (Group07)
IP_ : Command for axis movement
H_ : Code for specifying as the cutter compensation value(1–3digits)
(H code)

- **Cutter compensation
cancel
(offset mode cancel)**

G40 IP_ ;

G40 : Cutter 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_

Explanations

- **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 cutter compensation command (G41 or G42, nonzero dimension words in the offset plane, and H code other than H0) 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 34 occurs.

When processing the start-up block and subsequent blocks, the CNC prereads two blocks. The second preread block is not indicated.

- **Offset mode**

In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03). If two or more blocks that do not move the tool (miscellaneous function, dwell, etc.) are processed in the offset mode, the tool will make either an excessive or insufficient cut. If the offset plane is switched in the offset mode, alarm 37 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 equipment 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 cutter compensation.**

When performing offset cancel, circular arc commands (G02 and G03) are not available. If a circular arc is commanded, an alarm (No. 034) is generated and the tool stops.

In the offset cancel, the control executes the instructions in that block and the block in the cutter 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 cutter compensation.

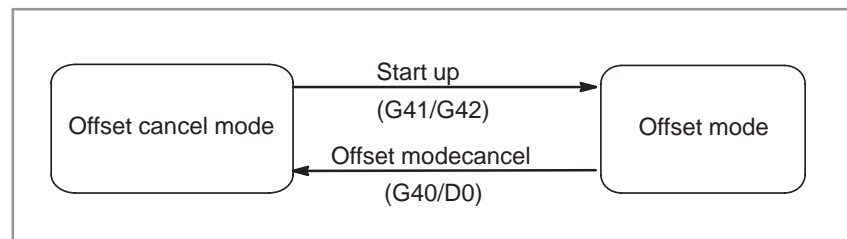


Fig.14.5 (b) Changing the offset mode

- **Change of the Cutter compensation value**

In general, the cutter compensation value shall be changed in the cancel mode, when changing tools. If the cutter compensation value is changed in offset mode, the vector at the end point of the block is calculated for the new cutter compensation value.

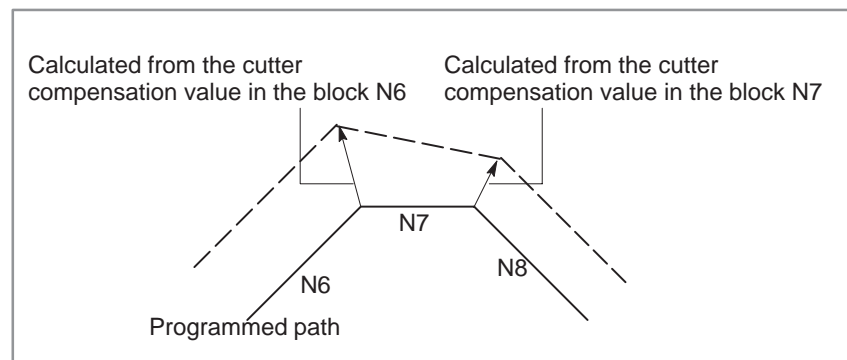


Fig.14.5 (c) Changing the cutter compensation value

- **Positive/negative cutter compensation value and tool center path**

If the offset amount 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.

The figure below shows one example. Generally, the offset amount is programmed to be positive (+).

When a tool path is programmed as in (1), if the offset amount is made negative (–), the tool center moves as in (2), and vice versa. Consequently, the same tape permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the offset amount. Applicable if start-up and cancel is A type. (See subsec. 14.6.2 and 14.6.4)

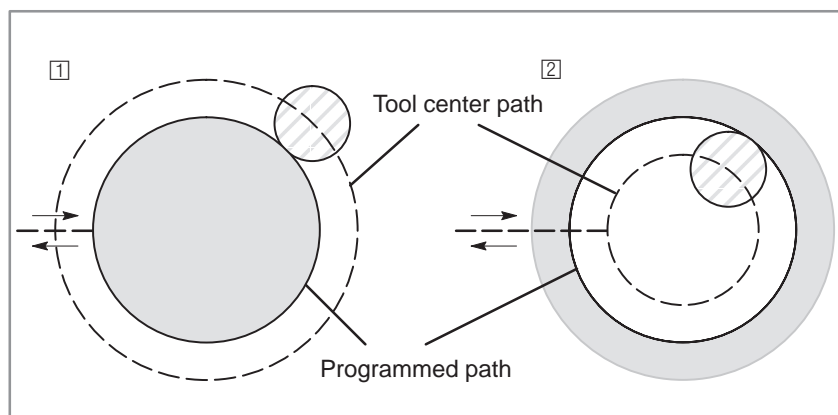


Fig.14.5 (d) Tool center paths when positive and negative cutter compensation values are specified

- **Cutter compensation value setting**

Assign a cutter compensation values to the H codes on the CRT/MDI panel. The table below shows the range in which cutter compensation values can be specified.

Increment system	Metric input	inch input
IS-B	0 to $\pm 999.999\text{mm}$	0 to $\pm 99.9999\text{inch}$
IS-C	0 to $\pm 999.9999\text{mm}$	0 to $\pm 99.99999\text{inch}$

NOTE

- 1 The cutter compensation value corresponding to offset No. 0, that is, H0 always means 0. It is impossible to set H0 to any other offset amount.
- 2 Cutter compensation C can be specified by D code with parameter (No. 036 #6) set to 1.

- **Offset vector**

The offset vector is the two dimensional vector that is equal to the cutter compensation value assigned by H 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 cutter compensation value**

Specify a cutter compensation value with a number assigned to it. The number consists of 1 to 3 digits after address H (H code). The H code is valid until another H code is specified. The H code is used to specify the tool offset value as well as the cutter compensation value.

When an H code for cutter compensation is specified during tool length compensation, the amount of tool length compensation remains as is. When an H code for tool length compensation is specified during cutter compensation, however, the cutter compensation, as well as tool length compensation, is changed. Do not modify the tool length compensation in cutter compensation mode.

The amount of tool length compensation and that of cutter compensation can be separately specified with the H and D codes by setting bit 6 of parameter No. 036 appropriately. Only the tool length compensation can be changed in cutter compensation. Even in this case, the tool length compensation is equal to the cutter compensation. That is, the setting of H10 is equal to that of D10.

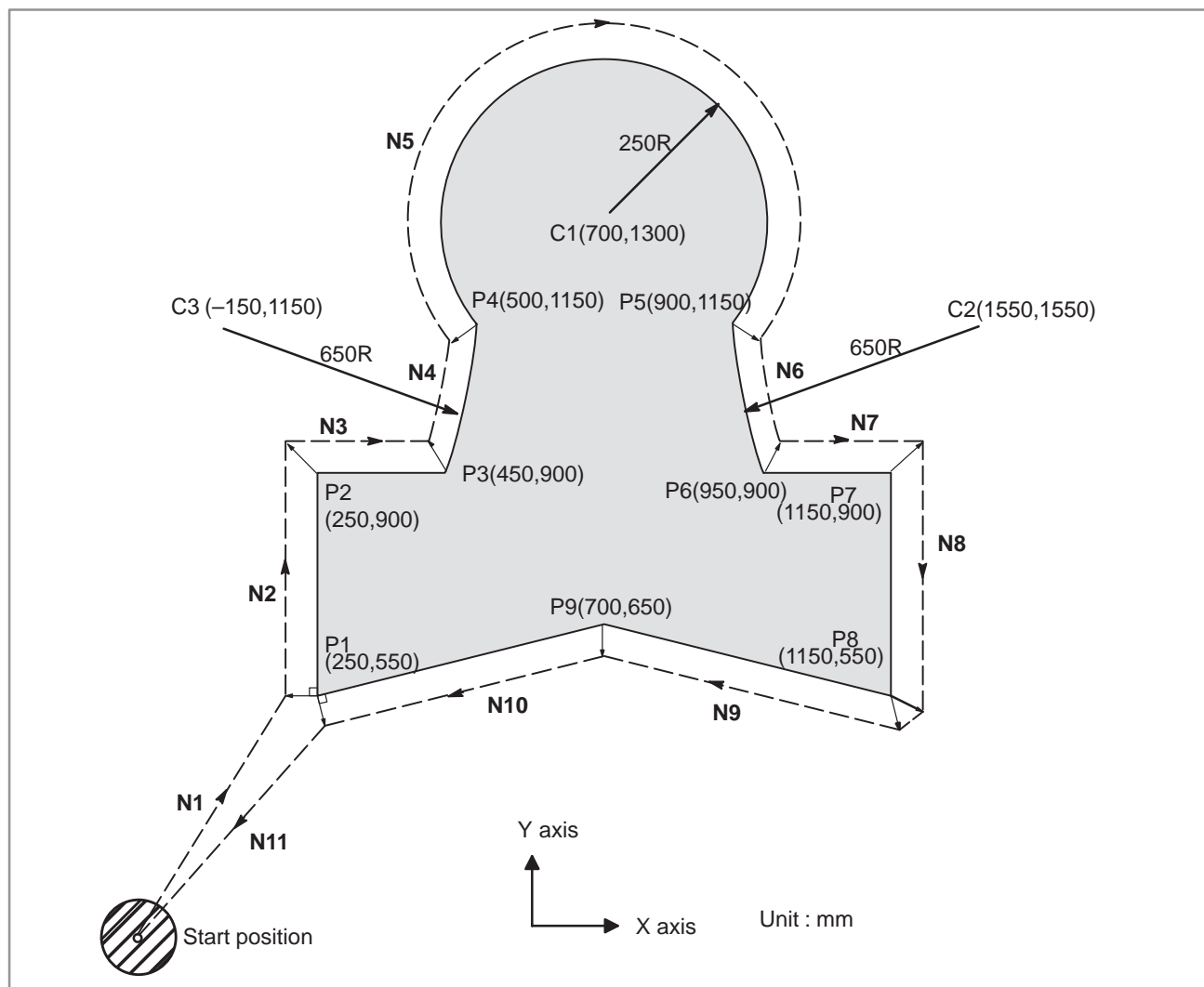
- **Plane selection and vector**

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 alarm (No. 37) is displayed and the machine is stopped.

Examples



G92 X0 Y0 Z0 ; Specifies absolute coordinates.

The tool is positioned at the start position (X0, Y0, Z0).

N1 G90 G17 G00 G41 H07 X250.0 Y550.0 ; Starts cutter 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 position (X0, Y0, Z0).

14.6

DETAILS OF CUTTER COMPENSATION C

This section provides a detailed explanation of the movement of the tool for cutter compensation C outlined in Section 14.5.

This section consists of the following subsections:

14.6.1 General

14.6.2 Tool Movement in Start-up

14.6.3 Tool Movement in Offset Mode

14.6.4 Tool Movement in Offset Mode Cancel

14.6.5 Interference Check

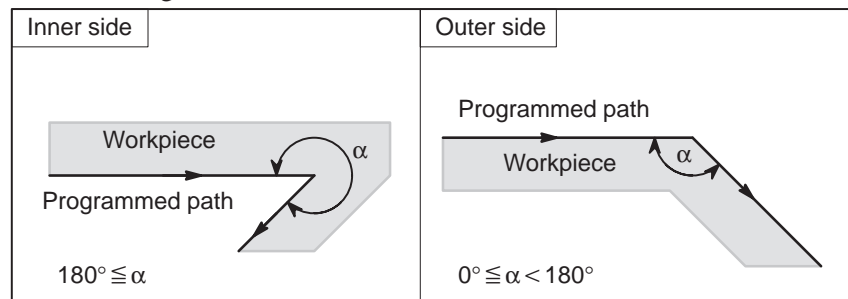
14.6.6 Over cutting by Cutter Compensation

14.6.7 Input command from MDI

14.6.1 General

• Inner side and outer side

When an angle of intersection created by tool paths specified with move commands for two blocks 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.”



• Meaning of symbols

The following symbols are used in subsequent figures:

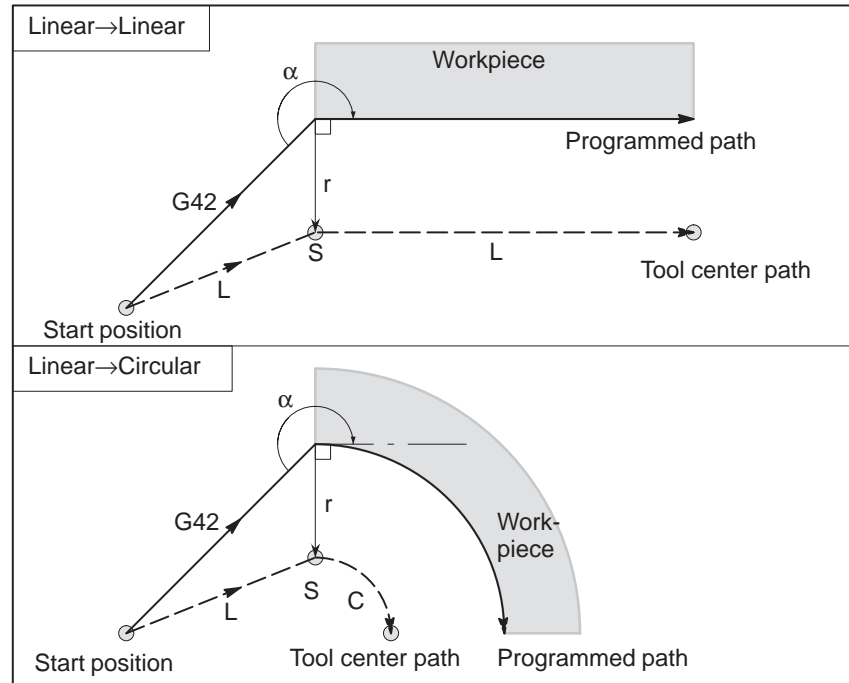
- *S* indicates a position at which a single block is executed once.
- *SS* indicates a position at which a single block is executed twice.
- *SSS* indicates a position at which a single block is executed three times.
- *L* indicates that the tool moves along a straight line.
- *C* indicates that the tool moves along an arc.
- *r* indicates the cutter 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*.
- \bigcirc indicates the center of the tool.

14.6.2 Tool Movement in Start-up

When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start-up):

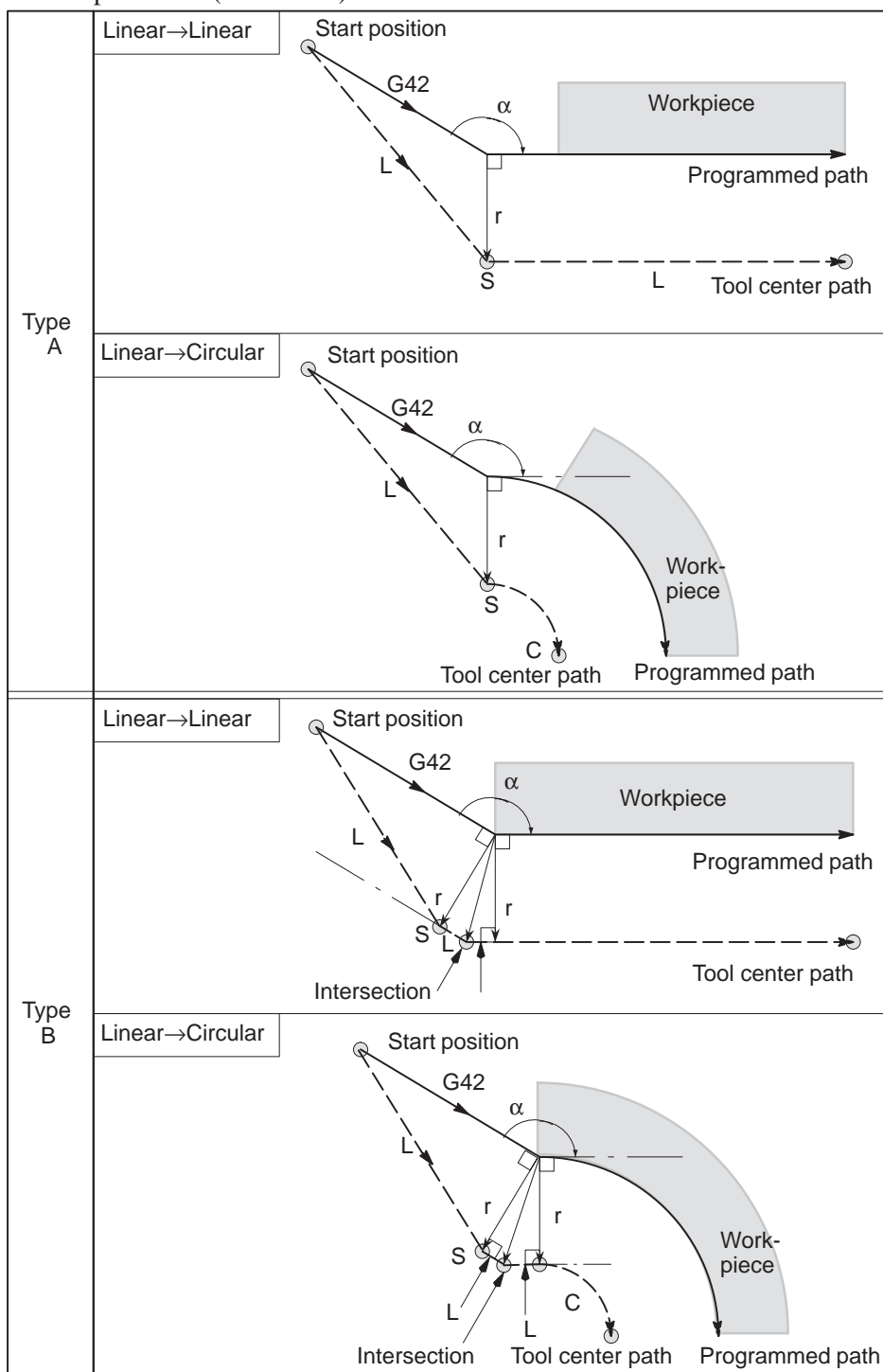
Explanations

- Tool movement around an inner side of a corner ($180^\circ \leq \alpha$)



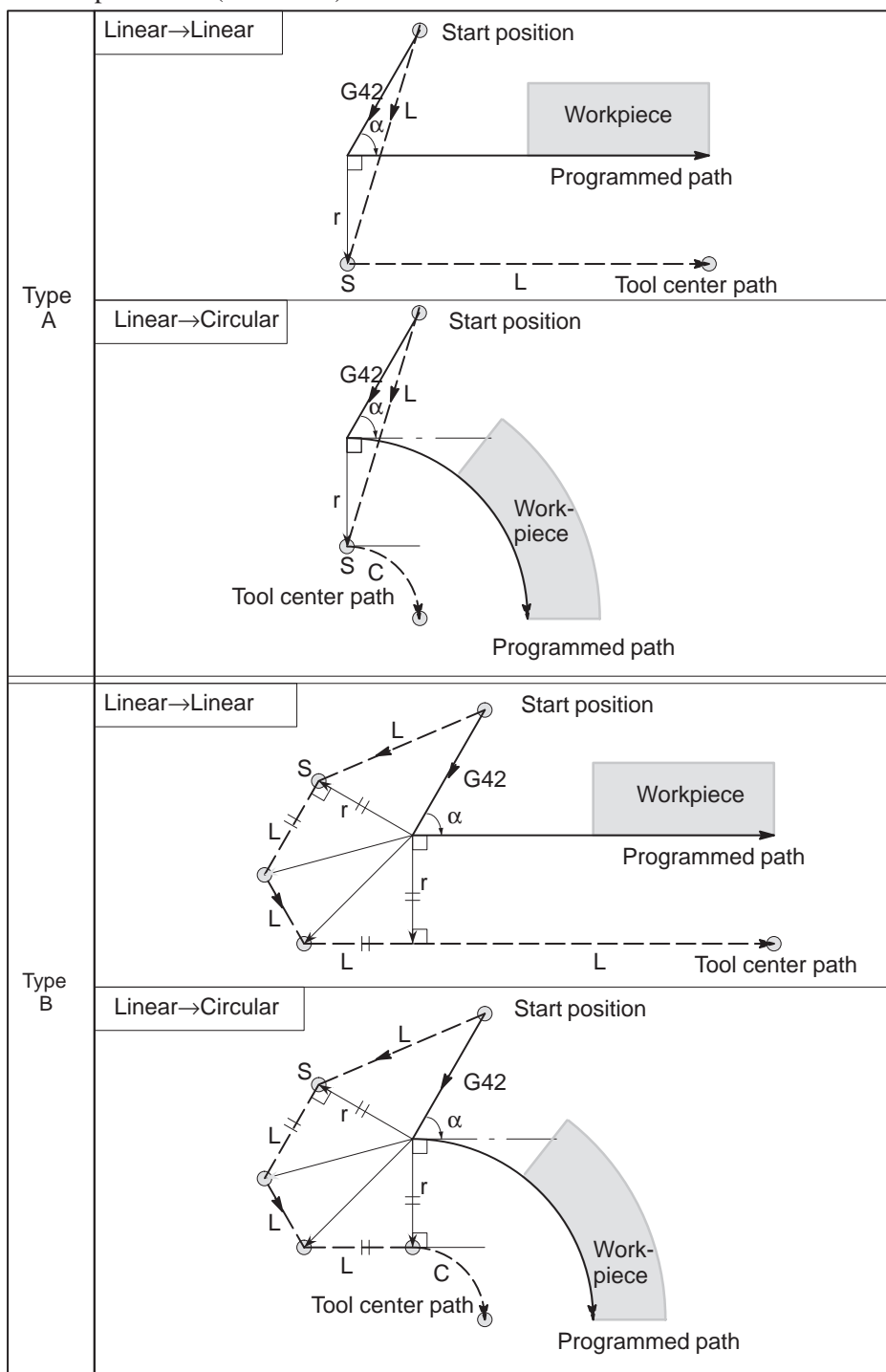
• **Tool movement around the outside of a corner at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)**

Tool path in start-up has two types A and B, and they are selected by parameter (No. 016#2).

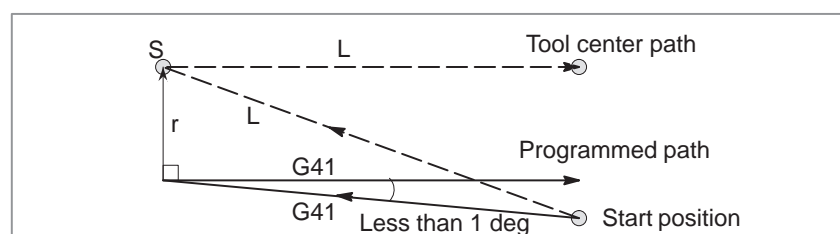


- Tool movement around the outside of an acute angle ($\alpha < 90^\circ$)

Tool path in start-up has two types A and B, and they are selected by parameter (No.016#2).



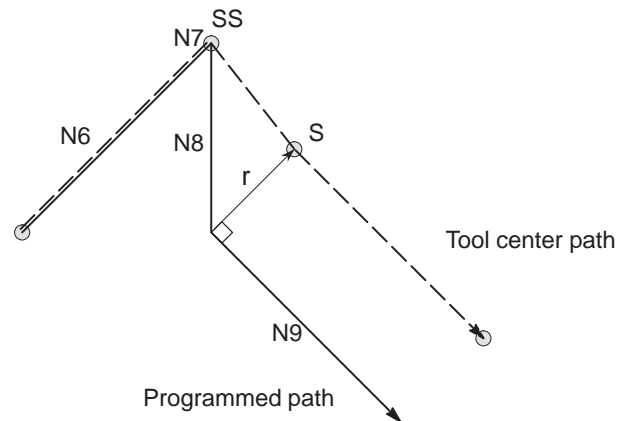
- Tool movement around the outside linear→linear at an acute angle less than 1 degree ($\alpha < 1^\circ$)



- **A block without tool movement specified at start-up**

If the command is specified at start-up, the offset vector is not created.

```
G91 G40 ... ;  
:  
N6 X100.0 Y100.0 ;  
N7 G41 X0 ;  
N8 Y-100.0 ;  
N9 Y-100.0 X100.0 ;
```

**NOTE**

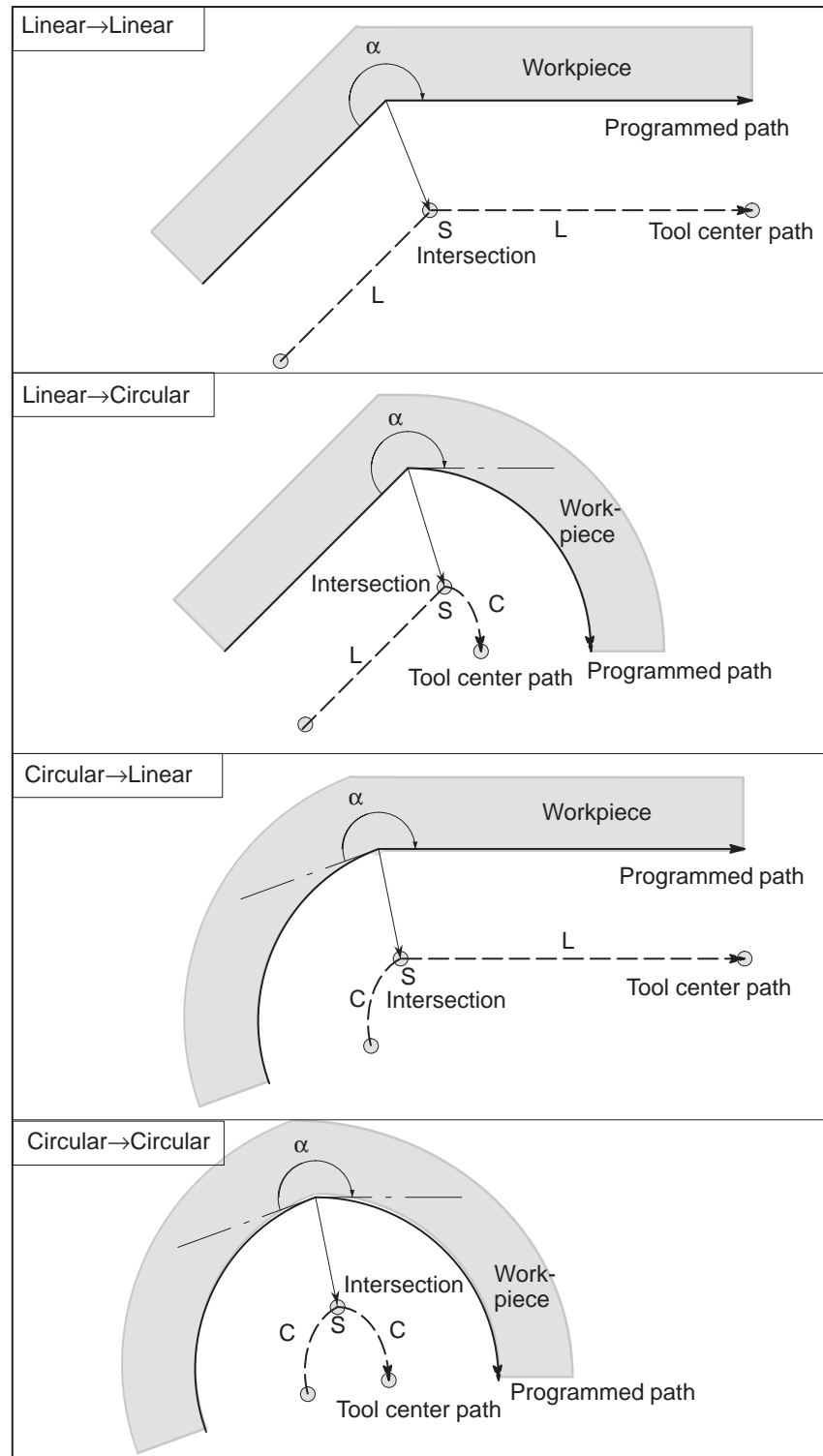
For the definition of blocks that do not move the tool, see Subsection 14.6.3.

14.6.3 Tool Movement in Offset Mode

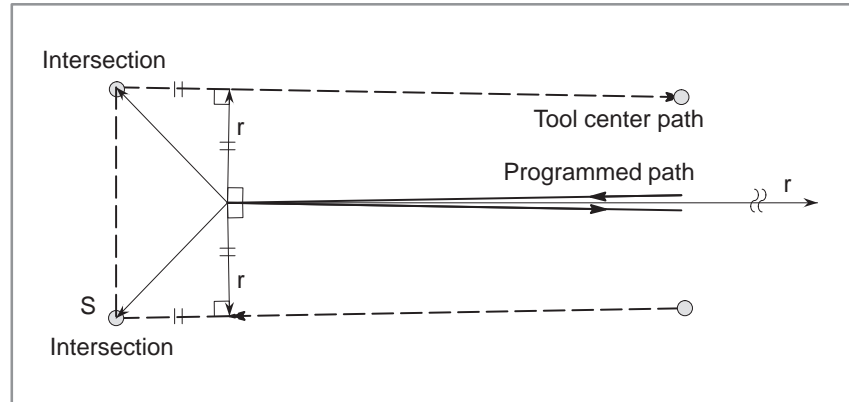
In the offset mode, the tool moves as illustrated below:

Explanations

- Tool movement around the inside of a corner ($180^\circ \leq \alpha$)

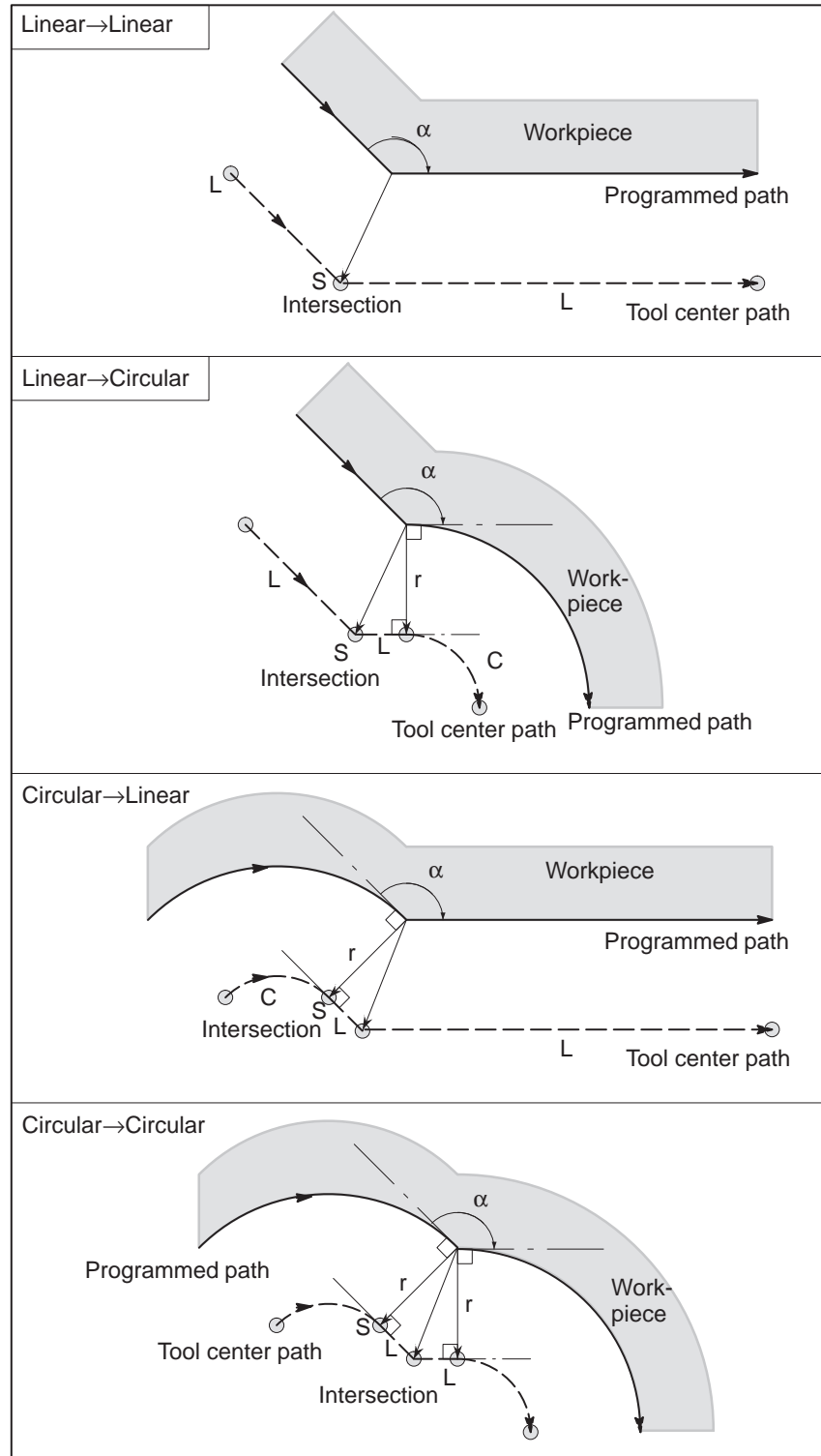


- **Tool movement around the inside ($\alpha < 1^\circ$) with an abnormally long vector, linear \rightarrow linear**

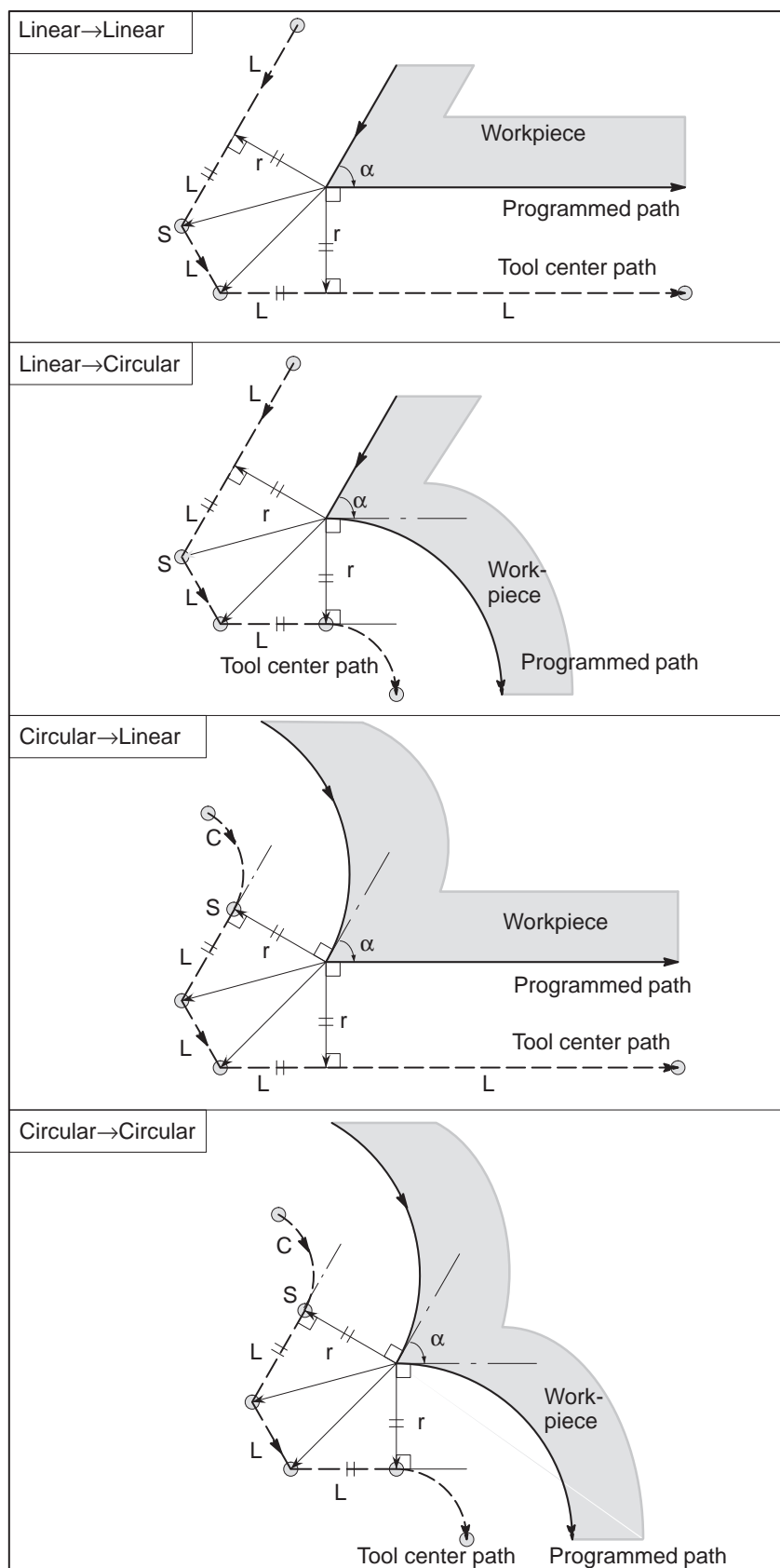


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

- Tool movement around the outside corner at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)



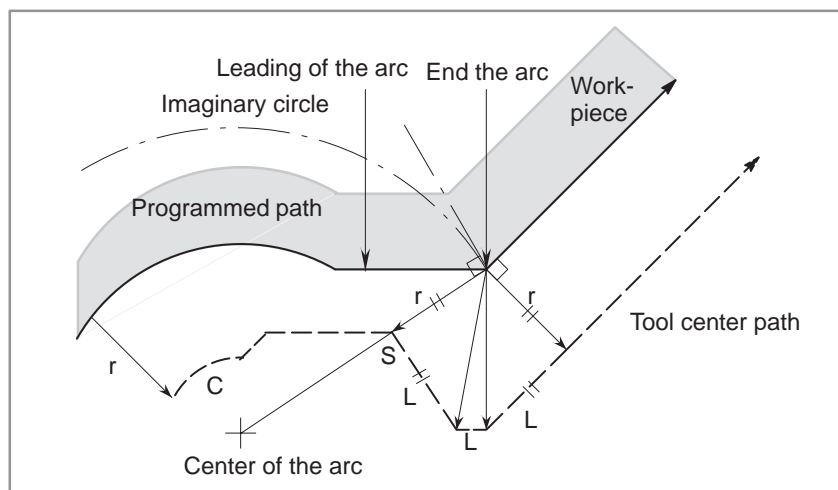
- Tool movement around the outside corner at an acute angle ($\alpha < 90^\circ$)



- When it is exceptional

End position for the arc is not on the arc

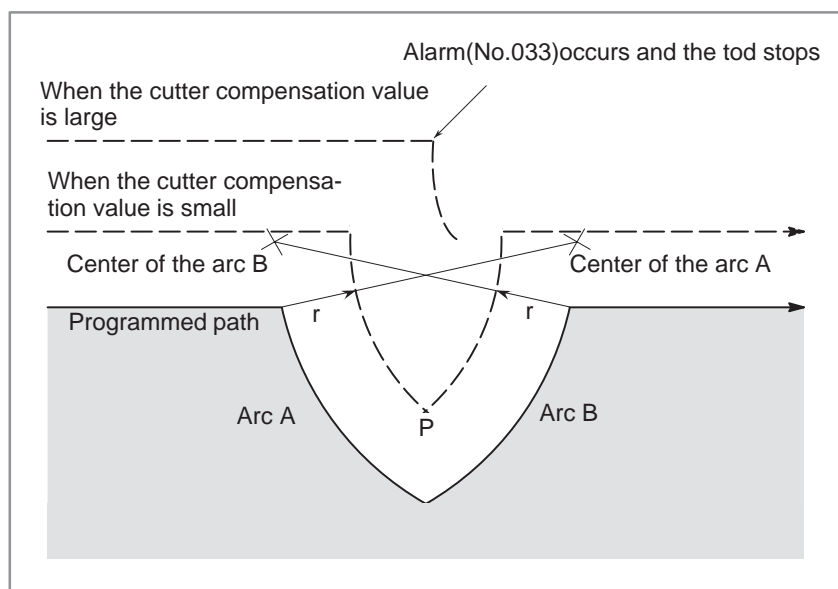
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system creates a vector and carries out compensation. The resulting tool center path is different from that created by applying cutter compensation to the programmed path in which the line leading to the arc is considered straight.



The same description applies to tool movement between two circular paths.

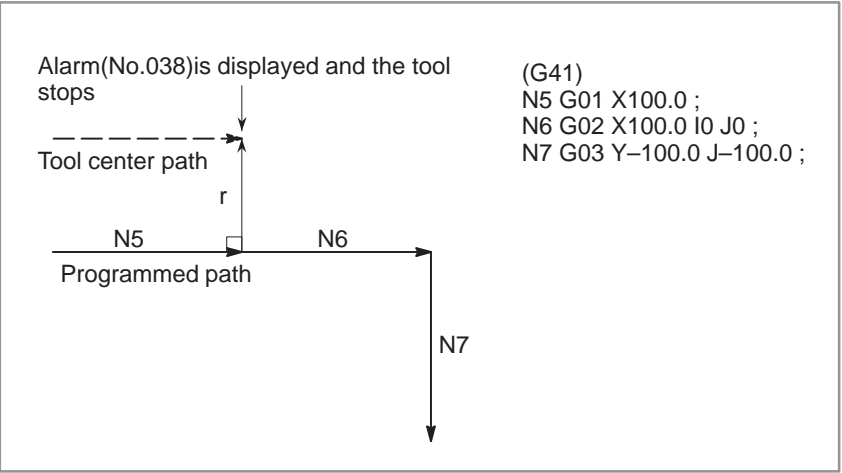
There is no inner intersection

If the cutter 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 cutter compensation. When this is predicted, alarm 33 occurs at the end of the previous block and the tool is stopped. In the example shown below, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for cutter compensation. If an excessively large value is specified, this intersection does not occur.



The center of the arc is identical with the start position or the end position

If the center of the arc is identical with the start position or end point, alarm (No. 038) is displayed, and the tool will stop at the end position of the preceding block.



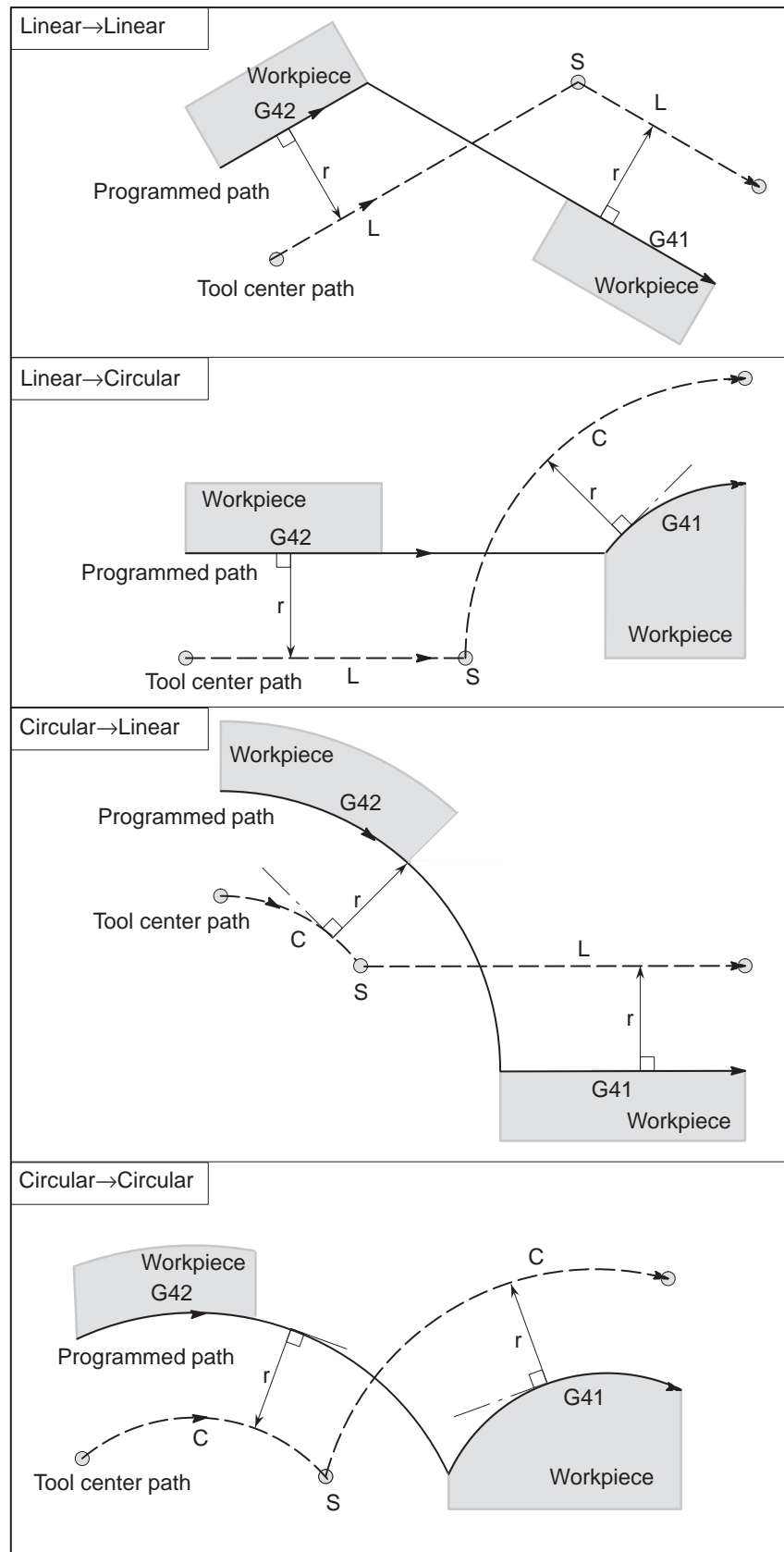
- Change in the offset direction in the offset mode**

The offset direction is decided by G codes (G41 and G42) for cutter radius and the sign of cutter compensation value as follows.

Gcode	Sign of offset amount	
	+	-
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

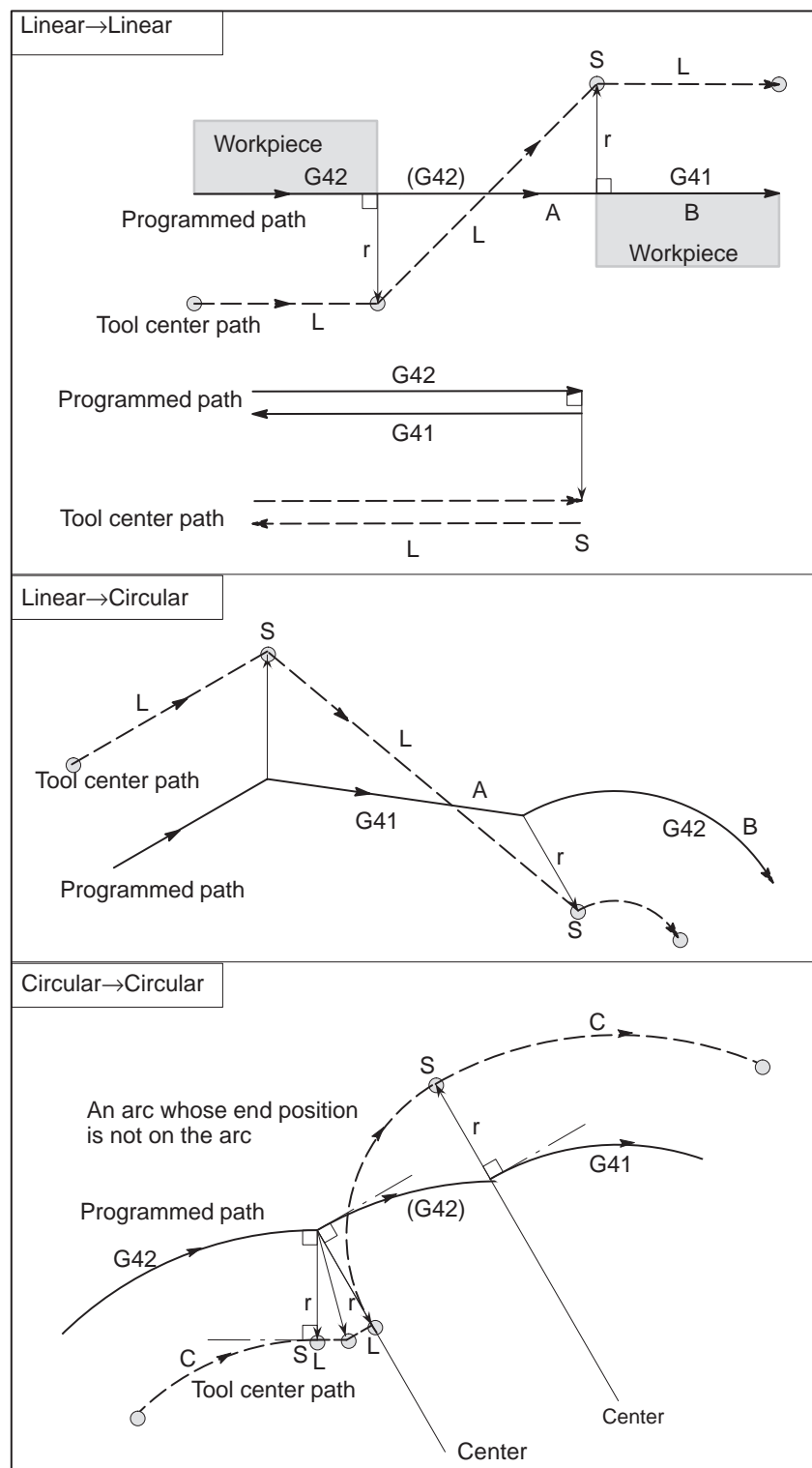
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block. However, the change is not available in the start-up block and the block following it.

Tool center path with an intersection



Tool center path without an intersection

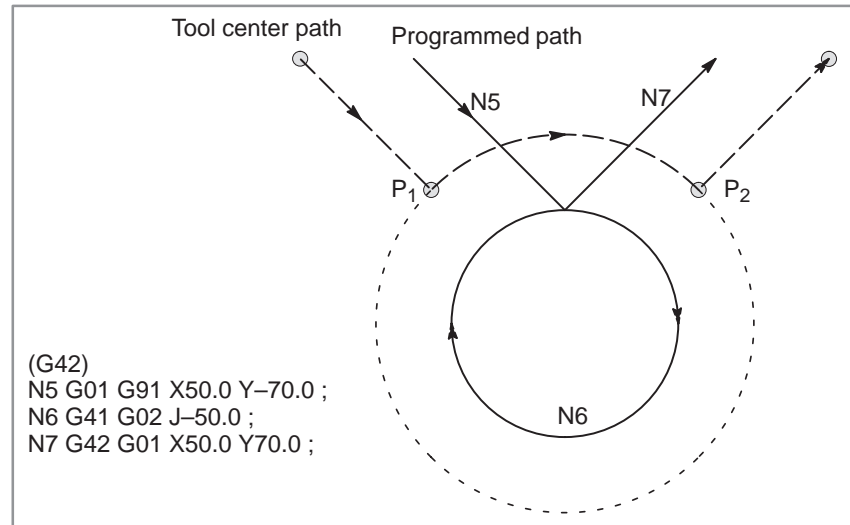
When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.



The length of tool center path larger than the circumference of a circle

Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I, J, and K this situation can occur.

In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from P₁ to P₂ as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.

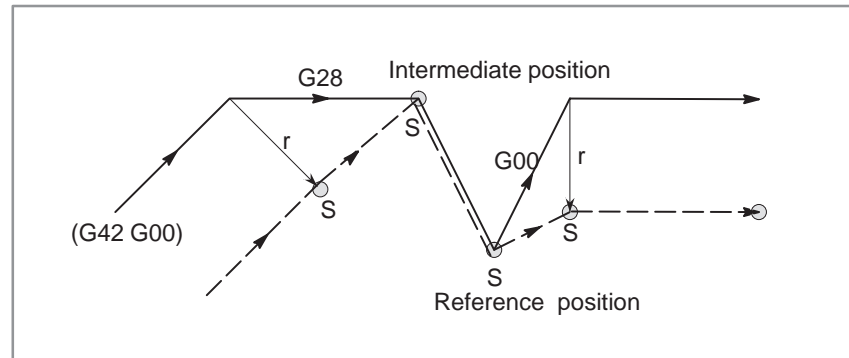


- **Temporary cutter compensation cancel**

If the following command is specified in the offset mode, the offset mode is temporarily canceled then automatically restored. The offset mode can be canceled and started as described in Subsections 15.6.2 and 15.6.4.

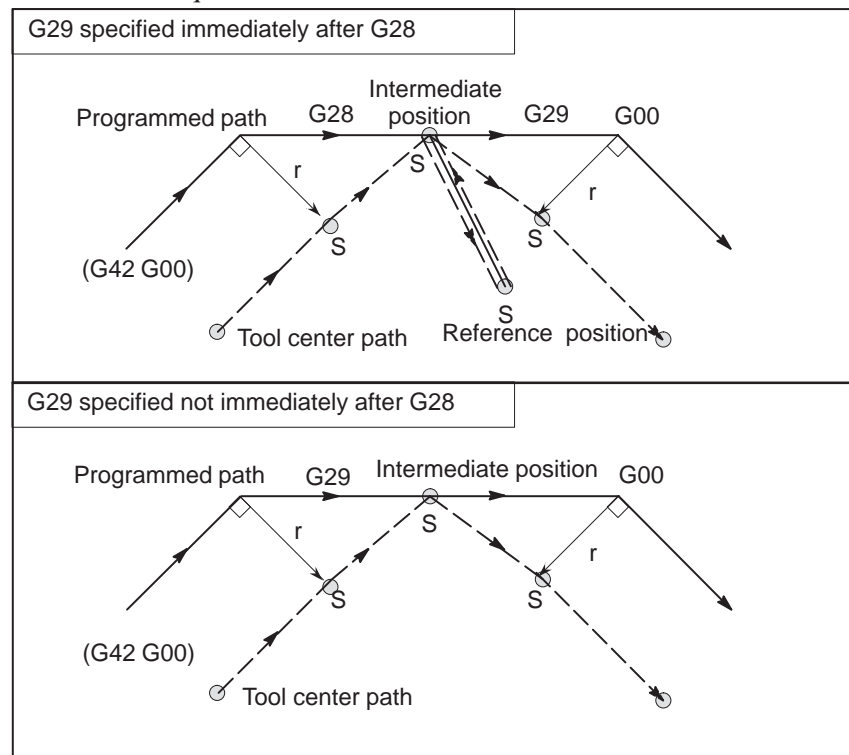
- **Specifying G28 (automatic return to the reference position) in the offset mode**

If G28 is specified in the offset mode, the offset mode is canceled at an intermediate position. If the vector still remains after the tool is returned to the reference position, the components of the vector are reset to zero with respect to each axis along which reference position return has been made.



- **Specifying G29 (automatic return from the reference position) in the offset mode**

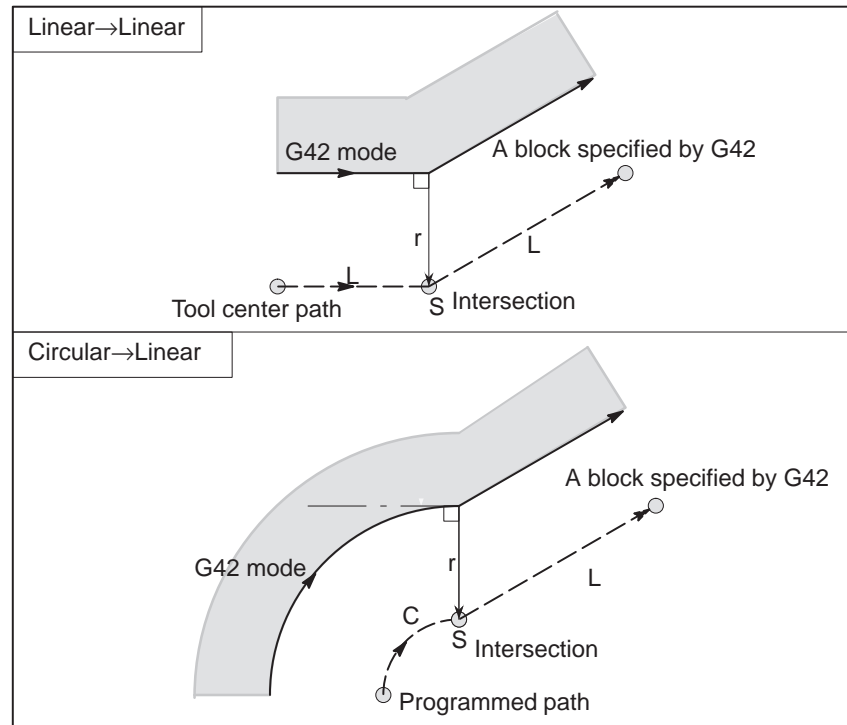
If G29 is commanded in the offset mode, the offset will be cancelled at the intermediate point, and the offset mode will be restored automatically from the subsequent block.



- **Cutter compensation G code in the offset mode**

The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining inner or outer side, by commanding the cutter compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct circular motion will not be obtained.

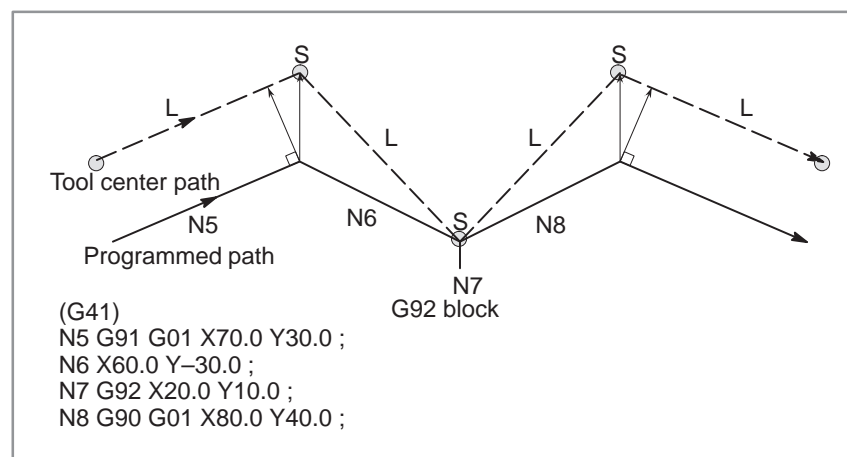
When the direction of offset is expected to be changed by the command of cutter compensation G code (G41, G42), refer to Subsec.15.6.3.



- **G92 command in the offset mode**

During offset mode, if G92 (absolute zero point programming) 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.



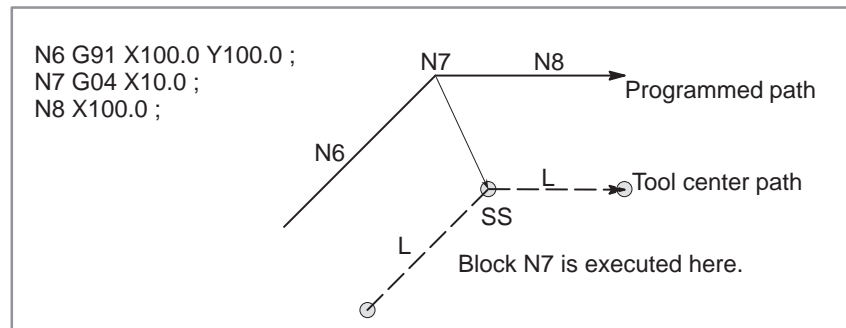
• A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

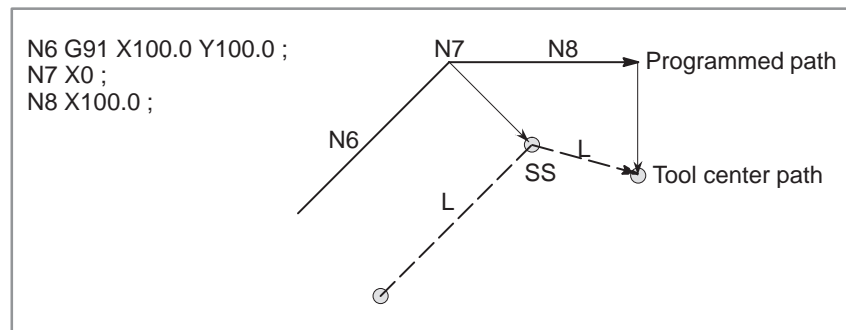
M05 ;	M code output	} These commands are of no movement.
S21 ;	S code output	
G04 X10.0 ; ...	Dwell	
G10 L11 P01 R10.0 ;	Cutter compensation value setting	
(G17) Z200.0 ; .	Move command not included in the offset plane.	
G90 ;	G code only	
G91 X0 ;	Move distance is zero.	

A block without tool movement specified in offset mode

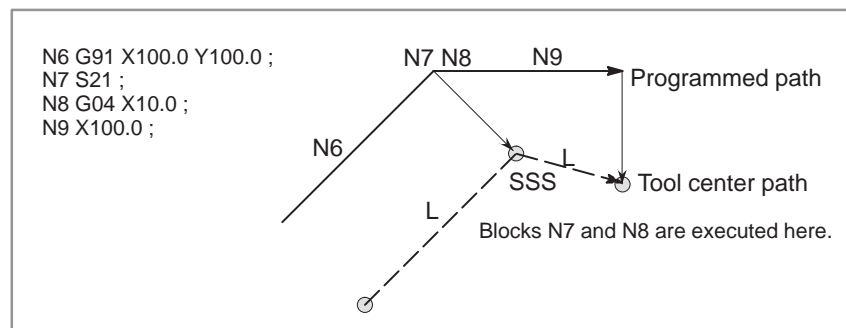
When a single block without tool movement is commanded in the offset mode, the vector and tool center path are the same as those when the block is not commanded. This block is executed at the single block stop point.



However, when the move distance is zero, even if the block is commanded singly, tool motion becomes the same as that when more than one block of without tool movement are commanded, which will be described subsequently.



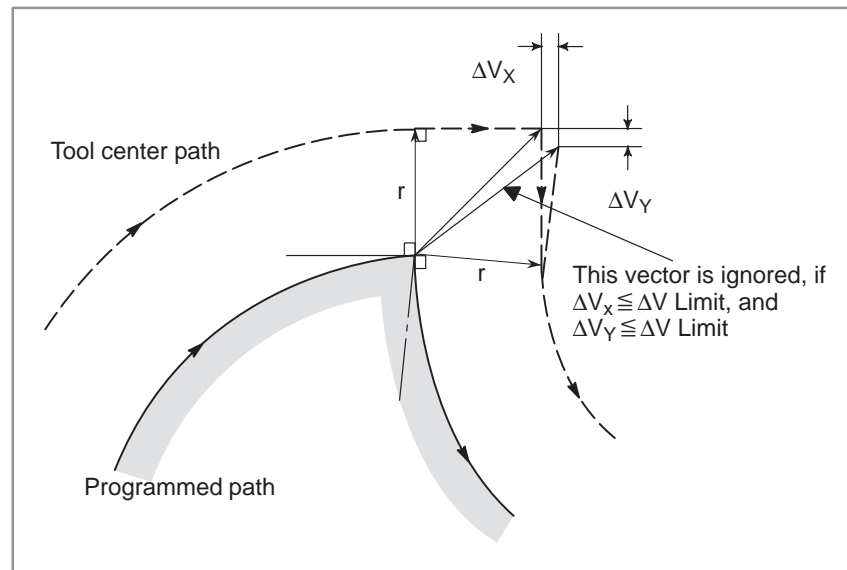
Two blocks without tool movement should not be commanded consecutively. If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.



• Corner movement

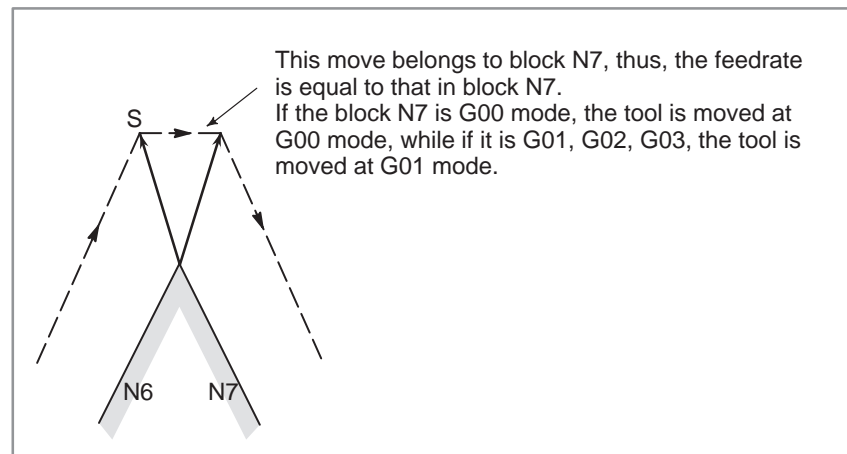
When two or more vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.

If these vectors almost coincide with each other, the corner movement isn't performed and the latter vector is ignored.



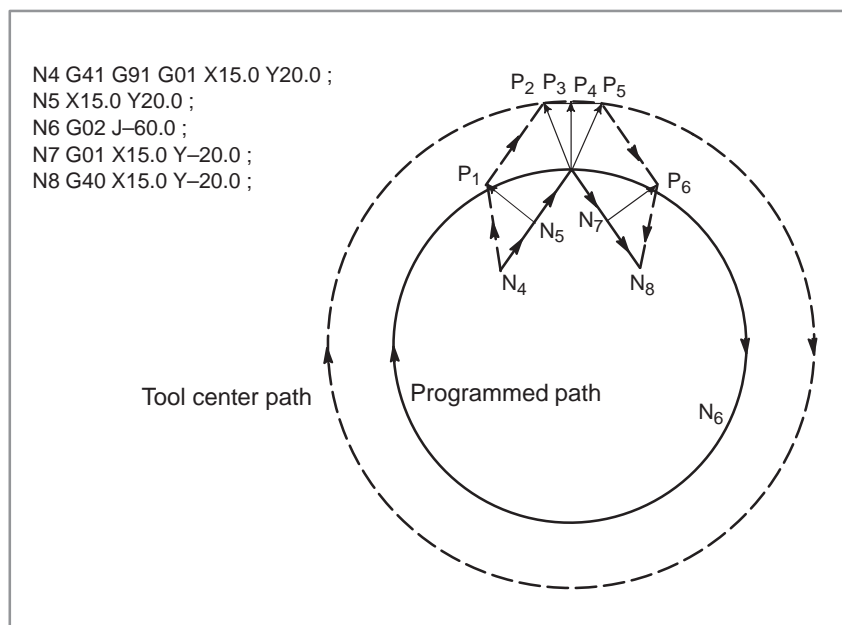
If $\Delta V_x \leq \Delta V_{limit}$ and $\Delta V_y \leq \Delta V_{limit}$, the latter vector is ignored. The ΔV_{limit} is set in advance by parameter (No. 557).

If these vectors do not coincide, a move is generated to turn around the corner. This move 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 N_6 is ignored.

- **Interruption of manual operation**

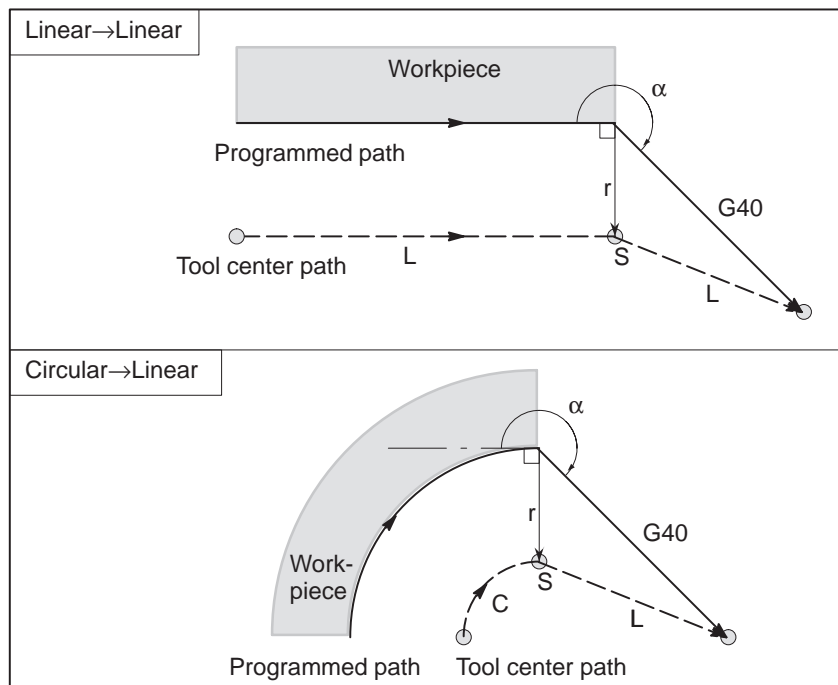
For manual operation during the cutter compensation, refer to Section III-3.5, "Manual Absolute ON and OFF."

14.6.4

Tool Movement in Offset Mode Cancel

Explanations

- Tool movement around an inside corner ($180^\circ \leq \alpha$)



● **Tool movement around an outside corner at an obtuse angle**
($90^\circ \leq \alpha < 180^\circ$)

Tool path has two types, A and B; and they are selected by parameter (No. 016#2).

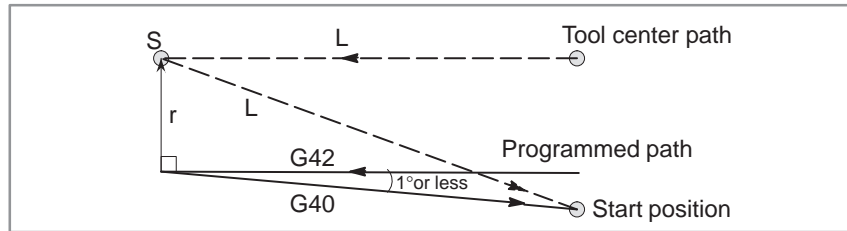
Type A	Linear→Linear	
	Circular→Linear	
Type B	Linear→Linear	
	Circular→Linear	

• **Tool movement around an outside corner at an acute angle ($\alpha < 90^\circ$)**

Tool path has two types, A and B : and they are selected by parameter (No. 016#2)

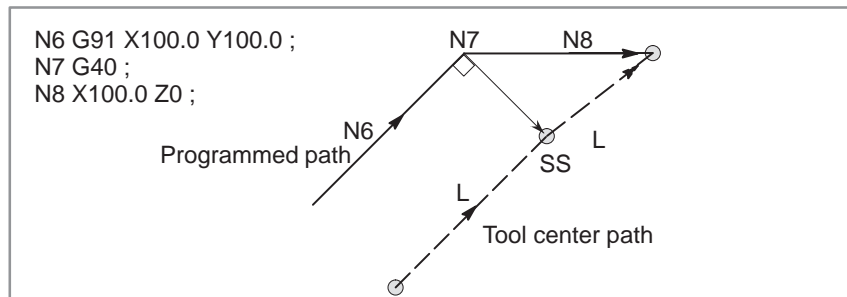
Type A	Linear→Linear	
	Circular→Linear	
Type B	Linear→Linear	
	Circular→Linear	

- **Tool movement around the outside linear → linear at an acute angle less than 1 degree ($\alpha < 1^\circ$)**



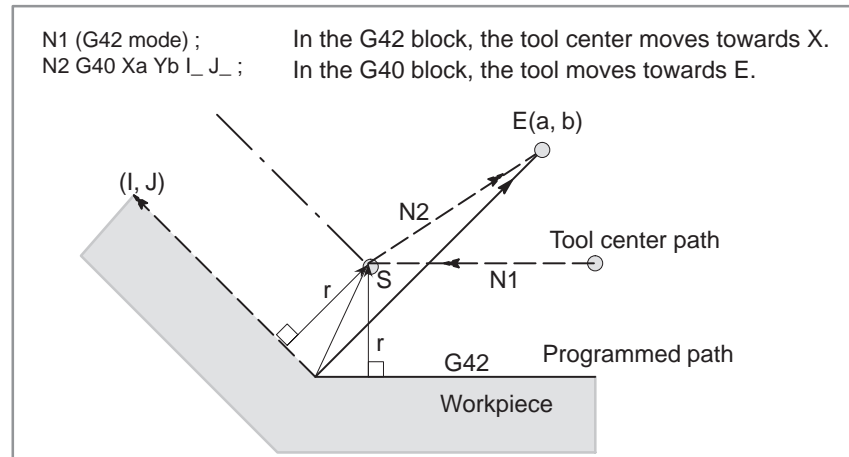
- **A block without tool movement specified together with offset cancel**

When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is cancelled in the next move command.

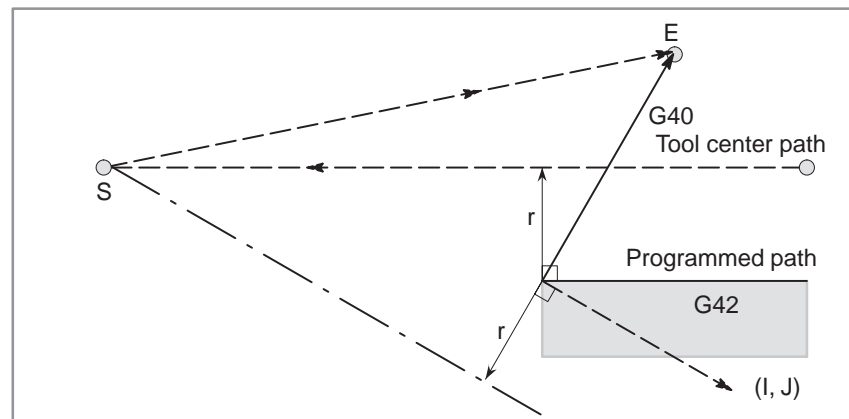


- **Block containing G40 and I_J_K_**
- **The previous block contains G41 or G42**

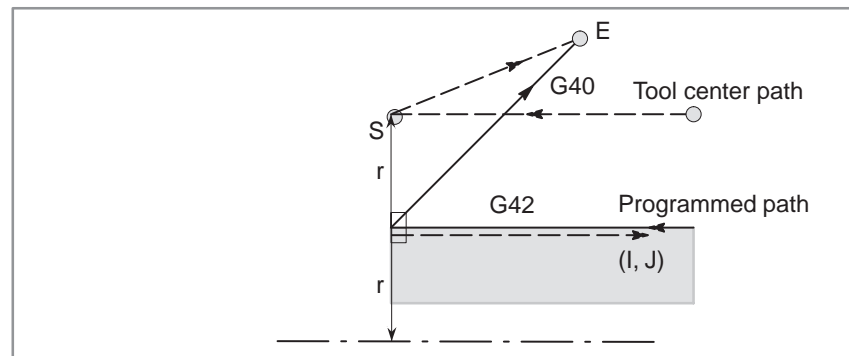
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified

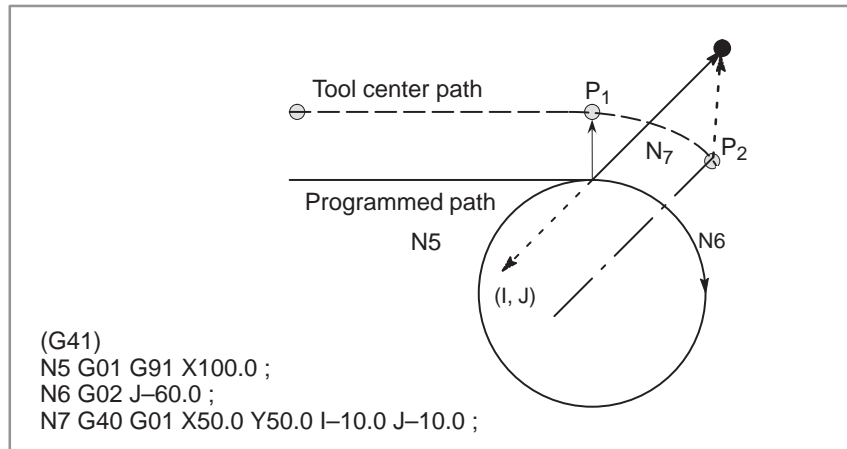


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



- **The 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 P1 to P2. The interference check function described in Subsection 14.6.5 may raise an alarm.



To make the tool trace a circle more than once, program two or more arcs.

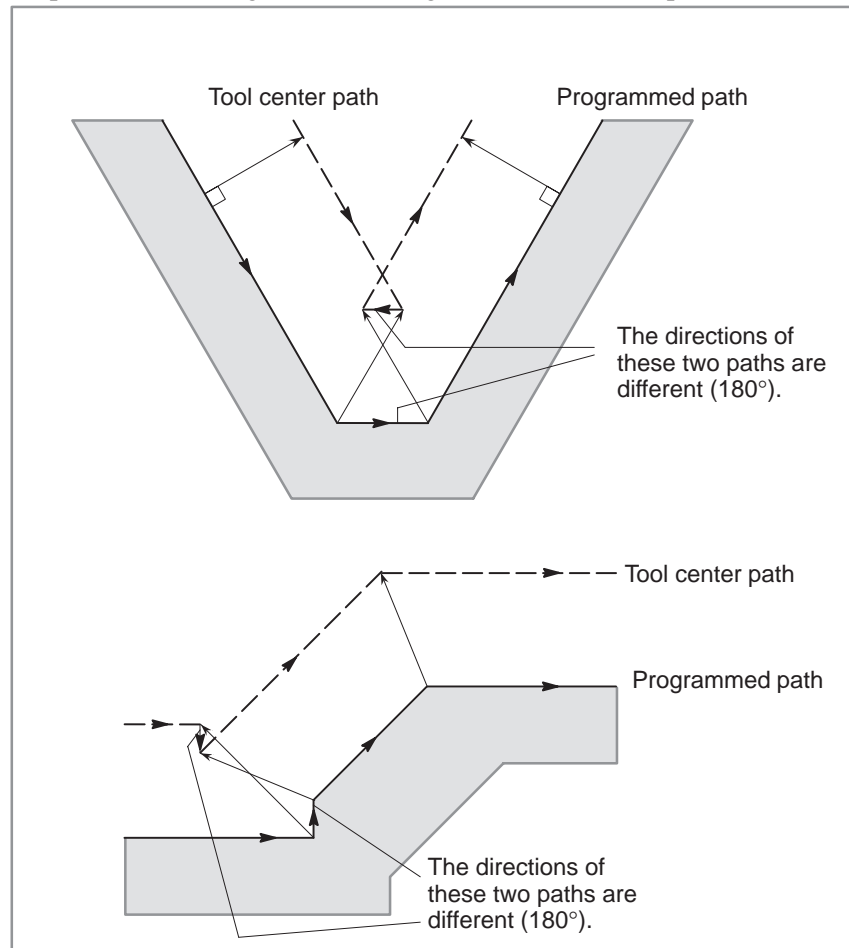
14.6.5 Interference Check

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.

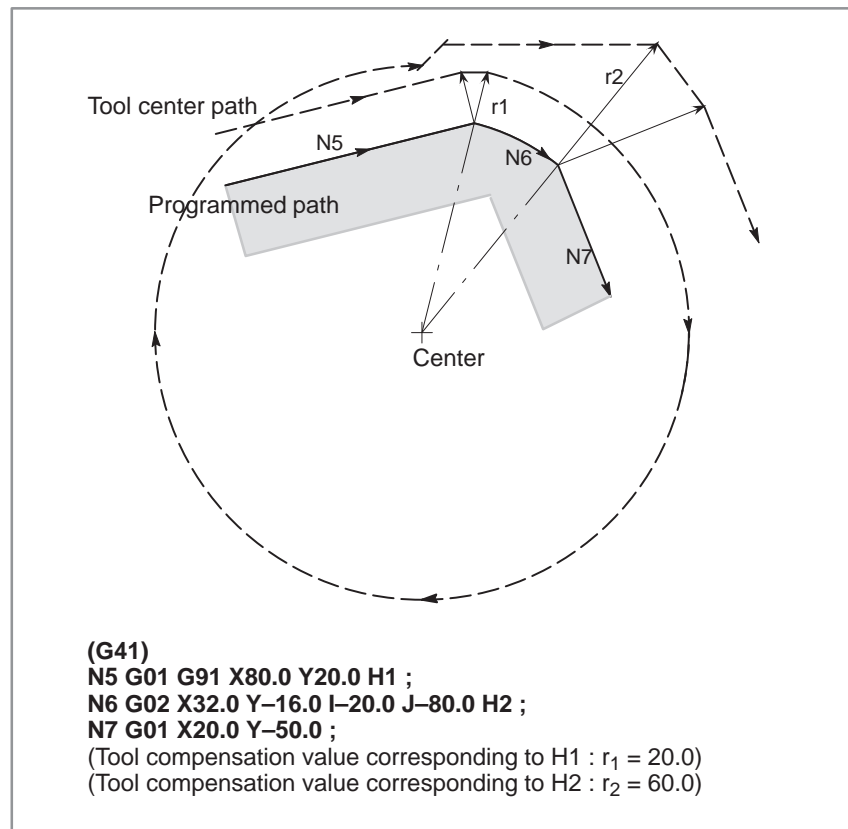
Explanations

- **Criteria for detecting interference**

- 1 The direction of the tool path is different from that of the programmed path (from 90 degrees to 270 degrees between these paths).



- ② In addition to the condition ①, the angle between the start point and end point on the tool center path is quite different from that between the start point and end point on the programmed path in circular machining (more than 180 degrees).



In the above example, the arc in block N6 is placed in the one quadrant. But after cutter compensation, the arc is placed in the four quadrants.

• Correction of interference in advance

- ① Removal of the vector causing the interference

When cutter compensation is performed for blocks A, B and C and vectors V_1 , V_2 , V_3 and V_4 between blocks A and B, and V_5 , V_6 , V_7 and V_8 between B and C are produced, the nearest vectors are checked first. If interference occurs, they are ignored. But if the vectors to be ignored due to interference are the last vectors at the corner, they cannot be ignored.

Check between vectors V_4 and V_5

Interference — V_4 and V_5 are ignored.

Check between V_3 and V_6

Interference — V_3 and V_6 are ignored

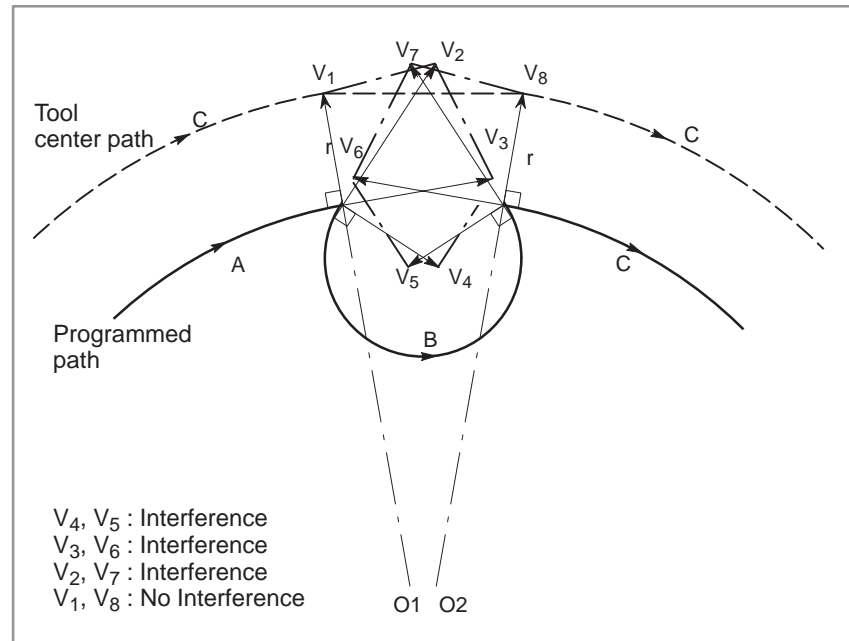
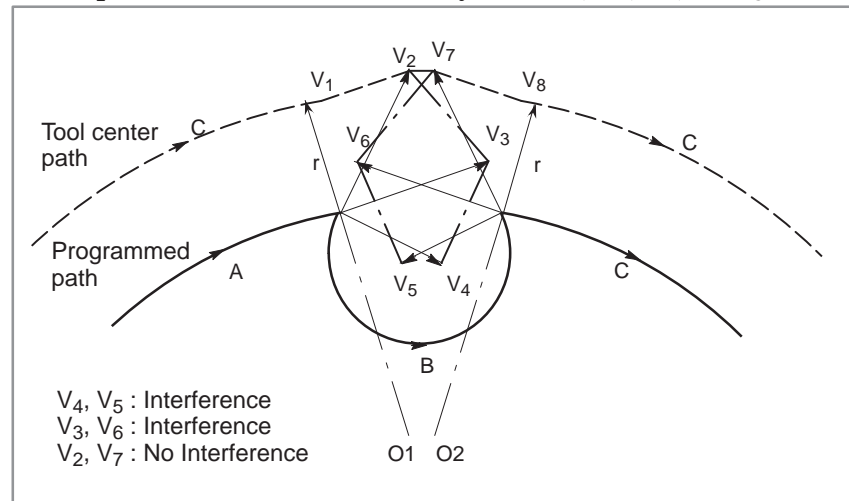
Check between V_2 and V_7

Interference — V_2 and V_7 are Ignored

Check between V_1 and V_8

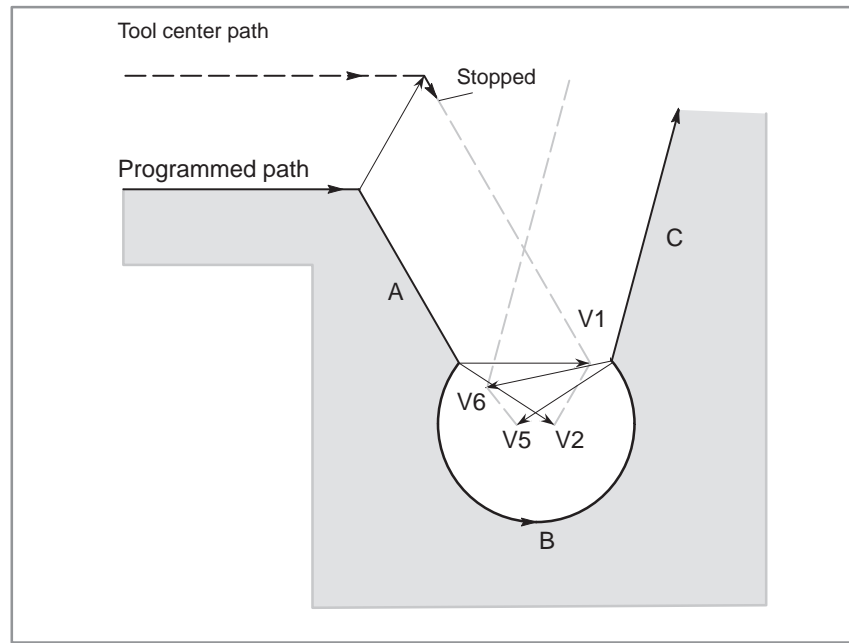
Interference — V_1 and V_8 are cannot be ignored

If while checking, a vector without interference is detected, subsequent vectors are not checked. If block B is a circular movement, a linear movement is produced if the vectors are interfered.

(Example 1) The tool moves linearly from V_1 to V_8 **(Example 2) The tool moves linearly from V_1, V_2, V_7 , to V_8** 

- ② If the interference occurs after correction ①, the tool is stopped with an alarm.

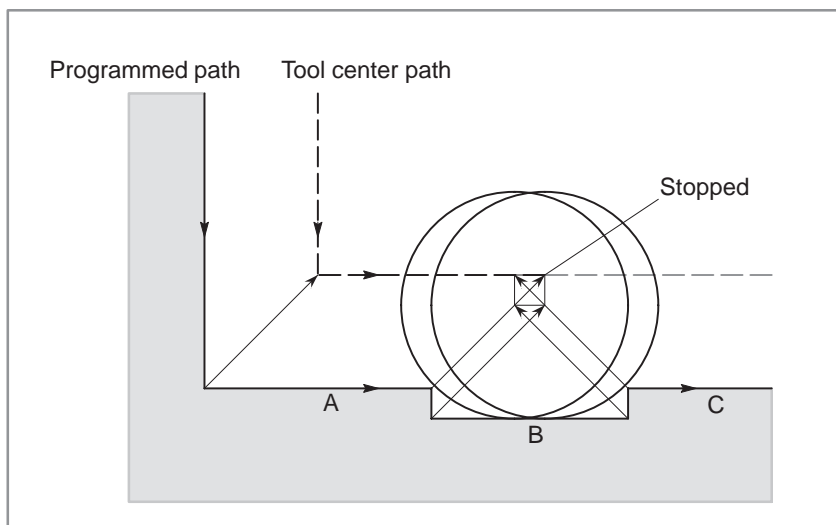
If the interference occurs after correction ① or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the alarm (No.41) is displayed and the tool is stopped immediately after execution of the preceding block. If the block is executed by the single block operation, the tool is stopped at the end of the block.



After ignoring vectors V_2 and V_5 because of interference, interference also occurs between vectors V_1 and V_6 . The alarm is displayed and the tool is stopped.

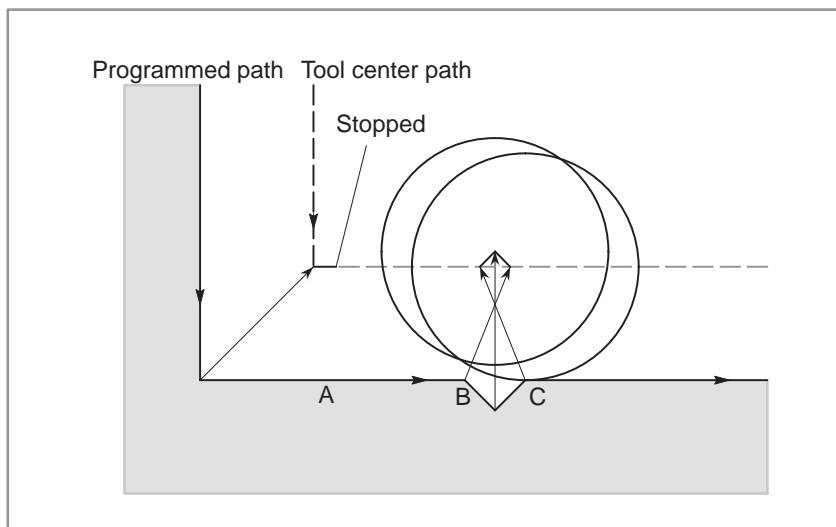
- When interference is assumed although actual interference does not occur

① Depression which is smaller than the cutter compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after cutter compensation the tool stops and an alarm is displayed.

2 Groove which is smaller than the cutter compensation value



In the same way as 1, the programmed path will be opposite to the tool path after cutter compensation has been applied. This state is determined to constitute interference. An alarm is issued and the machine stops.

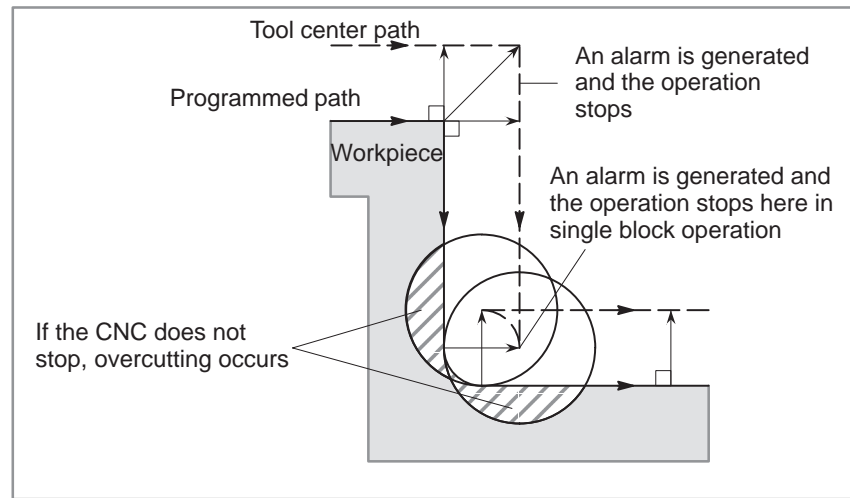
14.6.6

Overcutting by Cutter Compensation

Explanations

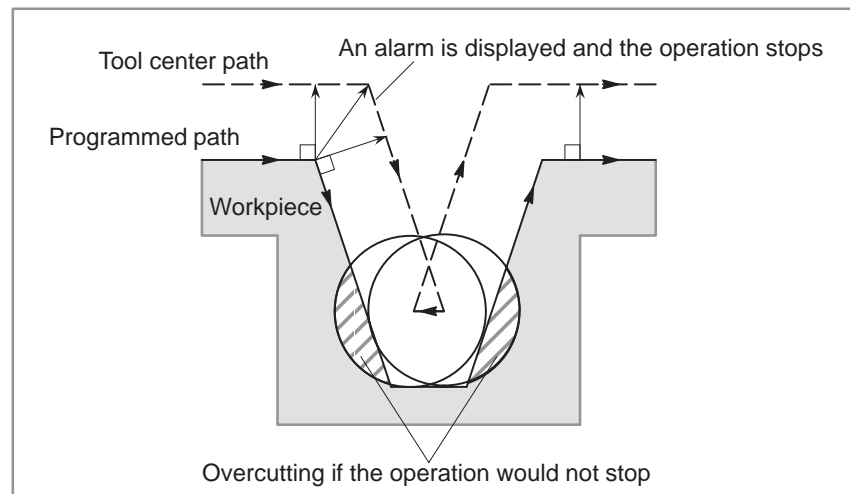
- **Machining an inside corner at a radius smaller than the cutter radius**

When the radius of a corner is smaller than the cutter radius, because the inner offsetting of the cutter will result in overcuttings, an alarm is displayed and the CNC stops at the start of the block. In single block operation, the overcutting is generated because the tool is stopped after the block execution.



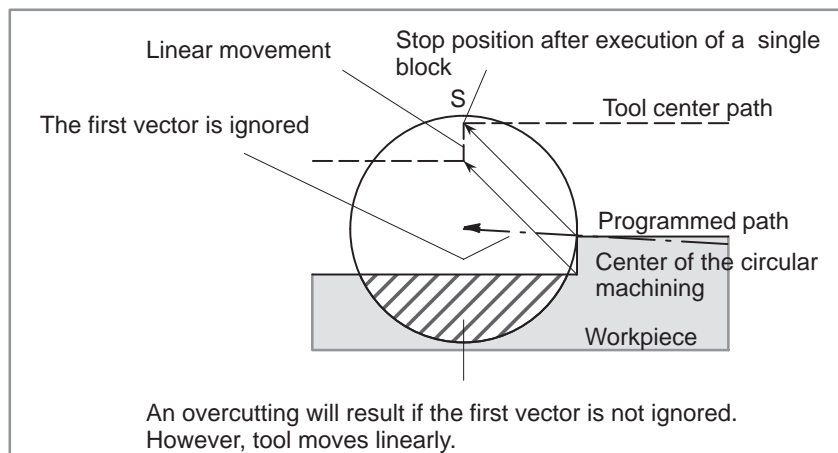
- **Machining a groove smaller than the tool radius**

Since the cutter compensation forces the path of the center of the tool 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.



- **Machining a step smaller than the tool radius**

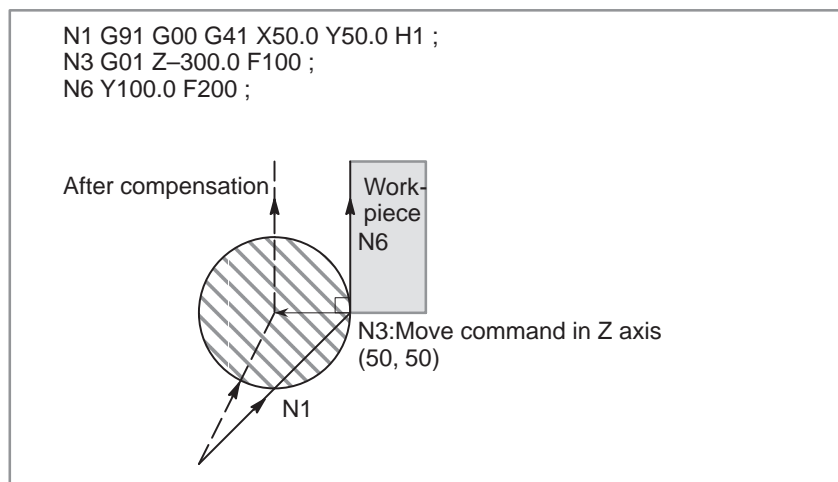
When machining of the step is commanded by circular machining in the case of a program containing a step smaller than the tool radius, the path of the center of tool with the ordinary offset becomes reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly 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.



- **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 cutter compensation is effected at some distance from the workpiece at the start of the machining.

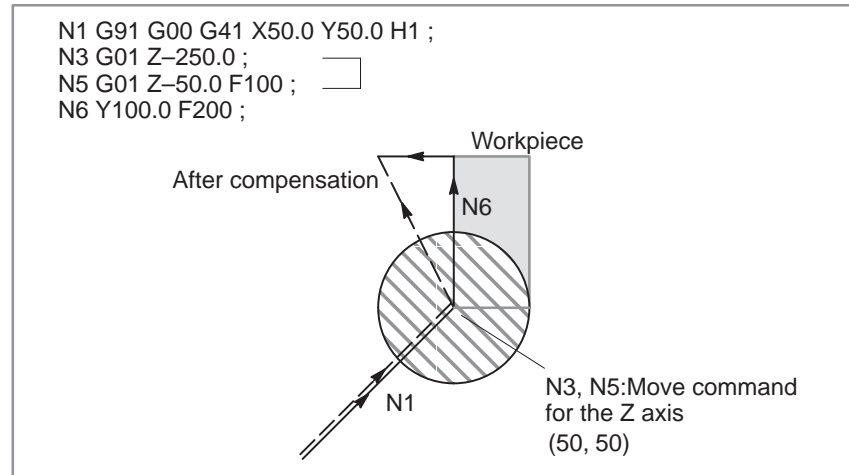
In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.



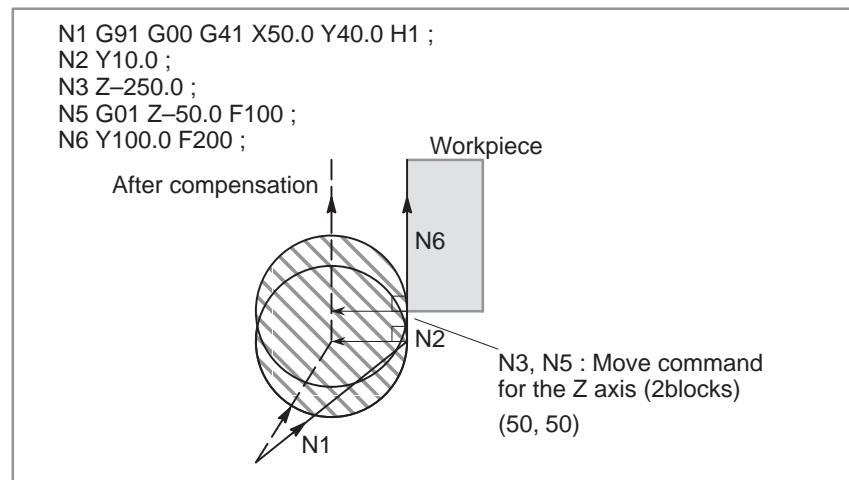
In the program example above, when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the figure above.

Then, if the block N3 (move command in Z axis) is divided as follows: As there are two move command blocks not included in the selected plane and the block N6 cannot be entered into the buffer storage, the tool center path is calculated by the information of N1 in the figure above. That is, the offset vector is not calculated in start-up and the overcutting may result.

The above example should be modified as follows:



The move command in the same direction as that of the move command after the motion in Z axis should be programmed.



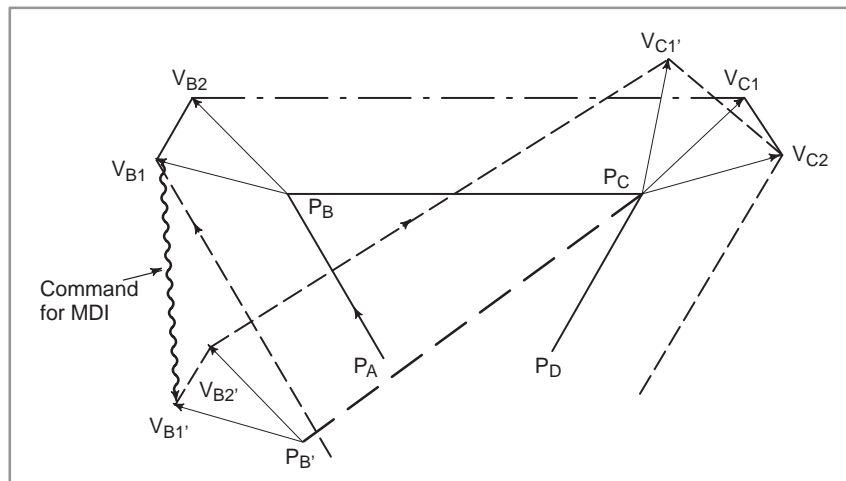
As the block with sequence No. N2 has the move command in the same direction as that of the block with sequence No. N6, the correct compensation is performed.

14.6.7 Input Command from MDI

Cutter compensation C is not performed for commands input from the MDI.

However, when automatic operation by absolute commands is temporarily stopped by the single block function, MDI operation is performed, then automatic operation starts again, the tool path is as follows :

In this case, the vectors at the start position of the next block are translated and the other vectors are produced by the next two blocks. Therefore, from next block but one, cutter compensation C is accurately performed.



When position P_A , P_B , and P_C are programmed in an absolute command, tool is stopped by the single block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are translated to V_{B1}' and V_{B2}' and offset vectors are recalculated for the vectors V_{C1} and V_{C2} between block P_B – P_C and P_C – P_D .

However, since vector V_{B2} is not calculated again, compensation is accurately performed from position P_C .

14.7
TOOL
COMPENSATION
VALUES, NUMBER
OF COMPENSATION
VALUES, AND
ENTERING VALUES
FROM THE
PROGRAM (G10)

Tool compensation values include tool geometry compensation values and tool wear compensation (Fig. 14.7).

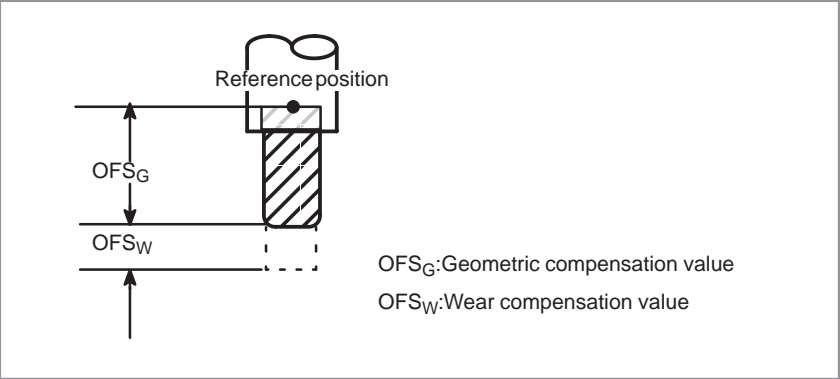


Fig.14.7 Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the CRT/MDI panel (see section III-9.1) or from a program. A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program. The value is used for tool length compensation, cutter compensation, or the tool offset.

Explanations

- Valid range of tool compensation values
- Number of tool compensation values and the addresses to be specified

Table 14.7(a) shows the valid input range of tool compensation values.

Table14.7 (a) The valid input range of tool compensation value

Increment system	Metric input	inch input
IS-B	0 to ±999.999mm	0 to ±99.9999inch
IS-C	0 to ±999.9999mm	0 to ±99.99999inch

The memory can hold 32,64,99, or 200 tool compensation values (option). Address D or H is used in the program. The address used depends on which of the following functions is used: tool length compensation(see section 14.1), tool offset (see section 14.3), cutter compensation B (see section 14.4), or cutter compensation C (see sections 14.5 and 14.6). The range of the number that comes after the address (D or H) depends on the number of tool compensation values : 0 to 32, 0 to 64, 0 to 99, or 0 to 200.

- **Tool compensation memory and the tool compensation value to be entered**

Tool compensation memory A or B can be used.

The tool compensation memory determines the tool compensation values that are entered (set) (Table 14.7 (b)).

Table14.7(b) Setting contents tool compensation memory and tool compensation value

Tool compensation memory	Tool compensation value
A	The tool compensation corresponding to the tool compensation number is used.
B	The tool geometry compensation, plus tool wear compensation, both corresponding to the tool compensation number, are used.

Format

The programming format depends on which tool compensation memory is used.

- **Input of tool compensation value by programing**

Table14.7(c) Setting format of Tool compensation memory and Tool compensation value

Tool compensation memory		Format
A	Tool compensation value	G10P_R_;
B	Geometry compensation value	G10L10P_R_;
	Wear compensation value	G10L11P_R_;

P : Number of tool compensation

R : Tool compensation value in the absolute command(G90) mode

Value to be added to the specified tool compensation value in the incremental command(G91) mode (the sum is also a tool compensation value.)

NOTE

To provide compatibility with the format of older CNC programs, the system allows L1 to be specified instead of L11.

Explanations

- **Scaling up or down along all axes at the same rate of magnification**
- **Scaling of each axis, programmable mirror image (negative magnification)**

Least input increment of scaling magnification is: 0.001 or 0.00001 It is depended on parameter (No. 036#07) which value is selected. If scaling P is not specified on the block of scaling (G51X_Y_Z_P_ ;), the scaling magnification set to parameter (No. 731) is applicable. If X,Y,Z are omitted, the tool position where the G51 command was specified serves as the scaling center.

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. First of all, set a parameter (No. 063#6) which validates each axis scaling (mirror image). Then, set parameter (No. 036#0, #1, #2) to enable scaling along each axis. Least input increment of scaling magnification of each axis (I, J, K) is 0.001 or 0.00001 (set parameter (No. 036#7)).

Magnification is set to parameter 5421 within the range ± 0.00001 to ± 9.99999 or ± 0.001 to ± 9.999

If a negative value is set, mirror image is effected.

If magnification I, J or K is not commanded, a magnification value set to parameter (No. 731, 732, 733) is effective. However, a value other than 0 must be set to the parameter.

NOTE

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

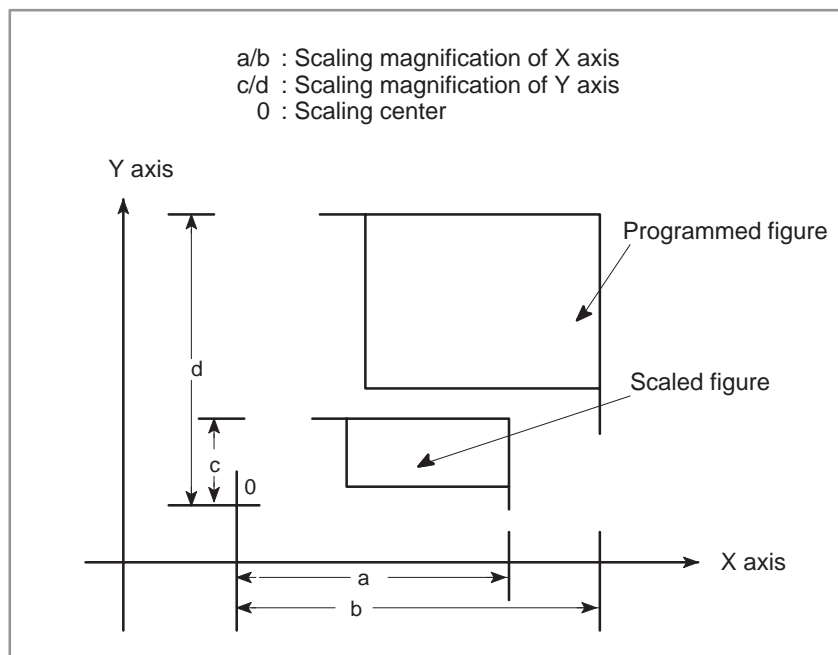


Fig.14.8 (b) Scaling of each axis

- **Scaling of circular interpolation**

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

When different magnifications are applied to axes and a circular interpolation is specified with radius R, it becomes as following figure 14.8 (c) (in the example shown below, a magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.).

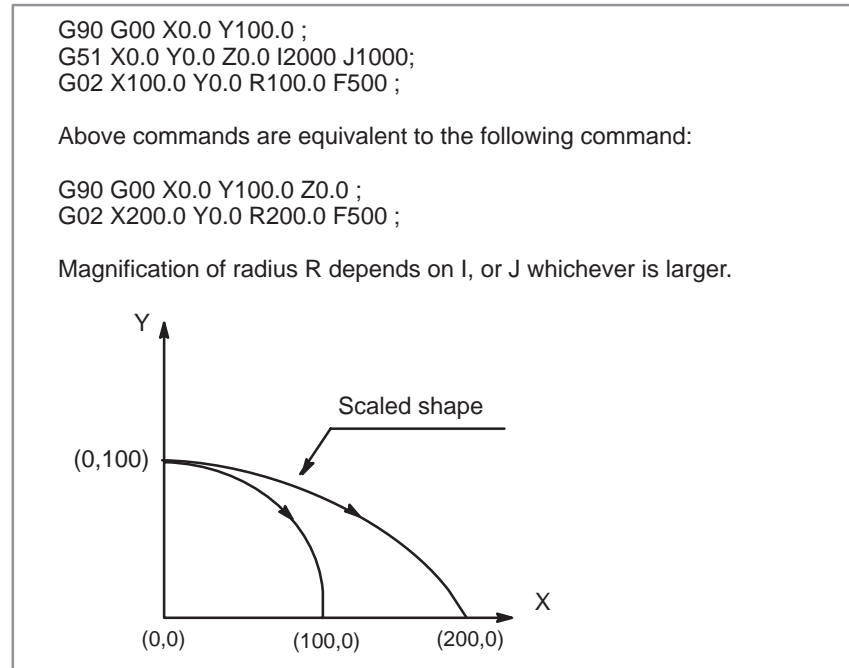


Fig.14.8 (c) Scaling for circular interpolation 1

When different magnifications are applied to axes and a circular interpolation is specified with I, J and K, it becomes as following figure 14.8 (d) (In the example shown below, a magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.).

```
G90 G00 X0.0 Y0.0 ;  
G51 X0.0 Y0.0 I2000 J1000;  
G02 X100.0 Y0.0 I0.0 J-100.0 F500 ;
```

Above commands are equivalent to the following commands.

```
G90 G00 X0.0 Y100.0;  
G02 X200.0 Y0.0 I0.0 J-100.0 F500 ;
```

In this case, the end point does not meet the radius, a linear section is included.

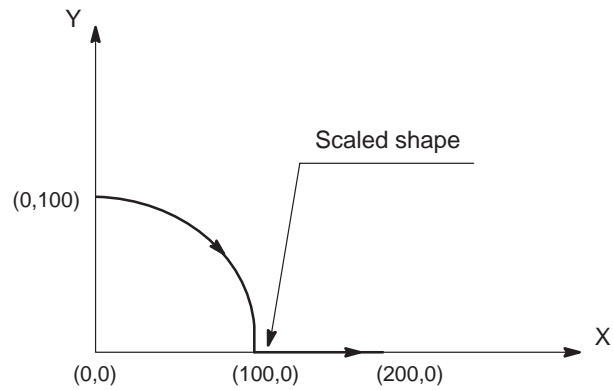


Fig.14.8(d) Scaling for circular interpolation 2

- **Invalid scaling**

This scaling is not applicable to cutter compensation values, tool length offset values, and tool offset values (Fig. 14.8 (e)).

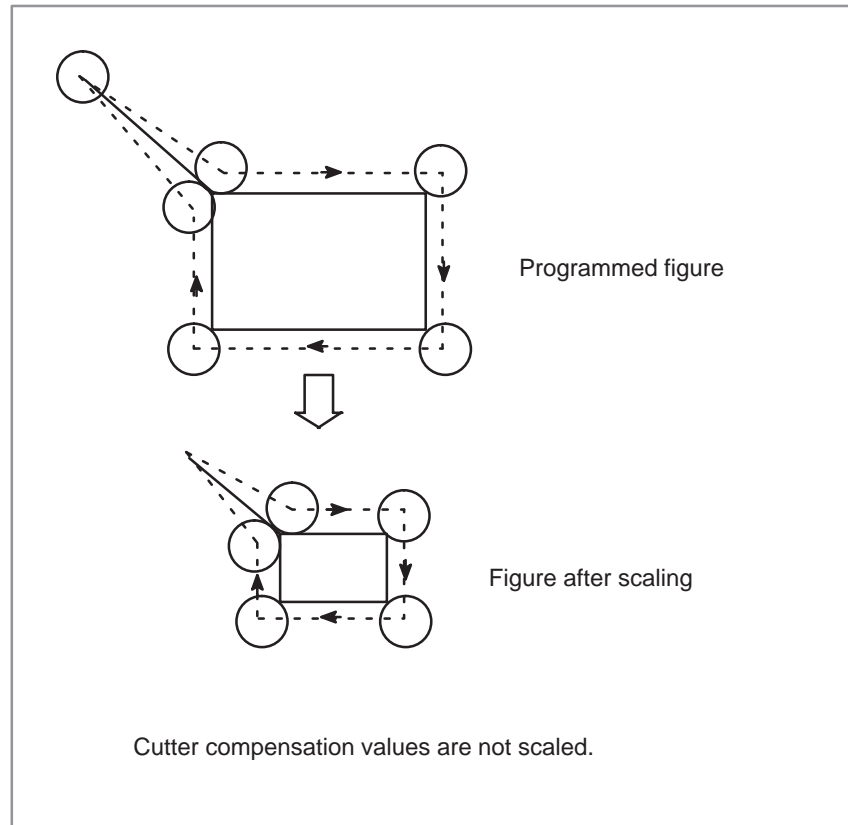


Fig.14.8(e) Scaling during cutter compensation

Scaling is not applicable to the Z-axis movement in case of the following canned cycle.

- Cut-in value Q and retraction value d of peck drilling cycle (G83, G73).
- Fine boring cycle (G76)
- Shift value Q of X and Y axes in back boring cycle (G87).

In manual operation, the travel distance cannot be increased or decreased using the scaling function.

WARNING

- 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 (G27, G28, G29, G30) or coordinate system setting (G92), 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 cutter compensation C. See the description of blocks that do not move the tool at subsection 14.6.3.

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) Cutter compensation C Offset direction is reversed.
 - (3) Coordinate system rotation Rotation angle is reversed.
 - (4) Cutter compensation B Offset direction is reversed. (Including G39)

Examples

Example of a mirror image program

Subprogram

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

Main program

```
N10 G00 G90;
N20 M98 P9000;
N30 G51 X50.0 Y50.0 I-1000 J1000;
N40 M98 P9000;
N50 G51 X50.0 Y50.0 I-1000 J-1000;
N60 M98 P9000;
N70 G51 X50.0 Y50.0 I1000 J-1000;
N80 M98 P9000;
N90 G50;
```

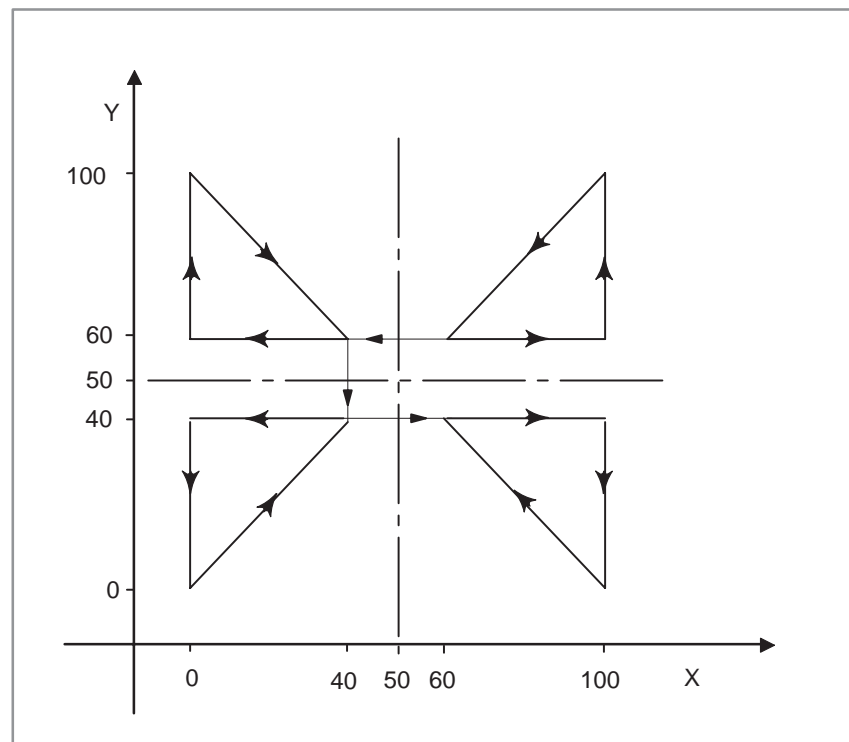


Fig. 14.8 (f) Example of a mirror image program

14.9

COORDINATE
SYSTEM ROTATION
(G68, G69)

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.

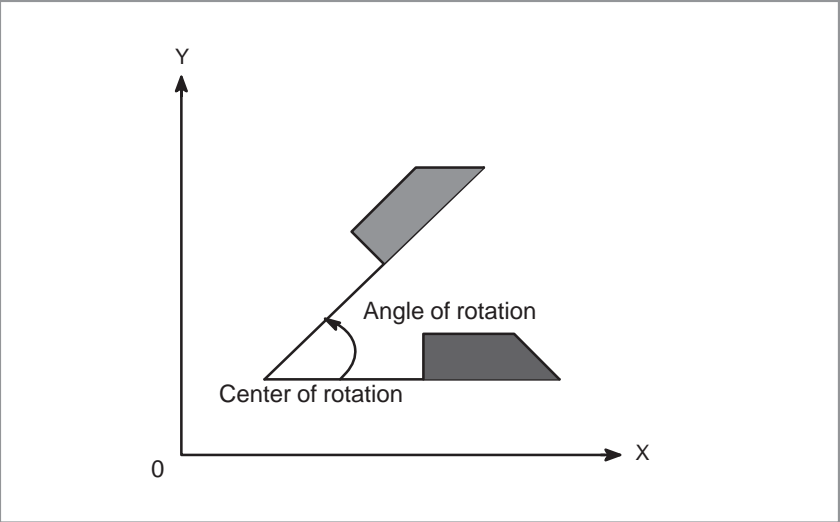


Fig. 14.9 (a) Coordinate system rotation

Format

Format	
<div><div><div><div>G17</div><div>G18</div><div>G19</div></div><div>⋮</div><div>G69 ; command</div></div><div><div>G68 α_β_R_ ;</div><div></div></div></div>	<div>Start rotation of a coordinate system.</div> <div><div></div><div>Coordinate system rotation mode (The coordinate system is rotated.)</div></div> <div>Coordinate system rotation cancel</div>
Meaning of command	
<div>G17 (G18 or G19) : Select the plane in which contains the figure to be rotated.</div> <div>α_β_ : Absolute command for two of the x_,y_,and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68.</div> <div>R_ : Angular displacement with a positive value indicates counter clockwise rotation. Bit 0 of parameter 041 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).</div> <div>Least input increment : 0.001 deg</div> <div>Valid data range : 360,000 to 360,000</div>	

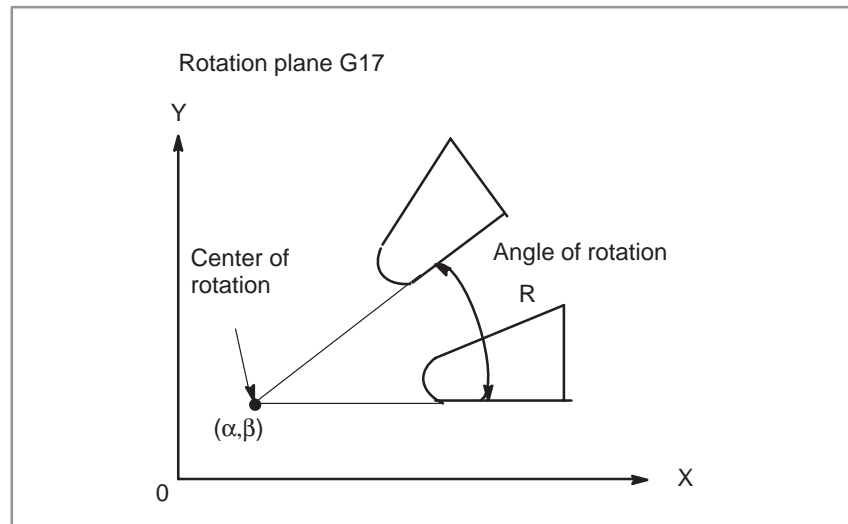


Fig.14.9 (b) Coordinate system rotation

NOTE

When a decimal fraction is used to specify angular displacement ($R_{_}$), the 1's digit corresponds to degree units.

Explanations

- **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 (G68). G17, G18 or G19 must not be designated in the mode of coordinate system rotation.

As for the incremental position commands designated between the G68 block and a block with an absolute command; it is regarded that the position where G68 was designated is the center of rotation (Fig. 14.9 (c)).

When $\alpha_{_}$ and $\beta_{_}$ are omitted, the position where G68 is commanded is set as the center of rotation.

When angle of rotation is omitted, the value set to parameter (No. 730) is regarded as the rotation angle. The coordinate system rotation is cancelled by G69;

G69 may be designated in the same block as the other commands. Tool offset, such as cutter compensation, tool length offset, or tool offset, is performed after the coordinate system is rotated for the command program.

WARNING

Use absolute programming to specify movement in a block in which G69 is specified.

```

N1 G92 X-50.0 Y-50.0 G69 G17 ;
N2 G68 X70.0 Y30.0 R60.0 ;
N3 G90 G01 X0 Y0 F200 ;
    (G91X50.0Y50.0)
N4 G91 X100.0 ;
N5 G02 Y100.0 R100.0 ;
N6 G03 X-100.0 I-50.0 J-50.0 ;
N7 G01 Y-100.0 ;
N8 G69 G90 X-50.0 Y-50.0 M02 ;

```

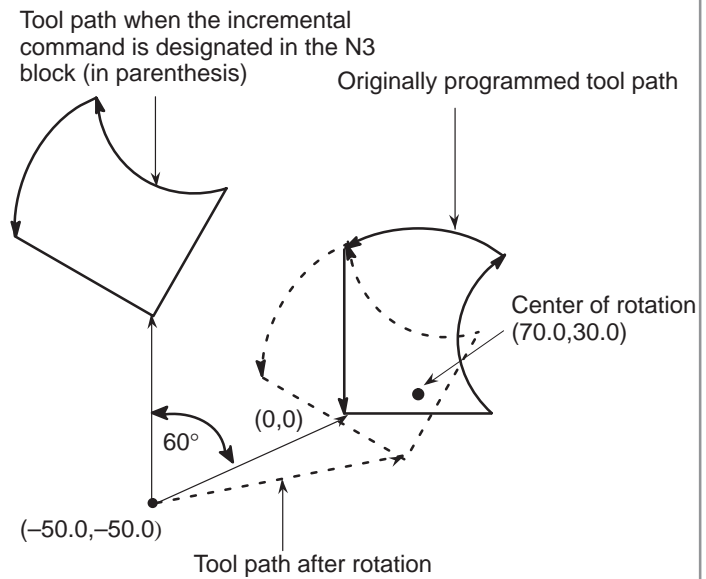


Fig.14.9(c) Absolute/incremental command during coordinate system rotation

Examples

• Cutter compensation C and coordinate system rotation

It is possible to specify G68 and G69 in cutter compensation C mode. The rotation plane must coincide with the plane of cutter compensation C.

```
N1 G92 X0 Y0 G69 G01 ;
N2 G42 G90 X100.0 Y100.0 F1000 H01 ;
N3 G68 R-30.0 ;
N4 G91 X200.0 ;
N5 G03 Y100.0 R100.0 J50.0 ;
N6 G01 X-200.0 ;
N7 Y-100.0 ;
N8 G69 G40 G90 X0 Y0 M30 ;
```

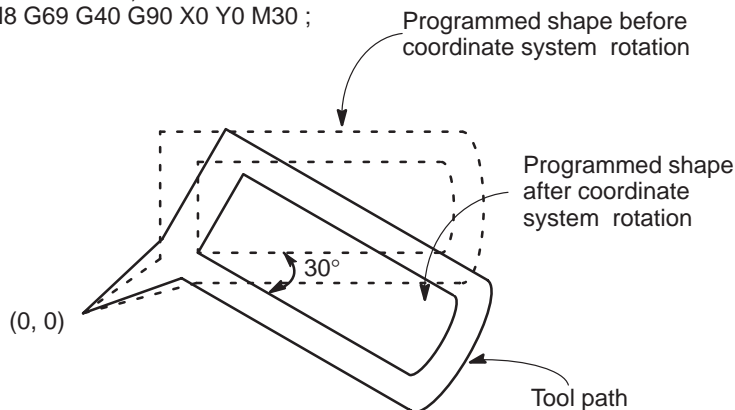


Fig. 14.9 (d) Cutter compensation C and coordinate system rotation

• Scaling and coordinate system rotation

If a coordinate system rotation command is executed in the scaling mode (G51 mode), the coordinate value (α, β) 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 (G68) should not be issued in cutter compensation C mode (G41, G42) on scaling mode (G51). The coordinate system rotation command should always be specified prior to setting the cutter compensation C mode.

1. When the system is not in cutter compensation mode C, specify the commands in the following order :

```
G51 ;   scaling mode start
G68 ;   coordinate system rotation mode start
      .
      .
      .
G69 ;   coordinate system rotation mode cancel
G50 ;   scaling mode cancel
```

2. When the system is in cutter compensation model C, specify the commands in the following order (Fig.14.9(e)) :

(cutter compensation C cancel)

G51 ; scaling mode start

G68 ; coordinate system rotation start

:

G41 ; cutter compensation C mode start

:

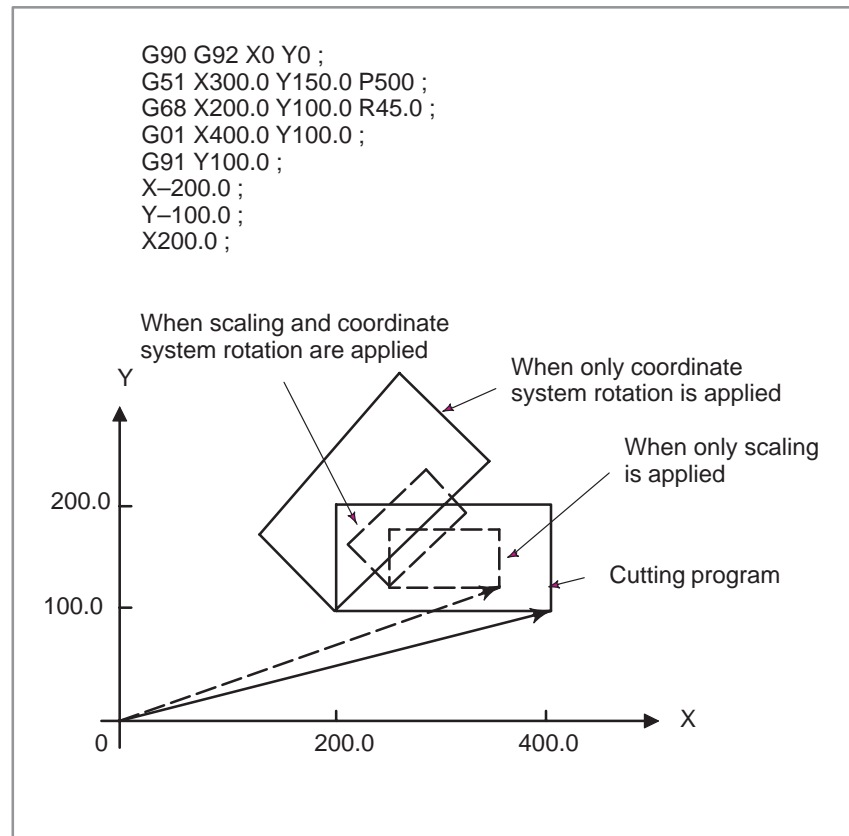


Fig. 14.9 (e) Scaling and coordinate system rotation in cutter compensation C mode

- **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 the bit (bit 0 of parameter 041) 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 G69 G17;
G01 F200 H01;
M98 P2100 ;
M98 P072200 ;
G69 G00 G90 X0 Y0 M30 ;

O 2200 G68 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 ;
```

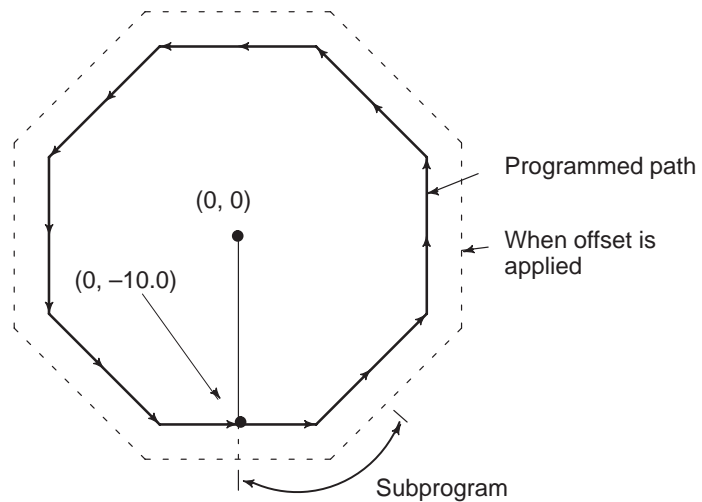


Fig. 14.9 (f) Coordinate system rotation command

14.10 INDEX TABLE INDEXING FUNCTION

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

Explanations

• Indexing position

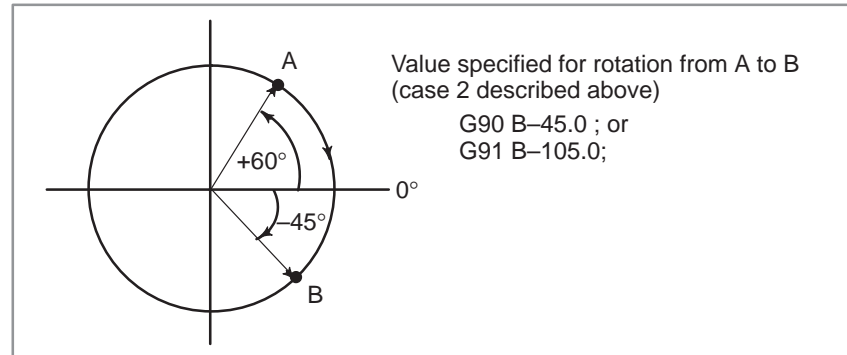
Specify an indexing position with the fourth axis address, generally A, B, or C.

The indexing position is specified by either of the following (depending on bit 4 of parameter 079):

1. Absolute value only
2. Absolute or incremental value depending on the specified G code: G90 or G91

A positive value indicates an indexing position in the counterclockwise direction. A negative value indicates an indexing position in the clockwise direction.

The minimum indexing angle of the index table is the value set to parameter 839. Only multiples of the least input increment can be specified as the indexing angle. If any value that is not a multiple is specified, an alarm (No. 135) occurs. Decimal fractions can also be entered. When a decimal fraction is entered, the 1's digit corresponds to degree units.



• Direction and value of rotation

The direction of rotation and angular displacement are determined by either of the following two methods. Refer to the manual written by the machine tool builder to find out which method is applied.

1. Using the miscellaneous function specified in parameter 249 (Address) (Indexing position) (Miscellaneous function);
Rotation in the negative direction (Address) (Indexing position);
Rotation in the positive direction (No miscellaneous functions are specified.)

An angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° when bit 2 of parameter 079 specifies this option.

For example, when G90 B400.0 (miscellaneous function); is specified at a position of 0, the table is rotated by 40° in the negative direction.

2. Using no miscellaneous functions

By setting to bits 2, 3, and 4 of parameter 079, operation can be selected from the following two options.

Select the operation by referring to the manual written by the machine tool builder.

(1) Rotating in the direction in which an angular displacement becomes shortest

This is valid only in absolute mode. A specified angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° when bit 2 of parameter 079 specifies this option.

For example, when G90 B400.0; is specified at a position of 0, the table is rotated by 40° in the positive direction.

(2) Rotating in the specified direction

In the absolute mode, the value set in bit 2 of parameter 079 determines whether an angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° .

In the incremental mode, the angular displacement is not rounded down.

For example, when G90 B720.0; is specified at a position of 0, the table is rotated twice in the positive direction, when the angular displacement is not rounded down.

• Feedrate

The table is always rotated around the indexing axis in the rapid traverse mode.

Dry runs cannot be executed for the indexing axis.

CAUTION

If a reset is made during indexing of the index table, a reference position return must be made before each time the index table is indexed subsequently.

NOTE

- 1 Specify the indexing command in a single block. If the command is specified in a block in which another controlled axis is specified, an alarm (No.136) occurs.
- 2 The waiting state which waits for completion of clamping or unclamping of the index table is indicated on diagnostic screen No. 701#2.
- 3 The miscellaneous function specifying a negative direction is processed in the CNC.
The relevant M code signal and completion signal are sent between the CNC and the machine.
- 4 If a reset is made while waiting for completion of clamping or unclamping, the clamp or unclamp signal is cleared and the CNC exits the completion wait state.

- **Indexing function and other functions**

Table14.10 Index indexing function and other functions

Item	Explanation
Relative position display	This value is rounded down when bit 1 of parameter 079 specifies this option.
Absolute position display	This value is rounded down when bit 2 of parameter 079 specifies this option.
Automatic return from the reference position (G29) 2nd reference position return (G30)	Impossible to return
Movement in the machine coordinate system	Impossible to move
Single direction positioning	Impossible to specify
2nd auxiliary function (B code)	Possible with any address other than B that of the indexing axis.
Operations while moving the indexing axis	Unless otherwise processed by the machine, feed hold, interlock and emergency stop can be executed. Machine lock can be executed after indexing is completed.
SERVO OFF signal	Disabled The indexing axis is usually in the servo-off state.
Incremental commands for indexing the index table	The workpiece coordinate system and machine coordinate system must always agree with each other on the indexing axis (the workpiece zero point offset value is zero.).
Operations for indexing the index table	Manual operation is disabled in the JOG, INC, or HANDLE mode. A manual reference position return can be made. If the axis selection signal is set to zero during manual reference position return, movement is stopped and the clamp command is not executed.

14.11 NORMAL DIRECTION CONTROL (G150, G151, G152)

When a tool with a rotation axis (fourth-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the fourth-axis is always perpendicular to the tool path (Fig. 14.11 (a)).

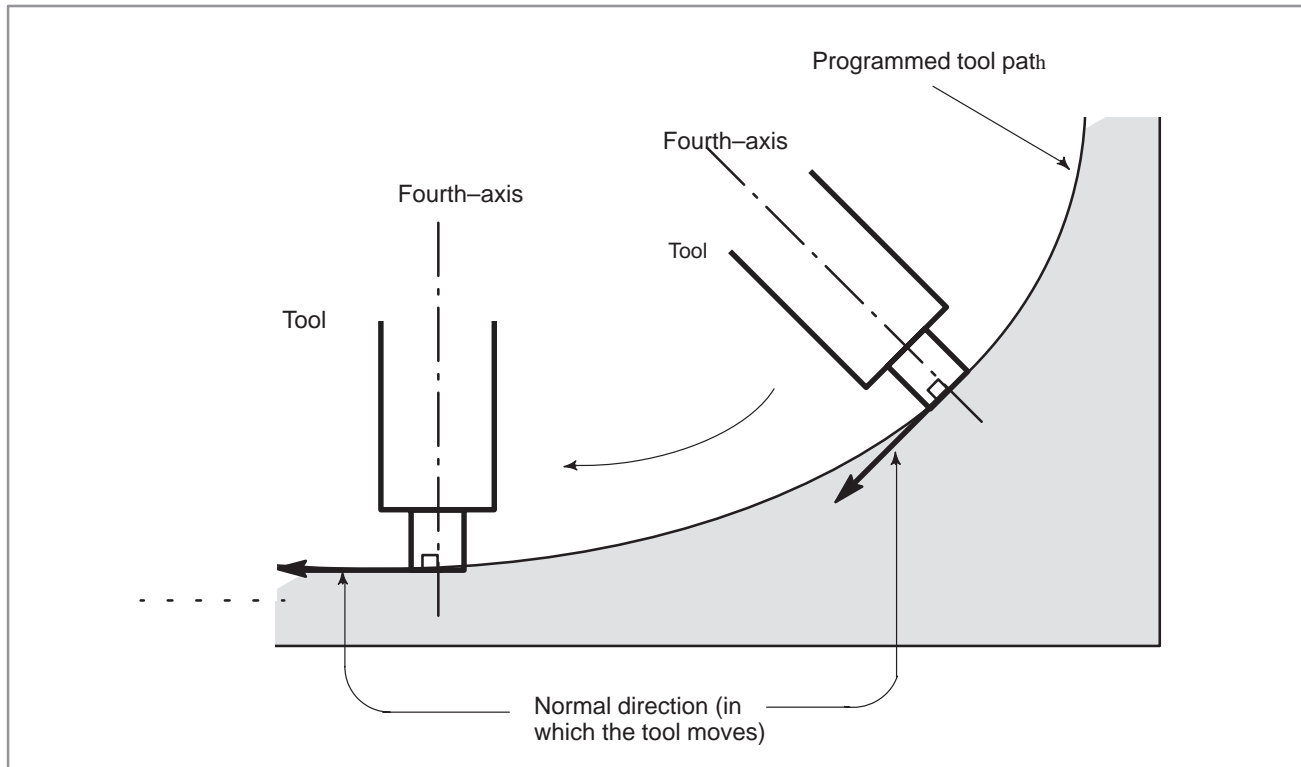


Fig. 14.11 (a) Sample movement of the tool

Format

G code	Function	Explanation
G151	Normal direction control left	If the workpiece is to the right of the tool path looking toward the direction in which the tool advances, the normal direction control left (G151) function is specified.
G152	Normal direction control right	After G151 or G152 is specified, the normal direction control function is enabled (normal direction control mode).
G150	Normal direction control cancel	When G150 is specified, the normal direction control mode is canceled.

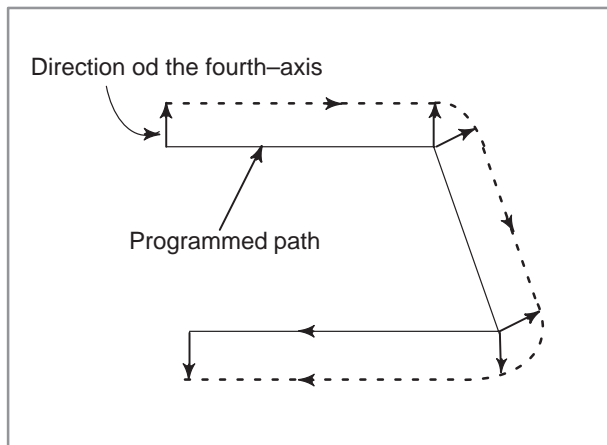


Fig. 14.11 (b) Normal direction control left (G151)

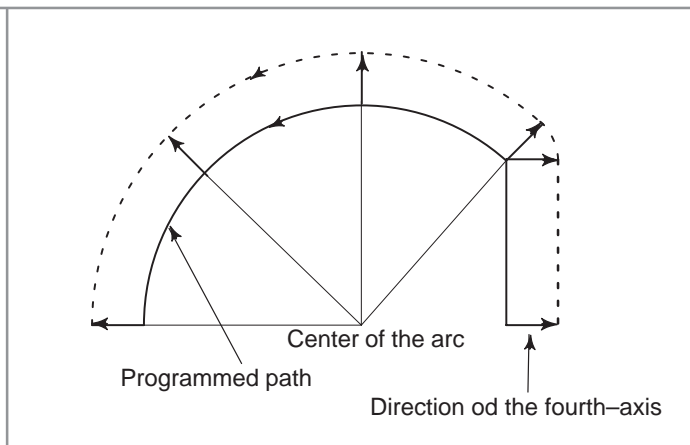


Fig. 14.11 (c) Normal direction control right (G152)

Explanations

• Angle of the fourth axis

When viewed from the center of rotation around the fourth-axis, the angular displacement about the fourth-axis is determined as shown in Fig. 14.11 (d). 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° .

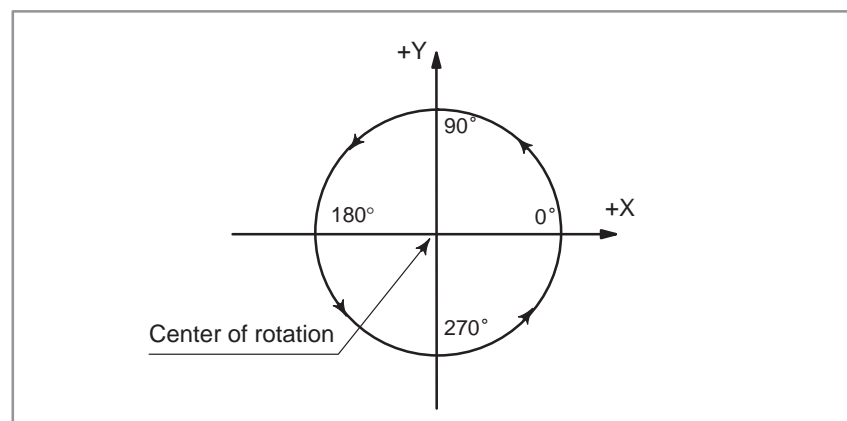


Fig. 14.11 (d) Angle of the fourth axis

• Normal direction control

When the cancel mode is switched to the normal direction control mode, the fourth-axis becomes perpendicular to the tool path at the beginning of the block containing G151 or G152.

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

In the cutter compensation mode, the tool is oriented so that the fourth-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.

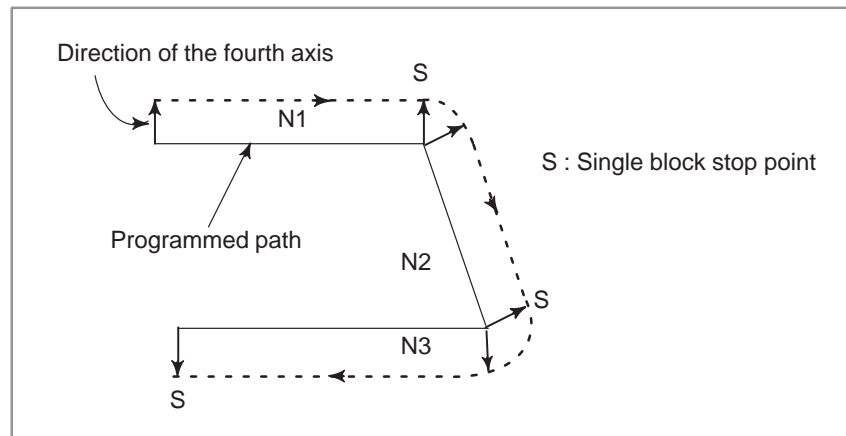


Fig. 14.11 (e) Point at which a single-block stop occurs in the normal direction control mode

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

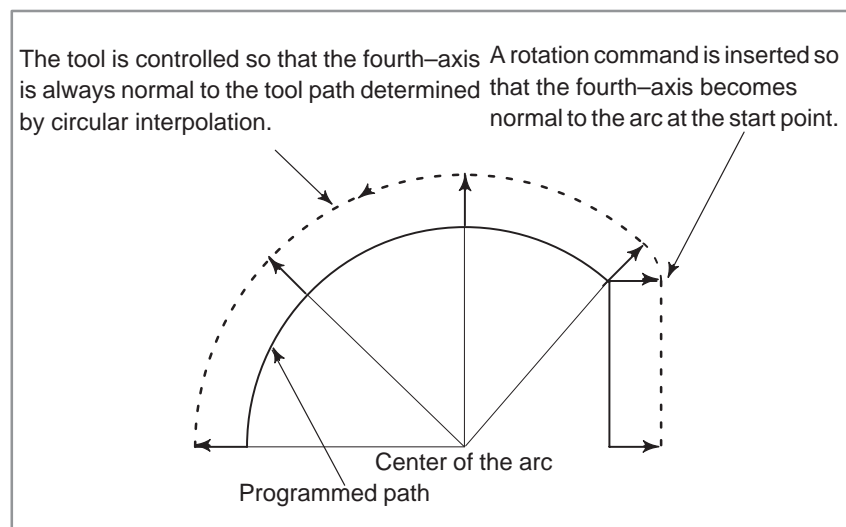


Fig. 14.11 (f) Normal direction control of the circular interpolation

CAUTION

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

● Fourth axis feedrate

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 683. 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 fourth axis during circular interpolation is defined by the following formula.

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

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

Amount of movement of the revolution :

The difference in angles at the beginning and the end of the block.

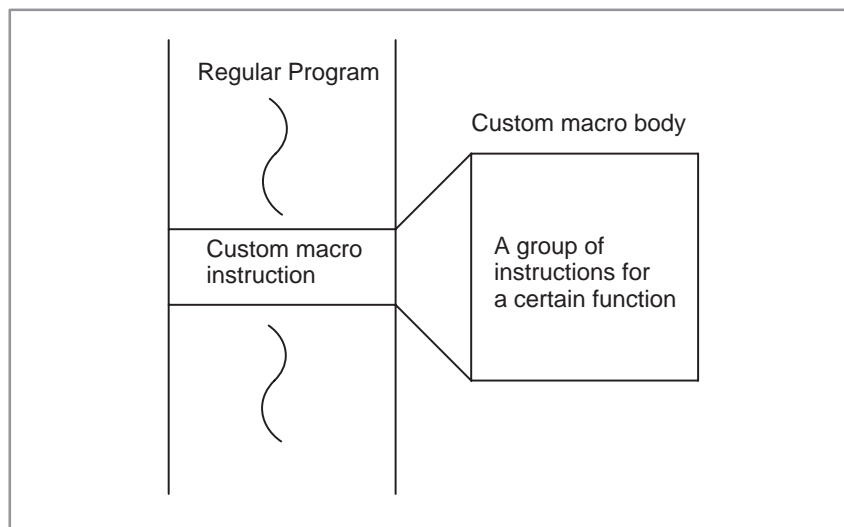
NOTE

- 1 Do not specify any command to the fourth-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 fourth axis with the actual position of the fourth axis on the machine using the coordinate system setting (G92) or the like.
- 3 The helical cutting option is required to use this function.
Helical cutting cannot be specified in the normal direction control mode.
- 4 Normal direction control cannot be performed by the G53 move command.

15

CUSTOM MACRO A

A function covering a group of instructions is stored in memory as same as a subprogram. The stored function is presented by one instruction, so that only the representative instruction need be specified to execute the function. This group of registered instructions is called a “custom macro body” and the representative instruction is called a “custom macro instruction”. The custom macro body may simply be called a macro. And the custom macro instruction may be called a macro call command.



Programmers need only remember representative macro instructions without having to remember all the instructions in a custom macro body. The three most significant points on custom macros are that variables can be used in the custom macro body, operations can be performed on variables and actual values can be assigned to the variables in custom macro instructions.

NOTE

Machine tool builders are requested to attach your custom macro program tape or program list to the CNC unit without fail. If it is necessary to replace part program storage memory due to a failure, FANUC serviceman or end users operators in charge of maintenance should know the contents of custom macro for the purpose of repairing the trouble immediately.

15.1 CUSTOM MACRO COMMAND

The custom macro command is the command to call the custom macro body.

15.1.1 M98 (Single Call)

Command format is as follows :

Format



M98 P__;
 Called macro body program No.

With the above command, the macro body specified by P is called.

15.1.2 Subprogram Call Using M Code

The subprogram can be called using M code set in parameter.

N_ G_ X_ - M98P <p> ;

instead of commanding as above, the same operation can be commanded using following command :

N_ G_ X_ - M <m> ;

The correspondence of M code <m> which calls subprogram and the program number <p> (O9001 to O9003) of the called subprogram shall be set by parameters (No. 0240 to No. 0242).

For subprogram call, a maximum of 3 among M03 to M255, except M30 and M code which does not buffer (parameter No. 111, 112) can be used.

NOTE

- 1 Similarly to M98, signal MF and M code are not output.
- 2 Delivery of argument is not possible.
- 3 Subprogram call M code used in the subprogram which is called by M or T code does not executes subprogram call but as an ordinal M code.

15.1.3 Subprogram Call Using T Code

When parameter (No. 0040 #5) is set beforehand, subprogram (O9000) can be called using T code.

N_G_X_ - T <t> ;

the above command results in the same operation of command of the following 2 blocks.

#149 = <t> ;

N_G_X_ - M98 P9000 ;

The T code t__ is stored in a common variable #0149 as an argument.

NOTE

- 1 It is not possible to command with a same block as that of subprogram call using M code.
- 2 Subprogram call T code used in the subprogram which is called by M or T code does not executes subprogram call but as an original T code.

15.1.4 G66 (Modal Call)

Format



G66 P _ ;
 ↑
 Called macro body program No.

The above command selects the macro modal call mode for NC. In other words, every time each block subsequent to the above command is executed, the macro designated by P is called. Also, an argument can be designated by a block subsequent to the above command. For this argument, refer to 15.1.3. The macro modal call mode is cancelled by the command below.

G67 ;

CAUTION

In MDI mode, the G66 command specifies macro modal calling mode. The G67 command cancels macro modal calling mode. Other commands, however, do not call a macro, even when executed between G66 and G67. Instead, they are executed as ordinary commands. If MDI operation B is specified, macros can be called in MDI mode.

NOTE

- 1 Blocks containing the G66 and G67 commands do not call a macro.
- 2 Commands other than O, N, and P are ignored in the G66 and G67 blocks.
- 3 A repetition count cannot be specified for macro modal calling. Only the four low-order digits of the values specified with P in a G66 block are valid.
- 4 The maximum nesting level for macro modal calling is 1. The maximum nesting level for subprogram calling is 4. However, the maximum total nesting level for macro modal calling and subprogram calling is 4.

15.1.5 Argument Specification

An argument means an actual value given to a variable employed in a called macro. An argument can be specified at all employable addresses except for O. The format of argument specification is the same as in normal CNC command. The limitation at each address as a normal CNC command, such as decimal point, sign, maximum number of digits, etc., is also applicable to the format.

The following table indicates the correspondence between argument specification addresses and variable numbers.

Table 15.1.5 (a) Correspondence between addresses and variable numbers

Variable number (value)	Variable number (flag)	Address	Remarks
#8004	#8104	I	
#8005	#8105	J	
#8006	#8106	K	
#8009	#8109	F	
#8010	#8110	G	
#8011	#8111	H	
#8013	#8113	M	
#8014	#8114	N	
#8016	#8116	P	
#8017	#8117	Q	
#8018	#8118	R	
#8019	#8119	S	
#8020	#8120	T	
#8024	#8124	X	
#8025	#8125	Y	
#8026	#8126	Z	

#8100's variables are flags to indicate whether an argument has been specified or not every call. These variables are 1 if an argument is specified, and 0 if no argument is specified.

#8000's variables show a specified value, if an argument is specified. However, they become as specified below, if no argument is specified.

(a) Reference in CNC command

The address itself is neglected.

(b) Reference in macro command and branch command.

This value is undefined. Use it after confirming #8100's numbers.

15.1.5 (b) Correspondence between G codes of the argument specification and variable numbers

Variable number (value)	Variable number (flag)	G code group number	G codes of the argument specification
#8030	#8130	00	One shot and others
#8031	#8131	01	G00, G01, G02, G03
#8032	#8132	02	G17, G18, G19
#8033	#8133	03	G90, G91
#8035	#8135	05	G94
#8036	#8136	06	G20, G21
#8037	#8137	07	G40, G41, G42
#8038	#8138	08	G43, G44, G49
#8039	#8139	09	G73, G74, G76, G80 to G89
#8040	#8140	10	G98, G99
#8041	#8141	11	G50, G51
#8042	#8142	12	G66, G67
#8045	#8145	15	G61, G62, G63, G64
#8046	#8146	16	G68, G69

If plural G codes are specified a arguments in the same block, a value is input to variables in each group as shown in Table 15.1.5 (b). In this case, #8010 is the smallest group number out of the numbers specified at a time. No value can institute into #8100's and 8000's variables.

15.2 CUSTOM MACRO BODY

In the custom macro body, the CNC command, which uses ordinary CNC command variables, calculation, and branch command can be used. The custom macro body starts from the program No. which immediately follows O and ends at M99.

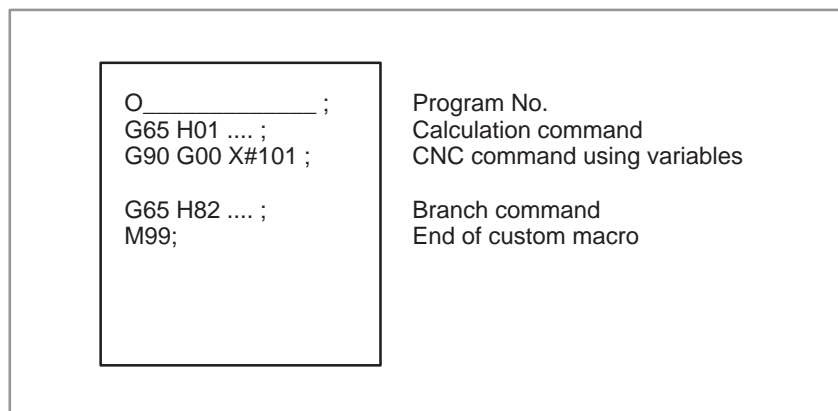


Fig.15.2 Construction of the custom macro body

15.2.1 Variables

A variable can be specified to make the macro flexible and versatile by applying the calculated variable when calling the macro or when executing the macro itself. Multiple variables are identified from each other by variable numbers.

(1) How to express variables

Variables are expressed by variable numbers following # as shown below.

#i (i = 1, 2, 3, 4

(Example) #5, #109, #1005

(2) How to quote variables

A numeral following an address can be replaced by a variable. Assume that <Address> #1 or <Address> - #1 is programmed, and it means that the variable value or its complement serves as the command value of the address.

(Example)

F#103F15 was commanded when #103=15

Z-#110 - Z-250 was commanded when #110=250

G#130 - G3 was commanded when #103=3.

When replacing a variable number with a variable, it is not expressed as “##100”, for example, but express as “#9100”. That is, “9” next to “#” indicates the substitute of the variable number, while the lower number to be replaced.

(Example)

If #100=105 and #105=-500, “X#9100” indicates that X-500 was commanded, and “X-#9100” indicates that X500 was commanded.

NOTE

- 1 No variable can be quoted at address O and N. Neither O#100 nor N#120 can be programmed.
- 2 It is not possible to command a value exceeding the maximum command value set in each address.
When #30=120, G#30 has exceeded the maximum command value.

15.2.2 Kind of Variables

Variables are sorted into common variables and system variables according to variable numbers, and their applications and characters differ from each other.

(1) Common variable #100 to #149 and #500 to #531

Common variables are common to main programs and each macro called from these main programs. That is, #i in a macro is equal to #i in another macro.

Common variables #100 to #149 are cleared when the power is turned off, and reset to "0" just after power was turned on. Common variables #500 to #531 are not cleared, even if power is turned off, and their values remain unchanged.

(2) System variable

The system variables are defined as variables whose applications remain fixed.

(a) Tool offset amount #1 to #99, #2000 to #2200

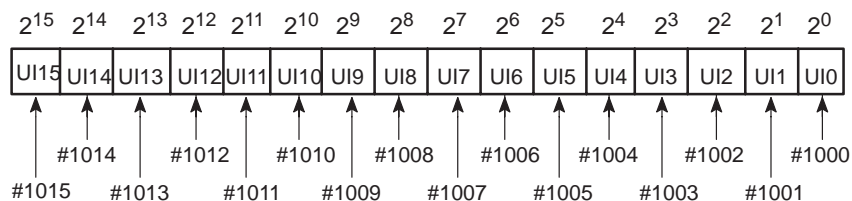
The offset amount can be known by reading system variable #1 to #99 values for tool offset amounts, and these values can be changed by substituting them into system variables #1 to #99.

Among these offset numbers 1 to 99, those which are not used as offset amounts can be treated as hold type common variables (#500 to #531).

The system variables #2001 to #2200 correspond to the tool offset numbers 1 to 200. They can be read and substituted as the same as #1 to #99. #2000 always can read 0.

(b) Interface input signals #1000 to #1015, #1032

Interface signals can be known, by reading system variables #1000 to #1015 for reading interface signals.



Input signal	Variable value
Contact closed	1
Contact opened	0

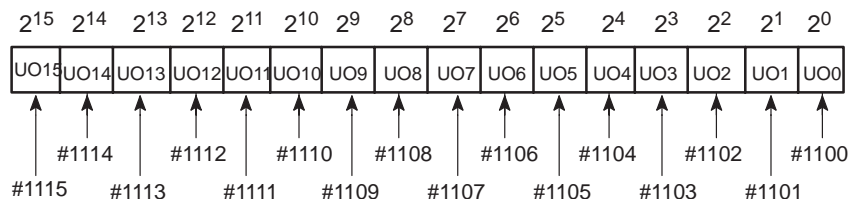
By reading system variable #1032, all the input signals can be read at once.

$$\#1032 = \sum_{i=0}^{15} \#(1000 + i) \times 2^i$$

NOTE

- 1 No value can be substituted into system variables #1000 to #1032.
- 2 System variables #1000 to #1015 can be displayed by diagnostic function.
DGNOS No.110 U10 to U17
DGNOS No.111 U18 to U15
- 3 System variables #1000 to #1032 can be used only when PMC is combined.

(c) Interface output signals #1100 to #1115, #1132, #1133 A value can be substituted into system variables #1100 to #1115 for sending the interface signals.



Input signal	Variable value
Contact closed	1
Contact opened	0

By substituting a value into system variable #1132, all output signals (UO0 to UO15) can be sent out at once.

$$\#1132 = \sum_{i=0}^{15} \#(1100 + i) \times 2^i$$

32 interface signals (UO100 to UO131) can be sent out by #1133 at once.

CAUTION

If any other number than '0' or '1' is substituted into system variables #1100 to #1115, it is treated as '1'.

NOTE

- 1 It is possible to read the values of system variables #1100 to #1133.
- 2 System variables #1100 to #1115 and #1133 can be displayed by diagnostic function.
 DGNOS No.162 UO0 to UO7
 No.163 UO8 to UO15
 No.196 UO100 to UO107
 No.197 UO108 to UO115
 No.198 UO116 to UO123
 No.199 UO124 to UO131
- 3 System variables #1100 to #1133 can be used only when PMC is combined.

(d) Clock information #3011, 3012

It is possible to know the year, month, day, hour, minute, and second by reading system variables #3011, #3012.

Kind	System variable
Year, Month, Day	#3011
Hour, Minute, Second	#3102

(Example) When it is May 20, 1987 4:17 5" PM
 #3012=19870520,
 #3012=161705

(e) Number of necessary parts, number of machined parts

Kind	System variable
Number of machined parts	#3901
Number of necessary parts	#3902

NOTE

Do not substitute a negative value.

(f) Modal information #4001 to #4120

It is possible to know the current values of modal information (modal command given till immediately preceding block) by reading values of system variables #4001 to #4120.

Variables	Modal information
#4001	G code (group 01)
#4002	G code (group 02)
#4003	G code (group 03)
...	
#4022	G code (group 22)
#4102	B code
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Sequence No.
#4115	Program No.
#4119	S code
#4120	T code

NOTE

The unit will be the one being used when the command is given.

(g) Position information #5001 to #5083

The position information can be known by reading system variables #5001 to #5083. The unit of position information is 0.001 mm in metric input and 0.0001 inch in inch input.

System variables	Position information	Reading while moving	Cutter and tool length compensation
#5001 #5002 #5003 #5004	Block end point position of X axis (ABSIO) Block end point position of Y axis Block end point position of Z axis Block end point position of 4th axis	Possible	Not considered. Position of tool nose (program command position)
#5021 #5022 #5023 #5024	X axis coordinate position (ABSMT) Y axis coordinate position Z axis coordinate position 4th axis coordinate position	Impossible	Considered. Position of tool reference point (Machine coordinate)
#5041 #5042 #5043 #5044	Present position of X axis (ABSOT) Present position of Y axis Present position of Z axis Present position of 4th axis	Impossible	Considered. Position of tool reference point (Absolute coordinate system)
#5061 #5062 #5063 #5064	Skip signal position of X axis (ABSKP) Skip signal position of Y axis Skip signal position of Z axis Skip signal position of 4th axis	Possible	Considered. Position of tool reference point
#5080 #5081 #5082 #5083	Value of cutter compensation Value of tool length compensation (X axis) Value of tool length compensation (Y axis) Value of tool length compensation (Z axis)	Possible	

NOTE

- 1 It is not possible to substitute any value into system variables #5001 to #5083.
- 2 When the skip signal doesn't turn on at G31, the skip signal position is the end point of that block.

15.2.3 Operation Instruction and Branch Instruction (G65)

General Form

G65HmP#i Q#j R#k ;

m : Indicates operation instruction and branch instruction at 01 to 99

#i : Variable name to which arithmetic result is loaded.

#j : Variable name 1 to be operated. A constant is also acceptable.

#k : Variable name 2 to be operated. A constant is also acceptable.

Meaning #1 = #j ⊕ #k

↑
Operator (Specified by Hm)

Example

P#100 Q#101 R#102#100=#101⊕#102

P#100 Q#101 R15#100=#101⊕15

P#100 Q-100 R#102#100=-100⊕#102

P#100 Q120 R-50#100=120⊕-50

P#100 Q-#101 R#102--#100=-#101⊕#012

CAUTION

- 1 No decimal point can be put to variable values. Therefore, the meaning of each value is the same as that designated without decimal point when quoted in each address.

(Example) #100 = 10

X#100 ----- 0.01 mm (metric input)

- 2 Those indicating an angle must be expressed by degree, and the least input increment is 1/1000 degree.

(Example) 100 ... 0.1°

NOTE

H code specified by G65 does not affect any selection of offset amount.

Table 15.2.3

G code	H code	Function	Definition
G65	H01	Definition, substitution	$\#i = \#j$
G65	H02	Addition	$\#i = \#j + \#k$
G65	H03	Subtraction	$\#i = \#j - \#k$
G65	H04	Product	$\#i = \#j \times \#k$
G65	H05	Division	$\#i = \#j \div \#k$
G65	H11	Logical sum	$\#i = \#j$. OR. $\#k$
G65	H12	Logical product	$\#i = \#j$. AND. $\#k$
G65	H13	Exclusive OR	$\#i = \#j$. XOR. $\#k$
G65	H21	Square root	$\#i = \sqrt{\#j}$
G65	H22	Absolute value	$\#i = \#j $
G65	H23	Remainder	$\#i = \#j - \text{trunc}(\#j / \#k) \times \#k$ (trunc : Discard fractions less than 1)
G65	H24	Conversion from BCD to binary	$\#i = \text{BIN}(\#j)$
G65	H25	Conversion from binary to BCD	$\#i = \text{BCD}(\#j)$
G65	H26	Combined multiplication/division	$\#i = (\#j \times \#k) \div \#l$
G65	H27	Combined square root 1	$\#i = \sqrt{\#J^2 + \#K^2}$
G65	H28	Combined square root 2	$\#i = \sqrt{\#J^2 - \#K^2}$
G65	H31	Sine	$\#i = \#j \cdot \text{SIN}(\#k)$
G65	H32	Cosine	$\#i = \#j \cdot \text{COS}(\#k)$
G65	H33	Tangent	$\#i = \#j \cdot \text{TAN}(\#k)$
G65	H34	Arctangent	$\#i = \text{ATAN}(\#j / \#k)$
G65	H80	Unconditional divergence	GOTO n
G65	H81	Conditional divergence 1	IF $\#j = \#k$, GOTO n
G65	H82	Conditional divergence 2	IF $\#j \neq \#k$, GOTO n
G65	H83	Conditional divergence 3	IF $\#j > \#k$, GOTO n
G65	H84	Conditional divergence 4	IF $\#j < \#k$, GOTO n
G65	H85	Conditional divergence 5	IF $\#j \leq \#k$, GOTO n
G65	H86	Conditional divergence 6	IF $\#j \geq \#k$, GOTO n
G65	H99	P/S alarm occurrence	P/S alarm number 500 + n occurrence

● Operation instruction

- (a) Definition and substitution of variable $\#i = \#j$
 G65 H01 P $\#i$ Q $\#j$;
 [Example] G65 H01 P $\#101$ Q1055 ; ($\#101=1005$)
 G65 H01 P $\#101$ Q $\#110$; ($\#101=\#110$)
 G65 H01 P $\#101$ Q $\#112$; ($\#101=-\#112$)
- (b) Addition $\#i = \#j + \#k$
 G65 H02 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H02 P $\#101$ Q $\#102$ R15 ; ($\#101=\#102+15$)
- (c) Subtraction $\#i = \#j - \#k$
 G65 H03 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H03 P $\#101$ Q $\#102$ R $\#103$; ($\#101=\#102-\#103$)
- (d) Product $\#i = \#j \times \#k$
 G65 H04 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H04 P $\#101$ Q $\#102$ R $\#103$; ($\#101=\#102 \times \#103$)
- (e) Division $\#i = \#j \div \#k$
 G65 H05 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H05 P $\#101$ Q102 R $\#103$; ($\#101=\#102 \div \#103$)
- (f) Logical sum $\#i = \#j. \text{OR}. \#k$
 G65 H11 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H11 P $\#101$ Q102 R $\#103$; ($\#101=\#102. \text{OR}. \#103$)
- (g) Logical product $\#i = \#j. \text{AND}. \#k$
 G65 H12 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H12 P $\#101$ Q $\#102$ R $\#103$;
 ($\#101=\#102. \text{AND}. \#103$)
- (h) Exclusive OR $\#i = \#j. \text{XOR}. \#k$
 G65 H13 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H13 P $\#101$ Q $\#102$ R $\#103$;
 ($\#101=\#102. \text{XOR}. \#103$)
- (i) Square root $\#i = \sqrt{\#j}$
 G65 H21 P $\#i$ Q $\#j$;
 [Example] G65 H21 P $\#101$ Q $\#102$; ($\#101=\sqrt{\#102}$)
- (j) Absolute value $\#i = |\#j|$
 G65 H22 P $\#i$ Q $\#j$;
 [Example] G65 H22 P $\#101$ Q $\#102$; ($\#101=|\#102|$)
- (k) Remainder $\#i = \#j - \text{trunc}(\#j/\#k) \times \#k$
 trunc : Discard fractions less than 1
 G65 H23 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H23 P $\#101$ Q $\#102$ R $\#103$;
 ($\#101=\#102 - \text{trunc}(\#102/\#103) \times \#103$)
- (l) Conversion from BCD to binary $\#i = \text{BIN}(\#j)$
 G65 H24 P $\#i$ Q $\#j$;
 [Example] G65 H24 P $\#101$ Q $\#102$; ($\#101=\text{BIN}(\#102)$)
- (m) Conversion from binary to BCD $\#i = \text{BCD}(\#j)$
 G65 H25 P $\#i$ Q $\#j$;
 [Example] G65 H25 P $\#101$ Q $\#102$; ($\#101=\text{BCD}(\#102)$)
- (n) Combined multiplication/division $\#i = (\#1 \times \#j) \div \#k$
 G65 H26 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H26 P $\#101$ Q $\#102$ R $\#103$;
 ($\#101=(\#101 \times \#102) (\div \#103)$)

(o) Combined square root 1 $\#i = \sqrt{\#j^2 + \#k^2}$

G65 H27 P#i Q#j R#k ;

[Example] G65 H27 P#101 Q#102 R#103 ;
(#101= $\sqrt{\#102^2 + 103^2}$)

(p) Combined square root 2 $\#i = \sqrt{\#j^2 - \#k^2}$

G65 H28 P#i Q#j R#k ;

[Example] G65 H28 P#101 Q#102 R#103 ;
(#101= $\sqrt{\#102^2 - 103^2}$)

(q) Sine $\#i = \#j \cdot \text{SIN}(\#k)$ (degree unit)

G65 H31 P#i Q#j R#k ;

[Example] G65 H31 P#101 Q#102 R#103 ; (#101=#102·SIN(#103))

(r) Cosine $\#i = \#j \cdot \text{COS}(\#k)$ (degree unit)

G65 H32 P#i Q#j R#k ;

[Example] G65 H32 P#101 Q#102 R#103 ;
(#101=#102·COS (#103))

(s) Tangent $\#i = \#j \cdot \text{TAN}(\#k)$ (degree unit)

G65 H33 P#i Q#j R#k ;

[Example] G65 H33 P#101 Q#102 R#103 ;
(#101=#102·TAN (#103))

(t) Arctangent $\#i = \text{ATAN}(\#j/\#k)$ (degree unit)

G65 H34 P#i Q#j R#k ; ($0^\circ \leq \#i < 360^\circ$)

[Example] G65 H34 P#101 Q#102 R#103 ;
(#101=ATAN (#102 / #103))

CAUTION

If either Q or R necessary for each arithmetic operation was not indicated, its value is calculated as '0'.

NOTE

- 1 Angle in (q) to (t) must be indicated by degree and the least input increment is 1/1000 degree.
- 2 All figures below decimal point are truncated if each arithmetic result includes decimal point.

● Branch instruction

(a) Unconditional branch

G65 H80 Pn ; n : Sequence number

[Example] G65 H80 P120 ; (Diverge to N120)

(b) Conditional divergence 1 #j. EQ. #k (=)

G65 H81 Pn Q#j R#k ; n : Sequence number

[Example] G65 H81 P1000 Q#101 R#102 ;

#101=#102, go to N1000

#101≠#102, go to next

(c) Conditional divergence 2 #j. NE. #k (≠)

G65 H82 Pn Q#j R#k ; n : Sequence number

[Example] G65 H82 P1000 Q#101 R#102 ;

#101≠#102, go to N1000

#101=#102, go to

(d) Conditional divergence 3 #j. GT. #k (>)

G65 H83 Pn Q#j R#k ; n : Sequence number

[Example] G65 H83 P1000 Q#101 R#102 ;

#101 > #102, go to N1000

#101 ≤ #102, go to next

(e) Conditional divergence 4 #j. LT. #k (<)

G65 H84 Pn Q#j R#k ; n : Sequence number

[Example] G65 H84 P1000 Q#101 R#102 ;

#101 < #102, go to N1000

#101 ≥ #102, go to next

(f) Conditional divergence 5 #j. GE. #k (≥)

G65 H85 Pn Q#j R#k ; n : Sequence number

[Example] G65 H85 P1000 Q#101 R#102 ;

#101 ≥ #102, go to N1000

#101 < #102, go to next

(g) Conditional divergence 6 #j. LE. #k (≤)

G65 H86 Pn Q#j R#k ; n : Sequence number

[Example] G65 H86 P1000 Q#101 R#102 ;

#101 ≤ #102, go to N1000

#101 > #102, go to next

(h) P/S alarm occurrence

G65 H99 Pn ; Alarm No. : 500+n

[Example] G65 H99 P15 ; P/S alarm 515 occurrence

NOTE

- 1 If positive numbers were designated as sequence numbers at branch designations, they are searched forward first and then, backward. If negative numbers were designated, they are searched backward first and then, forward.
- 2 Sequence number can also be designated by variables.
(Example) G65 H81 P#100 Q#101 R#102 ;
When conditions are satisfied, processing branches to the block having the sequence number designated with #100.

15.2.4

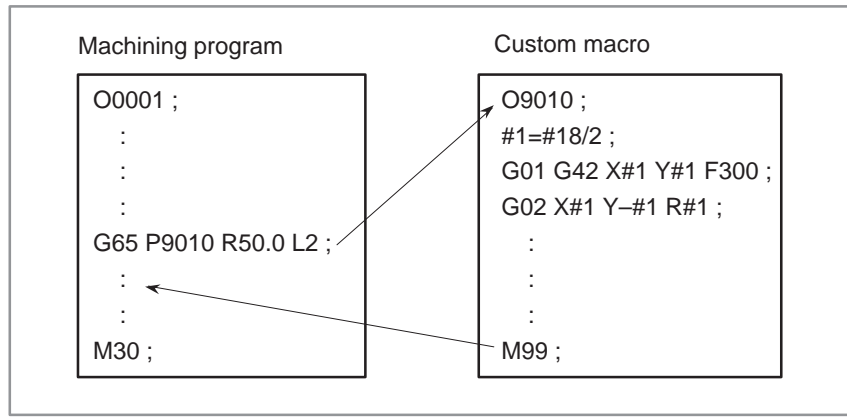
Notes on Custom Macro

- 1) How to input “#”
For standard MDI key, when “/ # EOB” key is depressed after address G, X, Y, Z, R, I, J, K, F, H, M, S, T, or P, # code is input
- 2) It is also possible to give a macro instruction in the MDI mode. However address data other than G65 are not displayed by keying operation.
- 3) Address H, P, Q and R of macro instruction must always be written after G65. Address O and N only are writable before G65.
H02 G65 P#100 Q#101 R#102 ; ----- Error
N100 G65 H01 P#100 Q10 ;----- Correct
- 4) Single block
Generally, the macro instruction block does not stop even if single block stop is turned on. However, by setting parameter No. 0011#5, it is possible to make single block effective. This is used for macro testing.
- 5) Variable values can be taken within a range of -2^{31} to $2^{31}-1$, but they are not displayed correctly, except for -99999999 to 99999999 . If they exceed the above range, they are displayed as *****.
- 6) It is possible to nest subprograms up to four times.
- 7) Since an integer only is employable as the variable value, in case the operation results with decimal numbers, the figures below decimal point truncated, if an arithmetic result contains a fraction part. Particularly be careful with the arithmetic sequence, accordingly.
[Example]
When #100=35, #101=10, #102=5, the following results.
#110=#100 ÷ #101 (=3)
#111=#110×#102 (=15)
#120=#100×#102 (=175)
#121=#120 ÷ #101 (=17)
#111=15 and #121=17
- 8) When a custom macro is loaded from a paper tape in the EIA code, ‘&’ code is treated as ‘#’, because there is no ‘#’ code in the EIA code.

16

CUSTOM MACRO B

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.



16.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 and X100.0.

With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;
G01 X#1 F300 ;
```

Explanation

- **Variable representation**

When specifying a variable, specify a number sign (#) followed by a variable number. Personal computers allow a name to be assigned to a variable, but this capability is not available for custom macros.

Example: #1

An expression can be used to specify a variable number. In such a case, the expression must be enclosed in brackets.

Example: #[#1+#2-12]

- **Range of variable values**

Local and common variables can have value 0 or a value in the following ranges :

-10^{47} to -10^{-29}

0

10^{29} to 10^{47}

If the result of calculation turns out to be invalid, an alarm No. 111 is issued.

- **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.

- **Undefined variable**

When the value of a variable is not defined, such a variable is referred to as a “null” variable. Variable #0 is always a null variable. It cannot be written to, but it can be read.

• Types of variables

Variables are classified into four types by variable number.

Table 16.1 Types of variables

Variable number	Type of variable	Function
#0	Always null	This variable is always null. No value can be assigned to this variable.
#1 – #33	Local variables	Local variables can only be used within a macro to hold data such as the results of operations. When the power is turned off, local variables are initialized to null. When a macro is called, arguments are assigned to local variables.
#100 – #149 (#199) #500 – #531 (#999)	Common variables	Common variables can be shared among different macro programs. When the power is turned off, variables #100 to #149 are initialized to null. Variables #500 to #531 hold data even when the power is turned off. As an option, common variables #150 to #199 and #532 to #999 are also available. However, when these values are using, the length of the tape that can be used for storage decreases by 8.5 m.
#1000 –	System variables	System variables are used to read and write a variety of NC data items such as the current position and tool compensation values.

NOTE

Common variables #150 to #199 and #532 to #999 are optional.

• Referencing variables

To reference the value of a variable in a program, specify a word address followed by the variable number. When an expression is used to specify a variable, enclose the expression in brackets.

Example: G01X[#1+#2]F#3;

A referenced variable value is automatically rounded according to the least input increment of the address.

Example:

When G00X#1; is executed on a 1/1000-mm CNC with 12.3456 assigned to variable #1, the actual command is interpreted as G00X12.346;.

To reverse the sign of a referenced variable value, prefix a minus sign (–) to #.

Example: G00X–#1;

When an undefined variable is referenced, the variable is ignored up to an address word.

Example:


When the value of variable #1 is 0, and the value of variable #2 is null, execution of G00X#1Y#2; results in G00X0;.

● **Displaying variable values**



Procedure for displaying variable values

Procedure

- 1

Press the  key to display the tool compensation screen.
- 2

Press the soft key **[MACRO]** to display the macro variable screen.
- 3

After press the  key, enter a variable number, then press  key.
The cursor moves to the position of the entered number.

VARIABLE		O1234 N1234	
NO.	DATA	NO.	DATA
100	123.456	108	
101	0.000	109	
102		110	
103		111	
104		112	
105		113	
106		114	
107		115	
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000	B	0.000
[OFFSET] [MACRO] [MENU] [WORK] []			

- When the value of a variable is blank, the variable is null.
- The mark ********* indicates an overflow (when the absolute value of a variable is greater than 99999999) or an underflow (when the absolute value of a variable is less than 0.0000001).

Limitations

Program numbers, sequence numbers, and optional block skip numbers cannot be referenced using variables.

Example:
Variables cannot be used in the following ways:
O#1;
/#2G00X100.0;
N#3Y200.0;

16.2 SYSTEM VARIABLES

System variables can be used to read and write internal NC data such as tool compensation values and current position data. Note, however, that some system variables can only be read. System variables are essential for automation and general-purpose program development.

Explanations

- **Interface signals**

Signals can be exchanged between the programmable machine controller (PMC) and custom macros.

Table 16.2(a) System variables for interface signals

Variable number	Function
#1000–#1015 #1032	A 16-bit signal can be sent from the PMC to a custom macro. Variables #1000 to #1015 are used to read a signal bit by bit. Variable #1032 is used to read all 16 bits of a signal at one time.
#1100–#1115 #1132	A 16-bit signal can be sent from a custom macro to the PMC. Variables #1100 to #1115 are used to write a signal bit by bit. Variable #1132 is used to write all 16 bits of a signal at one time.
#1133	Variable #1133 is used to write all 32 bits of a signal at one time from a custom macro to the PMC. Note, that values from –99999999 to +99999999 can be used for #1133.

For detailed information, refer to the Connection Manual (Function) (B-61393E-2).

- **Tool compensation values**

Tool compensation values can be read and written using system variables. Usable variable numbers depend on the number of compensation pairs, whether a distinction is made between geometric compensation and wear compensation, and whether a distinction is made between tool length compensation and cutter compensation. When the number of compensation pairs is not greater than 200, variables #2001 to #2400 can also be used.

Table 16.2(b) System variables for tool compensation memory A

Compensation number	System variable
1	#10001 (#2001)
:	:
200	#10200 (#2200)
:	:
400	#10400

Table 16.2(c) System variables for tool compensation memory B

Compensation number	Geometry compensation	Wear compensation
1	#11001 (#2201)	#10001 (#2001)
:	:	:
200	#11200 (#2400)	#10200 (#2200)
:	:	:
400	#11400	#10400

- **Macro alarms**

Table 16.2(d) System variable for macro alarms

Variable number	Function
#3000	When a value from 0 to 99 is assigned to variable #3000, the NC stops with an alarm. After an expression, an alarm message not longer than 26 characters can be described. The CRT screen displays alarm numbers by adding 500 to the value in variable #3000 along with an alarm message.

Example: #3000=1(TOOL NOT FOUND);

→ The alarm screen displays “001 TOOL NOT FOUND.”

- **Time information**

Time information can be read and written.

Table 16.2(e) System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 16–millisecond increments at all times. When the power is turned on, the value of this variable is reset to 0. When 65535 milliseconds is reached, the value of this timer returns to 0.
#3002	This variable functions as a timer that counts in 1–hour increments when the cycle start lamp is on. This timer preserves its value even when the power is turned off. When 1145324.612 hours is reached, the value of this timer returns to 0.
#3011	This variable can be used to read the current date (year/month/day). Year/month/day information is converted to an apparent decimal number. For example, March 28, 1993 is represented as 19930328.
#3012	This variable can be used to read the current time (hours/minutes/seconds). Hours/minutes/seconds information is converted to an apparent decimal number. For example, 34 minutes and 56 seconds after 3 p.m. is represented as 153456.

- **Automatic operation control**

The control state of automatic operation can be changed.

Table 16.2(f) System variable (#3003) for automatic operation control

#3003	Single block	Completion of an auxiliary function
0	Enabled	To be awaited
1	Disabled	To be awaited
2	Enabled	Not to be awaited
3	Disabled	Not to be awaited

- When the power is turned on, the value of this variable is 0.
- When single block stop is disabled, single block stop operation is not performed even if the single block switch is set to ON.

- When a wait for the completion of auxiliary functions (M, S, and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also, distribution completion signal DEN is not output.

Table 16.2(g) System variable (#3004) for automatic operation control

#3004	Feed hold	Feedrate override	Exact stop
0	Enabled	Enabled	Enabled
1	Disabled	Enabled	Enabled
2	Enabled	Disabled	Enabled
3	Disabled	Enabled	Enabled
4	Enabled	Enabled	Disabled
5	Disabled	Enabled	Disabled
6	Enabled	Disabled	Disabled
7	Disabled	Disabled	Disabled

- When the power is turned on, the value of this variable is 0.
- When feed hold is disabled:
 - When the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.
 - When the feed hold button is pressed then released, the feed hold lamp comes on, but the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled.
- When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override switch on the machine operator's panel.
- When exact stop check is disabled, no exact stop check (position check) is made even in blocks including those which do not perform cutting.

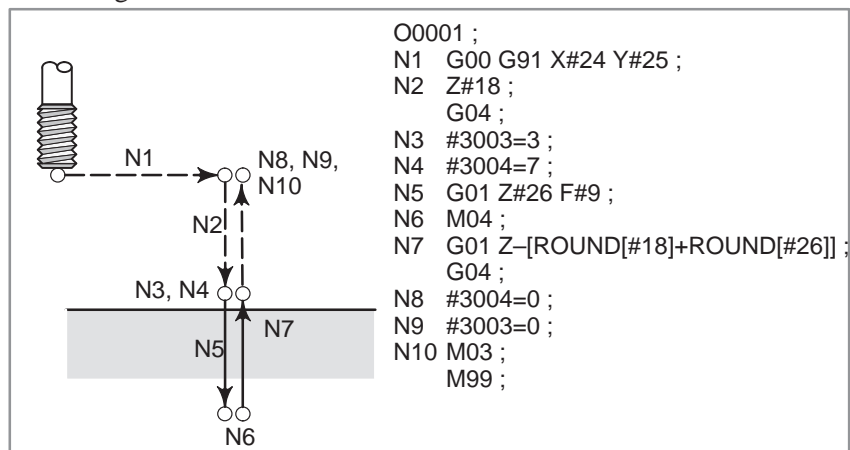


Fig. 16.2 Example of using variable #3004 in a tapping cycle

- **Settings**

Settings can be read and written. Binary values are converted to decimals.

#3005								
Setting	#15	#14	#13	#12	#11	#10	#9	#8
						TAPE	REV4	
Setting	#7	#6	#5	#4	#3	#2	#1	#0
	SEQ	ABS		INCH	ISO	TVON	REVS	REVX

REVX : X-axis mirror image on/off
 REVS : Y-axis mirror image on/off
 TVON : TV check on/off
 ISO : Output code, EIA/ISO
 INCH : Metric input/inch input
 ABS : Incremental programming/absolute programming
 SEQ : Sequence-number automatic insertion on/off
 REV4 : Fourth axis mirror image on/off
 TAPE : F10/11 tape format on/off

- **Number of machined parts**

The number (target number) of parts required and the number (completion number) of machined parts can be read and written.

Table 16.2 (h) System variables for the number of parts required and the number of machined parts

Variable number	Function
#3901	Number of machined parts (completion number)
#3902	Number of required parts (target number)

NOTE

Do not substitute a negative value.

- **Modal information**

Modal information specified in blocks up to the immediately preceding block can be read.

Table 16.2(i) System variables for modal information

Variable number	Function
#4001	G00, G01, G02, G03, G33 (Group 01)
#4002	G17, G18, G19 (Group 02)
#4003	G90, G91 (Group 03)
#4004	(Group 04)
#4005	G94, G95 (Group 05)
#4006	G20, G21 (Group 06)
#4007	G40, G41, G42 (Group 07)
#4008	G43, G44, G49 (Group 08)
#4009	G73, G74, G76, G80–G89 (Group 09)
#4010	G98, G99 (Group 10)
#4011	G50, G51 (Group 11)
#4012	G65, G66, G67 (Group 12)
#4014	G54–G59 (Group 14)
#4015	G61–G64 (Group 15)
#4016	G68, G69 (Group 16)
:	:
#4022	(Group 22)
#4102	B code
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code

Example:

When #1=#4001; is executed, the resulting value in #1 is 0, 1, 2, 3, or 33.

- **Current position**

Position information cannot be written but can be read.

Table 16.2(j) System variables for position information

Variable number	Position information	Coordinate system	Tool compensation value	Read operation during movement
#5001–#5004	Block end point	Workpiece coordinate system	Not included	Enabled
#5021–#5024	Current position	Machine coordinate system	Included	Disabled
#5041–#5044	Current position	Workpiece coordinate system		
#5061–#5064	Skip signal position			Enabled
#5081–#5084	Tool offset value			Disabled
#5101–#5104	Deviated servo position			

- The first digit (from 1 to 4) represents an axis number. Digit 1 corresponds to the X-axis, digit 2 to the Y-axis, digit 3 to the Z-axis, and digit 4 to the fourth axis.
- The tool offset value currently used for execution rather than the immediately preceding tool offset value is held in variables #5081 to 5088.
- The tool position where the skip signal is turned on in a G31 (skip function) block is held in variables #5061 to #5068. When the skip signal is not turned on in a G31 block, the end point of the specified block is held in these variables.
- When read during movement is “disabled,” this means that expected values cannot be read due to the buffering (preread) function.

- **Workpiece coordinate system compensation values (workpiece zero point offset values)**

Workpiece zero point offset values can be read and written.

Table 16.2 (k) System variables for workpiece zero point offset values

Variable number	Function
#2500	First axis external workpiece zero point offset value
#2501	First axis G54 workpiece zero point offset value
:	:
#2506	First axis G59 workpiece zero point offset value
#2600	Second axis external workpiece zero point offset value
#2601	Second axis G54 workpiece zero point offset value
:	:
#2606	Second axis G59 workpiece zero point offset value
#2700	Third axis external workpiece zero point offset value
#2701	Third axis G54 workpiece zero point offset value
:	:
#2706	Third axis G59 workpiece zero point offset value
#2800	Fourth axis external workpiece zero point offset value
#2801	Fourth axis G54 workpiece zero point offset value
:	:
#2806	Fourth axis G59 workpiece zero point offset value
#7001	First-axis workpiece zero point offset value (G54 P1)
:	:
#7004	Fourth-axis workpiece zero point offset value
#7021	First-axis workpiece zero point offset value (G54 P2)
:	:
#7024	Fourth-axis workpiece zero point offset value
:	:
#7941	First-axis workpiece zero point offset value (G54 P48)
:	:
#7944	Fourth-axis workpiece zero point offset value

NOTE

Variables #2500 to #2806 are optional variables for the workpiece coordinate systems. Variables #7001 to #7948 (G54 P1 to G54 P48) are optional variables for additional 48 workpiece coordinate systems.

16.3

ARITHMETIC AND LOGIC OPERATION

The operations listed in Table 16.3(a) can be performed on variables. The expression to the right of the operator can contain constants and/or variables combined by a function or operator. Variables #j and #K in an expression can be replaced with a constant. Variables on the left can also be replaced with an expression.

Table 16.3 (a) Arithmetic and logic operation

Function	Format	Remarks
Definition	#i=#j	
Sum Difference Product Quotient	#i=#j+#k; #i=#j-#k; #i=#j*#k; #i=#j/#k;	
Sine Cosine Tangent Arctangent	#i=SIN[#j]; #i=COS[#j]; #i=TAN[#j]; #i=ATAN[#j]/[#k];	An angle is specified in degrees. 90 degrees and 30 minutes is represented as 90.5 degrees.
Square root Absolute value Rounding off Rounding down Rounding up	#i=SQRT[#j]; #i=ABS[#j]; #i=ROUND[#j]; #i=FIX[#j]; #i=FUP[#j];	
OR XOR AND	#i=#j OR #k; #i=#j XOR #k; #i=#j AND #k;	A logical operation is performed on binary numbers bit by bit.
Conversion from BCD to BIN Conversion from BIN to BCD	#i=BIN[#j]; #i=BCD[#j];	Used for signal exchange to and from the PMC

Explanations

- **Angle units**

The units of angles used with the SIN, COS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

- **ATAN function**

After the ATAN function, specify the lengths of two sides separated by a slash. A result is found where $0 \leq \text{result} < 360$.

Example: When #1=ATAN[1]/[-1], the value of #1 is 135.0

- **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.

G01 X-#2 F300; Moves 2.346 mm.

G00 X[#1+#2];

Since $1.2345 + 2.3456 = 3.5801$, 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. $G00X-[\text{ROUND}[\#1]+\text{ROUND}[\#2]]$ must be specified to return the tool to the original position.

- Rounding up and down to an integer**

With NC, 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**

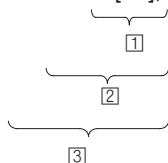
When a function is specified in a program, the first two characters of the function name can be used to specify the function.

**Example: ROUND → RO
FIX → FI**

- Priority of operations**

- ① Functions
- ② Operations such as multiplication and division (*, /, AND, MOD)
- ③ Operations such as addition and subtraction (+, -, OR, XOR)

Example) #1=#2+#3*SIN[#4];

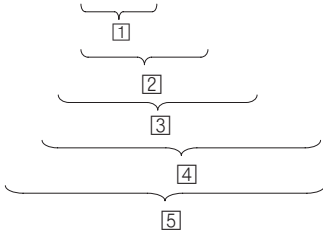


①, ②, and ③ indicate the order of operations.

● Bracket nesting

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, alarm No. 118 occurs.

Example) #1=SIN [[[#2+#3] *#4 +#5] *#6] ;



1 to 5 indicate the order of operations.

Limitations

● Brackets

Brackets ([,]) are used to enclose an expression. Note that parentheses are used for comments.

● Operation error

Errors may occur when operations are performed.

Table 16.3(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 = √b	1.24 × 10 ⁻⁹	3.73×10 ⁻⁹	
a = b + c a = b - c	2.33 × 10 ⁻¹⁰	5.32×10 ⁻¹⁰	(*2) Min $\left \frac{\varepsilon}{b} \right , \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] (*4)	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.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.

Example:

When an attempt is made to assign the following values to variables #1 and #2:

#1=9876543210123.456

#2=9876543277777.777

the values of the variables become:

#1=9876543200000.000

#2=9876543300000.000

In this case, when #3=#2-#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)

- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.

Example:

IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.

Therefore, instead find the difference between the two variables with IF[ABS[#1-#2]LT0.001].

Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).

- Also, be careful when rounding down a value.

Example:

When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997. Here, when #3=FIX[#2]; is specified, the resulting value of variable #1 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]

- **Divisor**

When a divisor of zero is specified in a division or TAN[90], alarm No. 112 occurs.

16.4 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.

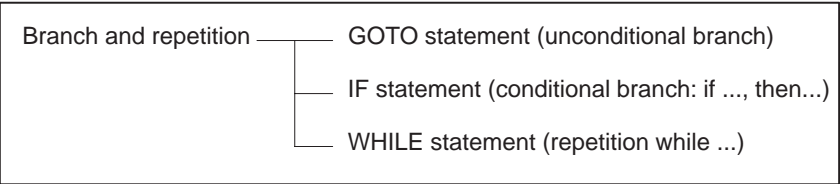
Explanations

- **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 011 is 1.
 - Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode (see Section 16.7).
 - Subprogram call blocks (blocks in which subprogram calls using M98, M codes, or T codes are specified) which only contain addresses O, N, P, and L have the same features as a macro statement.
 - Blocks containing M99 and addresses O, N, P, and L have the same features as a macro statement.
- **NC statements that have the same property as macro statements**

16.5

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:



16.5.1

Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 9999 is specified, alarm No. 128 occurs. A sequence number can also be specified using an expression.

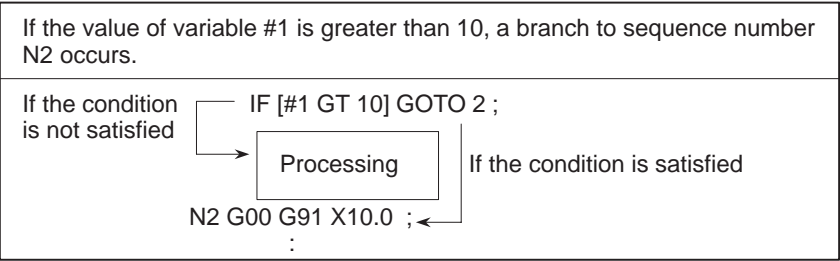
GOTO n ; n: Sequence number (1 to 9999)

Example:
GOTO1;
GOTO#10;

16.5.2

Conditional Branch (IF Statement)

Specify a conditional expression after IF. If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.



Explanations

- **Conditional expression**
A conditional expression must include an operator inserted between two variables or between a variable and constant, and must be enclosed in brackets ([,]). An expression can be used instead of a variable.

• Operators

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 inequality sign cannot be used.

Table 16.5.2 Operators

Operator	Meaning
EQ	Equal to (=)
NE	Not equal to (\neq)
GT	Greater than ($>$)
GE	Greater than or equal to (\geq)
LT	Less than ($<$)
LE	Less than or equal to (\leq)

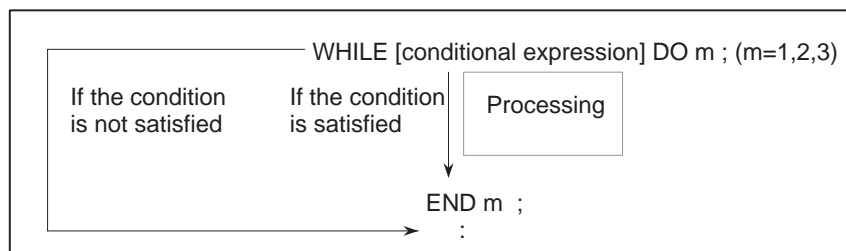
Sample program

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
```

16.5.3 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.

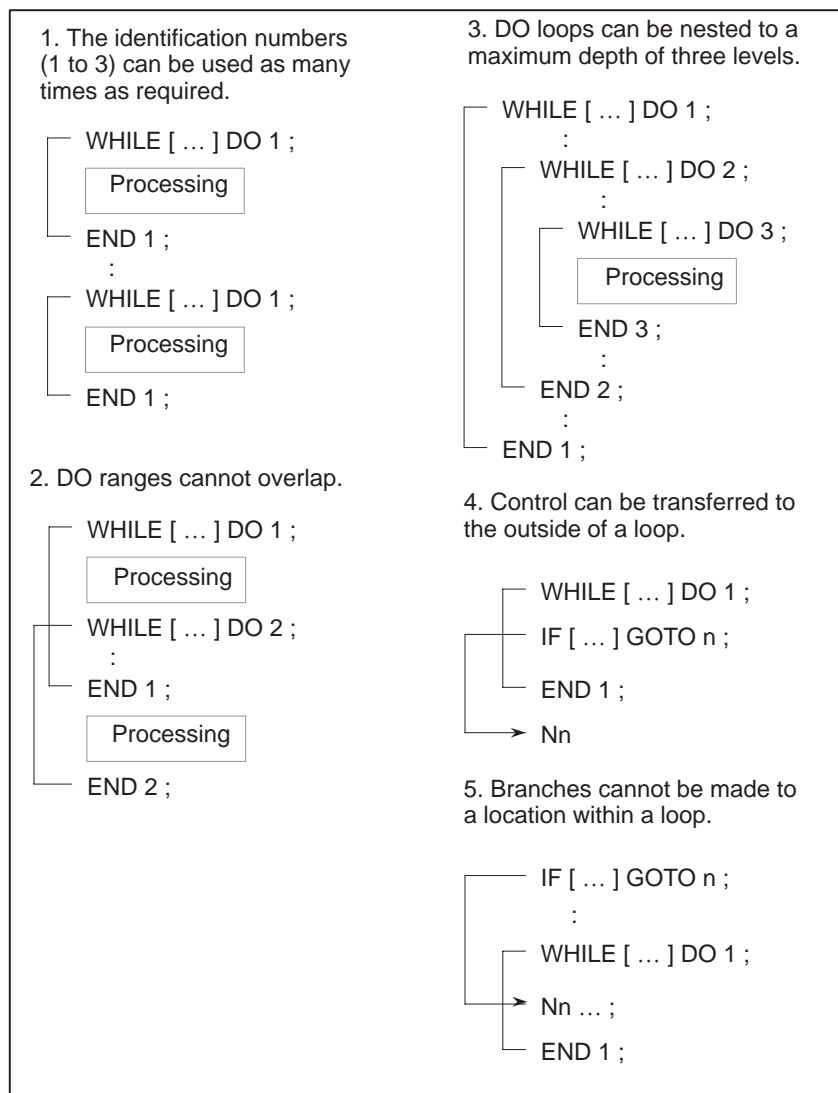


Explanations

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, alarm No. 126 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), alarm No. 124 occurs.



Limitations

• 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. Using the WHILE statement for repetition reduces processing time.

• Undefined variable

In a conditional expression that uses EQ or NE, a null value and zero have different effects. In other types of conditional expressions, a null value 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;
```


16.6 MACRO CALL

A macro program can be called using the following methods:

Macro call	—	Simple call ((G65)
	—	modal call (G66, G67)
	—	Macro call with G code
	—	Macro call with M code
	—	Subprogram call with M code
	—	Subprogram call with T code

Limitations

- **Differences between macro calls and subprogram calls**

Macro call (G65) differs from subprogram call (M98) as described below.

- With G65, an argument (data passed to a macro) can be specified. M98 does not have this capability.
- When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the subprogram is called after the command is executed. On the other hand, G65 unconditionally calls a macro.
- When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the machine stops in the single block mode. On the other hand, G65 does not stop the machine.
- With G65, the level of local variables changes. With M98, the level of local variables does not change.

16.6.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.

G65 P p L ℓ <argument-specification> ; P : Number of the program to call ℓ : Repetition count (1 by default) Argument : Data passed to the macro	
<div style="border: 1px solid black; padding: 5px; width: fit-content;"> O0001 ; : ; G65 P9010 L2 A1.0 B2.0 ; : ; M30 ; </div>	<div style="border: 1px solid black; padding: 5px; width: fit-content;"> O9100 ; #3=#1+#2 ; IF [#3 GT 360] GOTO 9 ; G00 G91 X#3 ; N9 M99 ; </div>

Explanations

- **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 9999 after address L. When L is omitted, 1 is assumed.
- By using argument specification, values are assigned to corresponding local variables.

- **Argument specification**

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

- Addresses G, L, N, O, and P cannot be used in arguments.
- Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.

Argument specification II

Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three-dimensional coordinates as arguments.

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	K ₃	#12	J ₇	#23
B	#2	I ₄	#13	K ₇	#24
C	#3	J ₄	#14	I ₈	#25
I ₁	#4	K ₄	#15	J ₈	#26
J ₁	#5	I ₅	#16	K ₈	#27
K ₁	#6	J ₅	#17	I ₉	#28
I ₂	#7	K ₅	#18	J ₉	#29
J ₂	#8	I ₆	#19	K ₉	#30
K ₂	#9	J ₆	#20	I ₁₀	#31
I ₃	#10	K ₆	#21	J ₁₀	#32
J ₃	#11	I ₇	#22	K ₁₀	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

Limitations

- **Format**

G65 must be specified before any argument.

- **Mixture of argument specifications I and II**

The NC 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.

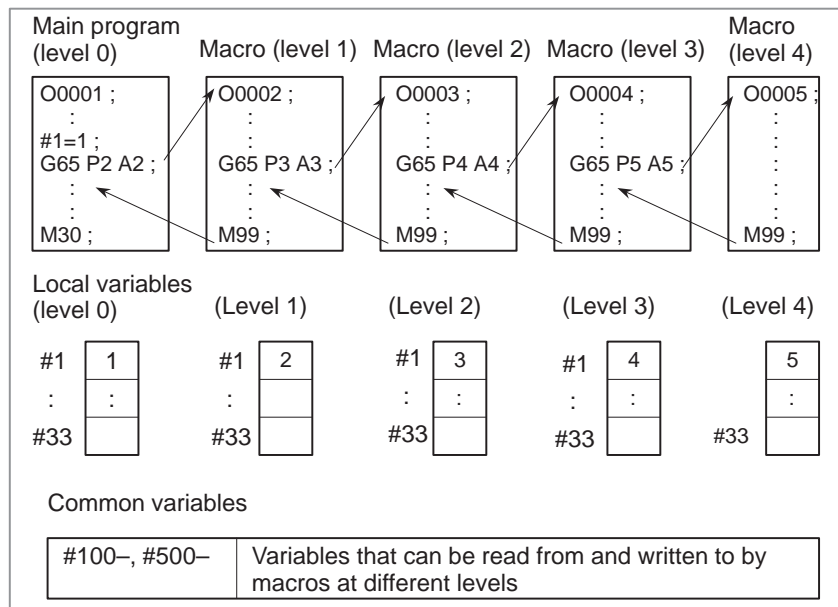
- **Position of the decimal point**

The units used for argument data passed without a decimal point correspond to the least input increment of each address. 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.

- **Call nesting**
- **Local variable levels**

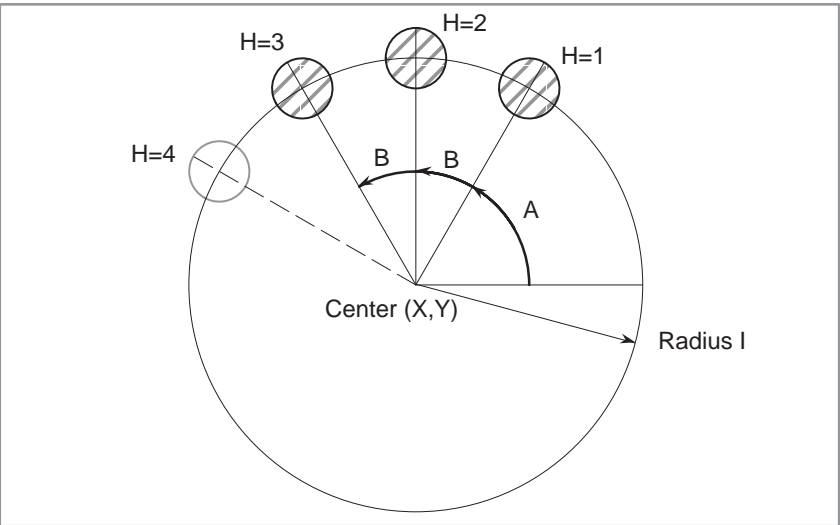
Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

- Local variables from level 0 to 4 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G65 or G66), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the NC.
- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program
(bolt hole circle)

A macro is created which drills H holes at intervals of B degrees after a start angle of A degrees along the periphery of a circle with radius I. The center of the circle is (X,Y). Commands can be specified in either the absolute or incremental mode. To drill in the clockwise direction, specify a negative value for B.



● Calling format

G65 P9100 Xx Yy Zz Rr Ff Ii Aa Bb Hh ;	
X:	X coordinate of the center of the circle (absolute or incremental specification) (#24)
Y:	Y coordinate of the center of the circle (absolute or incremental specification) (#25)
Z:	Hole depth (#26)
R:	Coordinates of an approach point (#18)
F:	Cutting feedrate (#9)
I :	Radius of the circle (#4)
A:	Drilling start angle (#1)
B:	Incremental angle (clockwise when a negative value is specified) (#2)
H:	Number of holes (#11)

- Program calling a macro program

```
O0002;
G90 G92 X0 Y0 Z100.0;
G65 P9100 X100.0 Y50.0 R30.0 Z-50.0 F500 I100.0 A0 B45.0 H5;
M30;
```

- Macro program (called program)

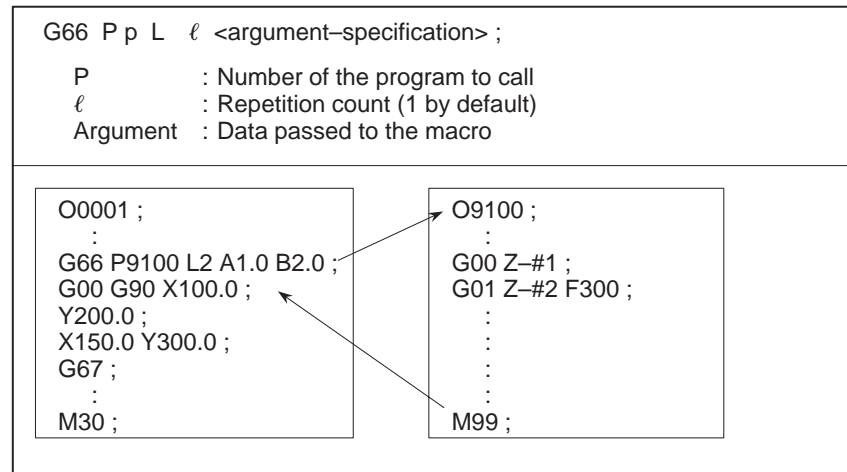
```
O9100;
#3=#4003; ..... Stores G code of group 3.
G81 Z#26 R#18 F#9 K0; (Note) ..... Drilling cycle.
Note: L0 can also be used.
IF[#3 EQ 90]GOTO 1; ..... Branches to N1 in the G90 mode.
#24=#5001+#24; ..... Calculates the X coordinate of the center.
#25=#5002+#25; ..... Calculates the Y coordinate of the center.
N1 WHILE[#11 GT 0]DO 1;
..... Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1]; ... Calculates a drilling position on the X-axis.
#6=#25+#4*SIN[#1]; ... Calculates a drilling position on the Y-axis.
G90 X#5 Y#6; . Performs drilling after moving to the target position.
#1=#1+#2; ..... Updates the angle.
#11=#11-1; ..... Decrements the number of holes.
END 1;
G#3 G80; ..... Returns the G code to the original state.
M99;
```

Meaning of variables:

#3: Stores the G code of group 3.
 #5: X coordinate of the next hole to drill
 #6: Y coordinate of the next hole to drill

16.6.2 Modal Call (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.



Explanations

• 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 9999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.

• Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

• Call nesting

Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

• Modal call nesting

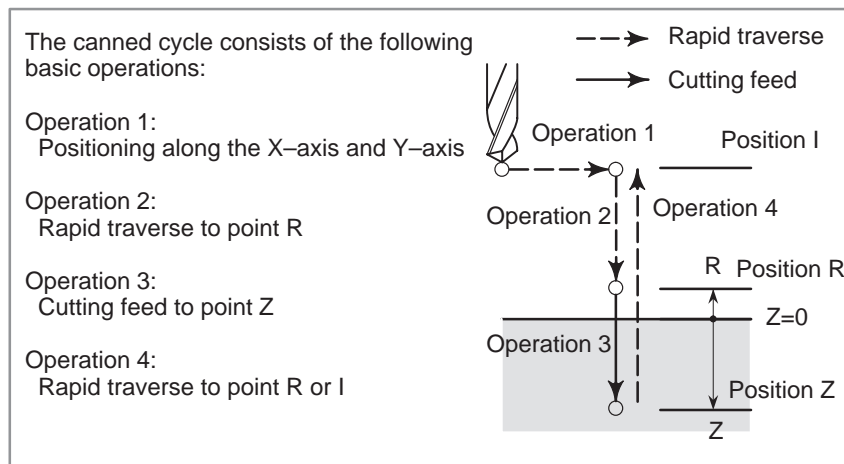
Modal calls can be nested by specifying another G66 code during a modal call.

Limitations

- In a G66 block, no macros can be called.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a miscellaneous 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.

Sample program

The same operation as the drilling canned cycle G81 is created using a custom macro and the machining program makes a modal macro call. For program simplicity, all drilling data is specified using absolute values.



• Calling format

```
G65 P9110 Xx Yy Zz Rr Ff Ll;
```

X: X coordinate of the hole (absolute specification only) (#24)
 Y: Y coordinate of the hole (absolute specification only) (#25)
 Z: Coordinates of position Z (absolute specification only) (#26)
 R: Coordinates of position R (absolute specification only) (#18)
 F: Cutting feedrate (#9)
 L: Repetition count

• Program that calls a macro program

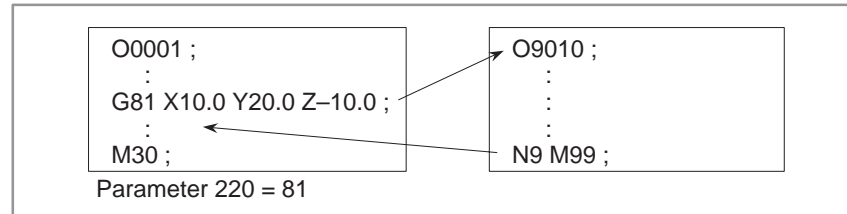
```
O0001;  
G28 G91 X0 Y0 Z0;  
G92 X0 Y0 Z50.0;  
G00 G90 X100.0 Y50.0;  
G66 P9110 Z-20.0 R5.0 F500;  
G90 X20.0 Y20.0;  
X50.0;  
Y50.0;  
X70.0 Y80.0;  
G67;  
M30;
```

• Macro program (program called)

```
O9110;  
#1=#4001; . . . . . Stores G00/G01.  
#3=#4003; . . . . . Stores G90/G91.  
#4=#4109; . . . . . Stores the cutting feedrate.  
#5=#5003; . . . . . Stores the Z coordinate at the start of drilling.  
G00 G90 Z#18; . . . . . Positioning at position R  
G01 Z#26 F#9; . . . . . Cutting feed to position Z  
IF[#4010 EQ 98]GOTO 1; . . . . . Return to position I  
G00 Z#18; . . . . . Positioning at position R  
GOTO 2;  
N1 G00 Z#5; . . . . . Positioning at position I  
N2 G#1 G#3 F#4; . . . . . Restores modal information.  
M99;
```

16.6.3 Macro Call Using 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).



Explanations

By setting a G code number from 1 to 255 used to call a custom macro program (9010 to 9019) in the corresponding parameter (220 to 229), the macro program can be called in the same way as with G65.

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**

Program number	Parameter number
O9010	220
O9011	221
O9012	222
O9013	223
O9014	224
O9015	225
O9016	226
O9017	227
O9018	228
O9019	229

- **Repetition**
- **Argument specification**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

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.

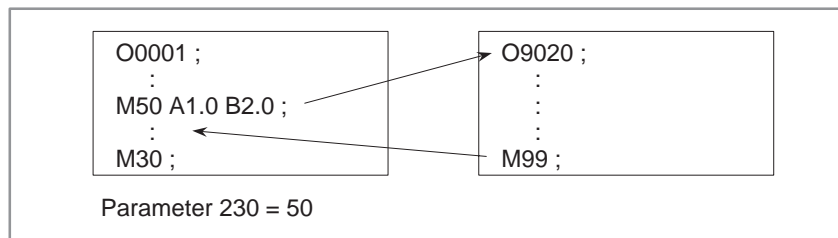
Limitations

- **Nesting of calls using G codes**

In a program called with a G code, no macros can be called using a G code. A G code in such a program is treated as an ordinary G code. In a program called as a subprogram with an M or T code, no macros can be called using a G code. A G code in such a program is also treated as an ordinary G code.

16.6.4 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).



Explanations

By setting an M code number from 1 to 255 used to call a custom macro program (9020 to 9029) in the corresponding parameter (230 to 239), the macro program can be called in the same way as with G65.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9020	230
O9021	231
O9022	232
O9023	233
O9024	234
O9025	235
O9026	236
O9027	237
O9028	238
O9029	239

- **Repetition**
- **Argument specification**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

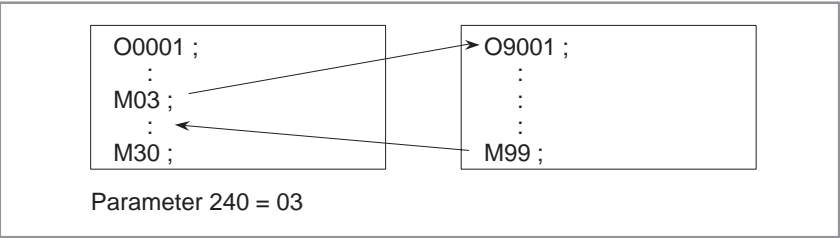
As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitations

- An M code used to call a macro program must be specified at the start of a block.
- In a macro called with a G code or in a program called as a subprogram with an M or T code, no macros can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

16.6.5 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).



Explanations

By setting an M code number from 1 to 255 used to call a subprogram in a parameter (240 to 242), the corresponding macro program (9001 to 9009) can be called in the same way as with M98.

- Correspondence between parameter numbers and program numbers

Program number	Parameter number
O9001	240
O9002	241
O9003	242

- Repetition
- Argument specification
- M code

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

Argument specification is not allowed.

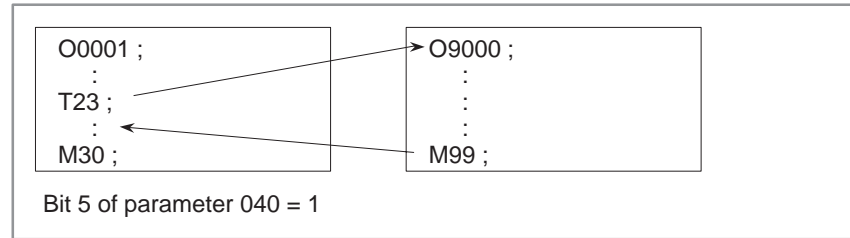
An M code in a macro program that has been called is treated as an ordinary M code.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

16.6.6 Subprogram Calls Using a T Code

By enabling subprograms (macro program) to be called with a T code in a parameter, a macro program can be called each time the T code is specified in the machining program.



Explanations

- Call

By setting bit 5 of parameter 040 to 1, the macro program O9000 can be called when a T code is specified in the machining program. A T code specified in a machining program is assigned to common variable #149.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using a T code. A T code in such a macro or program is treated as an ordinary T code.

16.6.7

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 240, and set 05 in parameter 241.

- **Variable value setting**

Set 0 in variables #501 to #505.

- **Program that calls a macro program**

```
O0001;  
T01 M06;  
M03;  
G04 X20.0;  
M05; ..... Changes #501.  
T02 M06;  
M03;  
G04 X20.0;  
M05; ..... Changes #502.  
T03 M06;  
M03;  
G04 X20.0;  
M05; ..... Changes #503.  
T04 M06;  
M03;  
G04 X20.0;  
M05; ..... Changes #504.  
T05 M06;  
M03;  
G04 X20.0;  
M05; ..... Changes #505.  
M30;
```

Macro program (program called)

O9001(M03); Macro to start counting
M01;
IF[#4120 EQ 0]GOTO 9; No tool specified
IF[#4120 GT 5]GOTO 9; Out-of-range tool number
#3002=0; Clears the timer.
N9 M03; Rotates the spindle in the forward direction.
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 the spindle.
M99;

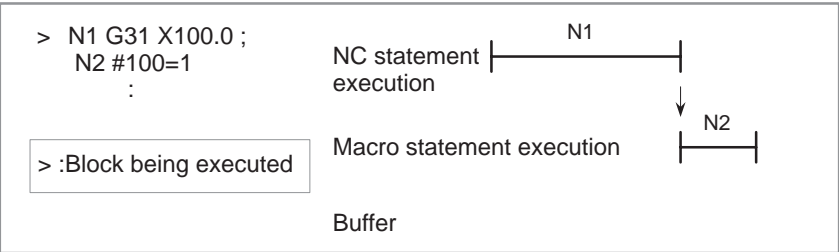
16.7

PROCESSING
MACRO
STATEMENTS

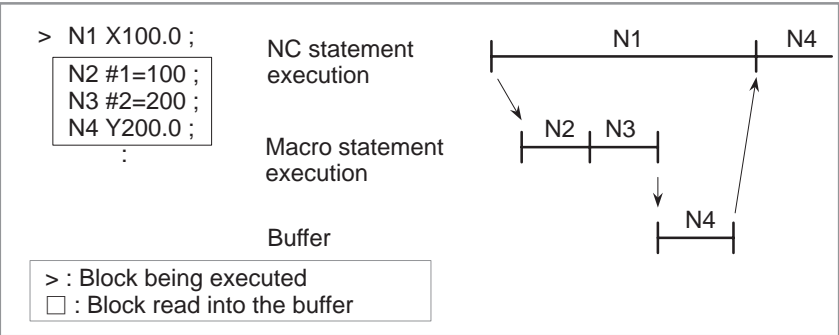
For smooth machining, the NC statement is pre-read to be performed next. This operation is referred to as buffering. In cutter compensation mode (G41, G42), the NC pre-reads NC statements two or three blocks ahead to find intersections. Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Blocks containing M00, M01, M02, or M30, blocks containing M codes for which buffering is suppressed by setting parameters 111 to 112, and blocks containing G31 are not pre-read.

Explanations

- When the next block is not buffered
(M codes that are not buffered, G31, etc.)

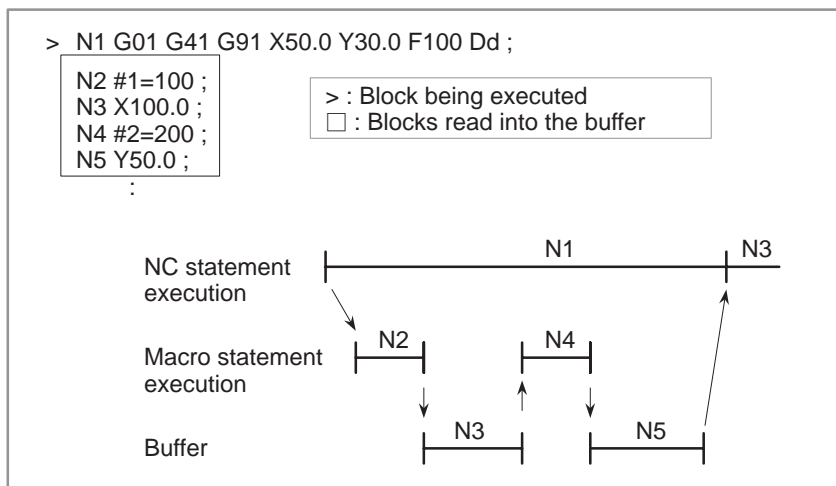


- Buffering the next block in other than cutter compensation mode (G41, G42) (normally pre-reading one block)



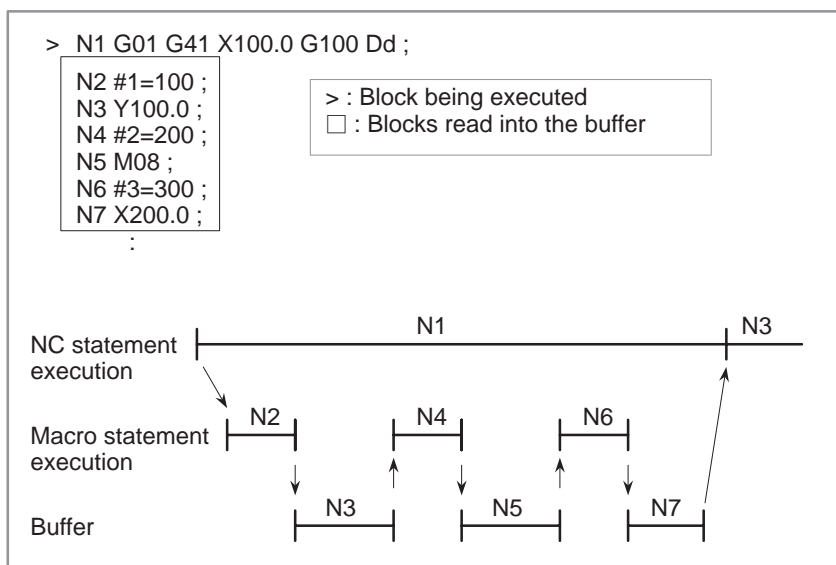
When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- **Buffering the next block in cutter compensation mode (G41, G42)**



When N1 is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. The macro statements (N2, N4) between N1 and N5 are processed during execution of N1.

- **When the next block involves no movement in cutter compensation C (G41, G42) mode**



When the NC1 block is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. Since N5 is a block that involves no movement, an intersection cannot be calculated. In this case, the NC statements in the next three blocks (up to N7) are read. The macro statements (N2, N4, and N6) between N1 and N7 are processed during execution of N1.

16.8 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.

16.9

LIMITATIONS



- **MDI operation**

Macro call can be specified in automatic operation. During automatic operation, however, it is impossible to switch to the MDI mode for a macro program call. Macro call can also be specified in MDI operation B.
- **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 (except blocks containing macro call commands, arithmetic operation commands, and control commands).
A block containing a macro call command (G65, G66, or G67) does not stop even when the single block mode is on. Blocks containing arithmetic operation commands and control commands can be stopped in single block mode by setting (bit 5 of parameter 011) to 1.
Single block stop operation is used for testing custom macro programs. Note that when a single block stop occurs at a macro statement in cutter compensation C 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**

Registered custom macro programs and subprograms should be protected from being destroyed by accident. By setting (bit 0 of parameter 389 #2) and (bit 4 of parameter 010) to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. When the entire memory is cleared (by pressing the  and  keys at the same time to turn on the power), the contents of memory such as custom macro programs are deleted.
- **Reset**

Local variables and common variables #100 to #199 are cleared to null values by a reset operation. They can be prevented from being cleared by setting, (bits 6 and 7 of parameter 040). System variables #1000 to #1133 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.
- **Display of the PROGRAM RESTART page**

As with M98, the M and T codes used for subprogram calls are not displayed.
- **Feed hold**

When a feed hold is enabled during execution of a macro statement, the machine stops after execution of the macro statement. The machine also stops when a reset or alarm occurs.
- **Constant values that can be used in <expression>**

+0.0000001 to +99999999
-99999999 to -0.0000001
The number of significant digits is 8 (decimal). If this range is exceeded, alarm No. 003 occurs.

16.10 EXTERNAL OUTPUT COMMANDS

In addition to the 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/punch interface.

Explanations

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**

POPEN

POPEN establishes a connection to an external input/output device. It must be specified before a sequence of data output commands. The NC outputs a DC2 control code.

• Data output command **BPRNT**

BPRNT [a #b [c] ...]

Number of significant decimal places

Variable

Character

The BPRNT command outputs characters and variable values in binary.

- (i) Specified characters are converted to corresponding codes according to the setting (ISO) that is output at that time.

Specifiable characters are as follows:

- **Letters (A to Z)**
- **Numbers**
- **Special characters (*, /, +, –, etc.)**

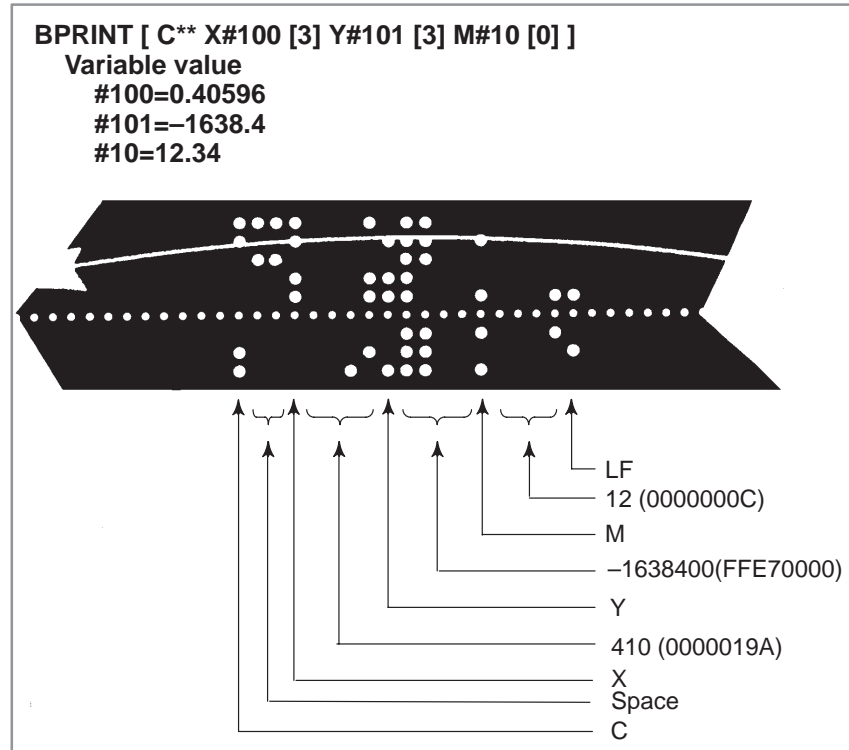
An asterisk (*) is output by a space code.

- (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.

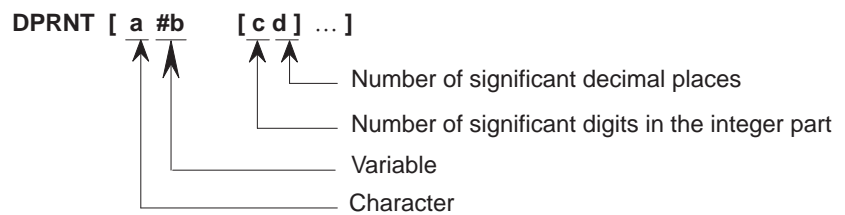
- (iii) When specified data has been output, an EOB code is output according to the ISO code settings on the parameter screen.

- (iv) Null variables are regarded as 0.

Examples



- **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 BPRINT 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.

One code is output for each of the specified number of digits, starting with the highest digit. For each digit, a code is output according to the settings (ISO). The decimal point is also output using a code set in the settings (ISO).

Each variable must be a numeric value consisting of up to eight digits. When high-order digits are zeros, these zeros are not output if (bit1 of parameter 040) is 1. If it 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 of parameter 040) is 0, a space code is output to indicate a positive number instead of +; if it is 1, no code is output.

Examples

DPRINT [X#2 [53] Y#5 [53] T#30 [20]]

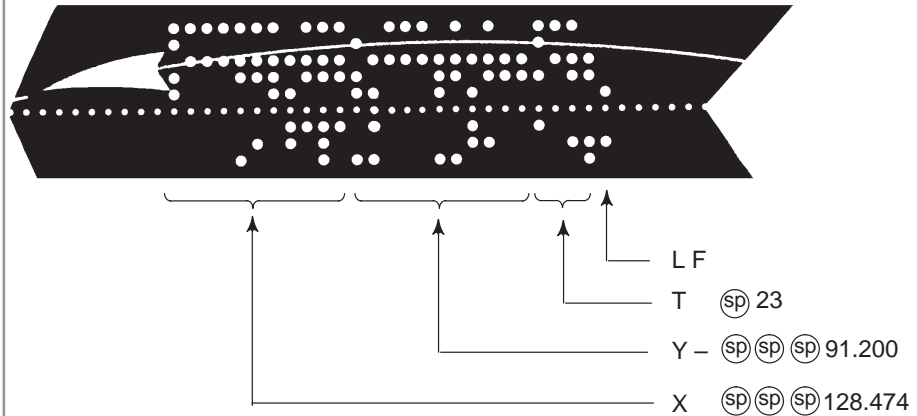
Variable value

#2=128.47398

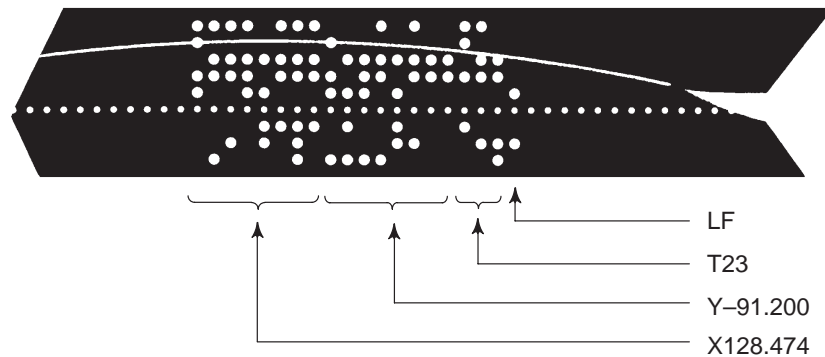
#5=-91.2

#30=123.456

① Parameter (No.040#1)=0



② Parameter (No.040#1)=1



- Close command PCLOS

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 channel use for parameter. According to the specification of this parameter, set data items (such as the baud rate) for the reader/punch interface. The remote buffer cannot be specified.

Specify that the reader/punch interface is used as the output device for punching. (Never specify output to the FANUC CASSETTE or floppy disks.)

To indicate the end of a line of data in ISO code, specify whether to use only an LF (bit 4 of parameter 057 is 0) or an LF and CR (bit 4 of parameter 057 is 1).

NOTE

- 1 It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.
- 2 Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.
- 3 When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. 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.

16.11 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

M96 P○○○○ ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Explanations

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.

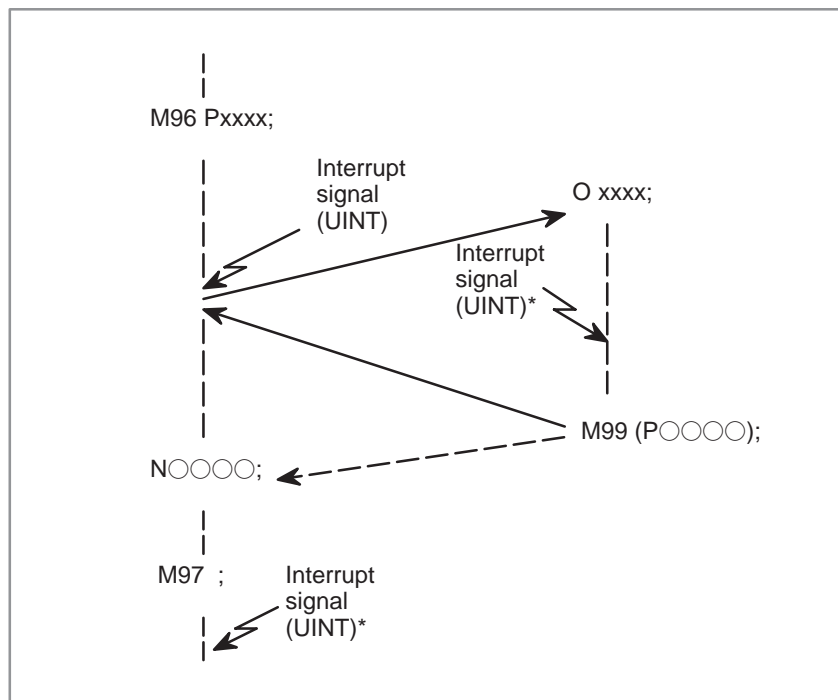


Fig.16.11 Interruption type sustom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx.

When the interrupt signal (UINT, marked by * in Fig. 16.11 is input during execution of the interrupt program or after M97 is specified, it is ignored.

16.11.1 Specification Method

Explanations

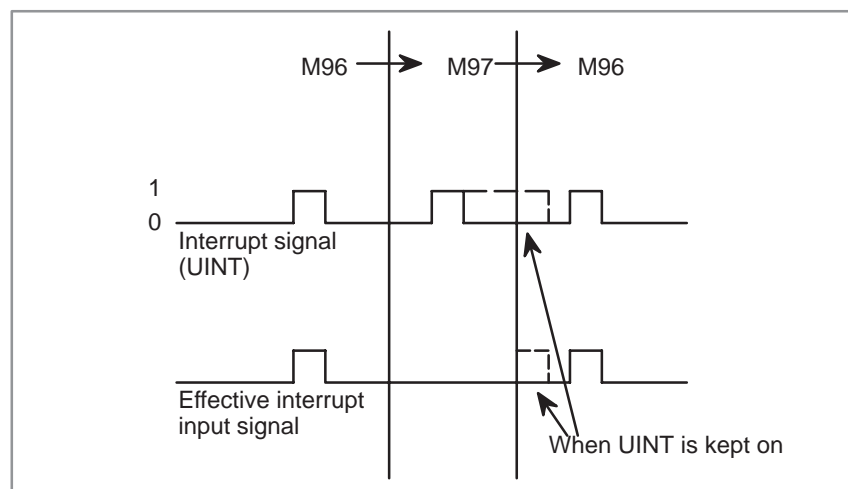
• Interrupt conditions

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

- **When memory operation or MDI operation B is selected**
- **When STL (start lamp) is on**
- **When a custom macro interrupt is not currently being processed**

• 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 NC is reset. After M97 is specified or the NC is reset, no custom macro interrupts are initiated even when the interrupt signal (UINT) is input. The interrupt signal (UINT) is ignored until another M96 command is specified.



The interrupt signal (UINT) becomes valid after M96 is specified. Even when the signal is input in M97 mode, it is ignored. When the signal input in M97 mode is kept on until M96 is specified, a custom macro interrupt is initiated as soon as M96 is specified (only when the status-triggered scheme is employed); when the edge-triggered scheme is employed, the custom macro interrupt is not initiated even when M96 is specified.

NOTE

For the status-triggered and edge-triggered schemes, see Item "Custom macro interrupt signal (UINT)" of Subsec. 16.11.2.

16.11.2

Details of Functions

Explanations

- **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 of parameter 056).

- (a) **Subprogram-type interrupt**

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**

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.

- **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 of parameter 056) 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 246 and 247 as follows:

Set the M code to enable custom macro interrupts in parameter 246, and set the M code to disable custom macro interrupts in parameter 247.

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 246 and 247.

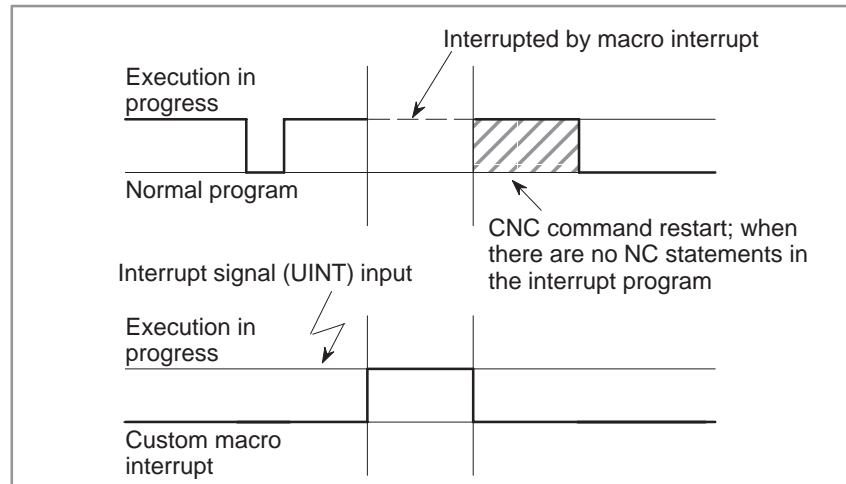
The M codes used for custom macro interrupt control are processed internally (they are not output to external units). However, in terms of program compatibility, it is undesirable to use M codes other than M96 and M97 to control custom macro interrupts.

- **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 of parameter 056) is used to select whether to perform interrupts even in the middle of a block or to wait until the end of the block.

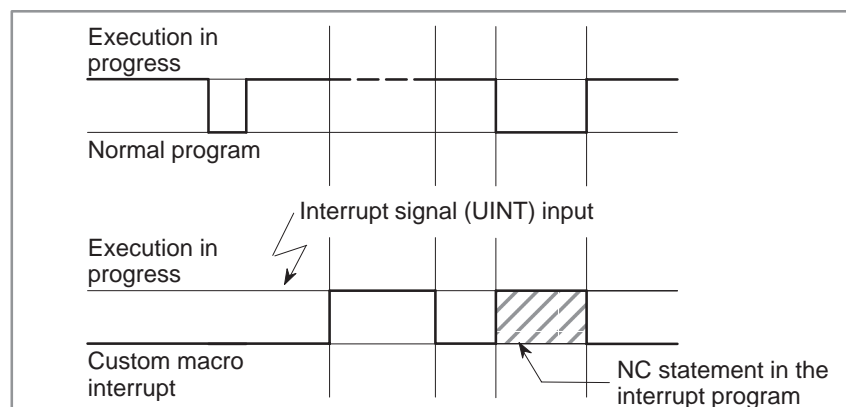
Type I
(when an interrupt is
performed even in the middle
of a block)

- (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.



Type II
(when an interrupt is
performed at the end of the
block)

- (i) If the block being executed is not a block that consists of several cycle operations such as a drilling canned cycle and automatic reference position return (G28), an interrupt is performed as follows:
 When an interrupt signal (UINT) is input, macro statements in the interrupt program are executed immediately unless an NC statement is encountered in the interrupt program. NC statements are not executed until the current block is completed.
- (ii) If the block being executed consists of several cycle operations, an interrupt is performed as follows:
 When the last movement in the cycle operations is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after all cycle operations are completed.



- **Conditions for enabling and disabling the custom macro interrupt signal**

The interrupt signal becomes valid after execution starts of a block that contains M96 for enabling custom macro interrupts. The signal becomes invalid when execution starts of a block that contains M97.

While an interrupt program is being executed, the interrupt signal becomes invalid. The signal become valid when the execution of the block that immediately follows the interrupted block in the main program is started after control returns from the interrupt program. In type I, if the interrupt program consists of only macro statements, the interrupt signal becomes valid when execution of the interrupted block is started after control returns from the interrupt program.

- **Custom macro interrupt during execution of a block that involves cycle operation**

For type I

Even when cycle operation is in progress, movement is interrupted, and the interrupt program is executed. If the interrupt program contains no NC statements, the cycle operation is restarted after control is returned to the interrupted program. If there are NC statements, the remaining operations in the interrupted cycle are discarded, and the next block is executed.

For type II

When the last movement of the cycle operation is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after cycle operation is completed.

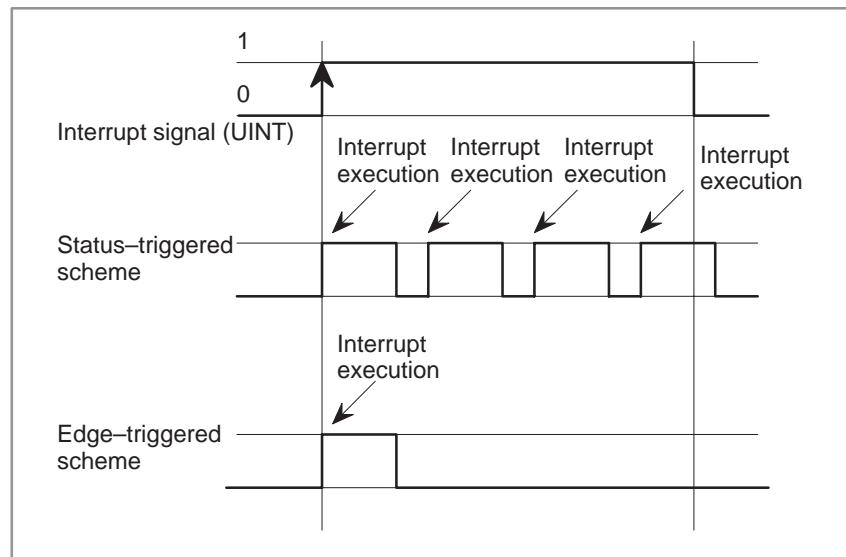
- **Custom macro interrupt signal (UINT)**

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 of parameter 056). 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.



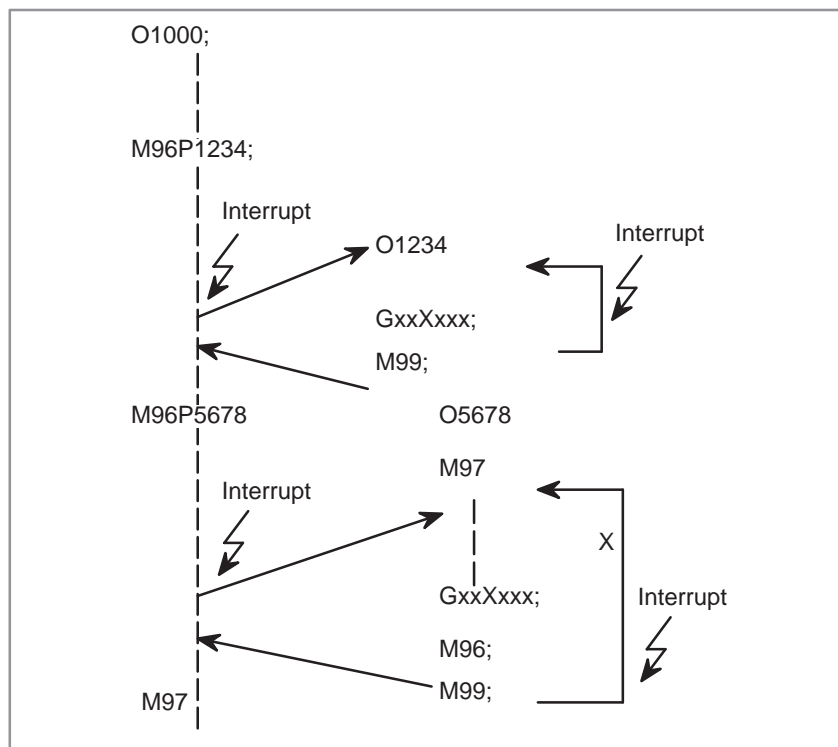
In the above example, an interrupt is executed four times when the status triggered scheme is used; when the edge-triggered scheme is used, the interrupt is executed just once.

- **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.

When a custom macro interrupt program is being executed, no interrupts are generated. To enable another interrupt, execute M99. When M99 is specified alone, it is executed before the preceding commands terminate. Therefore, a custom macro interrupt is enabled for the last command of the interrupt program. If this is inconvenient, custom macro interrupts should be controlled by specifying M96 and M97 in the program.

When a custom macro interrupt is being executed, no other custom macro interrupts are generated; when an interrupt is generated, additional interrupts are inhibited automatically. Executing M99 makes it possible for another custom macro interrupt to occur. M99 specified alone in a block is executed before the previous block terminates. In the following example, an interrupt is enabled for the Gxx block of O1234. When the signal is input, O1234 is executed again. O5678 is controlled by M96 and M97. In this case, an interrupt is not enabled for O5678 (enabled after control is returned to O1000).



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 G○○ is executed before M99 is recognized.)

- ① G○○ X○○○ ;
M99 ;
② G○○ X○○○ M99 ;

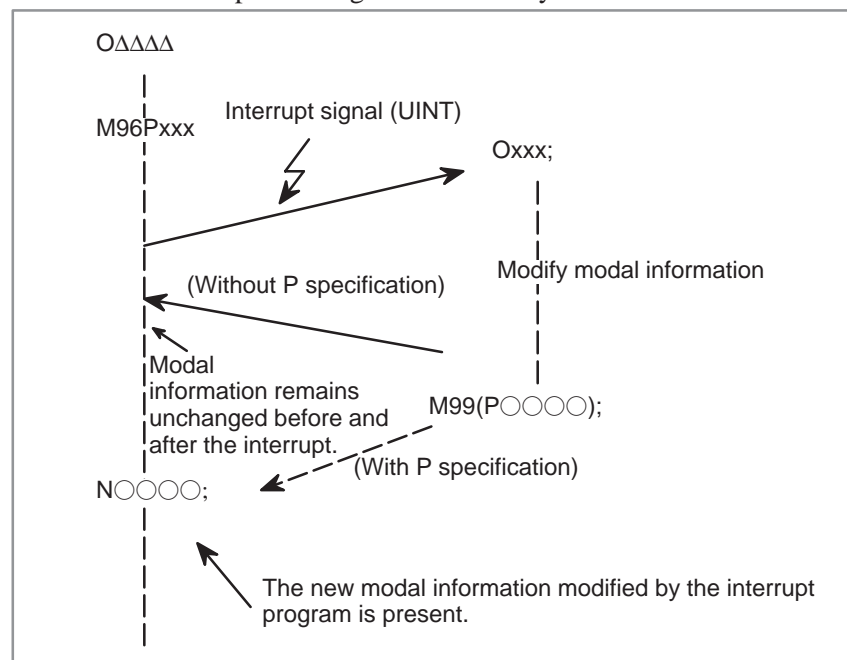
- **Custom macro interrupt and modal information**

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 Pxxxx, 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. Restoration of the old modal information present before the interrupt is not desirable. This is because after control is returned, some programs may operate differently depending on the modal information present before the interrupt. In this case, the following measures are applicable:

- 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.



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.

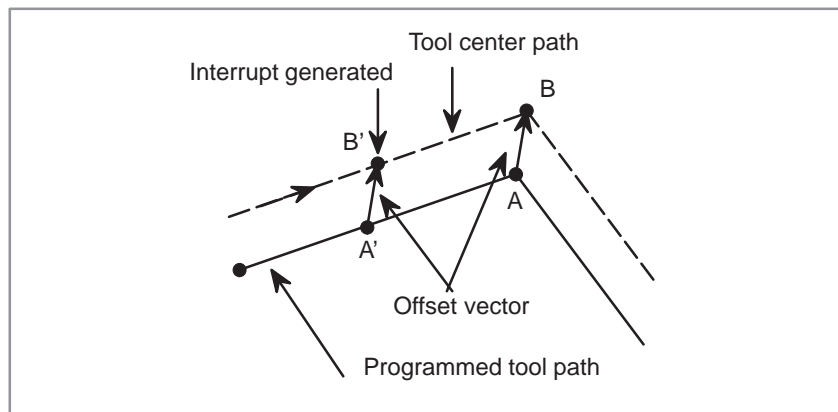
Modal information when control is returned by M99 P○○○○○

The new modal information modified by the interrupt program remains valid even after control is returned. The old modal information which was valid in the interrupted block can be read using custom macro system variables #4001 to #4120.

Note that when modal information is modified by the interrupt program, system variables #4001 to #4120 are not changed.

• System variables (position information values) for the interrupt program

- The coordinates of point A can be read using system variables #5001 and up until the first NC statement is encountered.
- The coordinates of point A' can be read after an NC statement with no move specifications appears.
- The machine coordinates and workpiece coordinates of point B' can be read using system variables #5021 and up and #5041 and up.



• 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 M99Pxxxx;, the modal call in the interrupt program remains valid.

• Custom macro interrupt and program restart

When the interrupt signal (UINT) is input while a return operation is being performed in the dry run mode after the search operation for program restart, the interrupt program is called after restart operation terminates for all axes. This means that interrupt type II is used regardless of the parameter setting.

17

PATTERN DATA INPUT FUNCTION

This function enables users to perform programming simply by extracting numeric data (pattern data) from a drawing and specifying the numerical values from the CRT/MDI panel.

This eliminates the need for programming using an existing NC language.

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


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

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

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

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

17.1 DISPLAYING THE PATTERN MENU

Pressing the  key and the soft key **[MENU]** is displayed on the following pattern menu screen.

MENU : HOLE PATTERN O0000 N00000

1. BOLT HOLE
2. GRID
3. LINE ANGLE
4. TAPPING
5. DRILLING
6. BORING
7. POCKET
8. PECK
9. TEST PATRN
10. BACK

SELECT = S O T

10:01:29 MDI

[OFFSET] [MACRO] [**MENU**] [] []

HOLE PATTERN : This is the menu title. An arbitrary character string consisting of up to 12 characters can be specified.

BOLT HOLE : This is the pattern name. An arbitrary character string consisting of up to 10 characters can be specified, including katakana.

The machine tool builder should specify the character strings for the menu title and pattern name using the custom macro, and load the character strings into program memory as a subprogram of program No. 9500.

- **Macro commands specifying the menu title**

Menu title : $C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 C_9 C_{10} C_{11} C_{12}$

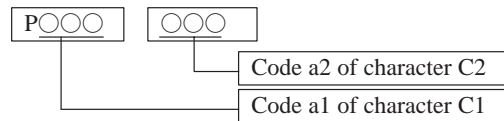
C_1, C_2, \dots, C_{12} : Characters in the menu title (12 characters)

Macro instruction

G65 H90 P_p Q_q R_r I_i J_j K_k :

H90: Specifies the menu title

p : Assume a₁ and a₂ to be the codes of characters C₁ and C₂. Then,



q : Assume a₃ and a₄ to be the codes of characters C₃ and C₄. Then,
 $q = a_3 10^3 + a_4$

r : Assume a₅ and a₆to be the codes of characters C₅ and C₆. Then,
 $r = a_5 10^3 + a_6$

i : Assume a₇ and a₈to be the codes of characters C₇ and C₈. Then,
 $i = a_7 10^3 + a_8$

j : Assume a₉ and a₁₀to be the codes of characters C₉ and C₁₀. Then,
 $j = a_9 10^3 + a_{10}$

k : Assume a₁₁ and a₁₂to be the codes of characters C₁₁ and C₁₂. Then,
 $k = a_{11} 10^3 + a_{12}$

Example) If the title of the menu is "HOLE PATTERN" then the macro instruction is as follows:

G65 H90 P072079 Q076069 R032080
 HO LE □ P
I065084 J084069 K082078;
 AT TE RN

For codes corresponding to these characters, refer to the table in 17.3.

- **Macro instruction describing the pattern name**

Pattern name: $C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 C_9 C_{10}$

C_1, C_2, \dots, C_{10} : Characters in the pattern name (10 characters)

Macro instruction

G65 H91 $P_n Q_q R_r I_i J_j K_k$;

H91: Specifies the menu title

n : Specifies the menu No. of the pattern name

$n=1$ to 10

q : Assume a_1 and a_2 to be the codes of characters C_1 and C_2 . Then,
 $q=a_1 \times 10^3 + a_2$

r : Assume a_3 and a_4 to be the codes of characters C_3 and C_4 . Then,
 $r=a_3 \times 10^3 + a_4$

i : Assume a_5 and a_6 to be the codes of characters C_5 and C_6 . Then,
 $i=a_5 \times 10^3 + a_6$

j : Assume a_7 and a_8 to be the codes of characters C_7 and C_8 . Then,
 $j=a_7 \times 10^3 + a_8$

k : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10} . Then,
 $k=a_9 \times 10^3 + a_{10}$

Example) If the pattern name of menu No. 1 is "BOLT HOLE" then the macro instruction is as follows.

G65 H91 P1 Q066079 R076084 I032072 J079076 K069032 ;
 BO LT □H OL E □

- **Pattern No. selection**

To select a pattern from the pattern menu screen, enter the corresponding pattern No. The following is an example.

The selected pattern No. is assigned to system variable #5900. The custom macro of the selected pattern can be started by starting a fixed program (external program No. search) with an external signal then referring to the system variable #5900 in the program.

NOTE

If each characters of P, Q, R, I, J, and K are not specified in a macro instruction, two spaces are assigned to each omitted character.

Example

Custom macros for the menu title and hole pattern names.

```
MENU : HOLE PATTERN                                O0000 N00000
  1.  BOLT HOLE
  2.  GRID
  3.  LINE ANGLE
  4.  TAPPING
  5.  DRILLING
  6.  BORING
  7.  POCKET
  8.  PECK
  9.  TEST PATRN
 10.  BACK

SELECT =                                           S   O   T
10:01:29                                           MDI
[ OFFSET ] [ MACRO ] [ MENU ] [   ] [   ]
```

```
O9500 ;
N1G65 H90 P072 079 Q076 069 R032 080 I 065 084 J 084 069 K082 078;    HOLE PATTERN
N2G65 H91 P1 Q066 079 R076 084 I 032 072 J 079 076 K069 032 ;        1.BOLT HOLE
N3G65 H91 P2 Q071 082 R073 068 ;                                       2.GRID
N4G65 H91 P3 Q076 073 R078 069 I 032 065 J 078071 K076069           3.LINE ANGLE
N5G65 H91 P4 Q084 065 R080 080 I 073 078 J 071 032 ;                 4.TAPPING
N6G65 H91 P5 Q068 082 R073 076 I 076 073 J 078 071 ;                 5.DRILLING
N7G65 H91 P6 Q066079 R082073 I 078 071 ;                               6.BORING
N8G65 H91 P7 Q080 079 R067 075 I 069 084 ;                             7.POCKET
N9G65 H91 P8 Q080069 R067075 ;                                          8.PECK
N10G65 H91 P9 Q084 069 R083 084 I032 080 J065 084 K082 078 ;        9.TEST PATRN
N11G65 H91 P10 Q066 065 R067 0750 ;                                     10.BACK
N12M99 ;
```

17.2

PATTERN DATA DISPLAY

When a pattern menu is selected, the necessary pattern data is displayed.

VAR. : BOLT HOLE			O9501 N0014
NO.	NAME	DATA	COMMENT
-- 500	TOOL	0	
501	KIJUN X	0	*BOLT HOLE
502	KIJUN Y	0	CIRCLE*
503	RADIUS	0	SET PATTERN
504	S. ANGL	0	DATA TO VAR.
505	HOLES NO	0	NO.500-505.
506		0	
507		0	
ACTUAL POSITION (RELATIVE)			
	X	0.000	Y 0.000
	Z	0.000	
NO. 500 =		S	0 T
10:00:48			
[OFFSET] [MACRO] [MENU] [] []			

BOLT HOLE : This is the pattern data title. A character string consisting of up to 12 characters can be set.

TOOL : This is the variable name. A character string consisting of up to 10 characters can be set.

***BOLT HOLE**

CIRCLE* : This is a comment statement. A character string can be displayed consisting of up to 8 lines, 12 characters per line.

(It is permissible to use katakana in a character string or line.)

The machine tool builder should program the character strings of pattern data title, pattern name, and variable name using the custom macro, and load them into the program memory as a subprogram whose No. is 9500 plus the pattern No. (O9501 to O9510).

- **Macro instruction specifying the pattern data title (the menu title)**

Menu title : $C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 C_9 C_{10} C_{11} C_{12}$

C_1, C_2, \dots, C_{12} : Characters in the menu title (12 characters)

Macro instruction

G65 H92 $P_n Q_q R_r I_i J_j K_k$;

H92 : Specifies the pattern name

p : Assume a_1 and a_2 to be the codes of characters C_1 and C_2 . Then,

$$p = a_1 \times 10^3 + a_2$$

See 17.3 for character codes.

q : Assume a_3 and a_4 to be the codes of characters C_3 and C_4 . Then,

$$q = a_3 \times 10^3 + a_4$$

r : Assume a_5 and a_6 to be the codes of characters C_5 and C_6 . Then,

$$r = a_5 \times 10^3 + a_6$$

i : Assume a_7 and a_8 to be the codes of characters C_7 and C_8 . Then,

$$i = a_7 \times 10^3 + a_8$$

j : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10} . Then,

$$j = a_9 \times 10^3 + a_{10}$$

k : Assume a_{11} and a_{12} to be the codes of characters C_{11} and C_{12} . Then,

$$k = a_{11} \times 10^3 + a_{12}$$

Example) Assume that the pattern data title is "BOLT HOLE." The macro instruction is given as follows:

G65 H92 P066079 Q076084 R032072 I079076 J069032 ;

BO LT □ H OL E

- **Macro instruction specifying the variable name**

Variable name : $C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 C_9 C_{10}$

C_1, C_2, \dots, C_{10} : Characters in the variable name (10 characters)

Macro instruction

G65 H93 $P_n Q_q R_r I_i J_j K_k$;

H93 : Specifies the variable name

n : Specifies the menu No. of the variable name

n=1 to 10

q : Assume a_1 and a_2 to be the codes of characters C_1 and C_2 . Then,

$$q = a_1 \times 10^3 + a_2$$

r : Assume a_3 and a_4 to be the codes of characters C_3 and C_4 . Then,

$$r = a_3 \times 10^3 + a_4$$

i : Assume a_5 and a_6 to be the codes of characters C_5 and C_6 . Then,

$$i = a_5 \times 10^3 + a_6$$

j : Assume a_7 and a_8 to be the codes of characters C_7 and C_8 . Then,

$$j = a_7 \times 10^3 + a_8$$

k : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10} . Then,

$$k = a_9 \times 10^3 + a_{10}$$

Example) Assume that the variable name of the variable No. 503 is "RADIUS." The macro instruction is given as follows:

G65 H93 P503 Q082065 R068073 I085083 ;

RA DI US

NOTE

Variable names can be assigned to 32 common variables #500 to #531, which are not cleared when the power is turned off.

- **Macro instruction to describe a comment**

One comment line: $C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 C_9 C_{10} C_{11} C_{12}$

C_1, C_2, \dots, C_{12} : Character string in one comment line (12 characters)

Macro instruction

G65 H94 P_n Q_q R_r I_i J_j K_k ;

H94 : Specifies the comment

p : Assume a_1 and a_2 to be the codes of characters C_1 and C_2 . Then,

$$p = a_1 \times 10^3 + a_2$$

See 17.7 for character codes.

q : Assume a_3 and a_4 to be the codes of characters C_3 and C_4 . Then,

$$q = a_3 \times 10^3 + a_4$$

r : Assume a_5 and a_6 to be the codes of characters C_5 and C_6 . Then,

$$r = a_5 \times 10^3 + a_6$$

i : Assume a_7 and a_8 to be the codes of characters C_7 and C_8 . Then,

$$i = a_7 \times 10^3 + a_8$$

j : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10} . Then,

$$j = a_9 \times 10^3 + a_{10}$$

k : Assume a_{11} and a_{12} to be the codes of characters C_{11} and C_{12} . Then,

$$k = a_{11} \times 10^3 + a_{12}$$

A comment can be displayed in up to eight lines. The comment consists of the first line to the eighth line in the programmed sequence of G65 H94 for each line.

Example) Assume that the comment is "BOLT HOLE." The macro instruction is given as follows:

G65 H94 P042066 Q079076 R084032 I072079 J076069;
 *B OL T□ HO LE

Examples

Macro instruction to describe a parameter title , the variable name, and a comment.

VAR. : BOLT HOLE			O9501 N0014
NO.	NAME	DATA	COMMENT
-- 500	TOOL	0	
501	KIJUN X	0	*BOLT HOLE
502	KIJUN Y	0	CIRCLE*
503	RADIUS	0	SET PATTERN
504	S. ANGL	0	DATA TO VAR.
505	HOLES NO	0	NO.500-505.
506		0	
507		0	
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000		
NO. 500 =		S	0 T
10:00:48			
[OFFSET] [MACRO] [MENU] [] []			

O9501 ;	
N1G65 H92 P066 079 Q076 084 R032 072 I 079 076 J069 032;	VAR:BOLT HOLE
N2G65 H93 P500 Q084 079 R079076;	#500 TOOL
N3G65 H93 P501 Q075 073 R074 085 I078 032 J088 032 ;	#501 KIJUN X
N4G65 H93 P502 Q075 073 R074 085 I 078 032 J089 032 ;	#502 KIJUN Y
N5G65 H93 P503 Q082 065 R068 073 I 085 083 ;	#503 RADIUS
N6G65 H93 P504 Q083 046 R032 065 I 078 071 J 076 032;	#504 S.ANGL
N7G65 H93 P505 Q072 079 R076 069 I 083 032 J078 079 K046 032 ;	#505 HOLES NO
N8G65 H94;	Comment
N9G65 H94 P042 066 Q079 076 R084 032 I072 079 J076 069 ;	*BOLT HOLE
N10G65 H94 R032 067 I073 082 J067 076 K069 042 ;	CIRCLE*
N11G65 H94 P083 069 Q084 032 080 065 I084 084 J069 082 K078 032 ;	SET PATTERN
N12G65 H94 P068 065 Q084 065 R032 084 I079 032 J086 065 K082046 ;	DATA NO VAR.
N13G65 H94 P078 079 Q046 053 R048 048 I045 053 J048 053 K046 032 ;	No.500-505
N14M99 ;	

17.3

CHARACTERS AND CODES TO BE USED FOR THE PATTERN DATA INPUT FUNCTION

Table. 17.3(a) Characters and codes to be used for the pattern data input function

Char-acter	Code	Comment	Char-acter	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		"	034	Quotation mark
H	072		#	035	Hash sign
I	073		\$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		'	039	Apostrophe
M	077		(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		^	092	
3	051		¥	093	Yen sign
4	052]	094	Right squar bracket
5	053		_	095	Underscore

Table 17.3 (b)Numbers of subprograms employed in the pattern data input function

Subprogram No.	Function
O9500	Specifies character strings displayed on the pattern data menu.
O9501	Specifies a character string of the pattern data corresponding to pattern No.1
O9502	Specifies a character string of the pattern data corresponding to pattern No.2
O9503	Specifies a character string of the pattern data corresponding to pattern No.3
O9504	Specifies a character string of the pattern data corresponding to pattern No.4
O9505	Specifies a character string of the pattern data corresponding to pattern No.5
O9506	Specifies a character string of the pattern data corresponding to pattern No.6
O9507	Specifies a character string of the pattern data corresponding to pattern No.7
O9508	Specifies a character string of the pattern data corresponding to pattern No.8
O9509	Specifies a character string of the pattern data corresponding to pattern No.9
O9510	Specifies a character string of the pattern data corresponding to pattern No.10

Table. 17.3 (c)Macro instructions used in the pattern data input function

G code	H code	Function
G65	H90	Specifies the menu title.
G65	H91	Specifies the pattern name.
G65	H92	Specifies the pattern data title.
G65	G93	Specifies the variable name.
G65	H94	Specifies the comment.

Table. 17.3 (d)System variables employed in the pattern data input function

System variable	Function
#5900	Pattern No. selected by user.

18

PROGRAMMABLE PARAMETER ENTRY (G10)

General

The values of parameters can be entered in a lprogram. 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

Format	
G10L50; Parameter entry mode setting N_P_; G11; Parameter entry mode cancel	
Meaning of command	
N_ : Parameter No. (4digids) or data No. for pitch errors compensation P_ : Parameter setting value (Leading zeros can be omitted.)	

Explanations

- **Parameter setting value (P_)**

Can not use a decimal point in a value set in a parameter (P_).
a decimal point cannot be used in a custom macro variable for P_ either.

WARNING

- 1 Do not fail to perform reference point return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
- 2 The canned-cycle mode must be cancelled before entering of parameters.

NOTE

Other NC statements cannot be specified while in parameter input mode.

Examples

To change the length of the effective area (in-position) along the X-axis

G10L50 ; Parameter entry mode
N500 P50 ;
G11 ; cancel parameter entry mode

19

MEMORY OPERATION USING Series 10/11 TAPE FORMAT

General

Memory operation of the program registered by Series 10/11 tape format is possible with setting of the setting parameter (TAPEF).

Explanations

Data formats for cutter compensation, subprogram calling, and canned cycles are different between the Series 0 and Series 10/11. The Series 10/11 data formats can be processed for memory operation. Other data formats must comply with the Series 0. When a value out of the specified range for the Series 0 is registered, an alarm occurs. Functions not available in the Series 0 cannot be registered or used for memory operation.

- **Address for the cutter compensation offset number**

Offset numbers are specified by address D in the Series 10/11. When an offset number is specified by address D, the modal value specified by address H is replaced with the offset number specified by address D.

- **Subprogram call**

If a subprogram number of more than four digits is specified, the four low-order digits are regarded as the subprogram number. If no repeat count is specified, 1 is assumed.

Table 19 (a) Subprogram call data format

CNC	Data format
Series 10/11	M98 P ○○○○○○ L ○○○○ ; P : Subprogram number L : Repetition count
Series 0	M98 P ○○○ □□□□ ; Repetition count Subprogram number

- **Address for the canned cycle repetition count**

The Series 10/11 and Series 0 use different addresses for the repeat count for canned cycles as listed in Table 19 (b).

Table 19 (b) Address for times of repetition of canned cycle

CNC	Address
Series 10/11	L
Series 0	K

20 HIGH SPEED CYCLE CUTTING

General

This function can convert the machining profile to a data group that can be distributed as pulses at high-speed by the macro compiler or macro executor. The function can also call and execute the data group as a machining cycle using the CNC command (G05 command).

Format

G05 P10000 L0000 ;

P10000 is number of the machining cycle to be called first:
P10001 to P10999

L0000 is repetition count of the machining cycle
(L1 applies when this parameter is omitted.) :
L1 to L999

Call and execute the data for the high speed cutting cycle specified by the macro compiler and macro executor using the above command.

Cycle data can be prepared for up to 999 cycles. Select the machining cycle by address P. More than one cycle can be called and executed in series using the cycle connection data in the header.

Specify the repetition count of the called machining cycle by address L. The repetition count in the header can be specified for each cycle.

The connection of cycles and their repetition count are explained below with an example.

Example) Assume the following:

Cycle 1 Cycle connection data 2 Repetition count 1

Cycle 2 Cycle connection data 3 Repetition count 3

Cycle 3 Cycle connection data 0 Repetition count 1

G05 P10001 L2 ;

The following cycles are executed in sequence:

Cycles 1, 2, 2, 2, 3, 1, 2, 2, 2, and 3

20.1 NUMBER OF CONTROL AXES

Four axes maximum (Four axes can be controlled simultaneously.).

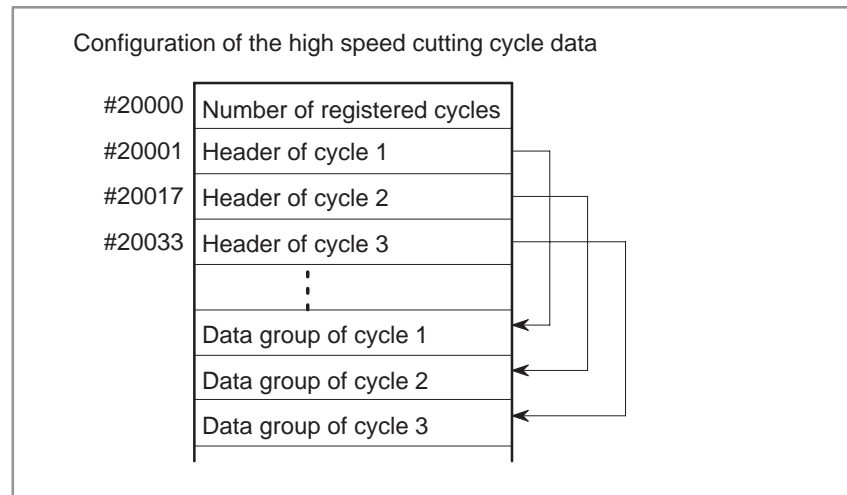
20.2 PULSE DISTRIBUTION

Set the number of pulses per cycle in parameter 055 #4 to #6 as a macro variable (#20000 to #85535) for high speed cycle cutting using the macro compiler and macro executor.

The unit for the number of pulses is the least input increment.

20.3 CONFIGURATION OF HIGH SPEED CYCLE CUTTING DATA

Data for the high speed cycle cutting is assigned to variables (#20000 to #85535) for the high speed cycle cutting by the macro compiler and macro executor.



20.3.1 Number of Registered Cycles

Specify the number of cycles (number of headers) of high-speed machining cycle data. Values from 1 to 999 can be specified.

20.3.2 Header

The header for high-speed cycle machining data has the following configuration:

Header configuration	
#20001/20017/20033..	Cycle repetition count
#20002/20018/20034..	Cycle connection data
#20003/20019/20035..	Number of data items
#20004/20020/20036..	Data type
#20005/20021/20037..	Variable assigned to the 1st axis data
#20006/20022/20038..	Variable assigned to the 2nd axis data
#20007/20023/20039..	Variable assigned to the 3rd axis data
#20008/20024/20040..	Variable assigned to the 4th axis data
#20009/20025/20041..	
#20010/20026/20042..	
#20011/20027/20043..	Total number of fixed data items for the 1st axis
#20012/20028/20044..	Total number of fixed data items for the 2nd axis
#20013/20029/20045..	Total number of fixed data items for the 3rd axis
#20014/20030/20046..	Total number of fixed data items for the 4th axis
#20015/20031/20047..	
#20016/20032/20048..	

Explanations

- **Cycle repetition count**

Specify the repetition count for this cycle. Values from 0 to 32767 can be specified. When 0 or 1 is specified, the cycle is executed once.

- **Cycle connection data**

Specify the number (1 to 999) of the cycle to be executed after this cycle. When no connection cycle exists because of the last cycle, specify 0.

- **Number of data items**

Specify the number of data items per cycle. Valid values are from 1 to 32767.

When a fixed data item is specified, the fixed data is repeated for the specified number of times in one cycle.

- **Data type**

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
-	-	-	-	r4	r3	r2	r1	-	-	-	-	t4	t3	t2	t1

The bits from t1 to t4, corresponding to the 1st to 4th axes, have the following meanings:

0: Distribution data is always constant.

1: Distribution data is variable or fixed.

When the distribution data is variable or fixed, the bits from r1 to r4, corresponding to the 1st to 4th axes, have the following meanings:

0: Distribution data is read forward.

1: Distribution data is read backwards.

Because the data consists of bits, it is necessary to use a binary-coded decimal value when setting it using the macro compiler and macro executor.

Example)

When constant data is assigned to the 1st and 2nd axes and variable data is assigned to the 3rd and 4th axes, #20004 = 12; (t4 and t3: 0, t2 and t1: 1)

- **Variables assigned to data for the 1st to 4th axes**

- **Constant data**

When the corresponding data type bit (t4 to t1) is 0, specify “distribution data value”.

- **Variable data**

When the corresponding data type bit (t4 to t1) is 1 and the total number of fixed data items = 0, specify “(Storing start data variable No. of the distribution data)/10”.

- **Fixed data**

When the corresponding data type bit (t4 to t1) is 1 and the total number of fixed data items is other than 0, specify “(Storing start data variable No. of the distribution data)/10”.

The applicable value for the variable data and fixed data is 2001 to 8553. It is not possible to start storing data in the executable format from a variable No. that is not a multiple of 10.

To read the distribution data backwards, set the variable No. of the data to be distributed last. For example, to read the distribution data in #25000 to #25999 backwards, set 25000 as the data assignment variable.

- **Total number of fixed data items for the first to 4th axes**

Set the length of the fixed data for the cycle.

The first address of the fixed data must be specified by the data assignment variable. When the total number of fixed data items = 0 and the corresponding data type bit (t4 to t1) is 1, the data is regarded as a variable data.

NOTE

- 1 When the high-speed machining function is used, an extended RAM is necessary. The length of tape that can be specified is limited to 80 meters.
- 2 An alarm is issued if the function is executed in the G41/G42 mode.
- 3 Single block stop, dry run/feedrate override, automatic acceleration/deceleration and handle interruption are disabled during high-speed cycle machining.
- 4 When the distribution cycle is 2 ms or 1 ms set the total number of distribution data items for one cycle to a multiple of the following values. This does not apply when the distribution cycle is 16 ms 8 ms or 4 ms.

If the total number is not a multiple of one of the following values, movement in the remaining cycle becomes zero.

Example)

When all 41 data items (distribution cycle: 1 ms) are specified, movement is zero in the remaining 3 ms.

Distribution cycle 2 ms: Multiple of 2

Distribution cycle 1 ms: Multiple of 4

Alarms

Alarm number	Descriptions
115	<p>The contents of the header are invalid. This alarm is issued in the following cases.</p> <ol style="list-style-type: none"> 1. The header corresponding to the number of the specified call machining cycle was not found. 2. A cycle connection data value is not in the valid range (0 to 999). 3. The number of data items in the header is not in the valid range (1 to 32767). 4. The first variable No. for storing data in the executable format is not in the valid range (#20000 to #85535). 5. The last variable No. for storing data in the executable format exceeds the limit (#85535). 6. The first variable No. for start data in the executable format overlaps with a variable No. used in the header.
178	High-speed cycle machining was specified in the G41/G42 mode.
179	The number of control axes specified in parameter 597 exceeds the maximum number.

21

SIMPLE SYNCHRONOUS CONTROL

General

It is possible to change the operating mode for two or more specified axes to either synchronous operation or normal operation by an input signal from the machine.

For example, The following operating modes are applicable to machines having two tables driven independently by separate control axes. The following example is of a machine with two tables driven independently by the Y axis and V axis. If the axis names are different, substitute a suitable name for the ones below.

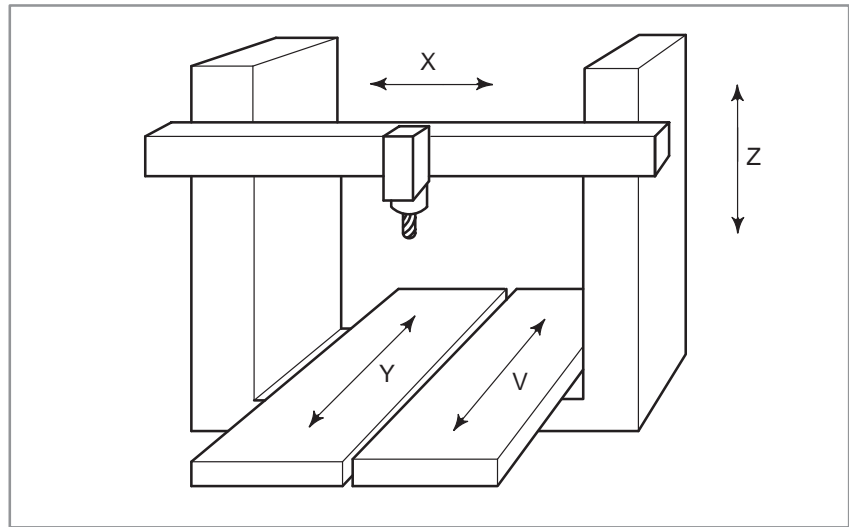


Fig. 21 Example of axis configuration of the machine operated by simple synchronous control

Explanations

- **Synchronous operation**

This mode is used for machining large workpieces that extend over two tables.

While operating one axis with a move command, it is possible to synchronously move the other axis. In the synchronous mode, the axis to which the move command applies is called the master axis, and the axis that moves synchronously with the master axis is called the slave axis. In this example, it is assumed that Y axis is the master axis and V axis is the slave axis. Here, the Y axis and the V axis move synchronously in accordance with program command Yyyyy issued to the Y axis (master axis).

Synchronous operation here means that the move command for the master axis is issued simultaneously to both the servo motor for the master axis and that for the slave axis. In synchronous operation, the servo motor for the slave axis is not compensated for the deviation which is always detected between the two servo motors.

Synchronous operation is possible during automatic operation, jog feed, manual handle feed using the manual pulse generator, and incremental feed, but is not possible during manual reference position return.

- **Normal operation**

This operating mode is used for machining different workpieces on each table. The operation is the same as in ordinary CNC control, where the movement of the master axis and slave axis is controlled by the independent axis address (Y and V). It is possible to issue the move commands to both the master axis and slave axis in the same block.

- ① The Y axis moves normally according to program command Yyyyy issued to the master axis.
- ② The V axis moves normally according to program command Vvvvv issued to the slave axis.
- ③ The Y axis and the V axis move simultaneously according to program command YyyyyVvvvv.

Both automatic and manual operations are the same as in ordinary CNC control.

CAUTION

- 1 When the automatic reference position return command (G28) and the 2nd/3rd/4th reference position return command (G30) are issued during synchronous operation, the V axis follows the same movement as the Y axis returns to the reference position. If the V axis is positioned at the reference position after the return movement is complete, the reference position return complete signal of the V axis goes on when that of the Y axis goes on.
As a rule, commands G28 and G30 must be issued in the normal operating mode.
- 2 The machine must be manually returned to the reference position in the normal operating mode.
- 3 Both the pitch error and backlash are compensated independently for the master axis and the slave axis.
- 4 Turn on the manual absolute switch (ABS = 1) during synchronous operation. If it is off, the slave axis may not move correctly.
- 5 When the machine is manually returned to the reference position during synchronous operation, both the master axis and the slave axis move synchronously until the acceleration movement is complete. However, grid detection thereafter is carried out independently.
- 6 The operation can be switched between synchronous operation and normal operation by using a signal input from the PMC.

NOTE

- 1 When the automatic reference position return check command (G27) is issued during synchronous operation, the V axis and Y axis move in tandem. If both the Y axis and the V axis have reached their respective reference positions after the movement is complete, the reference position return complete signals go on. If either axis is not at the reference position, an alarm is issued. As a rule, command G27 must be issued in the normal operating mode.
- 2 In synchronous axis control, commands that require no axis motion, such as the workpiece coordinate system setup command (G92) and the local coordinate system setup command (G52), are set to the Y axis by program command Yyyyy issued to the master axis.
- 3 For signals such as external deceleration, interlock, and machine lock, only the signals issued to the master axis are valid in the synchronous operating mode. Signals issued to other axes are ignored.
- 4 The axis which is to be the master axis must be specified by parameter No. 075#0, #1.
- 5 The slave axis is exclusively set to the 4th axis.
- 6 When a move command is issued to the slave axis during synchronous operation, a P/S alarm (No. 213) is issued.

22 ROTARY AXIS ROLL-OVER

General

The roll-over function prevents coordinates for the rotation axis from overflowing. The roll-over function is enabled by setting bit 1 of parameter 398 to 1.

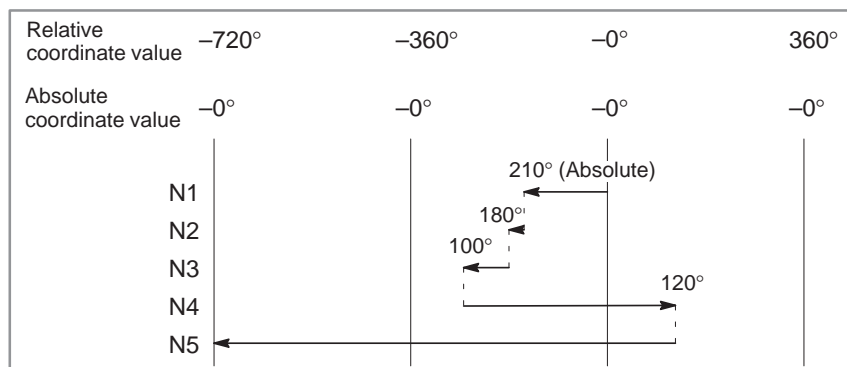
Explanations

For an incremental command, the tool moves the angle specified in the command. For an absolute command, the coordinates after the tool has moved are values set in parameter No. 860, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when bit 2 of parameter No. 398 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 3 of parameter No. 398 is set to 1.

Examples

Assume that axis A is the rotating 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 rotating axis, the axis moves as shown below.

G90 A0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 G90 A-150.0 ;	N1	-150	210
N2 G90 A540.0 ;	N2	-30	180
N3 G90 A-620.0 ;	N3	-80	100
N4 G91 A380.0 ;	N4	+380	120
N5 G91 A-840.0 ;	N5	-840	0



NOTE

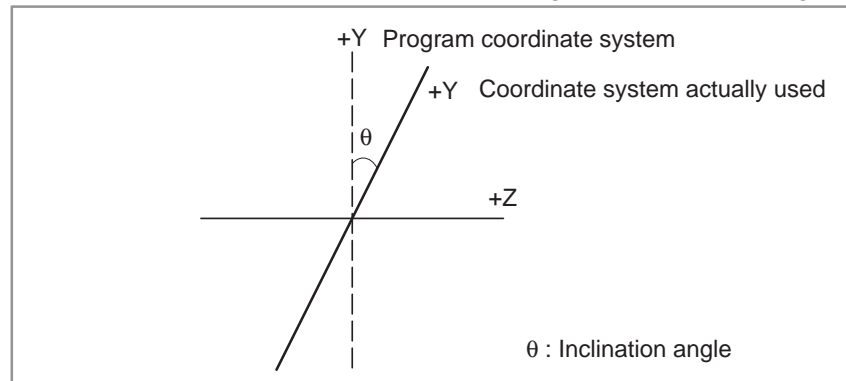
This function cannot be used together with the indexing function of the index table.

23

ANGULAR AXIS CONTROL (0-GSC, 0-GSD/II)

General

When the Y-axis makes an angle other than 90° with the Z-axis, the inclined axis control function controls the distance traveled along each axis according to the inclination angle. A program, when created, assumes that the Y-axis and Z-axis intersect at right angles. However, the actual distance traveled is controlled according to an inclination angle.



Explanations

The distance traveled along each axis is controlled according to the formulas described below.

The distance traveled along the Y-axis is determined by the following formula:

$$Y_a = Y_p / \cos \theta$$

The distance traveled along the Z-axis is corrected by the inclination of the Y-axis, and is determined by the following formula:

$$Z_a = Z_p - Y_p \cdot \tan \theta$$

The feedrate is determined as described below. The speed component along the Y-axis is determined by the following formula:

$$F_a = F_p / \cos \theta$$

Y_a , Z_a , F_a : Actual distance and speed

Y_p , Z_p , F_p : Programmed distance and speed

- **Method of use**

Parameter (No. 077#5) enables or disables the inclined axis control function. If the function is enabled, the distance traveled along each axis is controlled according to an inclination angle (No. 837).

Parameter (No. 077#4) enables Y-axis manual reference point return only with a distance along the Y-axis.

- **Absolute and relative position display**

An absolute and a relative position are indicated in the programmed Cartesian coordinate system.

Machine position display

- **Machine position display**

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle. However, when inch/metric conversion is performed, a position is indicated which incorporates inch/metric conversion applied to the results of inclination angle operation.

WARNING

- 1 After inclined axis control parameter setting, be sure to perform manual reference point return operation.
- 2 If a movement along the Z-axis occurs in Y-axis manual reference point return operation, be sure to perform reference point return operation starting with the Y-axis.
- 3 If an inclination angle close to 0° or $\pm 90^\circ$ is set, an error can occur. (A range from $\pm 20^\circ$ to $\pm 90^\circ$ should be used.)
- 4 Before a Z-axis reference point return check (G37) can be made, Y-axis reference point return operation must be completed.

24

ADVANCED PREVIEW CONTROL

General

This function is designed to enable high-speed, high-precision machining. This function can suppress the delay that is incurred by acceleration/deceleration and the servo system and which increases as a higher feedrate is used. Thus, the tool can exactly follow the specified values, minimizing the machining figure error. This function is optional.

Explanations

Using the command shown below, advanced preview control mode can be turned on and off. High-speed, high-precision machining is performed in advanced preview control mode.

G08 P_p ;

p = 1: Turns advanced preview control mode on.
= 0: Turns advanced preview control mode off.

Note) A block containing G08 must not include any other codes.

In advanced preview control mode, the following functions are enabled:

- 1 Cutting feed linear acceleration/deceleration before interpolation
- 2 Automatic corner deceleration
- 3 Block overlap
- 4 Look-ahead feed forward

For information about these functions, see the relevant sections. Note that dedicated parameters for advanced preview control are available.

NOTE

- 1 In advanced preview control mode, the following cannot be specified:
 - Control along the C-axis normal
 - Cylindrical interpolation
 - Polar coordinate specification
 - F 1-digit feed/threading/synchronous feed
 - Index table indexing
 - Rigid tapping
 - High-speed cycle machining/high-speed remote buffer
 - Handle interrupt
 - Program restart
 - Simple synchronous control
 - Interrupt-type custom macro
 - Automatic tool length measurement
 - Small-diameter peck drilling cycle
 - PMC-based axis control
 - Cs contour control
 - Skip cutting/high-speed skip cutting
 - Automatic signal-based reference position return
- 2 In advanced preview control mode, servo look-ahead feed forward control is used. For information about the servo software and parameters to be set, refer to the documentation provided with the servo system.
- 3 Advanced preview control mode is canceled by the input of a reset signal.

III. OPERATION

1

GENERAL



1.1 MANUAL OPERATION

Explanations

- **Manual reference position return**
(See Section III-3.1)

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.

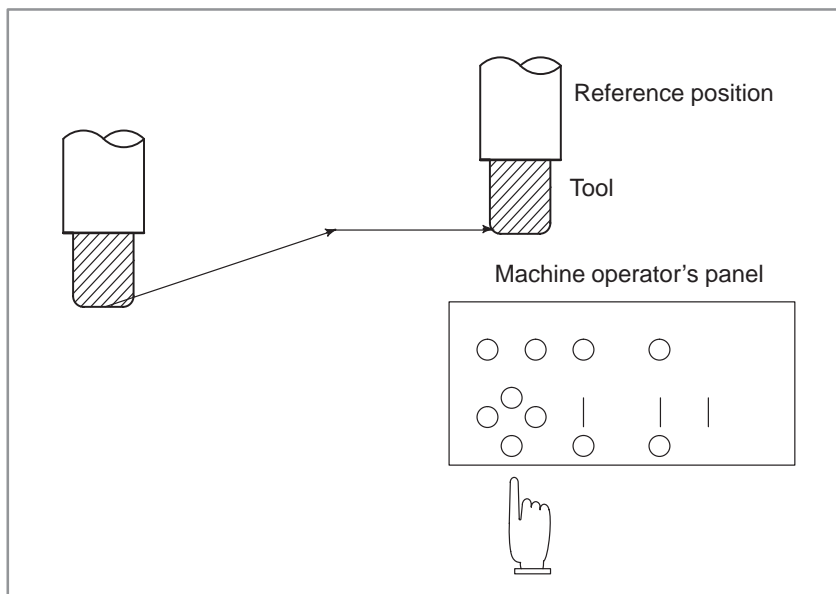


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.

This operation is called automatic reference position return (See Section II-6).

- **The tool movement by manual operation**

Using machine operator's panel switches, pushbuttons, or the manual handle, the tool can be moved along each axis.

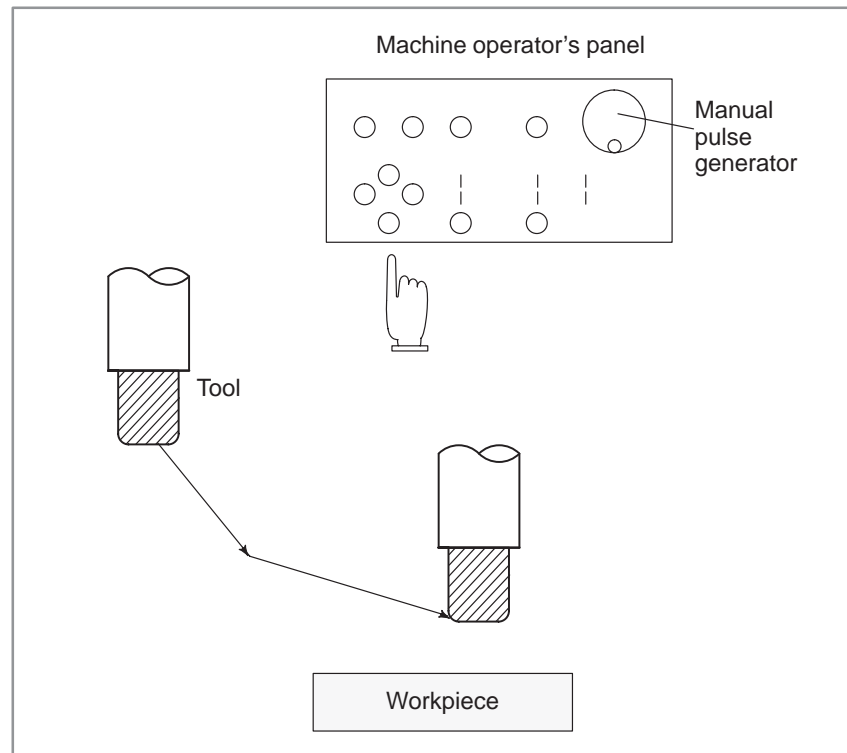


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

- (i) Jog feed (See Section III-3.2)
The tool moves continuously while a pushbutton remains pressed.
- (ii) Incremental feed (See Section III-3.3)
The tool moves by the predetermined distance each time a button is pressed.
- (iii) Manual handle feed (See Section III-3.4)
By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMMING— AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, DNC, and MDI operations. (See Section III-4).

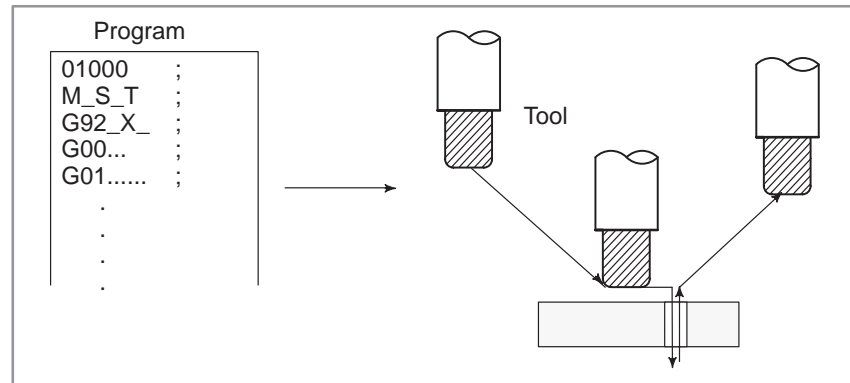


Fig. 1.2 (a) Tool Movement by Programming

Explanations

- **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.

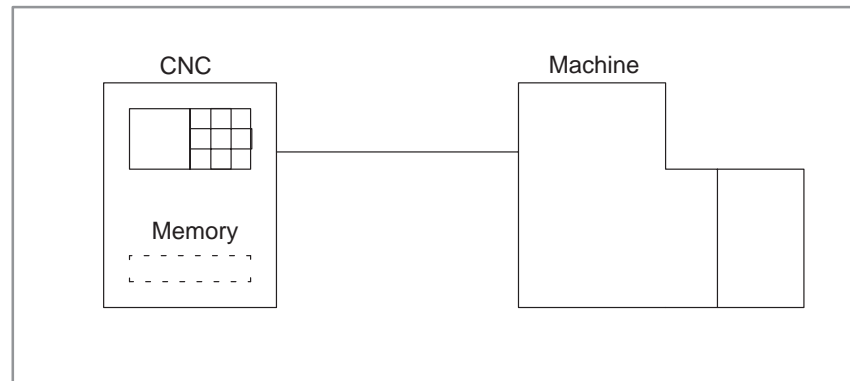


Fig. 1.2 (b) Tool Movement by Programming

- **DNC operation**

In this mode of operation, the program is not registered in the CNC memory. It is read from the connected I/O unit instead. This mode is useful when the program is too large to fit the CNC memory. It can also be used for high-speed machining by means of a remote buffer.

- **MDI operation**

After the program is entered, as an command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.

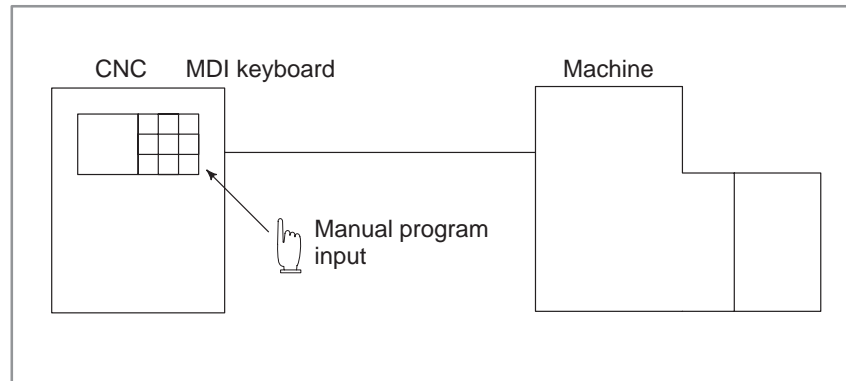


Fig. 1.2 (c) MDI operation

1.3 AUTOMATIC OPERATION

Explanations

- **Program selection**

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number (Section III-9.3).

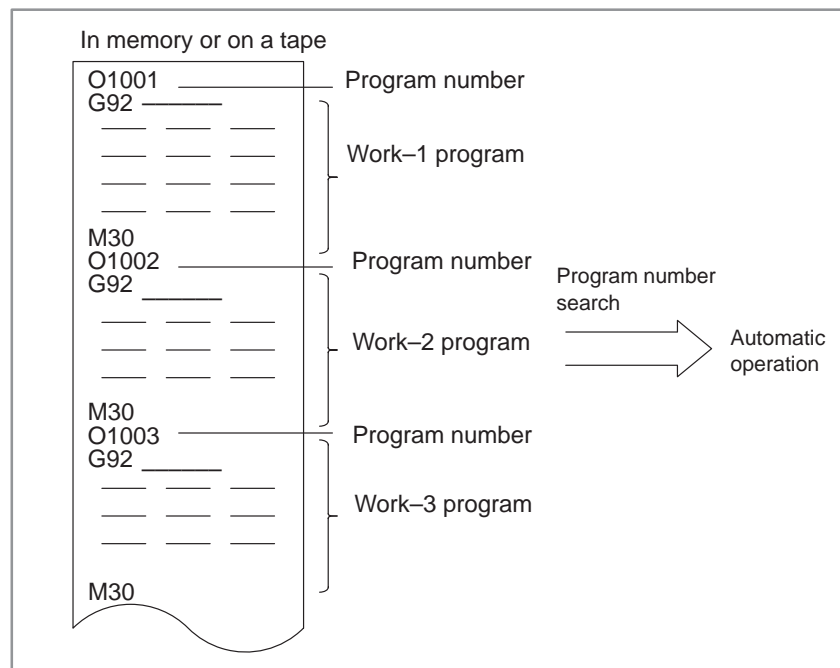


Fig. 1.3 (a) Program selection for automatic operation

- **Start and stop**
(See Section III-4)

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.

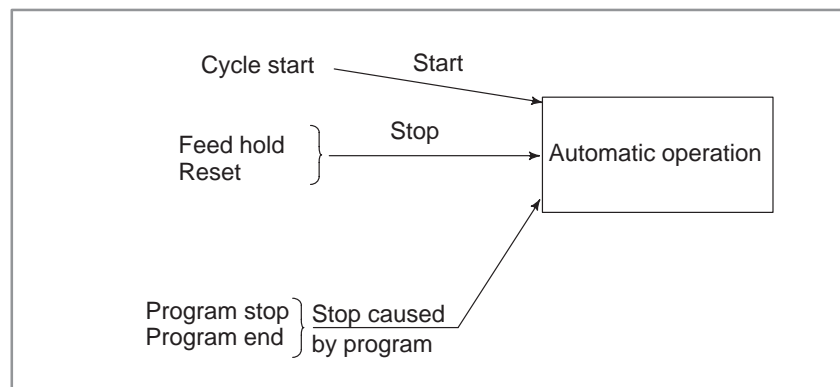


Fig. 1.3 (b) Start and stop for automatic operation

- **Handle interruption**
(See Section III-4.7)

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle.

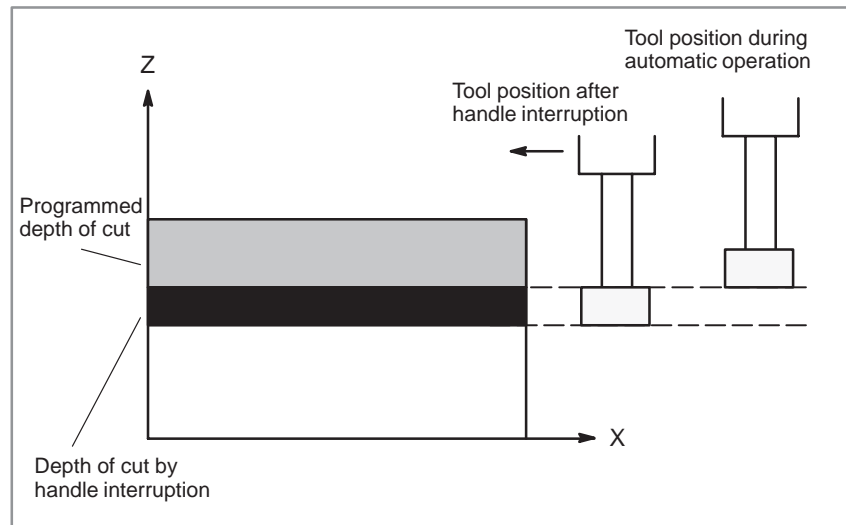


Fig. 1.3 (c) Handle interruption for automatic operation

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Section III-5).

1.4.1 Check by Running the Machine

Explanations

- **Dry run**
(See Section III-5.4)

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel.

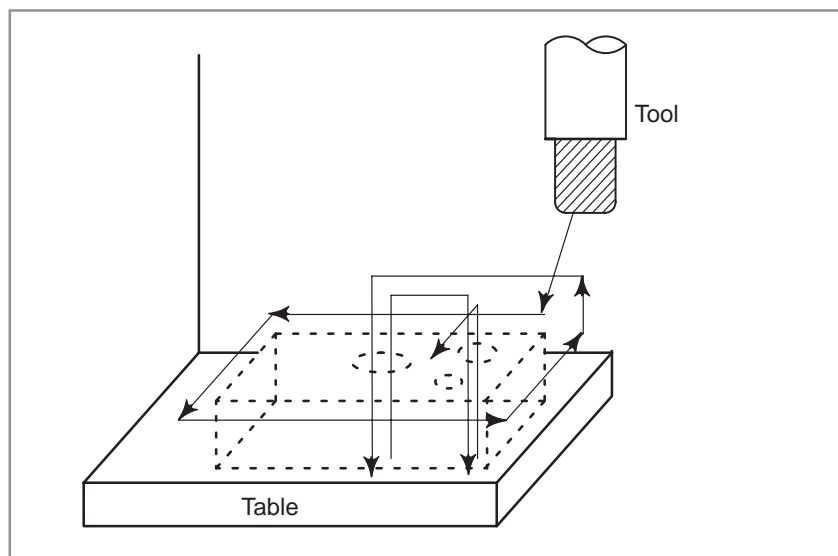


Fig. 1.4.1 (a) Dry run

- **Feedrate override**
(See Section III-5.2)

Check the program by changing the feed rate specified by the program.

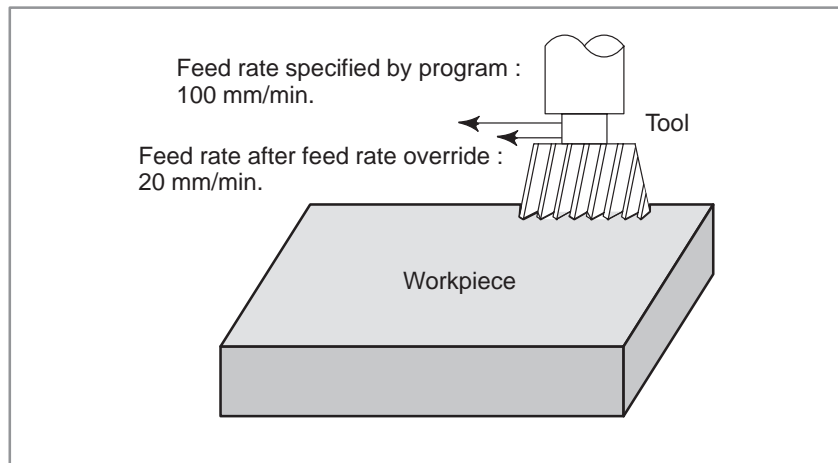


Fig. 1.4.1 (b) Feedrate override

- **Single block**
(See Section III-5.5)

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.

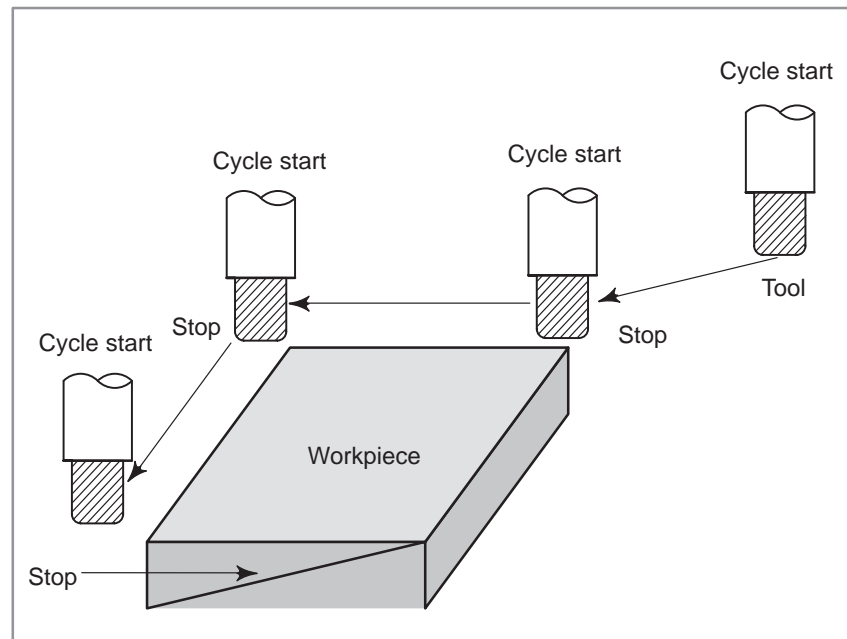


Fig. 1.4.1 (c) Single block

1.4.2

How to View the Position Display Change without Running the Machine

Explanations

- **Machine lock**
(See Sections III-5.1)

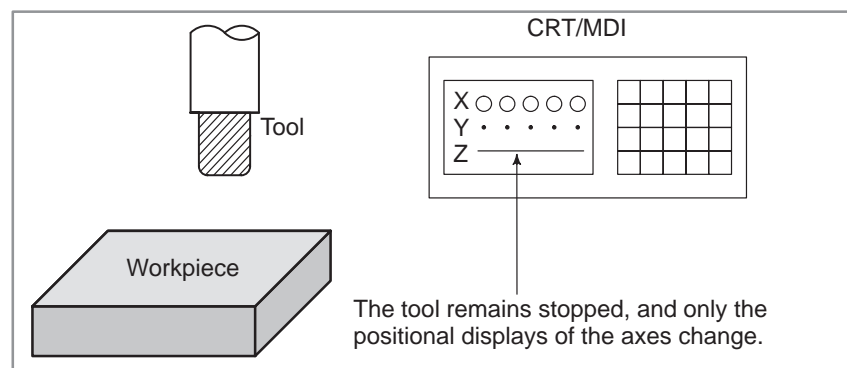


Fig. 1.4.2 Machine lock

- **Auxiliary function lock**
(See Section III-5.1)

When automatic running is placed into the auxiliary function lock mode during the machine lock mode, all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) are disabled.

1.5 EDITING A PART PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the CRT/MDI panel (See Section III-9). This operation can be executed using the part program storage/edit function.

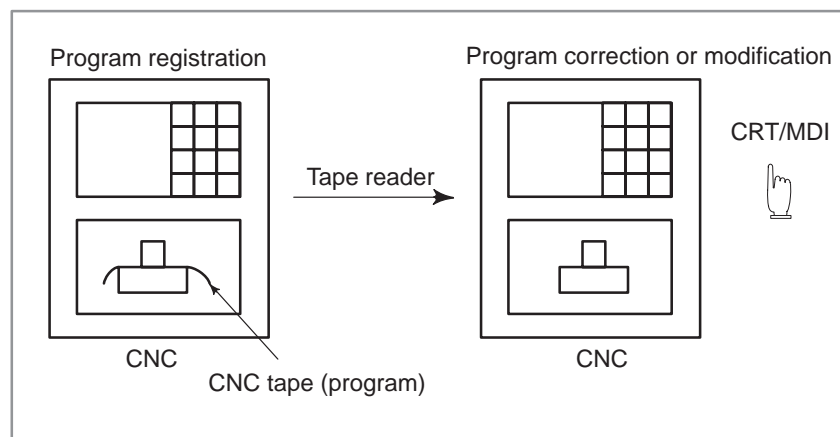


Fig. 1.5 Part program editing

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the CRT/MDI screen (See III-11).

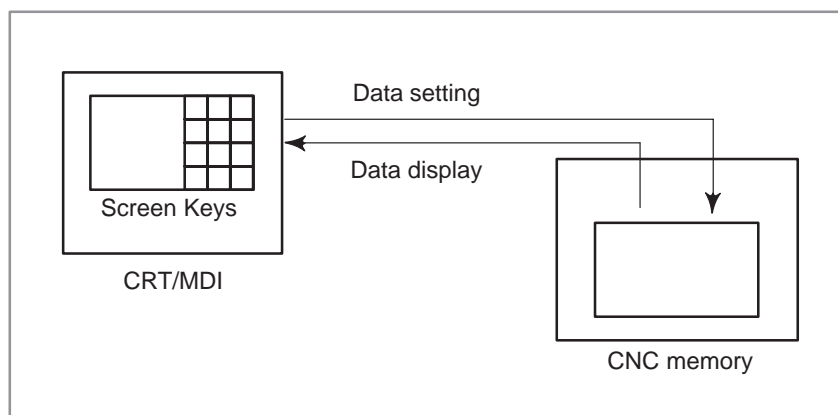


Fig. 1.6 (a) Displaying and setting data

Explanations

- Offset value

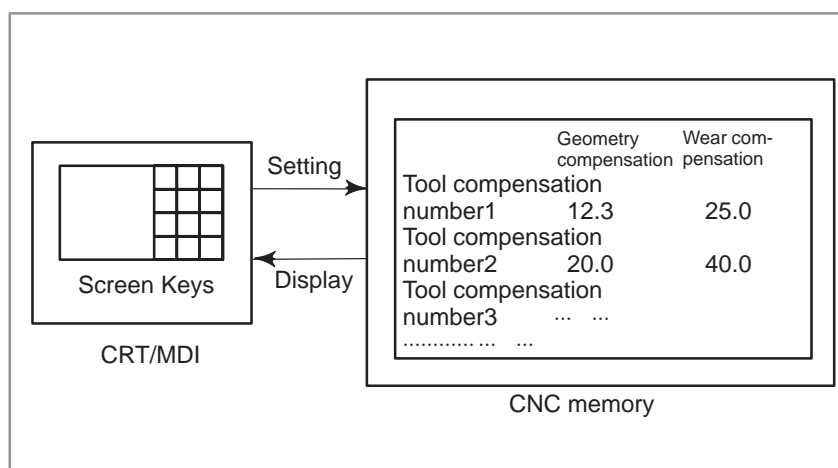


Fig. 1.6 (b) Displaying and setting offset values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions. By setting tool dimension data in CNC memory beforehand, 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 Section III-11.4.1).

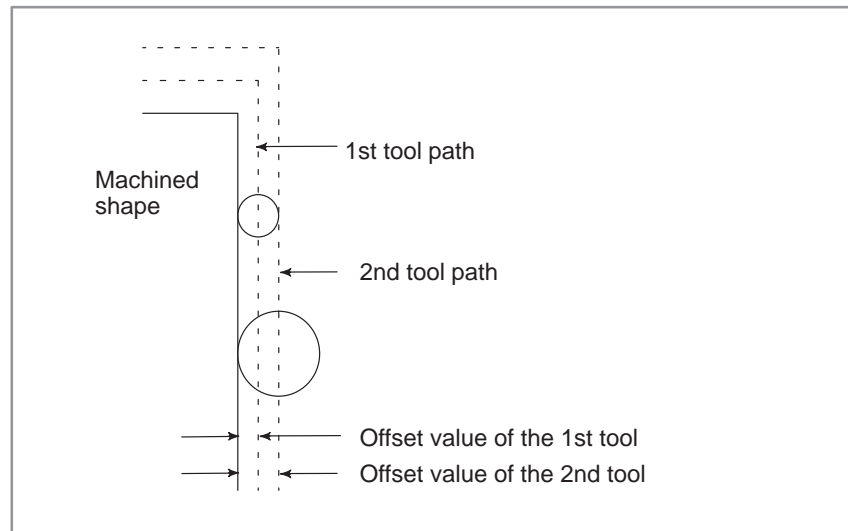


Fig. 1.6 (c) Offset value

- **Displaying and setting operator's setting data**

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.

For example, the following data can be set:

- Inch/Metric switching
- Selection of I/O devices
- Mirror image cutting on/off

The above data is called setting data (See Section III-11.5.3).

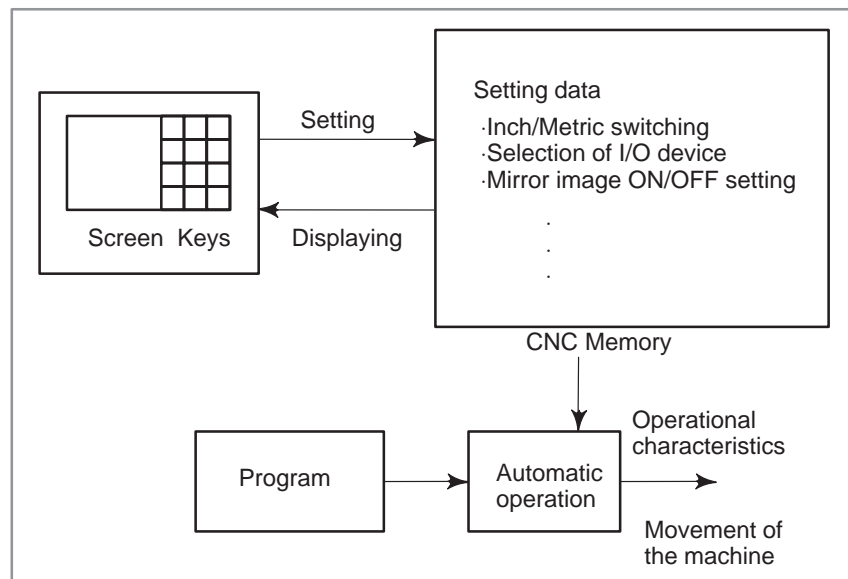


Fig. 1.6 (d) Displaying and setting operator's setting data

• Displaying and setting parameters

The CNC functions have versatility in order to take action in characteristics of various machines.

For example, CNC can specify the following:

- Rapid traverse rate of each axis
- Whether increment system is based on metric system or inch system.
- How to set command multiply/detect multiply (CMR/DMR)

Data to make the above specification is called parameters (See Section III-11.5.1).

Parameters differ depending on machine tool.

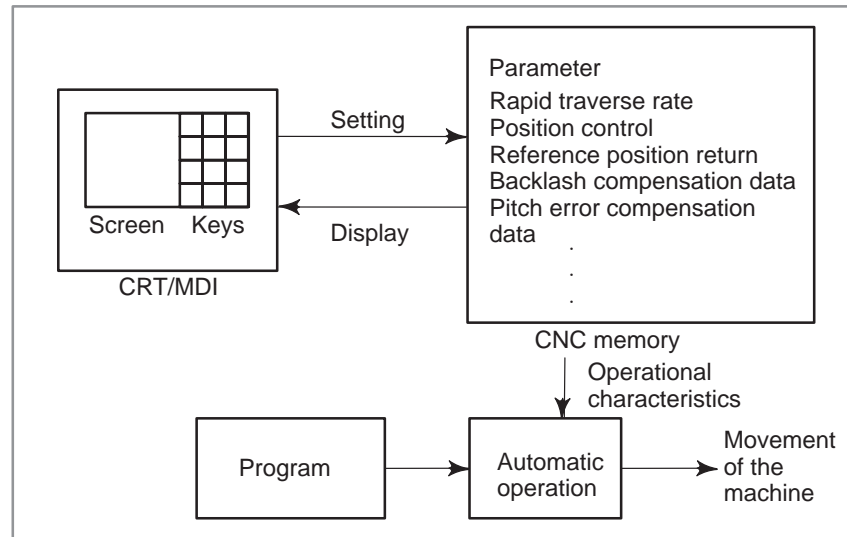


Fig. 1.6 (e) Displaying and setting parameters

• Data protection key

A key called the data protection key can be defined. It is used to prevent part programs from being registered, modified, or deleted erroneously (See Section III-11).

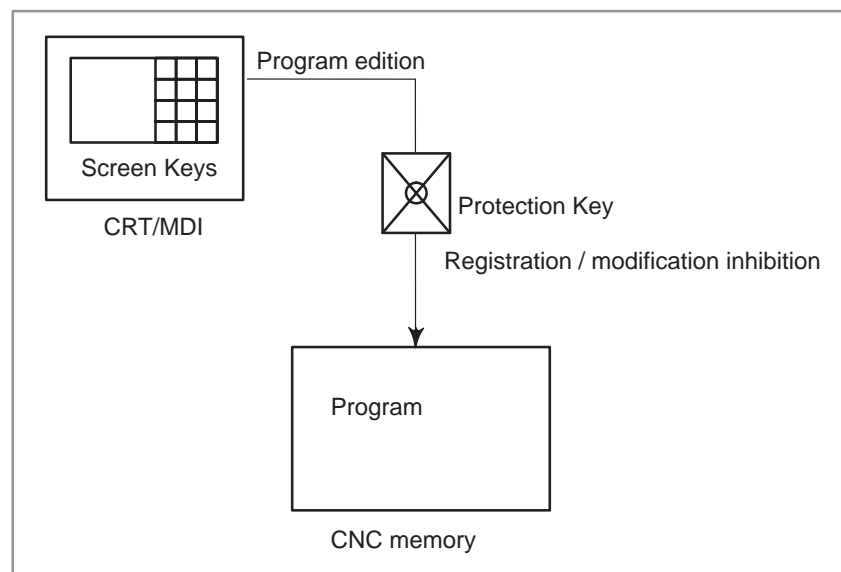
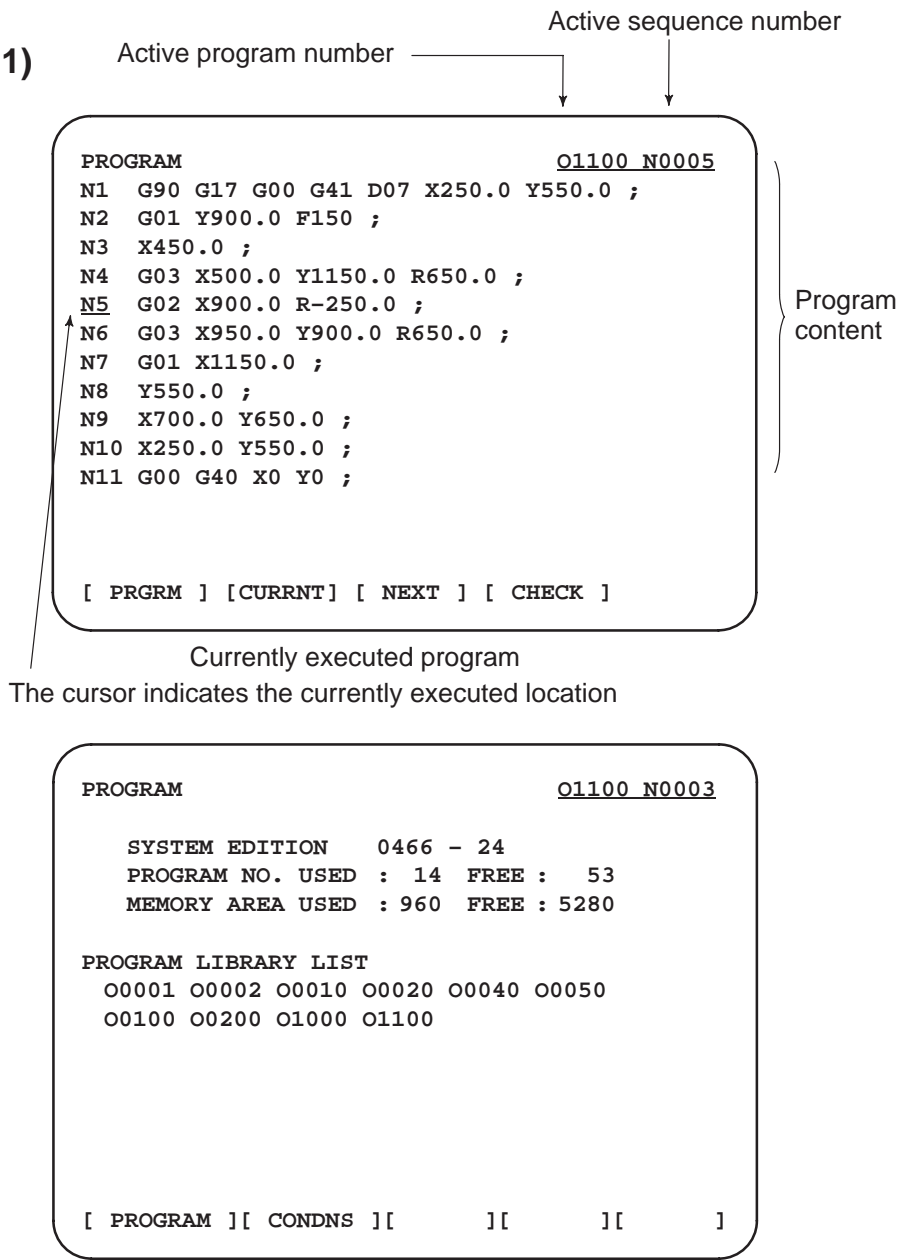


Fig. 1.6 (f) Data protection key

1.7 DISPLAY

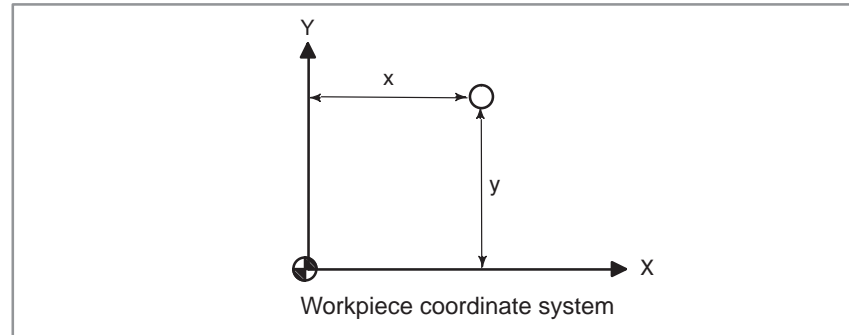
1.7.1 Program Display (See Subsections III-11.2.1 and III-11.3.1)

The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed.



1.7.2 Current Position Display (See Section III-11.1.1 to 11.1.3)

The current position of the tool is displayed with the coordinate values. The distance from the current position to the target position can also be displayed.



ACTUAL POSITION (ABSOLUTE)		O0003 N0003
X	150.000	
Y	300.000	
Z	100.000	
[ABS][REL][ALL][HNDL][]		

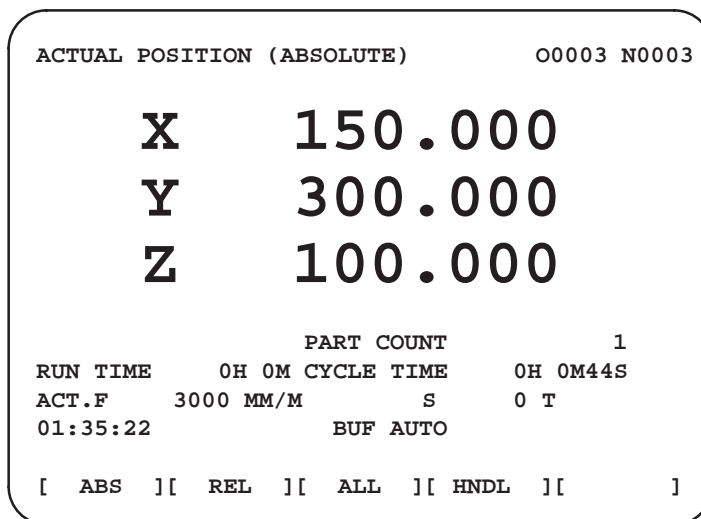
1.7.3 Alarm Display (See Section III-7.1)

When a trouble occurs during operation, error code and alarm message are displayed on CRT screen. See APPENDIX G for the list of error codes and their meanings.

ALARM MESSAGE		O1000 N0003
010	P/S ALARM	
		S 0T
09:52:24	ALARM	
[ALARM][OPR][MSG][]		

1.7.4 Parts Count Display, Run Time Display (See Section III-11.1.5)

When an option is selected, two types of run time and number of parts are displayed on the screen.



1.7.5 Graphic Display (See Section III-12)

Programmed tool movement can be displayed on the following planes:

- 1) XY plane
- 2) YZ plane
- 3) XZ plane
- 4) Three dimensional display

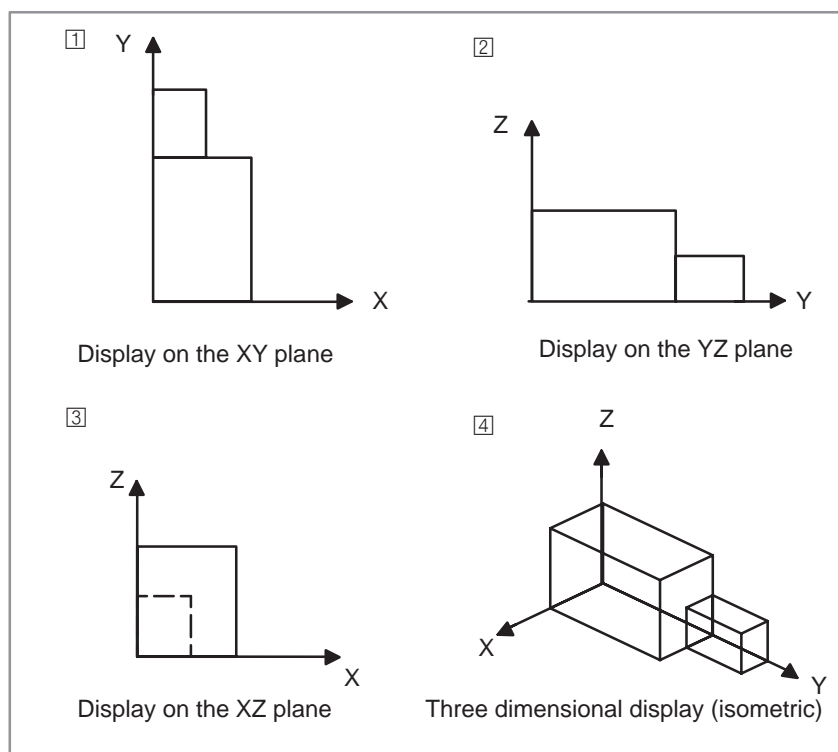


Fig. 1.7.5 Graphic display

1.8 DATA OUTPUT (SEE CHAPTER III-8)

Programs, offset values, parameters, etc. input in CNC memory can be output to paper tape, cassette, or a floppy disk for saving. After once output to a medium, the data can be input into CNC memory.

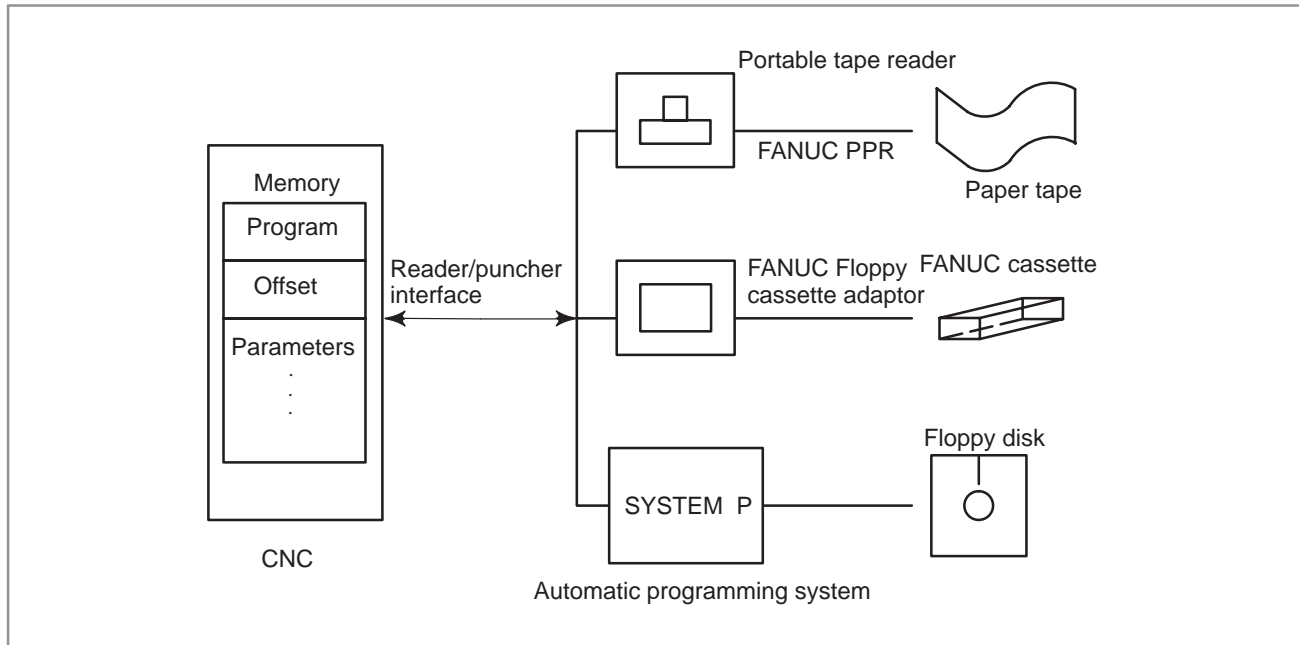
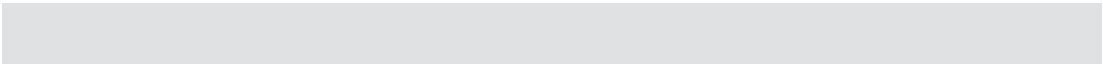


Fig. 1.8 Data output

2

OPERATIONAL DEVICES



The peripheral devices available include the CRT/MDI panel attached to the CNC, machine operator's panel and external input/output devices such as tape reader, PPR, floppy cassette, and FA card.

2.1

CRT/MDI PANELS AND LCD/MDI PANELS

Figs. 2.1 (a) to 2.1 (e) show the CRT/MDI and LCD/MDI panels.

- 9" small monochrome or color CRT/MDI panel (with softkey) Fig.2.1(a)
- 9" monochrome or color CRT/MDI panel (without softkey) Fig.2.1(b)
- 9" monochrome or color CRT/MDI panel (with full key) Fig.2.1(c)
- Thin type display/MDI pane Fig.2.1(d)
- 14" color CRT/MDI panel Fig.2.1(e)

External view

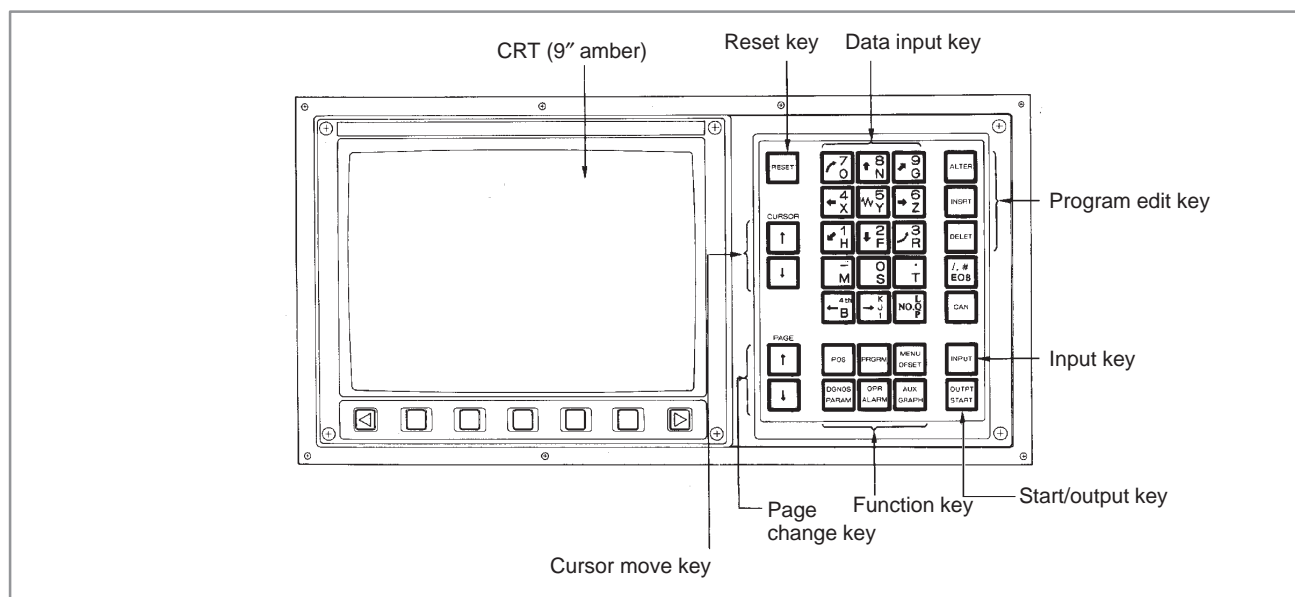


Fig. 2.1 (a) 9" small monochrome or color CRT/MDI (with softkey)

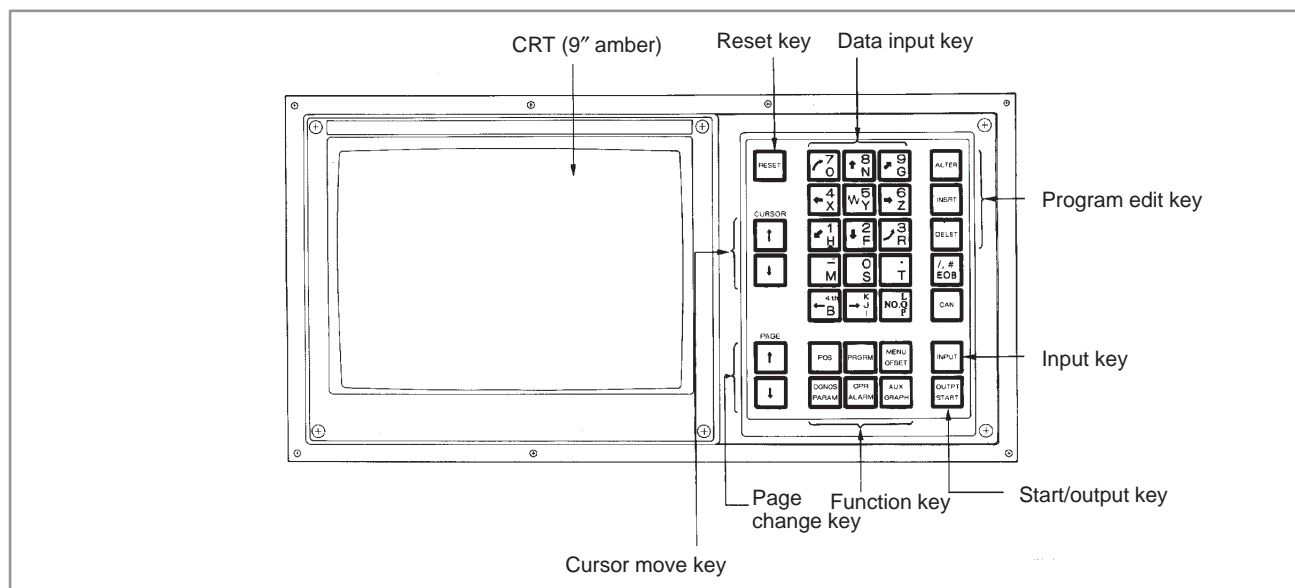


Fig. 2.1 (b) 9" monochrome or color CRT/MDI panel (without softkey)

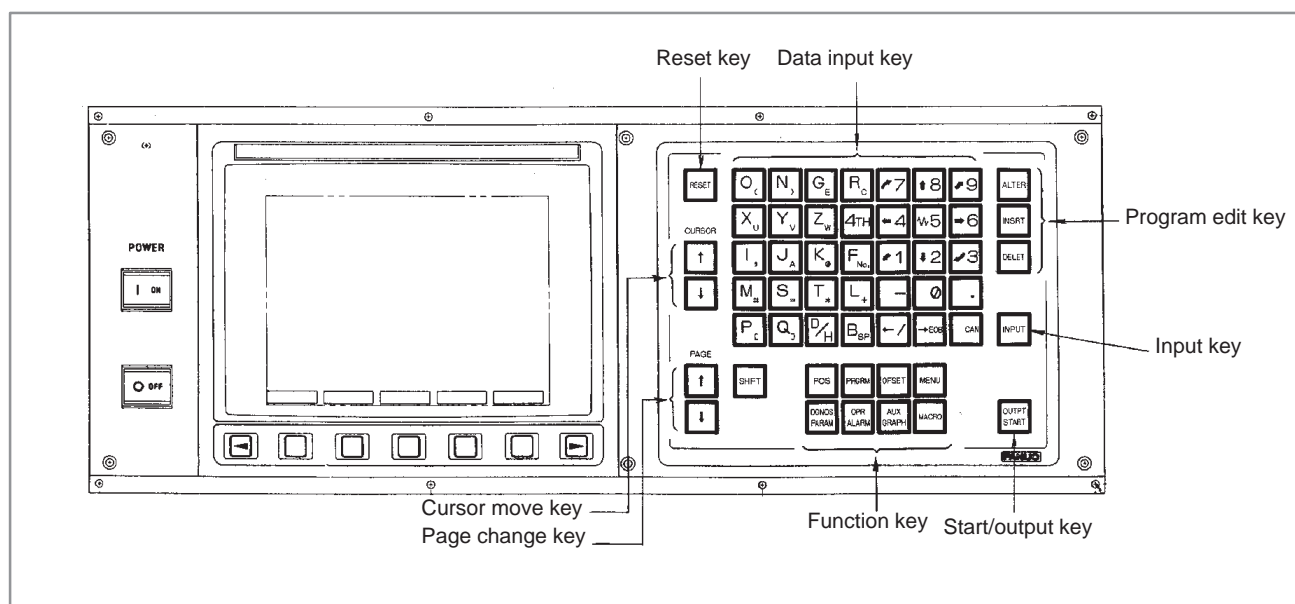


Fig. 2.1 (c) 9" monochrome or color CRT/MDI panel (with full key)

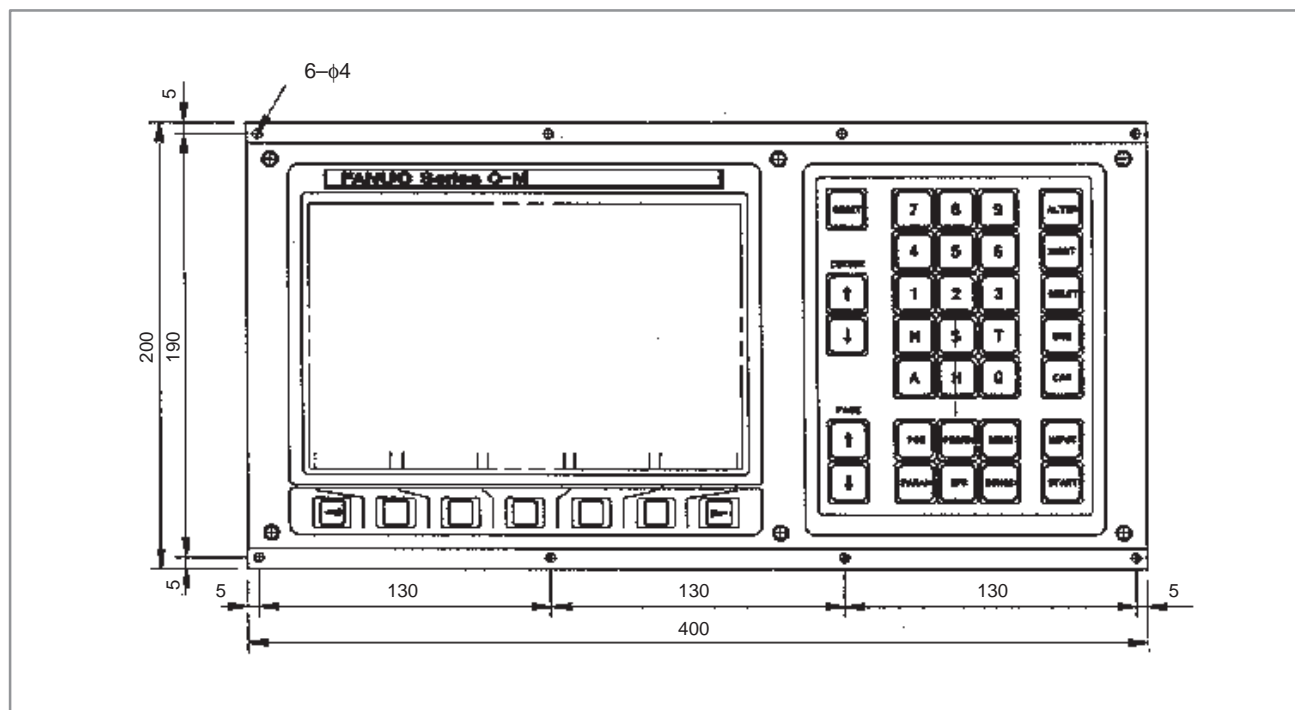


Fig. 2.1 (d) Thin type display/MDI panel

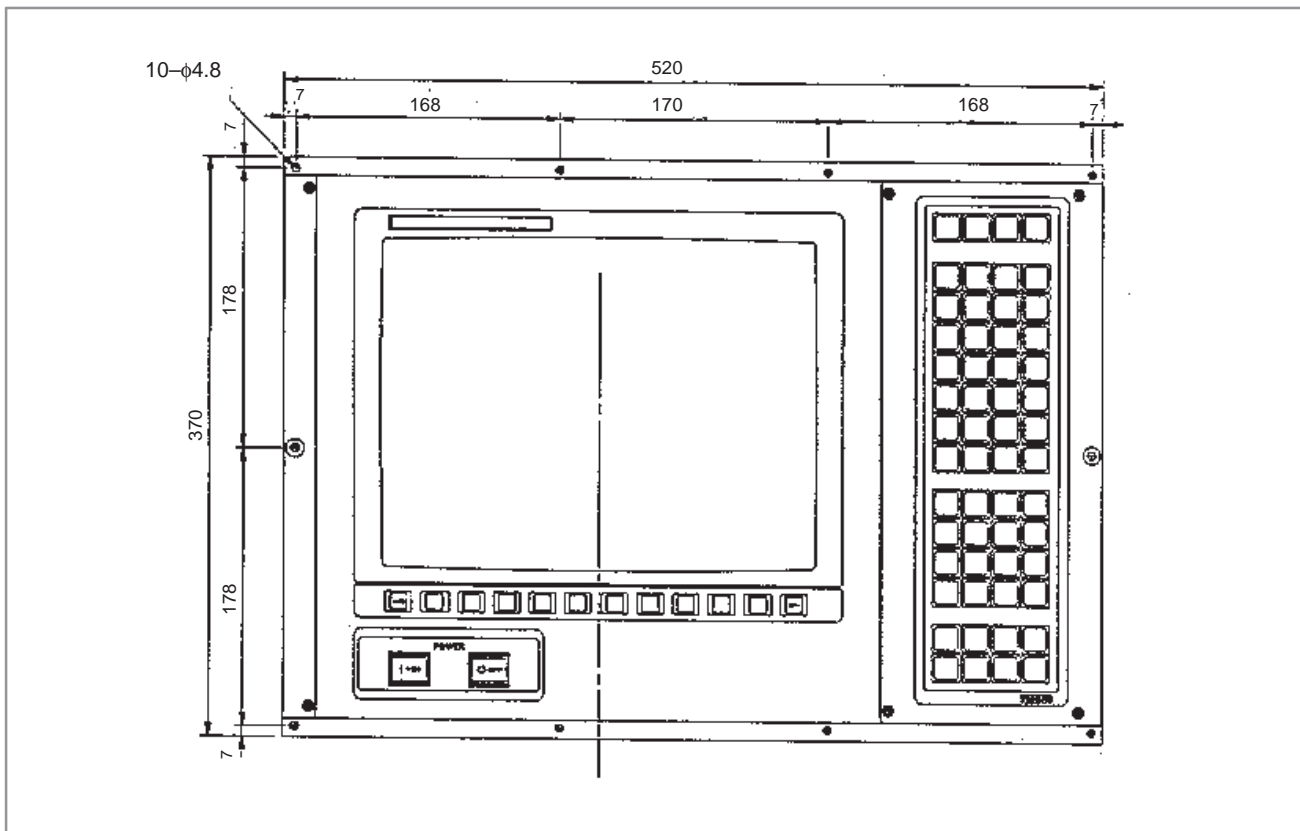


Fig. 2.1 (e) 14" color CRT/MDI panel

Explanation of the keyboard

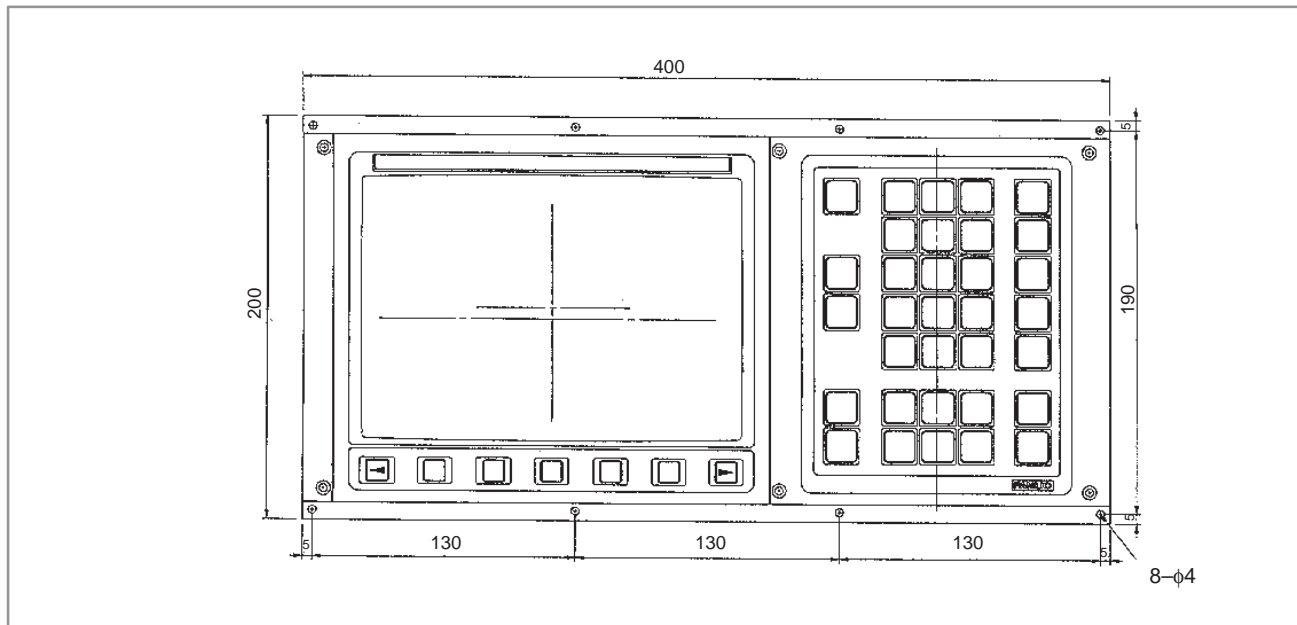


Fig. 2.1 (f) 9" small monochrome/or color CRT/MDI panel (with soft key)

Table 2.1 Explanation of the MDI keyboard


























Number	Name	Explanation
1	Power ON and OFF buttons  	Press these buttons to turn CNC power ON and OFF.
2	RESET key 	Press this key to reset the CNC, to cancel an alarm, etc.
3	START key 	This key is used to start MDI operation or automatic operation, depending on the machine. Refer to the manual provided by the machine tool builder. This key is also used to output data to an input/output unit.
4	Soft keys (option)	The soft keys have various functions, according to the Applications. The soft key functions are displayed at the bottom of the CRT screen.
5	Address and numeric keys   ...	Press these keys to input alphabetic, numeric, and other characters.
6	SHIFT key 	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters. Special character Λ is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered. (The standard MDI key panel does not have this key.)
7	INPUT key 	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the CRT screen. To copy the data in the key input buffer to the offset register, etc., press the <INPUT> key. This key is also used to input data from an input/output unit.

Table 2.1 Explanation of the MDI keyboard

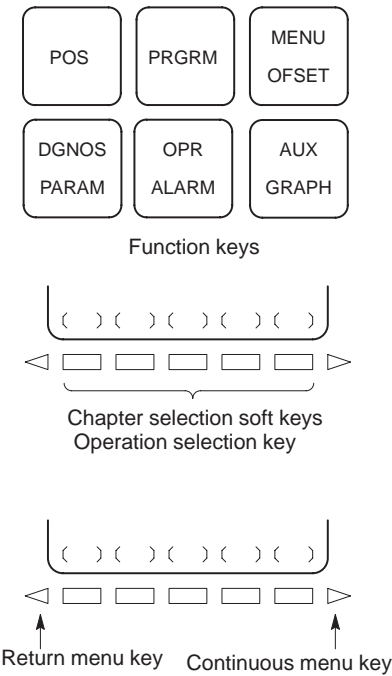
Number	Name	Explanation
8	Cancel key 	Press this key to delete the last character or symbol input to the key input buffer.
9	Program edit keys   	Press these keys when editing the program.  : Alteration  : Insertion  : Deletion
10	Function keys   ...	Press these keys to switch display screens for each function. See sec. 2.2 for details of the function keys.
11	Cursor move keys CURSOR  	There are two different cursor move keys.  This key is used to move the cursor in a downward or forward direction. The cursor is moved in small units in the forward direction.  This key is used to move the cursor in an upward or reverse direction. The cursor is moved in small units in the reverse direction.
12	Page change keys PAGE  	Two kinds of page change keys are described below.  This key is used to changeover the page on the CRT screen in the forward direction.  This key is used to changeover the page on the CRT screen in the reverse direction.

2.2

FUNCTION KEYS AND SOFT KEYS

2.2.1

General Screen Operations



- 1 Press a function key on the CRT/MDI panel. The chapter selection soft keys that belong to the selected function appear.
- 2 Press one of the soft keys. The screen for the selected chapter appears. If the soft key for a target chapter is not displayed, press the continuous menu key (next-menu key). In some cases, additional chapters can be selected within a chapter.
- 3 To redisplay the previous soft keys, press the return menu key.

The general screen display procedure is explained above. However, the actual display procedure varies from one screen to another. For details, see the description of individual operations.

If the soft keys are not provided, press a key having the equivalent function to select the desired chapter. The functions of the soft keys are automatically displayed, depending on the configuration. The functions are displayed, irrespective of the function configuration, if bit 7 of parameter 048 is specified accordingly.

2.2.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the CRT/MDI panel:

POS

Press this key to display the **position screen**.

PRGRM

Press this key to display the **program screen**.

MENU
OFFSET

Press this key to display the **offset screen**.

DGNOS
PARAM

Press this key to display the **parameter/diagnosis screen**.

OPR
ALARM

Press this key to display the **alarm screen**.

AUX
GRAPH

Press this key to display the **graphics screen**.

2.2.3

Key Input and Input Buffer

Explanations

- For standard key

When an address and a numerical 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 CRT screen.

The key input buffer begins with “ADDRESS”. When an address is input, the key input buffer begins with “NUMERIC”.

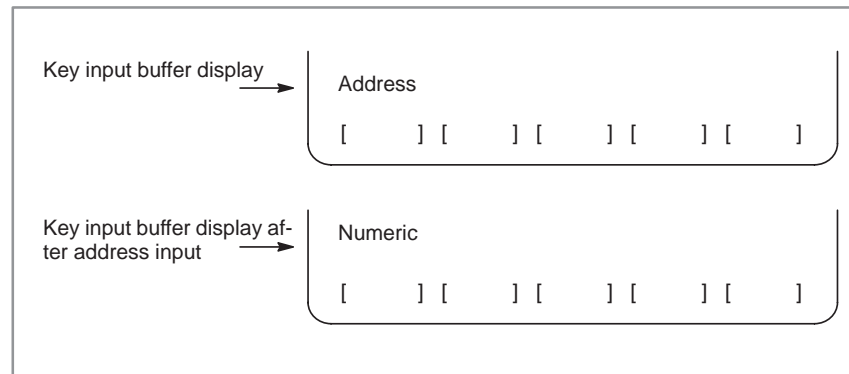
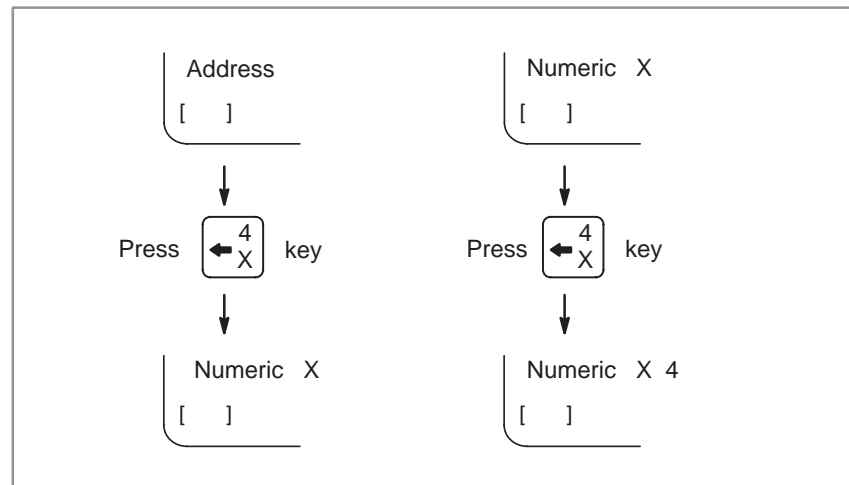


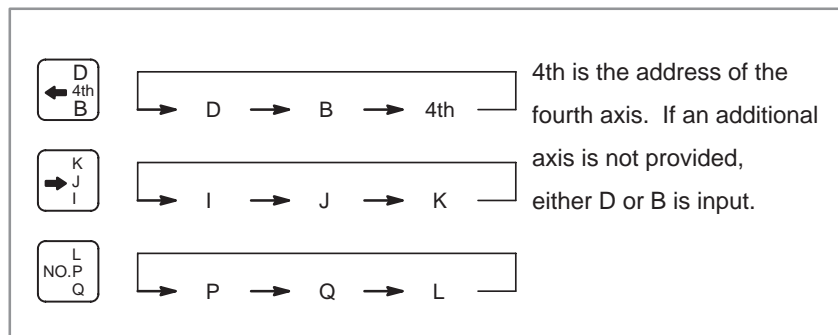
Fig. 2.2.3 (a) Key input buffer display

On the standard key panel, the same key is used to input both an address and a numeric value.

When the key input buffer begins with “ADDRESS”, pressing that key inputs the address. When the key input buffer begins “NUMERIC”, pressing that key inputs the numeric value.



Data of one word (address + numeric value) can be entered into the key input buffer at one time. The following data input keys are used to input addresses. Each time the key is pressed, the input address changes as shown below:



Pressing the **CAN** key deletes all the data input to the key input buffer.

When bit 7 of parameter 0394 is set to 1, each press of the **CAN** key deletes only the most recently entered character during data input using the parameter, diagnostic, or offset screen.

• For full key

When an address and a numerical 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 CRT screen.

A "<" is displayed at the end of the key input data indicating the input position of the next character.

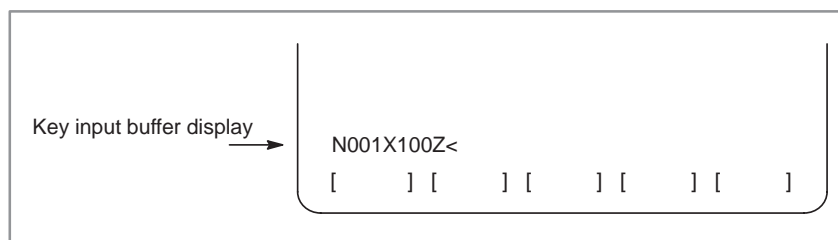


Fig. 2.2.3 (b) Key input buffer display

To input the lower character of the keys that have two characters inscribed on them, first press the **SHIFT** key and then the key in question.

When the **SHIFT** key is pressed, "<" indicating the next character input position changes to "Λ". Now lowercase characters can be entered (shift state).

When a character is input in shift status the shift status is canceled. Furthermore, if the **SHIFT** key is pressed in shift status, the shift status is canceled.

It is possible to input up to 32 characters at a time in the key input buffer. Press the **CAN** key to cancel the last 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.

2.2.4

14" CRT Soft Key Configuration

There are 12 soft keys in the 14" CRT/MDI panel. As illustrated below, the 5 soft keys on the right and those on the right and left edges operate in the same way as the 9" CRT, whereas the 5 keys on the left hand side are expansion keys dedicated to the 14" CRT.

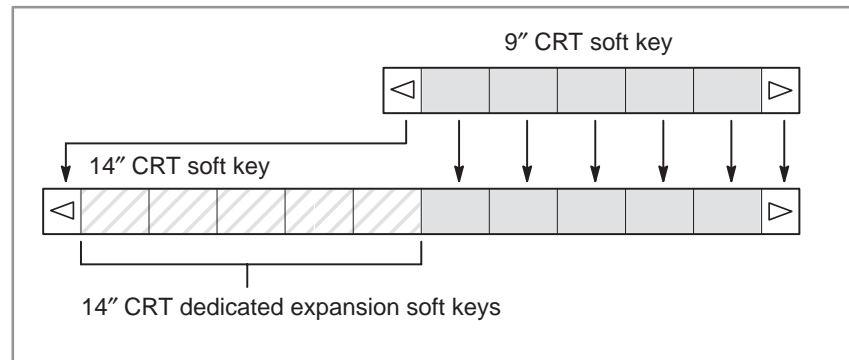


Fig. 2.2.4 CRT and LCD soft key configuration

Whenever a position display appears in the left half of the screen after a function key other than POS is pressed, the soft keys on the left half of the soft key display area are displayed as follows:



2.3 EXTERNAL I/O DEVICES

Five types of external input/output devices are available. This section outlines each device. For details on these devices, refer to the corresponding manuals listed below.

Table 2.3 External I/O device

Device name	Usage	Max. storage capacit	Reference manual
FANUC Handy File	Easy-to-use, multi function input/output device. It is designed for FA equipment and uses floppy disks.	3600m	B-61834E
FANUC Floppy Cassette	Input/output device. Uses floppy disks.	2500m	B-66040E
FANUC FA Card	Compact input/output device. Uses FA cards.	160m	B-61274E
FANUC PPR	Input/output device consisting of a paper tape reader, tape punch, and printer.	275m	B-58584E
Portable Tape Reader	Input device for reading paper tape.	—	Appendix H

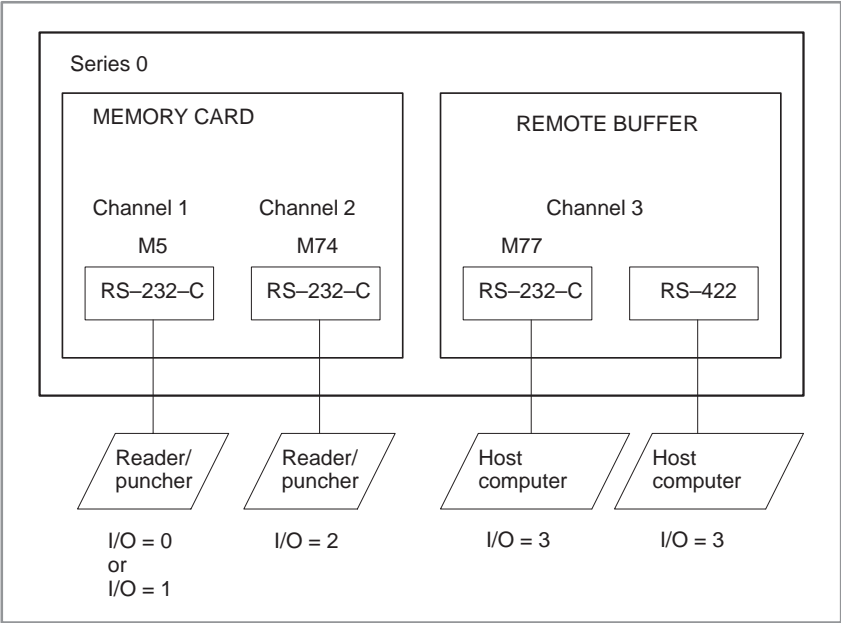
The following data can be input/output to or from external input/output devices:

- 1. Programs**
- 2. Offset data**
- 3. Parameters**
- 4. Custom macro common variables**

For how data is input and output, see Chapter 8.

Parameter

Before an external input/output device can be used, parameters must be set as follows.

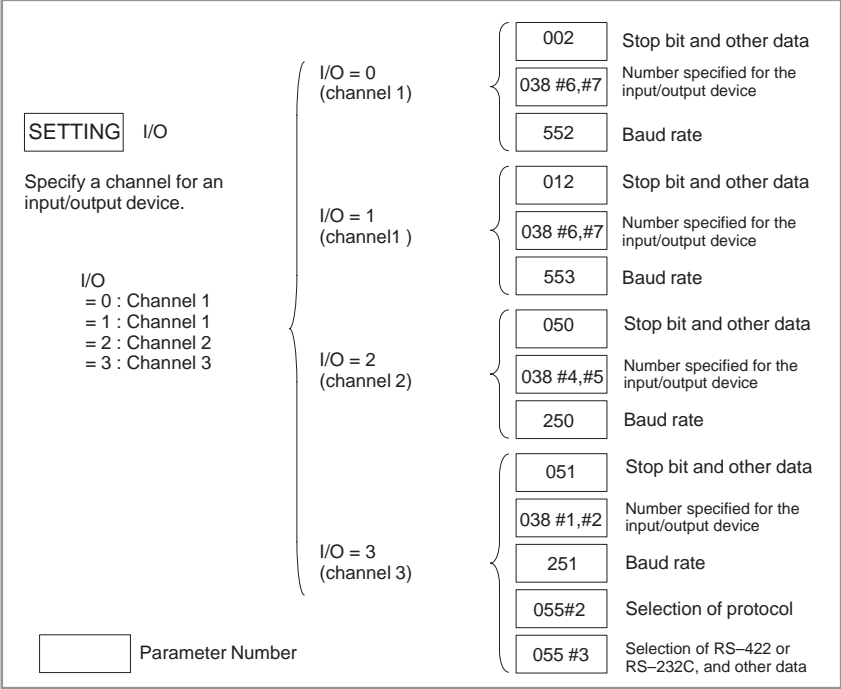


Series 0 has three channels of reader/punch interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting parameter I/O.

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.

For channel 1, two combinations of parameters to specify the input/output device data are provided.

The following shows the interrelation between the reader/punch interface parameters for the channels.

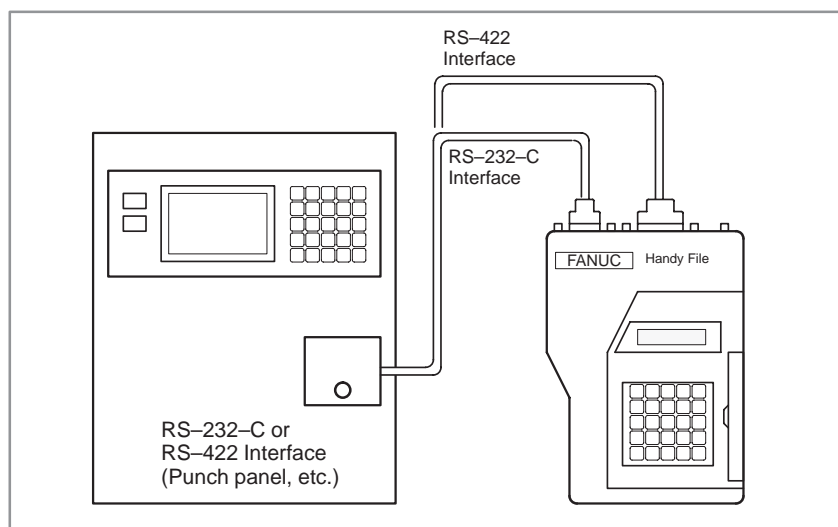


2.3.1 FANUC Handy File

The Handy File is an easy-to-use, multi function floppy disk input/output device designed for FA equipment. By operating the Handy File directly or remotely from a unit connected to the Handy File, programs can be transferred and edited.

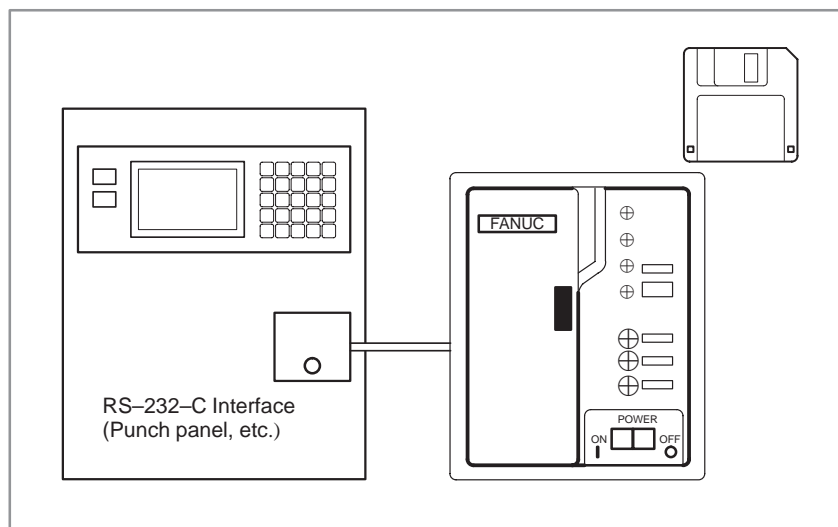
The Handy File uses 3.5-inch floppy disks, which do not have the problems of paper tape (i.e., noisy during input/output, easily broken, and bulky).

One or more programs (up to 1.44M bytes, which is equivalent to the memory capacity of 3600-m paper tape) can be stored on one floppy disk.



2.3.2 FANUC Floppy Cassette

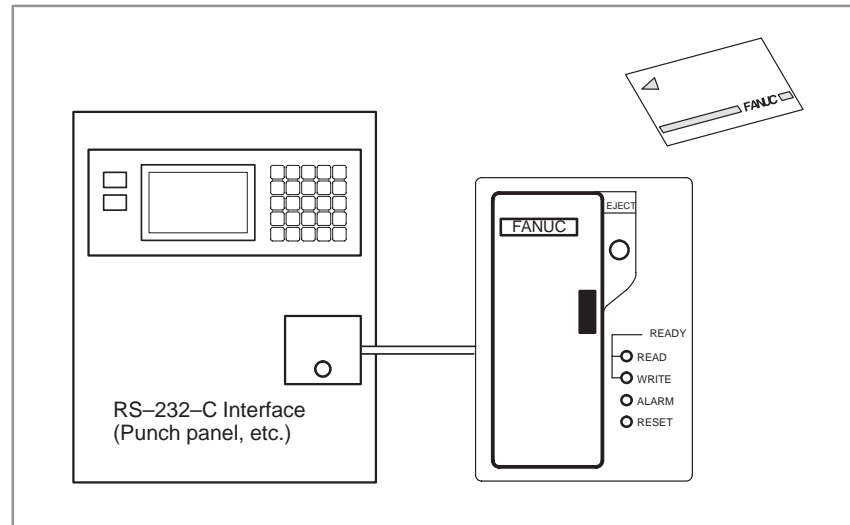
When the Floppy Cassette is connected to the NC, machining programs stored in the NC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the NC.



2.3.3 FANUC FA Card

An FA Card is a memory card used as an input medium in the FA field. It is compact, but has a large memory capacity with high reliability, and requires no special maintenance.

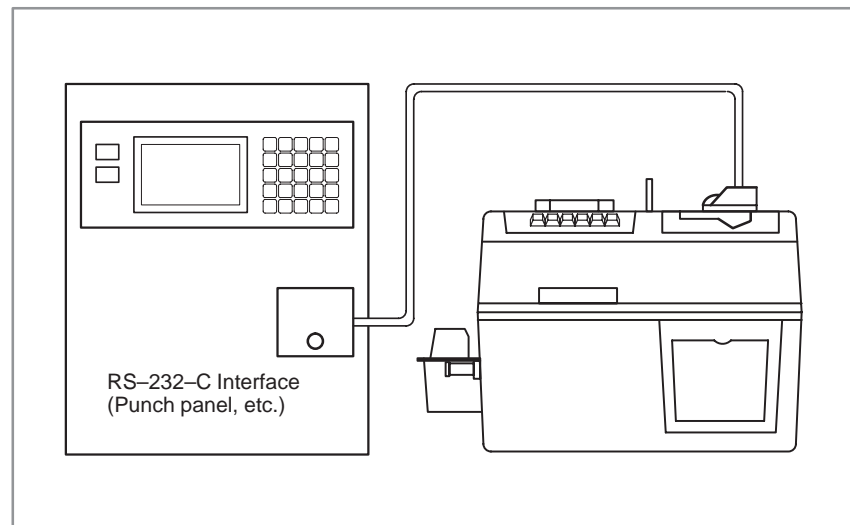
When an FA Card is connected to the CNC via the card adapter, NC machining programs stored in the CNC can be transferred to and saved in an FA Card. Machining programs stored on an FA Card can also be transferred to the CNC.



2.3.4 FANUC PPR

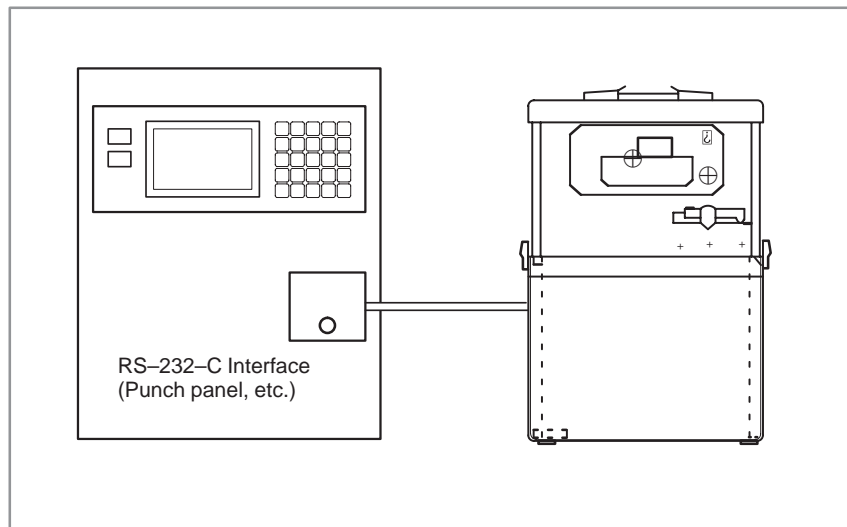
The FANUC PPR consists of three units: A printer, paper tape punch, and paper tape reader.

When the PPR is used alone, data can be read from the tape reader and printed or punched out. It is also possible to perform TH and TV checks on data that was read.



2.3.5 Portable Tape Reader

The portable tape reader is used to input data from paper tape.



2.4 POWER ON/OFF

2.4.1 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. If the software configuration screen is being displayed, a system failure may have occurred.

ACTUAL POSITION (ABSOLUTE)		00010 N0010	
X	123.456		
Y	363.233		
Z	0.000		
PART COUNT		16	
RUN TIME	0H57M	CYCLE TIME	0H 0M38S
ACT.F	3000 MM/M	S	0 T
16:17:47	BUF AUTO		
[ABS]	[REL]	[ALL]	[HNDL]

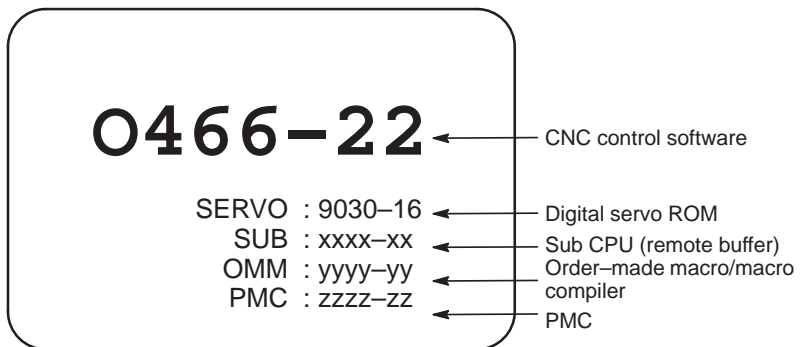
Position display (for 9" CRT)

- 4 Check that the fan motor is rotating.

WARNING

When pressing the <POWER ON> key, do not touch any other CRT/MDI or thin type display/MDI panel keys, until the positional or alarm screen is displayed. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

2.4.2 Display of Software Configuration



2.4.3 Power Disconnection

Procedure

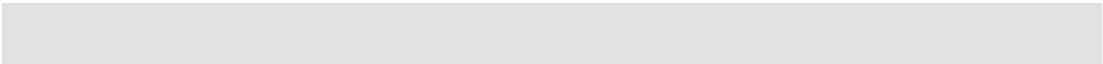
- 1 Check that the LED indicating the cycle start is off on the operator's panel.
- 2 Check that all movable parts of the CNC machine tool is stopping.
- 3 If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.
- 4 Continue to press the POWER OFF pushbutton for about 5 seconds.

NOTE

Refer to the machine tool builder's manual for turning off the power to the machine.

3

MANUAL OPERATION



MANUAL OPERATION are five kinds as follows :

- 3.1 Manual reference position return**
- 3.2 Jog feed**
- 3.3 Incremental feed**
- 3.4 Manual handle feed**
- 3.5 Manual absolute on and off**

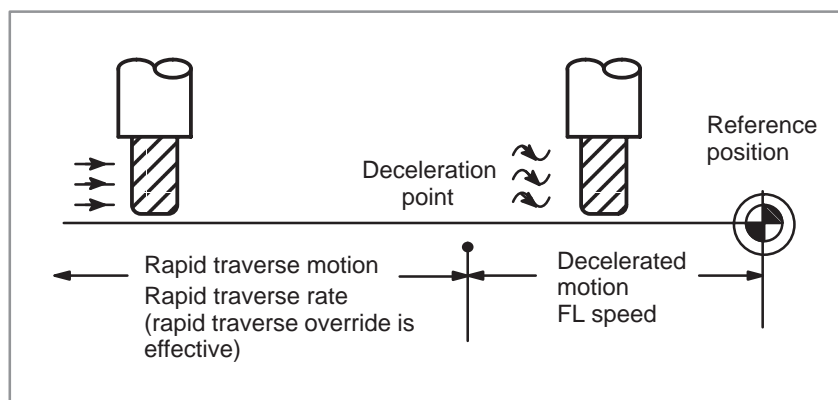
3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :

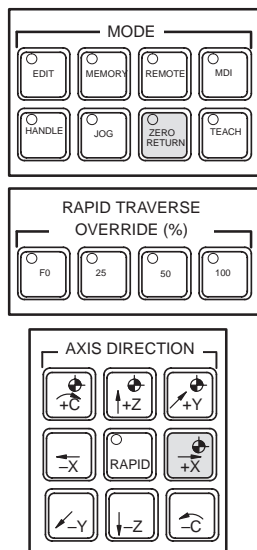
The tool is moved in the direction specified in parameter (bit 0 to #3 of No. 003) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 518 to 521, 533, and 534).

Fourstep 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 parameter (bit 4 of No.049).



Procedure for Manual Reference Position Return



- 1 Press the reference position return switch, one of the mode selection switches.
- 2 To decrease the feedrate, press a rapid traverse override switch. When the tool has returned to the reference position, the reference position return completion LED goes on.
- 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.
- 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.

[illegible]

Explanations

- **Automatic coordinate system setting**

If the parameter for automatic coordinate system setting (bit 7 of parameter 010) is specified, the coordinate system is determined automatically when a manual reference position return is made. If α , β , γ , and δ are specified in parameters 708 to 711, the system specifies a workpiece coordinate system such that the tip of the basic tool or a reference position on the tool holder, after reference position return, becomes $X = \alpha$, $Y = \beta$, $Z = \gamma$, and 4th = δ .

Restrictions

- **Moving the tool again**

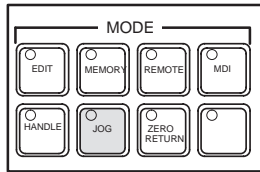
Once the REFERENCE POSITION RETURN COMPLETION LED lights at the completion of reference position return, the tool does not move unless the REFERENCE POSITION RETURN switch is turned off.
- **Reference position return completion LED**

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



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 Table 3.2.

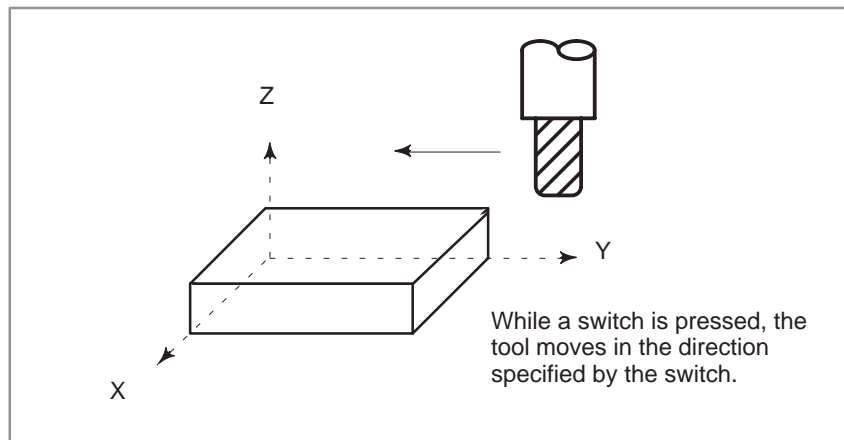
Table 3.2 Jog Feedrate

Rotary switch position	Feedrate		Rotary switch position	Feedrate	
	Metric input (mm/min)	Inch input (inch/min)		Metric input (mm/min)	Inch input (inch/min)
0	0	0	8	50	2.0
1	2.0	0.08	9	79	3.0
2	3.2	0.12	10	126	5.0
3	5.0	0.2	11	200	8.0
4	7.9	0.3	12	320	12
5	12.6	0.5	13	500	20
6	20	0.8	14	790	30
7	32	1.2	15	1260	50

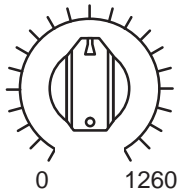
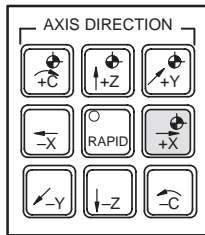
NOTE

The feedrate error (about $\pm 3\%$) affects on the feedrate in the table above.

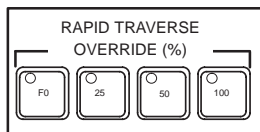
The jog feedrate can be adjusted with the jog feedrate override dial. Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate regardless of the position of the jog feedrate override dial. Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.049#4).



Procedure for Jog Feed



JOG FEED RATE OVERRIDE



- 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 direction the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in Table 3.2. 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.

Restrictions

- **Acceleration/deceleration for rapid traverse**
- **Change of modes**
- **Rapid traverse prior to reference position return**

Feedrate, time constant and method of automatic acceleration/deceleration for manual rapid traverse are the same as G00 in programmed command.

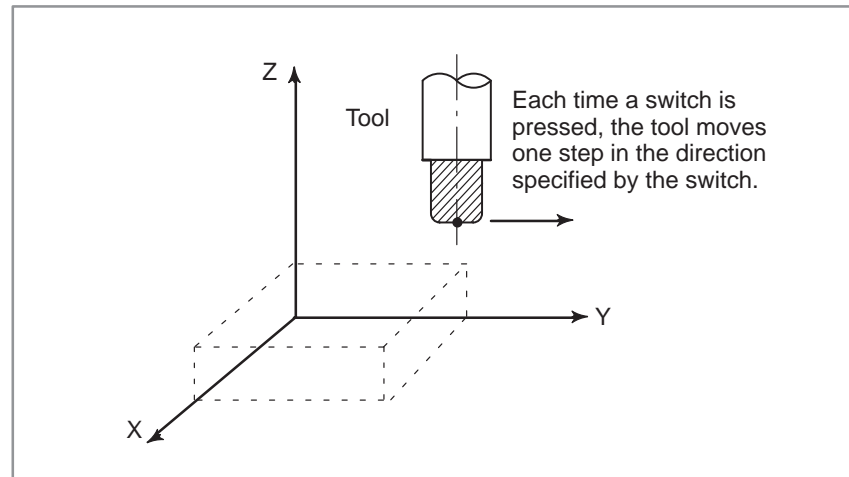
Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

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 parameter (No.010 #0).

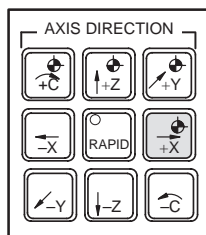
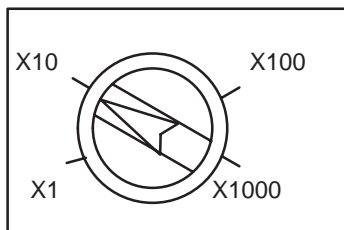
3.3 INCREMENTAL FEED

In the incremental (STEP) 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.

This mode is effective when a manual pulse generator is not connected.



Procedure for Incremental Feed



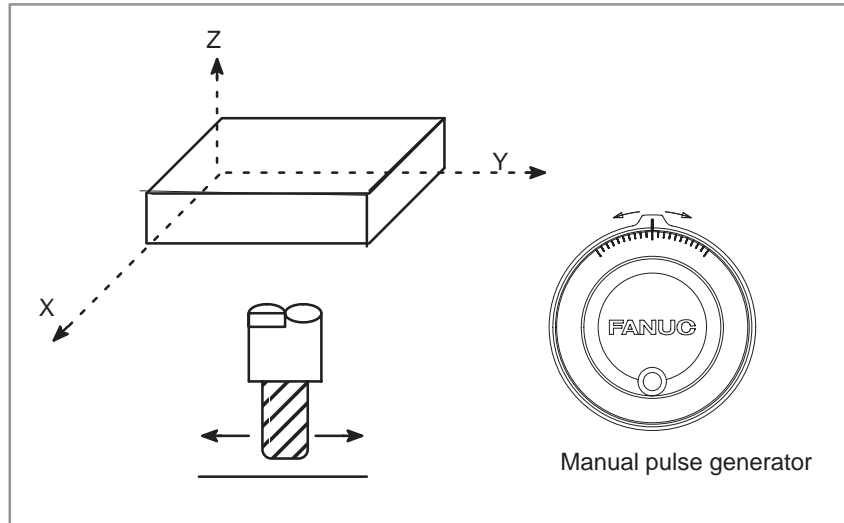
- 1 Press the STEP switch, one of the mode selection switches.
- 2 Select the distance to be moved for each step with the magnification dial.
- 3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.
- 4 If an axis direction selection switch is pressed after rapid traverse has been selected, movement is made 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.

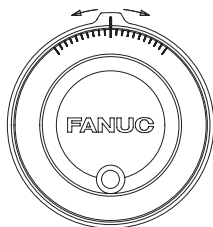
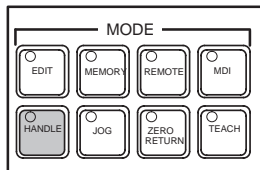
3.4 MANUAL HANDLE FEED

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. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (No. 121 and 699).



Procedure for Manual Handle Feed



Manual pulse generator

- 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.

Explanations

- **Availability of manual pulse generator in Jog mode**
- **Availability of manual pulse generator in TEACH IN JOG mode**
- **A command to the MPG exceeding rapid traverse rate**

Parameter (bit 0 of No. 013) enables or disables the manual handle feed in the JOG mode.

When the parameter (bit 0 of No. 013) is set 1, both manual handle feed and incremental feed are enabled.

Parameter (bit 6 of No. 002) enables or disables the manual handle feed in the TEACH IN JOG mode.

Parameter (bit 4 of No. 060) specifies as follows:

SET VALUE 0 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored. (The distance the tool is moved may not match the graduations on the manual pulse generator.)

SET VALUE 1 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC.
(No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

Restrictions

- **Number of MPGs**

Up to three manual pulse generators can be connected, one for each axis. The three manual pulse generators can be simultaneously operated.

WARNING

Rotating the handle quickly with a large magnification such as x100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

NOTE

Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

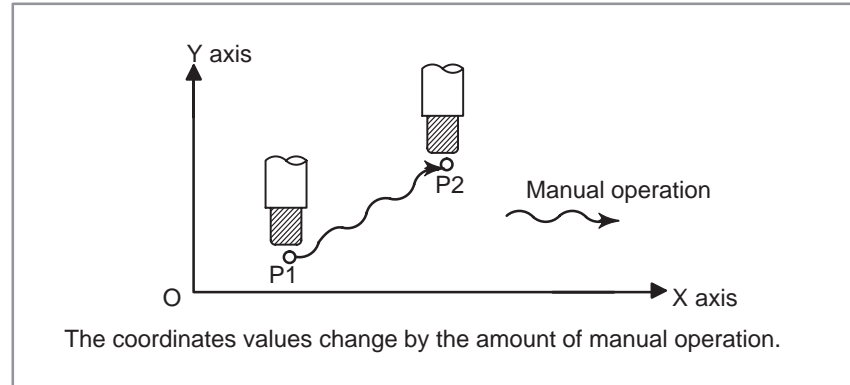


Fig. 3.5 (a) Coordinates with the switch ON

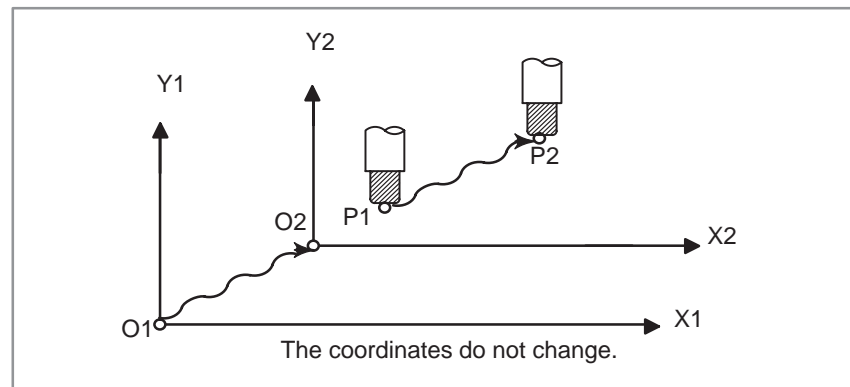


Fig. 3.5 (b) Coordinates with the switch OFF

Explanations

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

G01G90 X100.0Y100.0F010;	1
X200.0Y150.0	2
X300.0Y200.0	3

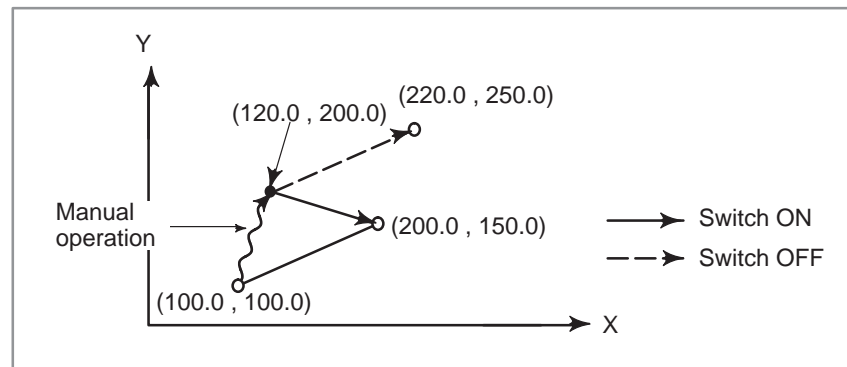
The subsequent figures use the following notation:

- Movement of the tool when the switch is on
- -→ Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

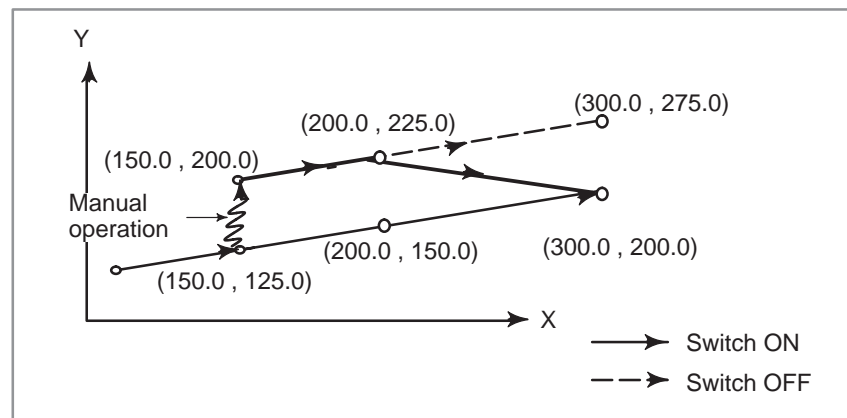
- **Manual operation after the end of block**

Coordinates when block 2 has been executed after manual operation (X-axis +20.0, Y-axis +100.0) at the end of movement of block 1.



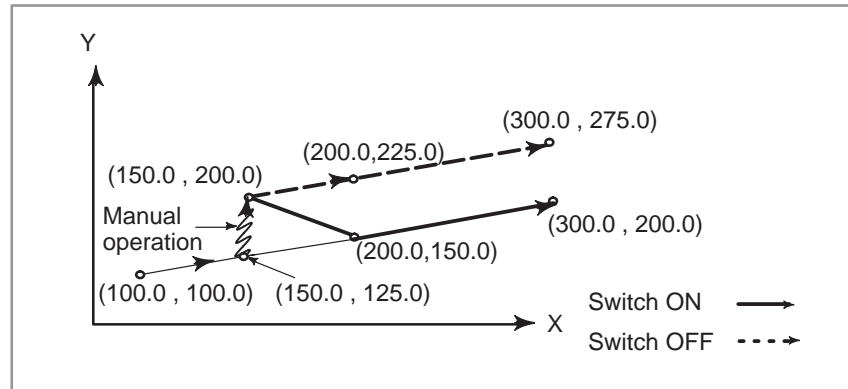
- **Manual operation after a feed hold**

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.



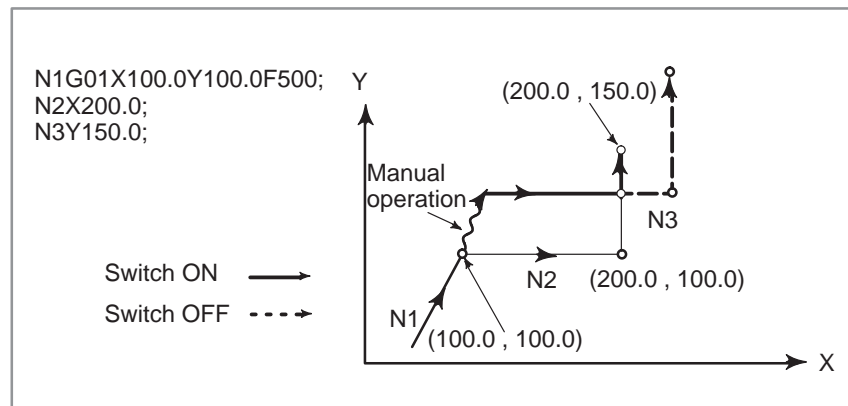
- When reset after a manual operation following a feed hold

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



- 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.

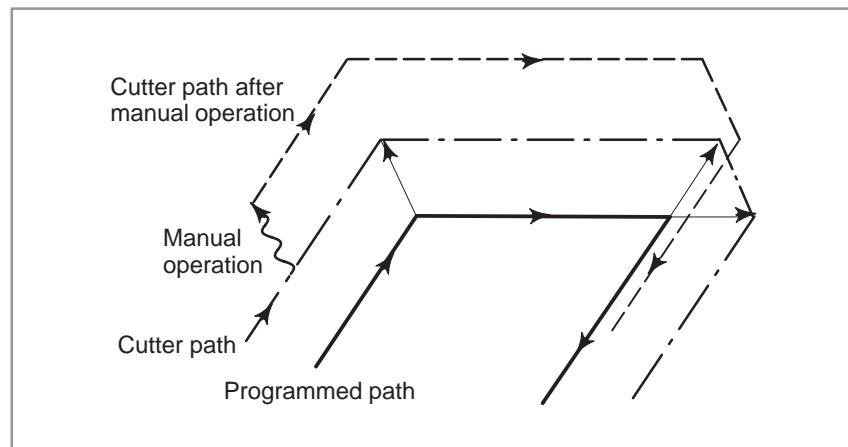


- When the next move block is an incremental
- Manual operation during cutter compensation

When the following commands are incremental commands, operation is the same as when the switch is OFF.

When the switch is OFF

After manual operation is performed with the switch OFF during cutter compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed. The amount of separation equals to the amount that was performed manually.

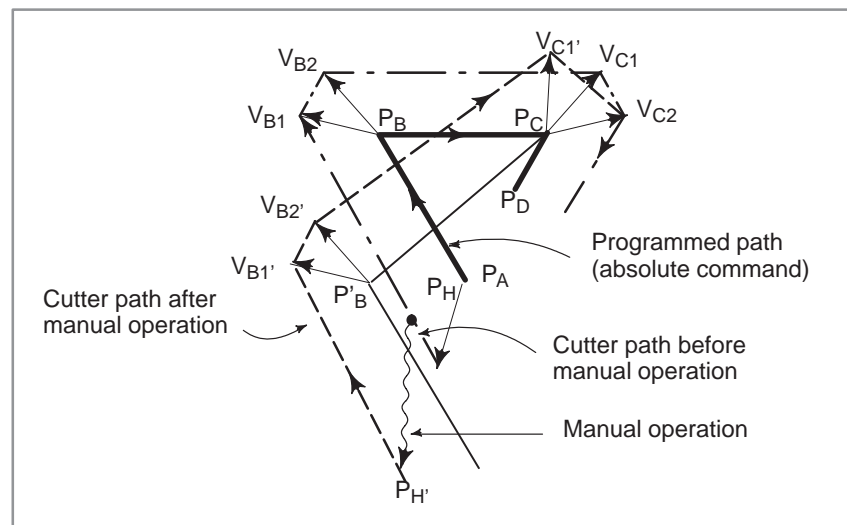


When the switch is ON during cutter compensation

Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the cutter 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.

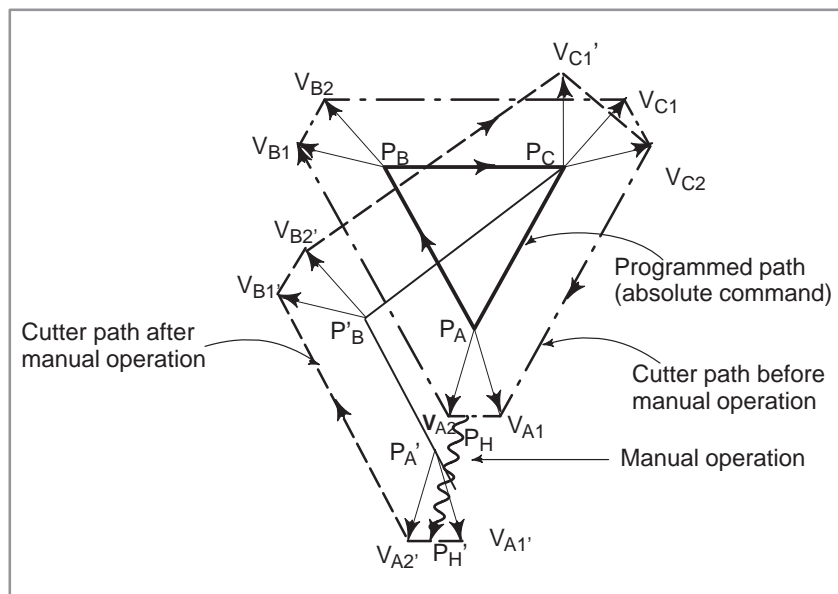
Manual operation performed in other than cornering

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

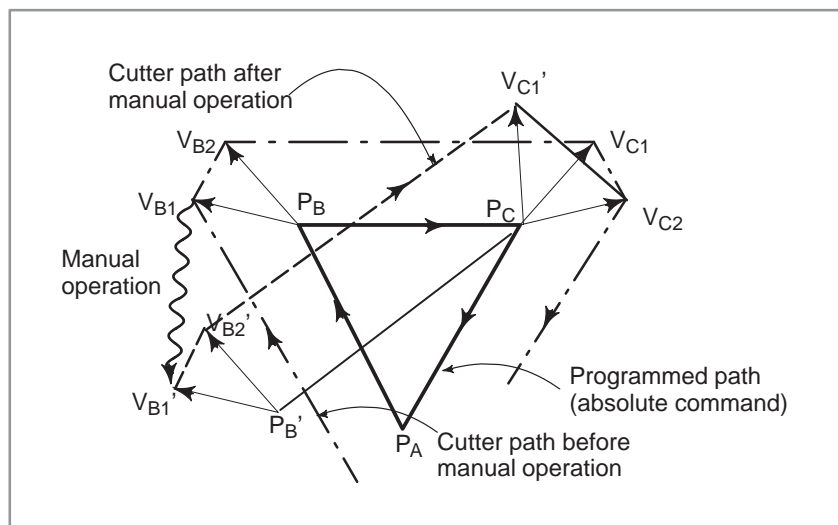
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2} , V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2} . Then correct cutter compensation is performed 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. Subsequent processing is the same as case a described above. An MDI operation can also be interveneted as well as manual operation. The movement is the same as that by manual operation.



4

AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.

This chapter explains the following types of automatic operation:

- **MEMORY OPERATION**

Operation by executing a program registered in CNC memory

- **MDI OPERATION**

Operation by executing a program entered from the MDI panel

- **DNC OPERATION**

Function for operating a machine while reading a program from an input/output unit

- **PROGRAM RESTART**

Restarting a program for automatic operation from an intermediate point

- **SCHEDULING FUNCTION**

Scheduled operation by executing programs (files) registered in an external input/output device (Handy File, Floppy Cassette, or FA Card)

- **SUBPROGRAM CALL FUNCTION**

Function for calling and executing subprograms (files) registered in an external input/output device (Handy File, Floppy Cassette, or FA Card) during memory operation

- **MANUAL HANDLE INTERRUPTION**

Function for performing manual feed during movement executed by automatic operation

- **MIRROR IMAGE**

Function for enabling mirror-image movement along an axis during automatic operation

- **SEQUENCE NUMBER SEARCH**

This function is usually used to search for a sequence number within a program and to start or resume the program from the block having that sequence number.

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 lamp 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 reset switch on the CRT/MDI panel 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.

Procedure for Memory Operation

Procedure

- 1 Press the **AUTO** mode selection switch.
- 2 Select a program from the registered programs. To do this, follow the steps below.
 - 2-1 Press  to display the program screen.
 - 2-2 Press address .
 - 2-3 Enter a program number using the numeric keys.
 - 2-4 Press the cursor key .
- 3 Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start lamp goes on. When automatic operation terminates, the cycle start lamp 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 lamp goes on and the cycle start lamp 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) The current operation, instigated by executing by an M, S, or T command, is continued.

b. Terminating memory operation

Press the  key on the CRT/MDI panel.

Automatic operation is terminated and the reset state is entered.

When a reset is applied during movement, movement decelerates then stops.

Explanations

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 CRT/MDI panel or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

• Optional block skip

When the optional block skip switch on the machine operator's panel is turned on, blocks containing a slash (/) are ignored.

Calling a subprogram stored in an external input/output device

A file (subprogram) in an external input/output device such as a Floppy Cassette can be called and executed during memory operation. For details, see Section 4.5.

4.2 MDI OPERATION

In the **MDI** mode, a program can be inputted in the same format as normal programs and executed from the MDI panel.


MDI operation is used for simple test operations.

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.





Procedure for MDI Operation – A

Example of X10.5 Y200.5;

One command block can be entered from the CRT/MDI for execution.

- 1 Press MDI key on the mode select switch.
- 2 Press the  button.
- 3 Press soft key [**MDI**] to display a screen with MDI at the top left.

PROGRAM (MDI)	(MODAL)	O9501 N9501
	G67 G00 F	
	G97 G17 R	
	G54 G90 P	
	G64 G22 Q	
	G69 G94 H	D
	G15 G21 M	
	G25 G40 S	
	G49 T	
	G80 B	
	G98	
	G50	
ADRS.	S	0 T
10:29:38	MDI	
[PRGRM]	[CURRNT]	[NEXT] [MDI] [RSTR]

- 4 Input "X 10.5" by address/numeric key.
- 5 Press  key.
The data, X 10.5, is input and displayed. If you are aware of an error in the keyed-in number before pressing the  key, press the  key and key in X and the correct number again.
- 6 Input "Y 200.5" by address/numeric key.
- 7 Press  key.
The data, Y200.5 is input and displayed. If you pressed wrong number keys, correct the operation following the instruction described above.

```

PROGRAM                                O9501 N9501
(MDI)                                (MODAL)
X      10.500 G67 G00 F
Y      200.500 G54 G17 R
                                G64 G90 P
                                G69 G22 Q
                                G15 G94 H
                                G25 G21 M
                                G40 S
                                G49 T
                                G80 B
                                G98
                                G50
ADDRS.                                S      0 T
10:30:25                                MDI
[ PRGRM ][ CURRNT ][ NEXT ] [ MDI ] [ RSTE ]





```

8 Press the  key.

Press the cycle start button on the machine operator's panel (depending on the machine tool).

Cancel before pressing the START button

To modify X10.5Y200.5 to X10.5, cancel Y200.5, by following the steps described below:

- 1 Press    keys.
- 2 Press the  or the cycle start button on the machine operator's panel.

WARNING


Modal G codes cannot be cancelled. Enter the correct data again.

Limitations

- A single MDI operation executes a single input block. Two or more blocks cannot be executed at one time.
- The end-of-block symbol (;) need not be entered.
- A subprogram call or macro call cannot be specified.
- In MDI operation, the setting (absolute) determines whether the commands are absolute or incremental. The G code (G90/G91) is ignored.
- The input block is cleared when the MDI operation is completed or when a reset is specified.

Procedure for MDI Operation – B

Procedure

- 1 Press the **MDI** mode selection switch.
- 2 Press the  function key on the CRT/MDI panel to select the program screen. The following screen appears:




```

PROGRAM ( MDI )                                O1234 N5678
O0000
%

G00  G90  G94  G40  G80  G50  G54  G69
G17  G22  G21  G49  G98  G67  G64
F      P      H      S
R      Q      M      T
<      S      0 T
11:26:11      MDI
[ PRGRM ][ CURRNT ][ NEXT ] [ MDI ] [ RSTR ]

```

Program number O0000 is entered 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. For program editing, see Chapter 9.
- 4 To entirely erase a program created in MDI mode, use one of the following methods:
 - a. Enter address , then press the  key on the MDI panel.
 - b. Alternatively, press the  key. In this case, set bit 7 of parameter 057 to 1 in advance.

- 5 To execute a program, set the cursor on the head of the program. (Start from an intermediate point is possible.) 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.

```

PROGRAM ( MDI )                                01234 N5678

O0000 G00 X100. Y200. ;
M03 ;
G01 Z120. F500 ;
M98 P9010 ;
G00 Z0 ;
%

(MODAL)
G00 G90 G94 G40 G80 G50 G54 G69
G17 G22 G21 G49 G98 G67 G64
F          P          H          S
R          Q          M          T
<          S          0 T
11:04:44          MDI
[ PRGRM ][ CURRNT ][ NEXT ] [ MDI ] [ RSTR ]

```

- 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 lamp goes on and the cycle start lamp goes off.

When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b. Terminating MDI operation

Press the  key on the CRT/MDI panel.

Automatic operation is terminated and the reset state is entered.

When a reset is applied during movement, movement decelerates then stops.

Explanations

• Erasing the program

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation –B, M30 does not return control to the beginning of the program (M99 performs this function).

Programs prepared in the **MDI** operation will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
- In **AUTO** mode, if memory operation is performed.
- In **EDIT** mode, if any editing is performed.
- Background editing is performed.

• Restart

After the editing operation during the stop of MDI operation was done, operation starts from the current cursor position.

- **Absolute/incremental command**

The setting (absolute) determines whether commands are absolute or incremental. If bit 5 of parameter 029 is set to 1, G90/G91 in the program is enabled. To call a subprogram or macro program during MDI operation, set bit 5 of parameter 029 to 1.

Explanations

- **Program registration**

Programs created in MDI mode cannot be registered.

- **Number of lines in a program**

A program can have as many lines as can fit on one page of the CRT screen.

A program consisting of up to six lines can be created. When parameter (No. 028 #3) is set to 0 to specify a mode that suppresses the display of continuous-state information, a program of up to 10 lines can be created. If the created program exceeds the specified number of lines, % (ER) is deleted (prevents insertion and modification).

- **Subprogram nesting**

Calls to subprograms (M98) can be specified in a program created in the MDI mode. This means that a program registered in memory can be called and executed during MDI operation. In addition to the main program executed by automatic operation, up to two levels of subprogram nesting are allowed (when the custom macro option is provided, up to four levels are allowed).

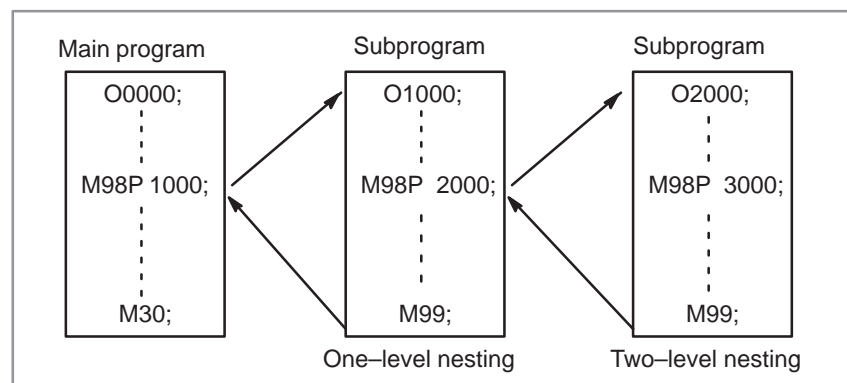


Fig. 4.2 Nesting level of subprograms called from the MDI program

- **Macro call**

When the custom macro option is provided, macro programs can also be created, called, and executed in the **MDI** mode. However, macro call commands cannot be executed when the mode is changed to **MDI** mode after memory operation is stopped during execution of a subprogram.

- **Memory area**

When a program is created in the **MDI** mode, an empty area in program memory is used. If the maximum number of programs has already been registered, or if the part program storage is full, a program cannot be created in **MDI** mode.

4.3 DNC OPERATION

In DNC operation, the machine is not operated by a program registered in memory of the CNC, instead being operated by a program read directly from a connected input/output unit. This mode is used when the program is too large to be registered in the memory of the CNC. This mode is also used for high-speed machining using a remote buffer.

For DNC operation with the Floppy Cassette directory display function (the Floppy Cassette directory display function is supported and the Floppy Cassette is specified as the input/output unit), see Chapter 8.

Procedure for DNC Operation

- 1 Select the MDI mode and specify the channel for the connected input/output unit in the I/O field on the setting screen.
- 2 Select AUTO mode.
- 3 Search for the beginning of the program of the input/output unit.
- 4 Input the DNCI signal. (For details of the actual operation, refer to the manual provided by the machine tool builder.)
- 5 Press the cycle start key.

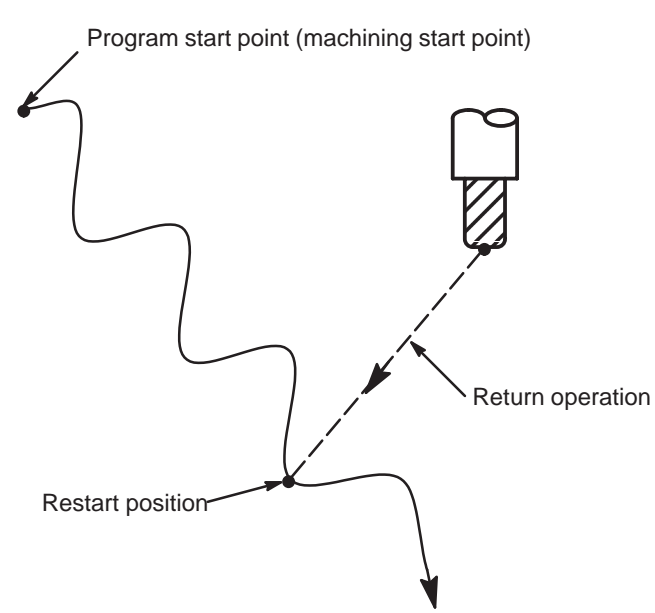
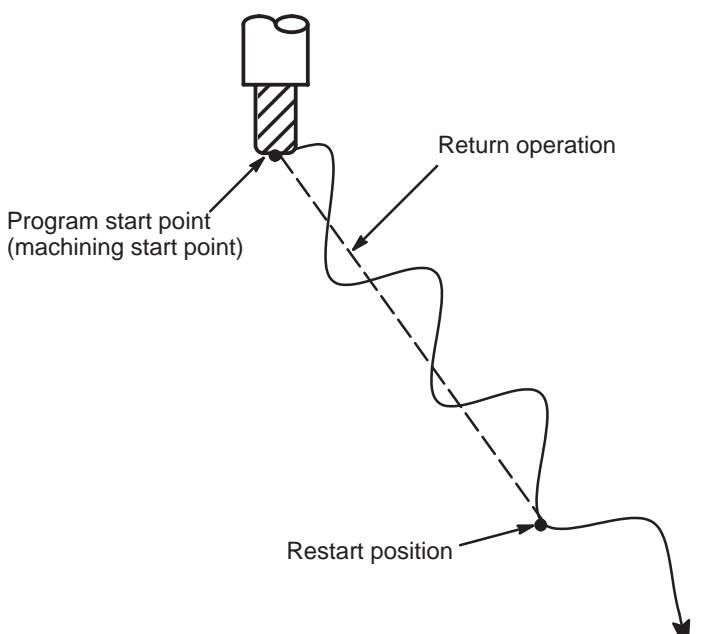
DNC operation starts. The operation can be stopped and resumed in the same way as for memory operation.

Explanations

- In DNC operation mode, the current program can call a subprogram registered in memory.
- In DNC operation mode, the current program can specify a custom macro. However, a repeat instruction or branch instruction cannot be programmed.
- In DNC operation mode, the current program can call a macro program registered in memory.
- In DNC operation mode, to return the system from the current subprogram or macro program to the calling program, a sequence number (M99P****) cannot be specified.
- If input/output channels 1 and 2 are used for DNC operation, the DC3 code is output and reading is stopped at the end of each block (each time the EOB is read). To enable continuous reading of the blocks, specify bit 7 of parameter 0390 accordingly.
- In DNC operation mode, to allow a program to be resumed specify bit 7 of parameter 0387 accordingly.
- In DNC operation mode, no program can be displayed. Only the current block and subsequent block can be displayed.
- In DNC operation mode, F is displayed as the address of the program number at the top right corner of the CRT screen.

4.4
PROGRAM RESTART

This function specifies Sequence No. of a block to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high-speed program check function. There are two restart methods: the P-type method and Q-type method.

P TYPE	Operation can be restarted anywhere. This restart method is used when operation is stopped because of a broken tool.
 <p>The diagram illustrates the P-type restart method. It shows a tool (represented by a vertical rectangle with a hatched section) moving along a path. The path starts at a point labeled 'Program start point (machining start point)'. The tool moves along a solid line, then a dashed line, and finally a solid line. At a point labeled 'Restart position', the tool stops. A dashed line labeled 'Return operation' shows the tool moving back to the 'Program start point (machining start point)'. After the return operation, the tool resumes its path from the 'Restart position'.</p>	
Q TYPE	Before operation can be restarted, the machine must be moved to the programmed start point (machining start point)
 <p>The diagram illustrates the Q-type restart method. It shows a tool (represented by a vertical rectangle with a hatched section) moving along a path. The path starts at a point labeled 'Program start point (machining start point)'. The tool moves along a solid line, then a dashed line, and finally a solid line. At a point labeled 'Restart position', the tool stops. A dashed line labeled 'Return operation' shows the tool moving back to the 'Program start point (machining start point)'. After the return operation, the tool resumes its path from the 'Restart position'.</p>	

Procedure for Program restart

Procedure 1




[P TYPE]

- 1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

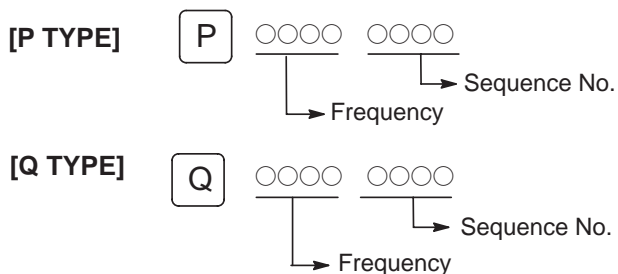
[Q TYPE]

- 1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
- 2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
- 3 If necessary, modify the offset amount.

Procedure 2

- 1 Turn the program restart switch on the machine operator's panel ON.
- 2 Press  on the CRT/MDI panel to display the desired program.
- 3 Find the program head.
- 4 **[P TYPE]** Input address P, then the sequence number of the block to be resumed. Press the cursor down  key.
[Q TYPE] Input address Q, then the sequence number of the block to be resumed. Press the cursor down  key.

If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number.



- 5 Once the search to resume the program has been completed, the CRT screen displays the program resume screen.

```

PROGRAM RESTART                                00100 N0123

(DESTINATION)  M010 015 050 *** ***
X   300.000    *** *** *** *** ***
Y   300.000    *** *** *** *** ***
Z   300.000    *** *** *** *** ***
                *** *** *** *** ***
                *** *** *** *** ***
(DISTANCE TO GO) *** *** *** *** ***
1 X   254.052
2 Y   243.988  T   ****
3 Z   249.368  S   ****
                B   ****
                                S   0 T
14:08:12          BUF AUTO          RSTR
[ PRGRM ][ CURRNT ][ NEXT ][ MDI ][ RSTR ]

```

DESTINATION shows the position at which machining is to restart. DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter No. 124 to 127) along which the tool moves to the restart position.

M: Thirty-five most recently specified M codes

T: Two most recently specified T codes

S: Most recently specified S code

B: Most recently specified B code

- 6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.
- 7 Check the screen for the M, S, T, and B codes to be executed. If they are found, execute the M, S, T, and B functions by the **MDI** mode. After execution, restore the previous mode.
- 8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.
- 9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings. Machining is then restarted.

Restrictions

- **P-type restart**

Under any of the following conditions, P-type restart cannot be performed:

 - When automatic operation has not been performed since the power was turned on
 - When automatic operation has not been performed since an emergency stop was released
 - When automatic operation has not been performed since the coordinate system was changed or shifted (change in an external offset from the workpiece reference point)
- **Restart block**

The block to be restarted need not be the block which was interrupted; operation can restart with any block. When P-type restart is performed, the restart block must use the same coordinate system as when operation was interrupted.
- **Single block**

When single block operation is ON during movement to the restart position, operation stops every time the tool completes movement along an axis. When operation is stopped in the single block mode, MDI intervention cannot be performed.
- **Manual intervention**

During movement to the restart position, manual intervention can be used to perform a return operation for an axis if it has not yet been performed for the axis. A return operation cannot be done further on axes for which a return has already been completed.
- **Reset**

Never reset during the time from the start of a search at restart until machining is restarted. Otherwise, restart must be performed again from the first step.
- **Manual absolute**

Regardless of whether machining has started or not, manual operation must be performed when the manual absolute mode is on.
- **Reference position return**

If no absolute-position detector (absolute pulse coder) is provided, be sure to perform reference position return after turning on the power and before performing restart.

WARNING

As a rule, the tool cannot be returned to a correct position under the following conditions. Special care must be taken in the following cases since none of them cause an alarm:

- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked. (ignore the Z-axis)
- When the mirror image is used.
- When manual operation is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.

4.5 SCHEDULING FUNCTION

The schedule function allows the operator to select files (programs) registered on a floppy-disk in an external input/output device (Handy File, Floppy Cassette, or FA Card) and specify the execution order and number of repetitions (scheduling) for performing automatic operation. It is also possible to select only one file from the files in the external input/output device and execute it during automatic operation.

This function is enabled if the Floppy Cassette directory display function is supported and the Floppy Cassette is specified as the input/output unit.

FILE DIRECTORY	
FILE NO.	FILE NAME
0001	O0010
0002	O0020
0003	O0030
0004	O0040

List of files in an external input/output device



Set file number and
number of repetitions.

ORDER	FILE NO	REPETITION
01	0002	2
02	0003	1
03	0004	3
04	0001	2

Scheduling screen




Executing automatic operation

Procedure for Scheduling Function


Procedure

• Procedure for executing one file

- 1 Press the **AUTO** switch on the machine operator's panel, then press the  function key on the MDI panel.
- 2 Press the rightmost soft key (continuous menu key), then press the **[FL. SDL]** soft key. A list of files registered in the Floppy Cassette is displayed on screen No. 1. To display more files that are not displayed on this screen, press the page key on the MDI panel. Files registered in the Floppy Cassette can also be displayed successively.

FILE DIRECTORY		01200 N1200	
CURRENT SELECTED : SCHEDULE			
NO.	FILE NAME	(METER)	VOL
0000	SCHEDULE		
0001	PARAMETER	65.6	
0002	ALL PROGRAM	1.9	
0003	00001	1.3	
0004	00002	1.3	
0005	00003	1.3	
0006	00004	1.3	
0007	00005	1.3	
0008	00100	1.9	
14:33:53		S	0 T
[SELECT]		AUTO	RSTR
[]		[]	[SCHDUL]

Screen No.1

- 3 Press the **[SELECT]** soft keys to display "SELECT FILE NO." (on screen No. 2). Enter a file number, then press the  key and the **[EXEC]** soft key. The file for the entered file number is selected, and the file name is indicated after "CURRENT SELECTED:".

FILE DIRECTORY		O0000 N0000	
CURRENT SELECTED:O0005			
NO.	FILE NAME	(METER)	VOL
0000	SCHEDULE		
0001	PARAMETER	65.6	
0002	ALL.PROGRAM	1.9	
0003	O0001	1.3	
0004	O0002	1.3	
0005	O0003	1.3	
0006	O0004	1.3	
0007	O0005	1.3	
0008	O0100	1.9	
SELECT FILE NO.=		7	
NUM.		S	0 T
14:46:01		AUTO	
[EXEC]	[]	[]	[[SCHDUL]

Screen No.2

- 4 Press the DNC operation switch on the machine operator's panel, then press the cycle start switch. The selected file is executed. For details on the DNC operation switch, refer to the manual supplied by the machine tool builder. The selected file number is indicated at the upper right corner of the screen as an F number (instead of an O number).

FILE DIRECTORY		F0007 N1200	
CURRENT SELECTED:00005			
15:00:16	BUF AUTO	S O T	
[SELECT]	[]	[]	[[SCHEDUL]]

Screen No.3

- Procedure for executing the scheduling function

- 1 Display the list of files registered in the Floppy Cassette. The display procedure is the same as in steps 1 and 2 for executing one file.
- 2 On screen No. 2, press the **[SELECT]** soft key to display "SELECT FILE NO."
- 3 Enter file number 0, and press the **[EXEC]** soft key. "SCHEDULE" is indicated after "CURRENT SELECTED:".
- 4 Press the leftmost soft key (return menu key) and the **[SCHEDUL]** soft key. Screen No. 4 appears.

SCHEDULE		F0000 N1200	
ORDER	FILE NO.	REQ.REP	CUR.REP
01	—		
02			
03			
04			
05			
06			
07			
08			
09			
10			
NUM.		S O T	
15:01:17		AUTO	
[]	[]	[CLEAR]	[DIR]

Screen No.4

Move the cursor and enter the file numbers and number of repetitions in the order in which to execute the files. At this time, the current number of repetitions "CUR.REP" is 0.

- 5 Press the DNC operation switch on the machine operator's panel, and then press the start switch. The files are executed in the specified order. When a file is being executed, the cursor is positioned at the number of that file.

The current number of repetitions CUR.REP is increased when M02 or M30 is executed in the program being run.

SCHEDULE		F0000 N0002	
ORDER	FILE NO.	REQ.REP	CUR.REP
01	0007	5	5
02	0003	23	23
03	0004	9999	22
04	0005	LOOP	0
05			
06			
07			
08			
09			
10			
NUM.		S	O T
19:31:53		AUTO	
[]	[CLEAR]	[DIR]

Screen No.5

Explanations

- **Specifying no file number**

If no file number is specified on screen No. 4 (the file number field is left blank), program execution is stopped at that point. To leave the file number field blank, press numeric key or then .

- **Endless repetition**

If a negative value is set as the number of repetitions, <LOOP> is displayed, and the file is repeated indefinitely.

- **Clear**

When the [CLEAR] and [EXEC] soft keys are pressed on screen No. 4, all data is cleared. However, these keys do not function while a file is being executed.

- **Return to the program screen**

When the leftmost soft key (return menu key) is pressed on screen No. 1, 2, 3, 4, or 5, the program screen is displayed.

Explanations

- **Number of repetitions** Up to 9999 can be specified as the number of repetitions. If 0 is set for a file, the file becomes invalid and is not executed.
- **Number of files registered** By pressing the page key on screen No. 4, up to 20 files can be registered.
- **M code** When M codes other than M02 and M30 are executed in a program, the current number of repetitions is not increased.
- **Displaying the floppy disk directory during file execution** During the execution of file, the floppy directory display of background editing cannot be referenced.
- **Restarting automatic operation** To resume automatic operation after it is suspended for scheduled operation, press the reset button.

4.6 SUBPROGRAM CALL FUNCTION

The subprogram call function is provided to call and execute subprogram files stored in an external input/output device (Handy File, FLOPPY CASSETTE, FA Card) during memory operation.

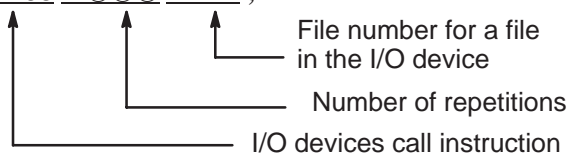
When the following block in a program in CNC memory is executed, a subprogram file in the external input/output device is called:

To execute this function, the Floppy Cassette directory display function is required.

Format

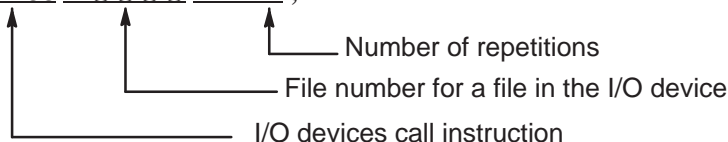
1. F0 tape format

M198 P○○○ ΔΔΔΔ ;



2. FS10/11 tape format

M198 P○○○○ LΔΔΔΔ ;



Explanations

The subprogram calling function is enabled when the Floppy Cassette (program file) is specified as the input/output unit.

When the custom macro B option is provided, either format 1 or 2 can be used. A different M code can be used for a subprogram call depending on the setting of parameter No.248. In this case, M198 is executed as a normal M code. The file number is specified at address P. If the parameter No. 063 #2 is set to 1, a program number can be specified. When a file number is specified at address P, Fxxxx is indicated instead of Oxxxx on the upper-right corner of the CRT screen.

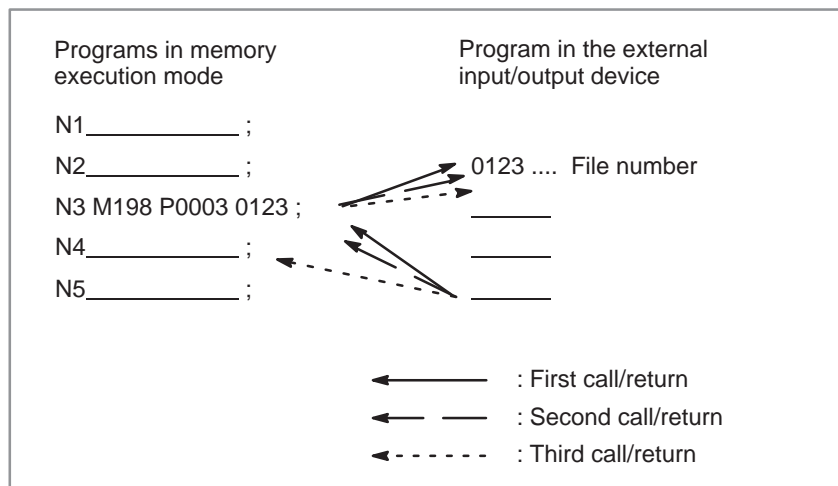


Fig. 4.6 Program flow when M198 is specified

NOTE

- 1 When M198 in the program of the file saved in a floppy cassette is executed, a P/S alarm (No.210) is given. When a program in the memory of CNC is called and M198 is executed during execution of a program of the file saved in a floppy cassette, M198 is changed to an ordinary M-code.
- 2 When MDI is intervened and M198 is executed after M198 is commanded in the AUTO mode, M198 is changed to an ordinary M-code.

4.7 MANUAL HANDLE INTERRUPTION

The movement by manual handle operation can be done by overlapping it with the movement by automatic operation in the automatic operation mode.

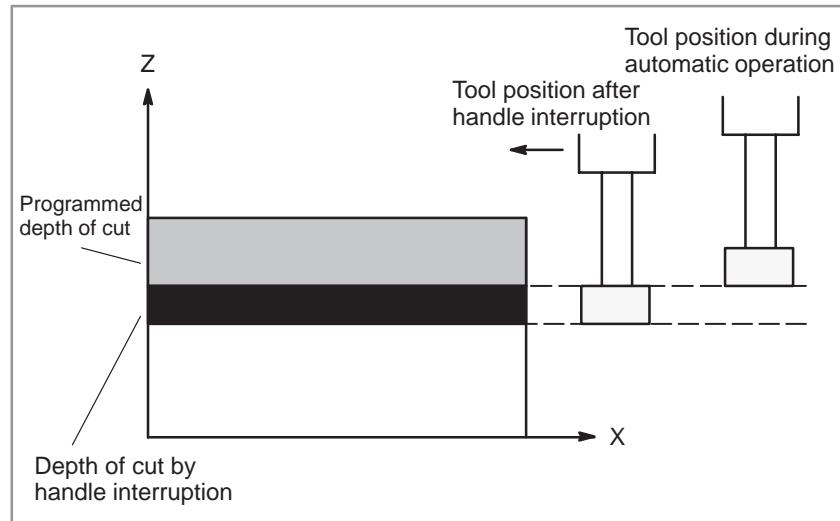


Fig. 4.7 Manual handle interruption

- Handle interruption axis selection signals
For the handle interruption axis selection signals, refer to the manual supplied by the machine tool builder.

During automatic operation, handle interruption is enabled for an axis if the handle interruption axis selection signal for that axis is on. Handle interruption is performed by turning the handle of the manual pulse generator.

WARNING

The travel distance by handle interruption is determined according to the amount by which the manual pulse generator is turned and the handle feed magnification (x1, x10, xM, xN).

Since this movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.

NOTE

The amount of travel by the handle interrupt is equal to the output unit. The amount of travel by the handle interrupt can be changed to the input unit by setting bit 2 of parameter 0075 accordingly. If this is specified, acceleration/deceleration is also applied to the amount of travel by the handle interrupt.

Explanations

• Relation with other functions

The following table indicates the relation between other functions and the movement by handle interrupt.

Signal	Relation
Machine lock	Machine lock is effective. The tool does not move even when this signal turns on.
Interlock	Interlock is effective. The tool does not move even when this signal turns on.
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 interrupt.

Display	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Handle interruption does not change relative coordinates.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

• Travel distance display

The move amount by the handle interrupt is displayed on the 4th page of the position display screen. The following 4 kinds of data are displayed concurrently.

HANDLE INTERRUPTION		O0000 N0200	
(INPUT UNIT)		(OUTPUT UNIT)	
X	69.594	X	69.594
Y	137.783	Y	137.783
Z	-61.439	Z	-61.439
(RELATIVE)		(DISTANCE TO GO)	
X	0.000	X	0.000
Y	0.000	Y	0.000
Z	0.000	Z	0.000
PART COUNT16			
RUN TIME0H 57M CYCLE TIME 0H 0M 38S			
S 0 T			
19:09:10 MDI			
[ABS]	[REL]	[ALL]	[HNDL]

- (a) INPUT UNIT : Handle interrupt move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

- (b) OUTPUT UNIT : Handle interrupt 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 interrupt move amount is cleared when the low speed reference position return (the first reference position return when power is turned on) ends every axis.

4.8 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the CRT/MDI panel.

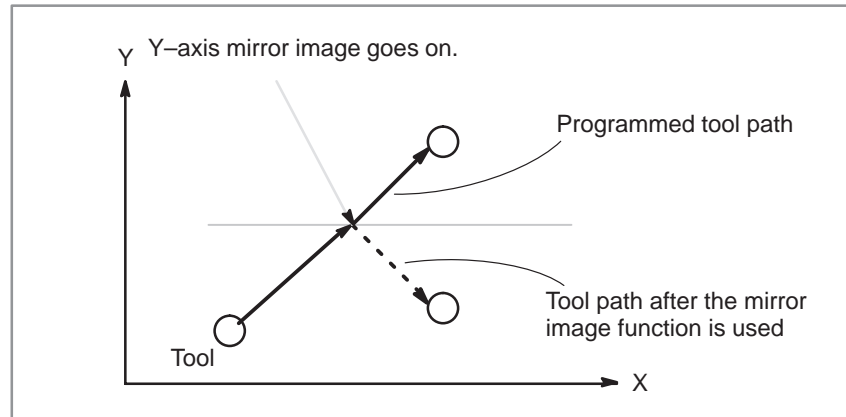


Fig. 4.8 Mirror image

Procedure for mirror image

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 DGNOS
PARAM function key.

2-3 Press the page key for chapter selection to display the setting screen.

PARAMETER

O0100 N0002

(SETTING 1)

REVX

=

0

REVV

=

0

TVON

=

0

ISO

=

1

(0:EIA

1:ISO

)

INCH

=

0

(0:MM

1:INCH)

I/O

=

0

ABS

=

0

(0:INC

1:ABS)

SEQ

=

0

CLOCK

97/07/07

19:32:00

S

0

T

NO. REVX =

19:32:51

MDI

[

PARAM

][

DGNOS

][

SV-PRM

][

]

2-4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (AUTO mode or MDI mode), then press the cycle start button to start automatic operation.

Explanations

For the mirror image switches, refer to the manual supplied by the machine tool builder.

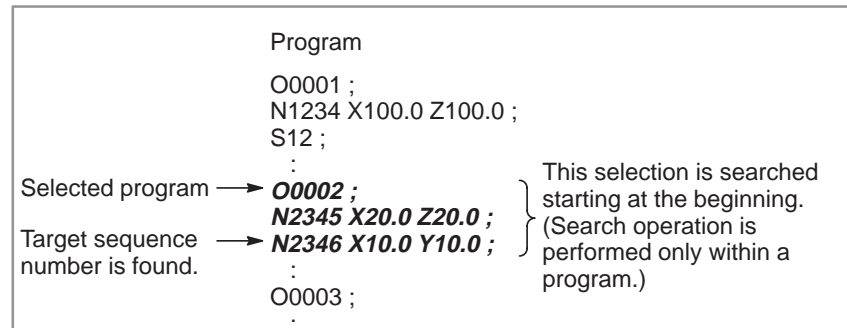
Restrictions

The direction of movement during manual operation, the direction of movement from an intermediate point to the reference position during automatic reference position return, the direction of approach during unidirectional positioning (G60), and the shift direction in a boring cycle (G76, G87) cannot be reserved.




4.9 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 2346 in a program (O0002) is searched for.



Procedure for sequence number search

- 1 Select **AUTO** mode.
- 2 Press  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 Key in address  .
- 5 Key in a sequence number to be searched for.
- 6 Press the cursor  key.
- 7 Upon completion of search operation, the sequence number searched for is displayed in the upper-right corner of the CRT screen.

Explanations

• 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, 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

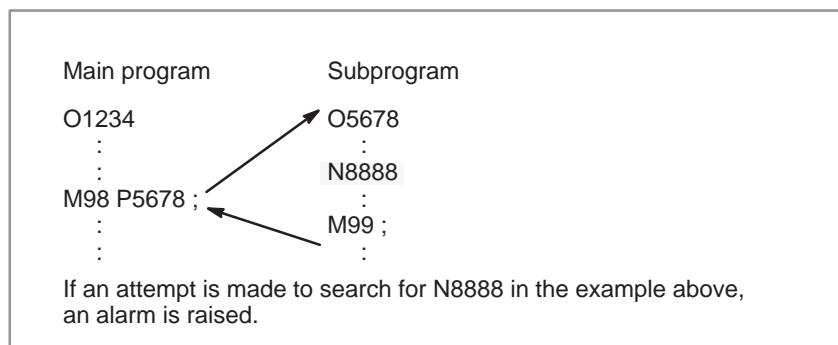
·P/S alarm (No. 003 to 010)

A P/S alarm 003 to 010 occurring during the sequence number search can be ignored if bit 1 of parameter 050 is specified accordingly.

Limitations

• Searching in sub-program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm (No. 060) is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.

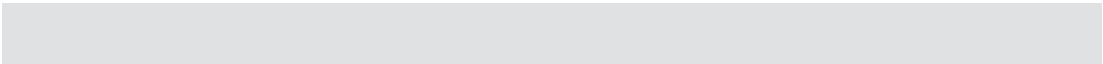


Alarm

Number	Contents
60	Command sequence number was not found in the sequence number search.

5

TEST OPERATION



The following functions are used to check before actual machining whether the machine operates as specified by the created program.

- 1. Machine Lock and Auxiliary Function Lock**
- 2. Feedrate Override**
- 3. Rapid Traverse Override**
- 4. Dry Run**
- 5. Single Block**

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 Z-axis machine lock, which stops the movement along Z-axis only. In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

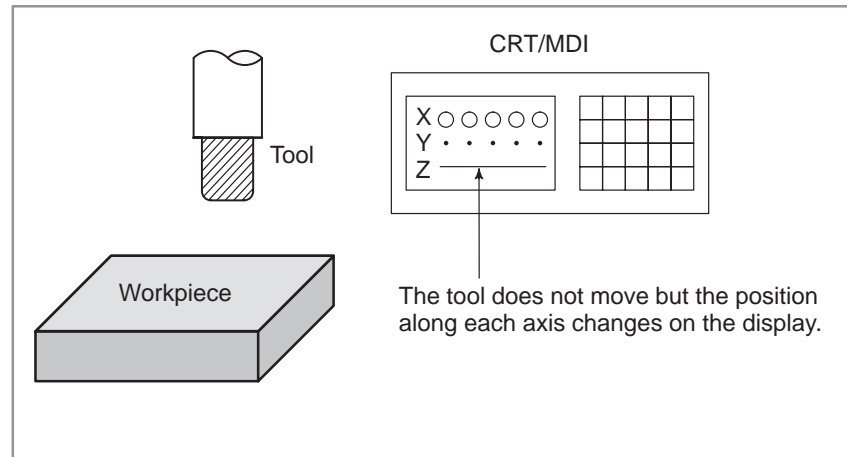


Fig. 5.1 Machine lock

Procedure for Machine Lock and Auxiliary Function Lock

• 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 Z-axis machine lock switch. Refer to the appropriate manual provided by the machine tool builder for machine lock.

The position relationship between the workpiece coordinates and machine coordinates may change after automatic operation by the machine lock function has been executed. If this occurs, reset the workpiece coordinate system by specifying the coordinate system setting command or by making a manual reference position return.

• Auxiliary Function Lock

Press the auxiliary function lock switch on the operator's panel. M, S, and T codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Explanations

• M, S, T command by only machine lock

M, S, and T commands are executed in the machine lock state.

• Reference position return under Machine Lock

When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

• M codes not locked by auxiliary function lock

M00, M01, M02, M30, M98, M99, M198, and M199 and commands are executed even in the auxiliary function lock state.

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.

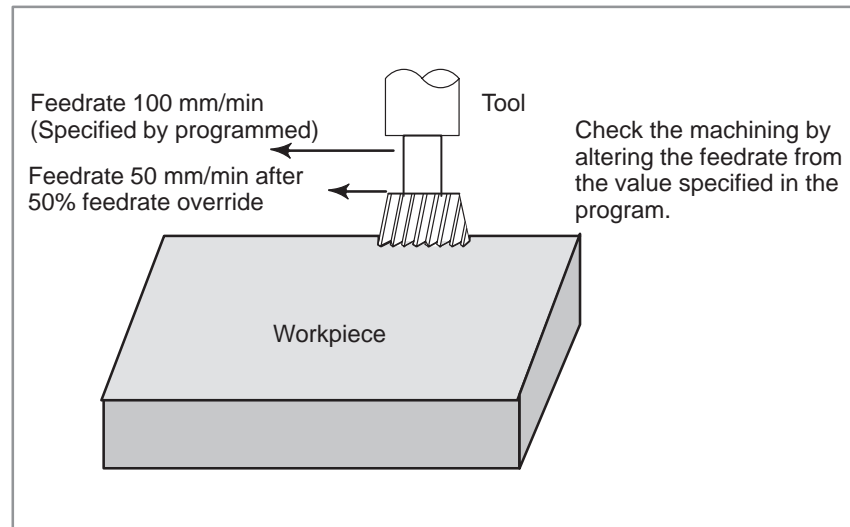
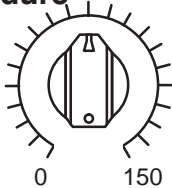


Fig. 5.2 Feedrate override

Procedure for Feedrate Override

Procedure



JOG FEED RATE OVERRIDE

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.

Explanations

- **Override Range**
- **Override during thread**

The override that can be specified ranges from 0 to 150% (10% steps). For individual machines, the range depends on the specifications of the machine tool builder.

During threading, the override is ignored and the feedrate remains as specified by program.

5.3 RAPID TRAVERSE OVERRIDE

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter (No. 533).

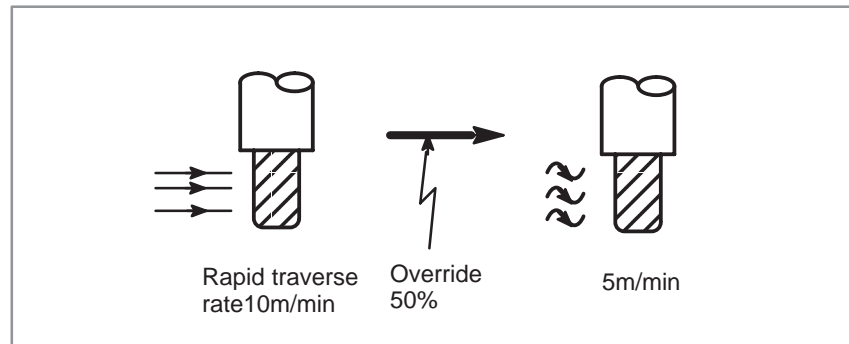
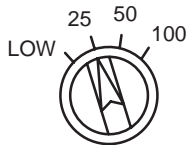


Fig. 5.3 Rapid traverse override

Rapid Traverse Override

Procedure



Rapid traverse override

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.

Explanations

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

- 1) Rapid traverse by G00
- 2) Rapid traverse during a canned cycle
- 3) Rapid traverse in G27, G28 and G30
- 4) Manual rapid traverse
- 5) Rapid traverse in manual reference position return

5.4
DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state taht the workpiece is removed from the table.

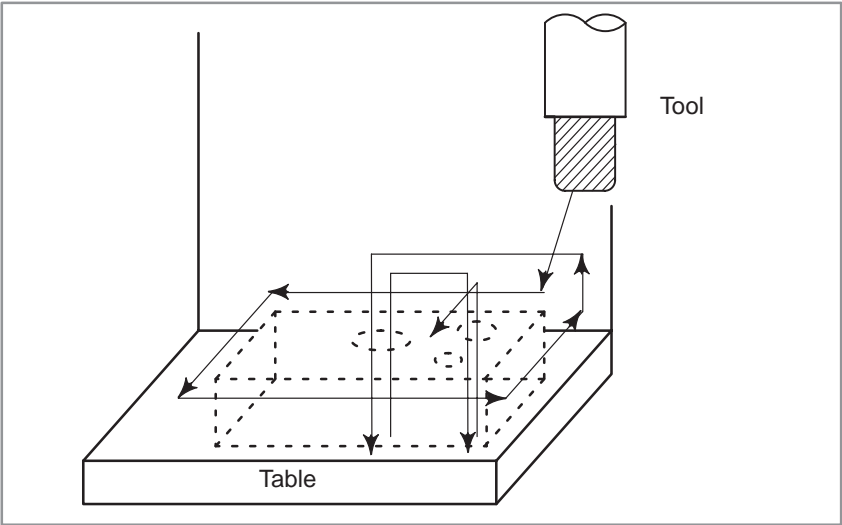


Fig. 5.4 Dry run

Procedure for 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.

Explanations

• Dry run feedrate



The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	Rapid traverse rate	Jog maximum feedrate
OFF	Jog feedrate or rapid traverse rate *1)	Jog feedrate

*1 : Jog feedrate when parameter (bit 6 of No. 001) is 1.
Rapid traverse rate when that parameter is 0.

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.

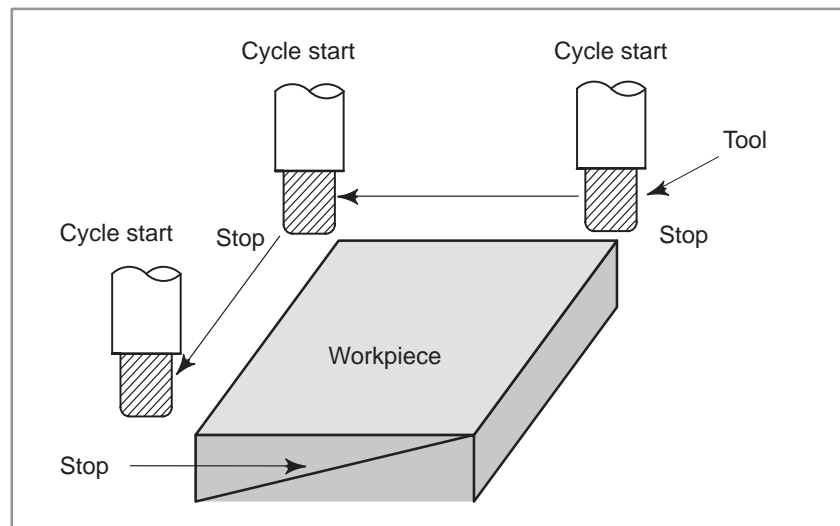


Fig. 5.5 (a) Single block

Procedure for 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.

Explanations

- **Reference position return and single block**

If G28 to G30 are issued, the single block function is effective at the intermediate point.

- **Single block during a canned cycle**

In a canned cycle, the single block stop points are the end of ,1 2, and 6 shown below. When the single block stop is made after the point 1 or 2, the feed hold LED lights.

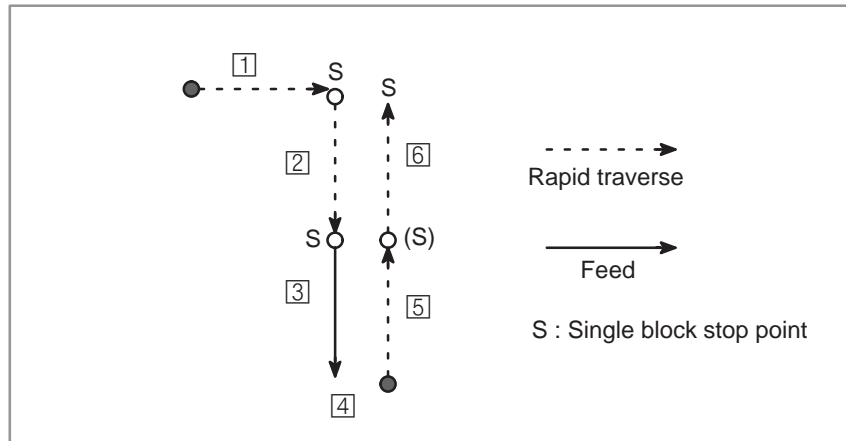


Fig. 5.5 (b) Single block during canned cycle

- **Subprogram call and single block**

Single block stop is not performed in a block containing M98P_; M99; or G65.

However, single block stop is even performed in a block with M98P_ or M99 command, if the block contains an address other than O, N or P.

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 Stroke check are available. This chapter describes emergency stop., overtravel check, and stroke check.

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

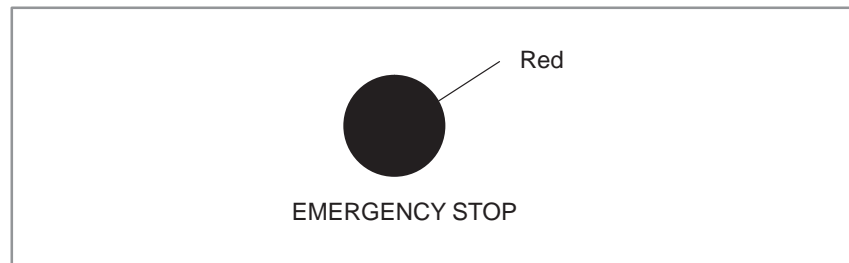


Fig. 6.1 Emergency stop

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.

Explanations

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

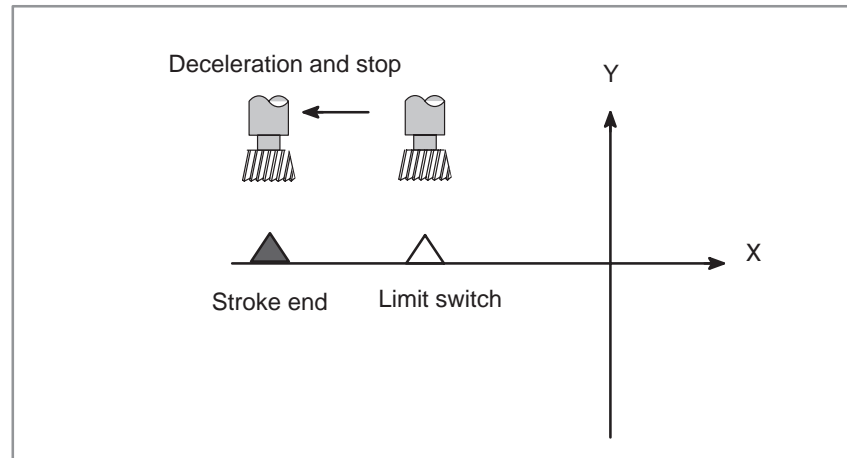


Fig. 6.2 Overtravel

Explanations

- **Overtravel during automatic operation**
- **Overtravel during manual operation**
- **Releasing overtravel**
- **Alarm**

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.

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.

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.

No.	Description
5n4	The tool has exceeded the hardware-specified overtravel limit along the positive nth axis (n : 1 to 3).
5n5	The tool has exceeded the hardware-specified overtravel limit along the negative nth axis (n : 1 to 3).

Setting bit 5 of parameter 0057 enables or disables the overtravel limit switch. For details, refer to the manual provided by the machine tool builder.

6.3

STROKE CHECK

Two areas which the tool cannot enter can be specified with stored stroke limit 1, 2 and stored stroke limit 3.

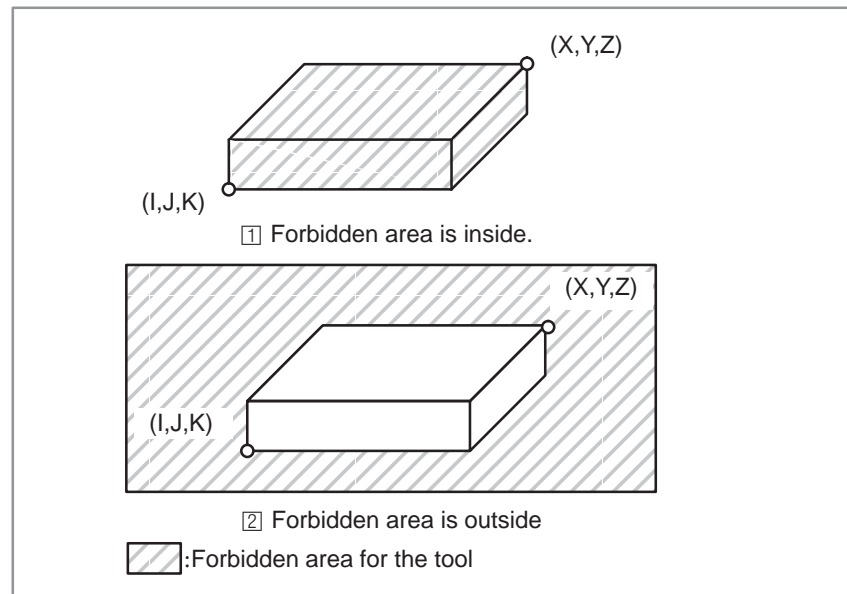


Fig. 6.3 (a) Stroke check

When the tool exceeds a stored stroke limit, 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.

Explanations

- **Stored stroke limit 1 and 2**
- **Stored stroke limit 3 (extended stored stroke check)**

Parameters (Nos. 700 to 707 or Nos. 743 to 750) 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.

Parameters (Nos. 804 to 809) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter (No. 024#4) selects either inside or outside as the forbidden area.

In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area. Each of G22; and G23; should be commanded independently of another commands in a block.

The command below creates or changes the forbidden area:

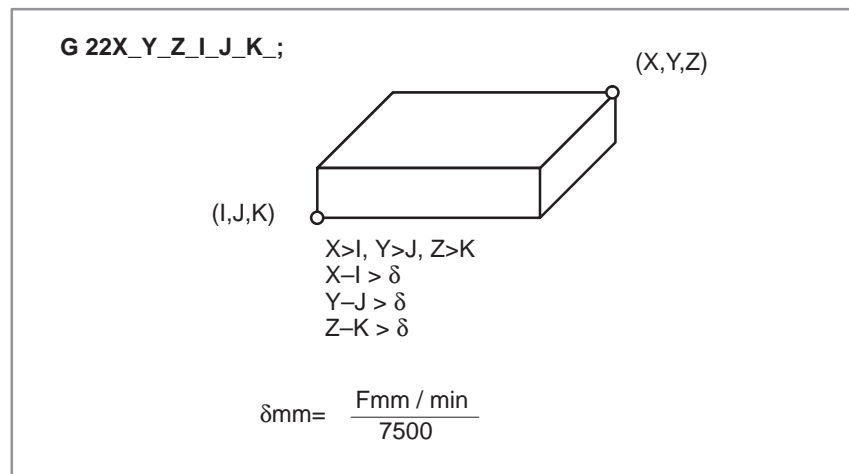


Fig. 6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

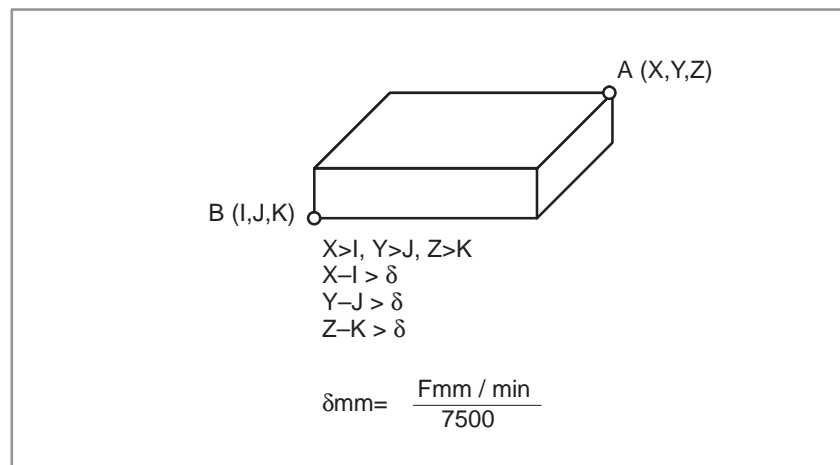


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

In limit 3, even if you mistake the order of the coordinate value of the two points, a rectangular, with the two points being the apexes, will be set as the area.

When you set the forbidden area through parameters (Nos. 804, 809), the data should be specified by the distance from the reference position in the least command increment (Output increment).

If it is set by a G22 command, specify the data by the distance from the reference position in the least input increment (Input increment.) The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

At power-on, G22 mode is selected.

- **Checkpoint for the forbidden area**

Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.

If point A (The top of the tool) is checked in Fig. 6.3 (d) , the distance “a” should be set as the data for the stored stroke limit function. If point B (The tool chuck) is checked, the distance “b” must be set. When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

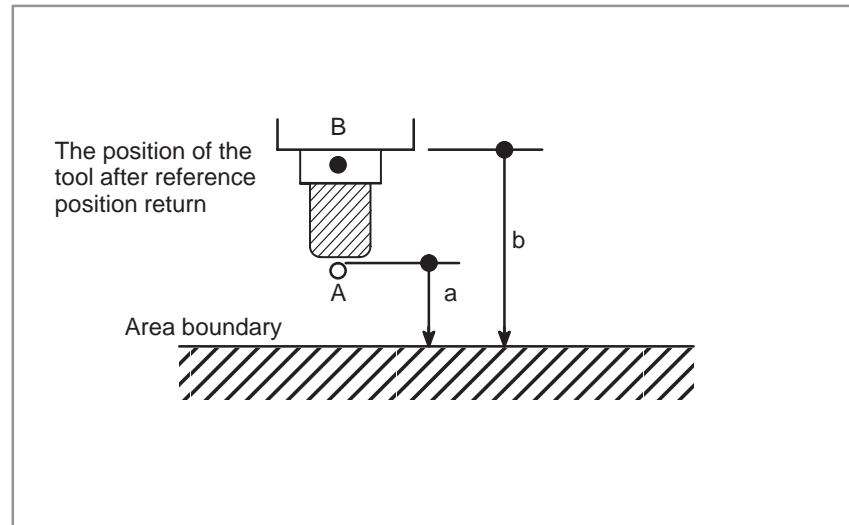


Fig. 6.3 (d) Setting the forbidden area

- **Forbidden area over-lapping**

Area can be set in piles.

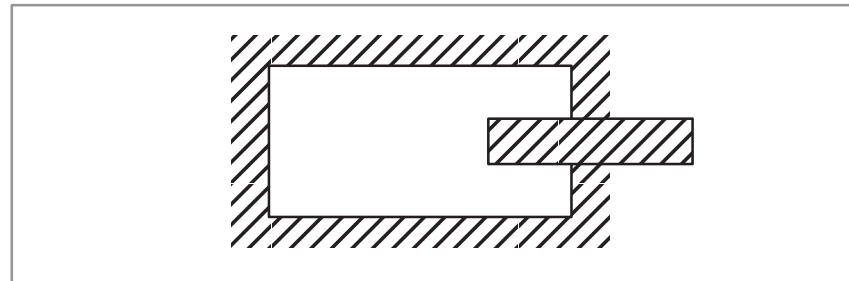


Fig. 6.3 (e) The forbidden area over lapping

Unnecessary limits should be set beyond the machine stroke.

- **Overrun amount of stored stroke limit**

If the maximum rapid traverse rate is F mm/minute, the maximum overrun amount, L mm, of the stored stroke limit is obtained from the following expression:

$$L \text{ mm} = F/7500$$

The tool enters the specified inhibited area by up to L mm. The tool can be stopped at a position up to L mm in front of the inhibited area if bit 7 of parameter 0076 is specified accordingly. In this case, the tool will not enter the inhibited area.

- **Effective time for a forbidden area**

Bit 3 of parameter 065 is used to select whether each stroke limit is enabled, either immediately after power-on or later, after a manual reference position return or reference position return by G28.

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 limit 3).

- **Releasing the alarms**

If a stroke check alarm occurs, manually retract the tool from the inhibited area in the direction opposite to the displayed alarm direction. Press the reset key to cancel the alarm. If the tool has entered two inhibited areas, thus preventing manual retraction of the tool, set the system to the emergency stop state. Change the stroke limit parameter and cancel the alarm. Then, manually retract the tool.

- **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.

NOTE

In setting a forbidden area, if the two points to be set are the same, the area is as follows:

- 1 When the forbidden area is limit 1, 2, all areas are forbidden areas.
- 2 When the forbidden area is limit 3, all areas are movable areas.

Alarms

Number	Message	Contents
5n0	OVER TRAVEL: +n	Exceeded the n-th axis (1–8) + side stored stroke limit 1 or 2.
5n1	OVER TRAVEL: –n	Exceeded the n-th axis (1–8) – side stored stroke limit 1 or 2.
5n2	OVER TRAVEL: +n	Exceeded the n-th axis (1–3) + side stored stroke limit 3.
5n3	OVER TRAVEL: –n	Exceeded the n-th axis (1–3) – side stored stroke limit 3.

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. 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-diagnostic function.

7.1 ALARM DISPLAY

Explanations

- Alarm screen

When an alarm occurs, the alarm screen appears.

```

ALARM MESSAGE                                00000 N0000

100  P/S ALARM
417  SERVO ALARM : X AXIS DGTL PARAM
427  SERVO ALARM : Y AXIS DGTL PARAM

NOT READY ALARM                                S    0 T
[ ALARM ][ OPR ][ MSG ][ MDI ][          ]

```

- Another method for alarm displays

In some cases, the alarm screen may not be displayed. Instead, ALARM will blink at the bottom of the screen.

```


PARAMETER                                00100 N0002

(SETTING 1)
REVS  = 0
REVY  = 0
TVON  = 0
ISO   = 1  (0:EIA  1:ISO )
INCH  = 0  (0:MM   1:INCH)
I/O   = 0
ABS   = 0  (0:INC  1:ABS)
SEQ   = 0

CLOCK 93/12/07
      19:34:32
NO. REVX =                                S    0 T
NOT READY ALARM                                MDI
[ PARAM ][ DGNOS ][          ][ SV-PRM ][          ]

```

In this case, display the alarm screen as follows:

- 1 Press the function  key .
- 2 Press the soft key [ALARM].

- **Reset of the alarm**

Error codes and messages indicate the cause of an alarm. To recover from an alarm, eliminate the cause and press the reset key.

- **Error codes**

The error codes are classified as follows:

No. 000 to 250: Program errors(*1)

No. 3n0 to 3n8: Absolute pulse coder (APC) alarms (*2)

No. 3n9: Serial pulse coder (SPC) alarms (*2)

No. 400 to 495: Servo alarms

No. 510 to 581: Overtravel alarms

No. 600 to 607: PMC alarms

No. 700 to 704: Overheat alarms

No. 910 to 988: System alarms

*1) For an alarm (No. 000 to 250) that occurs in association with background operation, the indication "xxxBP/S alarm" is provided (where xxx is an alarm number). Only a BP/S alarm is provided for No. 140.

*2) n is controlled axis number.




See the error code list in the Appendix G for details of the error codes.

7.2

CHECKING BY SELF-DIAGNOSTIC SCREEN

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. The state of the system can be checked by displaying the self-diagnostic screen.

Procedure for Diagnosis

- 1 Press the function key .
- 2 Press the soft key [DGNOS].
- 3 The diagnostic screen has more than 1 pages. Select the screen by the following operation.
 - (1) Change the page by the 1-page change key.
 - (2) • Press the  key.
 - Key input the number of the diagnostic data to be displayed.
 - Press the  key.

PARAMETER		O1224 N0000	
NO.	DATA	NO.	DATA
0700	00000000	0725	00000000
0701	00000000	0726	00000000
0710	00000000	0727	00000000
0711	00000000	0730	00000000
0712	00000000	0731	00000000
0720	00000000	0732	00000000
0721	00000000	0733	00000000
0722	00000000	0734	00000000
0723	00000000	0735	00000000
0724	00000000	0736	00000000
NO. 0700 =		S O T	
04:02:02		AUTO	
[PARAM][DGNOS][][SV-PRM][]	

	#7	#6	#5	#4	#3	#2	#1	#0
0700		CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN

When a digit is "1", the corresponding status is effective.

CFIN The M, S, O, or T function is being executed.

CMTN A move command in the cycle operation is being executed.

CDWL Dwell is being executed.

CINP An in-position check is being executed.

COVZ Override is at 0%.

CITL Interlock signal (STLK) is turned on.

CSCT Speed arrival signal of spindle is turned on.

	#7	#6	#5	#4	#3	#2	#1	#0
0701			CRST					

CRST One of the following : The reset button on the MDI panel, emergency stop, or remote reset is on.

	#7	#6	#5	#4	#3	#2	#1	#0
0712	STP	REST	EMS		RSTB			CSU

Indicates automatic operation stop or feed hold status. These are used for troubleshooting.

STP The flag which stops the automatic operation. This is set at the following condition.

- External reset signal is turned on.
- Emergency stop signal is turned on.
- Feed hold signal is turned on.
- Reset button on the CRT/MDI panel is turned on.
- The mode is changed to the manual mode, such as JOG, HANDLE/STEP, TEACH INJOG, TEACH IN HANDLE.
- Other alarm is generated.

REST This is set when one of the external reset, emergency stop, or reset button is set on.

EMS This is set when the emergency stop is set on.

RSTB This is set when the reset button is on.

CSU This is set when the emergency stop is turned on, or when the servo alarm has been generated.

8

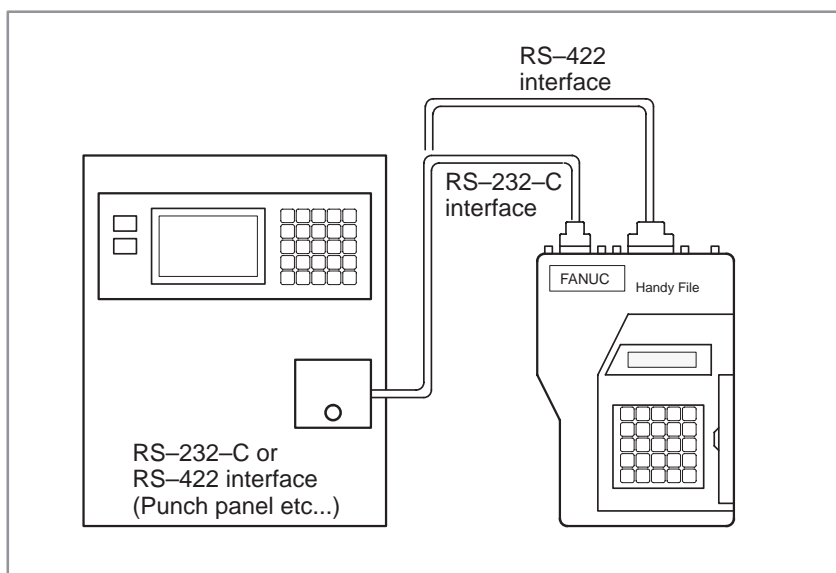
DATA INPUT/OUTPUT

NC data is transferred between the NC and external input/output devices such as the Handy File.

The following types of data can be entered and output :

- 1.Program
- 2.Offset data
- 3.Parameter
- 4.Pitch error compensation data
- 5.Custom macro common variable

Before an input/output device can be used, the input/output related parameters must be set.



8.1
FILES

Of the external input/output devices, the FANUC Handy File and FANUC Floppy Cassette use floppy disks as their input/output medium, and the FANUC FA Card uses an FA card as its input/output medium.

In this manual, an input/output medium is generally referred to as a floppy. However, when the description of one input/output medium varies from the description of another, the name of the input/output medium is used. In the text below, a floppy represents a floppy disk or FA card.

Unlike an NC tape, a floppy allows the user to freely choose from several types of data stored on one medium on a file-by-file basis.

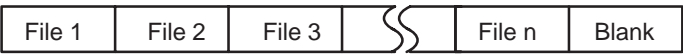
Input/output is possible with data extending over more than one floppy disk.

Explanations

• What is a File

The unit of data, which is input/output between the floppy and the CNC by one input/output operation (pressing the [READ] or [PUNCH] key), is called a “file”. When inputting CNC programs from, or outputting them to the floppy, for example, one or all programs within the CNC memory are handled as one file.

Files are assigned automatically file numbers 1,2,3,4 and so on, with the lead file as 1.



• Request for floppy replacement

When one file has been entered over two floppies, LEDs on the adaptor flash alternately on completion of data input/output between the first floppy and the CNC, prompting floppy replacement. In this case, take the first floppy out of the adaptor and insert a second floppy in its place. Then, data input/output will continue automatically.

Floppy replacement is prompted when the second floppy and later is required during file search-out, data input/output between the CNC and the floppy, or file deletion.

Floppy 1



Floppy 2



Since floppy replacement is processed by the input/output device, no special operation is required. The CNC will interrupt data input/output operation until the next floppy is inserted into the adaptor.

When reset operation is applied to the CNC during a request for floppy replacement, the CNC is not reset at once, but reset after the floppy has been replaced.

• Protect switch

The floppy is provided with the write protect switch. Set the switch to the write enable state. Then, start output operation.

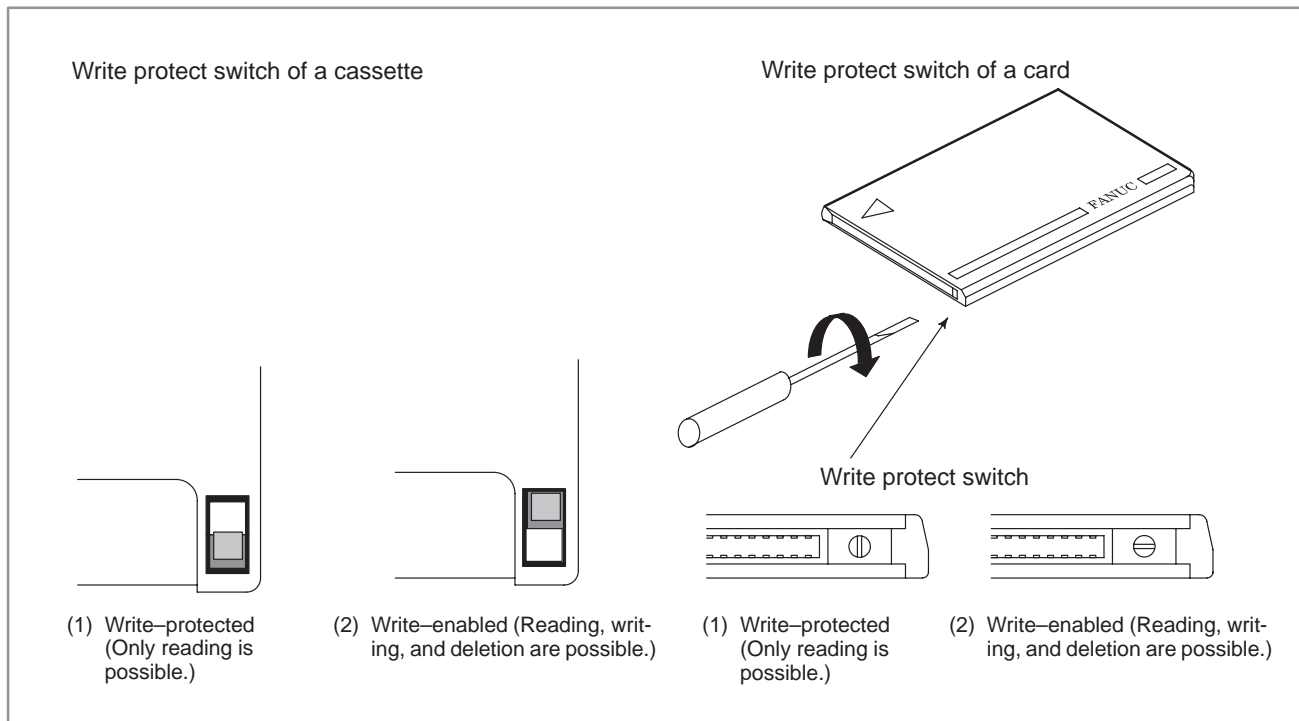


Fig. 8.1 Protect switch

• Writing memo

Once written in the cassette or card, data can subsequently be read out by correspondence between the data contents and file numbers. This correspondence cannot be verified, unless the data contents and file numbers are output to the CNC and displayed. The data contents can be displayed with display function for directory of floppy disk (See Section 8.8).

To display the contents, write the file numbers and the contents on the memo column which is the back of floppy.

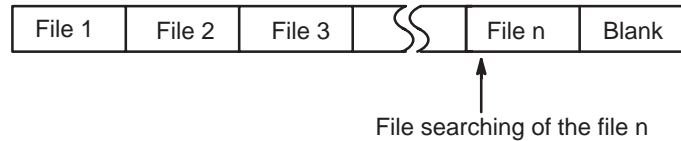
(Entry example on MEMO)

File 1	NC parameters
File 2	Offset data
File 3	NC program O0100
.	.
.	.
.	.
File (n-1)	NC program O0500
File n	NC program O0600

8.2 FILE SEARCH




When the program is input from the floppy, the file to be input first must be searched.

For this purpose, proceed as follows:



Procedure for File heading

Procedure

- 1 Press the **EDIT** or **AUTO** switch on the machine operator's panel.
- 2 Press function  key and display the program screen..
- 3 Enter address  key.
- 4 Enter the number of the file to search for.
 - N0
The beginning of the cassette or card is searched.
 - One of N1 to N9999
Of the file Nos. 1 to 9999, a designated file is searched.
 - N-9999
The file next to that accessed just before is searched.
 - N-9998
When N-9998 is designated, N-9999 is automatically inserted each time a file is input or output. This condition is reset by the designation of N1, N1 to 9999, or N-9999 or reset.
- 5 Press soft key  . The specified file is searched for.

Explanations

- **File search by N-9999**

The same result is obtained both by sequentially searching the files by specifying Nos. N1 to N9999 and by first searching one of N1 to N9999 and then using the N-9999 searching method. The searching time is shorter in the latter case.

Alarm

No.	Description
86	<p>The ready signal (DR) of an input/output device is off.</p> <p>An alarm is not immediately indicated in the CNC even when an alarm occurs during head searching (when a file is not found, or the like).</p> <p>An alarm is given when the input/output operation is performed after that. This alarm is also raised when N1 is specified for writing data to an empty floppy. (In this case, specify N0.)</p>

8.3

FILE DELETION

Files stored on a floppy can be deleted file by file as required.

Procedure for File Deletion

- Procedure
- 1

Insert the floppy into the input/output device so that it is ready for writing.
- 2

Press the EDIT switch on the machine operator's panel.
- 3

Press function

PRGRM

 key and display the program screen.
- 4

Enter address

N

 key.
- 5

Enter the number (from 1 to 9999) of the file to delete.
- 6

Press soft key

OUTPT
START

 .

The specified file is deleted.

Explanations

- File number after the file is deleted

When a file is deleted, the file numbers after the deleted file are each decremented by one. Suppose that a file numbered k was deleted. In this case, files are renumbered as follows:
Before deletion after deletion
1 to (k-1) . . . 1 to (k-1)
k Deleted
(k+1) to n . . . k to (n-1)
- Protect switch

Set the write protect switch to the write enable state to delete the files.

8.4

PROGRAM

INPUT/OUTPUT

8.4.1

Inputting a Program

This section describes how to load a program into the CNC from a floppy or NC tape.


Procedure for Inputting a Program

- Procedure
- 1


Make sure the input device is ready for reading.
- 2


Press the EDIT switch on the machine operator's panel.
- 3

When using a floppy, search for the required file according to the procedure in Section 8.2.
- 4

Press function  key and display the program screen.
- 5

After entering address O, specify a program number to be assigned to the program. When no program number is specified here, the program number used on the floppy or NC tape is assigned.
- 6

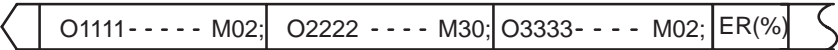
Press soft key  .
The program is input and the program number specified is assigned to the program.
- 7

To abandon the input at any point, press the  key.

Explanations

- Collation

If a program is input while the data protect key on the machine operator's panel turns ON, the program loaded into the memory is verified against the contents of the floppy or NC tape.
If a mismatch is found during collation, the collation is terminated with an alarm (P/S No. 79).
If the operation above is performed with the data protection key turns OFF, collation is not performed, but programs are registered in memory.
- Inputting multiple programs from an NC tape

When a tape holds multiple programs, the tape is read up to ER (or %).


- **Program numbers on a NC tape**

- ☐ When a program is entered without specifying a program number.
 - The O-number of the program on the NC tape is assigned to the program. If the program has no O-number, the N-number in the first block is assigned to the program.
 - When the program has neither an O-number nor N-number, the previous program number is incremented by one and the result is assigned to the program.
- ☐ When a program is entered with a program number
The O-number on the NC tape is ignored and the specified number is assigned to the program. When the program is followed by additional programs, the first additional program is given the program number. Additional program numbers are calculated by adding one to the last program.

- **Input with the soft keys**

The soft keys can be used to input a program.

This operation is enabled if the floppy disk directory display function is not supported or, if the function is supported, the Floppy Cassette is not specified as the input/output unit.

Procedure for program input with the soft keys

Procedure

- 1 Display the program screen in EDIT mode or background edit mode.
- 2 Press the **[I/O]** soft key.
- 3 Input address **[O]** , then the program number. If this step is skipped, the program number on the NC tape is automatically selected.
- 4 Press the **[READ]** soft key. To abandon input at any point, press the **[STOP]** soft key.
- 5 After input has been completed, press the **[CAN]** soft key to display the program screen again.

- **Program input in background edit mode**

Program input is identical to that in foreground edit mode. If the reset key is pressed to abandon input in the background while a program is being executed, however, the program execution is also halted. To input a program in background edit mode, use the soft keys.

Alarm




No.	Description
70	The size of memory is not sufficient to store the input programs
73	An attempt was made to store a program with an existing program number.
79	The verification operation found a mismatch between a program loaded into memory and the contents of the program on the floppy or NC tape.

8.4.2 Outputting a Program

A program stored in the memory of the CNC unit is output to a floppy or NC tape.

Procedure for Outputting a Program

Procedure

- 1 Make sure the output device is ready for output.
- 2 To output to an NC tape, specify the punch code system (ISO or EIA) using a parameter. To output the program to a floppy disk, select ISO.
- 3 Press the EDIT switch on the machine operator's panel.
- 4 Press function  key.
- 5 Enter address  key.
- 6 Enter a program number. If -9999 is entered, all programs stored in memory are output.
To output multiple programs by full MDI key at one time, enter a range as follows :
OΔΔΔΔ,O□□□□
Programs No.ΔΔΔΔ to No.□□□□ are output.
- 7 Press soft key  .
The specified program or programs are output.

Explanations (Output to a floppy)


- **File output location**
When output is conducted to the floppy, the program is output as the new file after the files existing in the floppy. New files are to be written from the beginning with making the old files invalid, use the above output operation after the N0 head searching.
- **An alarm while a program is output**
When P/S alarm 86 occurs during program output, the floppy is restored to the condition before the output.
- **Outputting a program after file heading**
When program output is conducted after N1 to N9999 head searching, the new file is output as the designated n-th position. In this case, 1 to n-1 files are effective, but the files after the old n-th one are deleted. If an alarm occurs during output, only the 1 to n-1 files are restored.
- **Efficient use of memory**
To efficiently use the memory in the cassette or card, output the program by setting parameter (No. 002#7, No. 012#7, No. 050#7, or 051#7) to 1. This parameter makes the feed is not output, utilizing the memory efficiently.
- **On the memo record**
Head searching with a file No. is necessary when a file output from the CNC to the floppy is again input to the CNC memory or compared with the content of the CNC memory. Therefore, immediately after a file is output from the CNC to the floppy, record the file No. on the memo.

- **Output with the soft keys**

The soft keys can be used to input a program.

This operation is enabled if the floppy disk directory display function is not supported or, if the function is supported, the Floppy Cassette is not specified as the input/output unit.


Procedure for program output with the soft keys

- 1 Display the program screen in EDIT mode or background edit mode.
- 2 Press the **[I/O]** soft key.
- 3 Input address , then the program number. If -9999 is entered as the program number, all programs in memory are output.
To output multiple programs by full MDI key at one time, enter a range as follows :
OΔΔΔΔ,O□□□□
Programs No.ΔΔΔΔ to No.□□□□ are output.
- 4 Press the **[PUNCH]** soft key. To abandon input at any point, press the **[STOP]** soft key.
- 5 After input has been completed, press the **[CAN]** soft key to display the program screen again.

- **Program output in background edit mode**

Program input is identical to that in foreground edit mode. If the reset key is pressed to abandon input in the background while a program is being executed, however, the program execution is also halted. A program that is currently selected in the foreground can also be output.

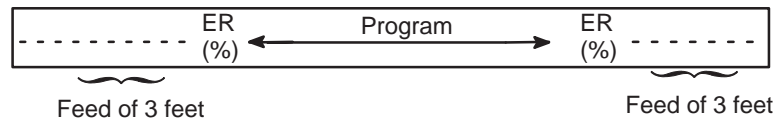
NOTE

Some machines also use the  key, used to start punching of the CNC tape in the background, as the cycle start key to start automatic operation. If such a machine is being used, punch the CNC tape in a mode other than automatic operation mode. (To check whether the key is used for both purposes, refer to the manual provided by the machine tool builder.)

Explanations (Output to an NC tape)

- **Format**

A program is output to paper tape in the following format:



If three-feet feeding is too long, press the key during feed punching to cancel the subsequent feed punching.

- **TV check**

A space code for TV check is automatically punched.

- **ISO code**

When a program is punched in ISO code, two CR codes are punched after an LF code.



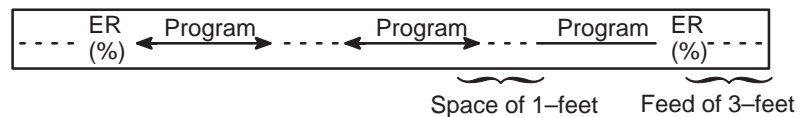
Only LF can be punched as the EOB if bit 7 of parameter 070 is specified accordingly.

- **Stopping the punch**

Press the key to stop punch operation.

- **Punching all programs**

All programs are output to paper tape in the following format.



The sequence of the programs punched is undefined.

8.5

OFFSET DATA INPUT AND OUTPUT



8.5.1 Inputting Offset Data

Offset data is loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as for offset value output. See section 8.5.2.

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.

Procedure for Inputting Offset data

Procedure

- 1 Make sure the input device is ready for reading
- 2 Press the EDIT switch on the machine operator's panel.
- 3 When using a floppy, search for the required file according to the procedure in Section 8.2.
- 4 Press function  key.and display the offset screen.
- 5 Press  key.



The input offset data will be displayed on the screen after completion of input operation.

8.5.2 Outputting Offset Data

All offset data is output in a output format from the memory of the CNC to a floppy or NC tape.

Procedure for Outputting Offset Data

Procedure

- 1 Make sure the output device is ready for output.
- 2 Specify the punch code system (ISO or EIA) using a parameter.
- 3 Press the EDIT switch on the machine operator's panel.
- 4 Press function  key.
- 5 Press soft key  .

Offset data is output in the output format described below.

Explanations

• Output format

Output format is as follows:

Format

- (1) For tool compensation memory A

G10 P_R_;

where P_: Offset No.

R_: Tool compensation amount

- (2) For tool compensation memory B

Setting/changing the geometric compensation amount

G10 L10 P_R_;

Setting/changing the wear compensation amount

G10 L11 P_R_;

The L1 command may be used instead of L11 for format compatibility of the conventional CNC.

• Output file name

When the floppy disk directory display function is used, the name of the output file is OFFSET.

8.6 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA



Parameters and pitch error compensation data are input and output from different screens, respectively. This chapter describes how to enter them.

8.6.1 Inputting Parameters

Parameters are loaded into the memory of the CNC unit from a floppy or NC tape. The input format is the same as the output format. See Section 8.6.2. 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 When using a floppy, search for the required file according to the procedure in Section 8.2.
- 3 Press the EMERGENCY STOP button on the machine operator's panel.
- 4 Press function  key and display the parameter screen.
- 5 Enter 1 in response to the prompt for writing parameters (PWE). Alarm P/S100 (indicating that parameters can be written) appears.
- 6 Press soft key  .
Parameters are read into memory. Upon completion of input, the "INPUT" indicator at the lower-right corner of the screen disappears.
- 7 Enter 0 in response to the prompt for writing parameters.
- 8 Turn the power to the NC back on.
- 9 Release the EMERGENCY STOP button on the machine operator's panel.

NOTE



For a full keyboard, before performing this procedure, set bit 3 of parameter No. 0038 to 1.

8.6.2 Outputting Parameters

All parameters are output in the defined format from the memory of the CNC to a floppy or NC tape.

Outputting parameters

Procedure

- 1 Make sure the output device is ready for output.
- 2 Specify the punch code system (ISO or EIA) using a parameter.
- 3 Press the EDIT switch on the machine operator's panel.
- 4 Press function  key, and display the parameter screen.
- 5 Press soft key  .

All parameters are output in the defined format.

Explanations

- **Output format**

Output format is as follows:

N .. P ;

N:Parameter No.

P:Parameter setting value .

- **Output file name**

When the floppy disk directory display function is used, the name of the output file is PARAMETER.

8.7 INPUTTING/ OUTPUTTING CUSTOM MACRO B COMMON VARIABLES

8.7.1 Inputting Custom Macro B Common Variables

The value of a custom macro B common variable (#500 to #999) is loaded into the memory of the CNC from a floppy or NC tape. The same format used to output custom macro B common variables is used for input. See Section 8.7.2. For a custom macro common variable to be valid, the input data must be executed by pressing the cycle start button after data is input. When the value of a common variable is loaded into memory, this value replaces the value of the same common variable already existing (if any) in memory.

Inputting custom macro common variables

Procedure

- 1 Input the program according to the procedure in Section 8.4.1.
- 2 Press the **AUTO** switch on the machine operator's panel upon completing input.
- 3 Press the cycle start button to execute the loaded program.
- 4 Display the macro variable screen to check whether the values of the common variables have been set correctly.

Explanations

- **Common variables**



The common variables (#500 to #531) can be input and output. When the option for adding a common variable is specified, values from #500 to #999 can be input and output. Common variables #100 to 199 cannot be input or output.

8.7.2 Outputting Custom Macro B Common Variable

Custom macro common variables (#500 to #999) stored in the memory of the CNC can be output in the defined format to a floppy or NC tape.

Outputting custom macro common variable

Procedure

- 1 Make sure the output device is ready for output.
- 2 Specify the punch code system (ISO or EIA) using a parameter.
- 3 Press the EDIT switch on the machine operator's panel.
- 4 Press function  key and display the macro variables.
- 5 Press function  key. Common variables are output in the defined format.

Explanations

• Output format

The output format is as follows:

```
%
;
#500=[25283*65536+65536]/134217728. .... ①
#501=#0; ..... ②
#502=0; ..... ③
#503= .....;
.....;
.....;
#531= .....;
M02;
%
```

- ① The precision of a variable is maintained by outputting the value of the variable as <expression>.
- ② Undefined variable
- ③ When the value of a variable is 0

• Output file name

When the floppy disk directory display function is used, the name of the output file is [MACRO VAR].

• Common variable

The common variables (#500 to #531) can be input and output.
When the option for adding a common variable is specified, values from #500 to #999 can be input and output.
Common variables #100 to 199 cannot be input or output.

8.8 DISPLAYING DIRECTORY OF FLOPPY DISK

On the floppy directory display screen, a directory of the FANUC Handy File, FANUC Floppy Cassette, or FANUC FA Card files can be displayed. In addition, those files can be loaded, output, and deleted.

```

DIRECTORY (FLOPPY)
NO.      FILE NAME
O0001  N00000
(METER) VOL

19:35:17
EDIT
[ SRHFIL ][ READ ][ PUNCH ][ DELETE ][ ]

```




This operation is enabled if the floppy disk directory display function is supported and the Floppy Cassette is specified as the input/output unit.

8.8.1 Displaying the Directory

Displaying the directory of floppy disk files

Procedure 1

Use the following procedure to display a directory of all the files stored in a floppy:

- 1 Press the EDIT switch on the machine operator's panel.
- 2 Press function  key .
- 3 Press soft key [FLOPPY].
- 4 Press page key  or  .
- 5 The screen below appears.



DIRECTORY (FLOPPY)		O0555 N0000
NO.	FILE NAME	(METER) VOL
0001	PARAMETER	65.6
0002	ALL. PROGRAM	1.9
0003	O0001	1.3
0004	O0002	1.3
0005	O0003	1.3
0006	O0004	1.3
0007	O005	1.3
0008	O0100	1.9
0009	O0555	1.3
19:36:51 EDIT		S 0 T
[SRHFIL][READ][PUNCH][DELETE][]		

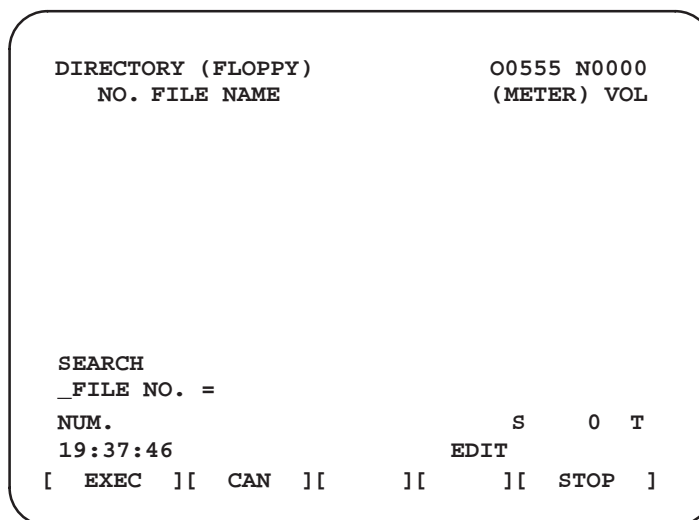
Fig. 8.8.1 (a)

- 6 Press a page key again to display another page of the directory.

Procedure 2

Use the following procedure to display a directory of files starting with a specified file number :

- 1 Press the EDIT switch on the machine operator's panel.
- 2 Press function  key.
- 3 Press soft key [FLOPPY].
- 4 Press soft key [F SRH].
- 5 Enter a file number, and press  key.
- 6 Press soft key[EXEC].
- 7 Press a page key to display another page of the directory.
- 8 Press soft key [CAN] to return to the soft key display shown in the screen of Fig 8.8.1(a).



DIRECTORY (FLOPPY) O0555 N0000
NO. FILE NAME (METER) VOL

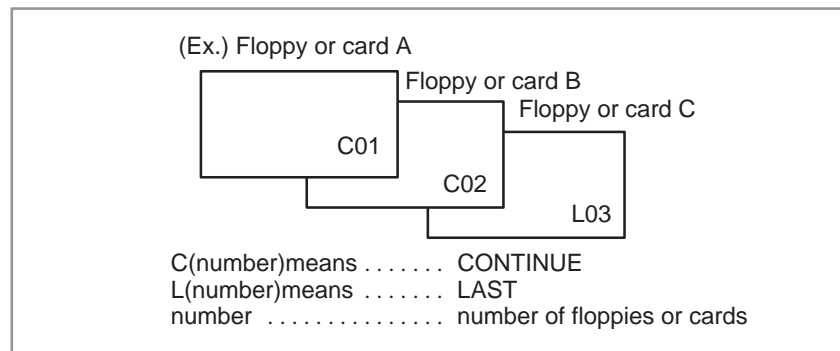
SEARCH
_FILE NO. =
NUM. S O T
19:37:46 EDIT
[EXEC][CAN][][STOP]

Fig. 8.8.1 (b)

Explanations

- Screen fields and their meanings

NO : Displays the file number
FILE NAME : Displays the file name.
(METER) : Converts and prints out the file capacity to paper tape length. You can also produce “(FEET)” by setting the INPUT UNIT to INCH of the setting data.
VOL. : When the file is multi-volume, that state is displayed.




8.8.2 Reading Files

The contents of the specified file number are read to the memory of NC.

Reading files

Procedure

- 1 Press the EDIT switch on the machine operator's panel.
- 2 Press function  key.
- 3 Press soft key **[FLOPPY]**.
- 4 Press soft key **[READ]**.

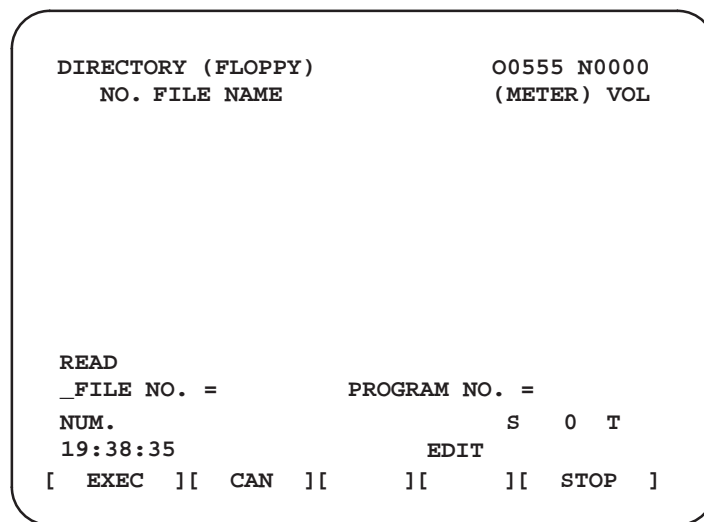




Fig. 8.8.2


- 5 Enter a file number.
- 6 Press function  key .
- 7 To modify the program number, enter the program number, then press function  key.
- 8 Press soft key **[EXEC]**. The file number indicated in the lower-left corner of the screen is automatically incremented by one.
- 9 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig. 8.8.1.(a).

8.8.3 Outputting Programs

Any program in the memory of the CNC unit can be output to a floppy as a file.

Outputting programs

Procedure


- 1 Press the EDIT switch on the machine operator's panel.
- 2 Press function  key.
- 3 Press soft key **[FLOPPY]**.
- 4 Press soft key **[PUNCH]**.

```

    DIRECTORY (FLOPPY)
    NO. FILE NAME
    00555 N0000
    (METER) VOL

PUNCH
  _FILE NO. =      _ PROGRAM NO. =
  NUM.
  19:39:17
  EDIT
[ EXEC ][ CAN ][ PRG-NO ][      ][ STOP ]


```

- 5 Enter a program number. To write all programs into a single file, enter -9999 in the program number field. In this case, the file name **“ALL.PROGRAM”** is registered.
- 6 Press function  key .

To output multiple programs by full MDI key at one time, enter a range as follows :

$$\triangle\triangle\triangle\triangle \rightarrow [\text{PRG-NO.}] \rightarrow \square\square\square\square \rightarrow [\text{PRG-NO.}]$$

Programs No. $\Delta\Delta\Delta\Delta$ to No. $\square\square\square\square$ are output.

- 7 Press soft key **[EXEC]**. The program or programs specified in step 5 are written after the last file on the floppy. To output the program after deleting files starting with an existing file number, key in the file number, then press function  key followed by soft key **[EXEC]**.
- 8 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig. 8.8.1 (a).


8.8.4

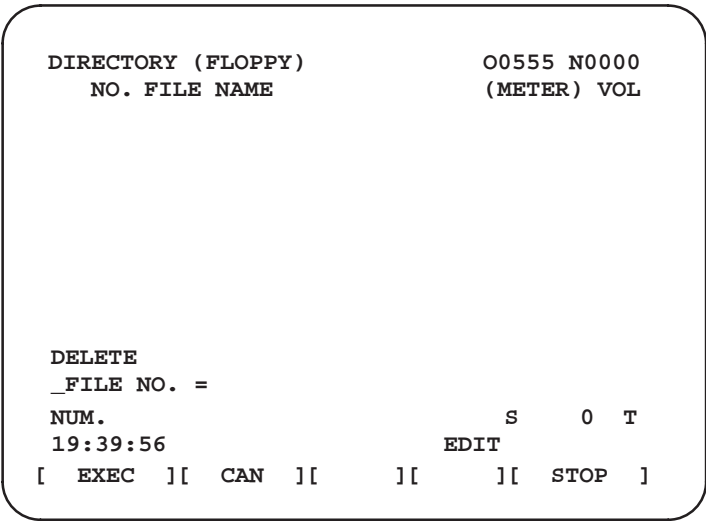
Deleting Files


The file with the specified file number is deleted.

Deleting files

Procedure



- 1 Press the EDIT switch on the machine operator’s panel.
- 2 Press function key  .
- 3 Press soft key **[FLOPPY]**.
- 4 Press soft key **[DELETE]**.



- 5 Specify the file to be deleted.
When specifying the file with a file number, type the number and press function  key.
- 6 Press soft key **[EXEC]**.
The file specified in the file number field is deleted. When a file is deleted, the file numbers after the deleted file are each decremented by one.
- 7 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig. 8.8.1 (a).

Restrictions

- **Inputting file numbers and program numbers with keys**
- **I/O devices**

If  +  keys are pressed without key inputting file number and program number, file number or program number shows blank. When 0 is entered for file numbers or program numbers, 1 is displayed.

To use channel 1 for the input/output unit, set the device of bits 6 and 7 of parameter 038 to the Floppy Cassette. To use channel 2, set the device of bits 4 and 5 of parameter 038 to the Floppy Cassette. To use channel 3, set the device of bits 1 and 2 of parameter 038 to the Floppy Cassette.

- **Significant digits**

For the numeral input in the data input area with FILE NO. and PROGRAM NO., only lower 4 digits become valid.

- **Collation**

When the data protection key on the machine operator's panel is ON, no programs are read from the floppy. They are verified against the contents of the memory of the CNC instead.

ALARM




No.	Contents
71	An invalid file number or program number was entered. (Specified program number is not found.)
79	Verification operation found a mismatch between a program loaded into memory and the contents of the floppy
86	The dataset-ready signal (DR) for the input/output device is turned off. (The no file error or duplicate file error occurred on the input/output device because an invalid file number, program number, or file name was entered.

8.8.5
Changing the File
Name

Change the name of the file having the specified file number.

Procedure for changing the file name

Procedure

- 1 Press the EDIT switch on the machine operator’s panel.
- 2 Press function key  .
- 3 Press soft key [FLOPPY].
- 4 Press soft key [RENAME].
- 5 Position the cursor to FILE NO. then enter the number of the file whose name is to be changed. Press the  key.
- 6 Position the cursor to NAME and key in a new file name. Then, press the  key.
- 7 Press soft key [EXEC].
- 8 To return to the previous screen, press the [CAN] soft key.

FILE DIRECTORY

NO.	FILE NAME	(METER) VOL
0001	PARAMETER	87.1
0002	ALL.PROGRAM	87.1
0003	O0001	1.9
0004	O0021	7.1
0005	O0041	7.1
0006	O0615	5.8
0007	O0651	9.1
0008	O0601	7.1
0009	O0645	5.8

RENAME

FILE NO. =NAME=

NUM. S 0 T0101

21:59:53 EDIT

(EXEC)

(CAN)

()

()

(STOP)

Limitations

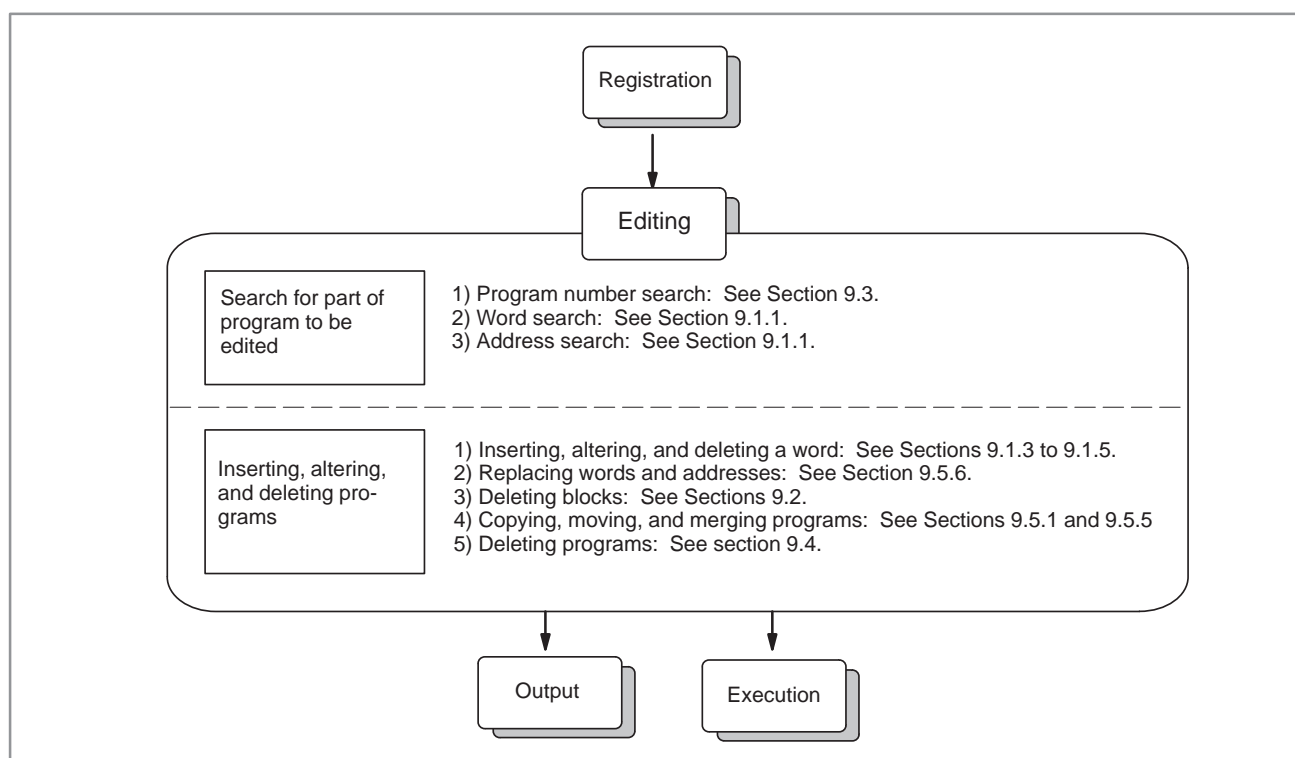
- Use channel 1 or 2 for the input/output unit and specify NEW INTERFACE as the device. These channels cannot be used for some connected input/output units.
- If the standard MDI key panel is being used, the file name will consist of an address and a numeric value.

9

EDITING PROGRAMS

General

This chapter describes how to edit programs registered in the CNC. Editing includes the insertion, modification, deletion, and replacement of words. Editing also includes deletion of the entire program and automatic insertion of sequence numbers. The extended part program editing function can copy, move, and merge programs. This chapter also describes program number search, sequence number search, word search, and address search, which are performed before editing the program.




9.1 INSERTING, ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, modifying, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

Procedure

- 1 Select **EDIT** mode.
- 2 Press function  key and display the program screen.
- 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 Search for a word to be modified.
 - Scan method
 - Word search method
- 5 Perform an operation such as altering, inserting, or deleting a word.

Explanations

- **Concept of word and editing unit**

A word is an address followed by a number. With a custom macro B, 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

(i) Program portion from an address to immediately before the next address

(ii) 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.

- **Data input during editing**


To insert or modify a word during editing, the following data is entered.

- **When the standard key panel is being used**

One word (a single alphabetic character followed by a numeric value or symbol) is entered.

- **Editing B with the standard key panel**


Even if the standard key panel is being used, editing B can be enabled by specifying bit 7 of parameter 018 accordingly. Two or more

addresses can be input at one time. If the  key is pressed after a single word (a single alphabetic character followed by a numeric value or symbol) is input, another word can be input. After all data has been entered, press the edit key to start editing.

To enable editing B, note the following:

- The  key is used to identify a breakpoint between words.

A program cannot be input or output while a program is displayed.
Input or output a program on the program directory screen.

- Input a program number as one word containing address O.
- Up to 32 characters can be entered at one time.
- Each time the  key is pressed, only the most-recently entered character is deleted.

- **When the full key panel is being used**

Two or more words, or a desired character string, can be input at one time.

WARNING


The user cannot continue program execution after altering, inserting, or deleting data of the program by suspending machining in progress by means of an operation such as a single block stop or feed hold operation during program execution. If such a modification is made, the program may not be executed exactly according to the contents of the program displayed on the screen after machining is resumed. So, when the contents of memory are to be modified by part program editing, be sure to enter the reset state or reset the system upon completion of editing before executing the program.

9.1.1 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

Procedure

- 1 Press the cursor key 







The cursor moves forward word by word on the screen; the cursor is displayed at a selected word. The cursor is positioned to the address of the selected word.

- 2 Press the cursor key 

The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

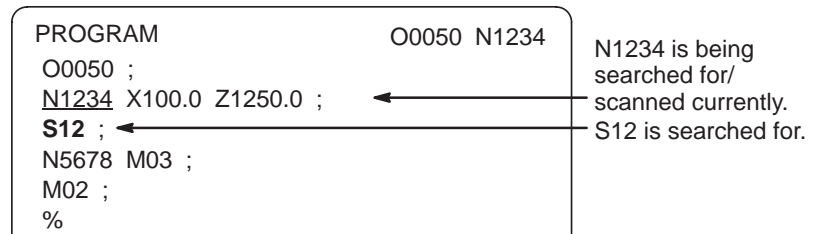
Example) When Z1250.0 is scanned

```
Program                                O0050 N1234
O0050 ;
N1234 X100.0 Z1250.0 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

- 3 Holding down the cursor key  or  scans words continuously.
- 4 Pressing the page key  displays the next page and searches for the first word of the page.
- 5 Pressing the page key  displays the previous page and searches for the first word of the page.
- 6 Holding down the page key  or  displays one page after another.

Procedure for searching a word

Example) of Searching for S12



1 Key in address **S** .

2 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.

3 Pressing the cursor key **↓** starts search operation.

Upon completion of search operation, the cursor is displayed at “S” of S12. Pressing the cursor key **↑** rather than the cursor **↓** key performs search operation in the reverse direction.

Procedure for searching an address

Example) of Searching for M03



1 Key in address **M** .

2 Press the cursor key **↓** .

Upon completion of search operation, the cursor is displayed at “M” of M03. Pressing the **↑** key rather than the **↓** key performs search operation in the reverse direction.

Alarm


Alarm number	Description
71	The word or address being searched for was not found.

9.1.2 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 two methods for heading the program pointer.

Procedure for Heading a Program

Method 1


- 1 Press  when the program screen is selected in EDIT mode.
When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2

- 1 Select **AUTO** or **EDIT** mode.
- 2 Press function  key and display the program.
- 3 Press the address key .
- 4 Press the cursor key .

9.1.3 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


Procedure

- 1 Search for or scan Z1250.0.

```
Program                                O0050 N1234
O0050 ;
N1234 X100.0 Z1250.0 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

← Z1250.0 is
searched for/
scanned.

- 2 Key in    .

- 3 Press the  key.


```
Program                                O0050 N1234
O0050 ;
N1234 X100.0 Z1250.0 T15 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

← T15 is inserted.

9.1.4

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

Procedure

- 1 Search for or scan T15.

```
Program                                O0050 N1234
O0050 ;
N1234 X100.0 Z1250.0 T15 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

← T15 is searched
for/scanned.

- 2 Key in    .

- 3 Press the  key.


```
Program                                O0050 N1234
O0050 ;
N1234 X100.0 Z1250.0 M15 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

← T15 is changed to
M15.

9.1.5

Deleting a Word

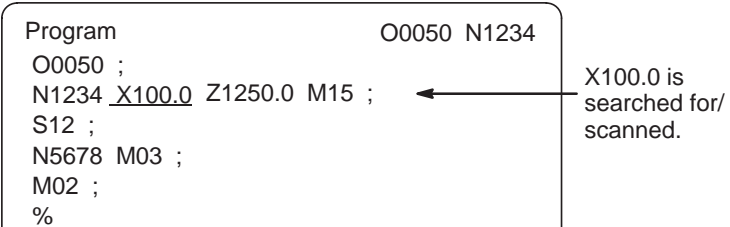
Procedure for deleting a word

- 1 Search for or scan a word to be deleted.
- 2 Press the  key.

Example of deleting X100.0

Procedure


- 1 Search for or scan X100.0.



```
Program O0050 N1234
O0050 ;
N1234 X100.0 Z1250.0 M15 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

X100.0 is searched for/
scanned.

- 2 Press the  key.



```
Program O0050 N1234
O0050 ;
N1234 Z1250.0 M15 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

X100.0 is deleted.

9.2

DELETING BLOCKS

A block or blocks can be deleted in a program.

9.2.1

Deleting a Block

The procedure below deletes a block up to its EOB code; the cursor advances to the address of the next word.

Procedure for deleting a block

- 1
- Search for or scan address N for a block to be deleted.
- 2
- Key in

EOB

.
- 3
- Press the

DELET

 key.

Example of deleting a block of No.1234

Procedure

- 1
- Search for or scan N1234.

Program

O0050 ;

N1234 Z1250.0 M15 ;

S12 ;

N5678 M03 ;

M02 ;

%

O0050 N1234

←

N1234 is searched for/ scanned.

- 2
- Key in

EOB

.

- 3
- Press the

DELET

 key.

Program

O0050 ;

S12 ;

N56789 M03 ;

M02 ;

%

O0050 N1234

←

Block containing N1234 has been deleted.

9.2.2 Deleting Multiple Blocks

The blocks from the currently displayed word to the block with a specified sequence number can be deleted.

Procedure for deleting multiple blocks

- 1 Search for or scan a word in the first block of a portion to be deleted.
- 2 Key in address N .
- 3 Key in the sequence number for the last block of the portion to be deleted.
- 4 Press the DELET key.

Example of deleting blocks from a block containing N01234 to a block containing N56789

Procedure

- 1 Search for or scan N1234.

Program O0050 ; <u>N1234</u> Z1250.0 M15 ; S12 ; N56789 M03 ; M02 ; %	O0050 N1234 ← N1234 is searched for/ scanned.
---	--

- 2 Key in N 5 6 7 8 .

Program O0050 ; <u>N1234</u> Z1250.0 M15 ; S12 ; N5678 M03 ; M02 ; %	O0050 N1234 } ← Underlined part is de- leted.
--	--

- 3 Press the DELET key.




Program O0050 ; M02 ; %	O0050 N1234 ← Blocks from block containing N1234 to block contain- ing N5678 have been deleted.
----------------------------------	--

9.3 PROGRAM NUMBER SEARCH




When memory holds multiple programs, a program can be searched for. There are two methods as follows.

Procedure for program number search

Method 1

- 1 Select **EDIT** or **AUTO** mode.
- 2 Press  key to display the program screen.
- 3 Key in address  .
- 4 Key in a program number to be searched for.
- 5 Press the cursor key  .
- 6 Upon completion of search operation, the program number searched for is displayed in the upper-right corner of the CRT screen
If the program is not found , P/S alarm No. 71 occurs.

Method 2

- 1 Select **EDIT** or **AUTO** mode.
- 2 Press  key to display the program screen.
- 3 Key in address  .
- 4 Press the cursor key  .
In this case, the next program in the directory is searched for .

Alarm

No.	Contents
59	The program with the selected number cannot be searched during external program number search.
71	The specified program number was not found during program number search.

9.4 DELETING PROGRAMS




Programs registered in memory can be deleted, either one program by one program or all at once. Also, More than one program can be deleted by specifying a range.

9.4.1 Deleting One Program

A program registered in memory can be deleted.

Procedure for deleting one program




Procedure

- 1 Select the **EDIT** mode.
- 2 Press  to display the program screen.
- 3 Key in address  .
- 4 Key in a desired program number.
- 5 Press the  key.
The program with the entered program number is deleted.

9.4.2 Deleting All Programs

All programs registered in memory can be deleted.



Procedure for deleting all programs

- 1 Select the **EDIT** mode.
- 2 Press  to display the program screen.
- 3 Key in address  .
- 4 Key in -9999.
- 5 Press edit key  to delete all programs.

9.4.3 Deleting More Than One Program by Specifying a Range

Programs within a specified range in memory are deleted.

Procedure for deleting more than one program by specifying a range

- 1 Select the **EDIT** mode.
- 2 Press  key to display the program screen.
- 3 Enter the range of program numbers to be deleted with address and numeric keys in the following format:
OXXXX,OYYYY
where XXXX is the starting number of the programs to be deleted and
YYYY is the ending number of the programs to be deleted.
- 4 Press edit key  to delete programs No. XXXX to No. YYYY.

Limitations

This operation is possible only when the full MDI key panel is being used.

9.5 EXTENDED PART PROGRAM EDITING FUNCTION

With the extended part program editing function, the operations described below can be performed using soft keys for programs that have been registered in memory.

Following editing operations are available :

- All or part of a program can be copied or moved to another program.
- One program can be merged at free position into other programs.
- A specified word or address in a program can be replaced with another word or address.

9.5.1

Copying an Entire Program

A new program can be created by copying a program.

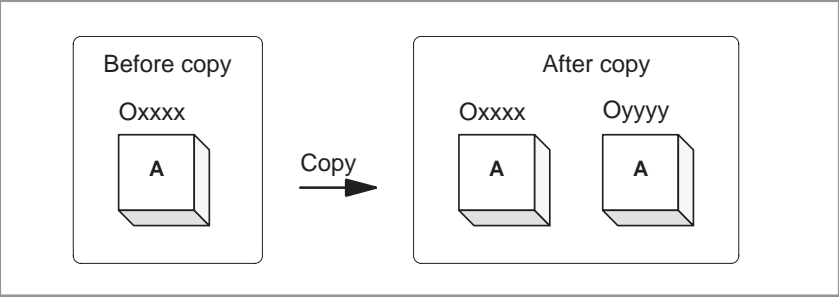



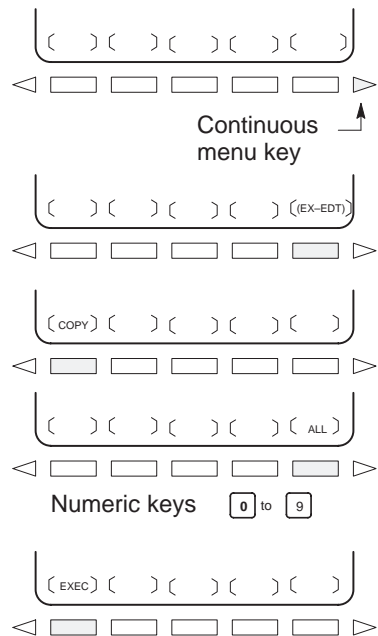


Fig. 9.5.1 Copying an entire program

In **Fig. 9.5.1**, the program with program number xxxx is copied to a newly created program with program number yyyy. The program created by copy operation is the same as the original program except the program number.

Procedure of copying an entire program

- 1 Enter the **EDIT** mode.
- 2 Press function  key to display the program screen .
- 3 Press the continuous menu key .
- 4 Press soft key **[EX-EDT]**.
- 5 Check that the screen for the program to be copied is selected and press soft key **[COPY]**.
- 6 Press soft key **[ALL]**.
- 7 Enter the number of the new program (with only numeric keys) and press the  key.
- 8 Press soft key **[EXEC]**.



9.5.2

Copying Part of a Program

A new program can be created by copying part of a program.

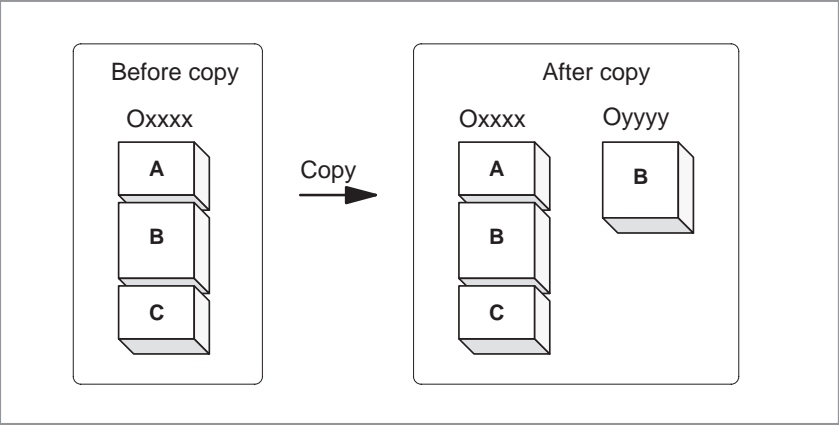
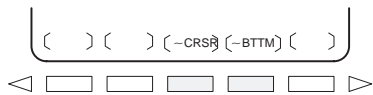
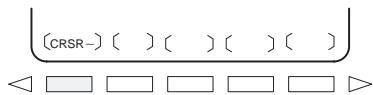


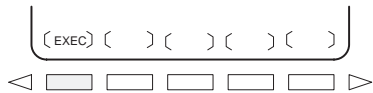
Fig. 9.5.2 Copying part of a program

In **Fig. 9.5.2**, part B of the program with program number xxxx is copied to a newly created program with program number yyyy. The program which No.is xxxx remains unchanged after copy operation.

Procedure for copying part of a program



Numeric keys to



- 1 Perform steps 1 to 5 in subsection 9.5.1.
- 2 Move the cursor to the start of the range to be copied and press soft key **[CRSR~]**.
- 3 Move the cursor to the end of the range to be copied and press soft key **[~CRSR]** or **[~BTM]** (in the latter case, the range to the end of the program is copied regardless of the position of the cursor).
- 4 Enter the number of the new program (with only numeric keys) and press the key.
- 5 Press soft key **[EXEC]**.

9.5.3

Moving Part of a Program

A new program can be created by moving part of a program.

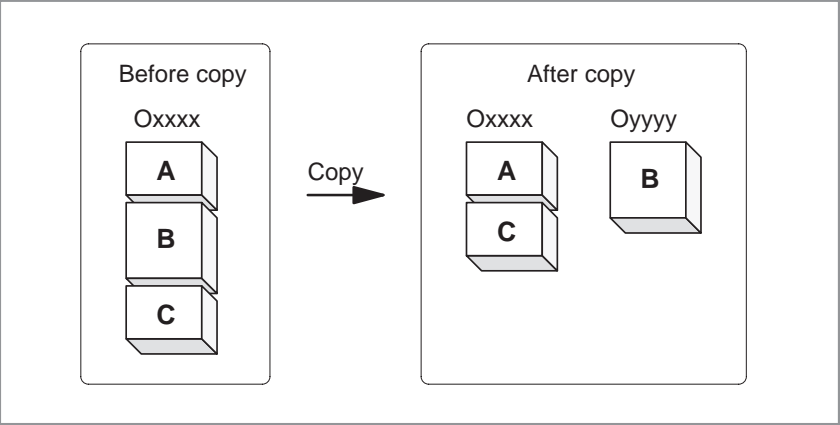


Fig. 9.5.3 Moving part of a program
In **Fig. 9.5.3**, part B of the program with program number xxxx is moved to a newly created program with program number yyyy; part B is deleted from the program with program number xxxx.

Procedure for moving part of a program

- () (MOVE) () () ()

< [] [] [] [] [] >

(CRSR~) () () () () ()

< [] [] [] [] [] >

() () () (~CRSR) (~BTM) () ()

< [] [] [] [] [] >

Numeric keys 0 to 9

(EXEC) () () () () ()

< [] [] [] [] [] >

1 Perform steps 1 to 4 in subsection 9.5.1.

2 Check that the screen for the program to be moved is selected and press soft key **[MOVE]**.

3 Move the cursor to the start of the range to be moved and press soft key **[CRSR~]**.

4 Move the cursor to the end of the range to be moved and press soft key **[~CRSR]** or **[~BTM]** (in the latter case, the range to the end of the program is copied regardless of the position of the cursor).

5 Enter the number of the new program (with only numeric keys) and press the **INPUT** key.

6 Press soft key **[EXEC]**.
- 510 —

9.5.4 Merging a Program

Another program can be inserted at an arbitrary position in the current program.

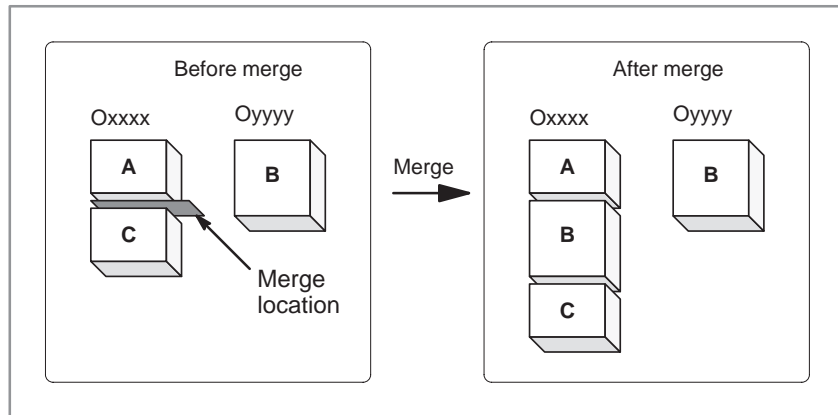
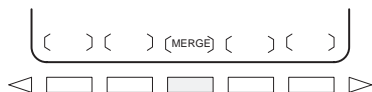


Fig. 9.5.4 Merging a program at a specified location

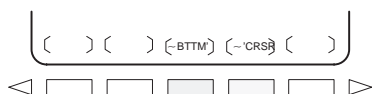
In **Fig. 9.5.4**, the program with program number XXXX is merged with the program with program number YYYY. The OYYYY program remains unchanged after merge operation.

Procedure for merging a program

- 1 Perform steps 1 to 4 in subsection 9.5.1.



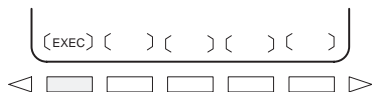
- 2 Check that the screen for the program to be edited is selected and press soft key **[MERGE]**.



- 3 Move the cursor to the position at which another program is to be inserted and press soft key **[~'CRSR]** or **[~BTTM]** (in the latter case, the end of the current program is displayed).

Numeric keys to

- 4 Enter the number of the program to be inserted (with only numeric keys) and press the **INPUT** key.



- 5 Press soft key **[EXEC]**.
The program with the number specified in step 4 is inserted before the cursor positioned in step 3.

9.5.5 Supplementary Explanation for Copying, Moving and Merging

Explanations

- **Setting an editing range**

The setting of an editing range start point with **[CRSR ~]** can be changed freely until an editing range end point is set with **[~CRSR]** or **[~BTM]**. If an editing range start point is set after an editing range end point, the end point must be reset.

The setting of an editing range start point and end point remains valid until an operation is performed to invalidate the setting.

One of the following operations invalidates a setting:

- An edit operation other than address search, word search/scan, and search for the start of a program is performed after a start point or end point is set.
- Processing is returned to operation selection after a start point or end point is set.

- **Without specifying a program number**

In copying program and moving program, if **[EXEC]** is pressed without specifying a program number after an editing range end point is set, a program with program number 00000 is registered as a work program. This 00000 program has the following features:

- The program can be edited in the same way as a general program. (Do not run the program.)
- If a copy or move operation is newly performed, the previous information is deleted at execution time, and newly set information (all or part of the program) is reregistered. (In merge operation, the previous information is not deleted.)
- When the program becomes unnecessary, delete the program by a normal editing operation.

- **Editing when the system waiting for a program number to be entered**

When the system is waiting for a program number to be entered, no edit operation can be performed.

Restrictions

- **Number of digits for program number**

If a program number is specified by 5 or more digits, a format error is generated.

Alarm

Alarm no.	Contents
70	Memory became insufficient while copying or inserting a program. Copy or insertion is terminated.
101	The power was interrupted during copying, moving, or inserting a program and memory used for editing must be cleared.

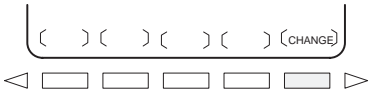
9.5.6

Replacement of Words and Addresses

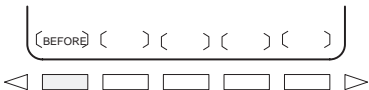
Replace one or more specified words.

Replacement can be applied to all occurrences or just one occurrence of specified words or addresses in the program.

Procedure for hange of words or addresses

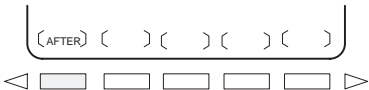


1 Perform steps 1 to 4 in subsection 9.5.1.



2 Press soft key **[CHANGE]**.

3 Enter the word or address to be replaced.



4 Press soft key **[BEFORE]**.

5 Enter the new word or address.



6 Press soft key **[AFTER]**.

7 Press soft key **[EXEC]** to replace all the specified words or addresses after the cursor.

Press soft key **[1-EXEC]** to search for and replace the first occurrence of the specified word or adress after the cursor.

Press soft key **[SKIP]** to only search for the first occurrence of the specified word or address after the cursor.

Explanations

- Replace X100 with Y200

[CHANGE] X 1 0 0 [BEFORE] Y 2 0 0
[AFTER][EXEC]

- Replace X100Y200 with X30

[CHANGE] X 1 0 0 Y 2 0 0
[BEFORE] X 3 0 [AFTER][EXEC]

- Replace IF with WHILE

[CHANGE] I F [BEFORE] W H I L E
[AFTER] [EXEC]

- Replace X with ,C10

[CHANGE] X [BEFOR] , C 1 0 [AFTER][EXEC]

Explanations

- Replacing custom macros

The following custom macro B words are replaceable:

IF, WHILE, GOTO, END, DO, BPRNT, DPRINT, POPEN, PCLOS

The abbreviations of custom macro B words can be specified.

When abbreviations are used, however, the screen displays the abbreviations as they are key input, even after soft key **[BEFORE]** and **[AFTER]** are pressed.

Restrictions

- The number of characters for replacement
- The characters for replacement

Up to 15 characters can be specified for words before or after replacement. (Sixteen or more characters cannot be specified.)

Words before or after replacement must start with a character representing an address. (A format error occurs.)

9.6 EDITING OF CUSTOM MACRO B

Unlike ordinary programs, custom macro B 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 section 10.1 for the comments of a program.

Explanations

• Editing unit

When editing a custom macro B 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 CRT 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 B word

When a custom macro B word is altered or inserted, the first two characters or more can replace the entire word.

Namely,

WHILE → WH or WD	GOTO → GO	XOR → XO	AND → AN
SIN → SI	COS → CO	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

(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.

9.7 BACKGROUND EDITING

Editing a program while executing another program is called background editing. The method of editing is the same as for ordinary editing (foreground editing).

During background editing, all programs cannot be deleted at once.

Procedure for background editing

- 1 Press function  key to display the program screen.
- 2 Press the rightmost soft key (continuous menu key), then press soft key **[BG-EDT]**.
The background editing screen is displayed (PROGRAM (BG-EDIT) is displayed at the top left of the screen).
- 3 Edit a program on the background editing screen in the same way as for ordinary program editing.
- 4 After editing is completed, press soft key **[BG-END]**.

Explanations

- **Alarms during background editing**

Alarms that may occur during background editing do not affect foreground operation. Conversely, alarms that may occur during foreground operation do not affect background editing. In background editing, if an attempt is made to edit a program selected for foreground operation, a BP/S alarm (No. 140) is raised. On the other hand, if an attempt is made to select a program subjected to background editing during foreground operation (by means of subprogram calling or program number search operation using an external signal), a P/S alarm (Nos. 059, 078) is raised in foreground operation. As with foreground program editing, P/S alarms occur in background editing. However, to distinguish these alarms from foreground alarms, BP/S is displayed in the data input line on the background editing screen.

NOTE

- 1 If the available part program storage is 80 m or less, free space in memory is used for background editing. A program to be subjected to background editing is copied into the free area in memory, then the original program is deleted. Subsequently, editing starts. Background editing can be executed if the part program storage has sufficient free area to which the target program can be copied and if program registration is allowed in terms of number. If background editing is repeated, the number of deleted areas will increase. To use these deleted areas efficiently, memory must be reorganized.
- 2 If the available part program storage is 120 m or more, background editing can be executed while leaving the registered programs as is. Such editing may create an unused area in memory. The unused area can be deleted by reorganizing the memory however.
- 3 If the reset key is pressed to abandon program input or output in background editing, the machining in the foreground will also be halted. To input or output a program in the background, therefore, use the soft keys. To halt the input or output, press the **[STOP]** soft key.
- 4 If a reset by M02/M30 of the machining program in the foreground is executed during program input or output in background editing, program input or output is halted. Program input or output can be prevented from being halted by the reset in the foreground if bit 2 of parameter 076 is specified accordingly.
- 5 In background editing, program input or output by the external activation signal (MINP) or input/output unit external control is inhibited.

9.8 REORGANIZING MEMORY


If the available part program storage is 120 m or more, or if the background editing function is supported, repeated program editing will create many small, unused areas in memory. Reorganizing memory arranges these unused areas into a single, contiguous area that can be used by programs.

Procedure for Reorganizing Memory

Procedure 1 (reset key)

Press the emergency stop, external reset, or reset key. The procedure for reorganizing memory is automatically started.

Procedure 2 (soft key)

- 1 Select **EDIT** mode.
- 2 Press the  key to display the program.
- 3 Press the **[LIB]** soft key.
- 4 Press the **[REORGANIZE]** soft key.

NOTE

- 1 One memory is reorganized, the system searches for the beginning of the selected program and the cursor is returned to that point.
- 2 If the power is turned off during the memory reorganization, alarm 101 occurs when the power is subsequently turned on. Before turning the power off after resetting an alarm, first check whether memory reorganization has been completed. While memory reorganization is being performed, EDIT blinks at the bottom right corner of the screen.
- 3 As described in procedure 1, above, the memory reorganization procedure is automatically started when a reset is performed. Memory reorganization can be prevented from being started by a reset if bit 0 of parameter 056 is specified accordingly.
- 4 Memory reorganization cannot be executed during background editing.

10

CREATING PROGRAMS



Programs can be created using any of the following methods:

- MDI keyboard
- PROGRAMMING IN TEACH IN MODE
- CONVERSATIONAL PROGRAMMING INPUT WITH GRAPHIC FUNCTION
- MENU PROGRAMMING FUNCTION
- CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION
- AUTOMATIC PROGRAM PREPARATION DEVICE (FANUC SYSTEM P)




This chapter describes creating programs using the MDI panel, TEACH IN mode, menu programming function, and conversational programming with graphic function. This chapter also describes the automatic insertion of sequence numbers.

10.1 CREATING PROGRAMS USING THE MDI PANEL

Programs can be created in the EDIT mode using the program editing functions described in Chapter 9.

Procedure for Creating Programs Using the MDI Panel

Procedure




- 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 Chapter 9.

Explanations


• Comments in a program

For the full key type MDI panel, comments can be written in a program using the control in/out codes.

Example) O0001 (FANUC SERIES 0) ;
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.

Note the following to enter a comment:






- Control-in code “)” cannot be registered by itself.
- Comments entered after the  key is pressed must not begin with a number, space, or address O.
- If an abbreviation for a macro is entered, the abbreviation is converted into a macro word and registered (see Section 9.6).
- Address O and subsequent numbers, or a space can be entered but are omitted when registered.


10.2 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 550.

Procedure for automatic insertion of sequence numbers

Procedure

- 1 Set 1 for SEQUENCE (see subsection 11.5.3).
- 2 Enter the **EDIT** mode.
- 3 Press  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.



```

PROGRAM                                00040 N0012

00040 ;
N10 G92 X0 Y0 Z0 ;
N12
%

(                                     S    0  T
19:42:18                             EDIT
[ PRGRM ][ LIB ][FLOPPY][          ][ C.A.P. ]

```

- 10 · In the example above, if N12 is not necessary in the next block, pressing the  key after N12 is displayed deletes N12.
- To insert N100 in the next block instead of N12, enter N100 and press  after N12 is displayed. N100 is registered and initial value is changed to 100.

10.3 CREATING PROGRAMS IN TEACH IN MODE








When the playback option is selected, the **TEACH IN JOG** mode and **TEACH IN HANDLE** mode are added. In these modes, a machine position along the X, Y, and Z axes obtained by manual operation is stored in memory as a program position to create a program.

The words other than X, Y, and Z, which include O, N, G, R, F, C, M, S, T, P, Q, and EOB, can be stored in memory in the same way as in **EDIT** mode.

Procedure for Creating Programs in TEACH IN Mode

Procedure

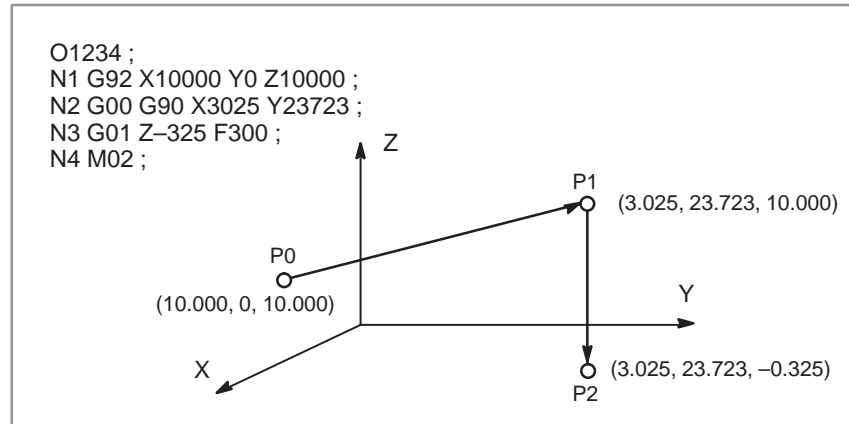
The procedure described below can be used to store a machine position along the X, Y, and Z axes.

- 1 Select the **TEACH IN JOG** mode or **TEACH IN HANDLE** mode.
- 2 Move the tool to the desired position with jog or handle.
- 3 Press  key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the machine position along each axis is to be registered (inserted).
- 4 Key in address  .
- 5 Press the  key. Then a machine position along the X axis is stored in memory.
(Example) X10.521 Machine position (for mm input)
X10521 Data stored in memory
- 6 Similarly, key in  , then press the  key. Then a machine position along the Y axis is stored in memory. Further, key in  , then press the  key. Then a machine position along the Z axis is stored in memory.

The coordinates registered in this way are coordinates without a decimal point. If the calculator-type decimal point input is selected, the decimal point is added to the coordinates.

Procedure example for creating the program in TEACH IN MODE

Explanations



- 1 Set the setting data **SEQUENCE NO.** to 1 (on). (The incremental value parameter (No. 550) is assumed to be "1".)
- 2 Select the **TEACH IN HANDLE** mode.
- 3 Make positioning at position P0 by the manual pulse generator.
- 4 Select the program screen.

- 5 Enter program number O1234 as follows:

O 1 2 3 4 INSRT

This operation registers program number O1234 in memory.

Next, press the following keys:

EOB INSRT

An EOB (;) is entered after program number O1234. Because no number is specified after N, sequence numbers are automatically inserted for N0 and the first block (N1) is registered in memory.

- 6 Enter the P0 machine position for data of the first block as follows:

G 9 2 INSRT X INSRT Y INSRT Z
 INSRT EOB INSRT

This operation registers G92X10000Y0Z10000; in memory. The automatic sequence number insertion function registers N2 of the second block in memory.

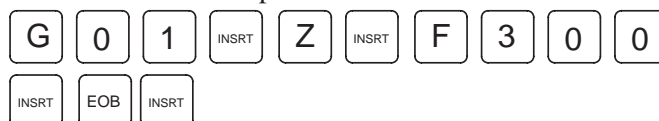
- 7 Position the tool at P1 with the manual pulse generator.
- 8 Enter the P1 machine position for data of the second block as follows:

G 0 0 INSRT G 9 0 X INSRT Y
 INSRT EOB INSRT

This operation registers G00G90X3025Z23723; in memory. The automatic sequence number insertion function registers N3 of the third block in memory.

9 Position the tool at P2 with the manual pulse generator.

10 Enter the P2 machine position for data of the third block as follows:



This operation registers G01Z -325 F300; in memory.

The automatic sequence number insertion function registers N4 of the fourth block in memory.

11 Register M02; in memory as follows:



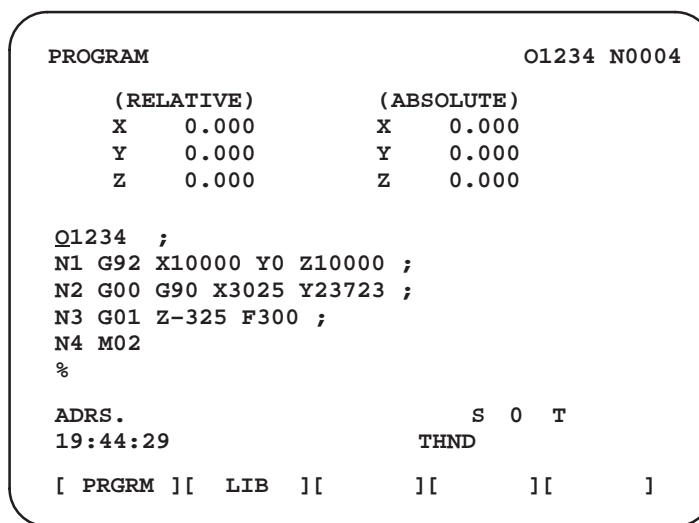
N5 indicating the fifth block is stored in memory using the automatic sequence number insertion function. Press the key to delete it.

This completes the registration of the sample program.

Explanations

- **Checking contents of the memory**

The contents of memory can be checked in the **TEACH IN** mode by using the same procedure as in **EDIT** mode.



- **Registering a position with compensation**

When a value is keyed in after keying in address , , or , then the key is pressed, the value keyed in for a machine position is added for registration. This operation is useful to correct a machine position by key-in operation.



- **Registering commands other than position commands**

Commands to be entered before and after a machine position must be entered before and after the machine position is registered, by using the same operation as program editing in **EDIT** mode.

10.4 MENU PROGRAMMING

When a program is created in EDIT mode, the G code menu is displayed on the screen.

Procedure for Menu Programming

- 1 Select EDIT mode then press the  function key. The program screen is displayed.
- 2 Press the address key . The G code menu is displayed in the lower half of the screen. For example, if G is keyed in after inserting N30, the following G code menu is displayed:

```


PROGRAM                                O0100 N0030
O0100 ;
N10 G92 X0 Y0 Z0 ;
N20 G00 G10. Y-50. ;
N30 ;
%

G00 : POSITIONING
G01 : LINEAR IPL.
G02 : CIRCULAR IPL. CW
G03 : CIRCULAR IPL. CCW
G04 : DWELL
G <

10:37:37                               S    0    T
                                           EDIT
[PRGRM ][ LIB ][FLOPPY][                ][ C.A.P. ]

```

- 3 The G code menu after G05 can be sequentially indicated by pushing the page key.


- 4 When a G code selected from the menu is input, The standard format of the one block corresponding to the G code is indicated. For example, when selecting G01, key in 0 and 1, and then press  key. G01 is inserted to the memory as shown below, and the standard format of the G01 block is indicated on the screen.





















```

PROGRAM                                O0100 N0030
O0100 ;
N10 G92 X0 Y0 Z0 ;
N20 G00 X10. Y-50. ;
N30 G01;
%

G01  X(Z,Y) --- Y(X,Z) --- F---
                                     M--- S--- T---
< 10:38:27                        EDIT S 0 T
[PRGRM ][ LIB ][FLOPPY][ ][ C.A.P. ]

```

- 5 Observing this format, enter an address and numeric value, then press the  key. Repeat this step as many times as necessary to insert a block. If the following is keyed in, for example:

the following screen is displayed:

```

PROGRAM                                O0100 N0040
O0100 ;
N10 G92 X0 Y0 Z0 ;
N20 G00 X10. Y-50. ;
N30 G01 X200. Y-60. F500 T12 ;
N40
%

< 10:39:27                        S 0 T
                        EDIT
[PRGRM ][ LIB ][FLOPPY][ ][ C.A.P. ]

```


10.5 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

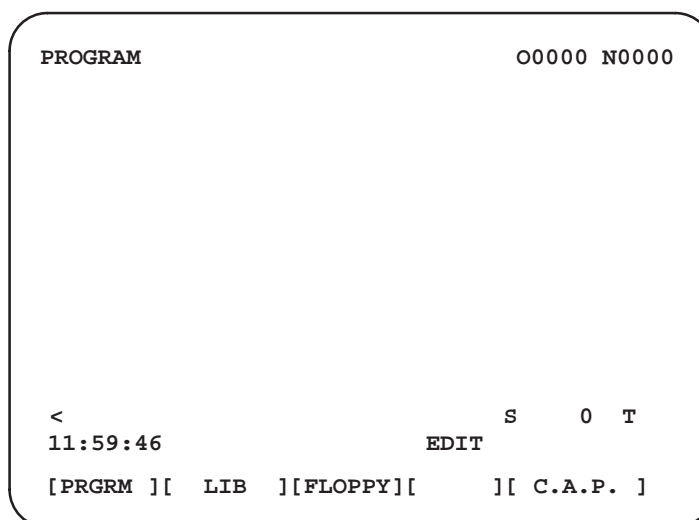
Programs can be created block after block on the conversational screen while displaying the G code menu.






Blocks in a program can be modified, inserted, or deleted using the G code menu and conversational screen.


Procedure for Conversational Programming with Graphic Function ◀ Creating a block ▶

Procedure 1

- 1 Enter the **EDIT** mode.
- 2 Press  key. If no program is registered, the following screen is displayed. If a program is registered, the program currently selected is displayed.



- 3 Key in the program number of a program to be registered after keying in address O, then press . For example, when a program with program number 10 is to be registered, key in   , then press . 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 soft keys different from those shown in step 2 are displayed, press the menu return key  to display the correct soft keys.

```

PROGRAM                                00010 N0000
G00 : POSITIONING
G01 : LINEAR IPL
G02 : CIRCULAR IPL. CW
G03 : CIRCULAR IPL. CCW
G04 : DWELL
G09 : EXACT STOP CHECK
G10 : OFFSET&TLC VALUE SETTING      (0)
G17 : XY PLANE
G18 : ZX PLANE
G19 : YZ PLANE
G20 : INCH
G21 : METRIC
ADRS.
14:26:15                             EDIT

[  MENU  ][           ][           ][           ][           ]

```

- 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 **[MENU]** to display a detailed screen for a keyed in G code. The figure below shows an example of detailed screen for G00.

```

PROGRAM                                00010 N0000
G00 : POSITIONING

      G00   G      G      G
      _X      Y
      Z
      H      OFFSET NO.
      M
      S
      T
      B
      :

      (X, Y, Z, )
      ↗
01:26:55                                EDIT
[  MENU  ][      ][      ][      ][      ]

```






When no keys are pressed, the standard details screen is displayed.

PROGRAM				O0010 N0000	
— G	G	G	G		
X		Y			
Z					
H		F			
R		M			
S		T			
B		I			
J		K			
P		Q			
L					
:					
01:28:46			EDIT		
[G.MENU]]]]]]]]]

- 7 Move the cursor to the block to be modified on the program screen.
- 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 Repeat this operation until all data required for the entered G code is entered.
- 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 leftmost soft key (the menu return key). The registered programs are converted to the conversational format and displayed.
- 12 Press the key to return to the program head.


Procedure for conversational programming with graphic function ◀ Modifying a block ▶

Procedure

- 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  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 **[MENU]**. 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 for conversational programming with graphic function ◀ Inserting a block ▶

Procedure

- 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 for conversational programming with graphic function ◀ Inserting a block ▶

Procedure

- 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.

11

SETTING AND DISPLAYING DATA

General

To operate a CNC machine tool, various data must be set on the CRT/MDI panel. 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.

This chapter describes the procedure, assuming that the soft keys are used to select a desired chapter. If the soft keys are not supported, press the key having the equivalent function two or more times to select the desired chapter.


Explanations

• Screen transition chart



MDI function keys
(Shaded keys () are described in this chapter.)

The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

See Chapter 12 for the screen that appears when function key  is pressed.

• Data protection key

The machine may have a data protection key to protect part programs. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

POSITION DISPLAY SCREEN

Screen transition triggered by the function key

POS

POS

Current position screen

(ABS) (REL) (ALL) (HNDL) ()

Position display of
work coordinate
system
⇒See subsec. 11.1.1.

Position displays
relative coordinate
system
⇒See subsec. 11.1.2.

Total position display
of each coordinate
system
⇒See subsec.11.1.3.

Manual handle in-
terruption
⇒See sec. 4.7.

Display of run time
and parts count
⇒See subsec. 11.1.5.

Display of run time
and parts count
⇒See subsec. 11.1.5.

Display of run time
and parts count
⇒See subsec. 11.1.5.

Display of run time
and parts count
⇒See subsec. 11.1.5.

Display of actual
speed
⇒See subsec. 11.1.4.

Display of actual
speed
⇒See subsec. 11.1.4.

Display of actual
speed
⇒See subsec. 11.1.4.

Display of actual
speed
⇒See subsec. 11.1.4.

Setting of relative
coordinate values
⇒See subsec. 11.1.2.

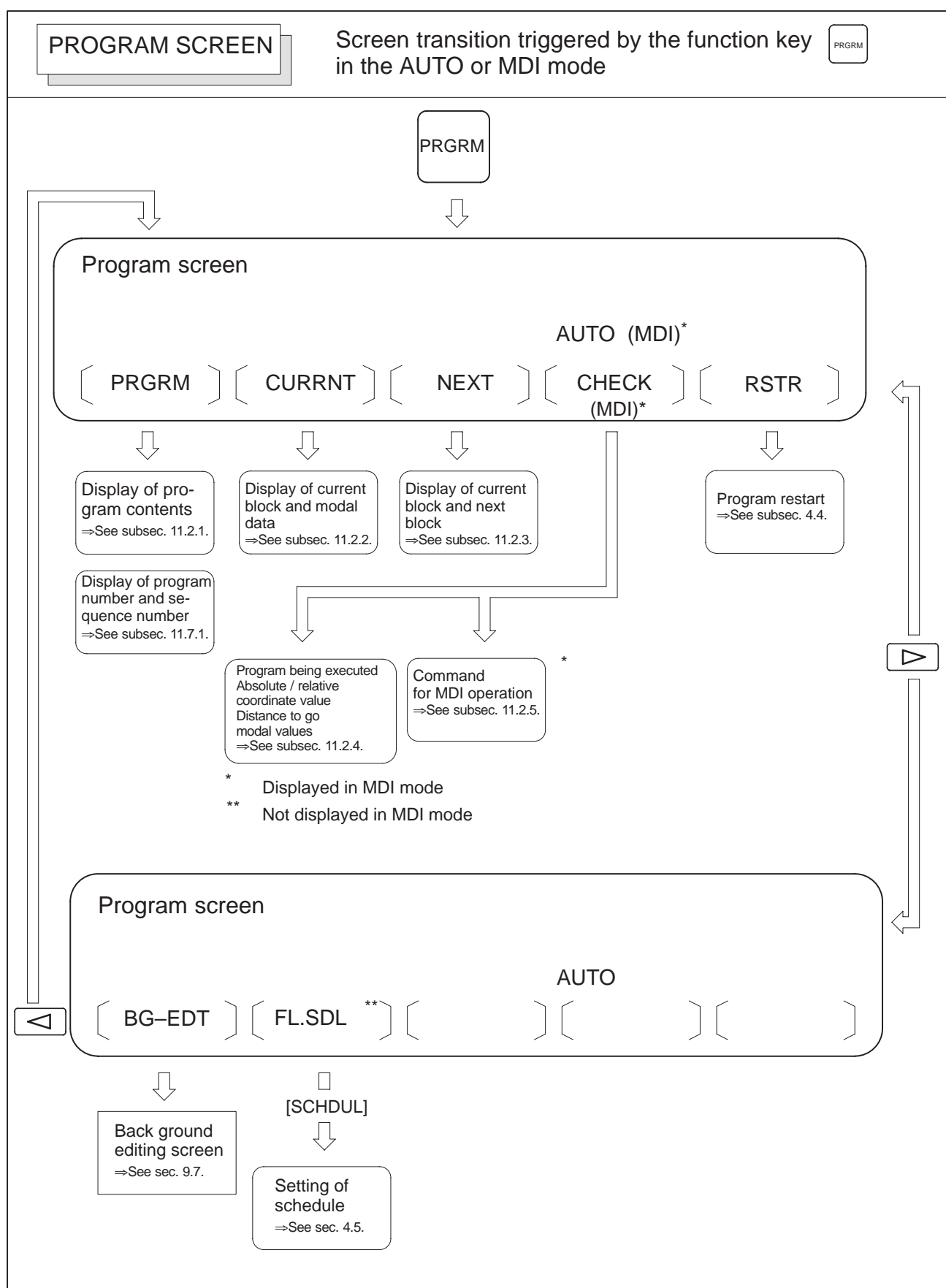
Display of
distance to go
⇒See subsec. 11.1.3.

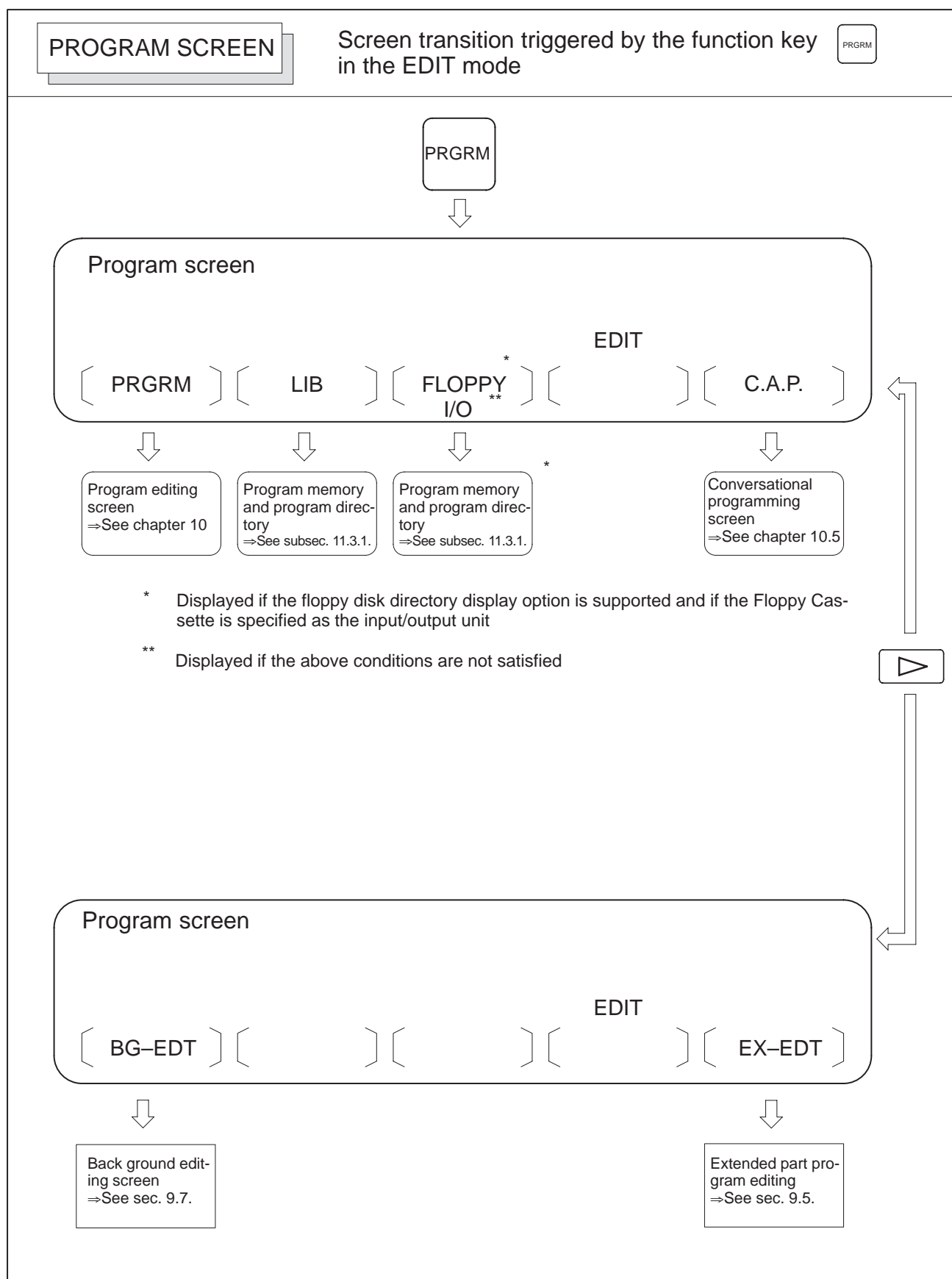
Current position screen

(MONI) () () () ()

Display of op-
erating monitor
⇒See 11.1.6

(This screen is displayed when the parameter (No. 0060#5) is set)





OFFSET SCREEN

Screen transition triggered by the function key

MENU
OFFSET

MENU
OFFSET

Tool offset value

[OFFSET] [MACRO] [MENU] [WORK] [TOOLLF]

Display of tool off-
set value
⇒See subsec. 11.4.1.

Display of
custom macro
variables
⇒See subsec. 11.4.4.

Display of pattern
menu
⇒See subsec. 11.4.5.

Display of work-
piece coordinate
system
⇒See subsec. 11.4.3.

Display of tool life
management data
⇒See subsec. 11.4.6.

Setting of tool off-
set data
⇒See subsec. 11.4.1.

Setting of macro
variables
⇒See subsec. 11.4.4.

Display of pattern
data
⇒See subsec. 11.4.5.

Setting of the work-
piece origin offset
value
⇒See subsec. 11.4.3.

Preset of tool life
counter
⇒See subsec. 11.4.6.

Tool length
measurement
⇒See subsec. 11.4.2.

Tool offset value

[] [] [] [WORK48] []

Displaying the
additional workpiece
coordinate systems
(48 systems)
⇒See subsec.
11.4.3.

Setting workpiece origin
offset value
⇒See subsec. 11.4.3.

PARAMETER/DIAGNOSTIC SCREEN

Screen transition triggered by the function key

DGNOS
PARAMDGNOS
PARAM

Parameter screen

[PARAM] [DGNOS] [] [SV-PRM*] []



Display of parameter screen
⇒see Subsec.11.5.1



Display of diagnosis screen
⇒See Sec. 7.2



Display of diagnosis screen
⇒See chapter 7



Setting of parameter
⇒see Subsec.11.5.1



Display of diagnosis screen
⇒See chapter 7

Setting of pitch error compensation data
⇒see Subsec.11.5.2

Display of setting data
⇒see Subsec. 11.5.3



Setting of setting data
⇒see Subsec. 11.5.3

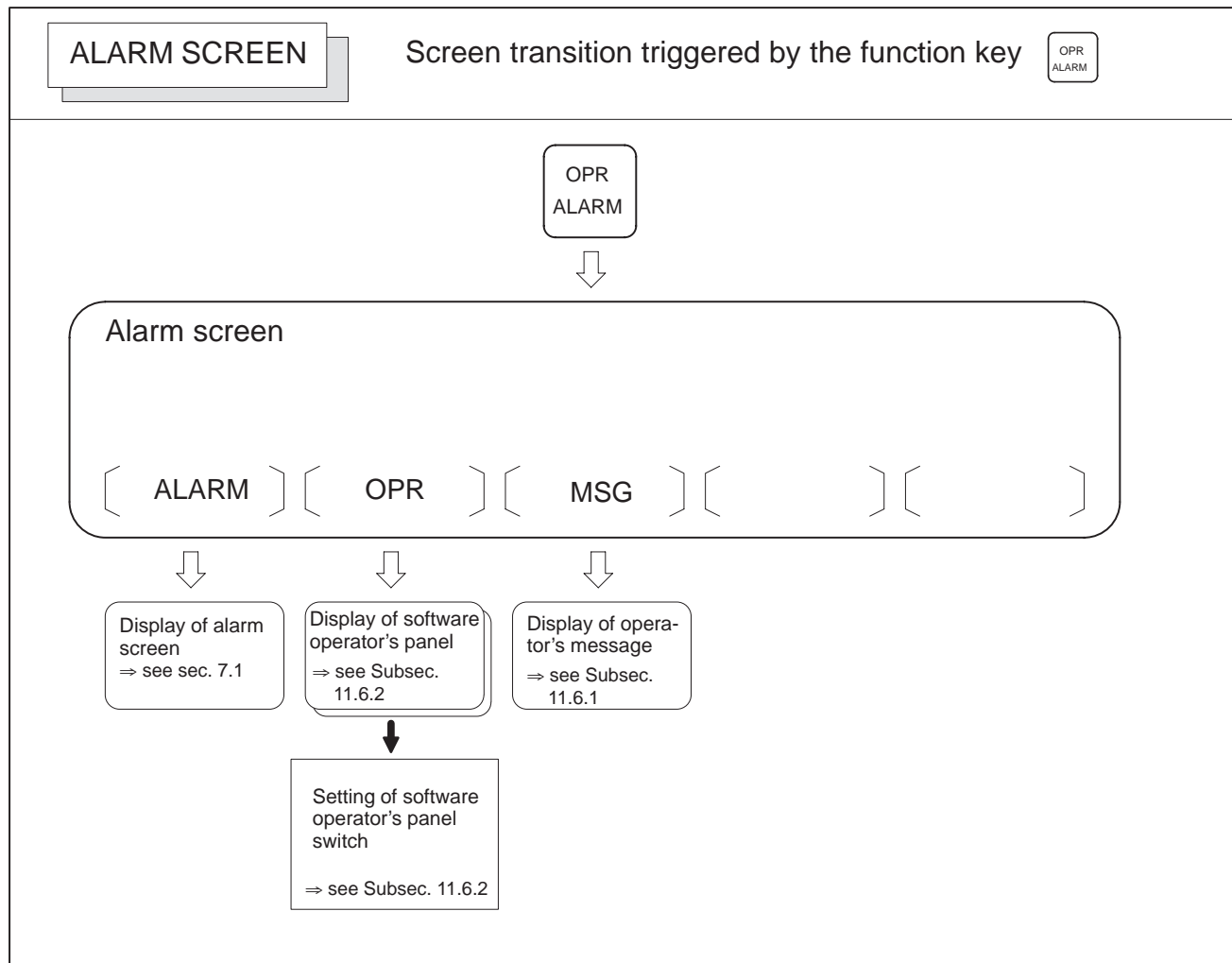
* The servo setting/adjustment screen can be suppressed if bit 0 of parameter 0389 is specified accordingly.

Setting of Sequence Number Comparison and Stop
⇒see Subsec. 11.5.4

Setting of parts count
⇒see Subsec. 11.5.5

Display of Run time, Parts count
⇒see Subsec. 11.5.5

Display of time and setting
⇒see Subsec. 11.5.5



- **Setting screens**

The table below lists the data set on each screen.

Table 11 Setting screen and data on them

No.	Setting screen	Contents of setting	Reference item
1	Tool offset value	Tool offset value Tool length offset value Cutter compensation value	Subsec. 11.4.1
		Tool length measurement	Subsec. 11.4.2
2	Setting data (handy)	Parameter write TV check Punch code Input unit (mm/inch) I/O channel Automatic insert of Sequence No. Conversion of tape format Incremental/absolute MDI travel distance	Subsec. 11.5.3
		Sequence number comparison and stop	Subsec. 11.5.4
		Mirror image	Subsec. 11.5.3
		Parts required	Subsec. 11.5.5
3	Macro variables	Custom macro common variables (#100 to #149) or (#100 to #199) (#500 to #531) or (#500 to #599)	Subsec. 11.4.4
4	Parameter	Parameter	Subsec. 11.5.1
		Pitch error compensation data	Subsec. 11.5.2
5	software operator's panel	Mode selection Jog feed axis selection Jog rapid traverse Axis selection for Manual pulse generator Multiplication for manual pulse generator Jog feedrate Feedrate override Rapid traverse override Optional block skip Single block Machine lock Dry run Protect key Feed hold	Subsec. 11.6.2
6	Tool life data (Tool life management)	Life count	Subsec. 11.4.6
7	Work coordinate system setting	Work origin offset value	Subsec. 11.4.3


11.1 SCREENS DISPLAYED BY FUNCTION KEY

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:

- Position display screen for the work coordinate system.
- Position display screen for the relative coordinate system.
- 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 screen for displaying the distance moved by handle interruption. See Section 4.7 for details on this screen.

11.1.1 Position Display in the Work 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.

- 1 Press function key POS.
- 2 Press soft key **[ABS]**.

ACTUAL POSITION (ABSOLUTE) 00010 N0000

X	123.456
Y	363.233
Z	0.000

RUN TIME	0H 1M	CYCLE TIME	0H 1M33S
ACT.F	3000 MM/M	S	0 T
01:35:22		BUF AUTO	

[ABS] [REL] [ALL] [HNDL] []

Explanations

- **Display including compensation values**

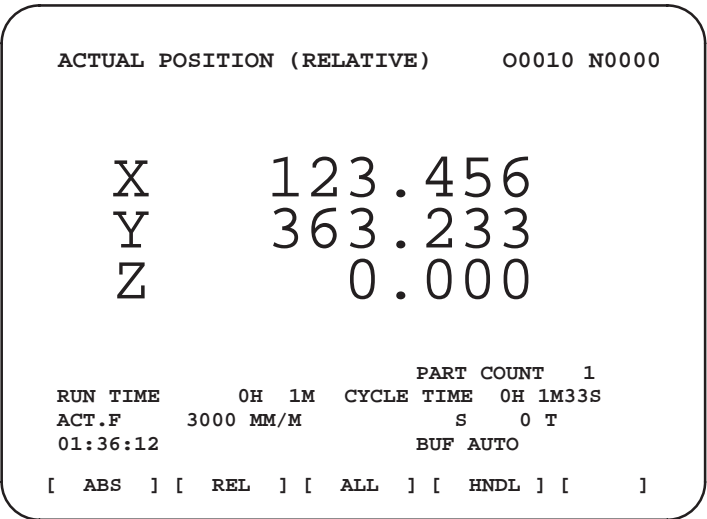
Bits 5 of parameter 018 can be used to select whether the displayed values include tool length offset.

11.1.2
Position Display in the
Relative Coordinate
System

Displays the current position of the tool in a relative coordinate system based on the coordinates 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.
On this screen, relative coordinates can be reset.

Display procedure for the current position screen with the relative coordinate system

- 1 Press function key POS .
- 2 Press soft key [REL].




Explanations


- **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:

Procedure to reset the axis coordinate to a specified value

- 1 Key in the address of the axis name (X, Y, etc.) on the relative coordinate screen. The entered axis address blinks. Two or more axis names can be input.
- 2 Press the  key. The relative coordinates of the axis having the blinking address are reset to 0.

Procedure to preset a value for a specified axis

- 1 Key in the desired axis name and value on the relative coordinate screen. The entered axis address blinks.
- 2 Press the  key. The relative coordinate of the axis with the blinking address is preset to the specified value.

To enable this operation, specify bit 0 of parameter 0064 accordingly. In this mode, a reset cannot be performed for the specified axis. To reset the coordinate, key in 0 as the preset value.

- **Display including compensation values**

Bit 1 of parameter 0001 can be used to select whether the displayed values include tool length offset and cutter compensation.

- **Presetting by setting a coordinate system**

Bit 1 of parameter 0002 is used to specify whether the displayed positions in the relative coordinate system are preset to the same values as in the workpiece coordinate system when a coordinate system is set by a G92 command or when the manual reference position return is made.

11.1.3 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.

Procedure for displaying overall position display screen

- 1 Press function key POS .
- 2 Press soft key **[ALL]**.

ACTUAL POSITION		O1000 N00010	
(RELATIVE)		(ABSOLUTE)	
X	246.912	X	123.456
Y	913.780	Y	456.890
Z	1578.246	Z	789.123
(MACHINE)		(DISTANCE TO GO)	
X	0.000	X	0.000
Y	0.000	Y	0.000
Z	0.000	Z	0.000
RUN TIME 0H 4M		PART COUNT 1	
ACT.F 0 MM/M		CYCLE TIME 0H 1M38S	
01:54:57		S 0 T	
		MDI	
[ABS]	[REL]	[ALL]	[HNDL] []

Explanations

● **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 work 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 AUTO 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 of parameter 063.

11.1.4 Actual Feedrate Display

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen by setting bit 2 of parameter 028. On a 14-inch CRT, the actual feedrate is always displayed.

Display procedure for the actual feedrate on the current position display screen

- 1 Press function key POS to display a current position display screen.

ACTUAL POSITION (ABSOLUTE) 00010 N0000

X	123.456
Y	363.233
Z	0.000

RUN TIME	0H 1M	CYCLE TIME	0H 1M33S
ACT.F	3000 MM/M	S	0 T
01:35:22	BUF AUTO		

[ABS] [REL] [ALL] [HNDL] []

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.

Explanations

- **Actual feedrate value**

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^n (fi)^2}$$

where

n : Number of axes

fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate

Fact : Actual feedrate displayed

The display unit: mm/min (metric input)
inch/min (Inch input)

- **Actual feedrate display of feed per revolution**

In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- **Actual 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 INCH/M

- **Actual feedrate display on the other screen**

The program check screen also displays the actual feedrate.

11.1.5 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

- 1 Press function key POS to display a current position display screen.

ACTUAL POSITION(RELATIVE)		00010 N0000	
X	123.456		
Y	363.233		
Z	0.000		
RUN TIME 0H15M		PART COUNT 5	
ACT.F 3000 MM/M		CYCLE TIME 0H 0M38S	
01:36:12		S 0 T0000	
		BUF AUTO	
[ABS] [REL] [ALL] [HNDL] [OPRT]			

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanations

• PART COUNT

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter 6710 is executed.

Key in address P, then press the CAN key. The value is reset to 0.

• RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

Key in address R, then press the CAN key. The value is reset to 0.

• 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 subsection 11.5.5.

• Setting of Machine parts and run time

The number of machined parts and run time cannot be set on current position display screens. They can be set on the setting screen.

• Incrementing the number of machined parts

Bit 0 (PCM) of parameter 219 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter is executed, or only each time an M code specified by parameter 6710 is executed.

11.1.6 Operating Monitor Display

This function displays the loads on the basic axes and serial spindle (first spindle). This function can also display the speed of the serial spindle (first spindle).

Procedure for manipulating the monitor display

Procedure

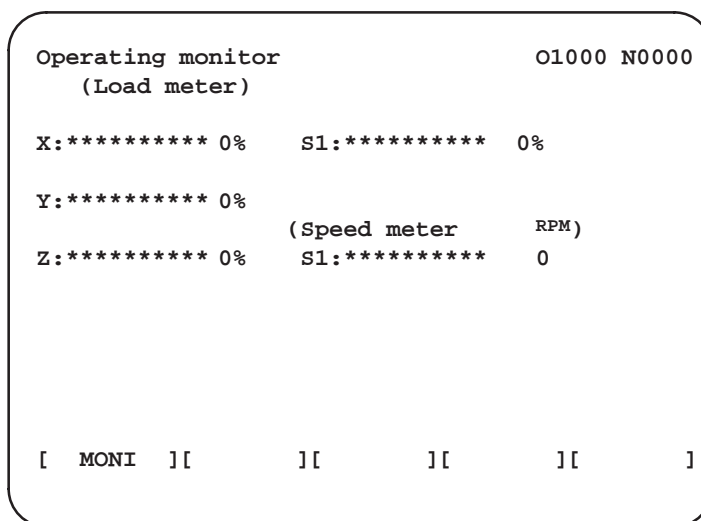
- 1 Display the current position display screen by pressing function key



- 2 Press the rightmost soft key [>].

- 3 Press the [MONI] soft key.

These steps display the operating monitor screen. The operating monitor screen can also be displayed by pressing the and page keys several times instead of performing steps 2 and 3.



- (1) Load meter (X to Z)

The load on each axis is indicated as a percentage of the rated torque and a string of asterisks (*). One asterisk represents 10%.

- (2) Load meter (S1)

The load on the first serial spindle is indicated as a percentage of the rated torque and a string of asterisks (*). One asterisk represents 20%.


- (3) Speed meter (S1)

The speed of the first serial spindle is indicated using RPM and a string of asterisks (*). One asterisk represents 10% of the maximum speed.


NOTE

- 1 Load data is displayed only for the basic axes.
- 2 Data is displayed only for the first serial spindle. No data is displayed for the second serial spindle or analog interface spindle.
- 3 Data is displayed when bit 5 of parameter No. 060 is set to 1.

11.2 SCREENS DISPLAYED BY FUNCTION KEY (IN AUTO MODE OR MDI MODE)

This section describes the screens displayed by pressing function key  in AUTO or MDI mode. The first four of the following screens display the execution state for the program currently being executed in AUTO or MDI mode and the last screen displays the command values for MDI operation in the MDI mode:

1. Program contents display screen
2. Current block display screen
3. Next block display screen
4. Program check screen
5. Program screen for MDI operation


Function key  can also be pressed in AUTO mode to display the program restart screen and scheduling screen.
See Section 4.4 for the program restart screen.
See Section 4.5 for the scheduling screen.

11.2.1 Program Contents Display

Displays the program currently being executed in AUTO mode.

Procedure for displaying the program contents

Procedure

- 1 Press function  key to display the program.
- 2 Press soft key **[PRGRM]**.
The cursor is positioned at the block currently being executed.

```

PROGRAM                                O2000 N0130
O2000 ;
N100 G92 X0 Y0 Z50. ;
N110 G91 G00 Y50. ;
N120 Z-50. ;
N130 G41 G17 H1 G01 X20. F3000 ;
N140 G02 J-25.5 ;
N150 X20. ;
N160 G02 X12.5 Y12.5 R12.5 ;
N170 G01 Y40. ;
N180 X30. Y30. ;
N190 G40 X50. ;

```

```

                                S      O T
16:59:40                        BUF AUTO
[ PRGRM ][CURRNT][ NEXT ][ CHECK ][ RSTR ]


```

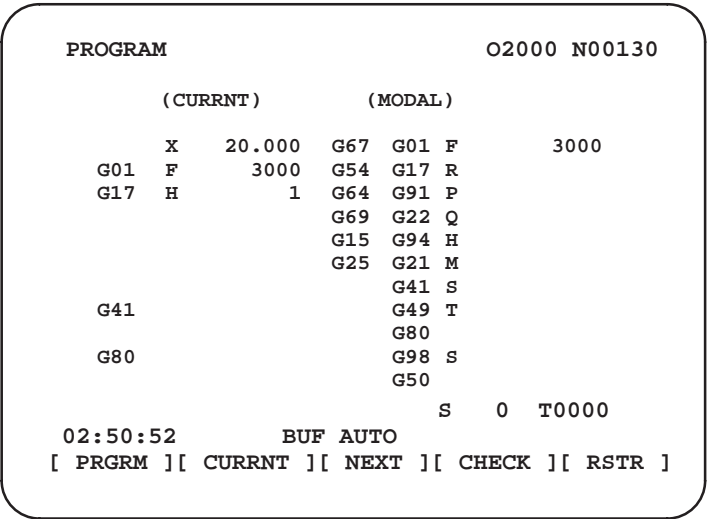

11.2.2

Current Block Display Screen

Displays the block currently being executed and modal data in the AUTO or MDI mode.

Procedure for displaying the current block display screen

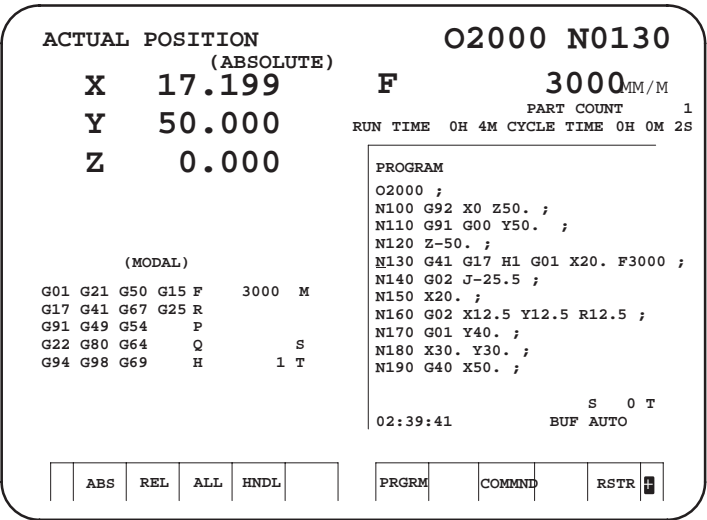
- 1 Press function key  .
- 2 Press soft key **[CURRNT]**.
The block currently being executed and modal data are displayed.



Explanations

- 14 inch CRT screen


The current block display screen is not provided for 14-inch CRTs. Press soft key **[PRGRM]** to display the contents of the program on the right half of the screen. The block currently being executed is indicated by the cursor. Modal data is displayed on the left half of the screen.

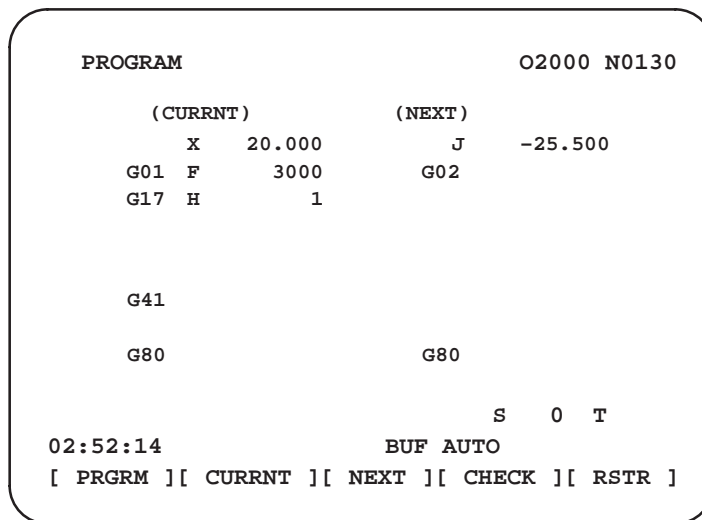


11.2.3 Next Block Display Screen

Displays the block currently being executed and the block to be executed next in the AUTO or MDI mode.

Procedure for displaying the next block display screen

- 1 Press function key  .
- 2 Press soft key **[NEXT]**.
The block currently being executed and the block to be executed next are displayed.




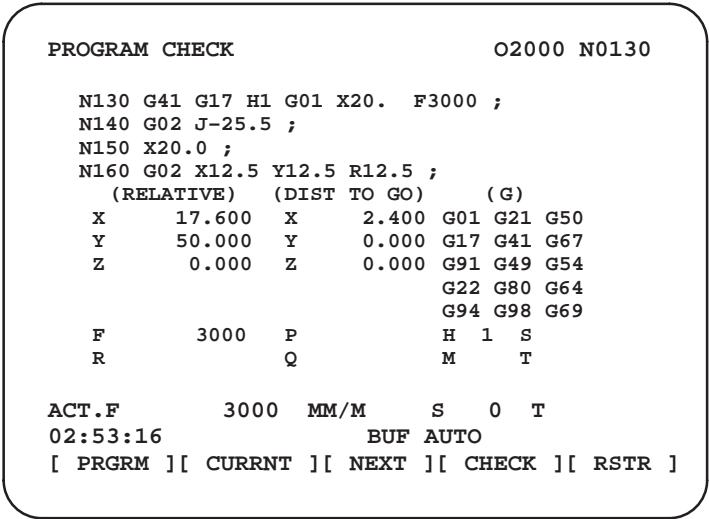
11.2.4

Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data in the AUTO mode.

Procedure for displaying the program check screen

- 1 Press function key  .
- 2 Press soft key **[CHECK]**.
The program currently being executed, current position of the tool, and modal data are displayed.



Explanations

- Program display

For the program currently being executed, the block currently being executed is displayed first.
- Current position display

The position in the workpiece coordinate system or relative coordinate system and the remaining distance are displayed. The absolute positions and relative positions are switched by parameter No. 028 #0.

- 14 inch CRT


The program check screen is not provided for 14-inch CRTs. Press soft key **[PRGRM]** to display the contents of the program on the right half of the screen. The block currently being executed is indicated by the cursor. The current position of the tool and modal data are displayed on the left half of the screen.

ACTUAL POSITION		O2000 N0130	
(ABSOLUTE)			
X	17.199	F	3000 MM/M
Y	50.000	PART COUNT 1	
Z	0.000	RUN TIME 0H 4M CYCLE TIME 0H 0M 2S	
(MODAL)		PROGRAM	
G01 G21 G50 G15 F 3000 M		O2000 ;	
G17 G41 G67 G25 R		N100 G92 X0 Z50. ;	
G91 G49 G54 P		N110 G91 G00 Y50. ;	
G22 G80 G64 Q S		N120 Z-50. ;	
G94 G98 G69 H 1 T		N130 G41 G17 H1 G01 X20. F3000 ;	
		N140 G02 J-25.5 ;	
		N150 X20. ;	
		N160 G02 X12.5 Y12.5 R12.5 ;	
		N170 G01 Y40. ;	
		N180 X30. Y30. ;	
		N190 G40 X50. ;	
		S 0 T	
		02:39:41 BUF AUTO	
ABS	REL	ALL	HNDL
PRGRM	COMMND	RSTR	

11.2.5
Program Screen for
MDI Operation

Displays the program input from the MDI and modal data in the MDI mode.

Procedure for displaying the program screen for MDI operation

- 1 Press function key  .
 - 2 Press soft key **[MDI]**.
The program input from the MDI and modal data are displayed.
- (1) MDI operation – A

PROGRAM

O2000 N0130

(MDI)

X 10.500

Y 200.500

(MODAL)

F 2000

G00 R

G17 P

G90 Q

G94 H 2

G21 M

G40 S 15

G49 T 02

G80

G98

G67

ADDRESS

MDI

[PRGRM][CURRNT][NEXT][MDI][]

(2) MDI operation – B

PROGRAM (MDI)

O1234 N5678

Q0000 G00 X100. Y200. ;

M03 ;

G01 Z120. F500 ;

M98 P9010 ;

G00 Z0 ;

%

(MODAL)

G00 G90 G94 G40 G80 G50 G54 G69

G17 G22 G21 G49 G98 G67 G64

F P H S

R Q M T

< S 0 T

11:04:44

MDI

[PRGRM][CURRNT][NEXT][MDI][RSTR]

Explanations




- **MDI operation**

See Section 4.2 for MDI operation.

- **Modal information**

The modal data is displayed when bit 7 (MDL) of parameter 3107 is set to 1. On a 14-inch CRT, however, the contents of the program are displayed on the right half of the screen and the modal data is displayed on the left half of the screen, regardless of this parameter.

11.3 SCREENS DISPLAYED BY FUNCTION KEY (IN THE EDIT MODE)


This section describes the screens displayed by pressing function key  in the EDIT mode. Function key  in the EDIT mode can display the program editing screen and the library screen (displays memory used and a list of programs). Pressing function key  in the EDIT mode can also display the conversational graphics programming screen and the floppy file directory screen. See Chapter 9 for the program editing screen and conversational graphics programming screen. See Chapter 8 for the floppy file directory screen.

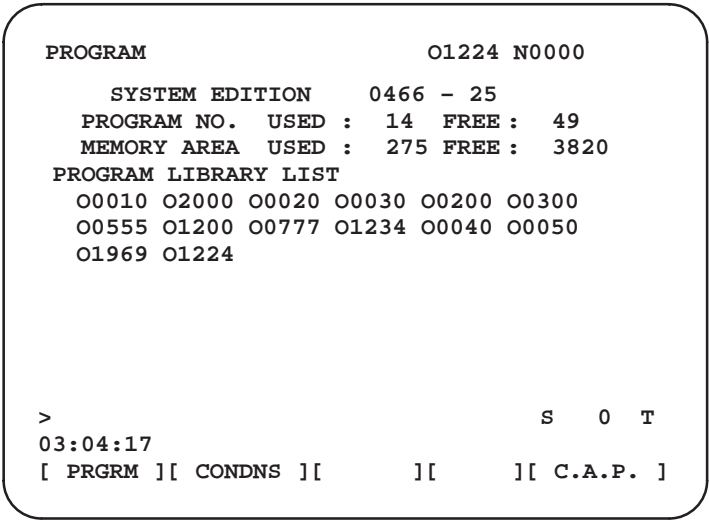
11.3.1

Displaying Memory
Used and a List of
Programs

Displays the number of registered programs, memory used, and a list of registered programs.

Procedure for displaying memory used and a list of programs

- 1 Select the **EDIT** mode.
- 2 Press function key  .
- 3 Press soft key [**LIB**].



Explanations

- Details of memory used
- PROGRAM NO. USED**

PROGRAM NO. USED : The number of the programs registered (including the subprograms)
- FREE*

: The number of programs which can be registered additionally.
- MEMORY AREA USED**

MEMORY AREA USED : The capacity of the program memory in which data is registered (indicated by the number of characters).
- FREE*

: The capacity of the program memory which can be used additionally (indicated by the number of characters).

● Program library list

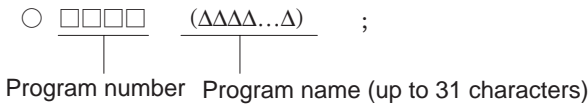
Program Nos. registered are indicated.
Also, the program name can be displayed in the program table by setting parameter No. 040#0.

```
PROGRAM                                01224 N0000
  SYSTEM EDITION                      0466 - 25
  PROGRAM NO.   USED :   14   FREE :   49
  MEMORY AREA  USED :  275   FREE :  3820
PROGRAM LIBRARY LIST
00010 (MACRO-GCODE.MAIN)
02000 (MACRO GCODE.SUB1)
00020 (TEST-PROGRAM.NO-1)
00030 (TEST-PROGRAM.F10-MACRO)
00200 (TEST-PROGRAM.OFFSET)
00300
00555 (INCH/MM CONVERT CHECK NO-1)
01200 (MACRO-MCODE.MAIN)

>
03:05:57   EDIT                      S    0    T
[ PRGRM ][ CONDNS ][           ][           ][ C.A.P. ]
```

● Program name

Always enter a program name between the control out and control in codes immediately after the program number.
Up to 31 characters can be used for naming a program within the parentheses. If 31 characters are exceeded, the exceeded characters are not displayed.
Only program number is displayed for the program without any program name.



● Software series

Software series of the system is displayed.
It is used for maintenance ; user is not required this information.

● Order in which programs are displayed in the program library list


Programs are displayed in the same order that they are registered in the program library list. However, if bit 4 of parameter 040 is set to 1, programs are displayed in the order of program number starting from the smallest one.

● Free area

Any unused areas created by background editing are counted as used program and used memory areas. When memory is reorganized (refer to Sec. 9.8), these unused areas are counted as free areas.

11.4 SCREENS DISPLAYED BY FUNCTION KEY



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 offset value
2. Workpiece origin offset value
3. Custom macro common variables
4. Pattern menu and pattern data
5. Tool life management data


This section also describes measurement of tool length and the sequence number comparison and stop function.

The pattern menu, pattern data, and tool life management data depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

11.4.1 Setting and Displaying the Tool Offset Value

Tool offset values, tool length offset values, and cutter compensation values are specified by D codes or H codes in a program. Compensation values corresponding to D codes or H codes are displayed or set on the screen.

Procedure for setting and displaying the cutter compensation value

- 1 Press function key  .
- 2 Press soft key **[OFFSET]**.
The screen varies according to the type of tool offset memory.

OFFSET		O1224 N0000	
NO.	DATA	NO.	DATA
001		009	0.000
002	0.000	010	12.269
003	5.000	011	10.230
004	0.000	012	-11.265
005	12.580	013	-8.562
006	0.000	014	0.000
007	0.000	015	0.000
008	0.000	016	0.000
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000		
NO. 013 = S O T			
03:22:13 MDI			
[OFFSET][MACRO][MENU][WORK][TOOLLF]			

Tool offset memory A

OFFSET / WEAR		O0000 N0000	
NO.	DATA	NO.	DATA
HW 001		HW 0.009	0.000
HW 002	0.000	HW 0.010	12.269
HW 003	5.000	HW 0.011	10.230
HW 004	0.000	HW 0.012	-11.265
HW 005	12.580	HW 0.013	-8.562
HW 006	0.000	HW 0.014	0.000
HW 007	0.000	HW 0.015	0.000
HW 008	0.000	HW 0.016	0.000
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000		
NO. 013 = S O T			
11:29:58 MDI			
[GEOM][MACRO][MENU][WORK][TOOLLF]			

Tool offset memory B

- 3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys. Press key and enter the compensation number for the compensation value to be set or changed and then press key.
- 4 Enter a compensation value, and key.

Explanations

- **Decimal point input**

A decimal point can be used when entering a compensation value.

- **Other method**

An external input/output device can be used to input or output a cutter compensation value. See Chapter 9. A tool length offset value can be set by measuring the tool length as described in the next subsection.

- **Tool offset memory**

There are tool offset memories A, B, and C, which are classified as follows:

Tool offset memory A

Tool geometry compensation and tool wear compensation are treated the same.

Tool offset memory B

D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated differently.

Tool offset memory C

Offset memory is divided into two areas: one area for tool length offset data (H), and the other for cutter compensation data (D).

- **Disabling entry of compensation values**

The entry of compensation values may be disabled by setting bit 0 and 1 of parameter 078 (not applied to tool offset memory A).

- **Incremental input**

The current compensation value can be incremented or decremented if bit 4 of parameter 001 is specified accordingly. If this is selected, key in a desired increment or decrement.

11.4.2 Tool Length Measurement

The length of the tool can be measured and registered as the tool length offset value by moving the reference tool and the tool to be measured until they touch the specified position on the machine.
The tool length can be measured along the X-, Y-, or Z-axis.

Procedure for tool length measurement

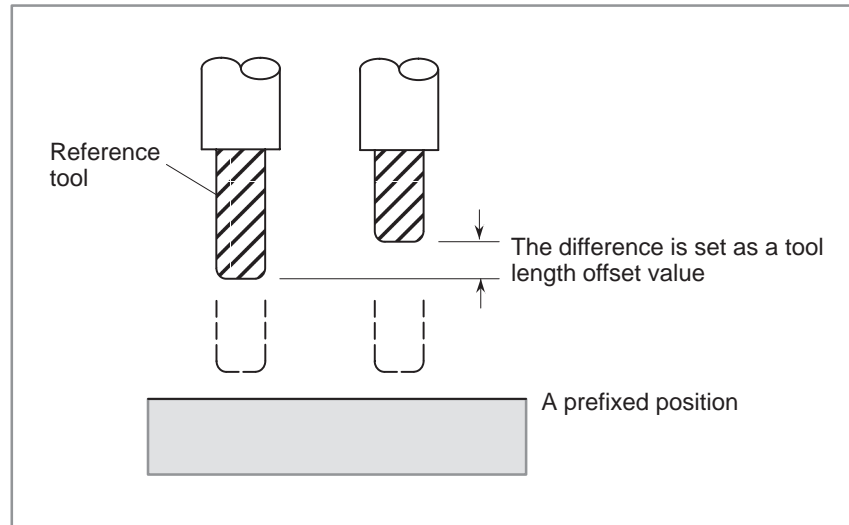
- 1 Use manual operation to move the reference tool until it touches the specified position on the machine (or workpiece.)
- 2 Press function key **POS** until the current position display screen with relative coordinates is displayed.

ACTUAL POSITION(RELATIVE)		O1000 N00010	
X	19.690		
Y	12.240		
Z	146.550		
RUN TIME 0H 4M		PART COUNT 17	
ACT.F 0 MM/M		CYCLE TIME 0H 0M 2S	
03:29:10		S 0 T	
		HNDL	
[ABS] [REL] [ALL] [HNDL] []			

- 3 Reset the relative coordinate for the Z-axis to 0 (see subsection 11.1.2 for details).
- 4 Press function key **MENU OFFSET** until the tool compensation screen is displayed.
- 5 Use manual operation to move the tool to be measured until it touches the same specified position. The difference between the length of the reference tool and the tool to be measured is displayed in the relative coordinates on the screen.
- 6 Move the cursor to the compensation number for the target tool (the cursor can be moved in the same way as for setting tool compensation values).
- 7 While holding down the **EOB** key, key in address **Z**.

If either **X** or **Y** key is depressed instead of **Z** key, the X or Y axis relative coordinate value is input as an tool length compensation value.


- 8 Press the key. The Z axis relative coordinate value is input and displayed as an tool length offset value.



11.4.3 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59 and G54 P1 to G54 P48) and external workpiece origin offset. The workpiece origin offset and external workpiece origin offset can be set on this screen.



Procedure for Displaying and Setting the Workpiece Origin Offset Value



- 1 Press function key  .
- 2 Press soft key **[WORK]** or **[WORK48]**.
The workpiece coordinate system setting screen is displayed.


WORK COORDINATES				O0001 N00000			
NO.		(SHIFT)		NO.		(G55)	
_00	X			02	X	0.000	
	Y	0.000			Y	0.000	
	Z	0.000			Z	0.000	
NO.		(G54)		NO.		(G56)	
01	X	0.000		03	X	0.000	
	Y	0.000			Y	0.000	
	Z	0.000			Z	0.000	
ADRS.				S O T			
03:36:12				MDI			
[WEAR][MACRO][MENU][WORK][TOOLLF]							

WORK COORDINATES				O1224 N0000			
P : 01		(G54)		P : 03		(G54)	
_01	X	0.000		03	X	0.000	
	Y	0.000			Y	0.000	
	Z	0.000			Z	0.000	
P : 02		(G54)		P : 04		(G54)	
02	X	0.000		04	X	0.000	
	Y	0.000			Y	0.000	
	Z	0.000			Z	0.000	
ADRS.				S O T			
03:38:13				MDI			
[][][][WORK48][]							

- 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 up  or page down  key.


Press  key and enter the workpiece coordinate system number (0: external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59, 1 to 48: workpiece coordinate systems G54 P1 to G54 P48), and then press  key.

- 4** Move the cursor to the workpiece origin offset to be changed.
- 5** Press the address key corresponding to the desired axis.
- 6** Enter a desired value by pressing numeric keys, then press  key .
The entered value is specified in the the workpiece origin offset value.
- 7** Repeat **4** and **6** to change other offset values.















11.4.4 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999). When the absolute value for a common variable exceeds 99999999, ***** is displayed. 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

- 1 Press function key  .
- 2 Press the soft key **[MACRO]**. The following screen is displayed:

VARIABLE		O1224 N0000	
NO.	DATA	NO.	DATA
100	00010.000	108	
101	00020.000	109	02564.146
102	00156.870	110	
103	0.000	111	25000.000
104	0.000	112	0.000
105	01523.579	113	
106	0.000	114	30000.000
107	00458.256	115	
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000		
NO. 114 =		S	O T
03:40:20		MDI	
[WEAR][MACRO][MENU][WORK][TOOLLF]			

- 3 Move the cursor to the variable number to set using either of the following methods:
 - Press the  key and enter the variable number. And then press  key.
 - 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  key.
- 5 To set a relative coordinate in a variable, press address key , , or , while holding down the  key. Then press  key.
- 6 To set a blank in a variable, just press  key after  key. The value field for the variable becomes blank. (Only for custom macro B)

11.4.5 Displaying Pattern Data and Pattern Menu

This subsection uses an example to describe how to display or set machining menus (pattern menus) created by the machine tool builder. Refer to the manual issued by the machine tool builder for the actual pattern menus and pattern data. See PROGRAMMING for the pattern data entry function.

Procedure for displaying the pattern data and the pattern menu

- 1 Press function key  .
- 2 Press the soft key **[MENU]**.
The following screen (pattern menu screen) is displayed:

```

MENU : HOLE PATTERN                                O9501 N0014

  1.  BOLT HOLE
  2.  GRID
  3.  LINE ANGLE
  4.  TAPPING
  5.  DRILLING
  6.  BORING
  7.  POCKET
  8.  PECK
  9.  TEST PATTERN
 10.  BACK

SELECT =                                           S      O  T
10:01:29                                           MDI
[ WEAR ][ MACRO ][ MENU ][ WORK ][ TOOLLF ]

```

- 3 Enter a pattern number and press  key.

In this example, press  , then press  key.

The following screen (pattern data screen) is displayed:

```


VAR. : BOLT HOLE                                O9501 N0014

NO.   NAME      DATA      COMMENT
500   TOOL      0
501   KIJUN X   0          *BOLT HOLE
502   KIJUN Y   0          CIRCLE*
503   RADIUS    0          SET PATTERN
504   S. ANGL   0          DATA TO VAR.
505   HOLES NO. 0          NO.500-505.
506                                     0
507                                     0

ACTUAL POSITION (RELATIVE)
      X      0.000      Y      0.000
      Z      0.000

NO. 500=                                           S      O  T
10:00:48                                           MDI
[ WEAR ][MACRO][ MENU ][ WORK ][ TOOLLF ]

```

- 4 Enter necessary pattern data and press .
- 5 After entering all necessary data, enter the **AUTO** mode and press the cycle start button to start machining.

Explanations

- **Explanation of the pattern menu screen**

HOLE PATTERN : Menu title

An optional character string can be displayed within 12 characters.

BOLT HOLE : Pattern name

An optional character string can be displayed within 10 characters.

The machine tool builder should program character strings of menu title and pattern name by custom macro, and load them into the program memory.

- **Explanation of the pattern data screen**

BOLT HOLE : Pattern data title

An optional character string can be displayed within 12 characters.

TOOL : Variable name

An optional character string can be displayed within 10 characters.

BOLT HOLE CIRCLE : Comment statement

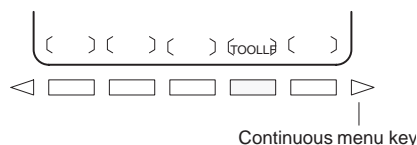
An optional character string can be displayed up to 12 characters/line by 8 lines.

The machine tool builder should program the character strings of variable name and comment statement by custom macro, and load them into the program memory.

11.4.6 Displaying and Setting Tool Life Management Data

Tool life data can be displayed to inform the operator of the current state of tool life management. Groups which require tool changes are also displayed. The tool life counter for each group can be preset to an arbitrary value. To register or modify tool life management data, a program must be created and executed. See Explanations in this section for details.

Procedure for display and setting the tool life management data



- 1 Press function key .
- 2 Press soft key **[TOOLLF]**.
- 3 One page displays data on two groups. Pressing page key or successively displays data on the following groups. Up to four group Nos., for which the Tool Change signal is being issued, are displayed at the bottom of each page. An arrow shown in the figure is displayed for five or more groups, if exists.

```

TOOL LIFE DATA :                O3000 N00060
                        SELECTED GROUP 000
GROUP 001 :  LIFE  0150  COUNT  0007
0034  0078    0012    0056
0090  0035    0026    0061
0000  0000    0000    0000
0000  0000    0000    0000

GROUP 002 :  LIFE  1400  COUNT  0000
0062  0024    0044    0074
0000  0000    0000    0000
0000  0000    0000    0000
0000  0000    0000    0000

TO BE CHANGED : 003 004 005 006 --->

NUM.                S  O  T
04:01:12            AUTO
[ WEAR ][ MACRO ][ MENU ][ WORK ][ TOOLLF ]

```

- 4 Press key and to display the page containing the data for a group, enter the group number and press key.
The cursor can be moved to an arbitrary group by pressing cursor key or .
- 5 To change the life count of a group, position the cursor to that group, key in a desired count, then press the key.

- 6 To reset the tool data of a group, position the cursor to that group, key in -9999, then press the key. All current execution data for the group selected with the cursor is cleared and the tool is considered as not being used.

Explanations

• Display contents

TOOL LIFE DATA :				O3000 N00060			
SELECTED GROUP 000							
__GROUP	001 :	LIFE	0150	COUNT	0007		
	0034	0078	0012	0056			
	0090	0035	0026	0061			
	0000	0000	0000	0000			
	0000	0000	0000	0000			
GROUP 002 :		LIFE	1400	COUNT	0000		
	0062	0024	0044	0074			
	0000	0000	0000	0000			
	0000	0000	0000	0000			
	0000	0000	0000	0000			
TO BE CHANGED		:	003	004	005	006	--->
NUM.				S O T			
04:01:12				AUTO			
[WEAR][MACRO][MENU][WORK][TOOLLF]							


- The first line is the title line.
- In the second line the group number of the current command is displayed.
When there is no group number of the current command, 0 is displayed.
- In lines 3 to 7 the tool life data of the group is displayed.
The third line displays group number, life and the count used.
The life count is chosen by parameter (No. 039#2) as either minutes(or hours) or number of times used.
In lines 4 to 5, tool numbers are displayed. In this case, the tool is selected in the order, 0034 → 0078 → 0012 → 0056 → 0090 ...
The meaning of each mark before the tool numbers is :
* : Shows the life has finished.
: Shows that the skip command has been accepted.
@ : Shows that the tool is currently being used.
The life counter counts for tool with @.
“*” is displayed when the next command is issued by the group to which it belongs.
- Lines 8 to 12 are next group life data to the group displayed in lines 3 to 7.
- In the thirteenth line the group number when the tool change signal is being emitted is displayed. The group number display appears in ascending order. When it cannot be completely displayed, “—>” is displayed.

11.5 SCREENS DISPLAYED BY FUNCTION KEY



Parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.

This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the Handy File (see Chapter 8).

If the  function key is pressed, the following data can be displayed and set:

- Pitch error compensation data
- Setting data
- Sequence number compensation and stop
- Run time and parts count
- Self-diagnostic data


See Chapter 7 for the diagnostic screens.

11.5.1 Displaying and Setting Parameters

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


- 1 When set the parameter, set 1 for **PARAMETER WRITE (PWE)** to enable writing. See the procedure for enabling/disabling parameter writing described below.
- 2 Press function key  .
- 3 Press soft key **[PARAM]** to display the parameter screen.

PARAMETER		O1224 N0000	
NO.	DATA	NO.	DATA
0001	00000000	0011	00000000
0002	00000011	0012	00000000
0003	00000000	0013	00000000
0004	01110111	0014	00000100
0005	01110111	0015	00000000
0006	01110111	0016	00000000
0007	00000000	0017	01111111
0008	00000000	0018	00000000
0009	00000000	0019	10000000
0010	10000000	0020	00000000

NO. 0001 =	S O T
04:02:02	MDI
[PARAM] [DGNOS] [] [SV-PRM] []

- 4 Move the cursor to the parameter number to be set or displayed in either of the following ways:
 - Press the  key and enter the variable number. And then press  key.
 - 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  key. The parameter is set to the entered value and the value is displayed.
- 6 When the parameter is set, set 0 for **PARAMETER WRITE (PWE)** to disable writing.

Procedure for enabling/displaying parameter writing

- 1 Select the **MDI** mode or enter state emergency stop.
- 2 Press function key  .
- 3 Press soft key [**PARAM**] to display the setting screen.

PARAMETER
O1224 N0000

(SETTING 2)

PWE = 1 (0:DISABLE 1:ENABLE)

REV4 = 0

TAPEF = 0

(SEQUENCE STOP)

PRGNO = 0

SEQNO = 0

PART TOTAL = 17

PART REQUIRED = 50






PART COUNT = 17

RUN TIME 0H 4M CYCLE TIME 0H 0M 2S

NO. PWE = = S 0 T

03:35:07

[PARAM][DGNOS][] [SV-PRM][]

- 4 Move the cursor to **PARAMETER WRITE (PWE)** using cursor keys.
- 5 Press , then press  key to enable parameter writing.
At this time, the CNC enters the P/S alarm state (No. 100).
- 6 After setting parameters, move the cursor to **PARAMETER WRITE (PWE)** and press  key, then press  key.
- 7 Depress the  key to release the alarm condition. If alarm No. 000 has occurred, however, turn off the power supply and then turn it on, otherwise the alarm is not released.

Explanations

- **Setting parameters with external input/output devices**
- **Parameters that require turning off the power**

See Chapter 9 for setting parameters with external input/output devices such as the Handy File.

Parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm 000. In this case, turn off the power, then turn it on again.

11.5.2 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated 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.

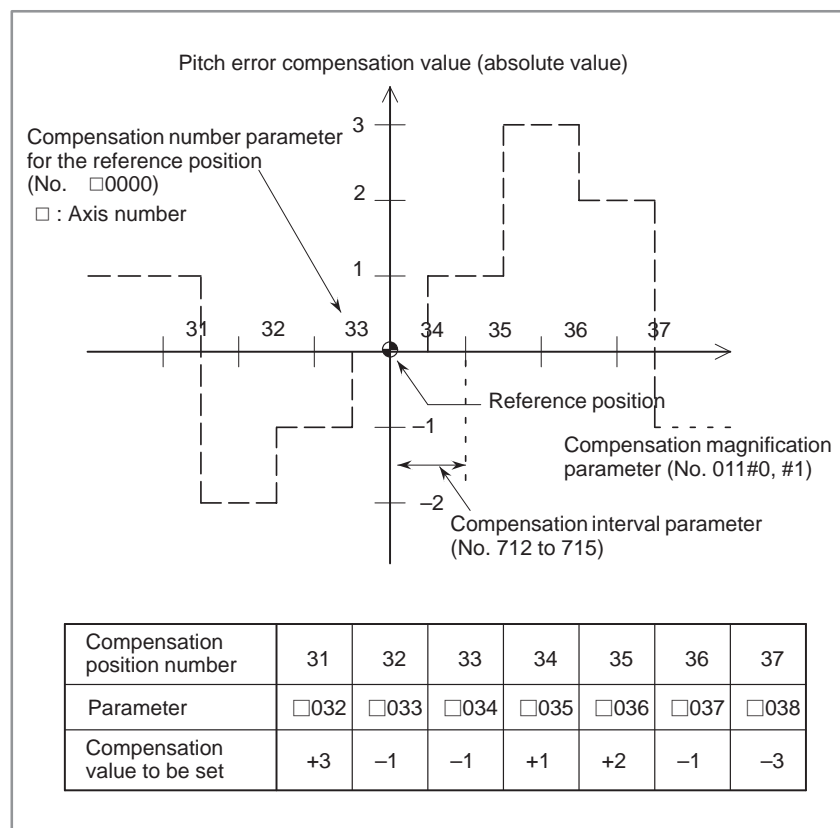
The pitch error compensation data is set according to the characteristics of the machine. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced.

In principle, the end user must not alter this data.

Pitch error compensation data can be set with external devices such as the Handy File (see Chapter 9). Compensation data can also be written directly with the MDI panel.

The following parameters must be set for pitch error compensation. Set the pitch error compensation value 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 (□ : axis number) : Parameter No. □000
- Pitch error compensation magnification : Parameter No. 011#0, #1
- Interval of the pitch error compensation points : Parameter No. 0712 to 0715
- Setting compensation value : Parameter No. □001 + compensation points

Explanations

- **Compensation point number** 128 compensation points from No. 0 to 127 are available for each axis. Specify the compensation number for the reference position of each axis in the corresponding parameter (Parameter n000, n: axis number).
- **Compensation value** Specify the compensation value in the corresponding parameter (Parameter n001 + compensation point number, n: axis number).

Restrictions

- **Compensation value range** Compensation values can be set within the range from $-7 \times$ compensation magnification (output unit) to $+7 \times$ compensation magnification (detection unit). The compensation magnification can be set 1, 2, 4, 8 in parameter No. 011#0, #1. The units of the compensation value can be changed to the detection units if bit 7 of parameter 035 is specified accordingly.
- **Intervals of compensation points** The pitch error compensation points are arranged with equally spaced. Set the space between two adjacent positions for each axis to the parameter (No. 712 to 715).
Valid data range is 0 to 99999999.
The minimum interval between pitch error compensation points is limited and obtained from the following equation:
Minimum interval of pitch error compensation points = maximum feedrate (rapid traverse rate) / 7500
Unit: mm, inches, deg, and mm/min, inches/min, deg/min
- **Pitch error compensation of the rotary axis** For the rotating axis, the interval between the pitch error compensation points shall be set to one per integer of the amount of movement (normally 360°) per rotation. The sum of all pitch error compensation amounts per rotation must be made to 0.
- **Conditions where pitch error compensation is not performed** Note that the pitch error is not compensated in the following cases:
 - When the machine is not returned to the reference position after turning on the power. If an absolute-position detector is provided and if the reference position has been determined, pitch error compensation is carried out.
 - If the interval between the pitch error compensation points is 0.

Explanations

• For linear axis (X axis)

· Machine stroke: -400 mm to +800 mm

· Interval between the pitch error compensation points: 50 mm

· No. of the compensation point of the reference position: 40

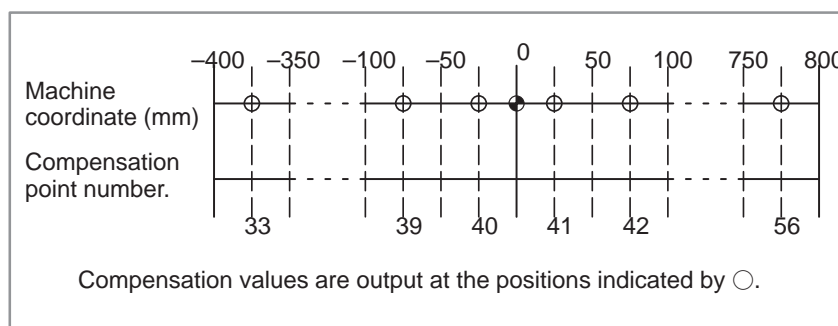
If the above is specified, the No. of the farthest compensation point in the negative direction is as follows:

No. of the compensation point of the reference position - (Machine stroke on the negative side/Interval between the compensation points) + 1
 $= 40 - 400/50 + 1 = 33$

No. of the farthest compensation point in the positive direction is as follows:

No. of the compensation point of the reference position + (Machine stroke on the positive side/Interval between the compensation points)
 $= 40 + 800/50 = 56$

The correspondence between the machine coordinate and the compensation point No. is as follows:



Therefore, set the parameters as follows:

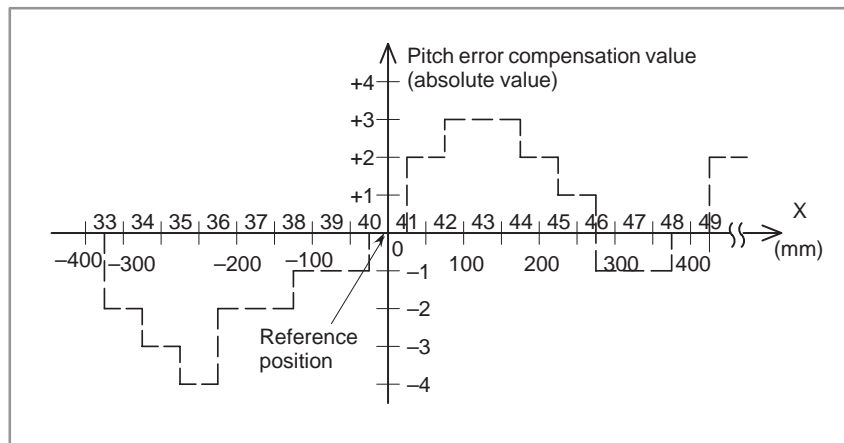
Parameter	Setting value
1000 : Compensation number for the reference position	40
011#0, #1 : Compensation magnification	#0=0, #1=0
0712 : Interval between pitch error compensation points	50000

The compensation amount is output at the compensation point No. corresponding to each section between the coordinates.

The following is an example of the compensation amounts.

Compensation position number	33	34	35	36	37	38	39	40	41
Parameter	1034	1035	1036	1037	1038	1039	1040	1041	1042
Compensation value	+2	+1	+1	-2	0	-1	0	-1	+2

Compensation position number	42	43	44	45	46	47	48	49	56
Parameter	1043	1044	1045	1046	1047	1048	1049	1050	1057
Compensation value	+1	0	-1	-1	-2	0	+1	+2	1



• For rotary axis (C axis)

· Amount of movement per rotation: 360°

· Interval between pitch error compensation points: 45°

· No. of the compensation point of the reference position: 60

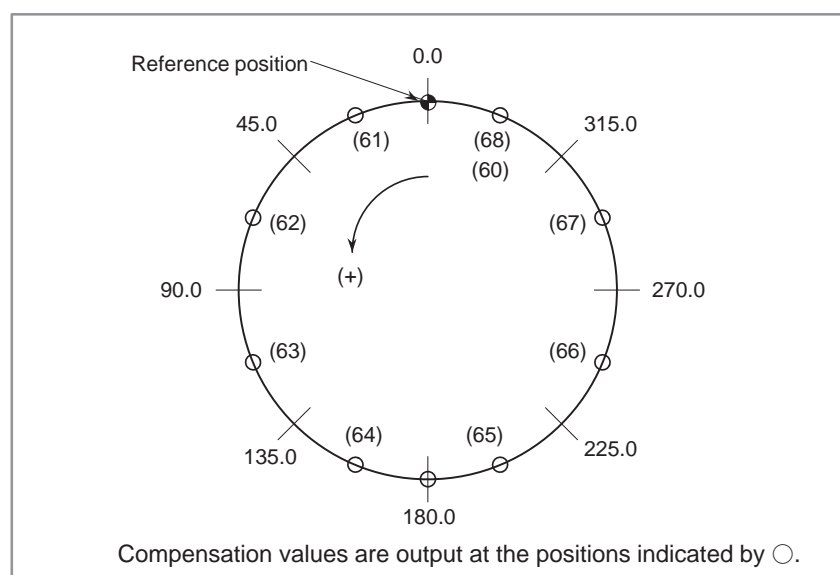
If the above is specified, the No. of the farthest compensation point in the negative direction for the rotating axis is always equal to the compensation point No. of the reference position.

The No. of the farthest compensation point in the positive direction is as follows:

No. of the compensation point of the reference position + (Move amount per rotation / Interval between the compensation points)

$$= 60 + 360/45 = 68$$

The correspondence between the machine coordinate and the compensation point No. is as follows:



If the sum of the compensation values for positions 61 to 68 is not 0, pitch error compensation values are accumulated for each rotation, causing positional deviation.

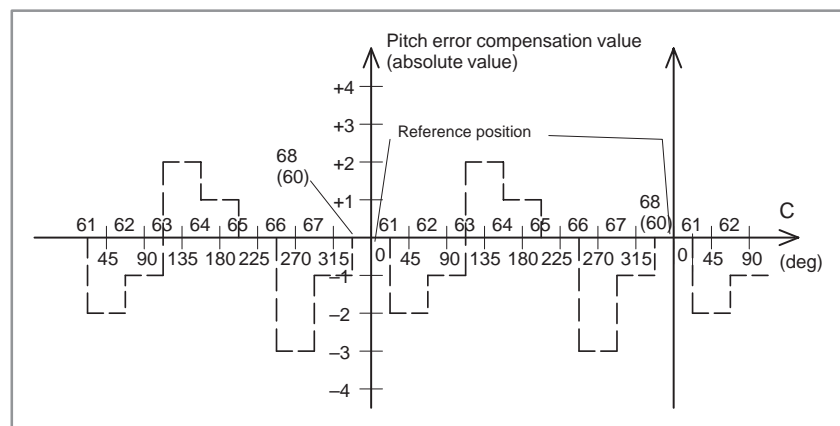
The same value must be set for compensation points 60 and 68.

Therefore, set the parameters as follows:

Parameter	Setting value
4000 : Compensation number for the reference position	60
011#0, #1 : Compensation magnification	#0=0, #1=0
715 : Interval between pitch error compensation points	45000

The following is an example of compensation amounts.

Compensation position number	60	61	62	63	64	65	66	67	68
Parameter	4061	4062	4063	4064	4065	4066	4067	4068	4069
Compensation value	+1	-2	+1	+3	-1	-1	-3	+2	+1



11.5.3 Displaying and Entering Setting Data




Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.

See Chapter 10 for automatic insertion of sequence numbers.

See subsection 11.4.4 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 [**PARAM**] to display the parameter 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.

PARAMETER
O1224 N0000

(SETTING 1)

—REVX = 0

REYV = 0

TVON = 0

ISO = 0 (0:EIA 1:ISO)

INCH = 0 (0:MM 1:INCH)

I/O = 0

ABS = 0 (0:INC 1:ABS)

SEQ = 0




CLOCK 97/01/10

03:30:18

NO. REVX = S 0 T

03:30:09 MDI

[PARAM][DGNOS][][SV-PRM][]

- 4 Move the cursor to the item to be changed by pressing cursor keys  or  .
- 5 Enter a new value and press  key.

Contents of settings

- **PARAMETER WRITE (PWE)**

Setting whether parameter writing is enabled or disabled.

0 : Disabled

1 : Enabled

- **TV CHECK (TV ON)**

Setting to perform TV check. 0 : No TV check




1 : Perform TV check

- **PUNCH CODE (ISO)** Setting code when data is output through reader puncher interface.
0 : EIA code output
1 : ISO code output
- **INPUT UNIT (INCH)** Setting a program input unit, inch or metric system
0 : Metric
1 : Inch
- **I/O CHANNEL (I/O)** Using channel of reader/puncher interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2
3 : Channel 3
- **SEQUENCE NUMBER
(SEQUENCE)** 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.
- **TAPE FORMAT (TAPE F)** Setting the F10/11 tape format conversion.
0 : Tape format is not converted.
1 : Tape format is converted.
See PROGRAMMING for the F10/11 tape format.
- **SEQUENCE STOP
(COMPENSATION AND
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
(MIRROR X, MIRROR Y,
MIRROR 4)** Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on
- **MDI travel command
(absolute)** Specifies whether a travel command executed in MDI operation is
absolute or incremental.
0 : Incremental command
1 : Absolute command

11.5.4 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

- 1 Select the **MDI** mode.
- 2 Press function key  .
- 3 Press soft key [**PARAM**].
- 4 Press page key  or  several times until the following screen is displayed.

PARAMETER
O1224 N0000

(SETTING 2)

_PWE = 1 (0:DISABLE 1:ENABLE)

REV4 = 0

TAPEF = 0

(SEQUENCE STOP)

PRGNO = 0

SEQNO = 0

PART TOTAL = 17

PART REQUIRED = 50

PART COUNT = 17

RUN TIME OH 4M CYCLE TIME OH OM 2S

NO. PWE = S 0 T

03:35:07 MDI

[PARAM][DGNOS][][SV-PRM][]

- 5 Enter in (PRGNO) for SEQUENCE STOP the number (1 to 9999) of the program containing the sequence number with which operation stops.
- 6 Enter in (SEQNO) for SEQUENCE STOP (with four or less digits) the sequence number with which operation is stopped.
- 7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

Explanations

- **Sequence number after the program is executed**

After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one. When the power is turned on, the setting of the sequence number is 0.
- **Exceptional blocks**

If the predetermined sequence number is found in a macro statement block, the execution does not stop at that block.

Example

```
N1 #1=1 ;  
N2 IF [#1 EQ 1] GOTO 08 ;  
N3 GOTO 09 ;  
N4 M98 P1000 ;  
N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.
- **Stop in the canned cycle**

If the predetermined sequence number is found in a block which has a canned-cycle command, the execution of the program stops after the return operation is completed.
- **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.

11.5.5 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. The data except for the total number of machined parts can be set on this screen. The time can be displayed and set on the screen.


Procedure for Displaying and Setting Run Time, Parts Count and Time

- 1 Select the MDI mode.
- 2 Press function key  .
- 3 Press chapter selection soft key [PARAM].
- 4 Press page key  or  several times until the following screen is displayed.

PARAMETER		O1224 N0000	
(SETTING 1)			
—REVS	= 0		
REVS	= 0		
TVON	= 0		
ISO	= 0	(0:EIA 1:ISO)	
INCH	= 0	(0:MM 1:INCH)	
I/O	= 0		
ABS	= 0	(0:INC 1:ABS)	
SEQ	= 0		
		CLOCK	97/01/10 03:30:18
NO. REVS	=	S	0 T
03:30:09		MDI	
[PARAM]	[DGNOS]	[SV-PRM]	[]

PARAMETER		O1224 N0000	
(SETTING 2)			
—PWE	= 1	(0:DISABLE 1:ENABLE)	
REV4	= 0		
TAPEF	= 0		
(SEQUENCE STOP)			
PRGNO	= 0		
SEQNO	= 0		
PART TOTAL	= 17		
PART REQUIRED	= 50		
PART COUNT	= 17		
RUN TIME		OH 4M CYCLE TIME	OH OM 2S
NO. PWE	=	S	0 T
03:35:07		MDI	
[PARAM]	[DGNOS]	[SV-PRM]	[]

- 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  key.

Display items

- **PARTS TOTAL** This value is incremented by one when M02, M30, or an M code specified by parameter 219 is executed. This value cannot be set on this screen. Set the value in parameter 779.
- **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. 600).
- **PARTS COUNT** This value is incremented by one when M02, M30, or an M code specified by parameter 219 is executed. 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.
- **OPERATING TIME** Indicates the total run time during automatic operation, excluding the stop and feed hold time.
- **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.
- **CUTTING 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.
- **TIME** Displays the current date and time. The date and time can be set on this screen. The date is displayed in year/month/day format. The time is displayed in hour:minute:second format (24-hour clock).

Explanations

- **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. 219) is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (parameter (No. 040#3) is set to 1). For details, see the manual issued by machine tool builders.

Restrictions

- **Run time and part count** 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.

Item	Maximum value	Item	Maximum value
Year	99	Hour	23
Month	12	Minute	59
Day	31	Second	59

11.6

SCREENS

DISPLAYED BY

FUNCTION KEY

OPR

ALARM

The alarm message and operator message can be displayed by pressing the

OPR

ALARM

 key. The software operator’s panel can also be displayed and specified. For details of how to display the alarm message, see Chapter 7.

11.6.1

Displaying Operator

Message

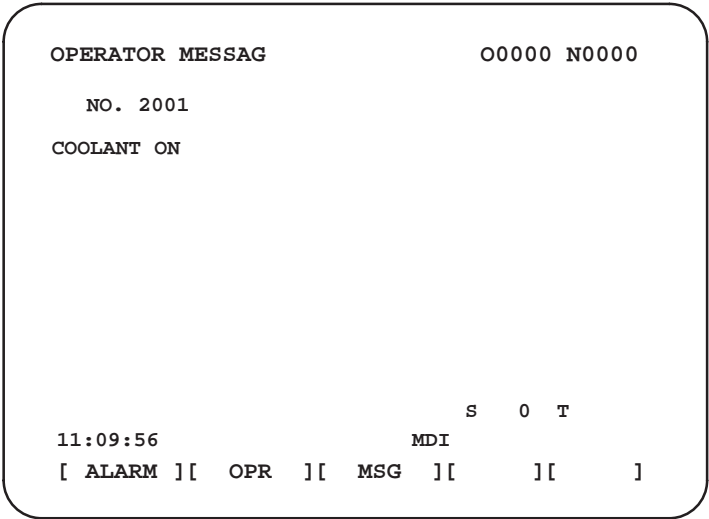
The operator message function displays a message on the PMC screen.

Procedure for Displaying Operator Message

- 1 Press function key

OPR

ALARM
- 2 Press soft key [MSG].



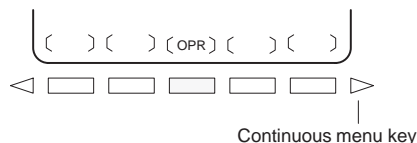
Explanations

- If the operator message display function is enabled, the screen is automatically switched to the operator message screen.
- A PMC command can also be used to clear the operator message.
- For details of the contents of the operator message, and how to clear the message, refer to the manual provided by the machine tool builder.

11.6.2 Displaying and Setting the Software Operator's Panel

With this function, functions of the switches on the machine operator's panel can be controlled from the CRT/MDI panel.
Jog feed can be performed using numeric keys.



Procedure for displaying and setting the software operator's panel







1 Press function key .

2 Press soft key **[OPR]**.

3 The screen consists of several pages.

Press page key  or  until the desired screen is displayed.

OPERATOR'S PANEL O1224 N0000











MODE : MDI AUTO EDIT HNDL JOG 
HANDLE AXIS :  HX HY HZ
HANDLE MULT. :  *1 *10 *100
RAPID OVRD. : 100% 50% 25% 
JOG FEED : 500 MM/MIN

FEED OVRD. : 100%  **

ACTUAL POSITION (ABSOLUTE)
X 0.000 Y 0.000
Z 0.000

03:59:30 S O T
MDI
[ALARM] [OPR] [MSG] [] []

OPERATOR'S PANEL O1224 N0000




BLOCK SKIP :  OFF ON
SINGLE BLOCK :  OFF  ON
MACHINE LOCK :  OFF  ON
DRY RUN :  OFF  ON
PROTECT KEY :  PROTECT  RELEASE
FEED HOLD :  OFF ON

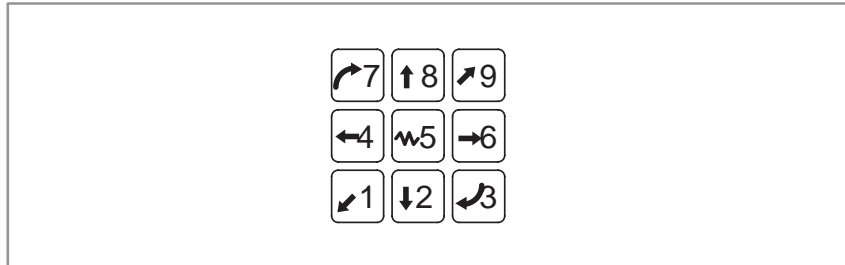
ACTUAL POSITION (ABSOLUTE)
X 0.000 Y 0.000
Z 0.000

04:00:18 S O T
MDI
[ALARM] [OPR] [MSG] [] []

4 Move the cursor to the desired switch by pressing cursor key  or



- 5 Push the cursor move 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 function key  together with an arrow key to perform jog rapid traverse.



Explanations

• Valid operations

The valid operations on the software operator's panel are shown below. Whether to use the MDI key or machine operator's panel for each group of operations can be selected by parameter 017#0 to #6.

Group1 : Mode selection

Group2 : Selection of jog feed axis, jog rapid traverse

Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification x1, x10, x100

Group4 : Jog federate, federate override, rapid traverse override

Group5 : Optional block skip, single block, machine lock, dry run

Group6 : Protect key

Group7 : Feed hold

• Display

The groups for which the machine operator's panel is selected by parameter 017#0 to #6 are not displayed on the software operator's panel.

• Screens on which jog feed is valid

When the screen indicates other than the software operator's panel screen and diagnostic 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. 130 to 137).

• General purpose switches

Eight optionally definable switches are added as an extended function of the software operator's panel. The name of these switches can be set by parameters as character strings of max. 8 characters. For the meanings of these switches, refer to the manual issued by machine tool builder.

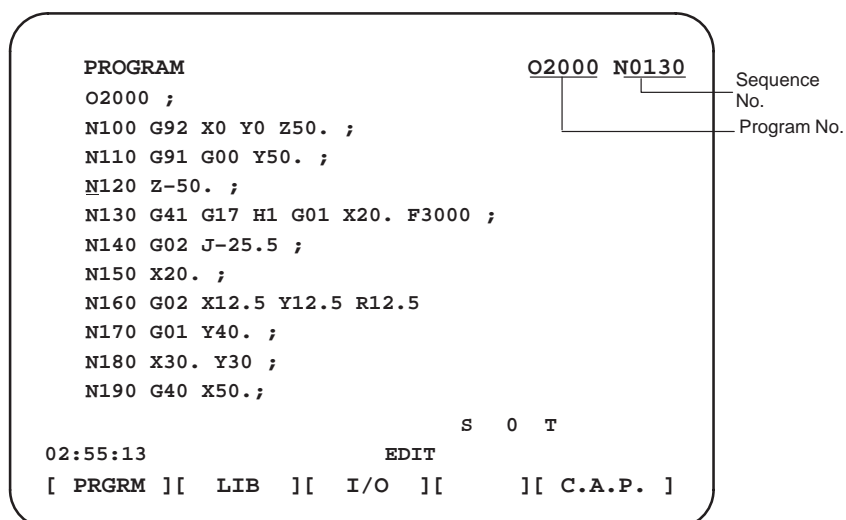
11.7 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING

The program number, sequence number, and current CNC status are always displayed on the screen except when the power is turned on or a system alarm occurs.

This section describes the display of the program number, sequence number, and status.

11.7.1 Displaying the Program Number and Sequence Number

The program number and sequence number are displayed at the top right on the screen as shown below.



The program number and sequence number displayed depend on the screen and are given below:

On the program screen in the EDIT mode on Background edit screen :
The program No. being edited and the sequence number just prior to the cursor are indicated.

Other than above screens :

The program No. and the sequence No. executed last are indicated.

Immediately after program number search or sequence number search :

Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

11.7.2 Displaying the Status and Warning for Data Setting

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the CRT screen allowing the operator to readily understand the operation condition of the system.

Explanations

Description of each displa

```

NOT READY          S    500 T0123
11:53:05  ALARM BAT BUF EDIT    INPUT
              (Display soft keys)

```

① Current mode

MDI : Manual data input
 AUTO : Automatic operation
 RMT : Automatic operation (Tape operation, or such like)
 EDIT : Memory editing
 HNDL : Manual handle feed
 JOG : Jog feed
 TJOG : TEACH IN JOG
 THND : TEACH IN HANDLE
 STEP : Manual incremental feed
 ZRN : Manual reference position return

② Alarm status

ALARM : Indicates that an alarm is issued.
 BAT : Indicates that the battery is low.

③ Current time

hh:mm:ss – Hours, minutes, and seconds

④ Other status display

INPUT : Indicates that data is being input.
 OUTPUT : Indicates that data is being output.
 SRCH : Indicates that a search is being performed.
 EDIT : Indicates that another editing operation is being performed
 (insertion, modification, etc.)
 COMPARE : Indicates that the program is being collated.
 LSK : Indicates that labels are skipped when data is input.
 RSTR : Indicates that the program is being restarted
 BUF : Indicates that the block to be executed next is being read.
 NOT READY : Indicates that the system is in the emergency stop state.

⑤ Actual spindle speed (S)

The actual spindle speed is displayed after S. (The threading function or synchronous feed function is required.)

⑥ T code

The specified T code is displayed.
 To display the actual spindle speed and T code, bit 2 of parameter 0014 must be specified accordingly.

12

GRAPHICS FUNCTION



Two graphic functions are available. One is a graphic display function, and the other is a dynamic graphic display function.

The graphic display function can draw the tool path specified by a program being executed. The graphic display function also allows enlargement and reduction of the display.

The dynamic graphic display function can draw a tool path and machining profile. In tool path drawing, automatic scaling and solid drawing are possible. In machining profile drawing, the status of machining in progress can be drawn through simulation. Blank figures can also be drawn.

This chapter mainly explains drawing procedures and drawing parameters for the following:

1. Drawing the tool path specified by a program being executed, with the graphic display function
2. Drawing the tool path with the dynamic graphic display function
3. Drawing the machining profile with the dynamic graphic display function

12.1

GRAPHICS DISPLAY

It is possible to draw the programmed tool path, which makes it possible to check the progress of machining, while observing the path on the CRT screen.


In addition, it is also possible to enlarge/reduce the screen.

Before drawing, graphic parameters must be set.

When the dynamic graphics function is used, the graphics function described in this section cannot be used. See Section 12.2 for the dynamic graphics function.

Graphics display procedure

Procedure

- 1
- Press function key .
- The graphic parameter screen shown below appears. (If this screen does not appear, press soft key [G.PRM].)

GRAPHIC PARAMETER

O9501 N0014

AXES

_P=

4

(XY=0, YZ=1, ZY=2, XZ=3, XYZ=4, ZXY=5)

RANGE

(MAX.)

X=

0

Y=

0

Z=

0

RANGE

(MIN.)

X=

0

Y=

0

Z=

0

SCALE

K=

70

GRAPHIC CENTER

X=

57500

Y=

75000

Z=

0

PROGRAM STOP

N=

0

AUTO ERASE

A=

1

NO.

P =

S

0

T

10:16:23

MDI


[G.PRM]

[GRAPH]

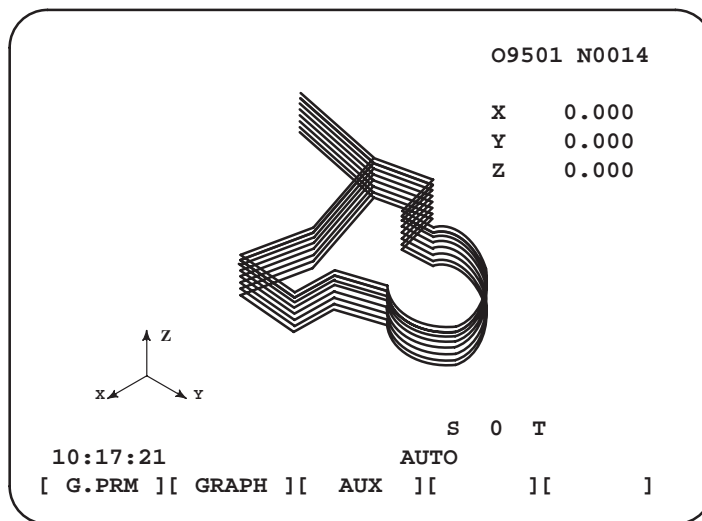
[AUX]

[]

[]

- 2
- Move the cursor with the cursor keys to a parameter to set.
- 3
- Enter data, then press the  key. (A decimal point cannot be entered.)
- 4
- Repeat steps 2 and 3 until all required parameters are specified.
- 5
- Press soft key [GRAPH].

- 6 Automatic operation is started and machine movement is drawn on the screen.



Explanations

- **RANGE**
(Actual graphic range)

The size of the graphic screen will be as follows:

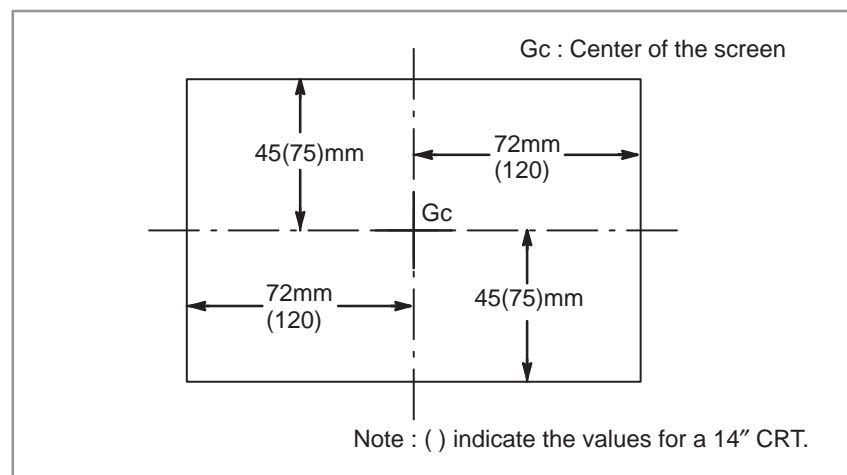


Fig. 12.1 (a) Graphic range

As shown in Fig.12.1 (a), the maximum graphics range is an area of approx. 144 mm (width) × 90 mm (height) for a 9" CRT and approx. 240 mm (width) × 150 mm (height) for a 14" CRT.

- **Setting the graphics range**

To draw a section of the program within the actual graphics range, set the graphics range using one of the following two methods:

- 1 Set the center coordinates of the range and the magnification.
- 2 Set the maximum and minimum coordinates for the range in the program.

Whether 1 or 2 is used depends on which parameters are set last. A graphics range which has been set is retained when the power is turned off.

1. Setting the center coordinate of the graphics range and graphics magnification

Set the center of the graphic range to the center of the screen. If the drawing range in the program can be contained in the actual graphics range, set the magnification to 1 (actual value set is 100).

When the drawing range is larger than the maximum graphics range or much smaller than the maximum graphics range, the graphics magnification should be changed. The graphics magnification is 0.01 to 100.00 times, which is usually determined as follows;

Graphics magnification = Graphics magnification (**H**), or graphics magnifications (**V**), whichever is smaller

Graphics magnification **H** = $\alpha / (\text{length on program to horizontal direction axis})$

Graphics magnification **V** = $\beta / (\text{length on program to vertical direction axis})$

α : 144mm (for 9" CRT)

β : 90mm

α : 240mm (for 14" CRT)

β : 150mm

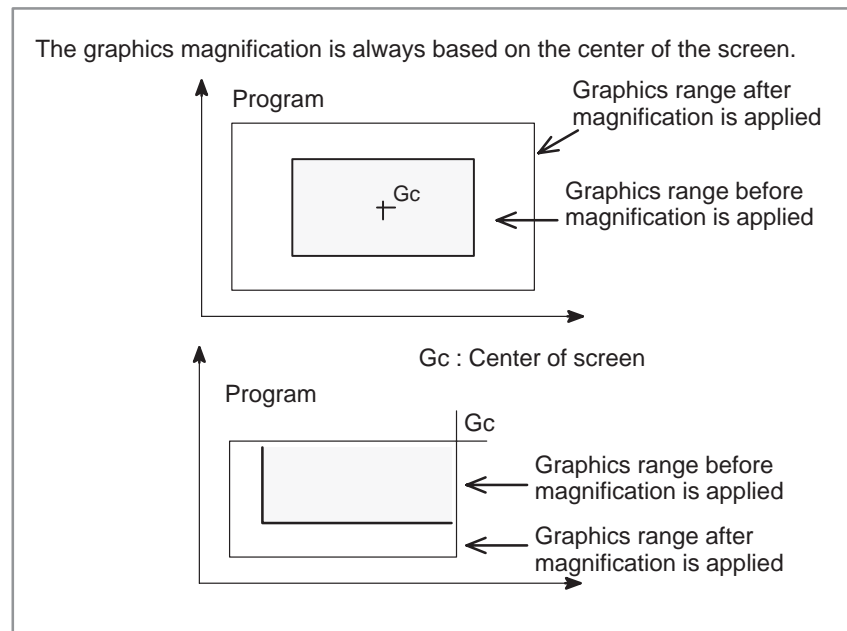


Fig. 12.1 (b) Applying graphics magnification (Example of enlargement)

2. Setting the maximum and minimum coordinates for the drawing range in the program

When the actual tool path is not near the center of the screen, method 1 will cause the tool path to be drawn out of the graphics range if graphics magnification is not set properly.

To avoid such cases, the following six graphic parameters are prepared;

Graphic range (Max.) X

Graphic range (Max.) Y

Graphic range (Max.) Z

Graphic range (Min.) X

Graphic range (Min.) Y

Graphic range (Min.) Z

With the above parameters, the center of screen (Gcx, Gcy, Gcz) is determined by the CNC as follows;

$$Gcx = (X (MAX.) + X (MIN.)) / 2$$

$$Gcy = (Y (MAX.) + Y (MIN.)) / 2$$

$$Gcz = (Z (MAX.) + Z (MIN.)) / 2$$

The unit of the value will be 0.001 mm or 0.0001 inch depending on the input unit.

Graphics magnification is applied automatically. When the graphics range is specified, the center coordinates and magnification do not need to be calculated.

• Work coordinate system and graphics

The graphic origin and graphic center point will not be changed even if the workpiece coordinate origin is changed.

In other words, the workpiece coordinate origin is always consistent with the graphic origin.

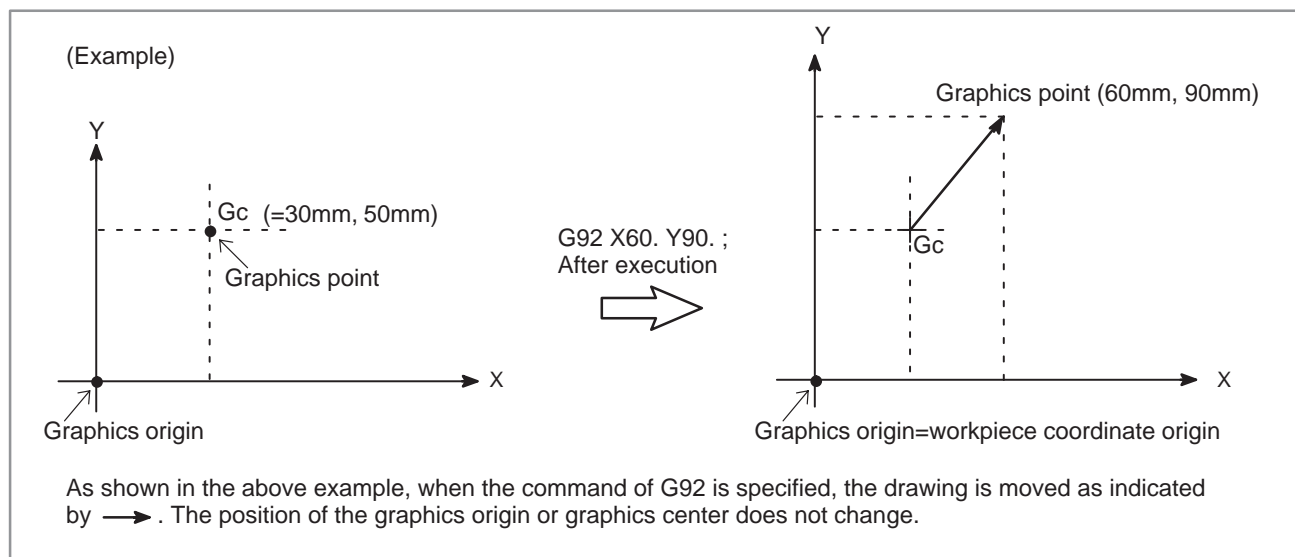


Fig. 12.1 (c) Workpiece coordinate origin and graphics origin

- **Graphics parameter**

- **AXES**

Specify the plane to use for drawing. The user can choose from the following six coordinate systems:

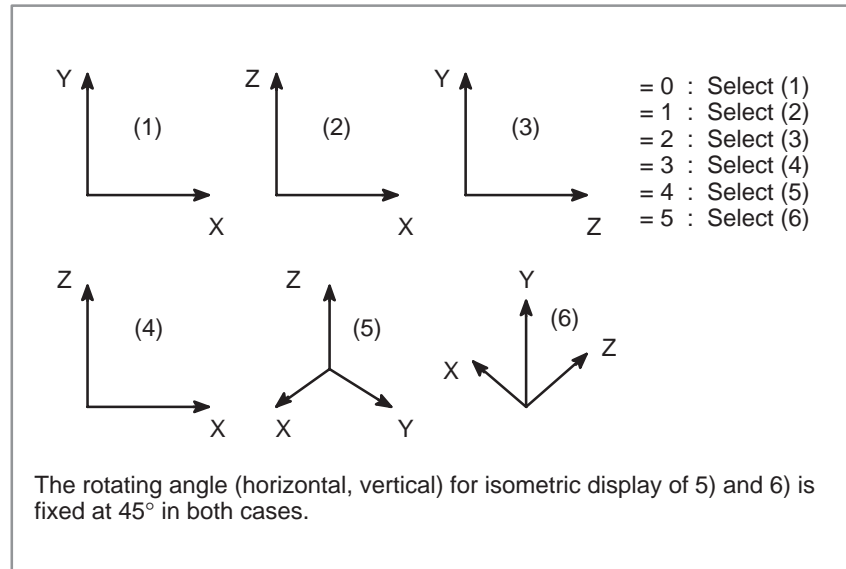


Fig. 12.1 (d) Coordinate system

- **RANGE (Max., Min.)**

Set the graphic range displayed on the screen by specifying maximum and minimum values along each axis.

X =Maximum value X=Minimum value

Y =Maximum value Y=Minimum value

Z =Maximum value Z=Minimum value

Valid range: -99999999 to +99999999

NOTE

- 1 The units are 0.001 mm or 0.0001 inch. Note that the maximum value must be greater than the minimum value for each axis.
- 2 When setting the graphics range with the graphics parameters for the maximum and minimum values, can not set the parameters for the magnification and screen center coordinates afterwards. Only the parameters set last are effective.

- **SCALE**

Set the graphic magnification

The setting range is 0 to 10000 (unit:0.01 time).

- **GRAPHIC CENTER**

X=

Y=

Z=

Set the coordinate value on the workpiece coordinate system at graphic center.

NOTE

- 1 When MAX. and MIN. of RANGE are set, the values will be set automatically once drawing is executed
- 2 When setting the graphics range with the graphics parameters for the magnification and screen center coordinates, do not set the parameters for the maximum and minimum values afterward. Only the parameters set last are effective.

- **PROGRAM STOP**

N=_

Set the sequence No. of the end block when necessary to partially draw.

This value is automatically cancelled and set to -1 once drawing is executed.

- **AUTO ERASE**

1 : Erase the previous drawing automatically when the automatic operation is started under reset condition.

2 : Not erase automatically.

- **Executing drawing only**

Since the graphic drawing is done when coordinate value is renewed during automatic operation, etc., it is necessary to start the program by automatic operation. To execute drawing without moving the machine, therefore, enter the machine lock state.

- **Deleting the previous drawing**

When the AUTO operation is started under reset condition, the program is executed after deleting the previous drawing automatically (Automatic deleting=1). It is possible not to delete the previous drawing by graphic parameter (Automatic deleting=0).

- **Drawing a part of a program**

When necessary to draw a part of a program, search the starting block to be drawn by the sequence No. search, and set the sequence No. of the end block to the PROGRAM STOP N= of the graphic parameter before starting the program under cycle operation mode.

- **Drawing using dashed lines and solid lines**

The tool path is shown with a dashed line (- - -) for rapid traverse and with a solid line (—) for cutting feed.

Restrictions

- **Feedrate**

In case the feed rate is considerably high, drawing may not be executed correctly, decrease the speed by dry-run, etc. to execute drawing.

12.2

DYNAMIC GRAPHIC DISPLAY

There are the following two functions in Dynamic Graphics.

Path graphic	This is used to draw the path of tool center commanded by the part program.
Solid graphic	This is used to draw the workpiece figure machined by tool movement commanded by the part program.

The path graphic function is used to precisely check the part program for drawing the tool path with a line. The solid graphic function is used to draw the workpiece figure to be machined with a program. Thus, it is easy to recognize roughly the part program. These two functions can be used freely by switching them.

12.2.1 Path Drawing

The path graphic feature calls a program from memory and draws the tool path specified by the program. This feature provides the following functions.

1. Drawing plane

The user can choose the drawing plane from four types of plane views, two types of isometric projection views, and biplane view.

2. Drawing rotation

When an isometric projection view is used, the drawing can be rotated horizontally and vertically.

3. Drawing enlargement and reduction

A drawing can be enlarged or reduced by specifying a magnification from 0.01 to 100 with respect to the actual size. In addition, a drawing can be automatically enlarged or reduced by setting maximum and minimum values.

4. Partial drawing

A range of the program can be drawn by specifying a starting sequence number and ending sequence number.

5. Programmed path and tool path drawing

The user can specify whether to apply tool length offset and cutter compensation to drawing. This way, either the actual programmed path or the tool path can be drawn.

6. Color

When a tool path is drawn on a color CRT screen, the colors used can be chosen from seven colors including white. The color of the tool path can be changed according to the T code.

7. Automatic scaling

The CNC automatically determines the maximum and minimum drawing coordinates for each program. This means that drawing can be performed with a magnification automatically determined according to these maximum and minimum values.

8. Partial enlargement drawing

Except for biplane views the user can enlarge all types of drawings by a factor of up to 100 while looking at the drawing that has been made.

9. Indicating the current tool position with a mark

The current tool position can be displayed on the screen.

10. Indicating the coordinates of the current position

The current position can also be indicated using coordinates.

11. Displaying coordinate axes and actual size dimensions lines


Coordinate axes and actual size dimension lines are displayed together with the drawing so that actual size can be referenced.

The first six functions above (1. to 6.) are available by setting the graphic parameters. The seventh to ninth functions (7. to 9.) are mainly executed using soft keys after drawing has been setup. The tenth function (10.) is enabled by setting a parameter. The eleventh function (11.) can be used at any time.

Path drawing procedure

Procedure

- 1 To draw a tool path, necessary data must be set beforehand.

So after press the function  key, press soft key **[PATH]**. The “PATH GRAPHIC (PARAMETER)” is displayed.

```

PATH GRAPHIC (PARAMETER-1)          O9501 N0014

AXES                P=                4
(XY=0, YZ=1, ZY=2, XZ=3, XYZ=4, ZXY=5, 2P=6)

ANGLE
  ROTATION          A=                0
  TILTING           A=                0
  SCALE             K=                0.00
  CENTER OR MAX./MIN.
    X= 0.000        Y= 0.000        Z= 0.000
    I= 0.000        J= 0.000        K= 0.000
  START SEQ. NO.   N=                0
  END SEQ. NO.     N=                0
NO.      P =                               S    0    T
10:20:36                               MDI
[ PARA-1 ][ PARA-2 ][ EXEC ][ SCALE ][ POS ]

```

```


PATH GRAPHIC (PARAMETER-2)          O9501 N0014

TOOL. COMP.      _P=                0
COLOR (0123456)
  PATH           P=                0
  TOOL           Q=                0
  AUTO CHANGE    R=                0



NO.      P =                               S    0    T
10:21:20                               MDI
[ PARA-1 ][ PARA-2 ][ EXEC ][ SCALE ][ POS ]

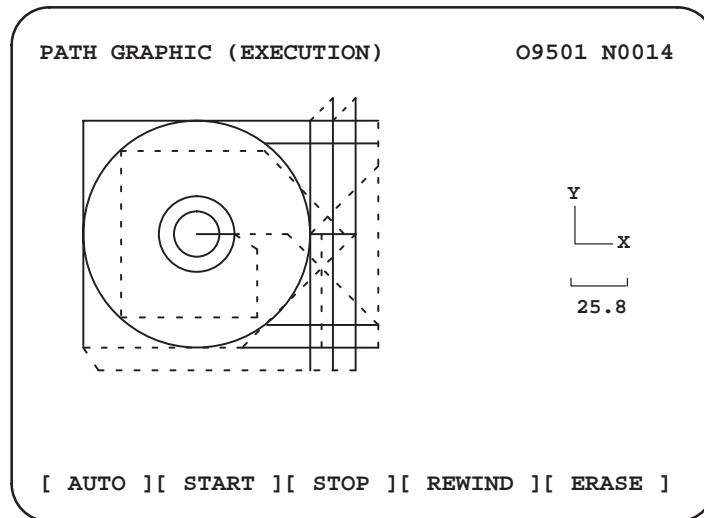
```

- 2 There are two screens for setting drawing parameters.
Press the soft key **[PARA-1]**, **[PARA-2]** according to the setting items for selecting screens.

- 3 Set the cursor to an item to be set by cursor keys.
- 4 Input numerics by numeric keys.
- 5 Press the  key.

The input numerics are set by these operations and the cursor automatically moves to the next setting items. The set data is held even after the power is turned off.

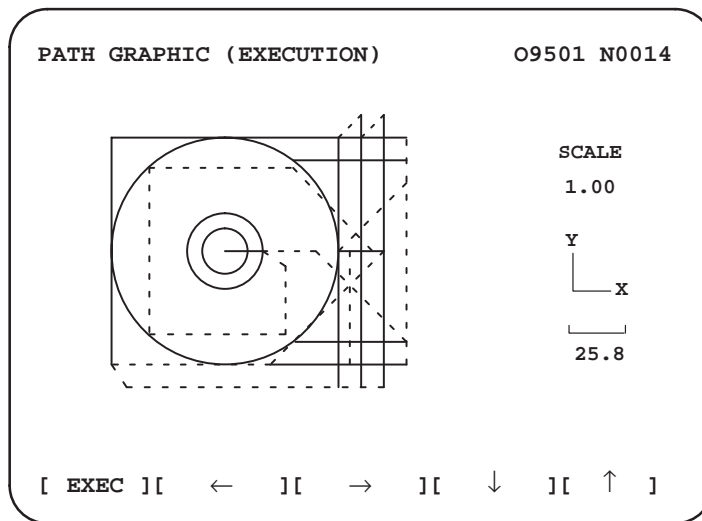
- 6 Set the operation mode to the AUTO mode, press function key , and call the part program which should be drawn.
- 7 After press function key , and press soft key **[PATH]** to redisplay the PATH GRAPHIC (PARAMETER) screen. Then press soft key **[EXEC]** to display the PATH GRAPHIC (EXECUTION) screen.



- 8 Press soft key **[AUTO]** or **[START]**. Pressing **[AUTO]** enables automatic scaling. See item 7 in introduction of path drawing and the description of soft key **[AUTO]** in Explanations for details. Drawing is now started. During drawing, the message "DRAWING" blinks at the lower-right corner of the screen.
- 9 Press soft key **[STOP]** to pause drawing. The indication of "STOP" blinks at the lower right corner on the screen. Press soft key **[START]** to start drawing. In addition, press soft key **[REWIND]** to redraw from the top of program before pressing soft key **[START]**.
- 10 Execute the last of part program (M02/M30) to end drawing. This will cause, blinking of the "DRAWING" light to turn off. The tool path view drawn can be retained until the power is turned off unless a new tool path view is drawn or soft key **[ERASE]** is pressed.

Partial enlargement

- 11** For partial drawing enlargement, display the PATH GRAPHIC (SCALE) screen by pressing the soft key **[ZOOM]** on the PATH GRAPHIC (PARAMETER) screen of **step 1** above. The tool path is displayed.



- 12** Perform positioning of marks displayed at the center of the screen to the center of the part enlarged using soft keys **[←]**, **[→]**, **[↓]**, and **[↑]**.
- 13** Set the relative magnification rate for the tool path view which is being drawn using the address keys “P” and “M”. When you press address key P or M, the following results:

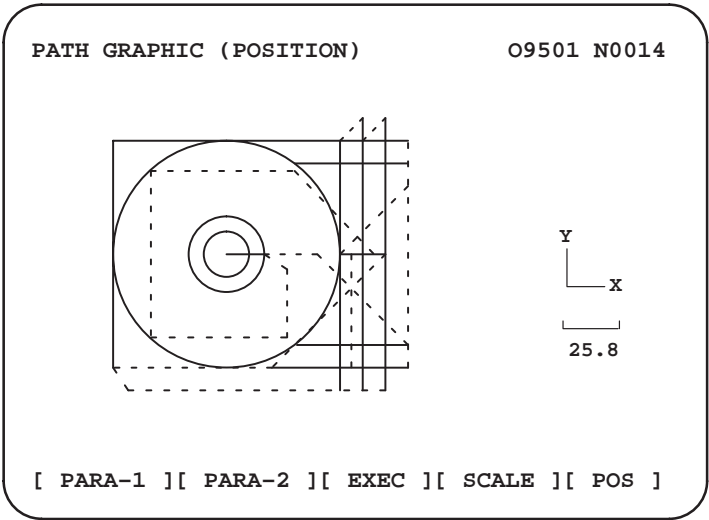
Address key	Function
P	The relative magnification rate increases by 0.1.
M	The relative magnification rate decreases by 0.1.

The relative magnification rate is continuously changed by keeping the address keys depressed. It is possible to magnify up to 100 times in reference to the actual dimensions.

- 14** Press the soft key **[EXEC]** after setting the relative magnification rate. Then, the screen automatically changes to “TOOL PATH (EXECUTION)” and the drawing of set partial enlargement view starts. The set partial enlargement status is valid until soft key **[AUTO]** or **[ERASE]** is pressed.

Mark display of current tool position

- 15 To display a mark at the current tool position, display the PATH GRAPHIC (POSITION) screen by pressing soft key [POS] on the PATH GRAPHIC (PARAMETER) screen of step 1 above. This mark blinks at the current tool center position on the tool path.



Explanations

- Drawing plane

The relationship between the setting value and drawing plane is as shown below:

Setting value	Drawing plane
0	Plane view (XY)
1	Plane view (YZ)
2	Plane view (ZY)
3	Plane view (XZ)
4	Isometric projection (XYZ)
5	Isometric projection (ZXY)
6	Biplane view (XY,XZ)

- Plane view (XY,YZ,ZY,XZ)

The following coordinate systems are selected.

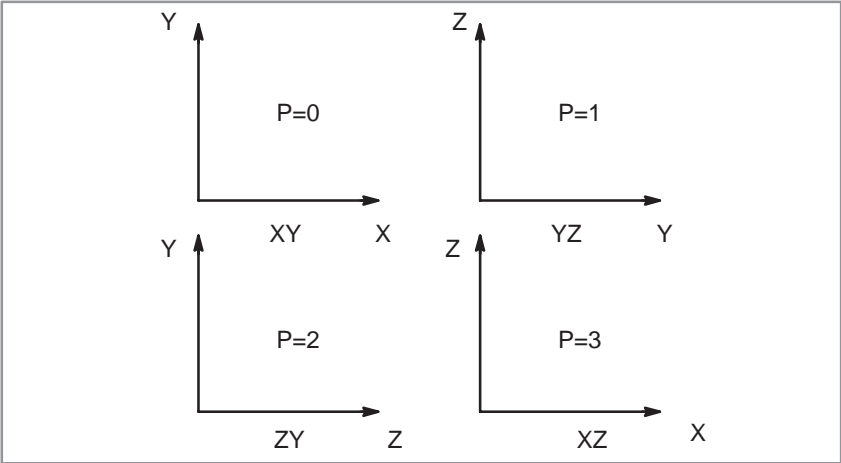


Fig. 12.2.1 (a) Coordinate systems for the plane view

- **Isometric projection (XYZ,ZXY)**

Projector view by isometric can be drawn.

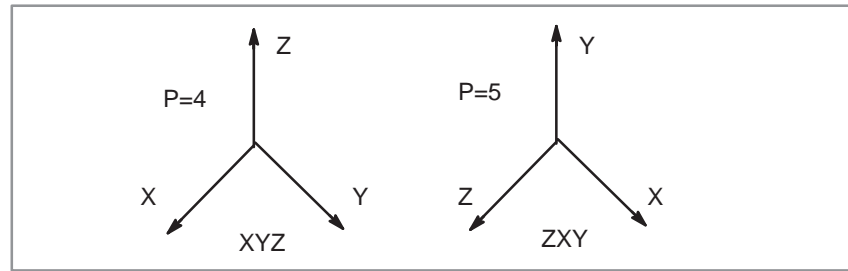


Fig. 12.2.1 (b) Coordinate systems for the isometric projection

- **Biplane view**

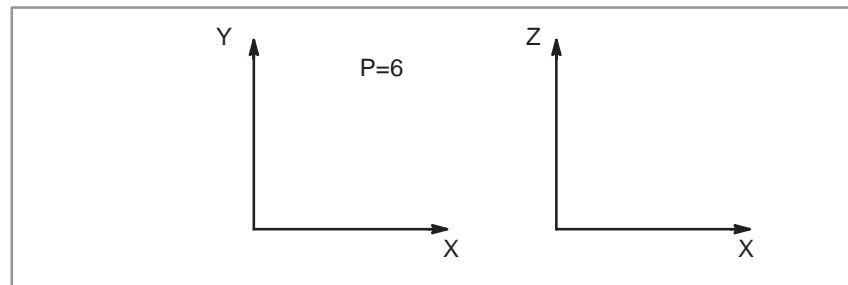


Fig. 12.2.1 (c) Coordinate systems for the biplane view

Biplanes (XY and XZ) can be drawn simultaneously. The maximum and minimum coordinate values must be set to draw the biplane view. The maximum and minimum coordinate values can also be set by performing automatic scaling

- **ANGLE**

The direction of the coordinate axis is set when the isometric projection is the setting of the drawing screen. The direction is set by horizontal and vertical rotation angles. The unit is expressed in degrees.

- **ROTATION**

The horizontal rotation angle is set in the range of -180° to $+180^\circ$ in reference to the vertical axis. Set a positive value for clockwise rotation of the coordinate axis. Thus, the direction of projection (visual arrow) becomes counterclockwise.

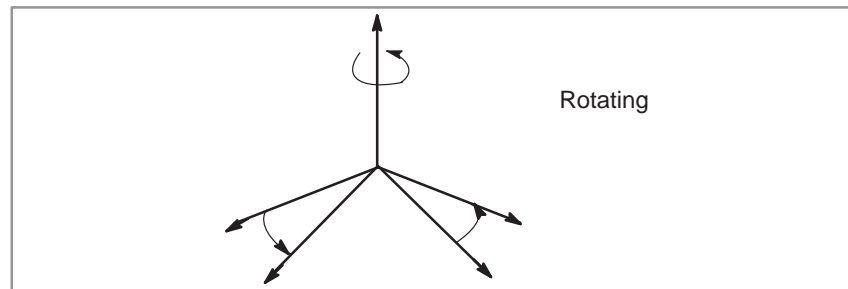


Fig. 12.2.1 (d) Rotating

- **TILTING**

The tilting angle of the vertical axis is set in the range of -90° to $+90^\circ$ in reference to the horizontal axis crossing the vertical axis at a right angle. When a positive value is set, the vertical axis slants to the other side of the graphic screen. Thus, the projection direction (arrow direction) becomes the horizontal direction.

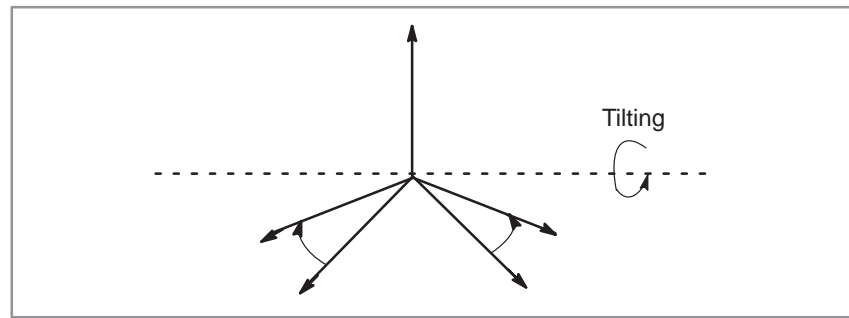


Fig. 12.2.1 (e) Tilting

- **SCALE**

Set the magnification rate of drawing from 0.01 to 100.00. When 1.0 is set, drawing is carried out in actual dimensions. When 0 is set, the drawing magnification rate is automatically set based on the setting of maximum and minimum coordinate values of drawing.

- **CENTER OR MAX./MIN.**

When a graphics (drawing) magnification of 0 is set, maximum coordinates on the X-axis, Y-axis, and Z-axis in the workpiece coordinate system must be set in addresses X, Y, and Z, and minimum coordinates must be set in addresses I, J, and K, to specify the graphics (drawing) range. For biplane view drawing, maximum and minimum coordinates for drawing must be specified.

When a drawing magnification other than 0 is set, the X, Y, and Z coordinates of the drawing center in the workpiece coordinate system must be set in addresses X, Y, and Z. Addresses I, J, and K are not used. The table below summarizes the setting requirements described above.

Setting the drawing magnification rate	Setting	
	Address X/Y/Z	Address I/J/K
Other than 0	Drawing center coordinate value of X, Y, and Z axes	Ignored
0 or biplane view drawing	Drawing maximum coordinate value of X, Y, and Z axes	Drawing minimum coordinate value of X, Y, and Z axes

- **START SEQ. NO. and END SEQ. NO.**

Set the start and end sequence numbers of drawing in four digits each. The part program for drawing is executed from the head and only the part enclosed by the start sequence and end sequence numbers is drawn. When 0 is set as the start sequence number, drawing is performed from the head of the program. In addition, when 0 is set as the end sequence number, drawing is performed up to the end of program. The sequence number is referred to regardless of either main program or subprogram.

- **TOOL COMP.**

It is possible to set whether the tool path is drawn by making the tool length offset or cutter compensation valid or invalid.

Setting value	Tool length offset or cutter compensation
0	Perform drawing by making tool compensation valid (an actual tool path is drawn.)
1	Perform drawing by making tool compensation invalid (A programmed path is drawn.)

Always set 0 before drawing when indicating the mark of the current tool position.

- **COLOR**

Specify the color of the tool path. In the case of monochrome display, it is not required to set it. The relationship between the setting value and color is as shown below:

Setting value	Color
0	White
1	Red
2	Green
3	Yellow
4	Blue
5	Purple
6	Light blue

- **PATH** Specify the color of the tool path.
- **TOOL** Specify the color of the current position mark of the tool.
- **AUTO CHANGE** Set if for changing the color of the tool path automatically according to the T-code command.

Setting value	Function
0	The color of the tool path is not changed.
1	The color of the tool path is changed automatically.

When 1 is set, the setting value of the color designation of PATH is incremented by 1 every time the T code is commanded. At the same time, the color of the tool path changes. If the setting value exceeds 6, it returns to 0.

- **Soft key functions on the “PATH GRAPHIC [EXECUTION]”screen**

Software key	Function
[AUTO]	Automatic scaling is performed. Obtain the maximum and minimum coordinates of the part program before performing drawing, specify them for the maximum and minimum values of drawing parameters, and set the drawing magnification rate to 0 before starting drawing. Thus, the tool path view is properly laid out on the screen.
[START]	Drawing starts. When the [START] is pressed while the drawing is not in STOP, the part program starts from the top of the part program. Press the [START] while the drawing is in stop to allow drawing to be carried out continuously.
[STOP]	Stop drawing. (Single block stop)
[REWIND]	Press this key to start drawing from the top of part program. Searches for the beginning of a part program.
[ERASE]	Erase the tool path view which has been drawn.

- **Graphic program**

No part program which has not been registered in memory can be drawn. Also, it is necessary that the M02 or M30 should be commanded at the end of the part program.

- **Mark for the tool current position**
The period of mark blinking is short when the tool is moving and becomes longer when the tool stops.
The mark indicating the current position of tool is displayed on the XY plane view when the biplane drawing is performed.
- **Position mark**
Parameter No. 058#5 is used to specify whether to use ■ or x as the mark for indicating the current tool position and the center of a partially enlarged drawing.
- **Display of the coordinate value**
Parameter No. 046#5 is used to specify whether to display the coordinates of the current position on the tool path drawing screen.
- **Changing the coordinate system**
If a program specifies a coordinate system change, parameter No.058#0 is used to specify whether to draw without changing the coordinate system or to draw by regarding the current drawing position as the current position in the new coordinate system.

Restrictions

- **Graphic condition**
If machine operation is not allowed, no drawing can be carried out. No drawing can be made during machine operation. The setting data and switches required for drawing are as shown below:

Setting data and switch	Status
Tool offset amount	Set it properly when performing drawing while the tool offset amount becomes valid.
Single block	Off
Optional block skip	Set it properly.
Feed hold	Off

- **Partial enlargement**
The partial enlargement can be carried out on the plane view and isometric projection view. No partial enlargement can be made in the drawing of the biplane view.
- **Tool current position**
In dynamic graphics display, drawing cannot be executed while the machine is operating even though this is possible in ordinary graphics display (see Section 12.1). However, after drawing is executed, the operator can see how the tool moves along the tool path by operating the machine while displaying the mark for the current position of the tool. It is necessary that the setting data and switches related to the machine operation should be the same status between drawing operation and machining operation for properly displaying the current position of tool on the drawn tool path.

12.2.2 Solid Graphics

The solid graphics draws the figure of a workpieces machined by the movement of a tool.

The following graphic functions are provided :

1. Solid model graphic

Solid model graphic is drawn by surfaces so that the machined figure can be recognized concretely.

2. Blank figure graphics

It is possible to draw a blank figure before machining.

A rectangular parallelepiped and a circular column or cylinder can be drawn. A circular column or cylinder parallel to the X-axis, Y-axis, or Z-axis can be selected.

3. Drawing of machining progress

It is possible to draw the progress of machining by simulation.

4. Drawing of final machined figure

It is possible to draw the final finish machined figure.

5. Changing of drawing direction

The user can choose from four drawing directions and eight tilting angles.

6. Plane view graphics

It is possible to draw XY plane views as well as solid model views. Height of the workpiece is discriminated by color for color display or brightness for monochrome display.

7. Tri-plane view graphic

In addition to a solid drawing, a triplane view can be drawn. The user can choose from four types of plane view and side view positions. The user can freely change the cross-section position of a side view.

8. Horizontal hole machining


It is possible to install tools in the direction which is parallel to the X or Y axis as well as the Z axis.

9. Tool change during machining

It is possible to change tools during machining by the part program command.

Solid graphics drawing procedure


Procedure

- 1 To draw a machining profile, necessary data must be set beforehand. So after press the function key  , press the soft key **[SOLID]**. Then the screen of “SOLID GRAPHIC (PARAMETER)” is displayed.

SOLID GRAPHIC (PARAMETER)

O9501 N0014

BLANK FORM	P=	2	
X= 0.000	Y=	0.000	Z= 0.000
I= 0.000	J=	0.000	K= 0.000
TOOL FORM	P=	0	
	R=	0.000	K= 0.000
PROJECTION	P=	0	
	Q=	0	R= 0
INTENSITY	P=	0	
(0123456)	Q=	0	R= 0
START SEQ.NO.	N=	0	
END SEQ.NO.	N=	0	
ANIM. SPEED	N=	0	
NO. X =			S 0 T
10:26:35		MDI	
[BLANK]	[EXEC]	[REVIEW]	[3-PLN] []

- 2 Use a cursor key to move the cursor to an item to be set.
- 3 Input numerics for the item at the cursor using the numeric key.
- 4 Press the  key.

Input numerics can be set by these operations and the cursor moves to the next setting item automatically. The set data is retained even if the power is turned off.

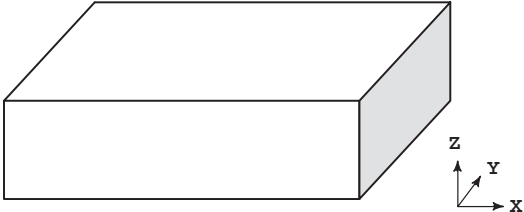
See Explanations for details on settings.

• SOLID GRAPHIC (BLANK)

- 5 To draw a blank figure, display the SOLID GRAPHIC (BLANK) screen by pressing soft key **[BLANK]** on the SOLID GRAPHIC (PARAMETER) screen of **step 1** above.

SOLID GRAPHIC (BLANK)


O9501 N0014

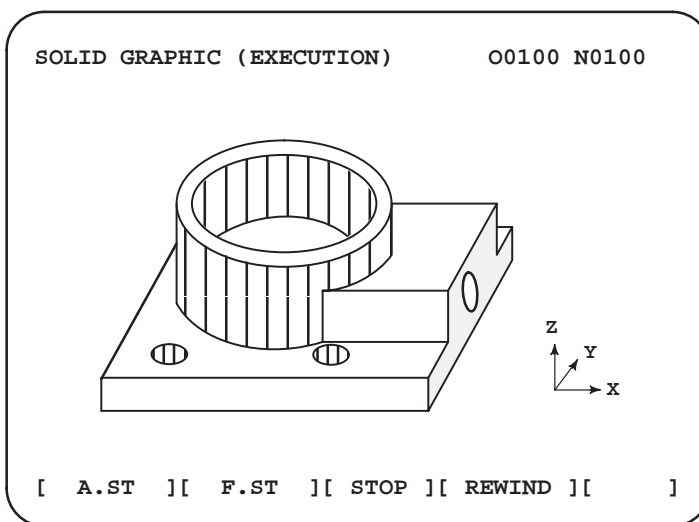


[ANEW] [+ ROT] [- ROT] [+TILT] [-TILT]

- 6 Press soft key **[ANEW]**. This allows the blank figure drawing to be performed based on the blank figure data set.
- 7 Press soft keys **[+ROT]** **[-ROT]** **[+TILT]**, and **[-TILT]**, when performing drawing by changing the drawing directions. Parameters P and Q for the drawing direction are changed and the figure is redrawn with the new parameters.

• SOLID GRAPHICS (EXECUTION)

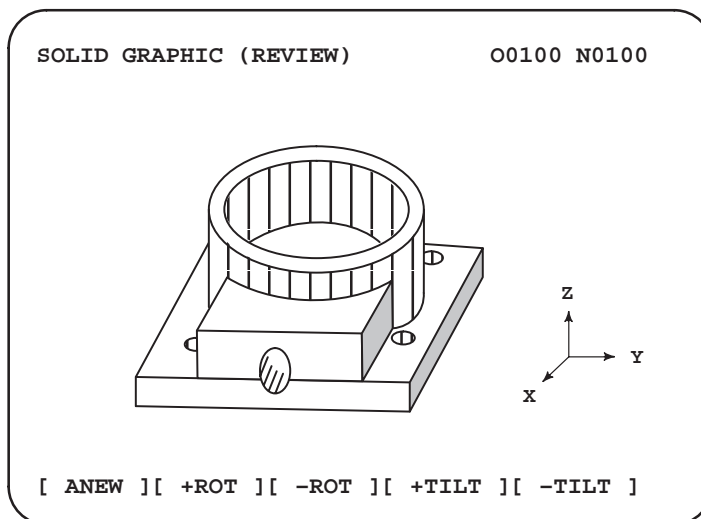
- 8 Set the operation mode to the **AUTO** mode, press function  key, and call the subject part program of drawing.
- 9 To draw a machining profile, display the SOLID GRAPHIC (EXECUTION) screen by pressing soft key **[EXEC]** on the SOLID GRAPHIC (PARAMETER) screen of **step 1** above.



- 10 Press either soft key **[A.ST]** or **[F.ST]**. When “STOP” is not displayed at the lower right corner of the screen, the program is executed from its head. “DRAWING” blinks at the lower right corner of screen during drawing. When **[A.ST]** is pressed, the status of machining in progress is drawn by simulation. When **[F.ST]** is pressed, the profile during machining is not drawn. Only the finished profile produced by the program is drawn. This allows drawing to be started.
- 11 Press soft key **[STOP]** to stop drawing temporarily. Drawing is stopped after drawing the current block and “STOP” blinks at the lower right corner of screen. Press soft key **[A.ST]** or **[F.ST]** when restarting drawing. Press soft key **[REWIND]** and then the **[A.ST]** or **[F.ST]** if redrawing from the head. It is possible to continue drawing after changing the solid graphic parameters in temporary stop.
- 12 When the end of program (M02 or M03) is executed, the drawing ends and the blinking of “DRAWING” stops. Then, the final finish figure is drawn on the screen. The drawn figure view is retained until the power is turned off as long as a new machine figure view is drawn.

● REVIEW

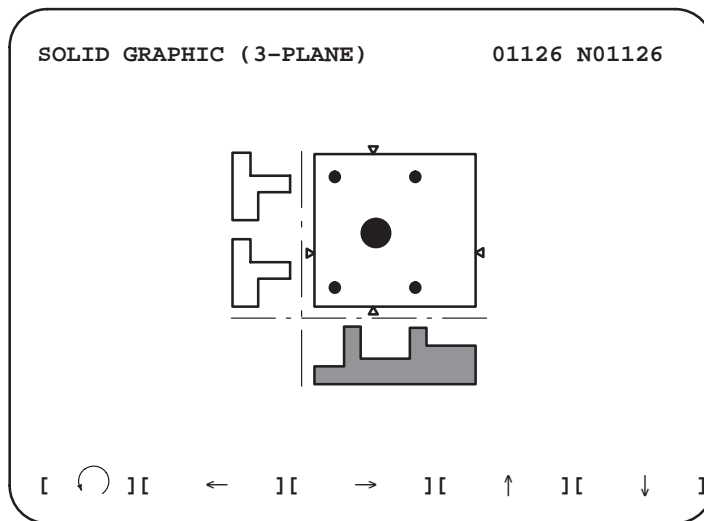
- 13** The color, intensity, or drawing direction of a machining figure which has been drawn can be changed and the figure redrawn.
To redraw the figure, first change the parameters for the color, intensity, or drawing direction on the SOLID GRAPHIC (PARAMETER) screen shown in **step 1**, then press soft key **[REVIEW]** to display the SOLID GRAPHIC (REVIEW) screen.



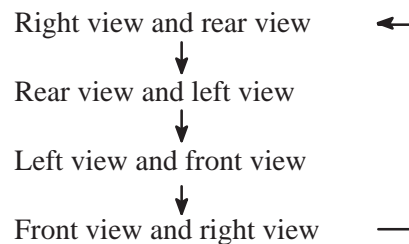
- 14** Press soft key **[ANEW]**. The machining figure is redrawn with the color, intensity, or drawing direction set in **step 13**.
- 15** To redraw the figure in a different direction, press soft key **[+ROT]**, **[-ROT]**, **[+TILT]**, or **[-TILT]**. Parameters P and Q for the drawing direction are changed and the figure is redrawn with the new parameters.

• Triplane view drawing

- 16** The machined figure can be drawn on the tri-plane view.
To draw a triplane view, press soft key **[3-PLN]** on the SOLID GRAPHIC (PARAMETER) screen of **step 1** above. The SOLID GRAPHIC (3-PLANE) screen appears.



- 17** Each time soft key [↺] is pressed, the side-view drawings displayed change as follows.



- 18** The sectional position of side view can be changed by the soft keys [←], [→], [↑], and [↓].

With the sectional position of the left/right side view, the marks ◀ and ▶ indicating the sectional position can be moved using the soft keys [←] and [→].

With the sectional position of rear/front side view, the marks ▲ and ▼ indicating the sectional position can be moved using the soft keys [↑], and [↓]. Keep the keys depressed to change sectional/views continuously.

Explanations

GRAPHICS PARAMETER

• BLANK FORM

◆ BLANK FORM (P)

Set the type of blank figure under P. The relationship between the setting value and figure is as follows:

P	Blank figure
0	Rectangular parallelepiped (Cubed)
1	Column or cylinder (parallel to Z-axis)
2	Column or cylinder (parallel to X-axis)
3	Column or cylinder (parallel to Y-axis)

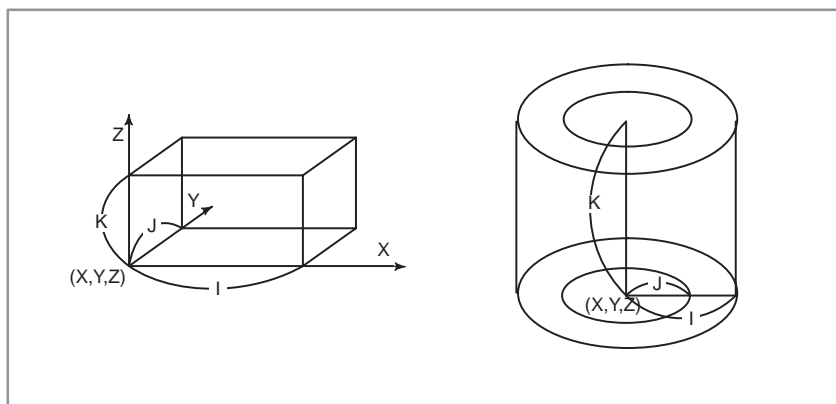
◆ Material positions (X,Y,Z)

Set the X-axis, Y-axis, and Z-axis coordinate values of standard point of materials in workpiece coordinate system to the addresses X, Y, and Z. The standard point of materials is the corner point in the negative direction in the case of rectangular parallelepiped blank figure and the center point of bottom in the case of column and cylinder materials.

◆ Material dimensions (I,J,K)

Set the dimensions of materials. The relationship between the addresses I, J, and K and setting value is as shown below:

Material	I	J	K
Rectangular	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Radius of circle	0	Length of column
Cylinder	Radius of external circle	Radius of internal circle	Length of cylinder



● TOOL FORM

- ◆ Machining tool orientation (P)

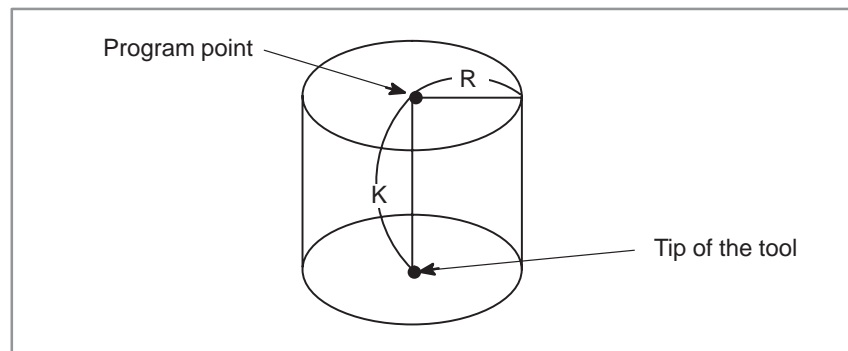
Set the machining direction of tools. The relationship between the setting value and machining direction is as shown below.

P	Machining direction of tools
0,1	Parallel to the Z-axis (perform machining from the + direction)
2	Parallel to the X-axis (perform machining from the + direction)
3	Parallel to the Y-axis (perform machining from the + direction)
4	Parallel to the Z-axis (perform machining from the – direction)
5	Parallel to the X-axis (perform machining from the – direction)
6	Parallel to the Y-axis (perform machining from the – direction)

- ◆ Dimensions of tools (R,K)

Set the dimensions of tool. The relationship between the displayed address and setting value is as shown below:

Address	Setting numerics
R	Radius of tool
K	Distance from the program point to tool tip (normally 0)



● PROJECTION

- ◆ Graphics method and direction (P)

The relationship between graphic method and direction and setting value is as shown below:

P	Graphic method and direction
0, 4	Oblique projection view (+ X-axis)
1, 5	Oblique projection view (+ Y-axis)
2, 6	Oblique projection view (– X-axis)
3, 7	Oblique projection view (–Y-axis)

This setting value can also be incremented or decremented by the soft keys **[+ROT]** or **[– ROT]**. In this case, if the setting value exceeds 7, it returns to 0. If it is smaller than 0, it becomes 7.

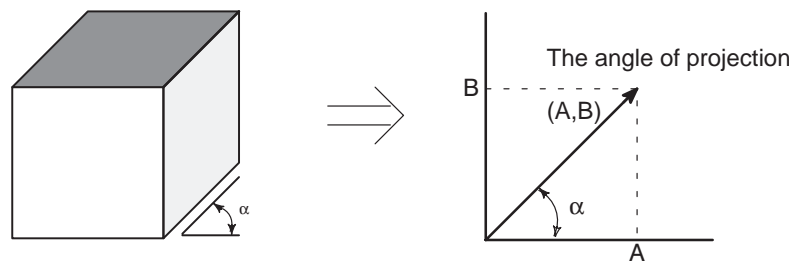
◆ Tilting angle (Q)

Set the slant direction of the projection axis in the case of oblique projection drawing. Moreover, plane view can be specified. The relationship between the setting value and slant direction is as shown below:

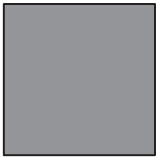

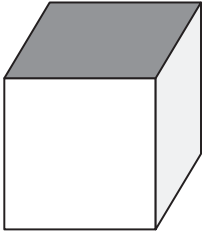
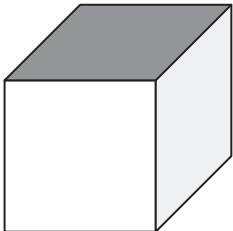
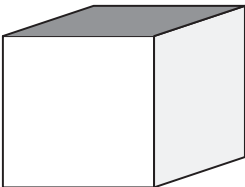
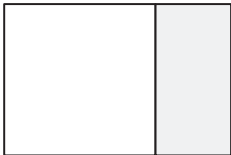
Q	Slant direction
3	Plane view
2	(0,2) direction ; $\alpha=90^\circ$ in the figure below
1	(1,2) direction ; $\alpha=60^\circ$ in the figure below
0	(2,2) direction ; $\alpha=45^\circ$ in the figure below
-1	(2,1) direction ; $\alpha=30^\circ$ in the figure below
-2	(2,0) direction ; $\alpha=0^\circ$ in the figure below

This setting value can also be incremented or decremented by 1 using soft key **[+ TILT]** or **[-TILT]**.

Example) The angle of projection is changed by the set value.



Variations of the above figure determined by set values are shown below.

(i) Set value:3	(ii) Set value: 2	(iii) Set value: 1
 (Two-dimensional drawing)	 ($\alpha=90^\circ$)	 ($\alpha=60^\circ$)
(iv) Set value: 0	(v) Set value: -1	(vi) Set value: -2
 ($\alpha=45^\circ$)	 ($\alpha=30^\circ$)	 ($\alpha=0^\circ$)

◆ VERTICAL AXIS (R)

Set the direction of the vertical axis.

R	VERTICAL AXIS
0, 1	Z-axis
2	X-axis
3	Y-axis

The direction of the vertical axis which is set is effective by executing graph.

● INTENSITY

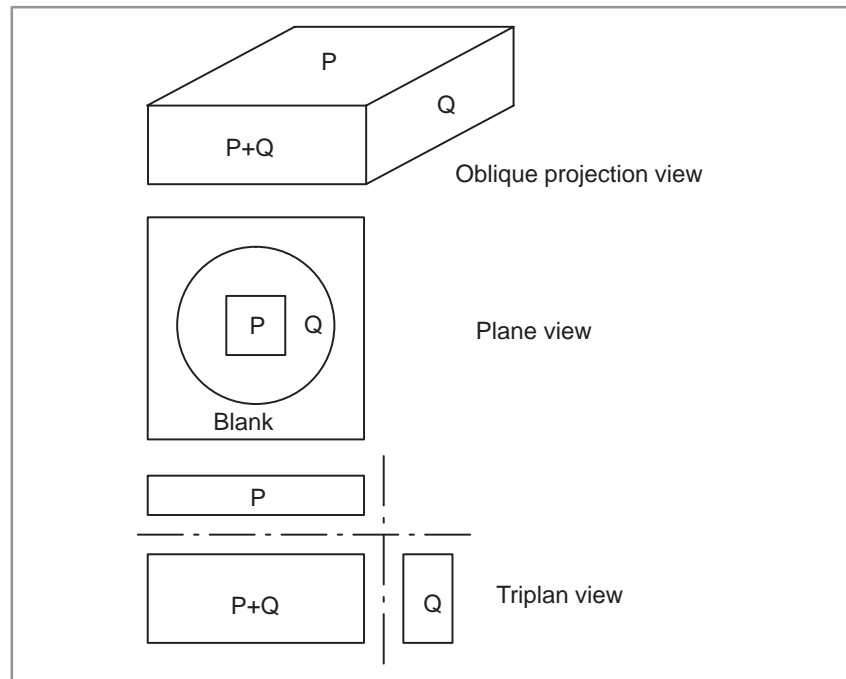
Specify the intensity of the drawing screen when performing drawing on the monochrome screen, and the color of the drawing screen when performing drawing on the color screen. The relationship between the setting, intensity, and color is as shown below:

However, when the plane view is displayed on the monochrome screen. The brighter surface, whichever is specified by P or Q becomes the top surface.

Setting value	Intensity	Color
0	Maximum brightness	White
1	Dark ↓ Light	Red
2		Green
3		Yellow
4		Blue
5		Purple
6		Light blue

The direction of the vertical axis which is set is effective by executing graph.

Address	Oblique projection view	Plane view	Triplane view
P	Upper surface	Upper surface	Upper/lower surface
Q	Side surface	Middle surface	Left/right surface
R	Ridge	Ridge	Ridge
Remarks	The intensity/color of front surface is between P and Q	Lower surface is blank	The intensity/color of plane view is between P and Q



- **START SEQ. NO. and END SEQ. NO.**

Specify the start sequence number and end sequence number of each drawing in a four-digit numeric. The subject part program is executed from the head. But only the part enclosed by the start sequence number and end sequence numeric is drawn. When 0 is set as the start sequence number, the program is drawn from its head. When 0 is set as the end sequence number, the program is drawn to its end. The comparison of sequence number is performed regardless of main program and subprogram.

- **ANIM. SPEED**

Set interval of animated simulation drawing ranging from 0 to 255. Every time the machining proceeds by the number set, the drawing is repeated. If 0 is set, drawing is repeated at every 1 block execution.

- **Soft key functions on the "SOLID GRAPHIC (EXECUTION)" screen**

Soft key	Function
[A.ST]	Simulate and draw the progress of machining.
[F.ST]	No figure during machining is drawn and only the final finish figure by that program is drawn.
[STOP]	When pressed, stops drawing at the end of block (single block stop).
[REWIND]	Press this key to perform drawing from the head of part program. Heading is performed automatically after execution of program end (M02/M30).

- **Graphics program**

No part program which has not been registered in memory can be drawn. It is also necessary that the M02 or M30 be commanded at the end of the part program.

- **Specifying the blank form and tool form in the part program**

It is possible to specify BLANK FORM and TOOL FORM in the part program. The command format is as shown below. If it is commanded during execution of drawing, the item corresponding to the screen of “SOLID GRAPHIC (PARAMETER)” is set and drawing continues with the set data.

- **Command of BLANK FORM**

G10 L90 P_X_Y_Z_I_J_K_ ;

The command value succeeding the address is the same as the numeric set to the address being displayed at the item of BLANK FORM of “SOLID GRAPHIC (PARAMETER)”. If BLANK FORM is commanded, drawing continues after a new blank figure is drawn.

- **Command of TOOL FORM**

G10 L91 P_R_K_ ;

The command value succeeding the address is the same as the numeric set to the address being displayed at the item of TOOL FORM of screen “SOLID GRAPHIC (PARAMETER)”. If 0 is set with the tool radius value, no machining simulation is performed thereafter.

- **Display of the coordinate value**

Parameter No. 046#5 is used to specify whether to display the coordinates of the current position on SOLID GRAPHIC screen.

- **TOOL COMP.**

In solid graphics, parameter No. 058#1 is used to specify whether to apply tool length offset.

- **Graphic method**

Parameter No. 058#2 is used to select whether to draw a triplane view with the third-angle or first-angle projection.

- **Ridge drawing**

Parameter No. 058#3 is used to specify whether to draw ridges in plane view drawing.

- **Display mode**

Parameter No. 058#4 is used to specify whether to display a solid graphics in the rough mode or in the fine mode. When a solid graphics is drawn in the fine mode, the drawing speed is slower than when drawn in the rough mode.

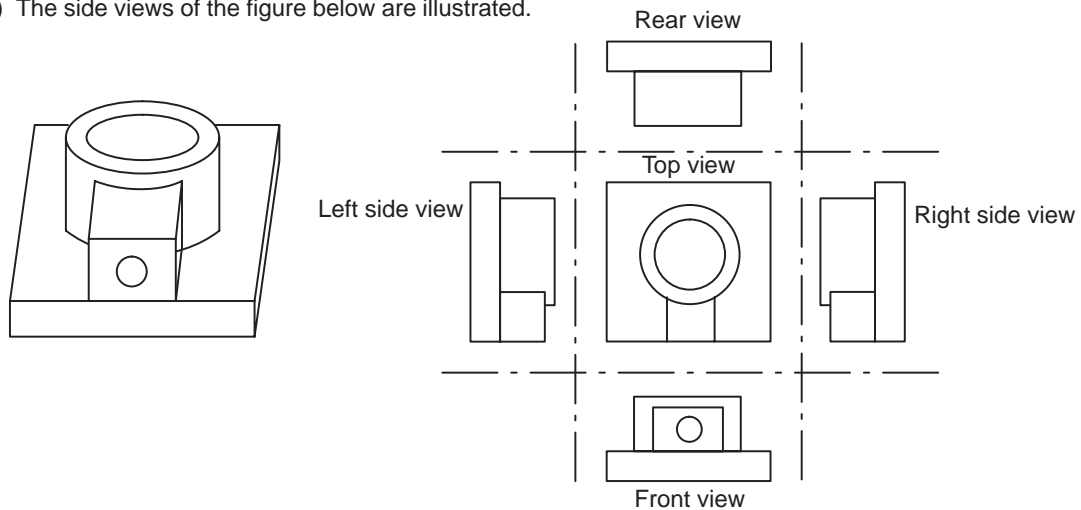
- **Cross section position change**

In triplane drawing, a value can be specified by parameter No. 253 for changing the position of the cross section while the soft key is held down. A value from 0 to 10 can be set. When 0 is specified, a value of 1 is used.

Examples

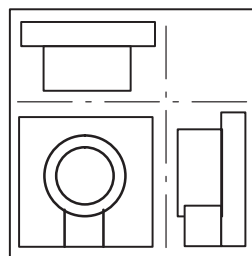
• Side view selection in triplane drawing

Example) The side views of the figure below are illustrated.

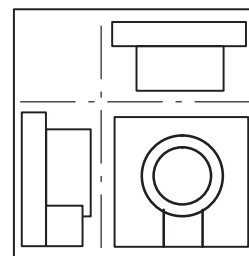


In the above figure, the side views displayed are switched as follows.

Right view and rear view



Rear view and left view



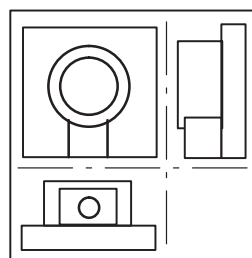
Press [↶]



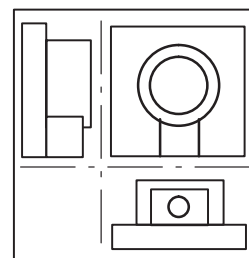
↑
Press [↶]

Press [↶] ↓

Front view and right view



Left view and front view

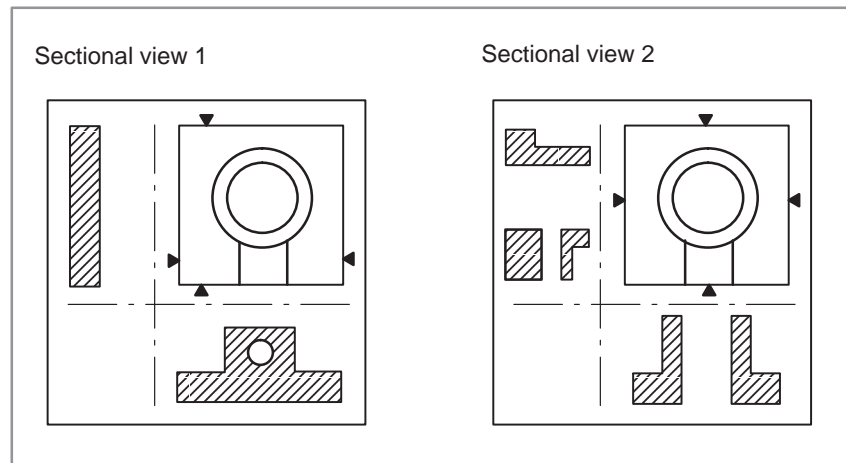


Press [↶]



- **Cross section position selection in triplane drawing**

Some examples of cross-sectional views are given below for the left view and front view shown on the previous page.



Limitations

- **Graphic condition**

If the machine operation is not enabled, no drawing can be made. No drawing can be made during operation of machine. The main setting data and switches needed on drawing are as shown below:

Setting data/switch	Status needed for drawing
Cutter compensation value	It is necessary to set the cutter compensation value properly. A valid tool length compensation value must be specified if a figure is drawn with tool length compensation enabled.
Single block	Off
Optional block skip	Properly set it.
Feed hold	Off

- **Tool form**

Tools which set the tool figure are limited to the cylinder figure (equivalent to flat end mill).

- **Helical interpolation**

In solid graphics, paths based on helical interpolation cannot be drawn.

13 DISPLAY AND OPERATION OF 00-MC

The CRT/MDI panel of 00-MC consists of a CRT display (14" color) and keyboard. Contents of display and operation by key input are completely different depending on whether the CNC screen or MMC screen is displayed on the CRT/MDI panel. In this manual, the operation when only the CNC screen is displayed is described. Refer to the manual of machine tool builder for MMC screen.

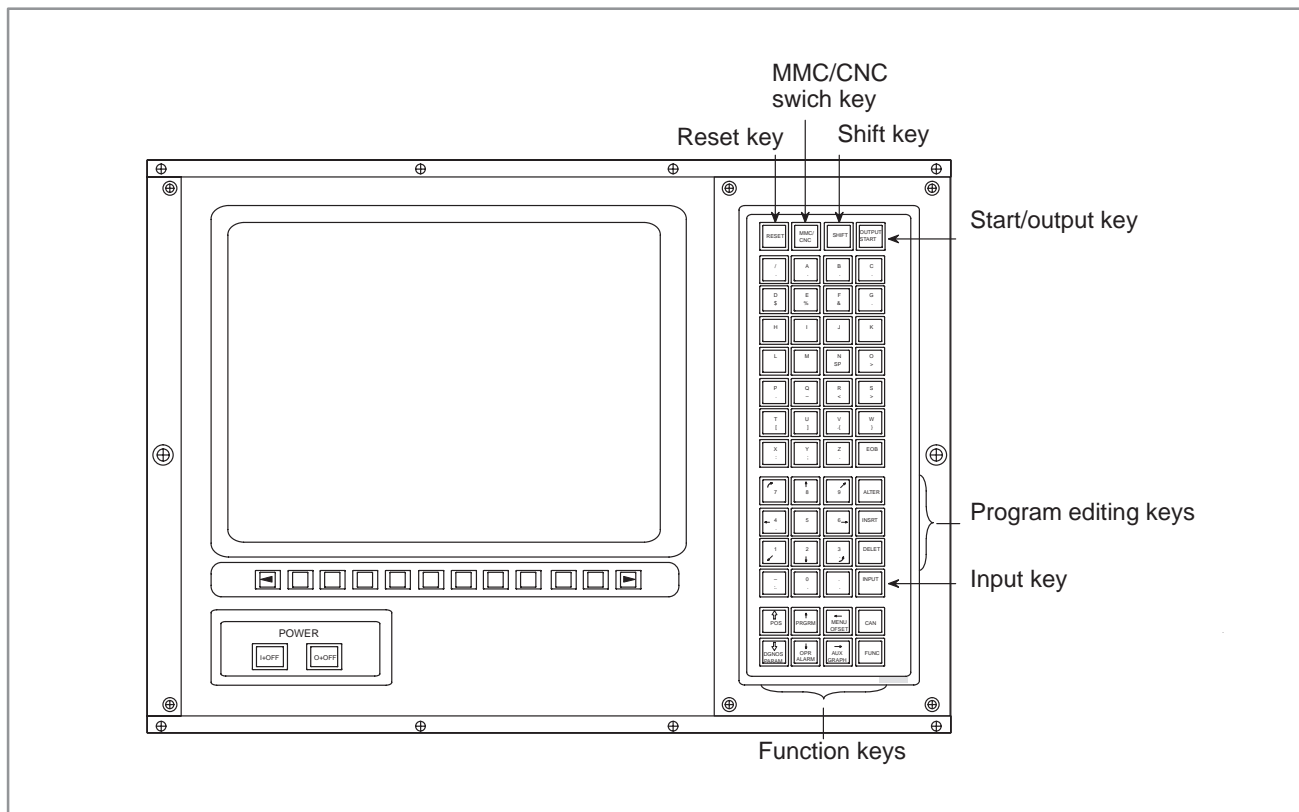


Fig. 13 MDI for 00-MC

13.1

DISPLAY

Press “CNC” key on the CRT/MDI panel to display the CNC screen when the MMC screen is displayed on the CRT display of the CRT/MDI panel. The CNC screen consists of a variable section and a fixed section. The variable section is the part that is surrounded by the frame at the bottom right, and its display contents are the same as displayed on the 9” CRT display of 0-MC. Therefore, the screen selected by function key, page key, cursor key, and soft key is displayed.

The fixed section is the rest of the above variable section, and its display contents are position data, operation time (optional), modal data, and S, T command value, as shown on the screen. Display items of this section cannot be changed by the screen selection operation. However, its display contents are always renewed.

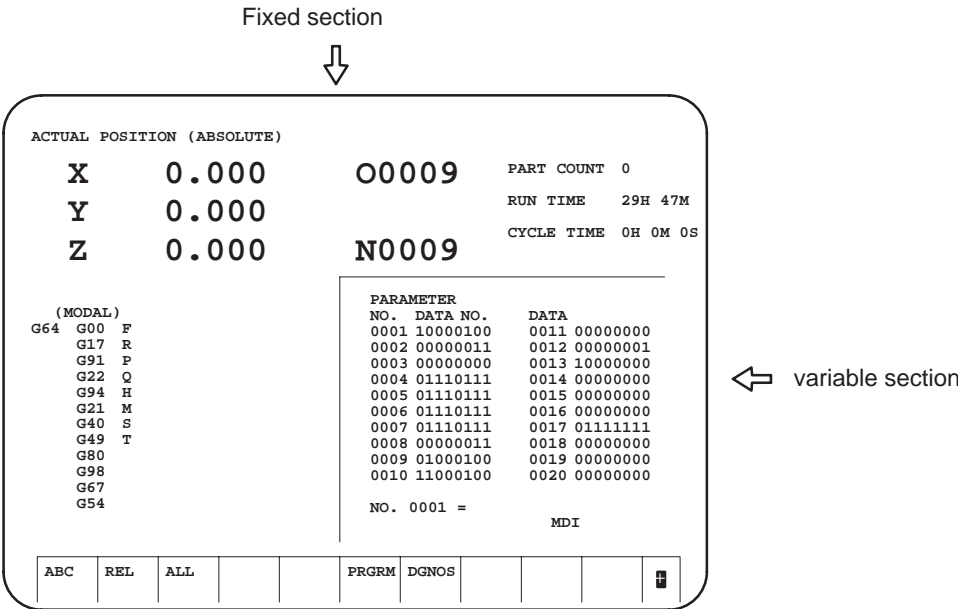









Fig. 13.1

13.2 OPERATION

Key operation can only be done when the CNC screen is displayed on the CRT display of the CRT/MDI panel. Address keys and numerical keys are independently arranged on 00-MB. However, inputting data is exactly the same as that of 0-MB. The page key  , cursor key

 , and selection key   on the software operator's panel are of combined use with the function key. Press the corresponding key for use as a page key, cursor key, and selection key on the software operator's panel. Press the corresponding key while pressing the "FUNC" key as the function key.

Five keys on the right half ten keys are effective for the variable section, and the other five keys on the left half are effective for selecting position display data in the fixed section.

When the number is specified by a method like the parameter screen, because there is no NO. key use the cursor key  instead.

IV. MAINTENANCE

1

METHOD OF REPLACING BATTERY



This chapter describes the method of replacing batteries as follows.


1.1 REPLACING CNC BATTERY FOR MEMORY BACK-UP

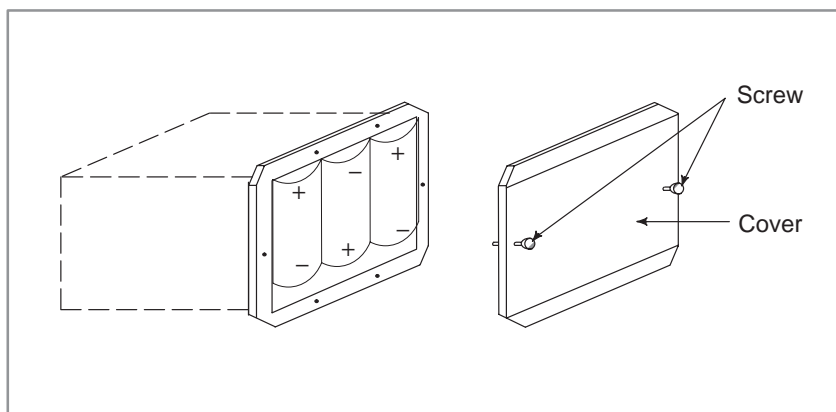
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

1.1 REPLACING CNC BATTERY FOR MEMORY BACK-UP

When the message “BAT” appears at the bottom of the screen, replace the backup batteries for the CNC memory according to the procedure described below.

Procedure for replacing CNC battery for memory back-up

- 1 Have three commercially available fresh alkaline R20 (D) batteries handy.
- 2 Switch on the machine (CNC). (When replacing the batteries, keep the CNC switched on. If you replace the batteries with the CNC switched off, the memory contents will be lost.)
- 3 Loosen the screws for the battery case cover and remove it. For the location of the battery case, refer to the relevant operating manual issued by the machine builder.
- 4 Replace the batteries in the battery case with the fresh ones you have handy. Be careful for the orientation of the batteries. Of the three batteries, the middle one is in the orientation opposite to the other two.
- 5 After replacing the batteries, put the battery case cover back in place.
- 6 Press the  key and ensure the message “BAT” disappears.



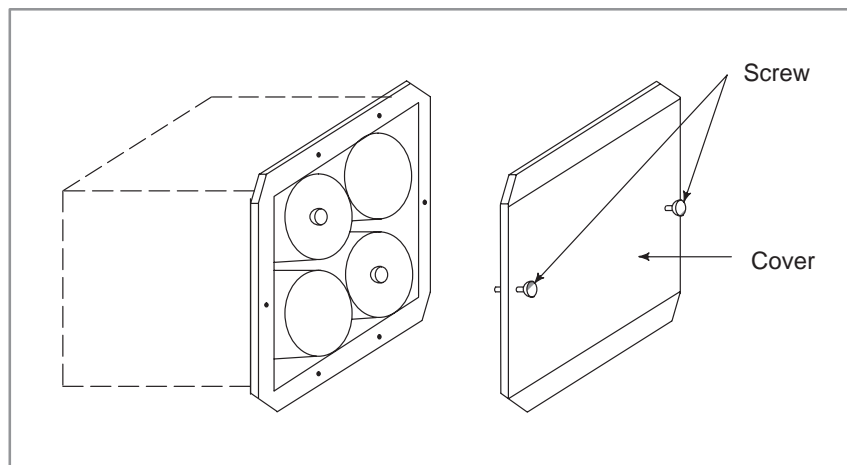
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

If absolute pulse coder alarm 3n7 (where n is an axis number) occurs, replace the batteries (alkaline) for the absolute pulse coder according to the procedure described below.

Procedure for replacing batteries for absolute pulse coder

Procedure

- 1 Have three commercially available fresh alkaline R20 (D) batteries handy.
- 2 Switch on the machine (CNC). (When replacing the batteries, keep the CNC switched on. If you replace the batteries with the CNC switched off, the memory contents will be lost.)
- 3 Loosen the screws for the battery case cover and remove it. For the location of the battery case, refer to the relevant operating manual issued by the machine builder.
- 4 Replace the batteries in the battery case with the fresh ones you have handy. Be careful for the orientation of the batteries. (Place the two batteries in the orientation opposite to the other two as shown below.)
- 5 After replacing the batteries, put the battery case cover back in place.
- 6 Switch the machine (CNC) off and on again, and ensure that alarm 3n7 does not occur.



APPENDIX

A

TAPE CODE LIST

ISO code										EIA code										Meaning		
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Without custom macro B	With custom macro B
0			○	○		○				0			○			○				Number 0		
1	○		○	○		○			○	1						○			○	Number 1		
2	○		○	○		○		○		2						○		○		Number 2		
3			○	○		○		○	○	3			○		○		○	○		Number 3		
4	○		○	○		○	○			4						○	○			Number 4		
5			○	○		○	○		○	5			○		○	○		○		Number 5		
6			○	○		○	○	○		6			○		○	○	○			Number 6		
7	○		○	○		○	○	○	○	7						○	○	○	○	Number 7		
8	○		○	○	○	○				8				○	○					Number 8		
9			○	○	○	○			○	9			○	○	○				○	Number 9		
A		○				○			○	a		○	○			○			○	Address A		
B		○				○		○		b		○	○			○		○		Address B		
C	○	○				○		○	○	c		○	○	○		○		○	○	Address C		
D		○				○	○			d		○	○			○	○			Address D		
E	○	○				○	○		○	e		○	○	○		○	○		○	Address E		
F	○	○				○	○	○		f		○	○	○		○	○	○		Address F		
G		○				○	○	○	○	g		○	○			○	○	○	○	Address G		
H		○			○	○				h		○	○		○	○				Address H		
I	○	○			○	○			○	i		○	○	○	○	○			○	Address I		
J	○	○			○	○		○		j		○		○		○		○	○	Address J		
K		○			○	○		○	○	k		○		○		○		○		Address K		
L	○	○			○	○	○			l		○				○		○	○	Address L		
M		○			○	○	○		○	m		○		○		○	○			Address M		
N		○			○	○	○	○		n		○				○	○		○	Address N		
O	○	○			○	○	○	○	○	o		○				○	○	○		Address O		
P		○		○		○				p		○		○		○	○	○	○	Address P		
Q	○	○		○		○			○	q		○		○	○	○				Address Q		
R	○	○		○		○		○		r		○			○	○			○	Address R		
S		○		○		○		○	○	s			○	○		○		○		Address S		
T	○	○		○		○	○			t			○			○		○	○	Address T		
U		○		○		○	○		○	u			○	○		○	○			Address U		
V		○		○		○	○	○		v			○			○	○		○	Address V		
W	○	○		○		○	○	○	○	w			○			○	○	○		Address W		
X	○	○		○	○	○				x			○	○		○	○	○	○	Address X		
Y		○		○	○	○			○	y			○	○	○	○				Address Y		
Z		○		○	○	○		○		z			○		○	○			○	Address Z		

ISO code										EIA code										Meaning		
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Without custom macro B	With custom macro B
DEL	○	○	○	○	○	○	○	○	○	Del		○	○	○	○	○	○	○	○		×	×
NUL						○				Blank						○					×	×
BS	○				○	○				BS			○		○	○		○			×	×
HT					○	○			○	Tab			○	○	○	○	○	○			×	×
LF or NL					○	○		○		CR or EOB	○					○						
CR	○				○	○	○		○	—											×	×
SP	○		○			○				SP				○		○					□	□
%	○		○			○	○		○	ER					○	○		○	○			
(○		○	○				(2-4-5)				○	○	○		○				
)	○		○		○	○			○	(2-4-7)		○			○	○		○				
+			○		○	○		○	○	+		○	○	○		○					×	
—			○		○	○	○		○	—		○				○						
:			○	○	○	○		○		—												
/	○		○		○	○	○	○	○	/			○	○		○			○			
.			○		○	○	○	○		.		○	○		○	○		○	○			
#	○		○			○		○	○	Parameter (No.0044)												
\$			○			○	○			—											Δ	○
&	○		○			○	○	○		&					○	○	○	○			Δ	○
▼			○			○	○	○	○	—											Δ	○
*	○		○		○	○		○		Parameter (No.0042)											Δ	
,	○		○		○	○	○			,			○	○	○	○		○	○			
;	○		○	○	○	○		○	○	—											Δ	Δ
<			○	○	○	○	○			—											Δ	Δ
=	○		○	○	○	○	○		○	Parameter (No.0043)											Δ	
>	○		○	○	○	○	○	○		—											Δ	Δ
?			○	○	○	○	○	○	○	—											Δ	○
@	○	○				○				—											Δ	○
"			○			○		○		—											Δ	Δ
[○	○		○	○	○		○	○	Parameter (No.0053)											Δ	
]	○	○		○	○	○	○		○	Parameter (No.0054)											Δ	

NOTE

- 1 The symbols used in the remark column have the following meanings.
 - (Space) : The character will be registered in memory and has a specific meaning.
If it is used incorrectly in a statement other than a comment, an alarm occurs.
 - × : The character will not be registered in memory and will be ignored.
 - △ : The character will be registered in memory, but will be ignored during program execution.
 - : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
 - : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.
- 2 Codes not in this table are ignored if their parity is correct.
- 3 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.
- 4 A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.



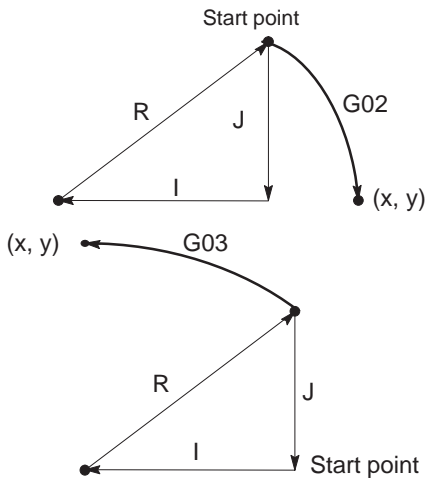
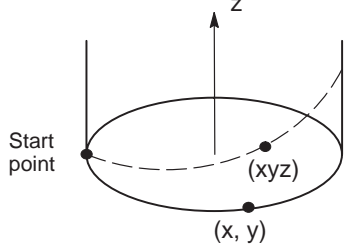
B LIST OF FUNCTIONS AND TAPE FORMAT

Some functions cannot be added as options depending on the model.
In the tables below, IP _; presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_Y_Z_A_).

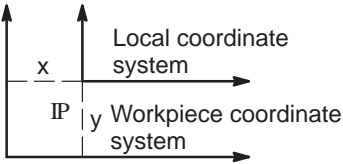
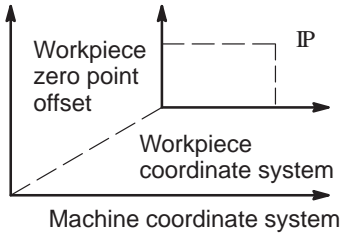
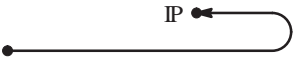
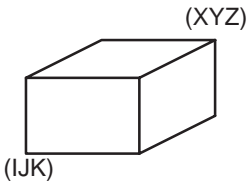
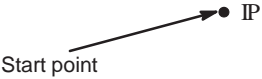
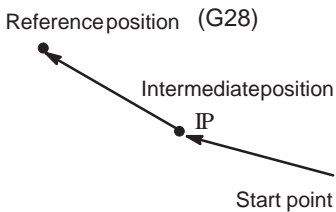
x = 1st basic axis (X usually)

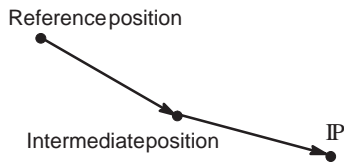
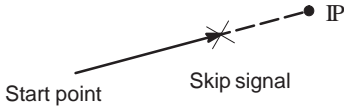
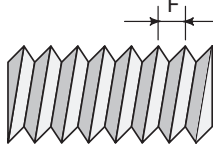
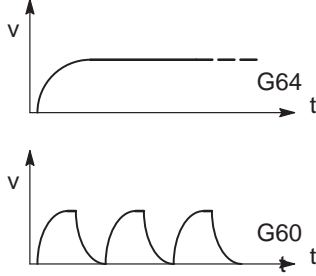
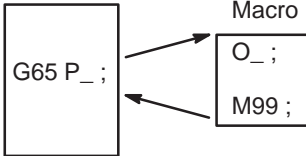
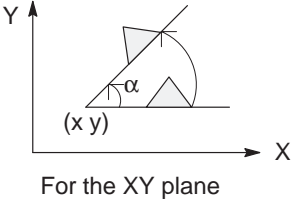
y = 2nd basic axis (Y usually)

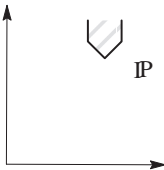
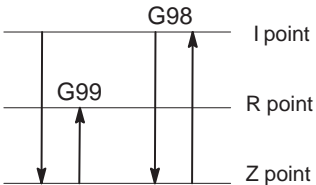
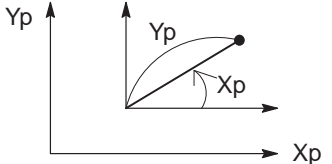
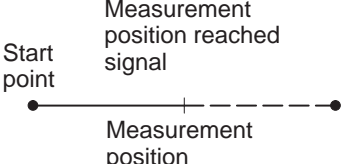
z = 3rd basic axis (Z usually)

Functions	Illustration	Tape format
Positioning (G00)		G00IP _ ;
Linear interpolation (G01)		G01IP _ 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 cutting (G02, G03)		$G17 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_ Y_ \left\{ \begin{matrix} R_ \\ I_ J_ \end{matrix} \right\} \alpha_ F_ ;$ $G18 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_ Z_ \left\{ \begin{matrix} R_ \\ I_ K_ \end{matrix} \right\} \alpha_ F_ ;$ $G19 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y_ Z_ \left\{ \begin{matrix} R_ \\ J_ K_ \end{matrix} \right\} \alpha_ F_ ;$ <p>α : Arbitrary address except a circular interpolation axis</p>
Dwell (G04)		$G04 \left\{ \begin{matrix} X_ \\ P_ \end{matrix} \right\} ;$

Functions	Illustration	Tape format
Exact stop (G09)		$G09 \left\{ \begin{matrix} G01_ \\ G02_ \\ G03_ \end{matrix} \right\} -;$
Change of offset value by program (G10)		Tool offset value (offset memory B) $G10 L10 P_ R_;$ Wear offset value (offset memory B) $G10 L11 P_ R_;$ Workpiece origin offset value $G10 L2 P_ IP_;$ $G10 L20 P_ IP_;$
Cutter compensation (G40 – G42)		$\left\{ \begin{matrix} G17 \\ G18 \\ G19 \end{matrix} \right\} \left\{ \begin{matrix} G40 \\ G41 \\ G42 \end{matrix} \right\} IP_ \left\{ \begin{matrix} H_ \\ D_ \end{matrix} \right\}$ H, D : Cutter compensation number G40 : Cancel
Tool length offset (G43, G44, G49)		$\left\{ \begin{matrix} G43 \\ G44 \end{matrix} \right\} \alpha_ H_;$ $\left\{ \begin{matrix} G43 \\ G44 \end{matrix} \right\} H_;$ H : Tool offset number G49 : Cancel
Tool offset (G45 – G48)		$\left\{ \begin{matrix} G45 \\ G46 \\ G47 \\ G48 \end{matrix} \right\} IP_ D_;$ D : Tool offset number P : Scale factor
Scaling (G50, G51)		$G51 IP_ P_;$ P : G50 ; Cancel

Functions	Illustration	Tape format
Local coordinate system setting (G52)		G52 IP_;
Machine coordinate system selection (G53)		G53 IP_;
Workpiece coordinate system selection (G54 – G59)		$\left\{ \begin{array}{c} G54 \\ \vdots \\ G59 \end{array} \right\} IP_;$
Single direction positioning (G50)		G60 IP_;
Inch/millimeter conversion (G20, G21)		G20 ; Inch input G21 ; Millimeter input
Stored stroke check (G22, G23)		G22 X_ Y_ Z_ I_ J_ K_ ; G23 ; Cancel
Reference position return check (G27)		G27 IP_;
Reference position return (G28) 2nd, 3rd, 4th reference position return (G30)		G28 IP_ ; G30P $\left\{ \begin{array}{c} P2 \\ P3 \\ P4 \end{array} \right\} IP_ ;$

Functions	Illustration	Tape format
Return from reference position to start point (G29)	 <p>Reference position</p> <p>Intermediate position</p> <p>IP</p>	G29 IP _ ;
Skip function (G31)	 <p>Start point</p> <p>Skip signal</p> <p>IP</p>	G31 IP _ F _ ;
Thread cutting (G33)	 <p>F</p>	Equal lead thread cutting G33 IP _ F _ ;
Cutting mode, Exact stop mode, Tapping mode, Automatic corner override mode	 <p>G64</p> <p>G60</p>	G64 _ ; Cutting mode G60 _ ; Exact stop mode G62 _ ; Automatic corner override mode G63 _ ; Tapping mode
Custom macro (G65, G66, G67)	 <p>G65 P _ ;</p> <p>Macro</p> <p>O _ ;</p> <p>M99 ;</p>	One-shot call G65P <argument>; P: Program number Modal call G66 P <argument> G67; cancel
Coordinate system rotation (G68, G69)	 <p>Y</p> <p>X</p> <p>(x y)</p> <p>α</p> <p>For the XY plane</p>	$G68 \left\{ \begin{array}{l} G17 X_ Y_ \\ G18 Z_ X_ \\ G19 Y_ Z_ \end{array} \right\} R_{\underline{\alpha}} ;$ <p>G69 ; Cancel</p>

Functions	Illustration	Tape format
Canned cycles (G73, G74, G80 – G89)	Refer to II.14. FUNCTIONS TO SIMPLIFY PROGRAMMING	G80 ; Cancel <div> <div>G73</div> <div>G74</div> <div>G76</div> <div>G81</div> <div>:</div> <div>G89</div> </div> <div> <div>X_ Y_ Z_ P_ Q_ R_ F_ K_ ;</div> </div>
Absolute/incremental programming (G90/G91)		G90_ ; Absolute command G91_ ; Incremental command G90_ G91_ ; Combined use
Change of workpiece coordinate system (G92)		G92 IP_ ;
Initial point return / R point return (G98, G99)		G98-- ; G99-- ;
Constant surface speed control (G96, G97)	m/min or feet/min	G96 S_ ; G97 ; Cancel
Polar coordinate input (G15, G16)	<p>Local coordinate system</p>  <p>Workpiece coordinate system</p>	G17 G16 Xp_ Yp ... ; G18 G16 Zp_ Xp ... ; G19 G16 Yp_ Zp ... ; G15 ; Cancel
Automatic tool length measurement (G37)		G37 Z... ;

C RANGE OF COMMAND VALUE

Linear axis

- In case of millimeter input, feed screw is millimeter

	Increment system	
	IS-B	IS-C
Least input increment	0.001 mm	0.0001 mm
Least command increment	0.001 mm	0.0001 mm
Max. program-mable dimension	±99999.999 mm	±9999.9999 mm
Max. rapid traverse Notes	100000 mm/min	24000 mm/min
Feedrate range Notes	1 to 100000 mm/min	1 to 12000 mm/min
Incremental feed	0.001, 0.01, 0.1, 1 mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

- In case of inch input,
feed screw is millimeter

	Increment system	
	IS-B	IS-C
Least input increment	0.0001 inch	0.00001 inch
Least command increment	0.001 mm	0.0001 mm
Max. program-mable dimension	±9999.9999 inch	±393.70078 inch
Max. rapid traverse Notes	100000 mm/min	24000 mm/min
Feedrate range Notes	0.01 to 4000 inch/min	0.01 to 480 inch/min
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.9999 inch	0 to ±99.99999 inch
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

- In case of inch input,
feed screw is inch

	Increment system	
	IS-B	IS-C
Least input increment	0.0001 inch	0.00001 inch
Least command increment	0.0001 inch	0.00001 inch
Max. program-mable dimension	±9999.9999 inch	±999.99999 inch
Max. rapid traverse Notes	4000 inch/min	960 inch/min
Feedrate range Notes	0.01 to 4000 inch/min	0.01 to 480 inch/min
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.9999 inch	0 to ±99.99999 inch
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

- In case of millimeter input, feed screw is inch

	Increment system	
	IS-B	IS-C
Least input increment	0.001 mm	0.0001 mm
Least command increment	0.0001 inch	0.00001 inch
Max. program-mable dimension	±99999.999 mm	±9999.9999 mm
Max. rapid traverse Notes	4000 inch/min	960 inch/min
Feedrate range Notes	1 to 100000 mm/min	1 to 12000 mm/min
Incremental feed	0.001, 0.01, 0.1, 1 mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

Rotation axis

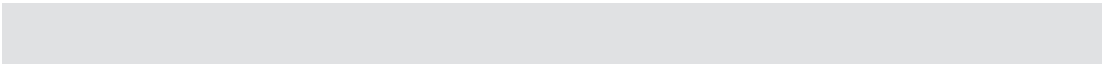
	Increment system	
	IS-B	IS-C
Least input increment	0.001 deg	0.0001 deg
Least command increment	0.001 deg	0.0001 deg
Max. program-mable dimension	±99999.999 deg	±9999.9999 deg
Max. rapid traverse Notes	100000 deg/min	24000 deg/min
Feedrate range (metric input) Notes	1 to 100000 deg/min	1 to 12000 deg/min
Feedrate range (inch input) Notes	1 to 6000 deg/min	1 to 600 deg/min
Incremental feed	0.001, 0.01, 0.1, 1 deg/step	0.0001, 0.001, 0.01, 0.1 deg/step

NOTE

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.

D

NOMOGRAPHS



D.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2 , as shown in Fig. D.1 (a), due to automatic acceleration and deceleration. Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

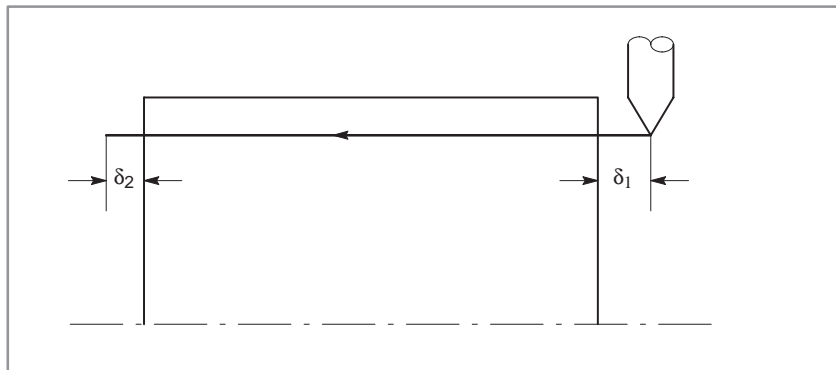


Fig. D.1 (a) Incorrect thread position

Explanations

• How to determine δ_2

$$\delta_2 = T_1 V \text{ (mm)} \dots\dots\dots (1)$$

$$V = \frac{1}{60} RL$$

T_1 : Time constant of servo system (sec)

V : Cutting speed (mm/sec)

R : Spindle speed (rpm)

L : Thread feed (mm)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

• How to determine δ_1

$$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V \dots\dots\dots (2)$$

$$\alpha = \exp(-\frac{t}{T_1}) \dots\dots\dots (3)$$

T_1 : Time constant of servo system (sec)

V : Cutting speed (mm/sec)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

The lead at the beginning of thread cutting is shorter than the specified lead L , and the allowable lead error is ΔL . Then as follows.

$$a = \frac{\Delta L}{L}$$

When the value of “ a ” is determined, the time lapse until the thread accuracy is attained. The time “ t ” is substituted in (2) to determine δ_1 : Constants V and T_1 are determined in the same way as for δ_2 . Since the calculation of δ_1 is rather complex, a nomography is provided on the following pages.

- How to use nomograph

First specify the class and the lead of a thread. The thread accuracy, a , will be obtained at [1], and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when $V = 10\text{mm/s}$ will be obtained at [2]. Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/s can be obtained at [3].

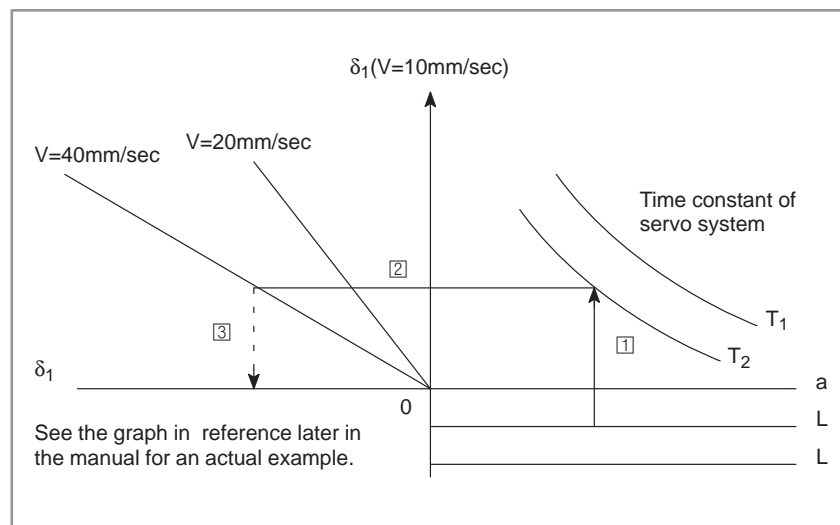


Fig. D.1 (b) Nomograph

NOTE

The equations for δ_1 , and δ_2 are for when the acceleration/deceleration time constant for cutting feed is 0.

D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

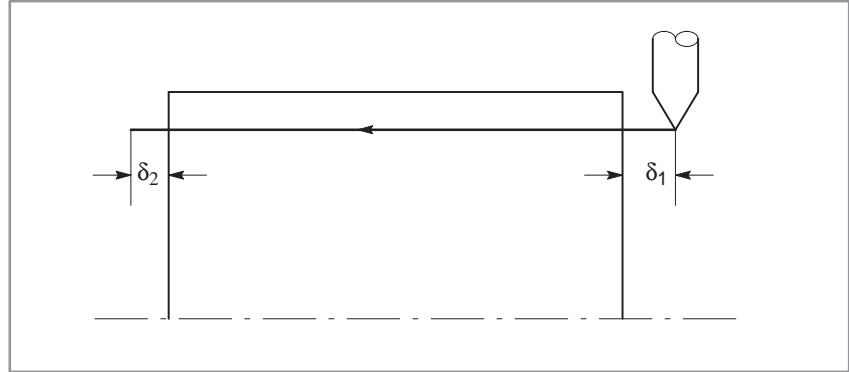


Fig. D.2 Incorrect threaded portion

Explanations

- How to determine δ_2

$$\delta_2 = \frac{LR}{1800 * } \text{ (mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

- How to determine δ_1

$$\begin{aligned} \delta_1 &= \frac{LR}{1800 * } (-1 - \ln a) \text{ (mm)} \\ &= \delta_2 (-1 - \ln a) \text{ (mm)} \end{aligned}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

Following a is a permitted value of thread.

a	-1 - ln a
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

Examples

R=350rpm

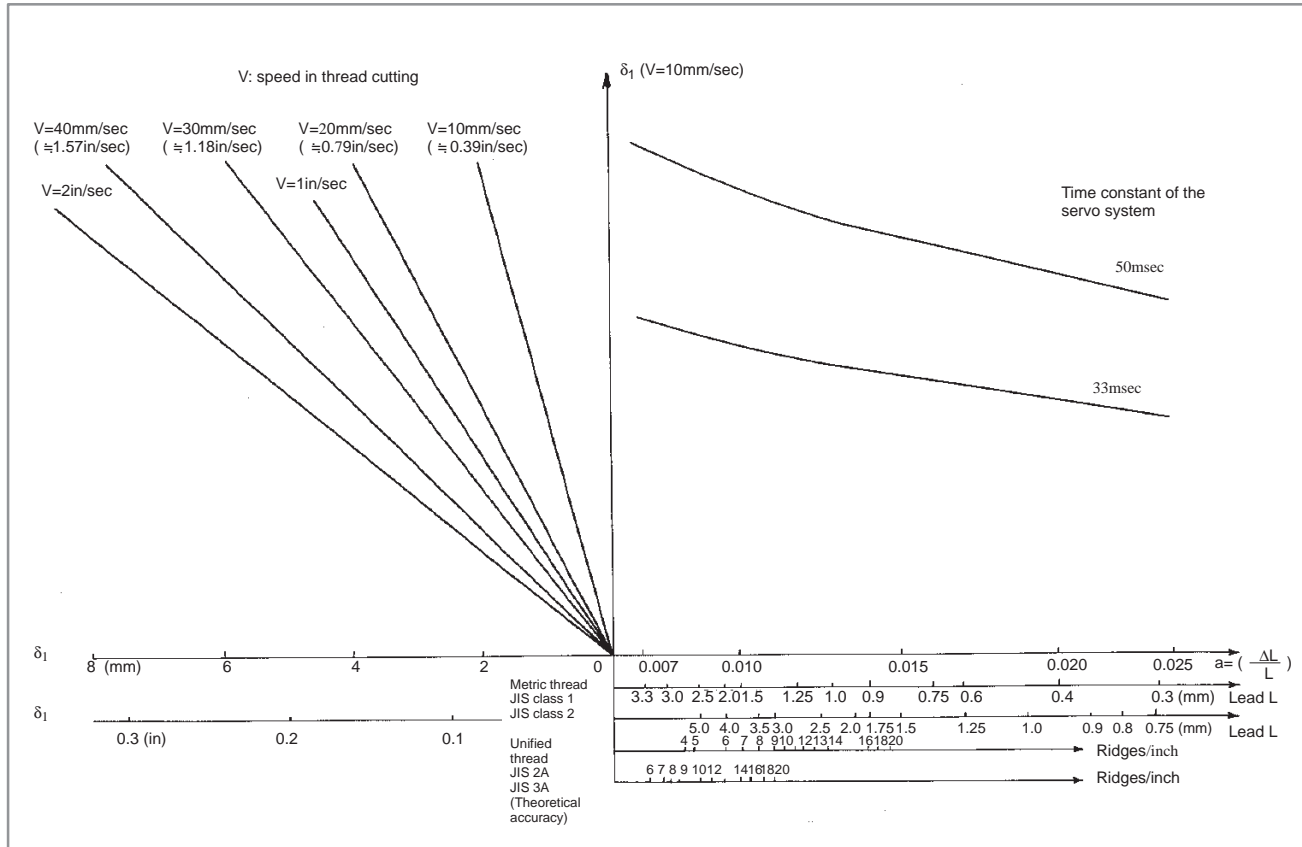
L=1mm

a=0.01 then

$$\delta_2 = \frac{350 \times 1}{1800} = 0.194 \text{ (mm)}$$

$$\delta_1 = \delta_2 \times 3.605 = 0.701 \text{ (mm)}$$

● Reference



Nomograph for obtaining approach distance δ_1

D.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. D.3 (a).

Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

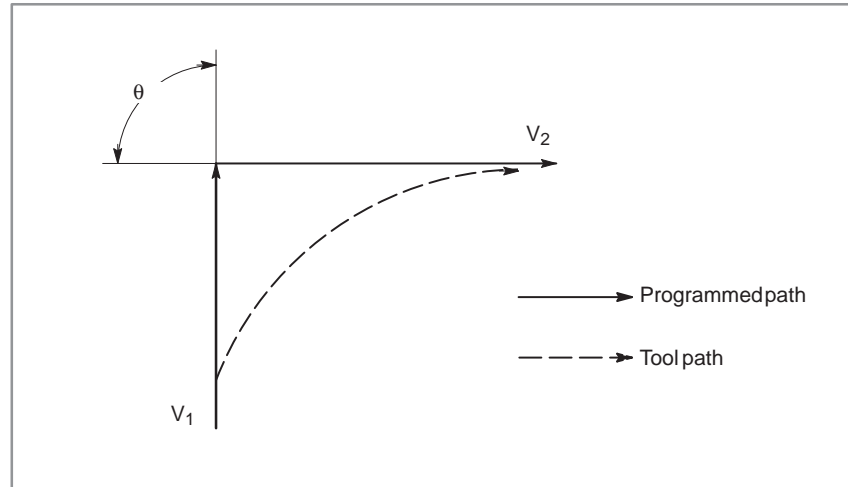


Fig. D.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:

- Feedrate (V_1 , V_2)
- Corner angle (θ)
- Exponential acceleration / deceleration time constant (T_1) at cutting ($T_1 = 0$)
- Presence or absence of buffer register.

The above parameters are used to theoretically analyze the tool path.

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.

Analysis

The tool path shown in Fig. D.3 (b) is analyzed based on the following conditions:

Feedrate is constant at both blocks before and after cornering.

The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

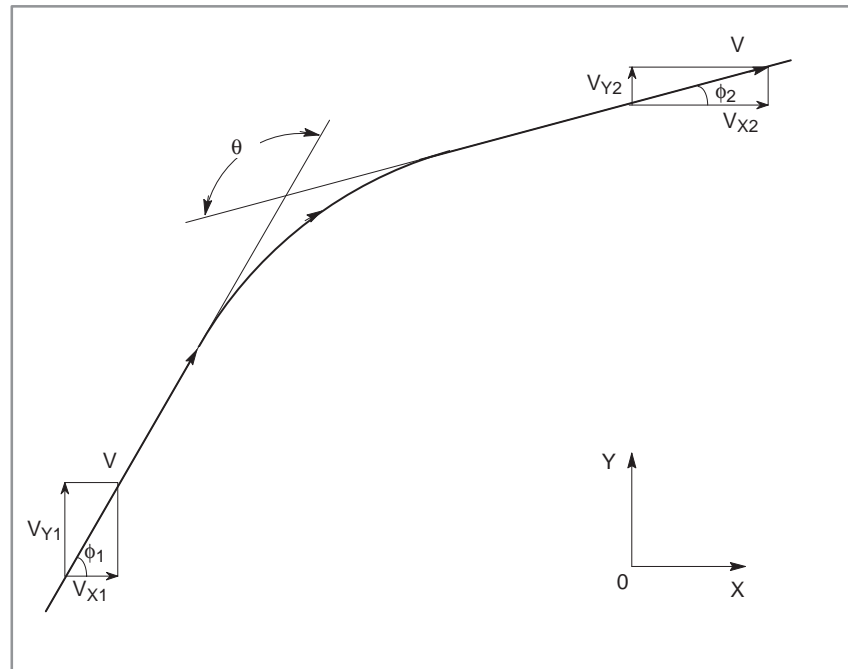


Fig. D.3 (b) Example of tool path

• Description of conditions and symbols

$$V_{X1} = V \cos \phi_1$$

$$V_{Y1} = V \sin \phi_1$$

$$V_{X2} = V \cos \phi_2$$

$$V_{Y2} = V \sin \phi_2$$

- | | | |
|----------|---|--|
| V | : | Feedrate at both blocks before and after cornering |
| V_{X1} | : | X-axis component of feedrate of preceding block |
| V_{Y1} | : | Y-axis component of feedrate of preceding block |
| V_{X2} | : | X-axis component of feedrate of following block |
| V_{Y2} | : | Y-axis component of feedrate of following block |
| θ | : | Corner angle |
| ϕ_1 | : | Angle formed by specified path direction of preceding block and X-axis |
| ϕ_2 | : | Angle formed by specified path direction of following block and X-axis |

- Initial value calculation

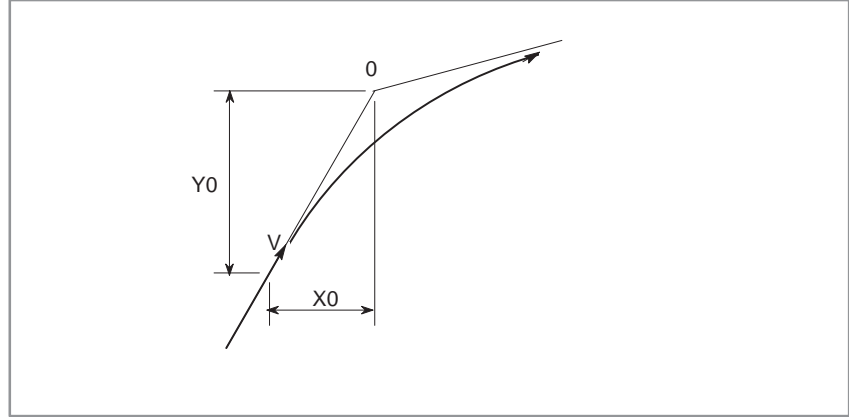


Fig. D.3 (c) Initial value

The initial value when cornering begins, that is, the X and Y coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

$$X_0 = V_{x1}(T_1 + T_2)$$

$$Y_0 = V_{y1}(T_1 + T_2)$$

T_1 : Exponential acceleration / deceleration time constant. ($T=0$)

T_2 : Time constant of positioning system (Inverse of position loop gain)

- Analysis of corner tool path

The equations below represent the feedrate for the corner section in X-axis direction and Y-axis direction.

$$V_x(t) = \frac{V_{x1}-V_{x2}}{T_1-T_2} \{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{x2}$$

$$V_y(t) = \frac{V_{y1}-V_{y2}}{T_1-T_2} \{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

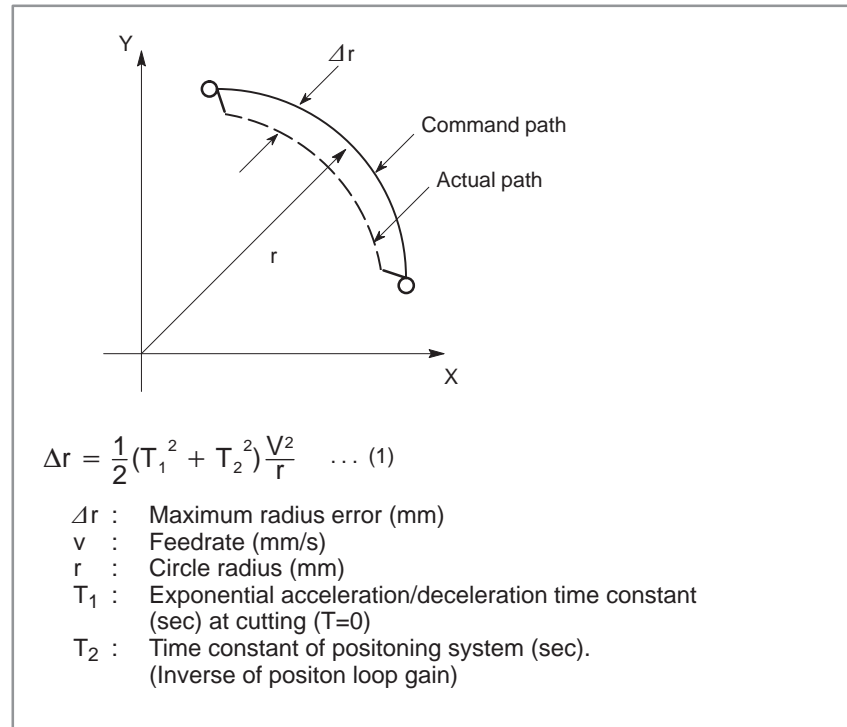
$$\begin{aligned} X(t) &= \int_0^t V_x(t) dt - X_0 \\ &= \frac{V_{x2}-V_{x1}}{T_1-T_2} \{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{x2}(T_1 + T_2 - t) \\ Y(t) &= \int_0^t V_y(t) dt - Y_0 \\ &= \frac{V_{y2}-V_{y1}}{T_1-T_2} \{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{y2}(T_1 + T_2 - t) \end{aligned}$$

D.4

RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, sepecially 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.

E STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

Parameter No. 046 bit6 is used to select whether resetting the CNC places it in the cleared state or in the reset state (0: reset state/1: cleared state). The symbols in the tables below mean the following :

○ : The status is not changed or the movement is continued.

× : The status is cancelled or the movement is interrupted.

Item		When turning power on	Cleared	Reset
Setting data	Offset value	○	○	○
	Data set by the MDI setting operation	○	○	○
	Parameter	○	○	○
Various data	Programs in memory	○	○	○
	Contents in the buffer storage	×	×	○ : MDI mode × : Other mode
	Display of sequence number	○	○ (Note 1)	○ (Note 1)
	One shot G code	×	×	×
	Modal G code	Initial G codes. (The G20 and G21 codes return to the same state they were in when the power was last turned off.)	Initial G codes. (G20/G21 are not changed.)	○
	F	Zero	Zero	○
	S, T, M	×	○	○
	K (Number of repeats)	×	×	×
Work coordinate value		Zero	○	○

Item		When turning power on	Cleared	Reset
Action in operation	Movement	×	×	×
	Dwell	×	×	×
	Issuance of M, S and T codes	×	×	×
	Tool length compensation	×	Depending on parameter No.001#3	○ : MDI mode Other modes depend on parameter No. 001#3.
	Cutter compensation	×	×	○ : MDI mode × : Other modes
	Storing called subprogram number	×	× (Note 2)	○ : MDI mode × : Other modes(Note 2)
Output signals	CNC alarm signal AL	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm
	Reference position return completion LED	×	○ (× : Emergency stop)	○ (× : Emergency stop)
	S, T and B codes	×	○	○
	M code	×	×	×
	M, S and T strobe signals	×	×	×
	Spindle revolution signal (S analog signal)	×	○	○
	CNC ready signal MA	ON	○	○
	Servo ready signal SA	ON (When other than servo alarm)	ON (When other than servo alarm)	ON (When other than servo alarm)
	Cycle start LED (STL)	×	×	×
	Feed hold LED (SPL)	×	×	×

NOTE

- 1 When heading is performed, the main program number is displayed as the sequence number.
- 2 When a reset is performed during execution of a subprogram, control returns the main program. Execution cannot be started from the middle of the subprogram.

F

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	Hash sign
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		^	092	
3	051		¥	093	Yen sign
4	052]	094	Right square bracket
5	053		_	095	Underscore

G ALARM LIST

1) Program errors (P/S alarm)

Number	Meaning	Contents and remedy
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign “-” was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign “.” input error (Sign “-” was input after an address with which it cannot be used. Or two or more “-” signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point “.” input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
008	ILLEGAL USE OF PROGRAM END	An attempt was made to execute EOR (%) because there was not M02, M30, or M99 at the end of the program. Correct the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G-CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
014	CAN NOT COMMAND G95	A synchronous feed is specified without the option for threading / synchronous feed.
015	TOO MANY AXES COMMANDED	The number of the commanded axes exceeded that of simultaneously controlled axes.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 876.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
025	CANNOT COMMAND F0 IN G02/G03	F0 (fast feed) was instructed by F1 –digit column feed in circular interpolation. Modify the program.

Number	Meaning	Contents and remedy
027	NO AXES COMMANDED IN G43/G44	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
029	ILLEGAL OFFSET VALUE	The offset values specified by H code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER	The offset number specified by D/H code for tool length offset or cutter compensation is too large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	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.
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for cutter compensation C. Modify the program.
034	NO CIRC ALLOWED IN ST-UP/EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation. Modify the program.
035	CAN NOT COMMANDED G39	G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. Modify the program.
036	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN CRC	G40 is commanded on the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
041	INTERFERENCE IN CRC	Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program.
042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program.
043	ILLEGAL T-CODE COMMAND	In the DRILL-MATE, a T code was not specified together with the M06 code in a block. Alternatively, the Tcode was out of range.
044	G27-G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 is commanded in canned cycle mode. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.
050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.

Number	Meaning	Contents and remedy
051	MISSING MOVE AFTER CHF/CNR	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01. Modify the program.
053	TOO MANY ADDRESS COMMANDS	For systems without the arbitrary angle chamfering or corner R cutting, a comma was specified. For systems with this feature, a comma was followed by something other than R or C. Correct the program.
055	MISSING MOVE VALUE IN CHF/CNR	In the arbitrary angle chamfering or corner R block, the move distance is less than chamfer or corner R amount.
058	END POINT NOT FOUND	In a arbitrary angle chamfering or corner R cutting block, a specified axis is not in the selected plane. Correct the program.
059	PROGRAM NUMBER NOT FOUND	In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background editing.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option) or, 200 (option). Delete unnecessary programs and execute program registration again.
073	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.
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	A level larger than the maximum allowable nesting level of subprogram calls was specified. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. An attempt was made to call a program being edited in the background processing mode. Correct the program.
079	PROGRAM VERIFY ERROR	In memory or program collation, a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.

Number	Meaning	Contents and remedy
080	G37 ARRIVAL SIGNAL NOT ASSERTED	In the automatic tool length measurement function (G37), the measurement position reach signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter (value ϵ). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37	Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program.
082	H-CODE NOT ALLOWED IN G37	H code and automatic tool compensation (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37	In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
090	REFERENCE RETURN INCOMPLETE	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.
091	MANUAL RETURN IMPOSSIBLE DURING PAUSE	A manual return to the reference position was impossible because of the program being at pause. Press the reset button to cause a manual return.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G28 (automatic reference position return) or G27 (reference position return check) did not return to the reference position.
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.)
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.)
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.) Perform automatic operation.

Number	Meaning	Contents and remedy
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE (parameter writing enabled) is set to 1. Set it to 0, then reset the system.
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. When this alarm occurs, set the PWE parameter to 1, then switch on the power while holding down the <DELET> key. All programs will be deleted.
109	D/S ALARM	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation turns out to be invalid, an alarm No.111 is issued. -10^{47} to -10^{-29} , 0, 10^{-29} to 10^{47}
112	DIVIDED BY ZERO	Division by zero was specified. (including $\tan 90^\circ$)
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	Custom macro A specified an undefined H code in a G65 block. There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high-speed cycle machining. The header contents are improper. This alarm is given in the following cases: High speed cycle machining <ol style="list-style-type: none"> 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The last storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data overlaps with the variable number used in the header. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.

Number	Meaning	Contents and remedy
119	ILLEGAL ARGUMENT	The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL-CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	Custom macro A specified an undefined H code in a G65 block. <Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, $1 \leq n \leq 3$ is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
135	ILLEGAL ANGLE COMMAND	The index table indexing positioning angle was instructed in other than an integral multiple of the value of the minimum angle. Modify the program.
136	ILLEGAL AXIS COMMAND	In index table indexing. Another control axis was instructed together with the B axis. Modify the program.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
141	CAN NOT COMMAND G51 IN CRC	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting .
143	SCALED MOTION DATA OVERFLOW	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling magnification.
144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.

Number	Meaning	Contents and remedy
148	ILLEGAL SETTING DATA	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714)
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value in the tool life management. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group of the tool life management commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group in the tool life management exceeds the maximum value registerable. Modify the number of tools.
153	T-CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
154	NOT USING TOOL IN LIFE GROUP	When the group is not commanded in the tool life management, H99 or D99 was commanded. Correct the program.
155	ILLEGAL T-CODE IN M06	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group of the tool life management is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups in the tool life management to be set exceeds the maximum allowable value. Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of "G07.1 rotation-axis name radius of cylinder."
176	IMPROPER G-CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning: G28,, G73, G74, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G52,G92, 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error is occurred in the high-speed remote buffer.
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAMETER SETTING ERROR	The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value.
180	COMMUNICATION ERROR (REMOTE BUF)	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.

Number	Meaning	Contents and remedy
181	FORMAT ERROR IN G81 BLOCK (Hobbing machine)	G81 block format error 1) T (number of teeth) has not been instructed. 2) Data outside the command range was instructed by either T, L, Q or P. Modify the program.
182	G81 NOT COMMANDED (Hobbing machine)	G83 (C axis servo lag quantity offset) was instructed though synchronization by G81 has not been instructed. Correct the program.
183	DUPLICATE G83 (COMMANDS) (Hobbing machine)	G83 was instructed before canceled by G82 after compensating for the C axis servo lag quantity by G83.
184	ILLEGAL COMMAND IN G81 (Hobbing machine)	A command not to be instructed during synchronization by G81 was instructed. 1) A C axis command by G00, G27, G28, G29, G30, etc. was instructed. 2) Inch/Metric switching by G20, G21 was instructed.
185	RETURN TO REFERENCE POINT (Hobbing machine)	G81 was instructed without performing reference position return after power on or emergency stop. Perform reference position return.
186	PARAMETER SETTING ERROR (Hobbing machine)	Parameter error regarding G81 1) The C axis has not been set to be a rotary axis. 2) A hob axis and position coder gear ratio setting error
190	ILLEGAL AXIS SELECT	In the constant surface speed control, the specified axis command (P) contains an illegal value. Correct the program.
194	SPINDLE COMMAND IN SYNCHRO-MODE	Cs contour control or rigid tapping was specified during serial spindle synchronous control. Correct the program.
195	SPINDLE CONTROL MODE SWITCH	The serial spindle control mode was not switched. Check the PMC ladder program.
197	C-AXIS COMMANDED IN SPINDLE MODE	A command for Cs-axis movement was issued when the current control mode is not serial spindle Cs contour control. Check the PMC ladder program or machining program.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value is out of the range or is not specified. Modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tap, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (G74) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid mode DI signal is not ON when G84 (G74) is executed though the rigid M code (M29) is specified. Consult the PMC ladder diagram to find the reason the DI signal (DGNG061.1) is not turned on. Modify the program.

Number	Meaning	Contents and remedy
206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Correct the program.
210	CAN NOT COMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation.
211	CAN NOT COMMAND HIGH-SPEED SKIP	A high-speed skip (G31) was specified during the feed-per-rotation or rigid tapping mode. Correct the program.
212	ILLEGAL PLANE SELECT	The arbitrary angle chamfering or a corner R is commanded or the plane including an additional axis. Correct the program.
213	ILLEGAL COMMAND IN SYNCHRO-MODE	Any of the following alarms occurred in the operation with the simple synchronization control. 1) The program issued the move command to the slave axis. 2) The program issued the manual continuous feed/manual handle feed/incremental feed command to the slave axis. 3) The program issued the automatic reference position return command without executing the manual reference position return after the power was turned on. 4) The difference between the position error amount of the master and slave axes exceeded the value specified in parameter .
214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
222	DNC OP. NOT ALLOWED IN BG.-EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.
224	RETURN TO REFERENCE POINT	Reference position return has not been performed before the automatic operation starts. Perform reference position return.
230	R CODE NOT FOUND	The infeed quantity R has not been instructed for the G160 block of the canned grinding cycle. Or the R command value is negative. Correct the program.
250	SIMULTANEOUS M06 AND Z-AXIS MOVEMENT NOT ALLOWED	A tool change (M06) and a Z-axis movement were specified simultaneously in the DRILL MATE. Correct the program.

2) Background edit alarm

Number	Meaning	Contents and remedy
???	BACKGROUND EDIT ALARM	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	SELECTED PROGRAM ALARM	It was attempted to select or delete in the background a program being selected in the foreground. (Note) Use background editing correctly.

NOTE

A background editing alarm may occur even during MDI operation-B. This is because MDI operation-B uses the background editing function.

3) Absolute pulse coder (APC) alarm

Number	Meaning	Contents and remedy
3n0	nth-axis origin return	Manual reference position return is required for the nth-axis.
3n1	APC alarm: nth-axis communication	nth-axis APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
3n2	APC alarm: nth-axis over time	nth-axis APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n3	APC alarm: nth-axis framing	nth-axis APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n4	APC alarm: nth-axis parity	nth-axis APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n5	APC alarm: nth-axis pulse error	nth-axis APC pulse error alarm. APC alarm.APC or cable may be faulty.
3n6	APC alarm: nth-axis battery voltage 0	nth-axis APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
3n7	APC alarm: nth-axis battery low 1	nth-axis axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
3n8	APC alarm: nth-axis battery low 2	nth-axis APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm.

4) Serial pulse coder (SPC) alarms

When either of the following alarms is issued, a possible cause is a faulty serial pulse coder or cable.

Number	Meaning	Contents
3n9	SPC ALARM: n AXIS PULSE CODER	The n axis (axis 1-8) pulse coder has a fault.
3n5	ZRN Impossible: n AXIS PULSE CODER	A reference position has not been established for the nth-axis serial pulse coder. Move the tool along the nth axis by more than a single rotation of the motor.

● **The details of serial pulse coder alarm No.3n9**

The details of serial pulse coder alarm No. 3n9 are displayed in the diagnosis display (No. 760 to 767, 770 to 777) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
760 to 767		CSAL	BLAL	PHAL	RCAL	BZAL	CKAL	SPHL

CSAL : The serial pulse coder is defective. Replace it.

BLAL : The battery voltage is low. Replace the batteries. This alarm has nothing to do with alarm (serial pulse coder alarm).

PHAL : The serial pulse coder or feedback cable is defective.
Replace the serial pulse coder or cable.

RCAL : The serial pulse coder is defective. Replace it.

BZAL : The pulse coder was supplied with power for the first time.
Make sure that the batteries are connected.
Turn the power off, then turn it on again and perform a reference position return. This alarm has nothing to do with alarm (serial pulse coder alarm).

CKAL : The serial pulse coder is defective. Replace it.

PHAL : The serial pulse coder or feedback cable is defective. replace the serial pulse coder or cable.

	#7	#6	#5	#4	#3	#2	#1	#0
770 to 777	DTERR	CRCERR	STBERR					

DTERR : The serial pulse coder encountered a communication error.
The pulse coder, feedback cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC-axis board

CRCERR : The serial pulse coder encountered a communication error.
The pulse coder, feedback cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC-axis board.

STBERR : The serial pulse coder encountered a communication error.
The pulse coder, feedback cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC-axis board.

5) Servo alarms

Number	Meaning	Contents and actions
400	SERVO ALARM: 1, 2TH AXIS OVERLOAD	1-axis, 2-axis overload signal is on. Refer to diagnosis display No. 720 or 721 for details.
401	SERVO ALARM: 1, 2TH AXIS VRDY OFF	1-axis, 2-axis servo amplifier READY signal (DRDY) went off.
402	SERVO ALARM: 3, 4TH AXIS OVERLOAD	3-axis, 4-axis overload signal is on. Refer to diagnosis display No. 722 or 723 for details.
403	SERVO ALARM: 3, 4TH AXIS VRDY OFF	3-axis, 4-axis servo amplifier READY signal (DRDY) went off.
404	SERVO ALARM: n-TH AXIS VRDY ON	Even though the n-th axis (axis 1-8) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the axis card and servo amplifier are connected.

Number	Meaning	Contents and actions
405	SERVO ALARM: ZERO POINT RETURN FAULT	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.
406	SERVO ALARM: 7, 8TH AXIS OVER LOAD 7, 8TH AXIS VRDY OFF	7-axis, 8-axis overload signal is on. Refer to diagnosis display No. 726 or 727 for details. 3-axis, 4-axis servo amplifier READY signal (DRDY) went off.
4n0	SERVO ALARM: n-TH AXIS – EXCESS ERROR	The position deviation value when the n-th axis stops is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n1	SERVO ALARM: n-TH AXIS – EXCESS ERROR	The position deviation value when the n-th axis moves is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n3	SERVO ALARM: n-th AXIS – LSI OVERFLOW	The contents of the error register for the n-th axis exceeded $\pm 2^{31}$ power. This error usually occurs as the result of an improperly set parameters.
4n4	SERVO ALARM: n-TH AXIS – DETECTION RELATED ERROR	N-th axis digital servo system fault. Refer to diagnosis display No. 720 and No.727 for details.
4n5	SERVO ALARM: n-TH AXIS – EXCESS SHIFT	A speed higher than 4000000 units/s was attempted to be set in the n-th axis. This error occurs as the result of improperly set CMR.
4n6	SERVO ALARM: n-TH AXIS – DISCONNECTION	Position detection system fault in the n-th axis pulse coder (disconnection alarm).
4n7	SERVO ALARM: n-TH AXIS – PARAMETER INCORRECT	This alarm occurs when the n-th axis is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 8n20 (motor form) is out of the specified limit. 2) A proper value (111 or -111) is not set in parameter No. 8n22 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 8n23 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 8n24 (number of position feedback pulses per motor revolution). 5) Parameters No. 8n84 and No. 8n85 (flexible field gear rate) have not been set. 6) An axis selection parameter (from No. 269 to 274) is incorrect. 7) An overflow occurred during parameter computation.
490	SERVO ALARM: 5TH AXIS OVER LOAD	5-axis, 6-axis overload signal is on. Refer to diagnosis display No. 724 or 725 for details.
491	SERVO ALARM: 5, 6TH VRDY OFF	5-axis, 6-axis servo amplifier READY signal (DRDY) went off.
494	SERVO ALARM: 5, 6TH AXIS VRDY ON	The axis card ready signal (MCON) for axes 5 and 6 is off, but the servo amplifier ready signal (DRDY) is not. Alternatively, when the power is applied, the DRDY is on, but the MCON is not. Ensure that the axis card and servo amplifier are connected.
495	SERVO ALARM: 5, 6TH AXIS ZERO POINT RETURN	This is a position control circuit error. It is likely that a return to the reference position failed because of an error in the NC or the servo system. Retry a return to the reference position.

NOTE

If an excessive spindle error alarm occurs during rigid tapping, the relevant alarm number for the tapping feed axis is displayed.

● **Details of servo alarm No.4n4**

The detailed descriptions of servo alarm number 414 are displayed with diagnosis numbers 720 to 727 in the sequence of axis numbers.

	#7	#6	#5	#4	#3	#2	#1	#0
720 to 727	OVL	LV	OVC	HCAL	HVAL	DCAL	FBAL	OFAL

OVL : An overload alarm is being generated.

(This bit causes servo alarm No. 400, 402, 406, 490).

LV : A low voltage alarm is being generated in servo amp.
Check LED.

OVC : A overcurrent alarm is being generated inside of digital servo.

HCAL : An abnormal current alarm is being generated in servo amp.
Check LED.

HVAL : An overvoltage alarm is being generated in servo amp.
Check LED.

DCAL : A regenerative discharge circuit alarm is being generated in servo amp.
Check LED.

FBAL : A disconnection alarm is being generated.
(This bit causes servo alarm No.4n6.)

OFAL : An overflow alarm is being generated inside of digital servo.

6) Spindle alarms

Number	Meaning	Contents and remedy
408	SPINDLE SERIAL LINK START FAULT	<p>This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle.</p> <p>The four reasons can be considered as follows:</p> <ol style="list-style-type: none"> 1) An improperly connected optic cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU-01 or AL-24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) This alarm does not occur after the system including the spindle control unit is activated.
409	SPINDLE ALARM DETECTION	<p>A spindle amplifier alarm occurred in a system with a serial spindle. The alarm is indicated as "AL-XX" (where XX is a number) on the display of the spindle amplifier. For details, see Section 14. Setting bit 7 of parameter No. 0397 causes the spindle amplifier alarm number to appear on the screen.</p>

7) Over travel alarms

Number	Meaning	Contents and remedy
5n0	OVER TRAVEL : +n	Exceeded the n-th axis + side stored stroke limit 1, 2.
5n1	OVER TRAVEL : -n	Exceeded the n-th axis - side stored stroke limit 1, 2.
5n2	OVER TRAVEL : +n	Exceeded the n-th axis + side stored stroke limit 3.
5n3	OVER TRAVEL : -n	Exceeded the n-th axis - side stored stroke limit 3.
5n4	OVER TRAVEL : +n	Exceeded the n-th axis + side hardware OT.
5n5	OVER TRAVEL : -n	Exceeded the n-th axis - side hardware OT.

8) Macro alarms

Number	Meaning	Contents and remedy
500 to 599	MACRO ALARM	This alarm is related to the custom macro, macro executor, or order-made macro (including conversational program inputs). Refer to the relevant manual for details. (The macro alarm number may coincide with an overtravel alarm number. However, they can be distinguished from each other because the overtravel alarm number is accompanied with the description of the alarm.

9) PMC alarms

Number	Meaning	Contents and remedy
600	PMC ALARM : INVALID INSTRUCTION	An invalid-instruction interrupt occurred in the PMC.
601	PMC ALARM : RAM PARITY	A PMC RAM parity error occurred.
602	PMC ALARM : SERIAL TRANSFER	A PMC serial transfer error occurred.
603	PMC ALARM : WATCHDOG	A PMC watchdog timer alarm occurred.
604	PMC ALARM : ROM PARITY	A PMC ROM parity error occurred.
605	PMC ALARM : OVER STEP	The maximum allowable number of PMC ladder program steps was exceeded.
606	PMC ALARM : I/O MODULE ASSIGNMENT	The assignment of I/O module signals is incorrect.
607	PMC ALARM : I/O LINK	An I/O link error occurred. The details are listed below.

Number	Details of PMC alarm (No. 607)
010	* Communication error (SLC (master) internal register error)
020	* An SLC RAM bit error occurred (verification error).
030	* An SLC RAM bit error occurred (verification error).
040	No I/O unit has been connected.
050	32 or more I/O units are connected.
060	* Data transmission error (no response from the slave)

Number	Details of PMC alarm (No. 607)
070	* Communication error (no response from the slave)
080	* Communication error (no response from the slave)
090	An NMI (for other than alarm codes 110 to 160) occurred.
130	* An SLC (master) RAM parity error occurred (detected by hardware).
140	* An SLC (slave) RAM parity error occurred (detected by hardware).
160	* SLC (slave) communication error <ul style="list-style-type: none"> * AL0 : Watchdog timer DO clear signal received * IR1 : CRC or framing error Watchdog timer alarm Parity error

Hardware errors are indicated with an asterisk (*).

10) Overheat alarms

Number	Meaning	Contents and remedy
700	OVERHEAT: CONTROL UNIT	Control unit overheat Check that the fan motor operates normally, and clean the air filter.

11) M-NET alarm

Number	Meaning	Contents and remedy
899	M-NET INTERFACE ALARM	This alarm is related to a serial interface for an external PLC. The details are listed below.

Number	Details of M-NET alarm (No. 899)
0001	Abnormal character (character other than transmission codes) received
0002	"EXT" code error
0003	Connection time monitor error (parameter No. 0464)
0004	Polling time monitor error (parameter No. 0465)
0005	Vertical parity or framing error detected
0257	Transmission time-out error (parameter No. 0466)
0258	ROM parity error
0259	Overflow error detected
Others	CPU interrupt detected

12) System alarms

(These alarms cannot be reset with reset key.)

Number	Meaning	Contents and remedy
910	MAIN RAM PARITY	This RAM parity error is related to low-order bytes. Replace the memory PC board.
911	MAIN RAM PARITY	This RAM parity error is related to high-order bytes. Replace the memory PC board.
912	SHARED RAM PARITY	This parity error is related to low-order bytes of RAM shared with the digital servo circuit. Replace the axis control PC board.
913	SHARED RAM PARITY	This parity error is related to high-order bytes of RAM shared with the digital servo circuit. Replace the axis control PC board.
914	SERVO RAM PARITY	This is a local RAM parity error in the digital servo circuit. Replace the axis control PC board.
915	LADDER EDITING CASSETTE RAM PARITY	This RAM parity error is related to low-order bytes of the ladder editing cassette. Replace the ladder editing cassette.
916	LADDER EDITING CASSETTE RAM PARITY	This RAM parity error is related to high-order bytes of the ladder editing cassette. Replace the ladder editing cassette.
920	WATCHDOG ALARM	This is a watchdog timer alarm or a servo system alarm for axis 1 to 4. Replace the master or axis control PC board.
921	SUB CPU WATCHDOG ALARM	This is a watchdog timer alarm related to the sub-CPU board or a servo system alarm for axis 5 or 6. Replace the sub-CPU board or the axis-5/6 control PC board.
922	7/8 AXIS SERVO SYSTEM ALARM	This is a servo system alarm related to axis 7 or 8. Replace the axis-7/8 control PC board.
930	CPU ERROR	This is a CPU error. Replace the master PC board.
940	PC BOARD INSTALLATION ERROR	PC board installation is incorrect. Check the specification of the PC board.
941	MEMORY PC BOARD CONNECTION ERROR	The memory PC board is not connected securely. Ensure that the PC board is connected securely.
945	SERIAL SPINDLE COMMUNICATION ERROR	The hardware configuration is incorrect for the serial spindle, or a communication alarm occurred. Check the hardware configuration of the spindle. Also ensure that the hardware for the serial spindle is connected securely.
946	SECOND SERIAL SPINDLE COMMUNICATION ERROR	Communication is impossible with the second serial spindle. Ensure that the second serial spindle is connected securely.
950	FUSE BLOWN ALARM	A fuse has blown. Replace the fuse (+24E; F14).
960	SUB CPU ERROR	This is a sub-CPU error. Replace the sub-CPU PC board.
998	ROM PARITY	This is a ROM parity error. Replace the ROM board in which the error occurred.

13) External alarm

Number	Meaning	Contents and remedy
1000	EXTERNAL ALARM	This alarm was detected by the PMC ladder program. Refer to the relevant manual from the machine builder for details.

14) Alarms Displayed on spindle Servo Unit

Alarm No.	Meaning	Description	Remedy
"A" display	Program ROM abnormality (not installed)	Detects that control program is not started (due to program ROM not installed, etc.)	Install normal program ROM
AL01	Motor overheat	Detects motor speed exceeding specified speed excessively.	Check load status. Cool motor then reset alarm.
AL02	Excessive speed deviation	Detects motor speed exceeding specified speed excessively.	Check load status. Reset alarm.
AL03	DC link section fuse blown	Detects that fuse F4 in DC link section is blown (models 30S and 40S).	Check power transistors, and so forth. Replace fuse.
AL04	Input fuse blown. Input power open phase.	Detects blown fuse (F1 to F3), open phase or momentary failure of power (models 30S and 40S).	Replace fuse. Check open phase and power supply regenerative circuit operation.
AL05	Control power supply fuse blown	Detects that control power supply fuse AF2 or AF3 is blown (models 30S and 40S).	Check for control power supply short circuit. Replace fuse.
AL-07	Excessive speed	Detects that motor rotation has exceeded 115% of its rated speed.	Reset alarm.
AL-08	High input voltage	Detects that switch is flipped to 200 VAC when input voltage is 230 VAC or higher (models 30S and 40S).	Flip switch to 230 VAC.
AL-09	Excessive load on main circuit section	Detects abnormal temperature rise of power transistor radiator.	Cool radiator then reset alarm.
AL-10	Low input voltage	Detects drop in input power supply voltage.	Remove cause, then reset alarm.
AL-11	Overvoltage in DC link section	Detects abnormally high direct current power supply voltage in power circuit section.	Remove cause, then reset alarm.
AL-12	Overcurrent in DC link section	Detects flow of abnormally large current in direct current section of power circuit	Remove cause, then reset alarm.
AL-13	CPU internal data memory abnormality	Detects abnormality in CPU internal data memory. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-15	Spindle switch/output switch alarm	Detects incorrect switch sequence in spindle switch/output switch operation.	Check sequence.
AL-16	RAM abnormality	Detects abnormality in RAM for external data. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-18	Program ROM sum check error	Detects program ROM data error. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-19	Excessive U phase current detection circuit offset	Detects excessive U phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-20	Excessive V phase current detection circuit offset	Detects excessive V phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.

Alarm No.	Meaning	Description	Remedy
AL-24	Serial transfer data error	Detects serial transfer data error (such as NC power supply turned off, etc.)	Remove cause, then reset alarm.
AL-25	Serial data transfer stopped	Detects that serial data transfer has stopped.	Remove cause, then reset alarm.
AL-26	Disconnection of speed detection signal for Cs contouring control	Detects abnormality in position coder signal (such as unconnected cable and parameter setting error).	Remove cause, then reset alarm.
AL-27	Position coder signal disconnection	Detects abnormality in position coder signal (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL-28	Disconnection of position detection signal for Cs contouring control	Detects abnormality in position detection signal for Cs contouring control (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL-29	Short-time overload	Detects that overload has been continuously applied for some period of time (such as restraining motor shaft in positioning).	Remove cause, then reset alarm.
AL-30	Input circuit overcurrent	Detects overcurrent flowing in input circuit.	Remove cause, then reset alarm.
AL-31	Speed detection signal disconnection motor restraint alarm or motor is clamped.	Detects that motor cannot rotate at specified speed or it is detected that the motor is clamped. (but rotates at very slow speed or has stopped). (This includes checking of speed detection signal cable.)	Remove cause, then reset alarm.
AL-32	Abnormality in RAM internal to LSI for serial data transfer. This check is made only when power is turned on.	Detects abnormality in RAM internal to LSI for serial data transfer. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-33	Insufficient DC link section charging	Detects insufficient charging of direct current power supply voltage in power circuit section when magnetic contactor in amplifier is turned on (such as open phase and defective charging resistor).	Remove cause, then reset alarm.
AL-34	Parameter data setting beyond allowable range of values	Detects parameter data set beyond allowable range of values.	Set correct data.
AL-35	Excessive gear ratio data setting	Detects gear ratio data set beyond allowable range of values.	Set correct data.
AL-36	Error counter over flow	Detects error counter overflow.	Correct cause, then reset alarm.
AL-37	Speed detector parameter setting error	Detects incorrect setting of parameter for number of speed detection pulses.	Set correct data.
AL-39	Alarm for indicating failure in detecting 1-rotation signal for Cs contouring control	Detects 1-rotation signal detection failure in Cs contouring control.	Make 1-rotation signal adjustment. Check cable shield status.
AL-40	Alarm for indicating 1-rotation signal for Cs contouring control not detected	Detects that 1-rotation signal has not occurred in Cs contouring control.	Make 1-rotation signal adjustment.

Alarm No.	Meaning	Description	Remedy
AL-41	Alarm for indicating failure in detecting position coder 1-rotation signal.	Detects failure in detecting position coder 1-rotation signal.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL-42	Alarm for indicating position coder 1-rotation signal not detected	Detects that position coder 1-rotation signal has not issued.	Make 1-rotation signal adjustment for signal conversion circuit.
AL-43	Alarm for indicating disconnection of position coder signal for differential speed mode	Detects that main spindle position coder signal used for differential speed mode is not connected yet (or is disconnected).	Check that main spindle position coder signal is connected to connector CN12.
AL-46	Alarm for indicating failure in detecting position coder 1-rotation signal in thread cutting operation.	Detects failure in detecting position coder 1-rotation signal in thread cutting operation.	Make 1-rotation signal adjustment for signal conversion circuit. Check cable shield status.
AL-47	Position coder signal abnormality	Detects incorrect position coder signal count operation.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL-48	Position coder 1-rotation signal abnormality	Detects that occurrence of position coder 1-rotation signal has stopped.	Make 1-rotation signal adjustment for signal conversion circuit.
AL-49	The converted differential speed is too high.	Detects that speed of other spindle converted to speed of local spindle has exceeded allowable limit in differential mode.	Check the position coder state of the other side.
AL-50	Excessive speed command calculation value in spindle synchronization control	Detects that speed command calculation value exceeded allowable range in spindle synchronization control.	Check parameters such as a position gain.
AL-51	Undervoltage at DC link section	Detects that DC power supply voltage of power circuit has dropped (due to momentary power failure or loose contact of magnetic contactor).	Remove cause, then reset alarm.
AL-52	ITP signal abnormality I	Detects abnormality in synchronization signal (ITP signal) with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL-53	ITP signal abnormality II	Detects abnormality in synchronization signal (ITP signal) with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL-54	Overload current alarm	Detects that excessive current flowed in motor for long time.	Check if overload operation or frequent acceleration/deceleration is performed.
AL-55	Power line abnormality in spindle switching/output switching	Detects that switch request signal does not match power line status check signal.	Check operation of magnetic contractor for power line switching. Check if power line status check signal is processed normally.

H OPERATION OF PORTABLE TAPE READER

Portable tape reader is the device which inputs the NC program and the data on the paper tape to CNC.

- Names and descriptions of each section

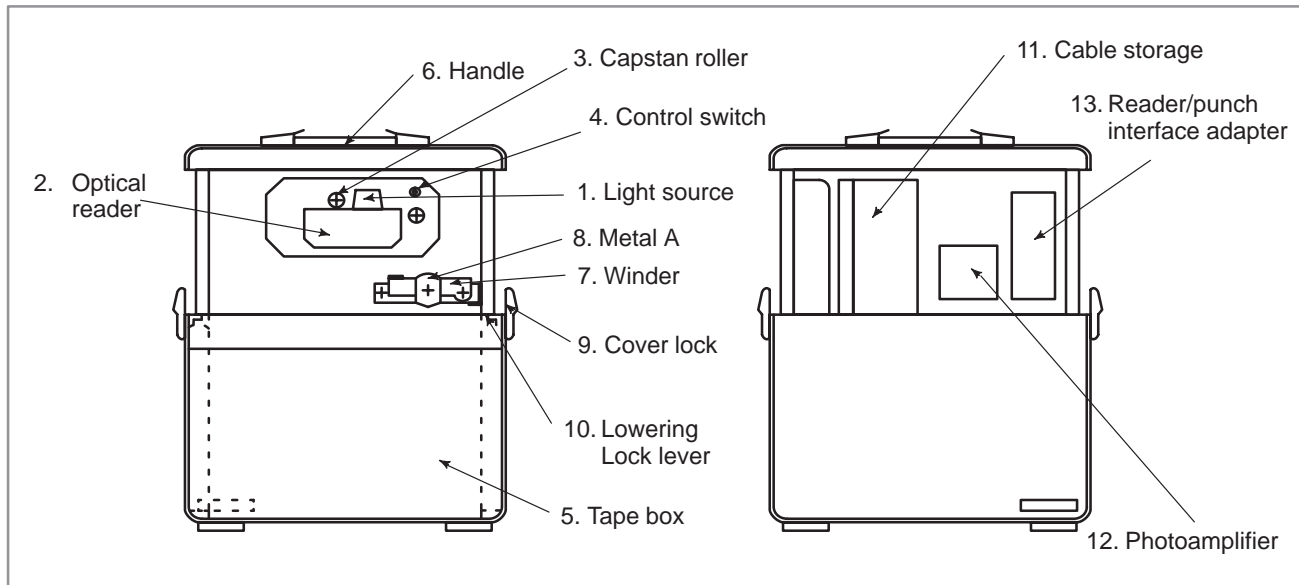


Table H Description of each section

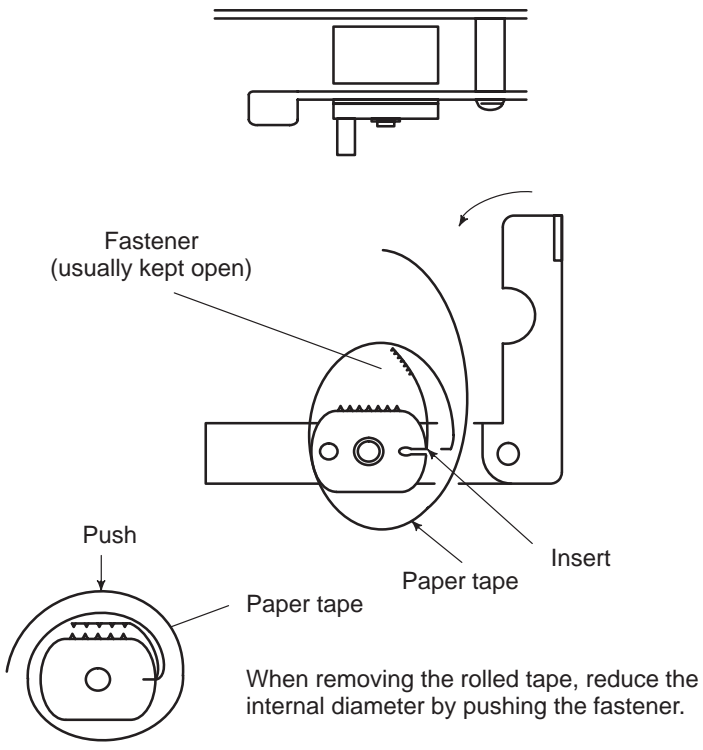
No.	Name	Descriptions
1	Light Sources	An LED (Light emitting diode) is mounted for each channel and for the feed hole (9 diodes in total). A built-in Stop Shoe functions to decelerate the tape. The light source is attracted to the optical reader by a magnet so that the tape will be held in the correct position. This unit can be opened upward, by turning the tape reader control switch to the RELEASE position (this turns off the magnet).
2	Optical Reader	Reads data punched on the tape, through a glass window. Dust or scratches on the glass window can result in reading errors. Keep this window clean.
3	Capstan Roller	Controls the feeding of tape as specified by the control unit.
4	Tape Reader Control Switch	A 3-position switch used to control the Tape Reader. RELEASE -- The tape is allowed to be free, or used to open the lightsource. When loading or unloading the tape, select this position. AUTO ----- The tape is set to fixed position by the Stop Shoe. The feed and stop of the tape is controlled by the CNC. To input data from tape, the Light Source must be closed and this position must be selected. MANUAL --- The tape can be fed in the forward reading direction. if another position is selected, the tape feed is stopped.
5	Tape Box	A Tape Box is located below the Tape Reader. A belt used to draw out a paper tape is located inside the box. The paper tape can easily be pulled out using this belt. The tape box accomodates 15 meters of tape.
6	Handle	Used to carry the tape reader.
7	Winder	Used to advance or rewind the tape.
8	Metal A	 <p>Fastener (usually kept open)</p> <p>Push</p> <p>Paper tape</p> <p>Insert</p> <p>Paper tape</p> <p>When removing the rolled tape, reduce the internal diameter by pushing the fastener.</p>
9	Cover lock	Be sure to use the lock for fastening the cover before carrying the tape reader.

Table H Description of each section

No.	Name	Descriptions
10	Lowering lock lever	When the tape reader is raised, the latch mechanism is activated to fix the tape reader. Thus, the tape reader is not lowered. The latch is locked with the lowering lock lever. The latch is therefore not unlocked even when the tape reader is raised with the handle. When the latch is locked, the lever is horizontal. To store the tape reader in the box, push the lever to release the lock, then raise the tape reader with the handle to unlock the latch. When the latch is unlocked, the tape reader can be stored in the box. When storing the tape reader, secure it with the cover lock.
11	Cable storage	Used to store rolled power and signal cables. The cable length is 1.5 m.
12	Photoamplifier	For the tape reader
13	Reader/punch interface adapter	200 VAC input and 5 VDC output power and reader/punch interface adapter PCB

Procedure for Operating the Portable Tape Reader

Preparations

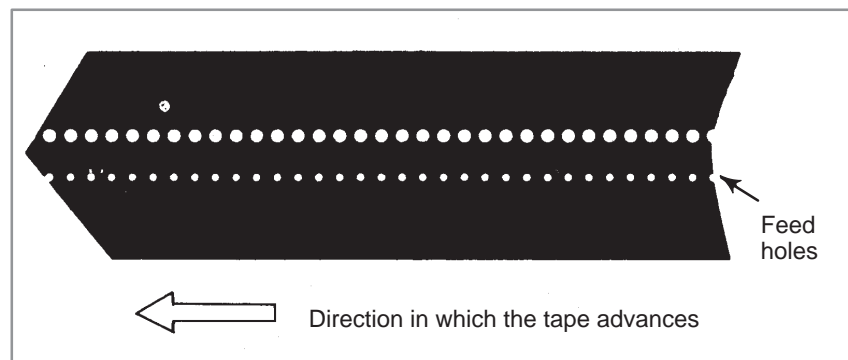
1 Unlock the cover locks **9**. Raise the tape reader with the handle **6** until it clicks, then lower the tape reader. The tape reader then appears and is secured. Check that the lowering lock levers **10** are horizontal.

2 Take out the signal and power cables from the cable storage **11** and connect the signal cable with the CNC reader/punch interface port and the power cable with the power supply.

Setting the tape

3 Turn the control switch to the **RELEASE** position.

4 Lift the Light Source Unit, and insert an NC tape between the gap. The tape must be positioned as shown in the figure, when viewed looking downward.



5 Pull the tape until the top of the tape goes past the Capstan roller.

6 Check that the NC tape is correctly positioned by the Tape Guide.

7 Lower the Light Source.

8 Turn the switch to the **AUTO** position.

9 Suspend the top and rear-end of the tape in the Tape Box.

Removing the tape

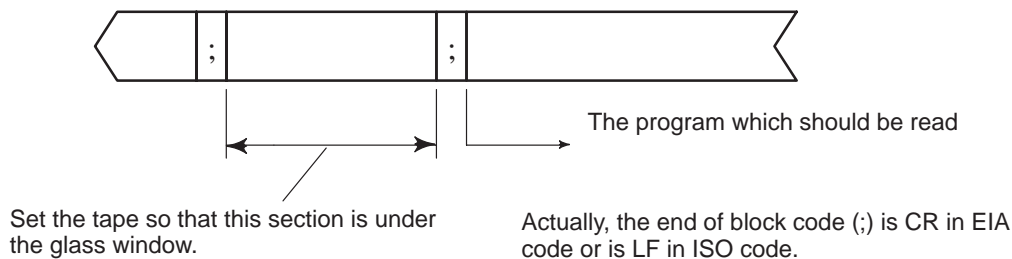
- 10** Turn the switch to the RELEASE position.
- 11** Lift the Light Source and remove the tape.
- 12** Lower the Light Source

Storage

- 13** Store the cables in the cable storage **11**.
- 14** Push the lowering lock lever **10** at both sides down.
- 15** Raise the tape reader with the handle **6** to unlock the latch, then gently lower it.
- 16** Lock the cover lock **9** and carry the tape reader with the handle **6**.

CAUTION**1 SETTING OF A TAPE**

When the NC tape is loaded, the Label Skip function activates to read but skip data until first End of Block code (CR in EIA code or LF in ISO code) is read. When loading an NC tape, the location within the tape, from which data reading should be started must properly be selected and the NC tape should be set as shown in the figure below.

**2 DISCONNECTION AND CONNECTION OF A PORTABLE TAPE READER CONNECTION CABLE**

Don't disconnect or connect CNC tape reader connection cable (signal cable) without turning off the CNC power supply, otherwise the PCB of the tape reader and master PCB of CNC controller may be broken. Turn off the CNC power supply before disconnecting or connecting the connection cable, accordingly.

Series 0-D SPECIFICATIONS

○ : Basic ● : Basic option ☆ : option
 ✱ : Function included in other option

Controlled axis

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Number of controlled axes	3 axes	○	—	—	—	○	—
	4 axes	—	○	○	○	—	○
Simultaneous controllable axes	3 axes	○	○	—	—	○	—
	4 axes	—	—	○	○	—	○
Additional axis control		—	—	—	○	—	○
Axis control by PMC	MAX. 2 axes	—	○	○	○	—	○
Cs-axis contouring control		—	—	—	○	—	○
Simple synchronous control		—	—	—	○	—	○
Angular axis control		—	—	—	—	—	○
Least input increment	0.001mm,0.001deg, 0.0001inch	○	○	○	○	○	○
Incremental system 1/10	0.0001mm,0.0001deg, 0.00001inch	—	—	—	○	○	○
Inch/metric conversion		—	○	○	○	○	○
Interlock		○	○	○	○	○	○
Machine lock		○	○	○	○	○	○
Emergency stop		○	○	○	○	○	○
Overtravel signal	X/Y/Z-axis	○	○	○	○	○	○
Stored stroke check 1		○	○	○	○	○	○
Stroke limit external setting		—	—	—	○	—	○
Stored stroke check 2		○	○	○	○	○	○
Stored stroke check 3	G22,G23	—	—	—	○	—	○
Mirror image	each axis	○	○	○	○	○	○
Follow up		○	○	○	○	○	○
Servo off		○	○	○	○	○	○
Mechanical handle feed		○	○	○	○	○	○
Backlash compensation		○	○	○	○	○	○
Stored pitch error compensation		○	○	○	○	○	○
Position switch		—	—	—	○	—	○

Operation

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Automatic operation (Memory)		○	○	○	○	○	○
DNC operation	included in Reader/puncher interface	○	○	○	○	○	○
MDI operation		○	○	○	○	○	○
MDI operation B		—	—	—	○	—	○
Scheduling function	Directory display of floppy cassette is need.	—	—	—	○	—	○
Program number search		○	○	○	○	○	○
Sequence number search		○	○	○	○	○	○
Sequence number comparison and stop		—	—	—	○	—	○
Program restart		—	—	—	○	—	○
Buffer register		○	○	○	○	○	○
Dry run		○	○	○	○	○	○
Single block		○	○	○	○	○	○
Manual continuous feed (JOG)		○	○	○	○	○	○
Manual reference position return		○	○	○	○	○	○
Setting the reference position without DOGs		○	○	○	○	○	○
Reference position setting with mechanical stopper		—	—	—	○	—	○
Manual handle feed	1 unit	○	○	○	○	—	○
	2/3 units	—	—	—	☆	—	☆
Manual handle feed rate	× 1, × 10, × m, × n m: to 127, n: to 1000	○	○	○	○	○	○
Handle interruption		—	—	—	○	—	○
Incremental feed	× 1, × 10, × 100, × 1000	○	○	○	○	○	○
Jog and handle simultaneous mode		○	○	○	○	○	○

Interpolation function

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Positioning	G00	○	○	○	○	○	○
Single direction positioning	G60	○	○	○	○	○	○
Exact stop mode	G61	○	○	○	○	○	○
Exact stop	G09	○	○	○	○	○	○
Linear interpolation		○	○	○	○	○	○
Circular interpolation	Multi quadrant is possible.	○	○	○	○	○	○
Dwell (per sec.)		○	○	○	○	○	○
Cylindrical interpolation		—	—	—	○	—	○
Helical interpolation		—	○	○	○	—	○
Helical interpolation axis extend		—	○	○	○	—	○
Thread cutting. synchronous feed	Position coder is need.	—	—	—	○	—	○
Skip function	G31	—	—	—	○	—	○
High-speed skip function		—	—	—	○	—	○
Reference position return	G28	○	○	○	○	○	○
Reference position return check	G27	○	○	○	○	○	○
2nd reference position return		○	○	○	○	○	○
3rd/4th reference position return		—	—	—	○	—	○
Normal direction control		—	—	—	○	○	○
Continuous dressing control		—	—	—	—	—	○
Infeed control		—	—	—	—	○	○
Index table indexing function		—	—	—	○	—	○
Rapid traverse rate	100 m/min	○	○	○	○	○	○
Rapid traverse override	F0,25,50,100%	○	○	○	○	○	○
Feed per minute	mm/min.	○	○	○	○	○	○
Feed par revolution	mm/rev	—	—	—	*	—	*
Tangential speed constant control		○	○	○	○	○	○
Cutting feedrate clamp		○	○	○	○	○	○
Automatic acceleration / deceleration	Rapid traverse : linear Cutting feed : exponential	○	○	○	○	○	○
Bell-shaped acceleration / deceleration for rapid traverse		—	—	—	○	—	○

Item	Specification	0–MD Package 1	0–MD Package 2	0–MD Package 3	0–MD II	0–GSD Package 1	0–GSD II
Linear acceleration / deceleration after cutting feed interpolation		—	—	—	○	—	○
Feedrate override	0 to 150%	○	○	○	○	○	○
F1–digit feed		—	—	—	○	—	○
Jog override	0 to 150%	—	—	—	○	—	○
Override cancel		○	○	○	○	○	○
External deceleration		—	—	—	○	—	○
Advanced preview control		—	—	—	○	—	○

Programming

Item	Specification	0–MD Package 1	0–MD Package 2	0–MD Package 3	0–MD II	0–GSD Package 1	0–GSD II
Tape code	EIA/ISO automatic recognition	○	○	○	○	○	○
Label skip		○	○	○	○	○	○
Parity check	Parity H, Parity V	○	○	○	○	○	○
Control in / out		○	○	○	○	○	○
Optional block skip	1 piece	○	○	○	○	○	○
	9 pieces	—	—	—	○	—	○
Max. programmable dimension	± 8 digits	○	○	○	○	○	○
Program number	O4 digits	○	○	○	○	○	○
Sequence number	N4 digits	○	○	○	○	○	○
Absolute / Incremental programming	It is possible to use in the same block.	○	○	○	○	○	○
Decimal point input / Pocket calculator type decimal point input programming		○	○	○	○	○	○
Plane selection	G17,G18,G19	○	○	○	○	○	○
Rotary axis designation	only added axes	○	○	○	○	○	○
Rotary axis roll–over function	only added axes	○	○	○	○	○	○
Polar coordinate command		—	—	—	○	—	○
Coordinate system setting		○	○	○	○	○	○
Automatic coordinate system setting		○	○	○	○	○	○
Workpiece coordinate system	G52,G53,G54 to G59	○	○	○	○	○	○
Workpiece coordinate pair addition	48 pairs Note)	—	—	—	○	—	○
Manual absolute on and off		○	○	○	○	○	○

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Optional angle chamfering / corner R		—	—	—	○	—	○
Programmable data input	G10 (Programmable input of offset)	—	○	○	○	○	○
Sub program call	two-fold nested	○	○	○	○	○	○
Custom macro A		○	○	○	○	○	○
Custom macro B	Editing is not available.	—	—	○	○	—	○
Addition of custom macro common variables	only custom macro B	—	—	—	○	—	○
Pattern data input		—	—	—	○	—	○
Interruption type custom macro		—	—	—	○	—	○
Canned cycles for drilling		○	○	○	○	—	○
Canned cycles for grinding		—	—	—	—	○	○
Small-hole peck drilling cycle		—	—	—	○	—	—
Circular interpolation by R programming		○	○	○	○	○	○
Automatic corner override		—	—	—	○	—	○
Block overlap		—	—	—	○	—	○
Automatic corner deceleration		—	—	—	○	—	○
Feedrate clamp by circular radius		—	—	—	○	—	○
Scaling		—	—	—	○	—	○
Coordinate system rotation		—	—	—	○	—	○
Programmable mirror image		—	—	—	*	—	*
Menu programming		—	—	—	○	—	○
Tape format for FS10/11		—	—	—	○	—	○

Miscellaneous function/spindle function

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Auxiliary function	M3 digit	○	○	○	○	○	○
2nd auxiliary function	B6 digit	○	○	○	○	—	○
Auxiliary function lock		○	○	○	○	○	○
High speed M/S/T interface		○	○	○	○	○	○
Multiple command of auxiliary function	3 pieces (M code only)	○	○	○	○	○	○
Spindle function	S analog/ serial output	○	○	○	○	○	○
Constant surface speed control		—	—	—	○	—	○
Spindle speed override	0 to 120%	○	○	○	○	○	○
Analog voltage control by PMC		○	○	○	○	○	○
1st spindle orientation		—	—	—	○	—	○
1st spindle speed range switch		—	—	—	○	—	○
2nd spindle orientation		—	—	—	○	—	○
2nd spindle speed range switch		—	—	—	○	—	○
Spindle synchronous control		—	—	—	○	—	○
Simple spindle synchronous control		—	—	—	○	—	○
Rigid tapping	Position coder is need.	—	○	○	○	—	○

Tool function/tool offset function

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Tool function		○	○	○	○	○	○
Tool offset memory	± 6 digits 32 pieces	—	—	—	—	○	—
	± 6 digits 64 pieces	○	○	○	—	—	—
	± 6 digits 400 pieces	—	—	—	○	—	○
Tool offset memory B	Tool geometry / wear offset	—	—	—	○	—	○
Tool length compensation		○	○	○	○	○	○
Tool position offset		—	—	—	○	—	○
Cutter compensation C		○	○	○	○	—	○
Tool life management		—	—	—	○	—	○
Automatic tool length measurement		—	—	—	○	—	○
Tool length measurement		—	—	—	○	—	○

Editing operation

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Part program storage length	10m	—	—	—	—	○	—
	120m	○	○	—	—	—	—
	320m	—	—	○	○	—	○
Registered programs	63 pieces	○	○	○	—	○	—
	200 pieces	—	—	—	○	—	○
Tape editing		○	○	○	○	○	○
Program protect		○	○	○	○	○	○
Background editing		—	○	○	○	—	○
Extended part program editing		—	—	○	○	—	○
Play back		—	—	—	○	—	○

Setting / display

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Status display		○	○	○	○	○	○
Clock function		—	—	—	○	—	○
Current position display		○	○	○	○	○	○
Program display	Program name : 31 characters	○	○	○	○	○	○
Parameter setting and display		○	○	○	○	○	○
Self-diagnosis function		○	○	○	○	○	○
Alarm display		○	○	○	○	○	○
Run hour and parts count display		—	○	○	○	—	○
Actual cutting feed display		○	○	○	○	○	○
Display of spindle speed and T code at all screens	Position coder is need.	—	—	—	○	—	○
Directory display of floppy cassette		—	—	—	○	—	○
Graphic display		—	—	—	☆	—	☆
Servo setting screen		○	○	○	○	○	○
Spindle setting screen		○	○	○	○	○	○
Servo waveform display		—	—	—	*	—	*
Software operator's panel		—	○	○	○	—	○
Software operator's panel general purpose switch		—	○	○	○	—	○
English display		○	○	○	○	○	○

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Japanese (Chinese characters) display		—	—	○	○	—	○
German / French display		—	—	○	○	—	○
Italian display		—	—	○	○	—	○
Chinese display		○	○	○	○	○	○
Spanish display		—	—	○	○	—	○
Korean display		—	—	—	☆	—	☆
Data protection key		○	○	○	○	○	○

Data input / output

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Reader / puncher interface	Reader/puncher (Ch.1) interface	○	○	○	○	○	○
	Reader/puncher (Ch.2) interface	—	—	—	○	—	○
External I/O device control		—	—	○	○	—	○
External data input		—	○	○	○	○	○
External key input		—	—	—	○	—	○
External workpiece number search	15 pieces	○	○	○	○	○	○
External program number search	1 to 9999	○	○	○	○	○	○

Others

Item	Specification	0-MD Package 1	0-MD Package 2	0-MD Package 3	0-MD II	0-GSD Package 1	0-GSD II
Status output signals		○	○	○	○	○	○
9" monochrome CRT		○	○	○	○	○	○
PMC-L	Basic command: 6.0μs Max. steps: 5000	○	○	—	●	○	●
PMC-M	Basic command: 2.0μs Max. steps: 8000	—	—	○	●	—	●
Internal I/O card	DI/DO: 104/72 points, source/sink	○	○	○	○	○	○
I/O Unit-MODEL A	DI/DO Max. : 1024/1024 points	—	—	—	☆	—	☆







































NOTE

Part program storage length is decreased by 9.75m.

J

CORRESPONDENCE BETWEEN ENGLISH KEY AND
SYMBOLIC KEY

Table J Correspondence between english key and symbolic key (Series 0)

Name	English key	Symbolic key	Name	English key	Symbolic key
RESET key			OPERATION/ ALARM key		
PAGE UP key			AUXILIARY/ GRAPHIC key		
PAGE DOWN key			MACRO key		
SHIFT key			ALTER key		
POSITION key			INSERT key		
PROGRAM key			INPUT key		
MENU key			OUTPUT/START key		
OFFSET key			DELETE key		
MENU/OFFSET key			CANCEL key		
DIAGNOS/ PARAM key					

≪ Number ≫

14" CRT soft key configuration, 402

≪ A ≫

Absolute and incremental programming (G90, G91), 94
 Actual feedrate display, 545
 Adding workpiece coordinate systems, 88
 Advanced preview control, 371
 Alarm and self-diagnosis functions, 464
 Alarm display, 389, 465
 Alarm list, 654
 Altering a word, 500
 Angular axis control (0-GSC, 0-GSD/II), 369
 Argument specification, 283
 Arithmetic and logic operation, 308
 Automatic corner deceleration, 68
 Automatic grinding wheel diameter compensation after dressing, 182
 Automatic insertion of sequence numbers, 521
 Automatic operation, 380, 423
 Automatic override for inner corners (G62), 60
 Automatic tool length measurement (G37), 196
 Auxiliary function, 115
 Auxiliary function (M function), 116

≪ B ≫

Background editing, 516
 Boring cycle (G85), 151
 Boring cycle (G86), 153
 Boring cycle (G88), 157
 Boring cycle (G89), 159
 Boring cycle back boring cycle (G87), 155
 Branch and repetition, 313

≪ C ≫

Canned cycle, 133
 Canned cycle cancel (G80), 161, 171
 Canned grinding cycle (0-GSC, 0-GSD/II), 172
 Change of the cutter compensation value, 215
 Changing the file name, 492
 Changing workpiece coordinate system, 85
 Character-to codes correspondence table, 653
 Characters and codes to be used for the pattern data input function, 355
 Check by running the machine, 382
 Checking by self-diagnostic screen, 467

Checking the minimum grinding wheel diameter (0-GSC, 0-GSD/II), 182
 Circular interpolation (G02, G03), 37
 Command for machine operations – miscellaneous function, 20
 Compensation function, 191
 Conditional branch (IF statement), 313
 Configuration of high speed cycle cutting data, 362
 Constant lead threading (G33), 45
 Constant surface speed control (G96, G97), 102
 Continuous-feed surface grinding cycle (G78), 177
 Controlled axes, 26, 27
 Conversational programming with graphic function, 528
 Coordinate system, 81
 Coordinate system on part drawing and coordinate system specified by CNC – coordinate system, 15
 Coordinate system rotation (G68, G69), 266
 Coordinate value and dimension, 93
 Copying an entire program, 508
 Copying part of a program, 509
 Corner offset circular interpolation (G39), 212
 Correspondence between english key and symbolic key, 685
 Creating programs, 519
 Creating programs in TEACH IN mode, 523
 Creating programs using the MDI panel, 520
 CRT/MDI panels and LCD/MDI panels, 393
 Current block display screen, 550
 Current position display, 389
 Custom macro A, 279
 Custom macro B, 297
 Custom macro body, 285
 Custom macro command, 280
 Cutter compensation B (G39 to G42), 205
 Cutter compensation cancel (G40), 213
 Cutter compensation left (G41), 208
 Cutter compensation right (G42), 210
 Cutting feed, 54
 Cutting feed block overlap, 67
 Cutting feed linear acceleration/deceleration before interpolation, 64
 Cutting feedrate control, 58
 Cutting speed – spindle speed function, 19
 Cylindrical interpolation (G107), 42

≪ D ≫

Data input/output, 469
 Data output, 391
 Decimal point programming, 99
 Deleting a block, 502
 Deleting a word, 501
 Deleting all programs, 505
 Deleting blocks, 502

Deleting files, 490
 Deleting more than one program by specifying a range, 506
 Deleting multiple blocks, 503
 Deleting one program, 505
 Deleting programs, 505
 Details of cutter compensation C, 224
 Details of functions, 339
 Direct constant–dimension plunge grinding cycle (G77), 175
 Display, 388, 621
 Display and operation of 00–MC, 620
 Display of run time and parts count, 546
 Display of software configuration, 409
 Displaying and entering setting data, 579
 Displaying and setting custom macro common variables, 566
 Displaying and setting data, 385
 Displaying and setting parameters, 572
 Displaying and setting pitch error compensation data, 574
 Displaying and setting run time, parts count, and time, 583
 Displaying and setting the software operator's panel, 587
 Displaying and setting the workpiece origin offset value, 564
 Displaying and setting tool life management data, 569
 Displaying directory of floppy disk, 485
 Displaying memory used and a list of programs, 557
 Displaying operator message, 586
 Displaying pattern data and pattern menu, 567
 Displaying the directory, 486
 Displaying the pattern menu, 347
 Displaying the program number and sequence number, 589
 Displaying the program number, sequence number, and status, and warning messages for data setting, 589
 Displaying the status and warning for data setting, 590
 DNC operation, 431
 Drilling cycle counter boring cycle (G82), 145
 Drilling cycle, spot drilling (G81), 143
 Dry run, 454
 Dwell (G04), 63
 Dynamic graphic display, 598

«E»

Editing a part program, 384
 Editing of custom macro B, 515
 Editing programs, 493
 Emergency stop, 458
 Exact stop (G09, G61) cutting mode (G64) tapping mode (G63), 59
 Extended part program editing function, 507
 External I/O devices, 403
 External motion function (G81), 186

External output commands, 333

«F»

FANUC FA Card, 406
 FANUC Floppy Cassette, 405
 FANUC Handy File, 405
 FANUC PPR, 406
 Feed functions, 49
 Feed–feed function, 13
 Feedrate clamp based on arc radius, 74
 Feedrate override, 452
 File deletion, 473
 File search, 472
 Files, 470
 Fine boring cycle (G76), 141
 Function keys, 399
 Function keys and soft keys, 398
 Functions to simplify programming, 132

«G»

G65 (modal call), 282
 General flow of operation of CNC machine tool, 5
 General screen operations, 398
 Graphic display, 390
 Graphics display, 592
 Graphics function, 591
 Grinding–wheel wear compensation by continuous dressing (0–GSC, 0–GSD/II), 181

«H»

Header, 362
 Heading a program, 498
 Helical cutting (G02, G03), 41
 High speed cycle cutting, 360
 High–speed peck drilling cycle (G73), 137
 How to indicate command dimensions for moving the tool – absolute, incremental commands, 18
 How to view the position display change without running the machine, 383

«I»

In–feed grinding along the Y and Z axes at the end of table swing (0–GSC, 0–GSD/II), 183
 Inch/metric conversion (G20, G21), 98
 Incorrect threaded length, 643
 Increment system, 27
 Incremental feed, 415

Index table indexing function, 272
 Input command from MDI, 257
 Inputting a program, 474
 Inputting and outputting parameters and pitch error compensation data, 481
 Inputting custom macro B common variables, 483
 Inputting offset data, 479
 Inputting parameters, 481
 Inputting/outputting custom macro B common variables, 483
 Inserting a word, 499
 Inserting, altering and deleting a word, 494
 Interference check, 249
 Intermittent-feed surface grinding cycle (G79), 179
 Internal circular cutting feedrate change, 62
 Interpolation function, 32
 Interruption type custom macro, 337

«J»

Jog feed, 413

«K»

Key input and input buffer, 400
 Kind of variables, 286

«L»

Left-handed rigid tapping cycle (G74), 167
 Left-handed tapping cycle (G74), 139
 Limitations, 332
 Linear interpolation (G01), 35
 List of functions and tape format, 634
 Local coordinate system, 90

«M»

M98 (single call), 280
 Machine coordinate system, 82
 Machine lock and auxiliary lock, 451
 Macro call, 317
 Macro call using an M code, 325
 Macro call using G code, 324
 Macro Instruction and Branch Instruction (G65), 291
 Macro statements and NC statements, 312
 Manual absolute ON and OFF, 418
 Manual handle feed, 416
 Manual handle interruption, 443
 Manual operation, 376, 410

Manual reference position return, 411
 Maximum stroke, 27
 MDI operation, 426
 Memory operation, 424
 Memory operation using Series 10/11 tape format, 359
 Menu programming, 526
 Merging a program, 511
 Method of replacing battery, 625
 Mirror image, 446
 Modal call (G66), 322
 Moving part of a program, 510
 Multiple M commands in a single block, 117

«N»

Name of axes, 27
 Next block display screen, 551
 Nomographs, 642
 Normal direction control (G150, G151, G152), 275
 Notes on custom macro, 296
 Notes on reading this manual, 7
 Number of control axes, 361
 Number of registered cycles, 362

«O»

Offset data input and output, 479
 Operating monitor display, 547
 Operation, 622
 Operation of portable tape reader, 673
 Operational devices, 392
 Optional angle chamfering and corner rounding, 184
 Outputting a program, 476
 Outputting custom macro B common variable, 484
 Outputting offset data, 480
 Outputting parameters, 482
 Outputting programs, 489
 Overall position display, 544
 Overcutting by cutter compensation, 254
 Overtravel, 459
 Overview of cutter compensation C (G40 to G42), 218

«P»







Part drawing and tool movement, 14
 Parts count display, run time display, 390
 Path drawing, 598
 Pattern data display, 351
 Pattern data input function, 346
 Peck drilling cycle (G83), 147

Peek rigid tapping cycle (G84 or G74), 169
 Plane selection, 92
 Plunge grinding cycle (G75), 173
 Polar coordinate command (G15, G16), 95
 Portable tape reader, 407
 Position display in the relative coordinate system, 542
 Position display in the work coordinate system, 541
 Positioning (G00), 33
 Positive/negative cutter compensation value and tool center path, 216
 Power disconnection, 409
 Power on/off, 408
 Preparatory function (G function), 28
 Processing macro statements, 330
 Program check screen, 552
 Program components other than program sections, 120
 Program configuration, 21, 118
 Program contents display, 549
 Program display, 388
 Program input/output, 474
 Program number search, 504
 Program restart, 432
 Program screen for MDI operation, 554
 Program section configuration, 123
 Programmable parameter entry (G10), 357
 Pulse distribution, 361

$\ll R \gg$

Radius direction error at circle cutting, 650
 Range of command value, 639
 Rapid traverse, 53
 Rapid traverse bell-shaped acceleration/deceleration, 76
 Rapid traverse override, 453
 Reading files, 488
 Reference position, 77
 Reference position (machine-specific position), 14
 Register, change and delete of tool life management data, 110
 Registering custom macro programs, 331
 Reorganizing memory, 518
 Repetition (WHILE statement), 314
 Replacement of words and addresses, 513
 Replacing batteries for absolute pulse coder, 627
 Replacing CNC battery for memory back-up, 626
 Rigid tapping, 164
 Rigid tapping (G84), 165
 Rotary axis roll-over, 368

$\ll S \gg$

Safety function, 457
 Sample program, 328
 Scaling (G50, G51), 260
 Scheduling function, 436
 Screens displayed by function key  , 571
 Screens displayed by function key  , 559
 Screens displayed by function key  , 586
 Screens displayed by function key  , 540
 Screens displayed by function key  (in auto mode or MDI mode), 548
 Screens displayed by function key  (in the edit mode), 556
 Selecting a workpiece coordinate system, 84
 Selection of tool used for various machining – tool function, 20
 Sequence number comparison and stop, 581
 Sequence number search, 448
 Series 0-D specifications, 677
 Setting a workpiece coordinate system, 83
 Setting and displaying data, 532
 Setting and displaying the tool offset value, 560
 Simple calculation of incorrect thread length, 645
 Simple call (G65), 317
 Simple synchronous control, 365
 Single block, 455
 Single direction positioning (G60), 34
 Skip function (G31), 47
 Small-diameter peck drilling cycle, 187
 Solid graphics, 607
 Specification method, 338
 Specifying the spindle speed value directly (S5-digit command), 101
 Specifying the spindle speed with a binary code, 101
 Spindle speed function (S function), 100
 Status when turning power on, when clear and when reset, 651
 Stroke check, 460
 Subprogram, 129
 Subprogram call function, 441
 Subprogram call using an M code, 326
 Subprogram call using M code, 280
 Subprogram call using T code, 281
 Subprogram calls using a T code, 327
 Supplementary explanation for copying, moving and merging, 512

Switch between cutter compensation left and cutter compensation right, 214

System variables, 301

« *T* »

Tape code list, 631

Tapping cycle (G84), 149

Test operation, 450

Testing a program, 382

The second auxiliary functions (B codes), 117

Tool compensation values, number of compensation values, and entering values from the program (G10), 258

Tool figure and tool motion by program, 24

Tool function (T function), 106

Tool length measurement, 562

Tool length offset (G43, G44, G49), 192

Tool life, 114

Tool life management command in a machining program, 111

Tool life management data, 109

Tool life management function, 108

Tool movement along workpiece parts figure–interpolation, 12

Tool movement by programming–automatic operation, 378

Tool movement in offset mode, 229

Tool movement in offset mode cancel, 243

Tool movement in start-up, 225

Tool movement range – stroke, 25

Tool offset (G45 to G48), 200

Tool path at corner, 647

Tool selection function, 107

Turning on the power, 408

« *U* »

Unconditional branch (GOTO statement), 313

« *V* »

Variables, 285, 298

« *W* »

Word search, 496

Workpiece coordinate system, 83

Revision Record

FANUC Series 0/00/0-Mate FOR MACHINING CENTER OPERATOR'S MANUAL (B-61404E)

05	Aug., '94	•All pages are revised				
04	Aug., '92	•Addition of common variable addition •Alteration of RS-232-C/RS-422 Interface •Alteration of Parameters •Alteration of Error Code List				
03	Sep., '90	•Correction of errors •Addition of "APPENDIX 7 RS-232-C/RS-422 INTERFACE"	07	Aug., '97		<ul style="list-style-type: none"> • Addition of Safety Precautions • Addition of Series 0-D • Addition of Appendix I "Series 0-D SPECIFICATIONS" • Correction of errors
02	Mar., '89	•Addition of 0-Mate MC function	06	May, '97		<ul style="list-style-type: none"> • Addition of following items <ul style="list-style-type: none"> <input type="checkbox"/> Cutting feed linear acceleration/deceleration before interpolation <input type="checkbox"/> Cutting feed block overlap <input type="checkbox"/> Automatic corner deceleration <input type="checkbox"/> Feedrate clamp based on arc radius <input type="checkbox"/> Rapid traverse bell-shaped acceleration/deceleration <input type="checkbox"/> Small-diameter peck drilling cycle <input type="checkbox"/> Operating monitor display <input type="checkbox"/> Correspondence between English key and Symbolic key • Correction of errors
01	Oct., '88	_____				
Edition	Date	Contents	Edition	Date	Contents	

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