



GE Fanuc Automation

Computer Numerical Control Products

*Series 0 / 00 / 0-Mate
for Lathe*

Operator's Manual

GFZ-61394E/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. The operator's manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

WARNING

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

NOTE

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

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

3

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied 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**

For some units, the chapter covering daily maintenance in the operator's manual or programming manual describes the fuse replacement procedure.

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.

For this reason, only those personnel who have received approved safety and maintenance training may perform this work.

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

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

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	15
1.3 PART DRAWING AND TOOL MOVEMENT	16
1.3.1 Reference Position (Machine-Specific Position)	16
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System	17
1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands ..	20
1.4 CUTTING SPEED – SPINDLE SPEED FUNCTION	23
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION	24
1.6 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION	25
1.7 PROGRAM CONFIGURATION	26
1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM	29
1.9 TOOL MOVEMENT RANGE – STROKE	30
2. CONTROLLED AXES	31
2.1 CONTROLLED AXES	32
2.2 NAMES OF AXES	33
2.3 INCREMENT SYSTEM	34
2.4 MAXIMUM STROKES	35
3. PREPARATORY FUNCTION (G FUNCTION)	36
4. INTERPOLATION FUNCTIONS	40
4.1 POSITIONING(G00)	41
4.2 LINEAR INTERPOLATION (G01)	43
4.3 CIRCULAR INTERPOLATION (G02,G03)	44
4.4 POLAR COORDINATE INTERPOLATION (G112,G113)	48
4.5 CYLINDRICAL INTERPOLATION (G107)	52
4.6 CONSTANT LEAD THREADING (G32)	55
4.7 VARIABLE-LEAD THREAD CUTTING (G34)	59
4.8 CONTINUOUS THREAD CUTTING	60
4.9 SKIP FUNCTION (G31)	61
4.10 MULTI-STEP (0-GCC, 00-GCC, 0-GCD/II)	63
4.11 SKIP FUNCTION BY TORQUE LIMIT ARRIVAL SIGNAL	64

5. FEED FUNCTIONS	66
5.1 GENERAL	67
5.2 RAPID TRAVERSE	69
5.3 CUTTING FEED	70
5.4 DWELL (G04)	73
5.5 DWELL BY TURNING TIMES OF SPINDLE	74
6. REFERENCE POSITION	75
7. COORDINATE SYSTEM	78
7.1 MACHINE COORDINATE SYSTEM	79
7.2 WORKPIECE COORDINATE SYSTEM	80
7.2.1 Setting a Workpiece Coordinate System	80
7.2.2 Selecting a Workpiece Coordinate System	82
7.2.3 Changing Workpiece Coordinate System	83
7.2.4 Workpiece Coordinate System Shift	85
7.3 LOCAL COORDINATE SYSTEM	86
7.4 PLANE SELECTION	88
8. COORDINATE VALUE AND DIMENSION	89
8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)	90
8.2 INCH/METRIC CONVERSION (G20, G21)	91
8.3 DECIMAL POINT PROGRAMMING	92
8.4 DIAMETER AND RADIUS PROGRAMMING	94
9. SPINDLE SPEED FUNCTION	95
9.1 SPECIFYING THE SPINDLE SPEED WITH A BINARY CODE	96
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND)	96
9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)	96
9.4 SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)	100
9.5 SPINDLE POSITIONING FUNCTION	103
9.5.1 Spindle Orientation	103
9.5.2 Spindle Positioning	103
9.5.3 Canceling Spindle Positioning	105
10. TOOL FUNCTION (T FUNCTION)	106
10.1 TOOL SELECTION	107
10.2 SIMPLIFIED TOOL LIFE MANAGEMENT	108
10.2.1 Compensation Applied to A Programmed T code	109
10.3 TOOL LIFE MANAGEMENT	111
10.3.1 Program of Tool Life Data	111
10.3.2 Counting a Tool Life	113
10.3.3 Specifying a Tool Group in a Machining Program	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)	118
11.4	OUTPUTTING SIGNAL NEAR END POINT	119
12	PROGRAM CONFIGURATION	121
12.1	PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS	123
12.2	PROGRAM SECTION CONFIGURATION	126
12.3	SUBPROGRAM	132
13	FUNCTIONS TO SIMPLIFY PROGRAMMING	136
13.1	CANNED CYCLE (G90, G92, G94)	137
13.1.1	Outer Diameter/Internal Diameter Cutting Cycle (G90)	137
13.1.2	Thread Cutting Cycle (G92)	139
13.1.3	End Face Turning Cycle (G94)	142
13.1.4	How to Use Canned Cycles (G90, G92, G94)	145
13.2	MULTIPLE REPETITIVE CYCLE (G70 TO G76)	147
13.2.1	Stock Removal in Turning (G71)	147
13.2.2	Stock Removal in Facing (G72)	151
13.2.3	Pattern Repeating (G73)	152
13.2.4	Finishing Cycle (G70)	153
13.2.5	End Face Peck Drilling Cycle (G74)	157
13.2.6	Outer Diameter / Internal Diameter Drilling Cycle (G75)	158
13.2.7	Multiple Thread Cutting Cycle (G76)	159
13.2.8	Notes on Multiple Repetitive Cycle (G70 to G76)	163
13.3	CANNED CYCLE FOR DRILLING (G80 TO G89)	164
13.3.1	Front Drilling Cycle (G83) / Side Drilling Cycle (G87)	167
13.3.2	Front Tapping Cycle (G84) / Side Tapping Cycle (G88)	170
13.3.3	Front Boring Cycle (G85) / Side Boring Cycle (G89)	172
13.3.4	Canned Cycle for Drilling Cancel (G80)	173
13.3.5	Precautions to be Taken by Operator	173
13.3.6	Rigid Tapping	174
13.3.7	Counter Rigid Tapping	177
13.4	CANNED GRINDING CYCLE (0-GCC, 00-GCC, 0-GCD/II)	178
13.4.1	Traverse Grinding Cycle (G71)	178
13.4.2	Traverse Direct Fixed-Dimension Grinding Cycle (G72)	179
13.4.3	Oscillation Grinding Cycle (G73)	180
13.4.4	Oscillation Direct Fixed-Dimension Grinding Cycle (G74)	181
13.5	CHAMFERING AND CORNER R	182
13.6	MIRROR IMAGE FOR DOUBLE TURRET (G68, G69)	185
13.7	DIRECT DRAWING DIMENSIONS PROGRAMMING	186
14	COMPENSATION FUNCTION	191
14.1	TOOL OFFSET	192
14.1.1	Tool Geometry Offset and Tool Wear Offset	192
14.1.2	T code for Tool Offset	193
14.1.3	Tool Selection	193
14.1.4	Offset Number	193
14.1.5	Offset	194
14.2	OVERVIEW OF TOOL NOSE RADIUS COMPENSATION	197
14.2.1	Imaginary Tool Nose	197
14.2.2	Direction of Imaginary Tool Nose	199
14.2.3	Offset Number and Offset Value	200
14.2.4	Work Position and Move Command	202

14.2.5	Notes on Tool Nose Radius Compensation	207
14.3	DETAILS OF TOOL NOSE RADIUS COMPENSATION	210
14.3.1	General	210
14.3.2	Tool Movement in Start-Up	212
14.3.3	Tool Movement in Offset Mode	214
14.3.4	Tool Movement in Offset Mode Cancel	226
14.3.5	Interference Check	229
14.3.6	Overcutting by Tool Nose Radius Compensation	234
14.3.7	Correction in Chamfering and Corner Arcs	235
14.3.8	Input Command from MDI	237
14.3.9	General Precautions for Offset Operations	238
14.4	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	239
14.4.1	Tool Compensation and Number of Tool Compensation	239
14.4.2	Changing of Tool Offset Value	240
14.5	AUTOMATIC TOOL OFFSET (G36, G37)	241
15	CUSTOM MACRO A	244
15.1	CUSTOM MACRO COMMAND	245
15.1.1	M98 (Single call)	245
15.1.2	Subprogram Call Using M Code	245
15.1.3	Subprogram Call Using T code	246
15.2	CUSTOM MACRO BODY	247
15.2.1	Variables	247
15.2.2	Kind of Variables	248
15.2.3	Operation Instruction and Branch Instruction (G65)	253
15.2.4	Notes on Custom Macro	258
16	CUSTOM MACRO B	259
16.1	VARIABLES	260
16.2	SYSTEM VARIABLES	263
16.3	ARITHMETIC AND LOGIC OPERATION	268
16.4	MACRO STATEMENTS AND NC STATEMENTS	272
16.5	BRANCH AND REPETITION	273
16.5.1	Unconditional Branch (GOTO Statement)	273
16.5.2	Conditional Branch (IF Statement)	273
16.5.3	Repetition (While Statement)	274
16.6	MACRO CALL	277
16.6.1	Simple Call (G65)	277
16.6.2	Modal Call (G66)	281
16.6.3	Macro Call Using G Code	283
16.6.4	Macro Call Using an M Code	284
16.6.5	Subprogram Call Using an M Code	285
16.6.6	Subprogram Calls Using a T Code	286
16.6.7	Sample Program	286
16.7	PROCESSING MACRO STATEMENTS	288
16.8	REGISTERING CUSTOM MACRO PROGRAMS	290
16.9	LIMITATIONS	291
16.10	EXTERNAL OUTPUT COMMANDS	292
16.11	INTERRUPTION TYPE CUSTOM MACRO	296
16.11.1	Specification Method	297





16.11.2	Details of Functions	298
17.	PATTERN DATA INPUT FUNCTION	306
17.1	DISPLAYING THE PATTERN MENU	307
17.2	PATTERN DATA DISPLAY	311
17.3	CHARACTERS AND CODES TO BE USED FOR THE PATTERN DATA INPUT FUNCTION ..	315
18.	PROGRAMMABLE PARAMETER ENTRY (G10)	317
19.	MEMORY OPERATION BY SERIES 10/11 TAPE FORMAT	318
19.1	ADDRESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 10/11 TAPE FORMAT	319
19.2	EQUAL-LEAD THREADING	320
19.3	SUBPROGRAM CALLING	321
19.4	CANNED CYCLE	322
19.5	MULTIPLE REPETITIVE CANNED TURNING CYCLE	323
20.	HIGH SPEED CYCLE CUTTING	325
20.1	NUMBER OF CONTROL AXES	326
20.2	PULSE DISTRIBUTION	326
20.3	CONFIGURATION OF HIGH SPEED CYCLE CUTTING DATA	327
20.3.1	Number of Registered Cycles	327
20.3.2	Header	327
21.	POLYGONAL TURNING	330
22.	ROTARY AXIS ROLL-OVER	337
23.	ANGULAR AXIS CONTROL (0-GCC, 00-GCC, 0-GCD/II)	338
24.2	SYSTEMS CONTROL FUNCTION (0-TTC)	340
24.1	GENERAL	341
24.2	WAITING FOR TOOL POSTS	343
24.3	TOOL POST INTERFACE CHECK	345
24.3.1	General	345
24.3.2	Data Setting for the Tool Post Interference Check Function	345
24.3.3	Setting and Display of Interference Forbidden Areas for Tool Post Interference Checking	348
24.3.4	Conditions for Making a Tool Post Interference Check	350
24.3.5	Execution of Tool Post Interference Checking	351
24.3.6	Example of Making a Tool Post Interference Check	353
24.4	BALANCE CUT (G68, G69)	355
24.5	CUSTOM MACRO VARIABLES COMMON TO TOOL POSTS	356



III. OPERATION

1.	GENERAL	359
1.1	MANUAL OPERATION	360
1.2	TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION	362

1.3	AUTOMATIC OPERATION	364
1.4	TESTING A PROGRAM	366
1.4.1	Check by Running the Machine	366
1.4.2	How to View the Position Display Change without Running the Machine	367
1.5	EDITING A PART PROGRAM	368
1.6	DISPLAYING AND SETTING DATA	369
1.7	DISPLAY	372
1.7.1	Program Display (See Section III-11.2.1)	372
1.7.2	Current Position Display (See Section III-11.1.1 to 11.1.3)	373
1.7.3	Alarm Display (See Section III-7.1)	373
1.7.4	Parts Count Display, Run Time Display (See Section III-11.5.3)	374
1.7.5	Graphic Display (See Section III-12)	374
1.8	DATA OUTPUT	376
2.	OPERATIONAL DEVICES	377
2.1	CRT/MDI PANELS	378
2.2	FUNCTION KEYS AND SOFT KEYS	381
2.2.1	General Screen Operations	381
2.2.2	Function Keys	382
2.2.3	Key Input and Input Buffer	383
2.3	EXTERNAL I/O DEVICES	385
2.3.1	FANUC Handy File	387
2.3.2	FANUC Floppy Cassette	387
2.3.3	FANUC FA Card	388
2.3.4	FANUC PPR	388
2.3.5	Portable Tape Reader	389
2.4	POWER ON/OFF	390
2.4.1	Turning on the Power	390
2.4.2	Power Disconnection	391
3.	MANUAL OPERATION	392
3.1	MANUAL REFERENCE POSITION RETURN	393
3.2	JOG FEED	395
3.3	INCREMENTAL FEED	397
3.4	MANUAL HANDLE FEED	398
3.5	MANUAL ABSOLUTE ON AND OFF	400
4.	AUTOMATIC OPERATION	405
4.1	MEMORY OPERATION	406
4.2	MDI OPERATION	409
4.3	DNC OPERATION	414
4.4	SEQUENCE NUMBER SEARCH	415
4.5	PROGRAM RESTART	417
4.6	SCHEDULING FUNCTION	422
4.7	SUBPROGRAM CALL FUNCTION	427
4.8	MANUAL HANDLE INTERRUPTION	429
4.9	MIRROR IMAGE	432

5. TEST OPERATION	433
5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK	434
5.2 FEEDRATE OVERRIDE	435
5.3 RAPID TRAVERSE OVERRIDE	436
5.4 DRY RUN	437
5.5 SINGLE BLOCK	438
6. SAFETY FUNCTIONS	441
6.1 EMERGENCY STOP	442
6.2 OVERTRAVEL	443
6.3 STROKE CHECK	444
7. ALARM AND SELF-DIAGNOSIS FUNCTIONS	448
7.1 ALARM DISPLAY	449
7.2 CHECKING BY SELF-DIAGNOSTIC SCREEN	451
8. DATA INPUT/OUTPUT	453
8.1 FILES	454
8.2 FILE SEARCH	456
8.3 FILE DELETION	457
8.4 PROGRAM INPUT/OUTPUT	458
8.4.1 Inputting a Program	458
8.4.2 Outputting a Program	460
8.5 OFFSET DATA INPUT AND OUTPUT	463
8.5.1 Inputting Offset Data	463
8.5.2 Outputting Offset Data	464
8.6 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA	465
8.6.1 Inputting Parameters	465
8.6.2 Outputting Parameters	466
8.7 INPUTTING/OUTPUTTING CUSTOM MACRO B COMMON VARIABLES	467
8.7.1 Inputting Custom Macro B Common Variables	467
8.7.2 Outputting Custom Macro B Common Variable	468
8.8 DISPLAYING DIRECTORY OF FLOPPY DISK	469
8.8.1 Displaying the Directory	470
8.8.2 Reading Files	472
8.8.3 Outputting Programs	473
8.8.4 Deleting Files	474
9. EDITING PROGRAMS	477
9.1 INSERTING, ALTERING AND DELETING A WORD	478
9.1.1 Word Search	480
9.1.2 Heading a Program	482
9.1.3 Inserting a Word	483
9.1.4 Altering a Word	484
9.1.5 Deleting a Word	485
9.2 DELETING BLOCKS	486
9.2.1 Deleting a Block	486
9.2.2 Deleting Multiple Blocks	487

9.3	PROGRAM NUMBER SEARCH	488
9.4	DELETING PROGRAMS	489
9.4.1	Deleting One Program	489
9.4.2	Deleting All Programs	489
9.4.3	Deleting More than One Program by Specifying a Range	490
9.5	EXTENDED PART PROGRAM EDITING FUNCTION	491
9.5.1	Copying an Entire Program	492
9.5.2	Copying Part of a Program	493
9.5.3	Moving Part of a Program	494
9.5.4	Merging a Program	495
9.5.5	Supplementary Explanation for Copying, Moving and Merging	496
9.5.6	Replacement of Words and Addresses	498
9.6	EDITING OF CUSTOM MACROS B	500
9.7	BACKGROUND EDITING	501
9.8	REORGANIGING MEMORY	503
10	CREATING PROGRAMS	504
10.1	CREATING PROGRAMS USING THE MDI PANEL	505
10.2	AUTOMATIC INSERTION OF SEQUENCE NUMBERS	506
10.3	CREATING PROGRAMS IN TEACH IN MODE	508
10.4	MENU PROGRAMMING	511
10.5	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION	513
11	SETTING AND DISPLAYING DATA	517
11.1	SCREENS DISPLAYED BY FUNCTION KEY 	526
11.1.1	Position Display in the Workpiece Coordinate System	527
11.1.2	Position Display in the Relative Coordinate System	528
11.1.3	Overall Position Display	530
11.1.4	Actual Feedrate Display	532
11.1.5	Display of Run Time and Parts Count	533
11.1.6	Operating Monitor Display	535
11.2	SCREENS DISPLAYED BY FUNCTION KEY  (IN AUTO MODE OR MDI MODE)	536
11.2.1	Program Contents Display	536
11.2.2	Current Block Display Screen	537
11.2.3	Next Block Display Screen	538
11.2.4	Program Check Screen	539
11.2.5	Program Screen for MDI Operation	540
11.3	SCREENS DISPLAYED BY FUNCTION KEY  (IN THE EDIT MODE)	541
11.3.1	Displaying Memory Used and a List of Programs	541
11.4	SCREENS DISPLAYED BY FUNCTION KEY 	544
11.4.1	Setting and Displaying the Tool Offset Value	545
11.4.2	Direct Input of Tool Offset Value	548
11.4.3	Direct Input of Tool Offset measured B	550
11.4.4	Counter Input of Offset value	552
11.4.5	Setting the Workpiece Coordinate System Shifting Amount	553
11.4.6	Direct Measured Value Input for Work Coordinate System Shift	555
11.4.7	Y Axis Offset	557
11.4.8	Displaying and Setting the Workpiece Origin Offset Value	559

11.4.9	Displaying and Setting Custom Macro Common Variables	560
11.4.10	Displaying and Setting Tool Life Management Data	561
11.5	SCREENS DISPLAYED BY FUNCTION KEY 	564
11.5.1	Displaying and Entering Setting Data	565
11.5.2	Sequence Number Comparison and Stop	568
11.5.3	Displaying and Setting Run Time, Parts Count, and Time	570
11.5.4	Displaying and Setting Parameters	572
11.5.5	Displaying and Setting Pitch Error Compensation Data	574
11.6	SCREENS DISPLAYED BY FUNCTION KEY 	579
11.6.1	Displaying Operator Message	579
11.6.2	Displaying and Setting the Software Operator's Panel	580
11.7	DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING	582
11.7.1	Displaying the Program Number and Sequence Number	582
11.7.2	Displaying the Status and Warning for Data Setting	583
12	GRAPHICS FUNCTION	584
12.1	GRAPHICS DISPLAY	585
13	DISPLAY AND OPERATION OF 00-TC/00-GCC	591
13.1	DISPLAY	592
13.2	OPERATION	593
 IV. MAINTENANCE		
1	METHOD OF REPLACING BATTERY	597
1.1	REPLACING CNC BATTERY FOR MEMORY BACK-UP	598
1.2	REPLACING BATTERIES FOR ABSOLUTE PULSE CODER	599
 APPENDIX		
A	TAPE CODE LIST	603
B	LIST OF FUNCTIONS AND TAPE FORMAT	606
C	RANGE OF COMMAND VALUE	610
D	NOMOGRAPHS	613
D.1	INCORRECT THREADED LENGTH	614
D.2	SIMPLE CALCULATION OF INCORRECT THREAD LENGTH	616
D.3	TOOL PATH AT CORNER	618
D.4	RADIUS DIRECTION ERROR AT CIRCLE CUTTING	621

E. STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET	622
F. CHARACTER-TO-CODES CORRESPONDENCE TABLE	624
G. ALARM LIST	625
H. OPERATION OF PORTABLE TAPE READER	644
I. SERIES 0-D SPECIFICATIONS	649
J. CORRESPONDENCE BETWEEN ENGLISH KEY AND SYMBOLIC KEY	656

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 (Table1).

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 alarms, self-diagnosis, and procedures for replacing fuses and 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-61400E).

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—TC	0—TC	Series 0	
FANUC Series 0—TF	0—TF		
FANUC Series 0—TTC	0—TTC		
FANUC Series 0—GCC	0—GCC		
FANUC Series 00—TC	00—TC	Series 00	
FANUC Series 00—GCC	00—GCC		
FANUC Series 0—Mate TC	0—Mate TC	Series 0—Mate	
FANUC Series 0—TD	0—TD	Series 0—D	Series 0—D
FANUC Series 0—GCD	0—GCD		
FANUC Series 0—TD II	0—TD II	Series 0—D II	
FANUC Series 0—GCD II	0—GCD 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 0/00/0-Mate DESCRIPTIONS	B-61392E	
FANUC Series 0/00 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 Executor)	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

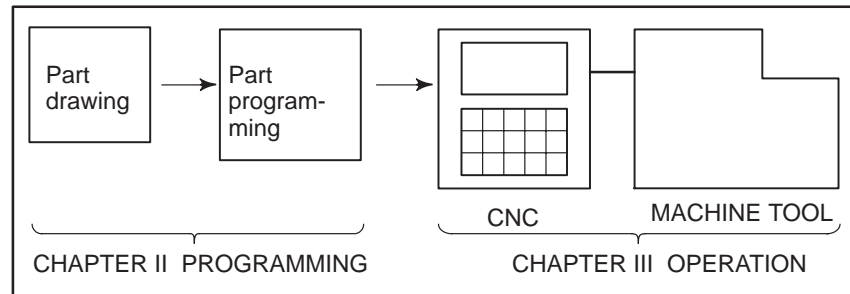
When machining the part using the CNC machine tool, first prepare the program, then operate the CNC machine by using the program.

- 1) First, prepare the program from a part drawing to operate the CNC machine tool.

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.



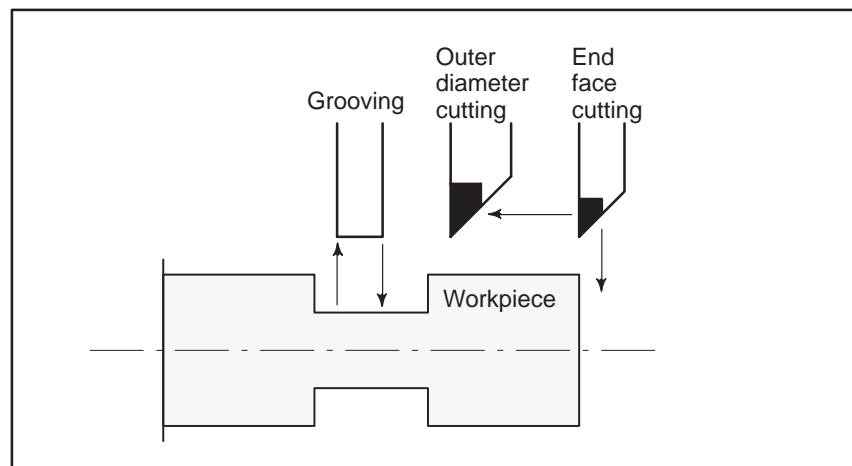
Before the actual programming, make the machining plan for how to machine the part.

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 Cutting procedure	1	2	3
	End face cutting	Outer diameter cutting	Grooving
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 cutting condition according to the workpiece figure, for each cutting.

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 the functions supported by the series and editions of the FANUC Series 0/00/0-Mate software indicated below. Note that the functions described in this manual may not be supported by series and editions of the software other than those listed below.

Model	Series	Edition
FANUC Series 0-TC	0666 0669	Edition 18 and later Edition 01 and later
FANUC Series 0-TF	0667	Edition 08 and later
FANUC Series 0-TTC	0680 0681 0682	Edition 16 and later Edition 16 and later Edition 16 and later
FANUC Series 0-GCC	0861	Edition 07 and later
FANUC Series 00-TC	0668	Edition 01 and later
FANUC Series 00-GCC	0862	Edition 01 and later
FANUC Series 0-Mate TC	0655	Edition 03 and later
FANUC Series 0-TD	0672	Edition 01 and later
FANUC Series 0-GCD	0881	Edition 01 and later
FANUC Series 0-TD II	0673	Edition 01 and later
FANUC Series 0-GCD II	0882	Edition 01 and later

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

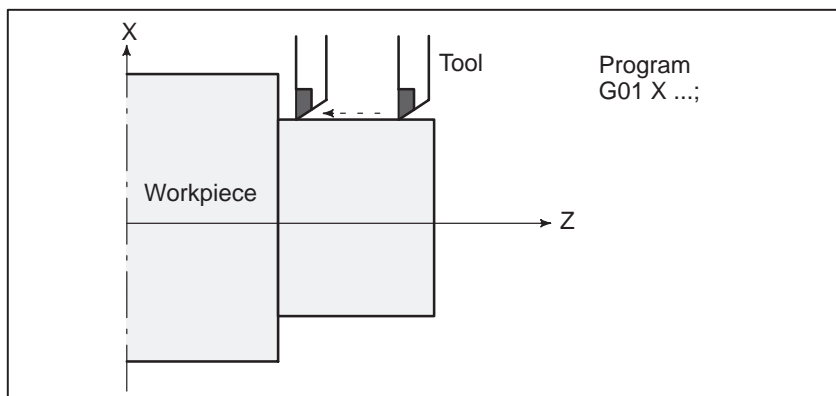


Fig. 1.1 (a) Tool movement along the straight line which is parallel to Z-axis

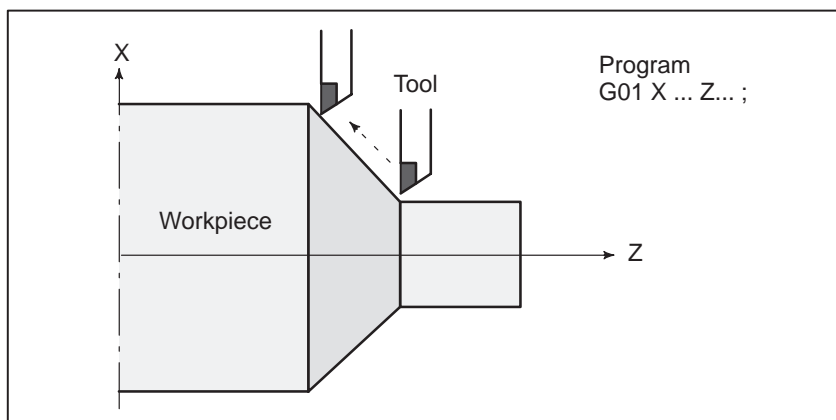


Fig. 1.1 (b) Tool movement along the taper line

- Tool movement along an arc

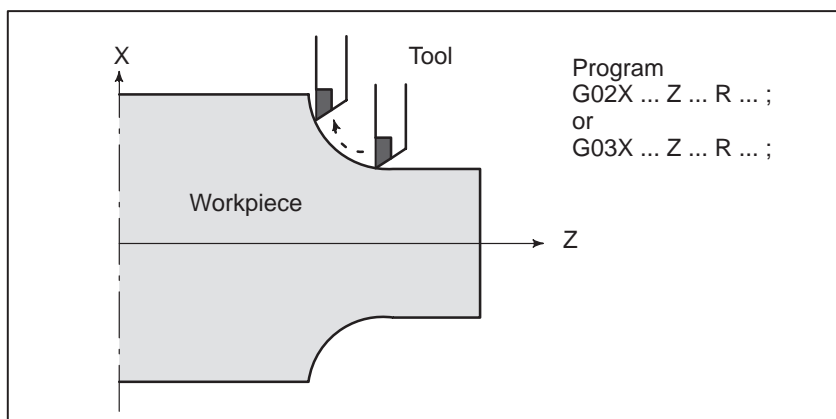


Fig. 1.1 (c) Tool movement along an arc

The term interpolation refers to an operation in which the tool moves along a straight line or arc in the way described above. Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

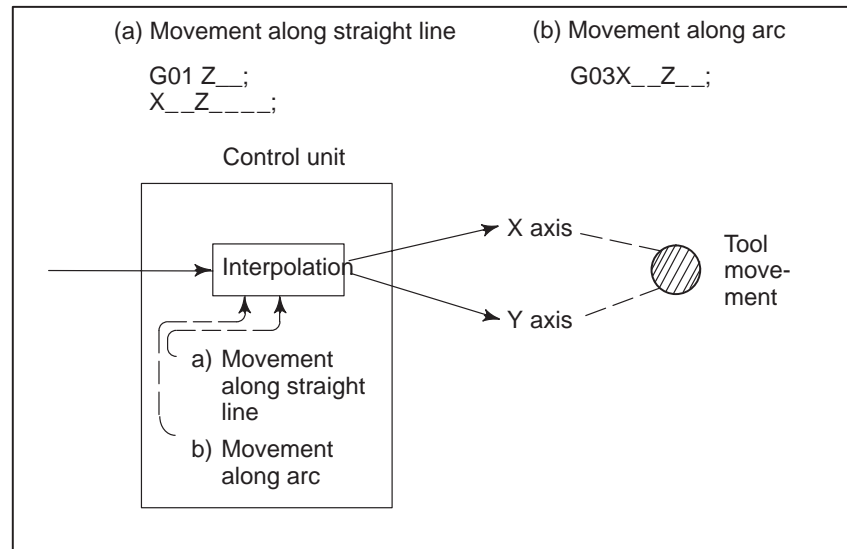


Fig. 1.1 (d) Interpolation function

NOTE

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

• Thread cutting

Threads can be cut by moving the tool in synchronization with spindle rotation. In a program, specify the thread cutting function by G32.

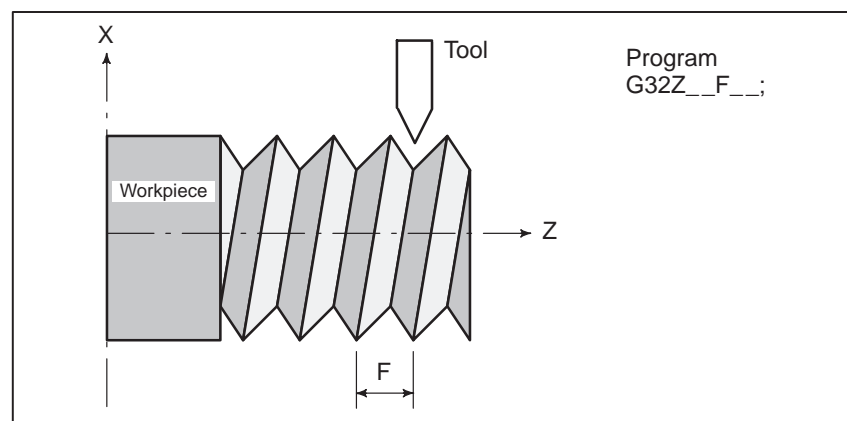


Fig. 1.1 (e) Straight thread cutting

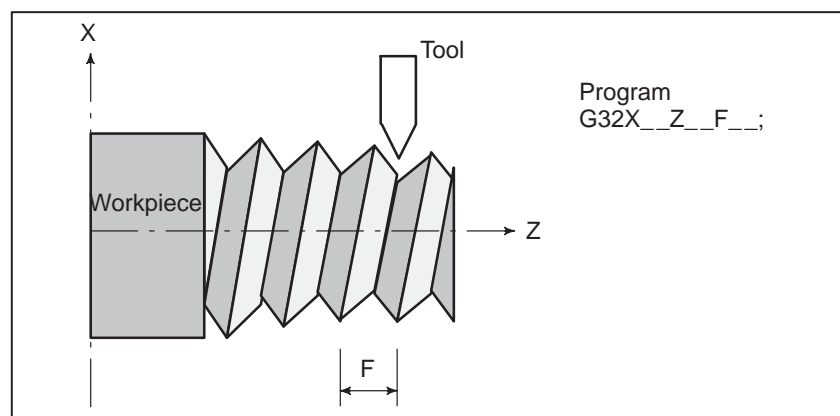


Fig. 1.1 (f) Taper thread cutting

1.2 FEED— FEED FUNCTION

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

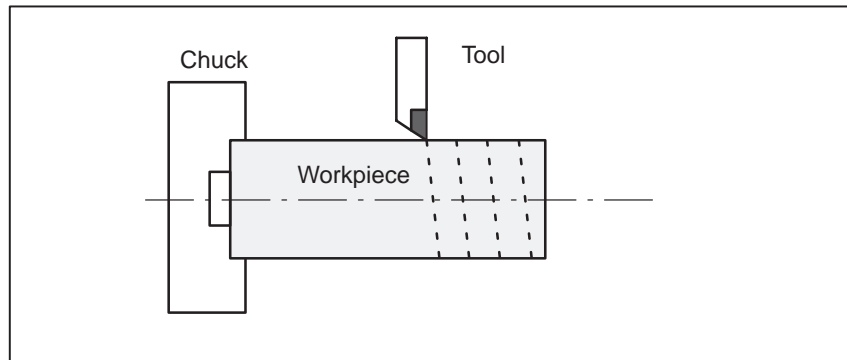


Fig. 1.2 Feed function

Feedrates can be specified by using actual numerics.

For example, the following command can be used to feed the tool 2 mm while the workpiece makes one turn :

F2.0

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.

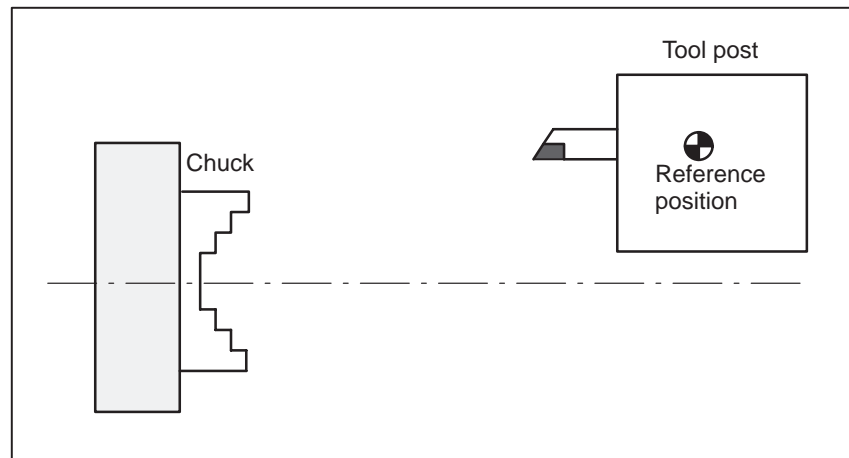


Fig. 1.3.1 Reference position

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

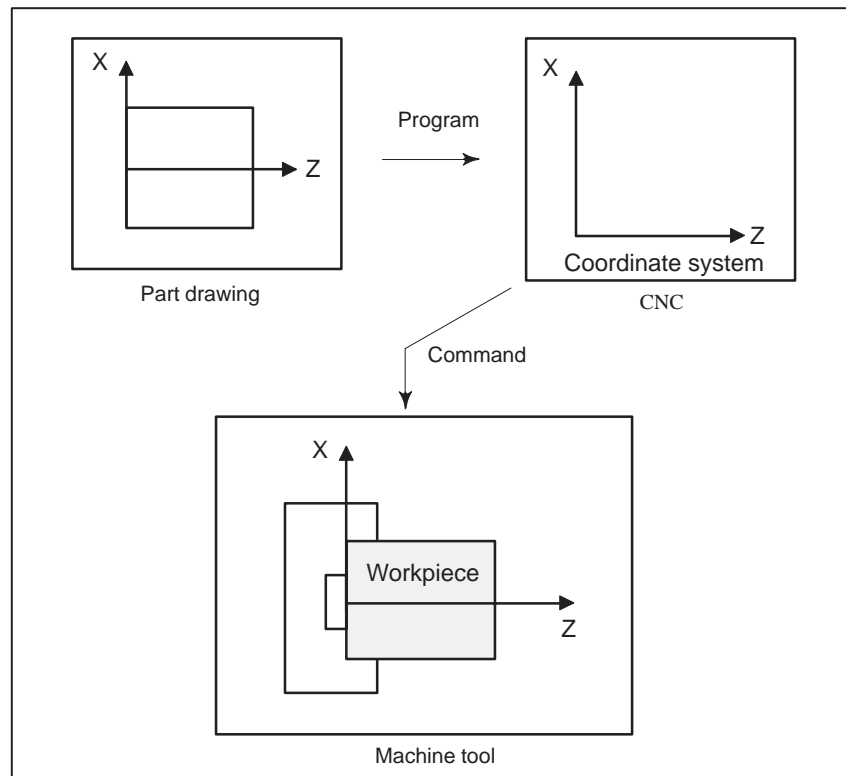


Fig. 1.3.2 (a) Coordinate system

Explanations

- **Coordinate system**

The following two coordinate systems are specified at different locations:
(See II-8)

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. 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.

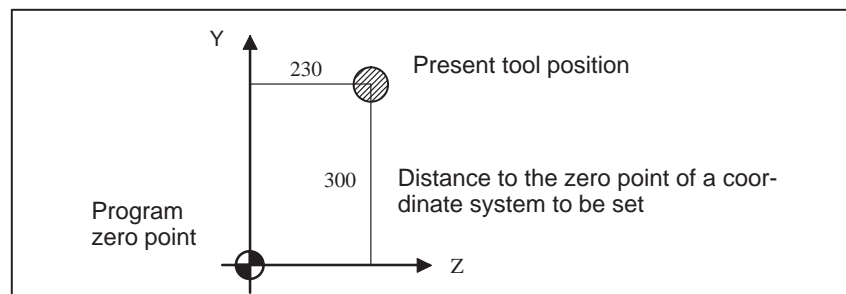


Fig. 1.3.2 (b) Coordinate system specified by the CNC

The 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**

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

1. When coordinate zero point is set at chuck face

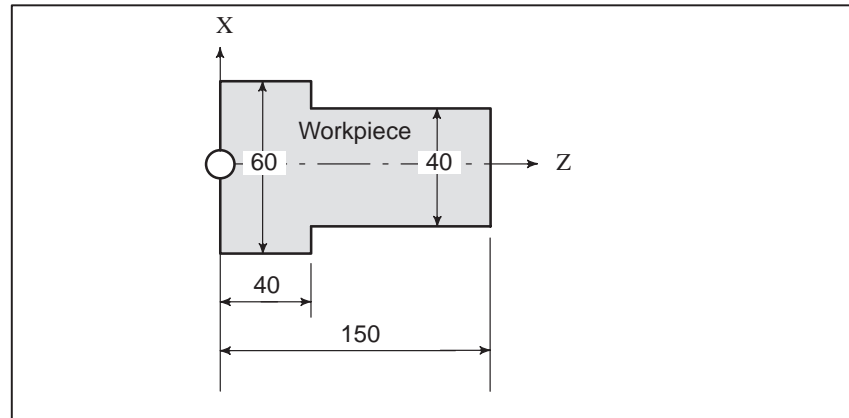


Fig. 1.3.2 (c) Coordinates and dimensions on part drawing

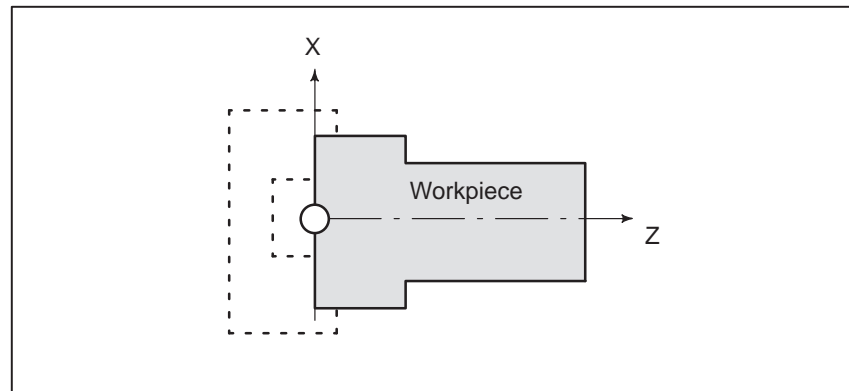


Fig. 1.3.2 (d) Coordinate system on lathe as specified by CNC (made to coincide with the coordinate system on part drawing)

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

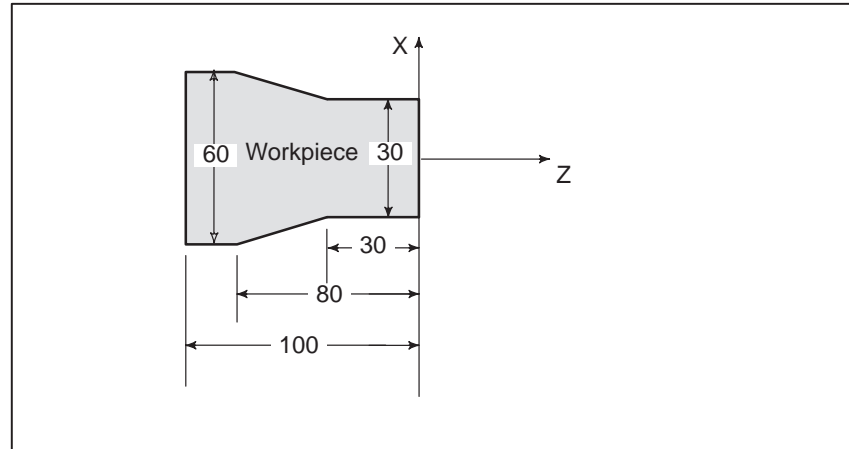
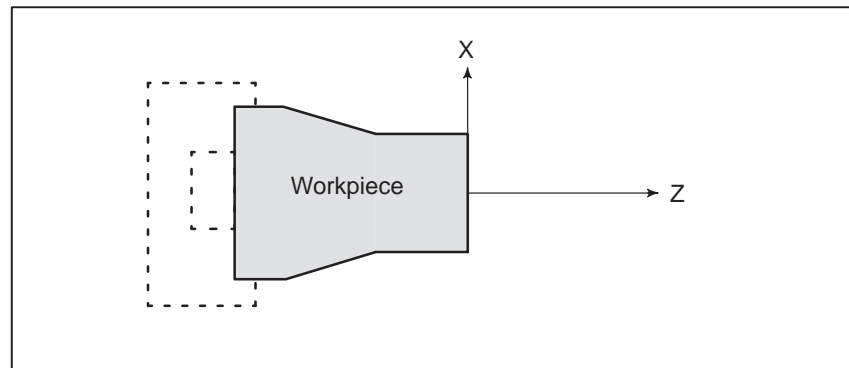


Fig. 1.3.2 (e) Coordinates and dimensions on part drawing



**Fig. 1.3.2 (f) Coordinate system on lathe as specified by CNC
(made to coincide with the coordinate system on part drawing)**

1.3.3

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

Explanations

- **Absolute commands**

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.

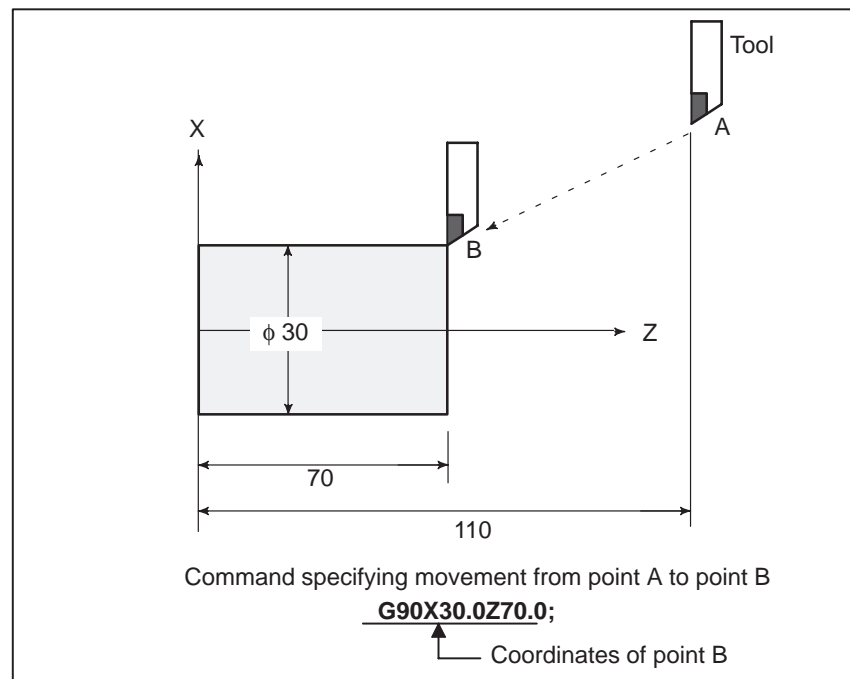


Fig. 1.3.3 (a) Absolute commands

- **Incremental commands**

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

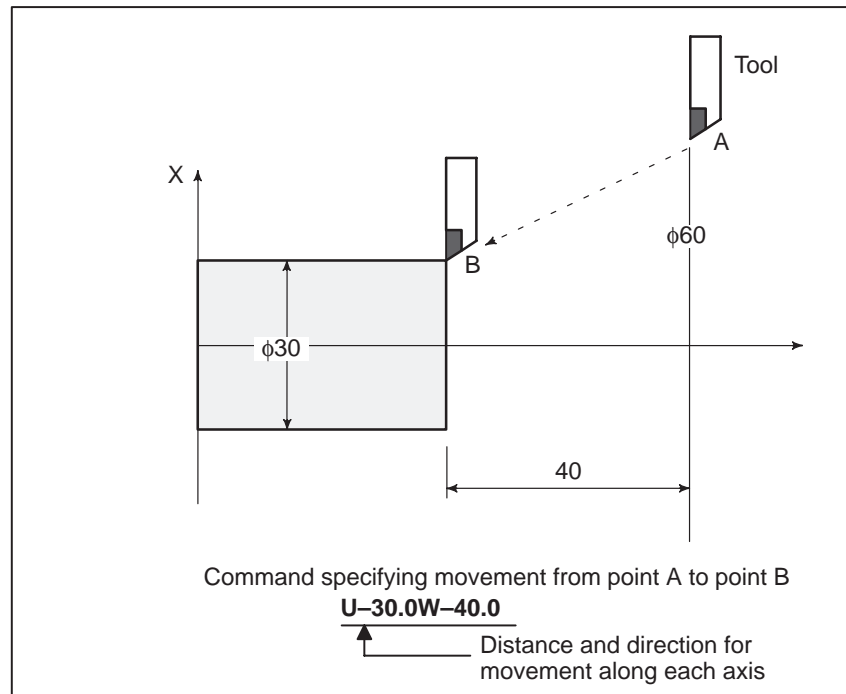


Fig. 1.3.3 (b) Incremental commands

- **Diameter programming / radius programming**

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

1. Diameter programming

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

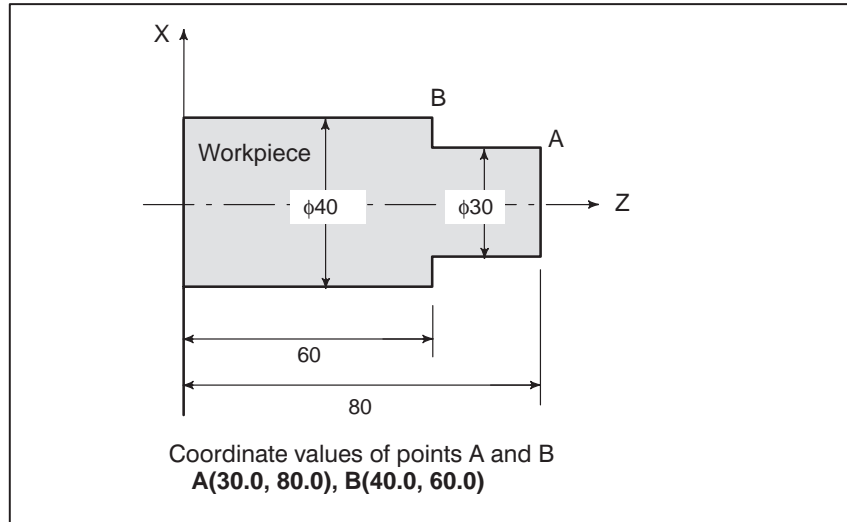


Fig. 1.3.3 (c) Diameter programming

2. Radius programming

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

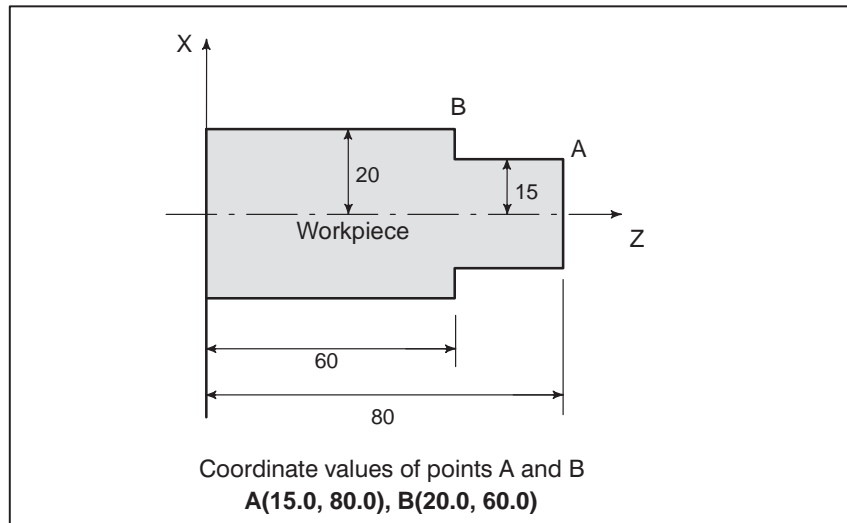


Fig. 1.3.3 (d) Radius programming

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.

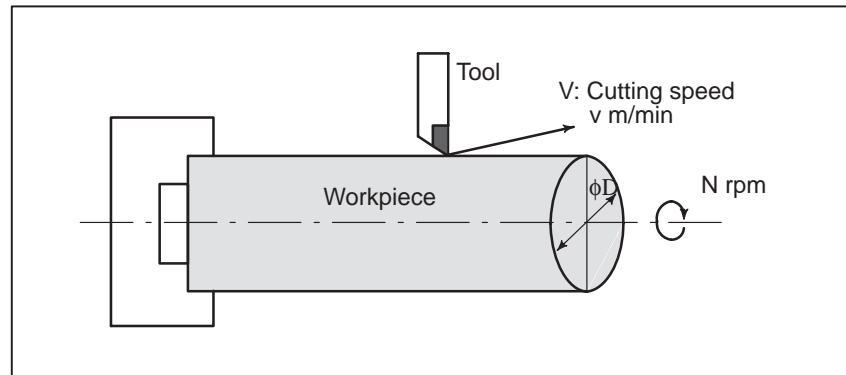


Fig. 1.4 Cutting speed

Examples

<When a workpiece 200 mm in diameter should be machined at a cutting speed of 30 mm/min. >

The spindle speed is approximately 478 rpm, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S478 ;

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

The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.

This function is called the constant surface speed control function (See II-9.3).

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION

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.

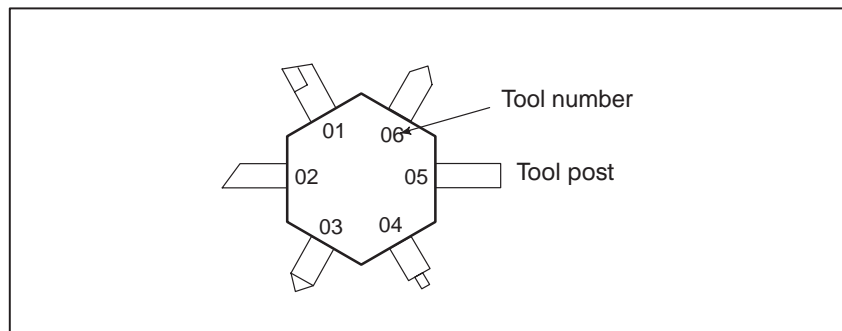


Fig. 1.5 Tool used for various machining

Examples

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

When the tool is stored at location 01 of the tool post, the tool can be selected by specifying **T0101**.

This is called the tool function (See II-10).

1.6 COMMAND FOR MACHINE OPERATIONS – 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.

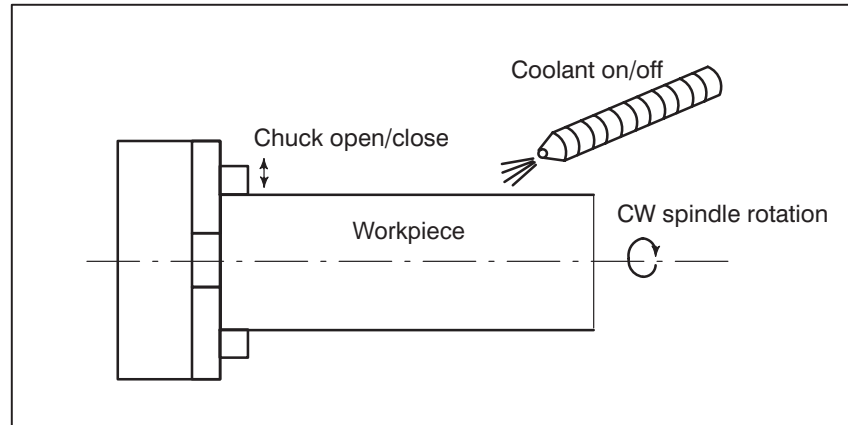


Fig. 1.6 Command for machine operations

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. (See II–11)

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

1.7 PROGRAM CONFIGURATION

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

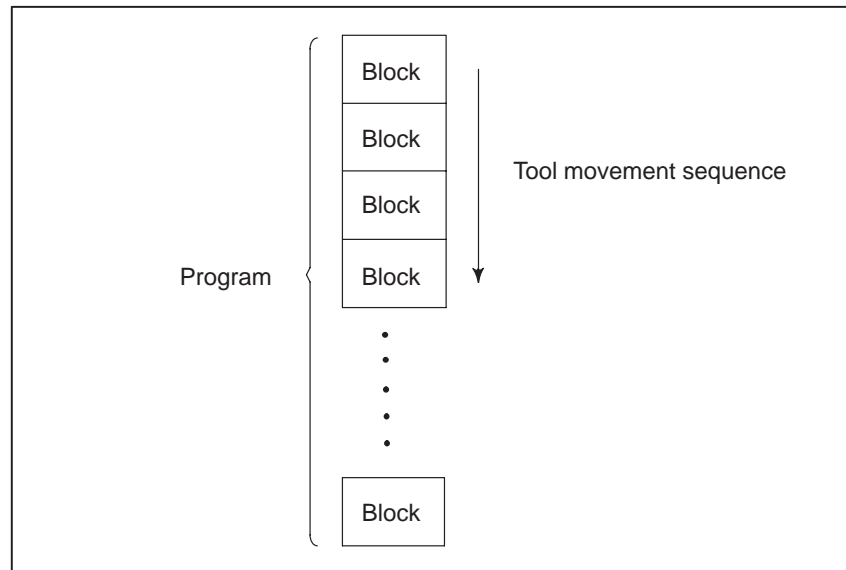


Fig. 1.7 Program configuration

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

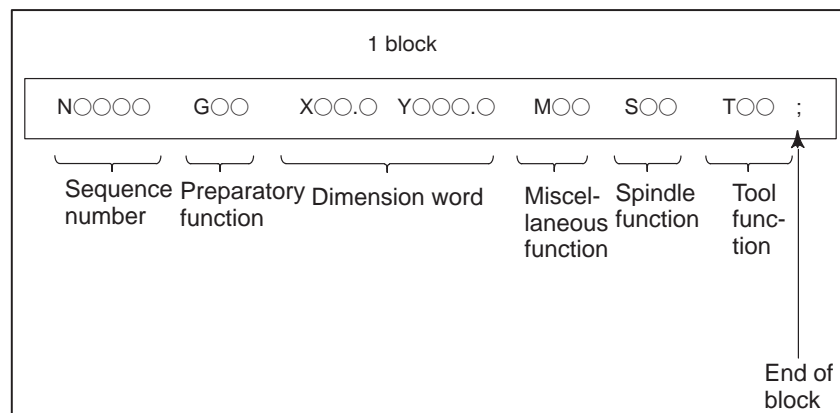


Fig. 1.7 (b) Block configuration

Each block starts with a sequence number which identifies the block, and ends with an end-of-block code which indicates 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

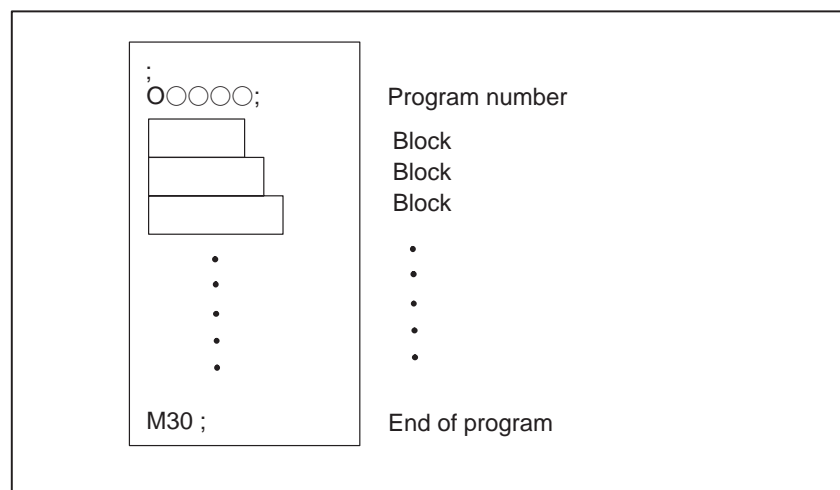
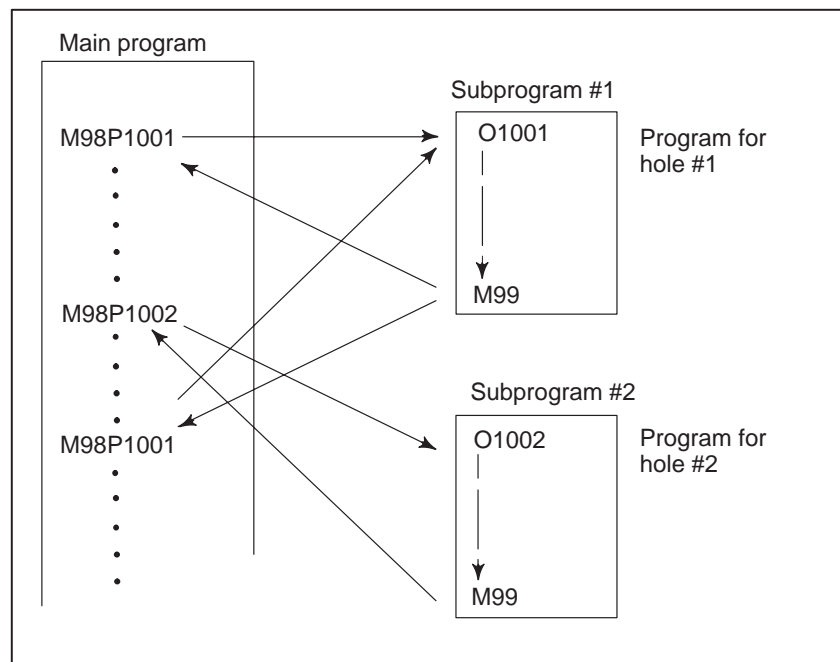


Fig. 1.7 (c) Program configuration

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

- **Tool offset**

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.

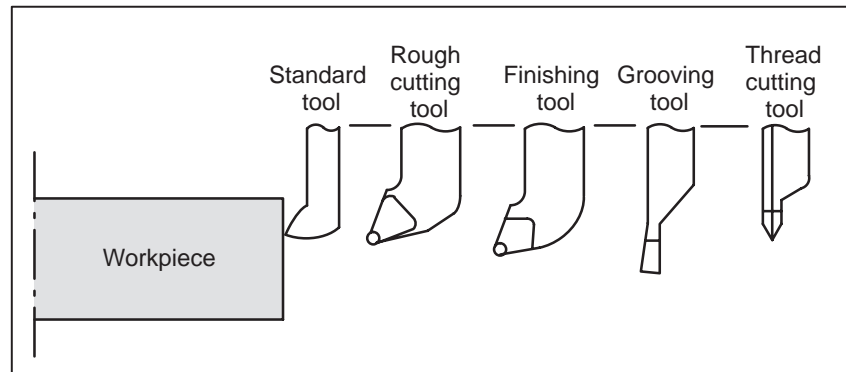
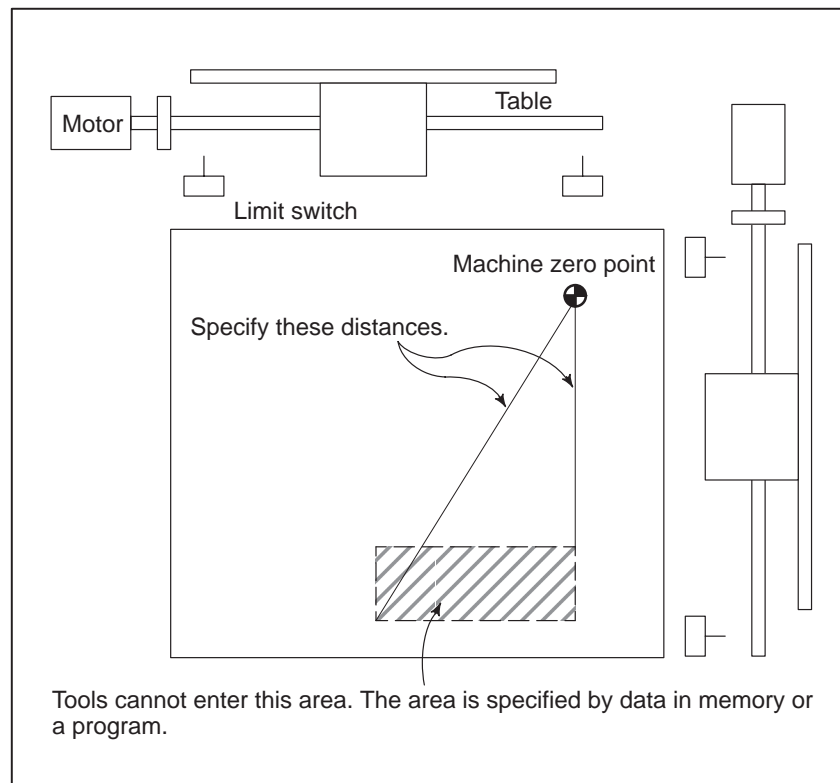


Fig. 1.8 Tool offset

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 16

Item	0-TC,0-TF,0-GCC 00-TC,00-GCC	0-Mate TC	0-TD/II, 0-GCD/II
Number of controlled basic axes	2	2	2
Increase in number of controlled axes (excluding PMC axes)	Up to 4	Up to 2	Up to 4
Number of simultaneously controlled basic axes	2	2	2
Increase in number of simultaneously controlled axes	Up to 2	Up to 2	Up to 4
PMC-based axis control	Up to 2 (*1)	None	None

*1 Amount by which number of controlled axes can be increased under PMC-based axis control. PMC-based axis control can be applied to a maximum of four axes.

• Series 18

Item	0-TTC	
	Tool post 1	Tool post 2
Number of controlled basic axes	2	2
Increase in number of controlled axes (excluding PMC axes)	Up to 4	Up to 3 (*1)
Number of simultaneously controlled basic axes	2	2
Increase in number of simultaneously controlled basic axes	Up to 4	Up to 3
PMC-based axis control	Up to 2 (*2)	None

*1 Including Cs contour control axes

*2 Maximum number of controlled axes to which PMC-based axis control can be applied. It is not possible to further increase the number of axes to which PMC-based axis control is applied.

NOTE

In manual operation (such as jog feed, incremental feed, and manual handle feed), the number of simultaneously controlled axes is 1. Note, however, that this number can be increased to 3 by setting bit 4 of parameter No. 049 to 1.

2.2 NAMES OF AXES

The following fixed axis names are used:

	First axis	Second axis	Third axis(*2)	Fourth axis(*3)
Axis name (absolute programming)	X	Z	C	Y
Incremental programming(*1)	U	W	H	V

*1 Used for incremental programming when G-code system A is used

*2 Command axis name for third axis. The display axis name varies with the setting of bit 0 of parameter No. 030.

*3 Command axis name for fourth axis. The display axis name varies with the setting of bit 4 of parameter No. 030.

Axis name alteration (0-GCC/00-GCC/0-GCD)

The command and display axis names of the third and fourth axes can be set to U, V, W, A, B, C, or Y. Using ASCII code, specify the axis name of the third axis in parameter No. 210, and the axis name of the fourth axis in parameter No. 211. When the axis name alteration function is used, only G-code system B or C can be used.

NOTE

When 0-TTC is used, and axis information such as current positions is displayed on the CRT screen, the displayed axis names may have a suffix for indicating a tool post number (X1 and X2, for example). Such suffixed axis names are displayed only to help identify the axis to which each tool post belongs. In actual programming, no suffixes are attached to axis names, X, Y, Z, U, V, W, A, B, and C.

2.3 INCREMENT SYSTEM

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inches, or degrees.

The increment system is classified into IS-B and IS-C (Tables 2.3.2(a) and 2.3.2 (b)). When IS-C is selected, however, the option for multiplying the increment system by 1/10 is required.

Table 2.3 (a) Increment system IS-B

		Least input increment	Least command increment
Metric system machine	mm input	0.001mm(Diameter)	0.0005mm
		0.001mm(Radius)	0.001mm
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.0005mm
		0.0001inch(Radius)	0.001mm
		0.001deg	0.001deg
Inch machine system	mm input	0.001mm(Diameter)	0.00005inch
		0.001mm(Radius)	0.0001inch
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.00005inch
		0.0001inch(Radius)	0.0001inch
		0.001deg	0.001deg

Table 2.3 (b) Increment system IS-C

		Least input increment	Least command increment
Metric system machine	mm input	0.0001mm(Diameter)	0.00005mm
		0.0001mm(Radius)	0.0001mm
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.00005mm
		0.0001inch(Radius)	0.0001mm
		0.0001deg	0.0001deg
Inch machine system	mm input	0.0001mm(Diameter)	0.000005inch
		0.0001mm(Radius)	0.00001inch
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.000005inch
		0.00001inch(Radius)	0.00001inch
		0.0001deg	0.0001deg

2.4 MAXIMUM STROKES

The maximum stroke controlled by this CNC is shown in the table below:
Maximum stroke=Least command increment ± 99999999

Table 2.4 (a) Maximum stroke

Increment system		Maximum strokes
IS-B	Metric machine system	± 99999.999 mm ± 99999.999 deg
	Inch machine system	± 9999.9999 inch ± 99999.999 deg
IS-C	Metric machine system	± 9999.9999 mm ± 9999.9999 deg
	Inch machine system	± 999.99999 inch ± 9999.9999 deg

NOTE

- 1 The unit in the table is a diameter value with diameter programming and a radius value in radius programming.
- 2 A command exceeding the maximum stroke cannot be specified.
- 3 The actual stroke depends on the machine tool.

3

PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.

G codes are divided into the following two types.

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

(Example)

G01 and G00 are modal G codes.

$$\left. \begin{array}{l} \text{G01X}_{\text{---}}; \\ \text{Z}_{\text{---}}; \\ \text{X}_{\text{---}}; \\ \text{G00Z}_{\text{---}}; \end{array} \right\} \text{G01 is effective in this range}$$

There are three G code systems : A,B, and C (Table 3). G-code system A is standard. G-code systems B and C are optional. Bit 5 of parameter No. 036 is used to specify whether G-code system B or C is to be used. When 0-GCC or 00-GCC is used, G-code systems B and C are also standard; which G-code system is used is specified with bits 1 and 5 of parameter No. 036. Basically, this manual assumes the use of G-code system A. In such cases, the use of G code system B or C is described.

Explanations


1. If the CNC enters the clear state (see bit 6 (CLER) of parameter 0045) when the power is turned on or the CNC is reset, the continuous-state G codes change as follows.
 - (1) G codes marked with  in Table 3 are enabled.
 - (2) When the system is cleared due to power-on or reset, whichever specified, either G20 or G21, remains effective.
 - (3) G22 is set when the system is cleared due to power-on. When the system is cleared due to reset, whichever specified, either G22 or G23, remains effective.
 - (4) Setting bit 0 (G01) of parameter (No.011#6) determines which code, either G00 or G01, is effective.
 - (5) Setting bit 3 (G91) of parameter (No.030#7) determines which code, either G90 or G91, is effective.
2. G codes of group 00 except G10 and G11 are single-shot G codes.
3. Alarm 010 is displayed when a G code not listed in the G code list is specified or a G code without a corresponding option is specified.
4. G codes of different groups can be specified in the same block. If G codes of the same group are specified in the same block, the G code specified last is valid.
5. If a G code of group 01 is specified in a canned cycle for drilling, the canned cycle is canceled in the same way as when a G80 command is specified. G codes of group 01 are not affected by G codes for specifying a canned cycle.
6. When G code system A is used for a canned cycle for drilling only the initial level is provided at the return point.
7. G codes are displayed for each group number.

Table 3 G code list (1/2)

G code			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation/Helical interpolation CW
G03	G03	G03		Circular interpolation/Helical interpolation CCW
G04	G04	G04	00	Dwell
G10	G10	G10		Data setting
G11	G11	G11		Data setting made cancel
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G36	G36	G36	00	Automatic tool compensation X
G37	G37	G37		Automatic tool compensation Z
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting, max. spindle speed setting
G52	G52	G52		Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G65	G65	G65	00	Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets ON
G69	G69	G69		Mirror image for double turrets OFF

Table 3 G code list (2/2)

G code			Group	Function
A	B	C		
G70	G70	G72	00	Finishing cycle (Other than 0–GCC, 00–GCC and 0–GCD/II)
G71	G71	G73		Stock removal in turning (Other than 0–GCC, 00–GCC and 0–GCD/II)
G72	G72	G74		Stock removal in facing (Other than 0–GCC, 00–GCC and 0–GCD/II)
G73	G73	G75		Pattern repeating (Other than 0–GCC, 00–GCC and 0–GCD/II)
G74	G74	G76		End face peck drilling (Other than 0–GCC, 00–GCC and 0–GCD/II)
G75	G75	G77		Outer diameter/internal diameter drilling (Other than 0–GCC, 00–GCC and 0–GCD/II)
G76	G76	G78		Multiple threading cycle (Other than 0–GCC, 00–GCC and 0–GCD/II)
G71	G71	G72	01	Traverse grinding cycle (For 0–GCC, 00–GCC and 0–GCD/II)
G72	G72	G73		Traverse direct constant–dimension grinding cycle (For 0–GCC, 00–GCC and 0–GCD/II)
G73	G73	G74		Oscillation grinding cycle (For 0–GCC, 00–GCC and 0–GCD/II)
G74	G74	G75		Oscillation direct constant–dimension grinding cycle (For 0–GCC, 00–GCC and 0–GCD/II)
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per revolution feed
—	G90	G90	03	Absolute programming
—	G91	G91		Incremental programming
—	G98	G98	11	Return to initial level (See Explanations 6)
—	G99	G99		Return to R point level (See Explanations 6)
G107	G107	G107	00	Cylindrical interpolation
G112	G112	G112	21	Polar coordinate interpolation mode
G113	G113	G113		Polar coordinate interpolation mode cancel
G250	G250	G250	20	Polygonal turning mode cancel
G251	G251	G251		Polygonal turning mode

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.

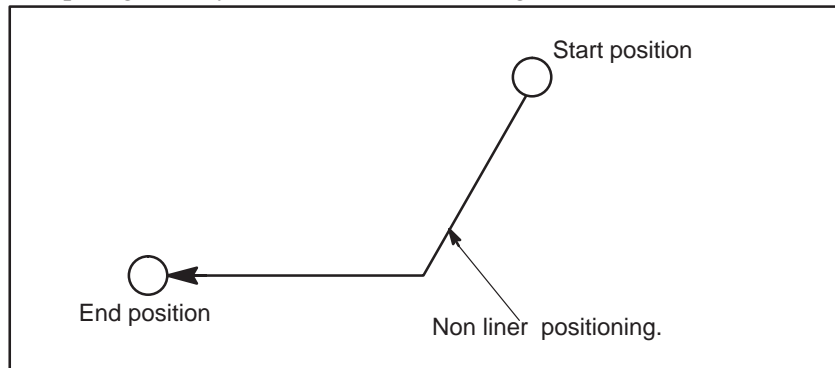
Format

G00IP_;

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.

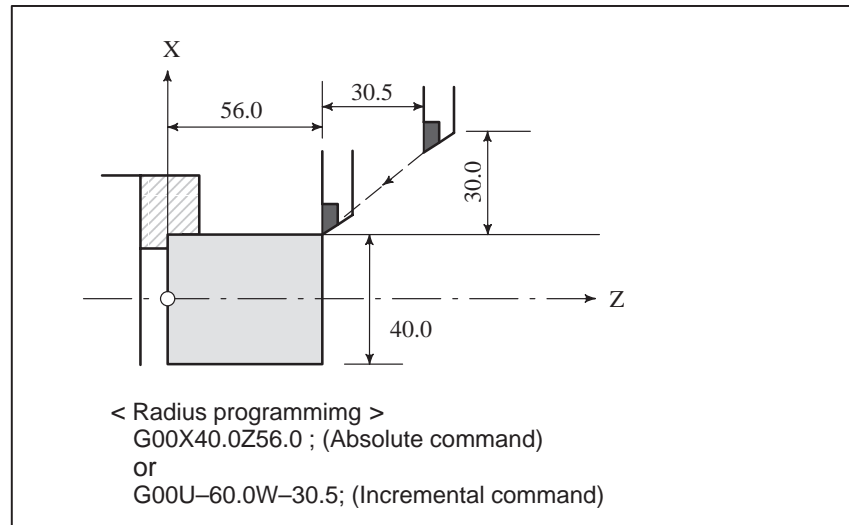


The rapid traverse rate in the G00 command is set to the parameter Nos. 518 to 521 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution 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 Nos. 500 to 503.

Examples



Restrictions

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

4.2 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.

For feed-per-minute mode under 2-axis simultaneous control, the feedrate for a movement along each axis as follows :

G01 α β Ff ;

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

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

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

Examples

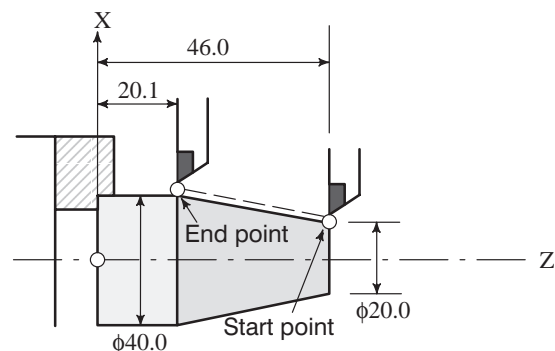
- Linear interpolation

< Diameter programming >

G01X40.0Z20.1F20 ; (Absolute command)

or

G01U20.0W-25.9F20 ; (Incremental command)



4.3 CIRCULAR INTERPOLATION (G02,G03)

Format

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

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

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

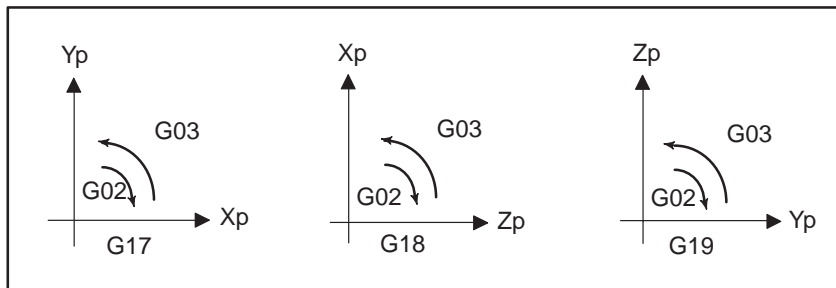
Table 4.3 Description of the Command Format

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 (set by parameter Nos. 279 and 280)
Yp_	Command values of Y axis or its parallel axis (set by parameter Nos. 279 and 280)
Zp_	Command values of Z axis or its parallel axis (set by parameter Nos. 279 and 280)
I_	Xp axis distance from the start point to the center of an arc with sign or radius value
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 value.
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.



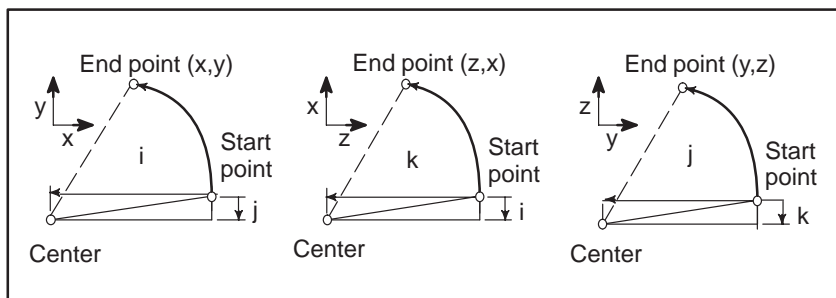
- **Distance moved on an arc**

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.

- **Full-circle programming**

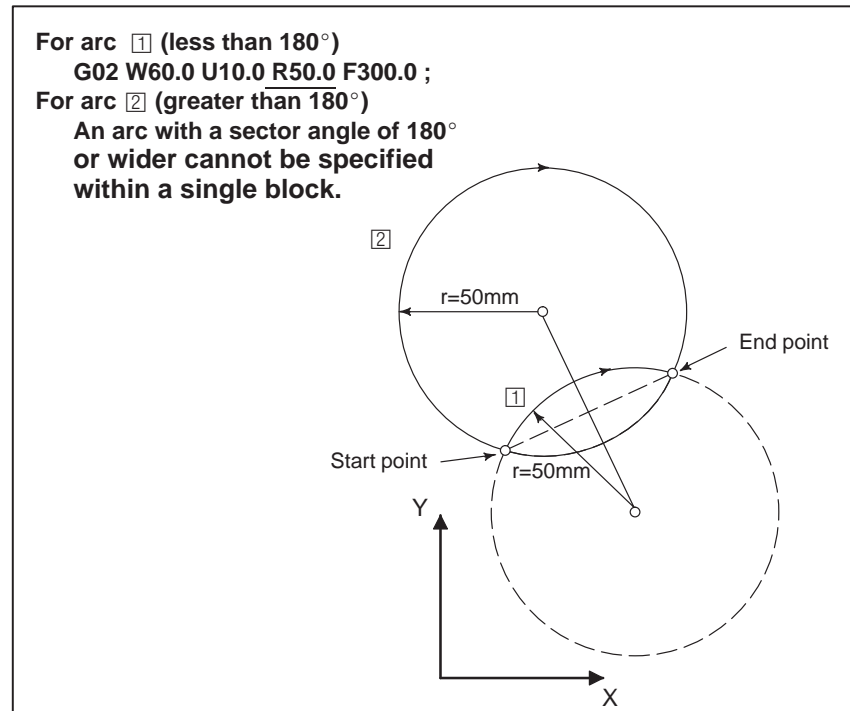
When X_p , Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.

G02I_; (for full-circle)

• Arc radius

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K. In this case, one arc is less than 180° , and the other is more than 180° are considered. An arc with a sector angle of 180° or wider cannot be specified. 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 tool nose radius 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.

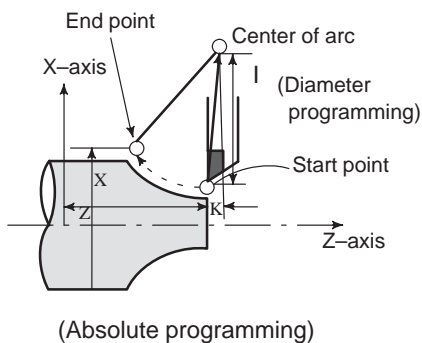
For example, when a ZX plane is specified in G-code B or C, specifying the X-axis or U-axis (parallel to the X-axis) causes alarm No. 028 to be generated.

When an arc having a center angle approaching 180° is specified in R, a center position calculation error may occur. In such a case, specify the center of the arc with I, J, and K.

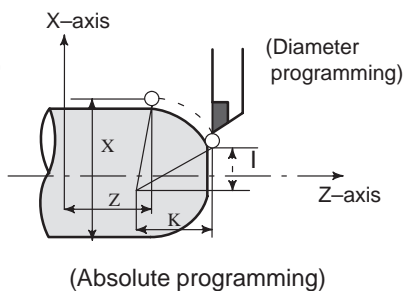
Examples

- Command of circular interpolation X, Z

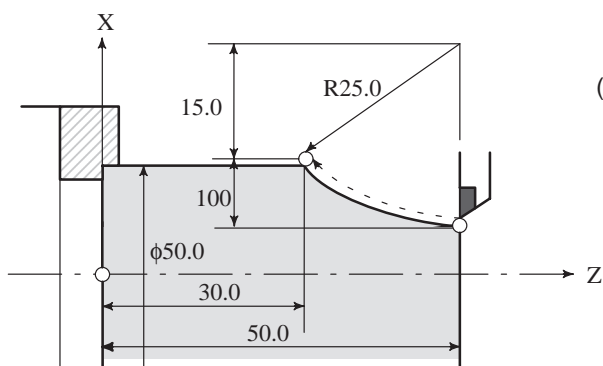
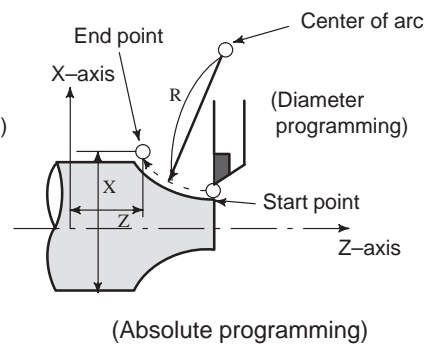
G02X_Z_I_K_F_;



G03X_Z_I_K_F_;



G02X_Z_R_F_;



(Diameter programming)
 G02X50.0Z30.0I25.0F0.3; or
 G02U20.0W-020.0I25.0F0.3; or
 G02X50.0Z30.0R25.0F0.3 or
 G02U20.0W-20.0R25.0F0.3;

4.4 POLAR COORDINATE INTERPOLATION (G112,G113)

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This method is useful in cutting a front surface and grinding a cam shaft on a lathe.

Format

- Specify G112 and G113 in Separate Blocks

G112 ;	Starts polar coordinate interpolation mode
G113 ;	Polar coordinate interpolation mode is cancelled

Explanations

- Polar coordinate interpolation plane**

G112 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 4.6 (a)). Polar coordinate interpolation is performed on this plane.

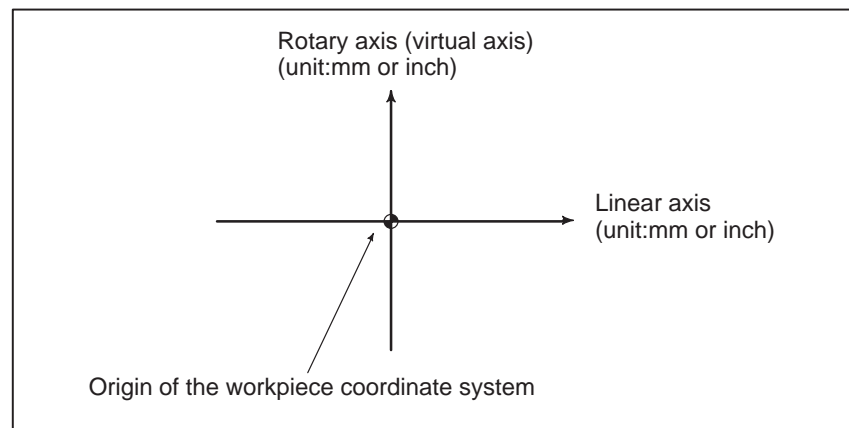


Fig. 4.4 Polar coordinate interpolation plane.

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G113).

The linear and rotation axes for polar coordinate interpolation must be set in parameters (Nos. 291 and 292) beforehand.

CAUTION

The plane used before G112 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G113 (canceling polar coordinate interpolation) is specified.

When the system is reset, polar coordinate interpolation is canceled and the plane specified by G17, G18, or G19 is used.

- **Distance moved and feedrate for polar coordinate interpolation**

The unit for coordinates on the hypothetical axis is the same as the unit for the linear axis (mm/inch)

The unit for the feedrate is mm/min or inch/min

In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotation axis is used as the axis address for the second axis (virtual axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotation axis regardless of the specification for the first axis in the plane. The virtual axis is at coordinate 0 immediately after G112 is specified. Polar interpolation is started assuming the angle of 0 for the position of the tool when G112 is specified.

Specify the feedrate as a speed (relative speed between the workpiece and tool) tangential to the polar coordinate interpolation plane (Cartesian coordinate system) using F.

- **G codes which can be specified in the polar coordinate interpolation mode**

G01 Linear interpolation
G02, G03 Circular interpolation
G04 Dwell
G40, G41, G42 .. Tool nose radius compensation
(Polar coordinate interpolation is applied to the path after cutter compensation.)
G65, G66, G67 .. Custom macro command
G98, G99 Feed per minute, feed per revolution

- **Circular interpolation in the polar coordinate plane**

The addresses for specifying the radius of an arc for circular interpolation (G02 or G03) in the polar coordinate interpolation plane depend on the first axis in the plane (linear axis).

- I and J in the Xp-Yp plane when the linear axis is the X-axis or an axis parallel to the X-axis.
- J and K in the Yp-Zp plane when the linear axis is the Y-axis or an axis parallel to the Y-axis.
- K and I in the Zp-Xp plane when the linear axis is the Z-axis or an axis parallel to the Z-axis.

The radius of an arc can be specified also with an R command.

- **Movement along axes not in the polar coordinate interpolation plane in the polar coordinate interpolation mode**

The tool moves along such axes normally, independent of polar coordinate interpolation.

- **Current position display in the polar coordinate interpolation mode**

Actual coordinates are displayed. However, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane (Cartesian coordinates).

- **Maximum cutting feedrate in polar coordinate interpolation mode**

In polar coordinate interpolation mode, the maximum cutting feedrate can be changed from that specified with parameter No. 527, used in most circumstances, to that specified with special parameter No. 663.

Restrictions

- **Coordinate system for the polar coordinate interpolation**
- **Tool nose radius compensation command**
- **Program restart**
- **Cutting feedrate for the rotation axis**

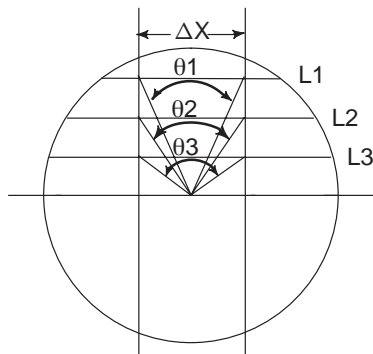
Before G112 is specified, a workpiece coordinate system) where the center of the rotary axis is the origin of the coordinate system must be set. In the G112 mode, the coordinate system must not be changed (G92, G52, G53, relative coordinate reset, G54 through G59, etc.).

The polar coordinate interpolation mode cannot be started or terminated (G112 or G113) in the tool nose radius compensation mode (G41 or G42). G112 or G113 must be specified in the tool nose radius compensation canceled mode (G40).

For a block in the G112 mode, the program cannot be restarted.

Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotation axis (C-axis) and the linear axis (X-axis). When the tool moves closer to the center of the workpiece, the C-axis component of the feedrate becomes larger and may exceed the maximum cutting feedrate for the C-axis (set in parameter No. 527), causing an alarm (see the figure below). To prevent the C-axis component from exceeding the maximum cutting feedrate for the C-axis, reduce the feedrate specified with address F or create a program so that the tool (center of the tool when tool nose radius compensation is applied) does not move close to the center of the workpiece.

WARNING



Consider lines L1, L2, and L3. ΔX is the distance the tool moves per time unit at the feedrate specified with address F in the Cartesian coordinate system. As the tool moves from L1 to L2 to L3, the angle at which the tool moves per time unit corresponding to ΔX in the Cartesian coordinate system increases from θ_1 to θ_2 to θ_3 .

In other words, the C-axis component of the feedrate becomes larger as the tool moves closer to the center of the workpiece. The C component of the feedrate may exceed the maximum cutting feedrate for the C-axis because the tool movement in the Cartesian coordinate system has been converted to the tool movement for the C-axis and the X-axis.

L :Distance (in mm) between the tool center and workpiece center when the tool center is the nearest to the workpiece center

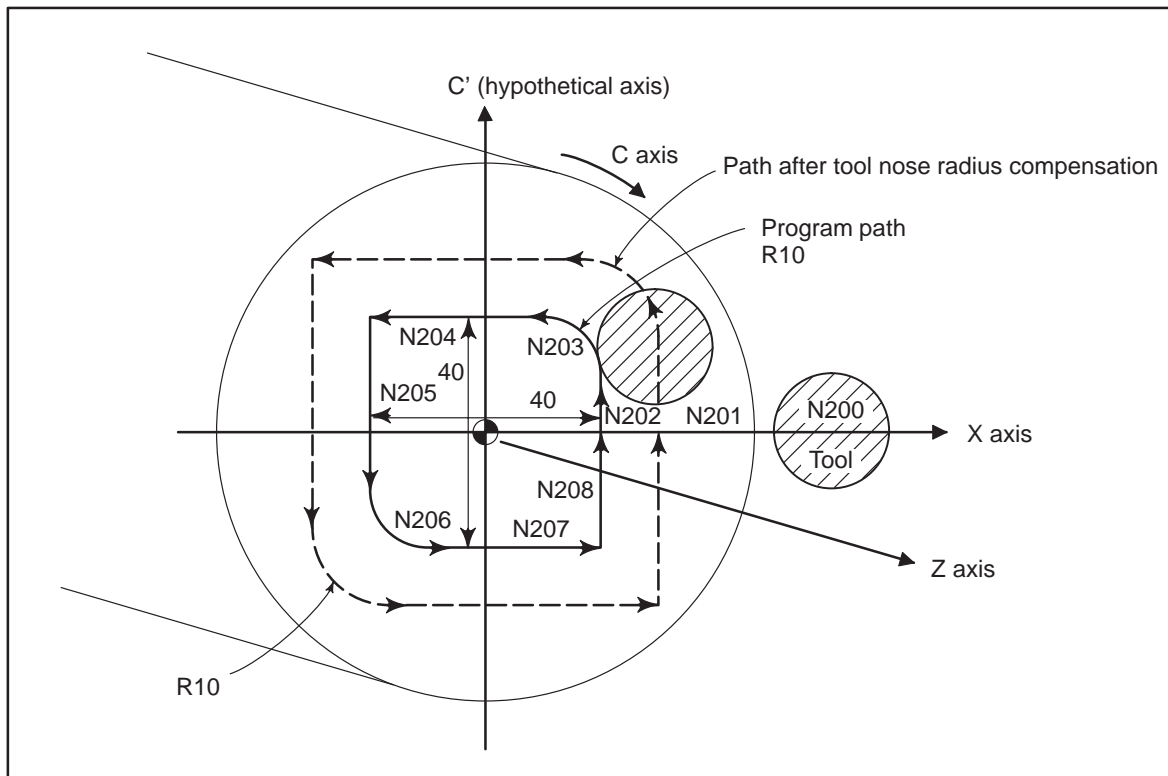
R :Maximum cutting feedrate (deg/min) of the C axis

Then, a speed specifiable with address F in polar coordinate interpolation can be given by the formula below. Specify a speed allowed by the formula. The formula provides a theoretical value; in practice, a value slightly smaller than a theoretical value may need to be used due to a calculation error.

$$F < L \times R \times \frac{\pi}{180} \text{ (mm/min)}$$

- **Clamping the feedrate about the rotation axis**

The feedrate about the rotation axis can be clamped by setting bit 2 of parameter No. 399 to 1, such that the value specified for the cutting feedrate is not exceeded. The tool can thus pass the proximity of the origin. The cutting feedrate may, however, vary slightly in the proximity of the zero point.

Examples**Example of Polar Coordinate Interpolation Program Based on X Axis (Linear Axis) and C Axis (Rotary Axis)**

X axis is by diameter programming, C axis is by radius programming.

O0001 ;

⋮

N010 T0101

⋮

N0100 G00 X120.0 C0 Z _ ; Positioning to start position

N0200 G12.1 ; Start of polar coordinate interpolation

N0201 G42 G01 X40.0 F _ ;

N0202 C10.0 ;

N0203 G03 X20.0 C20.0 R10.0 ;

N0204 G01 X-40.0 ;

N0205 C-10.0 ;

N0206 G03 X-20.0 C-20.0 I10.0 J0 ;

N0207 G01 X40.0 ;

N0208 C0 ;

N0209 G40 X120.0 ;

N0210 G13.1 ;

N0300 Z _ ;

N0400 X _ C _ ;

⋮

N0900M30 ;

Geometry program

(program based on cartesian coordinates on
X-C plane)

Cancellation of polar coordinate interpolation

4.5 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

⋮

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 Nos. 279 and 280 to specify whether the rotation axis is the X-, Y-, or Z-axis. 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 4.3.

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 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 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 (360°)

$$\left[\begin{array}{l} R \\ \left[\right] \end{array} \right. \begin{array}{l} \text{ : Workpiece radius} \\ \text{ : Rounded to the least input increment} \end{array}$$

Restrictions

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

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

- **Circular interpolation and tool nose radius compensation**

If the cylindrical interpolation mode is started when tool nose radius 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, G30) cannot be specified. Before positioning can be specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G107) cannot be performed in the positioning mode (G00).

- **Coordinate system setting**

In the cylindrical interpolation mode, a workpiece coordinate system 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.

- **Canned cycle for drilling during cylindrical interpolation mode**

Canned cycles for drilling, G81 to G89, cannot be specified during cylindrical interpolation mode.

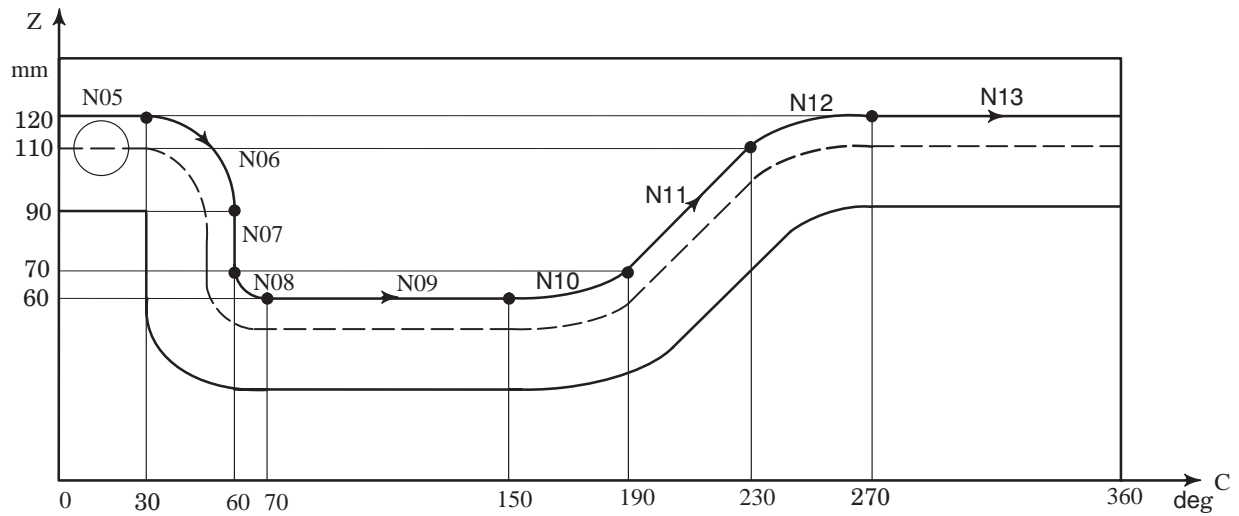
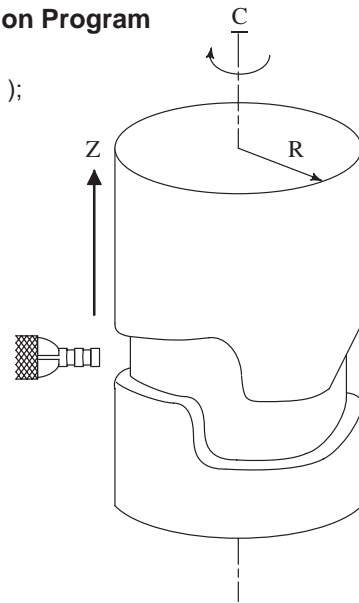
Examples

Example of a Cylindrical Interpolation Program

```

O0001 (CYLINDRICAL INTERPOLATION);
N01 G00 Z100.0 C0 T0101 ;
N02 G01 G18 W0 H0 ;
N03 G07.1 H57299 ;
N04 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 G07.1 C0 ;
N16 M30 ;

```



4.6 CONSTANT LEAD THREADING (G32)

Tapered screws and scroll threads in addition to equal lead straight threads can be cut by using a G32 command.

The spindle speed is read from the position coder on the spindle in real time and converted to a cutting feedrate for feed-per minute mode, which is used to move the tool.

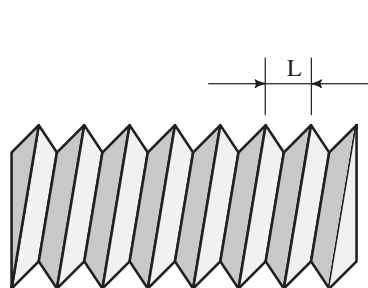


Fig.4.6 (a) Straight Thread

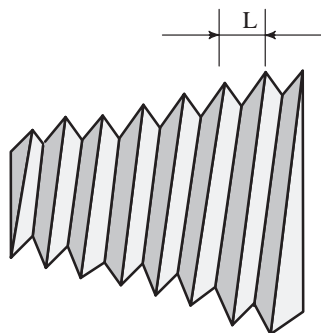


Fig.4.6 (b) Tapered Screw

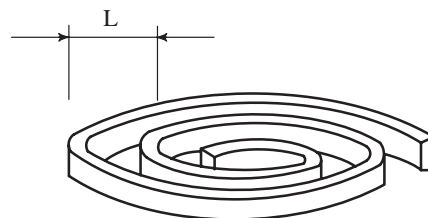


Fig.4.6 (c) Scroll Thread

Format

G32IP_F_;

IP_: End point

F_: Lead of the long axis
(always radius programming)

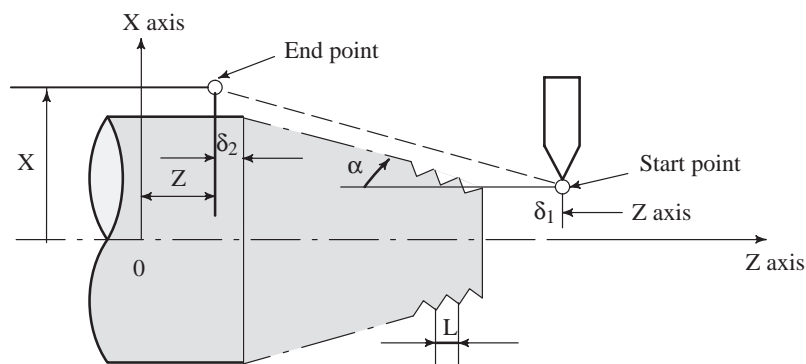


Fig. 4.6 (d) Example of Thread Cutting

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.

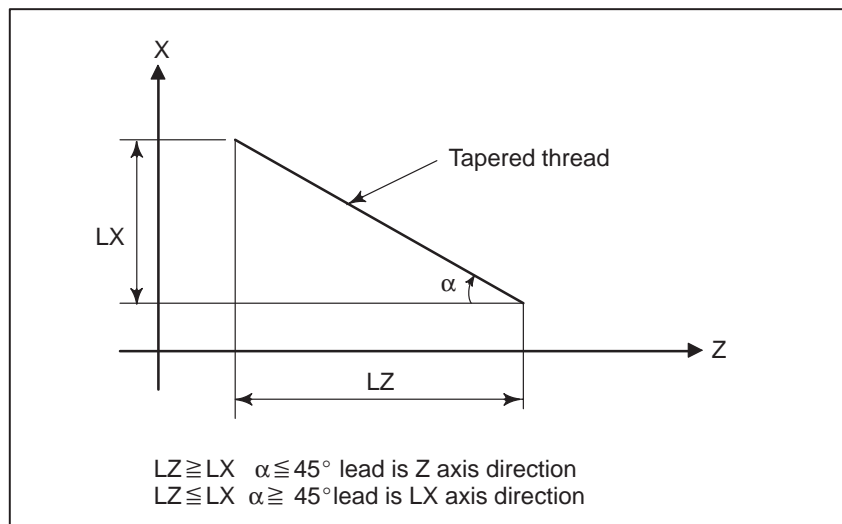


Fig. 4.6 (e) LZ and LX of a Tapered Thread

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.

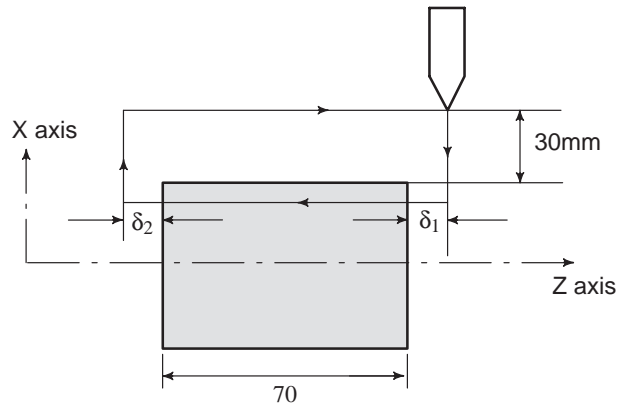
Table 4.6 (a) lists the ranges for specifying the thread lead.

Table 4.6 Ranges of lead size that can be specified

	Least command increment
mm input	0.0001 to 500.0000mm
Inch input	0.000001 to 9.999999inch

Explanations

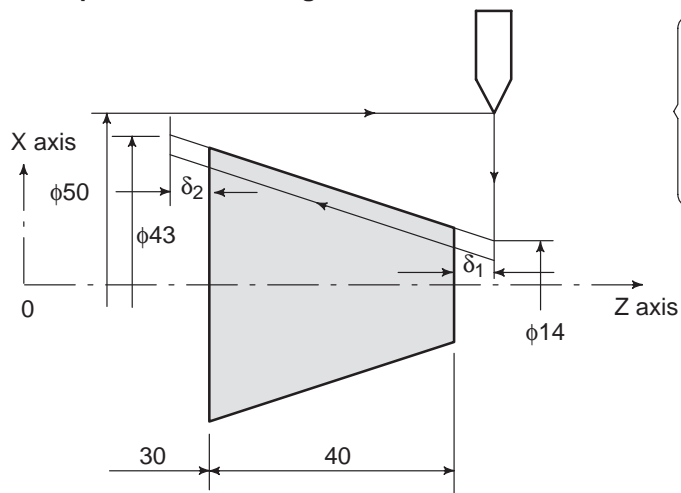
1. Straight thread cutting



The following values are used in programming :
 Thread lead :4mm
 $\delta_1=3\text{mm}$
 $\delta_2=1.5\text{mm}$
 Depth of cut :1mm (cut twice)
 (Metric input, Diameter programming)

```
G00 U-62.0 ;
G32 W-74.5 F4.0 ;
G00 U62.0 ;
W74.5 ;
U-64.0 ;
(For the second cut, cut 1mm more)
G32 W-74.5 ;
G00 U64.0 ;
W74.5 ;
```

2. Tapered thread cutting



The following values are used in programming :
 Thread lead : 3.5mm in the direction of the Z axis
 $\delta_1=2\text{mm}$
 $\delta_2=1\text{mm}$
 Cutting depth in the X axis direction is 1mm
 (Cut twice)
 (Metric input, Diameter programming)

```
G00 X 12.0 Z72.0 ;
G32 X 41.0 Z29.0 F3.5 ;
G00 X 50.0 ;
Z 72.0 ;
X 10.0 ;
(Cut 1mm more for the second cut)
G32 X 39.0 Z29.0 ;
G00 X 50.0 ;
Z 72.0 ;
```

WARNING

- 1 Feedrate override is effective (fixed at 100%) during thread cutting.
- 2 it is very dangerous to stop feeding the thread cutter. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while thread cutting. If the feed hold button is pressed during thread cutting, the tool will stop after a block not specifying thread cutting is executed as if the SINGLE BLOCK button were pushed. However, the feed hold lamp (SPL lamp) lights when the FEED HOLD button on the machine control panel is pushed. Then, when the tool stops, the lamp is turned off (Single Block stop status).
- 3 When the FEED HOLD button is again pushed during the first block not specifying thread cutting just after thread cutting block or when it has been continuously pushed, the tool stops at the block not specifying thread cutting.
- 4 When thread cutting is executed in the single block status, the tool stops after execution of the first block not specifying thread cutting.
- 5 When the mode was changed from automatic operation to manual operation during thread cutting, the tool stops at the first block not specifying thread cutting as when the feed hold button is pushed as mentioned in Note 3. However, when the mode is changed from one automatic operation mode to another, the tool stops after execution of the block not specifying thread cutting as for the single block mode in Note 4.
- 6 When the previous block was a thread cutting block, cutting will start immediately without waiting for detection of the 1-turn signal even if the present block is a thread cutting block.
However, the lead at the point where the blocks join is incorrect. To obtain the correct lead, the continuous threading option is required.
 G32Z _ F_ ;
 Z _; (A 1-turn signal is not detected before
 this block.)
 G32 ; (Regarded as threading block.)
 Z_ F_ ; (One turn signal is also not detected.)
- 7 Because the constant surface speed control is effective during scroll thread or tapered screw cutting and the spindle speed changes, the correct thread lead may not be cut. Therefore, do not use the constant surface speed control during thread cutting.
- 8 A movement block preceding the thread cutting block must not specify chamfering or corner R.
- 9 A thread cutting block must not specifying chamfering or corner R.
- 10 The spindle speed override is effective in thread cutting mode.
- 11 Thread cycle retract function is ineffective to G32.

4.7 VARIABLE-LEAD THREAD CUTTING (G34)

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

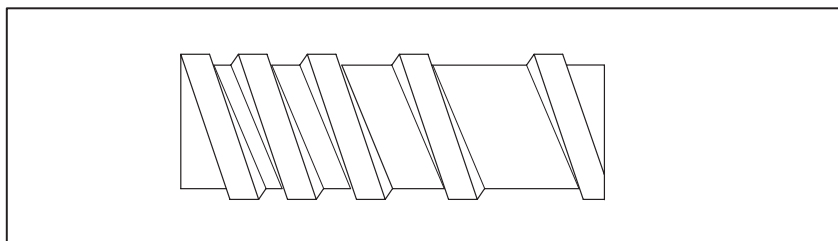


Fig. 4.7 (a) Variable-lead screw

Format

G34 IP_F_K_;
IP : End point
F : Lead in longitudinal axis direction at the start point
K : Increment and decrement of lead per spindle revolution

Explanations

Address other than K are the same as in straight/taper thread cutting with G32.

Table 4.7 (a) lists a range of values that can be specified as K.

Table 4.7 (a) Range of valid K values

	Valid range data
Metric input	± 0.0001 to ± 500.0000 mm/rev
Inch input	± 0.000001 to ± 9.999999 inch/rev

Alarm (No. 14) is produced, for example, when K such that the value in Table 4.7 (a) is exceeded is directed, the maximum value of lead is exceeded as a result of increase or decrease by K or the lead has a negative value.

WARNING

The "Thread Cutting Cycle Retract" is not effective for G34.

4.8 CONTINUOUS THREAD CUTTING

This function for continuous thread cutting is such that fractional pulses output to a joint between move blocks are overlapped with the next move for pulse processing and output (block overlap) .

Therefore, discontinuous machining sections caused by the interruption of move during continuously block machining are eliminated, thus making it possible to continuously direct the block for thread cutting instructions.

Explanations

Since the system is controlled in such a manner that the synchronism with the spindle does not deviate in the joint between blocks wherever possible, it is possible to performed special thread cutting operation in which the lead and shape change midway.

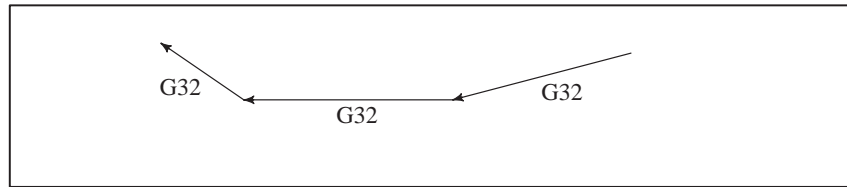


Fig. 4.8 (a) Continuous Thread Cutting

Even when the same section is repeated for thread cutting while changing the depth of cut, this system allows a correct machining without impairing the threads.

NOTE

- 1 Block overlap is effective even for G01 command, producing a more excellent finishing surface.
- 2 When extreme micro blocks continue, no block overlap may function.

4.9 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 #5068, as follows:

#5061 X axis coordinate value

#5062 Z axis coordinate value

#5063 3rd axis coordinate value

#5064 4th axis coordinate value

WARNING

Disable feedrate override, dry run, and automatic acceleration /deceleration when the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input. By setting bit 3 of parameter No.015, however, feedrate override, dry run, and automatic acceleration/deceleration can be enabled during movement based on the skip function. These functions are enabled when the feedrate per rotation is specified.

NOTE

- 1 If G31 command is issued while tool nose radius compensation 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.
- 2 For the high-speed skip option, executing G31 during feed-per-rotation mode causes P/S alarm 211 to be generated.

Examples

- The next block to G31 is an incremental command

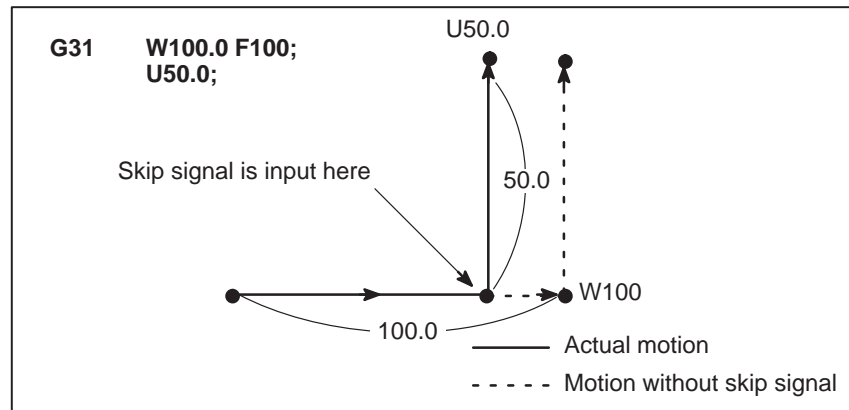


Fig. 4.9 (a) The next block is an incremental command

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

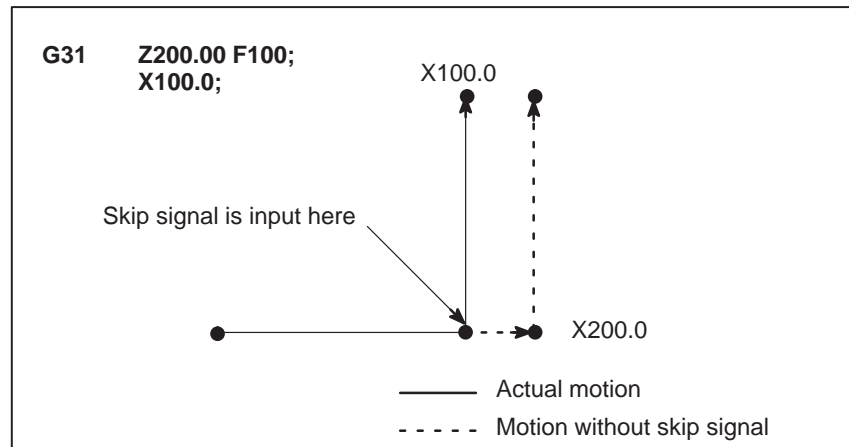


Fig. 4.9 (b) The next block is an absolute command for 1 axis

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

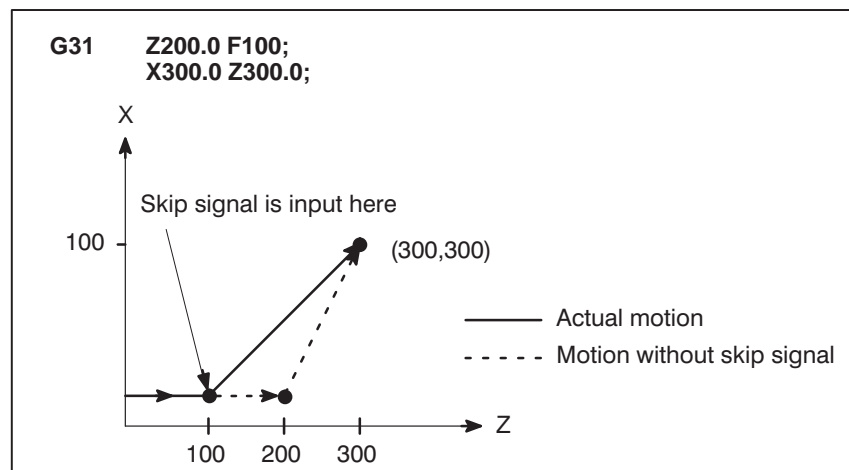


Fig. 4.9 (c) The next block is an absolute command for 2 axes

4.10

MULTI-STEP (0-GCC, 00-GCC, 0-GCD/II)

In a block specifying P1 to P4 after G31, the multi-step skip function stores coordinates in a custom macro variable when a skip signal (4-point) is turned on. In a block having Q1 to Q4 specified after G04, a dwell operation can be skipped by turning on the skip signal.

Parameters No. 033 and 034 can be used to select a 4-point skip signal.

One skip signal can be set to match multiple Pn or Qn (n=1,2,3,4) as well as to match a Pn or Qn on a one-to-one basis. Parameters No. 035 #0 to #3 can be used for dwell.

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

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

CAUTION

Dwell is not skipped when Qn is not specified and parameters No. 035#0-#3 are not set.

4.11

SKIP FUNCTION BY TORQUE LIMIT ARRIVAL SIGNAL

This function skips the remaining motion of the block by the torque limit arrival signal from the servo motor. It controls such functions as pressing a workpiece to the chuck or transferring a workpiece to another axis.

Format

G31 P99 Motion Command F feedrate ;

Explanations

When commanding the above with limiting the torque of the servo motor, the remaining motion of the block is skipped by asserting the torque limit arrival signal from the motor, and the next block is executed. The skip position is memorized in the ordinary system variables. G31 P99 is valid only on the commanded block.

Generally, the servo lag is not zero after the skip by the torque limit arrival signal. There are two methods to maintain the servo lag. One is leaving the lag as it is.

Another is retrieving the lag to the actual position and making it almost zero. In the latter case, the retrieved position is memorized as the skip position. The parameter (No.389#4) selects either.

● Program Example

G00 Z500.0 ;	Positioning to the start point
M□□ ;	Limiting the torque of servo motor
G31 P99 W22.0 F100.0 ;	Commanding the skip function
G01 Z500.0 ;	Re-positioning to the start point
M○○○ ;	Releasing the torque of servo motor

WARNING

- 1 The torque limit for the servo motor should be commanded before G31 P99. The torque limit can be achieved by using PMC window function. At least, one torque limit arrival signal of the connected servo motors is turned on, the skip is carried out.
- 2 The torque limit should be released at the point where the workpiece and the axis is not contacting each other. When resetting the G31 P99, it is interrupted but the torque limit is not released automatically.

NOTE

- 1 The ordinary skip signal is also active in G31 P99.
- 2 The in-position is not checked at the end of G31 P99.
- 3 The mirror image should be cancelled before commanding G31 P99.
- 4 The nose-R compensation should be cancelled before commanding G31 P99.
- 5 G31 P99 should not be commanded successively.
- 6 Dry-run, feedrate and automatic acc./dec. should be ignored during G31 P99. (Parameter No.015#5=0)
- 7 The servo excess error during stop at the axis is not checked while the torque limit arrival signal turns on.

5

FEED FUNCTIONS



5.1 GENERAL

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

• Feed functions

1. Rapid traverse

When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameters 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)).

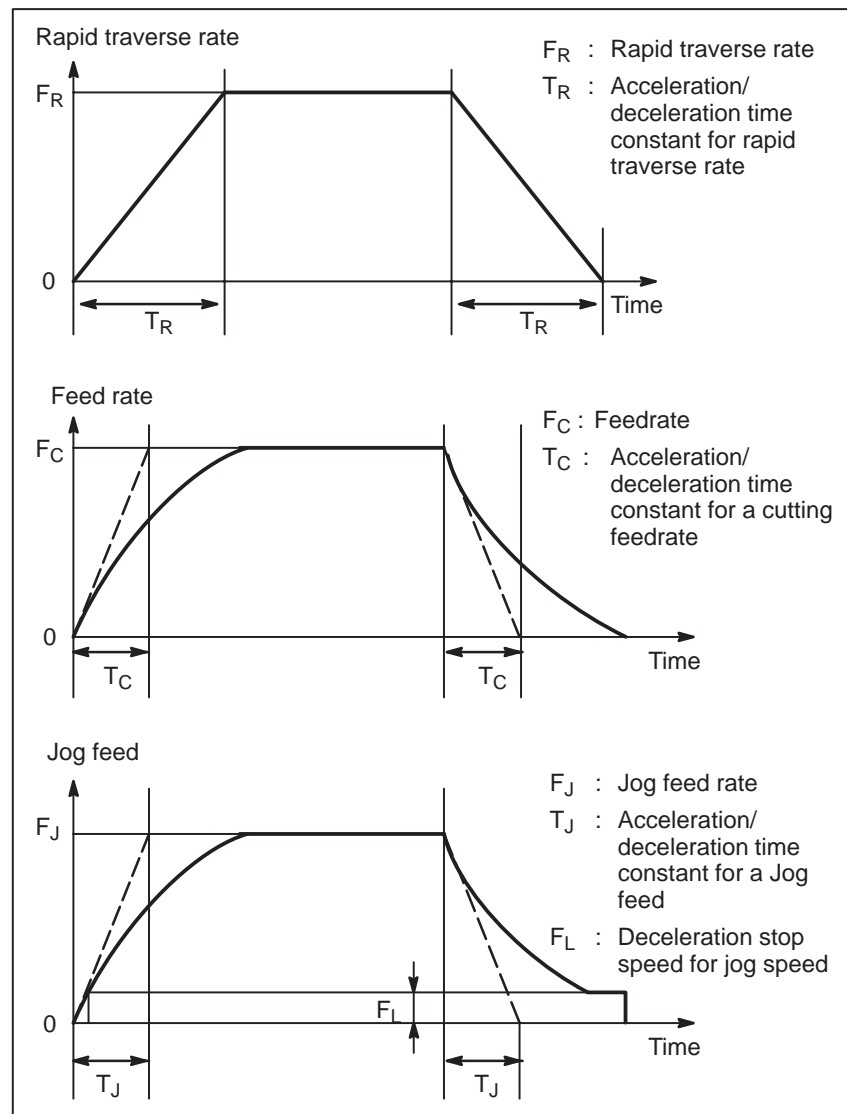


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- **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)).

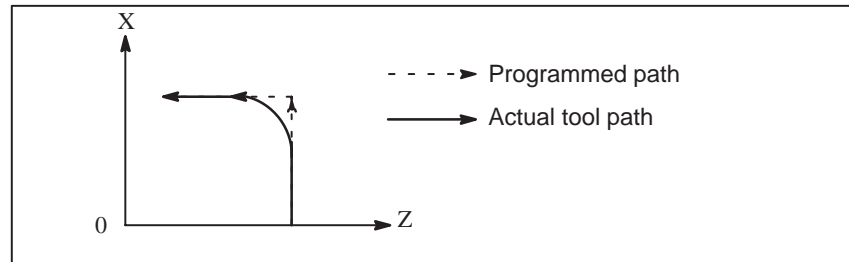


Fig. 5.1 (b) Example of Tool Path between Two Blocks

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

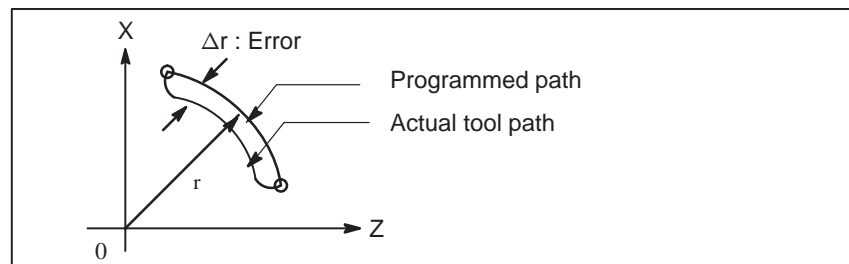


Fig. 5.1 (c) Example of Radial Error in Circular Interpolation

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

G00 IP_ ;

**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).

By setting bit 5 of parameter No. 020, however, the in-position check function can be disabled in each positioning block.

A rapid traverse rate is set for each axis by parameters 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 by parameter No. 533.

For detailed information, refer to the appropriate manual of the machine tool builder.

Command value range of the rapid traverse

	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.

Two modes of specification are available:

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

Format

Feed per minute

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

Feed per revolution

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

Explanations

- **Tangential speed constant control**

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

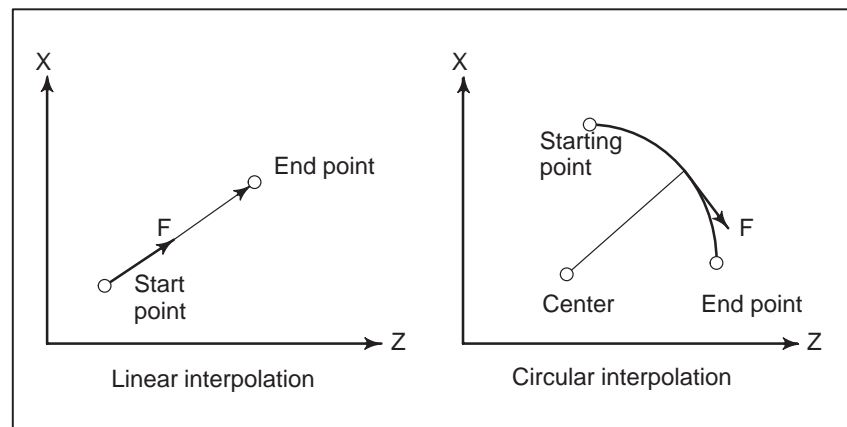


Fig. 5.3 (a) Tangential feedrate (F)

- **Feed per minute (G98)**

After specifying G98, the amount of feed of the tool per minute is to be directly specified by setting a number after F. G98 is a modal code. Once a G98 is specified, it is valid until G99 (feed per revolution) is specified. At power-on, the feed per revolution mode is set.

An override from 0% to 150% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

- **Feedrate command value range for feed per minute**

	Increment system	
	IS-B	IS-C
Metric output	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 output	0.01 to 4,000.00 inch/min 0.01 to 6,000.00 deg/min	0.01 to 480.00 inch/min 0.01 to 600.00 deg/min

- **Feedrate decimal point input for feed per minute**

The command value ranges indicated in the above table are also applicable when a decimal point is used in the specification of a feedrate for feed per minute. By setting bit 5 of parameter No. 077, however, the command value ranges for those cases where a decimal point is entered can be modified as indicated below. With 0-GCC and 00-GCC, the following command value ranges are applicable even if the parameter is not set. When a feedrate that falls outside the command value ranges shown below is required, specify the feedrate without entering a decimal point.

	Increment system	
	IS-B	IS-C
Metric output	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 output	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

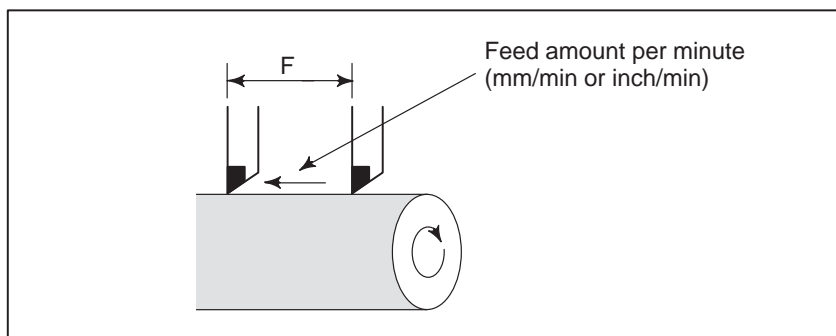


Fig. 5.3 (b) Feed per minute

WARNING

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

- **Feed per revolution (G99)**

After specifying G99, the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G99 is a modal code. Once a G99 is specified, it is valid until G98 (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.

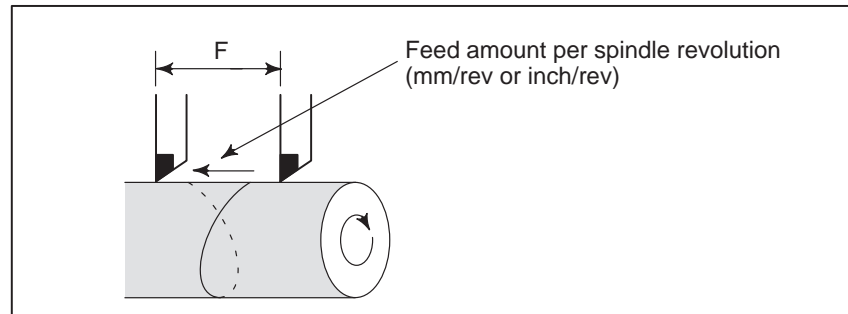


Fig. 5.3 (c) Feed per revolution

CAUTION

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

- **Feedrate command value range for feed per rotation**

	Increment system	
	IS-B	IS-C
Metric input	0.0001 to 500.0000 mm/rev	0.0001 to 500.0000 mm/rev
Inch input	0.000001 to 9.999999 inch/rev	0.000001 to 9.999999 inch/rev

The feedrate command value ranges for feed per rotation indicated in the table above indicate the values that can be specified; note that the actual feedrate is clamped to a maximum cutting feedrate.

- **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

DWELL (G04)

Format

Dwell G04 X_ ; or G04 U_ ; or G04 P_ ;
 X_ : Specify a time (decimal point permitted)
 U_ : 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.

Table 5.4 (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 5.4 (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

NOTE

Dwell and Tool offset can not be commanded in the same block.

5.5 DWELL BY TURNING TIMES OF SPINDLE

This function specifies the dwell interval by turning times of the spindle instead of the time interval.

This function is enabled by setting bit 0 of parameter No. 395 accordingly.

Format

```
(G99) G04 [ P _____  
             X _____  
             U _____ ] ;
```

Explanations

The dwell command by designating turning times of the spindle is possible when commanding G04 during the feed per revolution mode (G99). Turning times of the spindle is designated by P-code, X-code or U-code. The dwell is performed while the spindle rotates the designated times. The setting unit and range of turning times are as follows. The decimal point can be used in X-code and U-code.

	Unit	Range
Standard	0.001 rev	0 to 99999.999 rev
Input unit 1/10	0.0001 rev	0 to 9999.9999 rev

Example

When commanding G99 G04 U2.5;, the dwell is performed while the spindle rotates 2.5 times.

NOTE

- 1 In spite of the parameter setting, the dwell command during the feed per minute is regarded as the ordinary dwell by the time interval.
- 2 When setting the parameter (No.062#3) to "1" to expand the maximum spindle speed, This function is not used.

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. However, the first reference position matches the machine zero point.

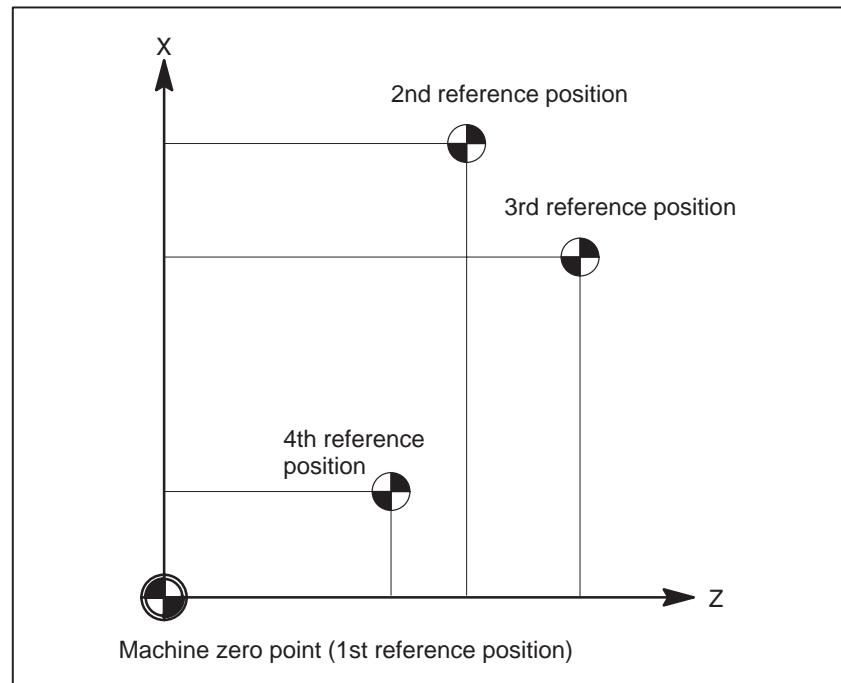


Fig. 6 (a) Machine zero point and reference positions

- **Reference position return**

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

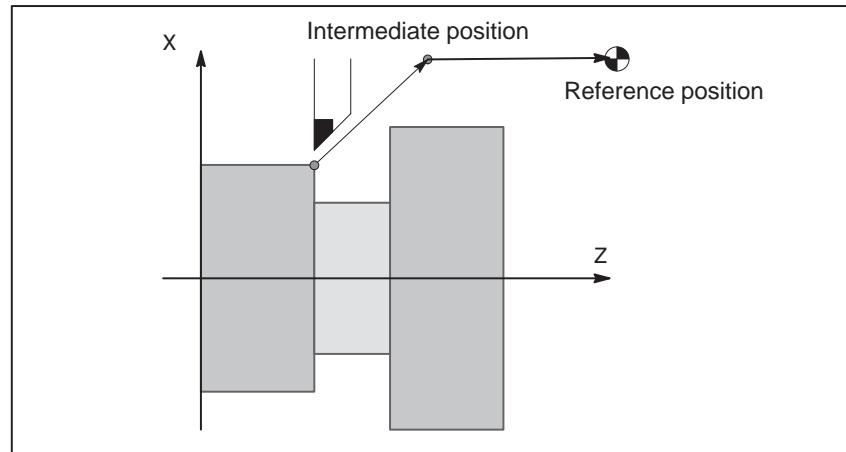


Fig. 6 (b) Reference position return

- **Reference position return check**

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

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)

- **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 tool nose radius compensation, and tool offset should be cancelled before executing this command.

- **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.

- **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 ZMIX (bit 0 to 3 of No. 003). 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 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 input unit. This is because the least input increment of the machine is smaller than its least command increment.

Reference

- **Manual reference position return**

See III-4.1.

7 COORDINATE SYSTEM

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

X_Z_

This command is referred to as a dimension word.

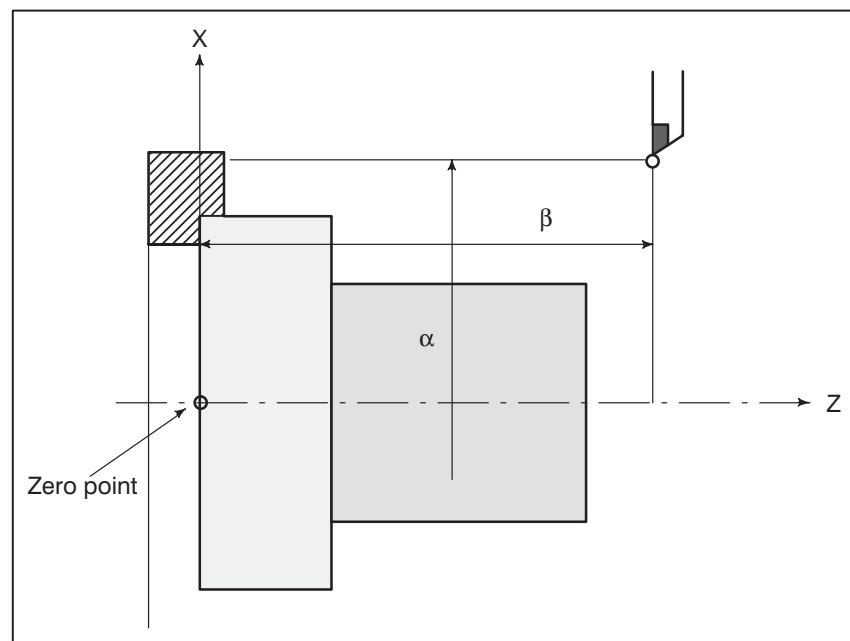


Fig. 7 Tool Position Specified by $X\alpha Z\beta$

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-4.1). 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 is valid, but the incremental command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

Restrictions

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

When the G53 command is specified, cancel the tool nose radius compensation 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 workpiece coordinate system option (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 G50

A workpiece coordinate system is set by specifying a value after G50 in the program.

(2) Automatic setting

If bit 7 of parameter No. 010 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed (see Part III-3.1.).

(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 enable the use of absolute programming, a workpiece coordinate system must be established using one of the methods described above.

Format

- Setting a workpiece coordinate system by G50

```
G50 IP_ ;
```

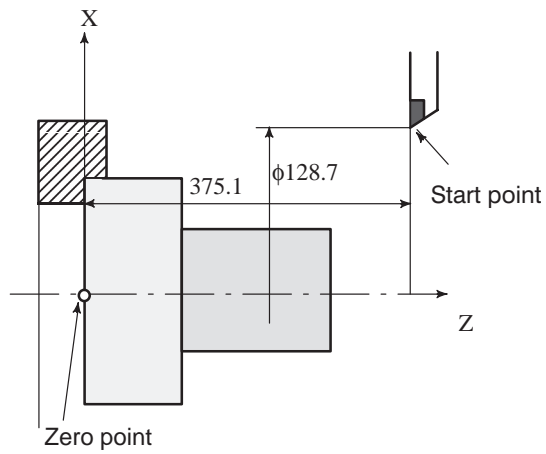
Explanations

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

Examples

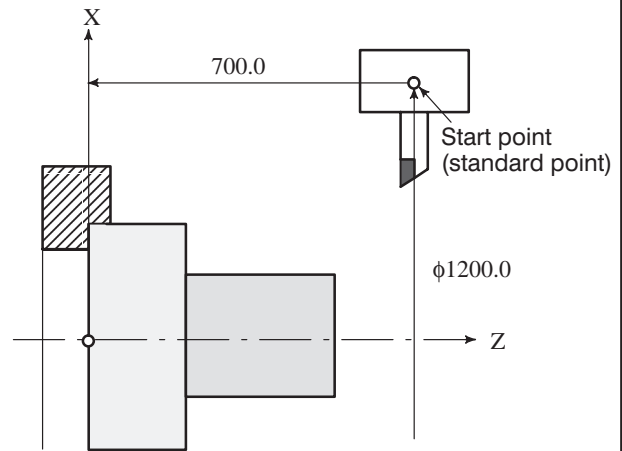
Example 1

Setting the coordinate system by the
G50X128.7Z375.1; command (Diameter designation)



Example 2

Base point
Setting the coordinate system by the
G50X1200.0Z700.0; command (Diameter designation)



7.2.2 Selecting a Workpiece Coordinate System

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

(1) Selecting a workpiece coordinate system set by G50 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

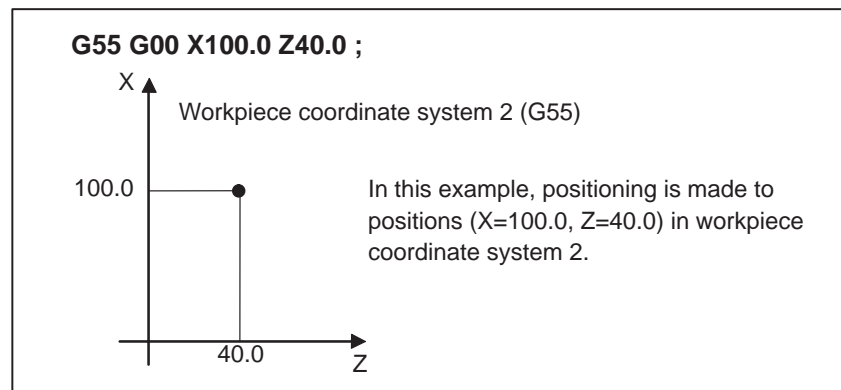


Fig. 7.2.2

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.

Three 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-11.4.6)
- (2) Programming by G10 or G50
- (3) Changing an external workpiece zero point offset value (refer to machine tool builder's manual)

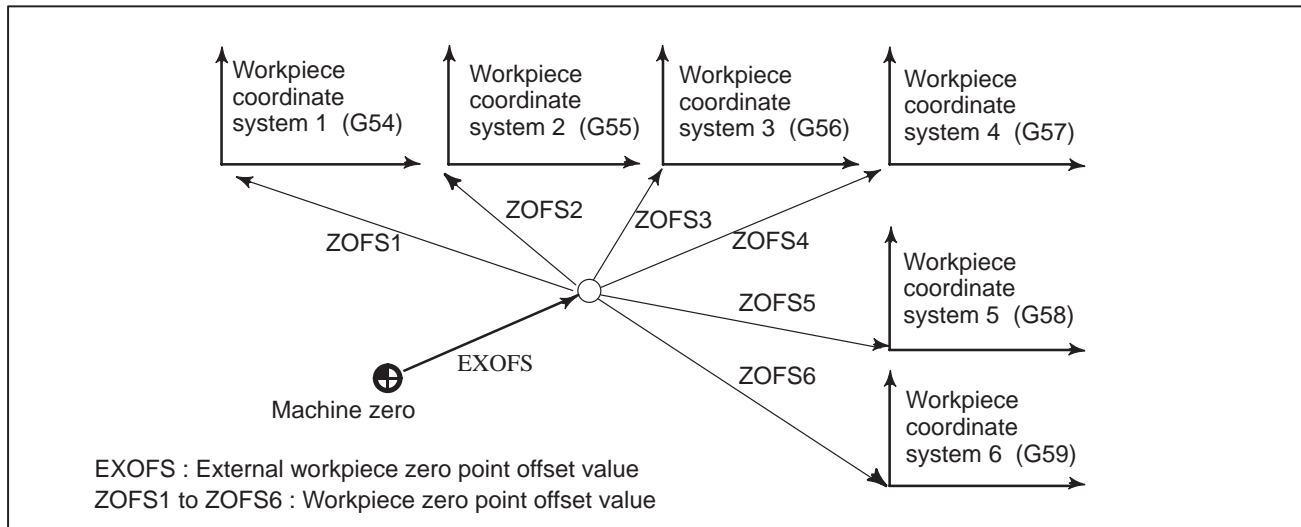


Fig. 7.2.3 Changing an external workpiece zero point offset value or workpiece zero point offset value

Format

• Changing by G10

G10 L2 Pp 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 G50

G50 IP _;

Explanations

• Changing by G10

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

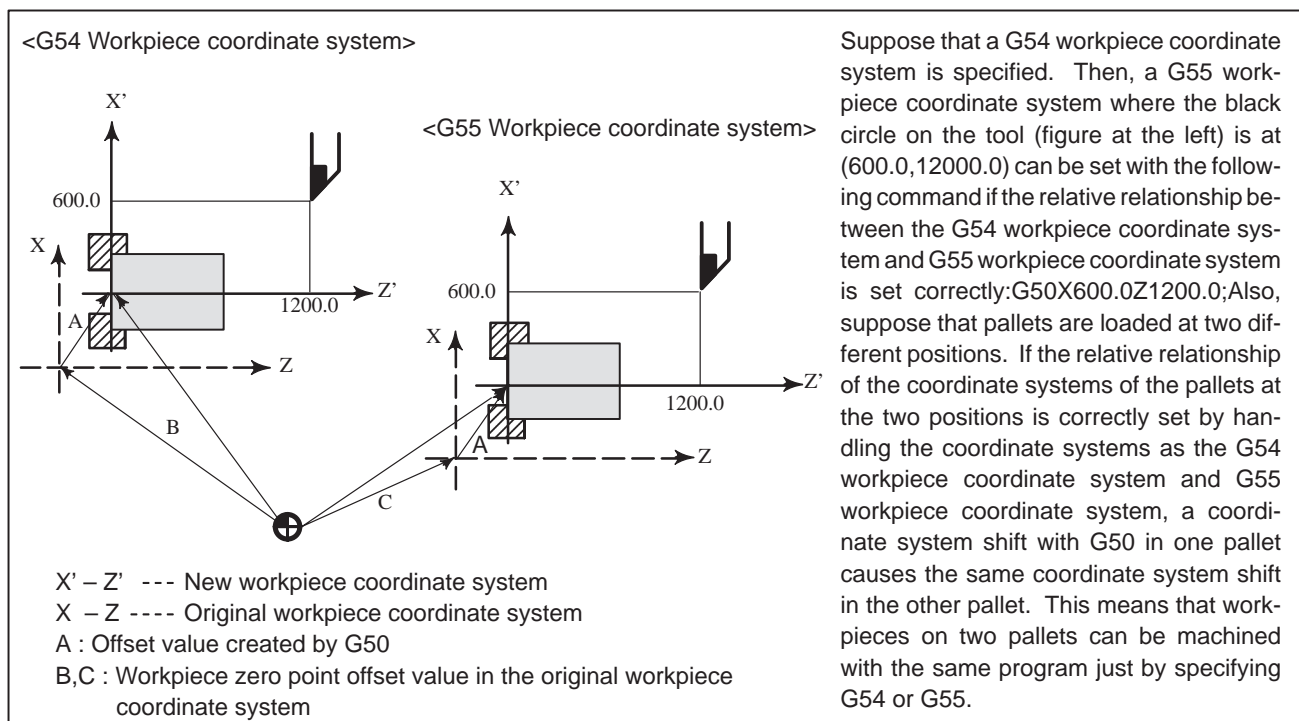
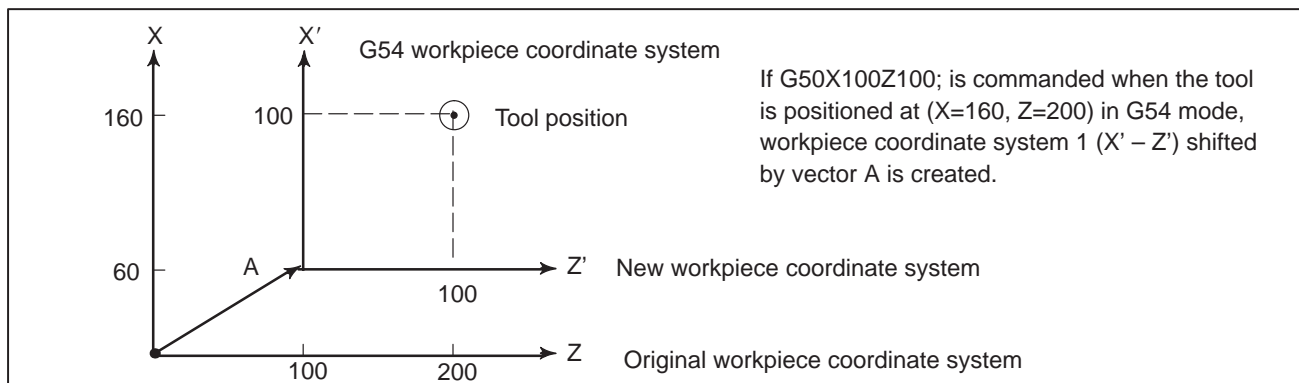
• Changing by G50

By specifying G50IP_;, 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_).

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

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.

Examples



7.2.4 Workpiece Coordinate System Shift

Explanations

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed work system, the set coordinate system can be shifted (see III-3.1).

Set the desired shift amount in the work coordinate system shift memory.

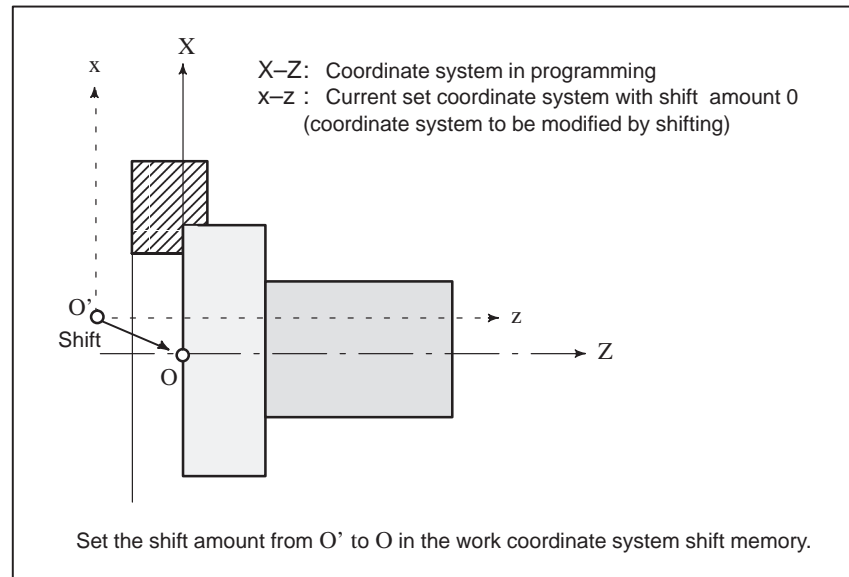


Fig. 7.2.4 Workpiece Coordinate System shift

See Section 11.1.1 of Part III for how to specify the distance the work coordinate system is shifted.

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

G52 IP _; Setting the local coordinate system

⋮

G52 IP 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.

Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the 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.

Workpiece coordinate system (G54 to G59) is optional function.

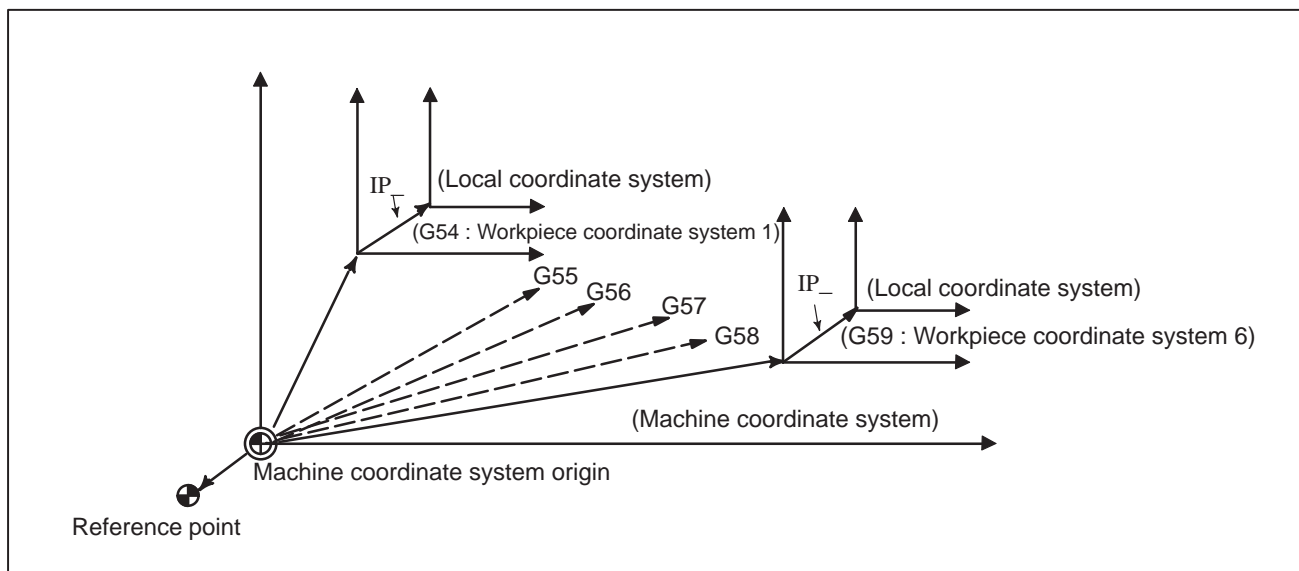


Fig. 7.3 Setting the local coordinate system

WARNING

- 1 The local coordinate system setting does not change the workpiece and machine coordinate systems.
- 2 When G50 is used to define a work coordinate system, if coordinates are not specified for all axes of a local coordinate system, the local coordinate system remains unchanged. If coordinates are specified for any axis of a local coordinate system, the local coordinate system is canceled.
- 3 G52 cancels the offset temporarily in tool nose radius compensation.
- 4 Command a move command immediately after the G52 block in the absolute mode.

7.4 PLANE SELECTION

Select the planes for circular interpolation and tool nose radius compensation 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.

Parameters No. 279 and 280 specifies whether 3rd axis or 4th axis is a basic axis (X-axis, Y-axis, or Z-axis) or an axis parallel to a basic axis. The plane is unchanged in the block in which G17, G18 or G19 is not commanded.

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

The movement instruction is irrelevant to the plane selection.

NOTE

Direct drawing dimension programming, chamfering, corner R, multiple repetitive canned cycle, and simple canned cycle are enabled only for the ZX plane.

Specifying these functions for other planes causes PS alarm No. 212 to be generated.

Examples

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

G17X_Y_ ; XY plane,

G17A_Y_ ; AY plane

G18X_Z_ ; ZX plane

X_Y_ ; Plane is unchanged (ZX plane)

G17 ; .. XY plane

G18 ; .. ZX plane

G17 A_ ; AY plane

G18Y_ ; ZX plane, Y axis moves regardless without any relation to the plane.

8

COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

8.2 INCH/METRIC CONVERSION (G20, G21)

8.3 DECIMAL POINT PROGRAMMING

8.4 DIAMETER AND RADIUS 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.

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

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

Format

- G code system A

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

- G code system B or C

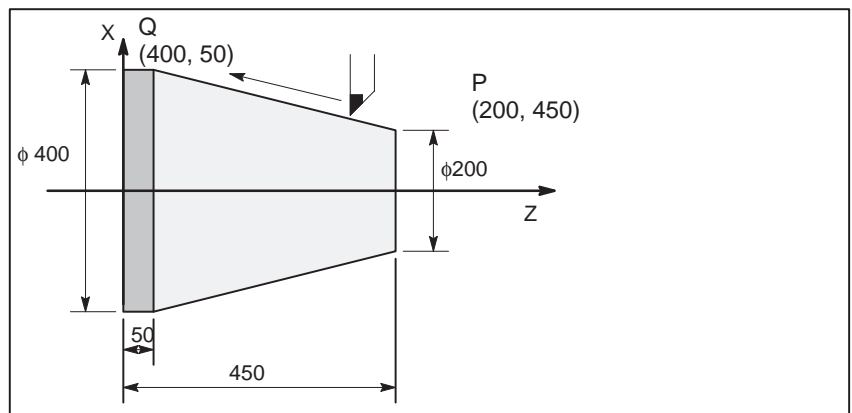
Absolute command G90 P ;

Incremental command G91 P ;

Examples

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

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



NOTE

- 1 Absolute and incremental commands can be used together in a block.
In the above example, the following command can be specified :
X400.0 W-400.0 ;
- 2 When both X and U or W and Z are used together in a block, the one specified later is effective.

8.2

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
- Workpiece 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 setting parameter (inch).

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, U, V, W, A, B, C, I, J, K, R, and F.

Explanations

There are two types of decimal point notation: calculator-type notation and standard notation.

When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters, inches or seconds. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal notation by using the (bit 7 of parameter 015). 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.

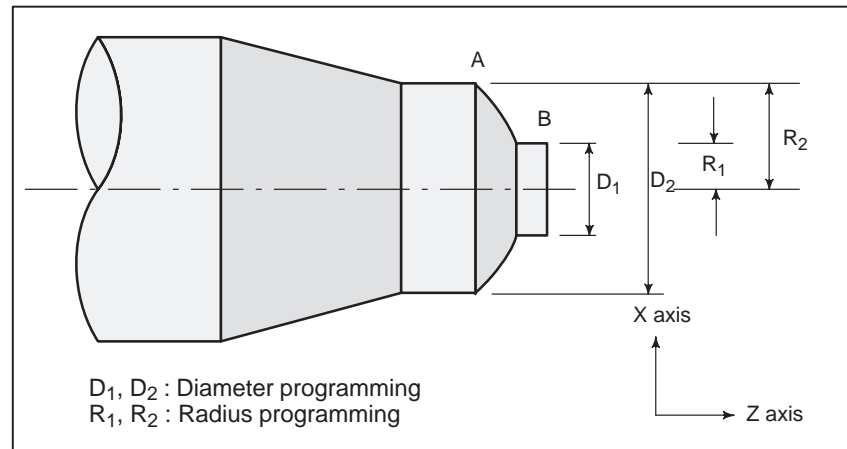
X123456.7; If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an alarm occurs.

8.4 DIAMETER AND RADIUS PROGRAMMING

Since the work cross section is usually circular in CNC lathe control programming, dimensions of X axis can be specified in two ways :

Diameter and Radius

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



Explanations

- **Notes on diameter programming/radius programming for each command**

Radius programming or diameter programming can be specified by parameter (No.019#2). When using diameter programming, note the conditions listed in the table 8.4(a).

Table 8.4 (a) Notes on specifying diameter value

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D_2 minus D_1 for tool path B to A.
Coordinate system setting (G50)	Specifies a coordinate value with a diameter value
Component of tool offset value	Parameter (No.001#4) determines either diameter or radius value
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.
Display of axis position	Displayed as diameter value

9

SPINDLE SPEED FUNCTION

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

In addition, the spindle can be rotated by a specified angle.

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)**
- 9.4 SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)**
- 9.5 SPINDLE POSITIONING**

9.1 SPECIFYING THE SPINDLE SPEED WITH A BINARY CODE

This spindle speed can be specified by address S followed by 2-digit code. A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the execution order when a move command and an S code command are in the same block.

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

The spindle speed can be specified directly by address S followed by a 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 S ;

↑Surface speed (m/min or feet/min)

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

- 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.

- Clamp of maximum spindle speed

G50 S_ ; The maximum spindle speed (rpm) follows S.

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 G97 command cancels the G96 mode. When constant surface speed control is applied, a spindle speed higher than the value specified in G50S_; (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.

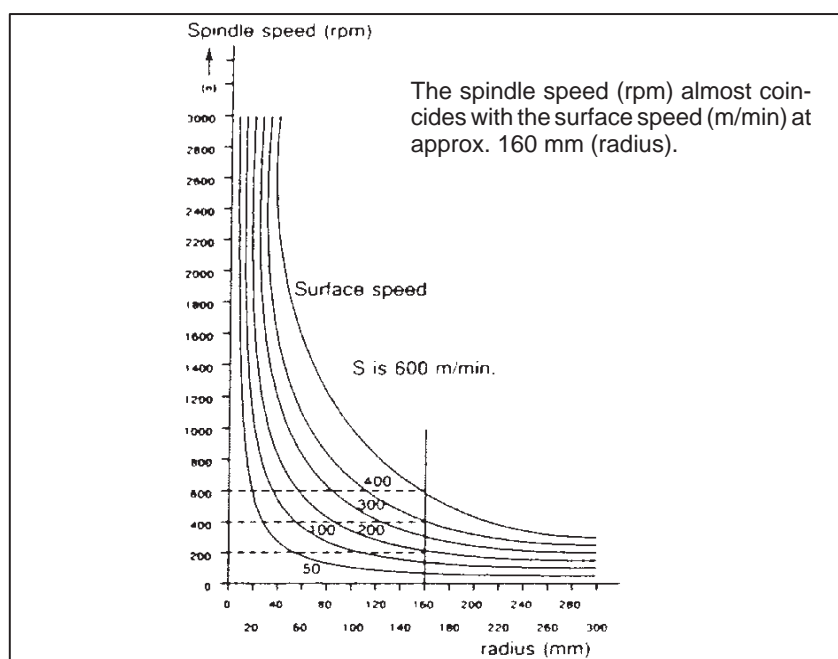


Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

- **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, Z axis, (axis to which the constant surface speed control applies) becomes zero.

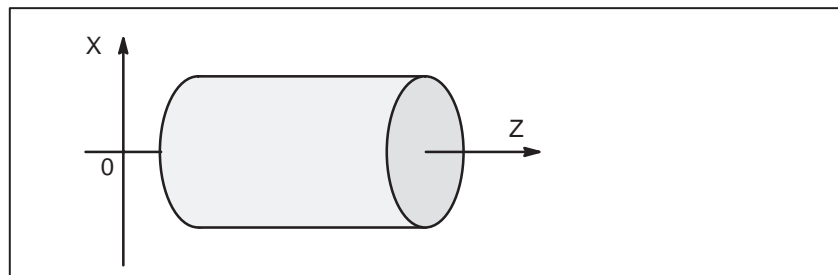
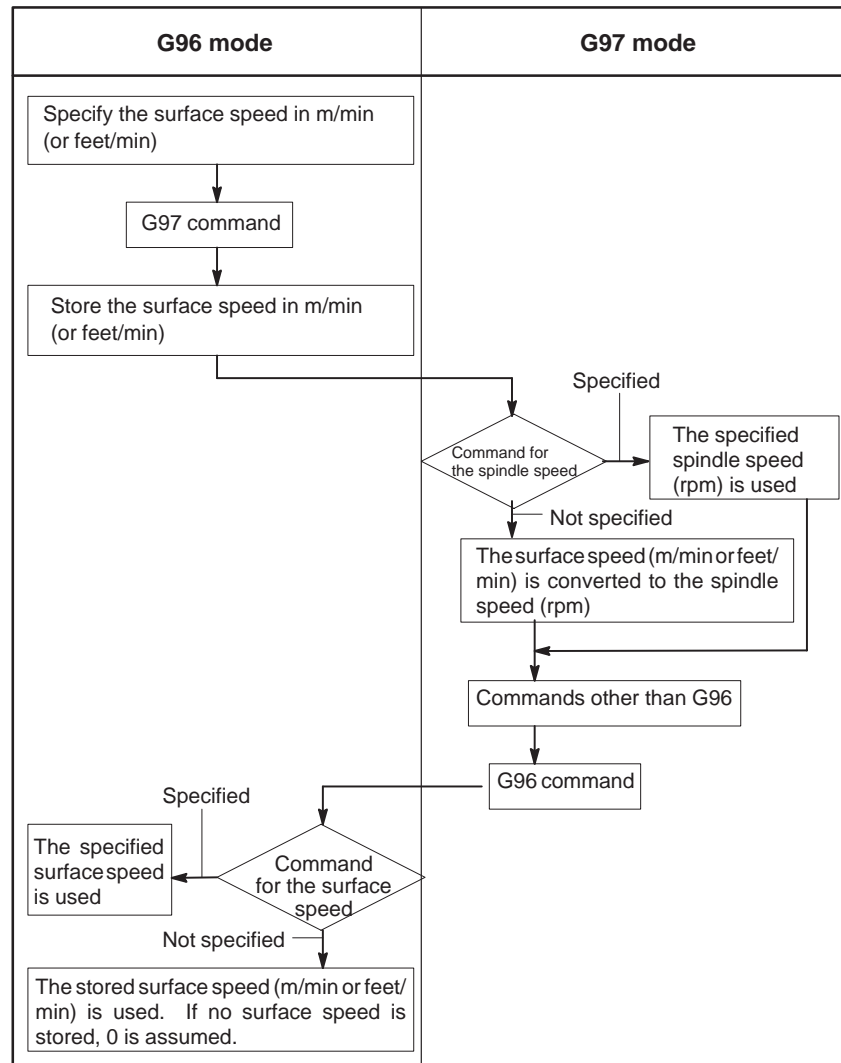


Fig. 9.3 (b) Example of the Workpiece Coordinate System for Constant Surface Speed Control

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



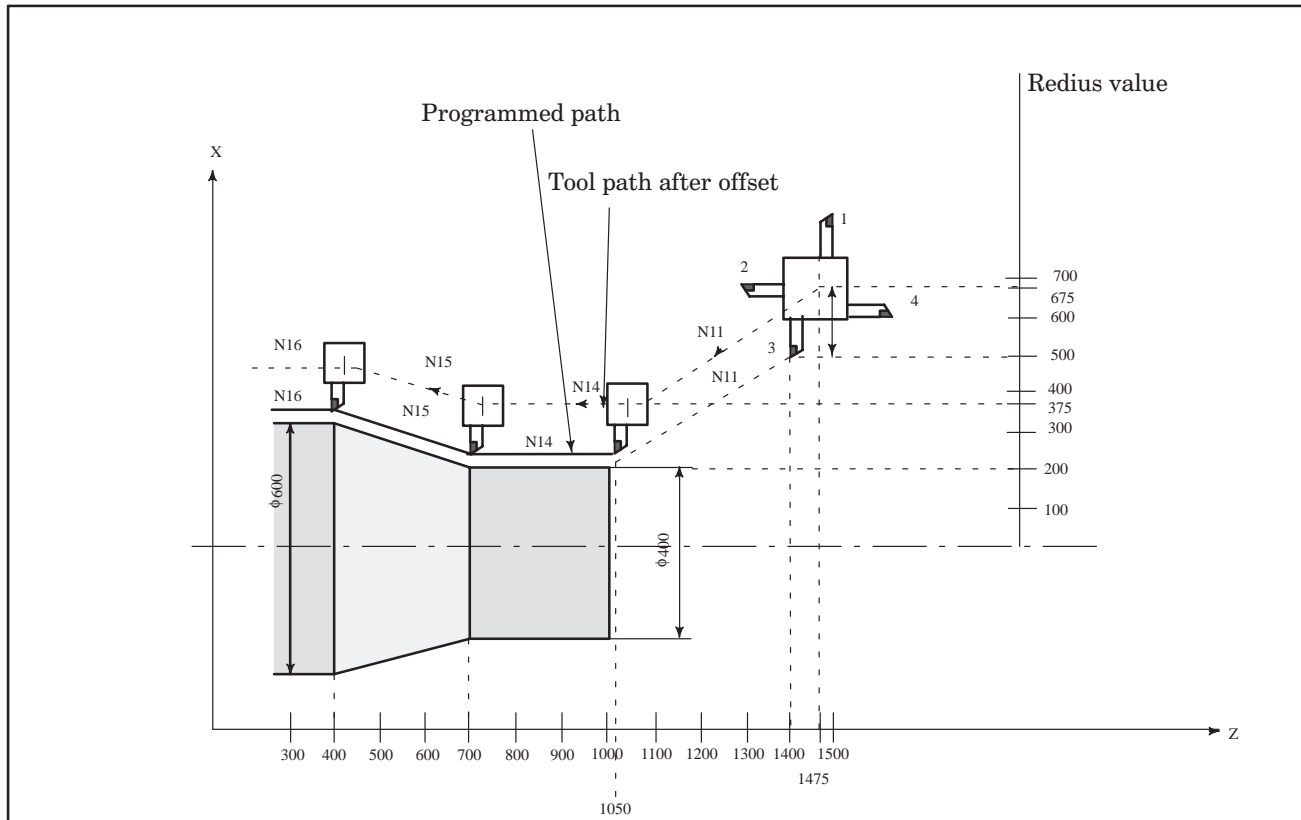
Restrictions

- **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.



Example

```

N8 G00 X1000.0Z1400.0 ;
N9 T33;
N11 X400.0Z1050.0;
N12 G50S3000 ; (Designation of max. spindle speed)
N13 G96S200 ;(Surface speed 200m/min)
N14 G01 Z 700.0F1.0 ;
N15 X600.0Z 400.0;
N16 Z ... ;

```

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

9.4

SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)

Format

With this function, an overheat alarm (No. 704) is raised when the spindle speed deviates from the specified speed due to machine conditions. This function is useful, for example, for preventing the seizure of the guide bushing.

G26 enables spindle speed fluctuation detection.

G25 disables spindle speed fluctuation detection.

G26 Pp Qq Rr ;	Spindle fluctuation detection on
G25 ;	Spindle fluctuation detection off

p : Time (in ms) from the issue of a new spindle rotation command (S command) to the start of checking whether the actual spindle speed is so fast that an overheat can occur.

When a specified speed is reached within the time period of P, spindle speed is checked at that time.

q : Tolerance (%) of a specified spindle speed

$$q = \frac{1 - \text{actual spindle speed}}{\text{specified spindle speed}} \times 100$$

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

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

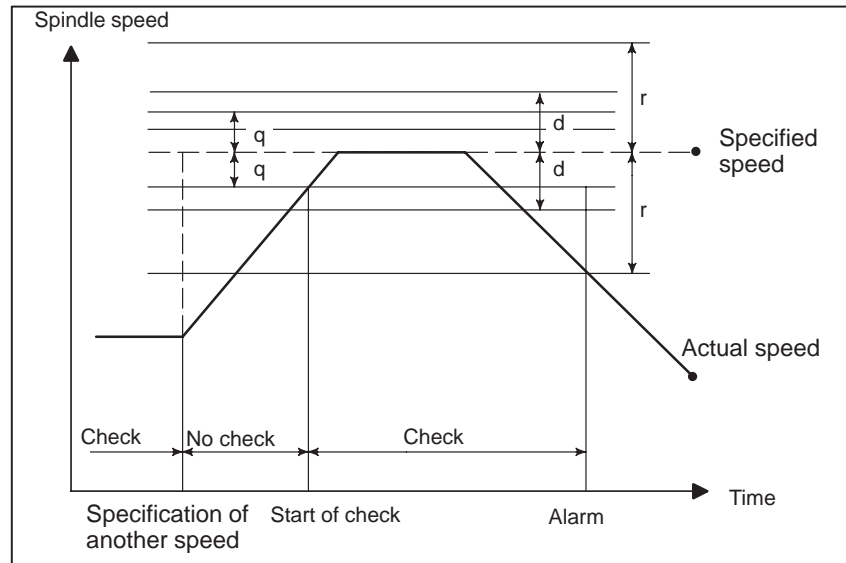
$$r = \frac{1 - \text{speed that can cause overheat}}{\text{specified spindle speed}} \times 100$$

G26 enables the spindle speed fluctuation detection function, and G25 disables the spindle speed fluctuation detection. Even if G25 is specified, p, q, and r are not cleared.

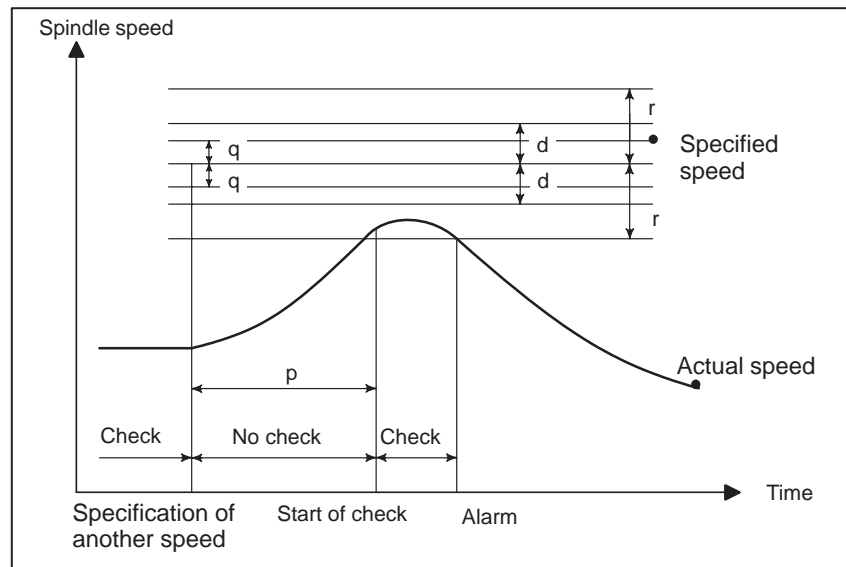
Explanations

The fluctuation of the spindle speed is detected as follows:

1. When an alarm is issued after a specified spindle speed is reached



2. When an alarm is issued before a specified spindle speed is reached



Specified speed :

(Speed specified by address S and five-digit value) \times (spindle override)

Actual speed : Speed detected with a position coder

p : Time elapses since the specified speed changes until a check starts.

q : (Percentage tolerance for a check to start) \times (specified speed)

r : (Percentage fluctuation detected as an alarm condition) \times (specified speed)

d : Fluctuation detected as an alarm (specified in parameter 4913)

An alarm is issued when the difference between the specified speed and the actual speed exceeds both r and d .

NOTE

- 1 When an alarm is issued in automatic operation, a single block stop occurs. The spindle overheat alarm is indicated on the CRT screen, and the alarm signal "SPAL" is output (set to 1 for the presence of an alarm). This signal is cleared by resetting.
- 2 Even when reset operation is performed after an alarm occurs, the alarm is issued again unless the cause of the alarm is corrected.
- 3 No check is made during spindle stop state (*SSTP = 0).
- 4 By setting the parameter (No. 564), an allowable range of speed fluctuations can be set which suppresses the occurrence of an alarm. However, an alarm is issued one second later if the actual speed is found to be 0 rpm.
- 5 The specified values of p, q, and r are set in the following parameters. If p, q, or r is omitted from a subsequent command, the value stored in the corresponding parameter is used.
 - p : Parameter No. 712
 - q : Parameter No. 531
 - r : Parameter No. 532
- 6 The units for specifying q or r can be changed to 0.1% by setting bit 0 of parameter No. 397 accordingly.

9.5 SPINDLE POSITIONING FUNCTION

In turning, the spindle connected to the spindle motor is rotated at a certain speed to rotate the workpiece mounted on the spindle. The spindle positioning function turns the spindle connected to the spindle motor by a certain angle to position the workpiece mounted on the spindle at a certain angle. The spindle is positioned about the C-axis.

The spindle positioning function involves the following three operations:

1. Canceling the spindle rotation mode and entering the spindle positioning mode (spindle orientation)
2. Positioning the spindle in the spindle positioning mode
3. Canceling the spindle positioning mode, and entering the spindle rotation mode

9.5.1 Spindle Orientation

When spindle positioning is first performed after the spindle motor is used for normal spindle operation, or when spindle positioning is interrupted, the spindle orientation is required.

Orientation permits the spindle to stop at a predetermined position.

Orientation is directed by the M code set in parameter No. 587. The direction of orientation can be set with a parameter. For the analog spindle, the direction is set in bit 2 of parameter 003).

For the serial spindle, it is set in bit 4 of parameter 6500.

9.5.2 Spindle Positioning

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

Positioning with a semi-fixed angle specified by an M code

Address M is followed by a 2-digit numeric. The specifiable value may be one of the six values from $M\alpha$ to $M(\alpha+5)$. Value α must be set in parameter No. 589 beforehand. The positioning angles corresponding to $M\alpha$ to $M(\alpha+5)$ are listed below. Value β must be set in parameter No. 590 beforehand.

M-code	Positioning angle	(Ex.) $\beta=30^\circ$
$M\alpha$	β	30°
$M(\alpha+1)$	2β	60°
$M(\alpha+2)$	3β	90°
$M(\alpha+3)$	4β	120°
$M(\alpha+4)$	5β	150°
$M(\alpha+5)$	6β	180°

Specify the command with incremental values. The direction of rotation can be specified in bit 1 of parameter 031.

Positioning with a given angle specified by address C or H

Specify the position using address C or H followed by a signed numeric value or numeric values. Addresses C and H must be specified in the G00 mode.

(Example) C-1000
H450

The end point must be specified with a distance from the program reference position (in absolute mode) using address C. Alternatively, the end point must also be specified with a distance from the start point to the end point (in incremental mode) using address H.

A numeric with the decimal point can be entered.

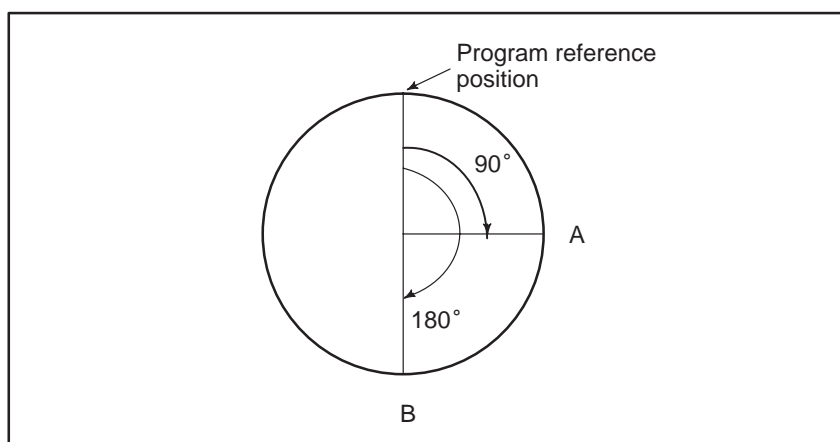
The value must be specified in degrees.

(Example) C35.0=C35 degrees

Program reference position

The position to which the spindle is oriented is assumed as the program reference position. The program reference position can be changed by setting of a coordinate system (G50) or automatic setting of a coordinate system (#bit 7 of parameter 010).

Feedrate for positioning



Command format		G code A		G code B and C	
		Address used	Command A-B in the above figure	Address used and G code	Command A-B in the above figure
Absolute command	Specify the end point with a distance from the program reference position.	C	C180.0 ;	G90C	G90C180.0;
Incremental command	Specify a distance from the start point to the end point.	H	H90.0 ;	G91C	G90C90.0 ;

Feedrate during positioning

The feedrate during positioning equals the rapid traverse speed specified in parameter No. 520. Linear acceleration/deceleration is performed. For the specified speed, an override of 100%,50%,25%,and F0 (parameter No. 585) can be applied.

Speed during orientation

The tool moves at the rapid traverse speed set in parameter No.520 until a sufficient speed for orientation is attained. After the speed for orientation has been attained, orientation is performed at the speed set in parameter No.586.

9.5.3 Canceling Spindle Positioning

When modes are to be switched from spindle positioning to normal spindle rotation, the M code set in parameter No. 588 is specified.

WARNING

- 1 Feed hold, dry run, machine lock, and auxiliary function lock cannot be performed during spindle positioning.
- 2 Parameter No. 589 must always be set even when positioning with a semi-fixed angle specified in an M-code is not performed. If the parameter is not set, M-codes from the M00 to M05 do not function properly.

NOTE

- 1 Specify spindle positioning alone in a block. A move command for the X or Z axis cannot be specified within the same block.
- 2 When emergency stop is applied during spindle positioning, spindle positioning stops. To resume it, restart with the orientation step.
- 3 The serial spindle Cs-axis contour control function and the spindle positioning function cannot be used at a time. If both options are specified, the spindle positioning function has priority.
- 4 The spindle positioning axis is indicated in pulses in the machine coordinate system.

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

By specifying a 2-digit/4-digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool. This is mainly used to select tools on the machine.

One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commandable with address T and the correspondence between the T codes and machine operations.

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

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

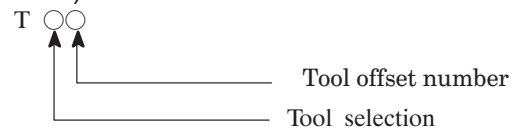
The selection of either sequence depends on the machine tool builder's specifications. Refer to the machine tool builder's manual for details.

The value after the T code indicates the desired tool. Part of the values is also used as the offset number indicating the compensation amount for tool offset.

The above can be used in either of two ways: bit 0 of parameter No. 014 is used to select the method to be used.

1. Last one digit of T-code designates the offset number.

(parameter 014#0=1)



2. Last two digits of T-code designate the offset number.

(parameter 014#0=0)



Explanations

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

Example(T2+2)

N1G00X100.0Z140.0 ;

N2T0313;(Select Tool No. 3 and Offset value No.13)

N3X40.0Z105.0;

Some machines use a 1-digit value to specify tool selection.






10.2
SIMPLIFIED TOOL
LIFE MANAGEMENT

By counting the number of M codes (M30 and M02) at the end of a program, the number of parts produced can be counted. When the total number of parts reaches a specified value (tool life), the tool is assumed to have reached the end of its service life. The life count is incremented, and the parts counter is cleared. According to the life count and parameter-specified compensation value, compensation is applied to the programmed T code.

Data display and setting related to tool life management

When the simplified tool life management function is enabled, the offset screen displays the following tool life management data page:

TOOL LIFE DATA		O0000 N0000	
NO.		DATA	
3910	TOOL LIFE		300
3911	PARTS COUNT (LIFE)		283
3912	LIEF COUNT		5
3913	PARTS COUNT (TOTAL)		1783
ACTUAL POSITION (RELATIVE)			
U	0.000	W	0.000
H	0.000	V	0.000
NO. 3910 =		S	0 T
MDI			
[TOOLLF]	[]	[]	[TOOLFM] []

- 1 Press the  function key. Select the tool life management data page by pressing the continuation menu key  then the **[TOOL LF]** soft key.
- 2 Using the cursor keys  and , position the cursor to the data item to be set.
Type in a desired value using the data input keys.
Press the  key. The value is set and displayed.

● **Explanation of each data item**

No. 3910: Tool life parts count

When the parts counter (No. 3911) reaches the value set with this item, the tool is assumed to have reached the end of its service life.

No. 3911: Parts counter

This counter is incremented by 1 each time M02 or M30 is executed. Each time the tool life period expires, this counter is cleared to 0.

No. 3912: Life count

This counter is used to indicate how many times the tool life period has expired. This counter is incremented by 1 each time the tool life period expires.

No. 3913: Total parts count

This count is the same as the parts count displayed on the POSITION screen. Usually, the following formula is satisfied:

(Total parts count) = (tool life parts count) x (life count) + (parts counter)

10.2.1

Compensation Applied to A Programmed T code

A programmed T code, T $\square\square\square\square$, consists of a tool selection number $\square\square$ and tool offset number $\square\square$. Compensation is applied to each number for execution as T $\bullet\bullet\bullet\bullet$.

Parameter No. 117 and No. 118 are used to specify compensation values for a tool selection number and tool offset number.

(1) Prior to the expiry of the first tool life period (when line count=0)

Tool offset number $\blacksquare\blacksquare = \square\square$

Tool selection number $\bullet\bullet = \square\square$

(2) After the first tool life period has expired (when life count = 1)

Tool offset number $\blacksquare\blacksquare = \square\square + (\text{offset number compensation value})$

Tool selection number $\bullet\bullet = \square\square + (\text{tool selection number compensation value})$

(3) After the second tool life period has expired (when life count=2)

Tool offset number $\blacksquare\blacksquare = \square\square + (\text{offset number compensation value}) \times 2$

Tool selection number $\bullet\bullet = \square\square + (\text{tool selection number compensation value}) \times 2$

(4) After the n-th tool life period has expired (when life count = n)

Tool offset number $\blacksquare\blacksquare = \square\square + (\text{offset number compensation value}) \times n$

Tool selection number $\bullet\bullet = \square\square + (\text{tool selection number compensation value}) \times n$

NOTE

- 1 When the programmed tool offset number and programmed tool selection number are both 0, compensation is not applied.
- 2 When a tool offset number after compensation ($\blacksquare\blacksquare$) or a tool selection number after compensation ($\bullet\bullet$) exceeds its maximum allowable value (set in parameter No.119 or No.120, respectively), the value of $\blacksquare\blacksquare$ or $\bullet\bullet$ is replaced with the remainder resulting from subtracting the maximum allowable value from the $\blacksquare\blacksquare$ or $\bullet\bullet$ value.
- 3 When, after compensation, a tool offset number ($\blacksquare\blacksquare$) exceeds the number of offset pairs, P/S alarm No. 030 is issued. Accordingly, a value not greater than the number of offset pairs must be set as the maximum tool offset number.
- 4 A T code (T $\bullet\bullet\bullet\bullet$), after compensation, is displayed in the T code all-time display field.

Examples

Parameters: Offset number compensation value (No. 117) = 8
Tool selection number compensation value (no. 118) = 10
Maximum allowable offset number (No. 119) = 16
Maximum allowable tool selection number (No. 120) = 99

Program	After first tool life period	After second tool life period
T0101	T1109	T2101
:	:	:
:	:	:
T0203	T1211	T2203
:	:	:
:	:	:
T0305	T1313	T2305
:	:	:
:	:	:
T0100	T1100	T2100
:	:	:
:	:	:
T0001	T0009	T0001

10.3 TOOL LIFE MANAGEMENT

Tools are classified into some groups. For each group, a tool life (time or frequency of use) is specified. Each time a tool is used, the time for which the tool is used is accumulated. When the tool life has been reached, the next tool previously determined in the same group is used. This function is called the tool life management function.

With 0-TTC, tool life management is performed for each tool post separately. So tool life management data is also set for each tool post.

10.3.1 Program of Tool Life Data

Format

Tools used sequentially in each group and their tool life are registered in the CNC as following program format of table 10.3.1.

Table 10.3.1 Program format of life management

Tape format	Meaning
O_____ ;	Program number
G10L3;	Start of setting tool life data
P____L_____ ;	P____:Group number (1 to 128) L____:Tool life (1 to 9999)
T_____ ;	(1) T :_____ Tool number
T_____ ;	(2) {
⋮	Tools are selected from
⋮	(n) (1)to (2) to ... to (n).
P____L_____ ;	} Data for the next group
T_____ ;	
T_____ ;	
⋮	
G11;	End of setting tool life data
M02(M30);	End of program

For the method of registering tool life data in CNC ,refer to OPERATION **11.4.10.**

Explanations

A tool life is specified either as the time of use (in minutes) or the frequency of use, which depends on the parameter setting parameter No. 039#2.

Up to 4300 minutes in time or 9999 times in frequency can be specified for a tool life.

The number of groups to be registered and the number of tools registered per group can be combined in three ways. One of the three combinations is set by a parameter No.039#0,#1.

	No.039#1	No.039#0	Number of groups	Number of tools
(1)	0	0	16	16
(2)	0	1	32	8
(3)	1	0	64	4

In any combination, up to 256 tools in total can be registered.

If there are not more than 16 groups and each group contain not more than 16 tools, for example, select combination (2) (1). Select combination if there are not more than 32 groups and each group contain not more than eight tools. To change the combination, changel the parameter, then set program is executed with the old tool group combination set in the NC. Whenever the parameter is changed, be sure to reexecute the group setting program.

The same tool number may appear anywhere any times in the program of tool life data.

Example

<pre> O0001 ; G10L3 ; P001L0150 ; T0011 ; T0132 ; T0068 ; P002L1400 ; T0061 ; T0241 ; T0134 ; T0074 ; P003L0700 ; T0012 ; T0202 ; G11 ; M02 ; </pre>	<div style="display: flex; align-items: center;"> <div style="font-size: 3em; margin-right: 10px;">}</div> <div>Data of group 1</div> </div> <div style="display: flex; align-items: center; margin-top: 10px;"> <div style="font-size: 3em; margin-right: 10px;">}</div> <div>Data of group 2</div> </div> <div style="display: flex; align-items: center; margin-top: 10px;"> <div style="font-size: 3em; margin-right: 10px;">}</div> <div>Data of group 3</div> </div>
------------------------------------------------------------------------------------------------------------------------------------------------------	----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------

Explanations

The group numbers specified in P need not be serial. They need not be assigned to all groups, either. When using two or more offset numbers for the same tool in the same process, set as follows;

Tape format	Meaning
P004L0500; T0101; T0105; T0108; T0206; T0203; T0202; T0209; T0304; T0309; P005L1200; T0405;	<p>The tools in group 4 are used from (1) to (2) to (3).</p> <p>(1) Each tool is used 500 times (or for 500 minutes).</p> <p>(2) When this group is specified three times in one process, the offset numbers are selected in the following orders:</p> <p>(3) Tools (1): 01→05→08 Tools (2): 06→03→02→09 Tools (3): 04→09</p>

10.3.2

Counting a Tool Life

Explanation

- **When a tool life is specified as the time of use (in minutes)**

In this case, the period during which the tool is actually used in cutting mode, between the specification of TΔΔ99 (ΔΔ= tool group number) and that of the next tool group or an ordinary tool number in a machining program, is counted in blocks of 4 seconds. The time taken for single-block stoppage, feed hold, rapid traverse, dwelling, and FIN wait is ignored.

Up to 4300 minutes can be specified for a life.

- **When a tool life is specified as the frequency of use**

Counting is performed for each process that is initiated by the cycle start of a machining program and ended when the NC is reset by the M02 or M03 command. The counters for tool groups used in a lprocess are incremented by one. Even when the same group is specified more than once in one process, the counter is incremented only by one. Up to 9999 can be set for a tool life.

Counting of a tool life is performed for each group. The life counter contents are not erased even when the power of CNC is cut off.

When a life is specified as the frequency of use, apply an external reset (ERS) signal to the CNC when M02 or M30 is executed.

10.3.3 Specifying a Tool Group in a Machining Program

In machining programs, T codes are used to specify tool groups as follows:

Tape format	Meaning
⋮ TΔΔ99; ⋮ ⋮ TΔΔ88; ⋮ M02(M300);	Ends the tool used by now, and starts to use the tool of the ΔΔ group. "99" distinguishes this specification from ordinary specification. Cancels the offset of the tool of the group. "88" distinguishes this specification from ordinary specification. Ends the machining program.

Explanations

Tape format	Meaning
T0199; ⋮ T0188; ⋮ T0508; ⋮ T0500; ⋮ T0299; ⋮ T0199; ⋮ ⋮	Ends the previous tool, and starts to use the tool of the 01 group. Cancels the offset of the tool of the 01 group. Ends the tool of the 01 group. Selects tool number 05 and offset number 08. Cancels the offset of tool number 05. Ends tool number 05, and starts to use the tool of the 02 group. Ends the tool of the 02 group, and starts to use the tool of the 01 group. If more than one offset number is specified for the tool, the second offset number is selected. Otherwise, the previous offset number is used.

11

AUXILIARY FUNCTION

General

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) .

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 address M followed by 3 digit value is specified, a code signal and strobe signal are transmitted. These signals are used for turning on/off the power to the machine.

In general, only one M code is valid in a block but up to three M codes can be specified in a block (although some machines may not allow that). The correspondence between M codes and functions is up to the machine tool builder.

All M codes are processed in the machine except for M98, M99, M198, M codes for calling a subprogram (parameters Nos. 240 to 242), and M codes for calling a custom macro (parameters Nos. 230 to 239). Refer to the appropriate manual issued by the machine tool builder.

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 12.3** 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.

NOTE

The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, ten M codes which do not buffer can be set by parameters (Nos. 111 and 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. Before this function can be used, bit 7 of parameter No. 065 must be set.

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. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.

M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.

Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.

Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.

Examples

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28G91U0W0 ; : : :	M40M50M60 ; G28G91U0W0 ; : : : : :

11.3 THE SECOND AUXILIARY FUNCTIONS (B CODES)

Indexing of the table is performed by address B and a following 8-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.

Explanations

- **Command range**
- **Command method**

0 to 99999999

1. The decimal point can be used for input.

Command	Output value
B10.	10000
B10	10

2. It is possible to change over the scale factor of B output, 1000 or 1 when the decimal point input is omitted, using the parameter (No.015#7).

Command	Output value
When DPI is 1:	B1 1000
When DPI is 0:	B1 1

3. It is possible to change over the scale factor of B output 1000 or 10000 when the decimal point input is omitted in the inch input system, using the parameter (No.032#5) When 015#7=1.

Command	Output value
When 032#5 is 1:	B1 10000
When 032#5 is 0:	B1 1000

Restrictions

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

11.4 OUTPUTTING SIGNAL NEAR END POINT

This function outputs the signal DEN2 when the tool approaches to the end point of a rapid traverse (G00) or linear interpolation (G01) block where a miscellaneous function (M-code) and a tolerable distance are commanded. As the miscellaneous function can be triggered near the end point by using the DEN2, it shortens the execution time.

Format

☐ G00 ☐ Motion Command , Q____ M____ ;
☐ G01 ☐

Motion Command: Ordinary Motion Command
,Q____: Tolerable Distance
M____: Ordinary miscellaneous function

Explanations

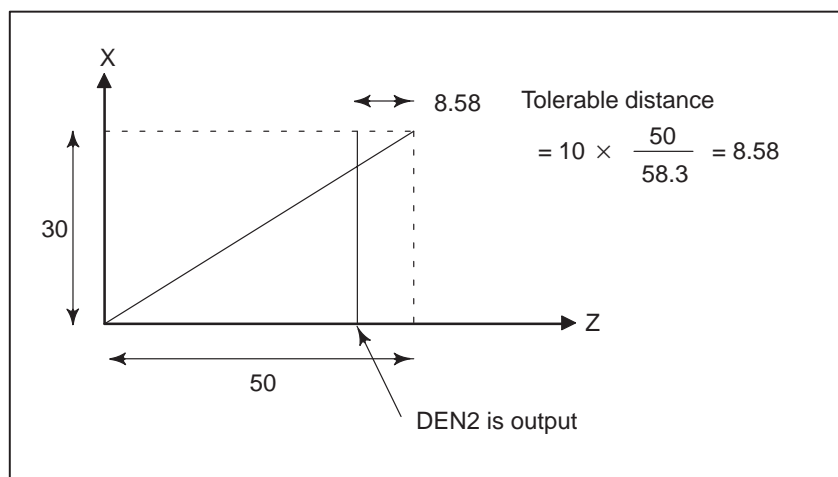
The miscellaneous function (M-code) and the tolerable distance ",Q" are commanded on the rapid traverse (G00) or linear interpolation (G01) block. The command unit of ",Q" is the input increment. The decimal point is permitted in ",Q". When "the distance to go" of the longest motion axis decreases to the tolerable distance defined by ",Q", the signal DEN2 which means the nearness of the end point is output. The tolerable distance of the longest motion is defined by the following formula.

Tolerable distance of longest motion =

$$\text{Command of ",Q"} \times \frac{\text{distance of longest motion}}{\text{Block distance}}$$

Program Example

G01 U60.0 W50.0 ,Q10.0 M123 ;



NOTE

- 1 The special M-codes which are managed internally such as M98 and M99, are not used for M-codes of this function.
- 2 This function is not used during the multiple turning cycles.
- 3 This function can not be commanded by MDI operation A.
- 4 The DEN2 keeps on till the completion signal (FIN) fo the miscellaneous function turns on. When the FIN inputs before reaching the end, the distribution end signal (DEN) is not output at the end.
- 5 When the function is used on the rapid traverse block of the simultaneous axes motion, there may be the error between the tolerance distance and the actual point where the DEN2 is output.

12 PROGRAM CONFIGURATION

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.

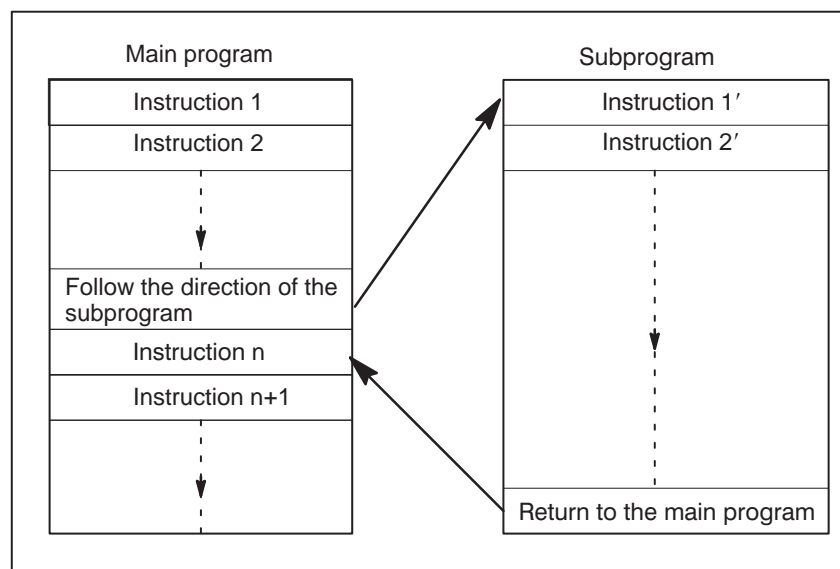


Fig. 12 (a) Main program and Subprogram

The CNC memory can hold up to 400 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine.

Program components

A program consists of the following components:

Table 12 (a) 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

The diagram illustrates the structure of a program file. It begins with a 'Tape start' symbol (%). This is followed by a 'Leader section' containing the word 'TITLE;'. Next is the 'Program start' symbol (O0001;). The 'Program section' is indicated by a bracket and includes a 'Comment section' (COMMENT;) and the 'M30;' command. The program concludes with a 'Tape end' symbol (%).

Fig. 12 (b) Program configuration

Program section configuration

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

Program section configuration	Program section
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z-32.0 H01 ;
:	:
Block n	Nn M2 ;
Program end	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.

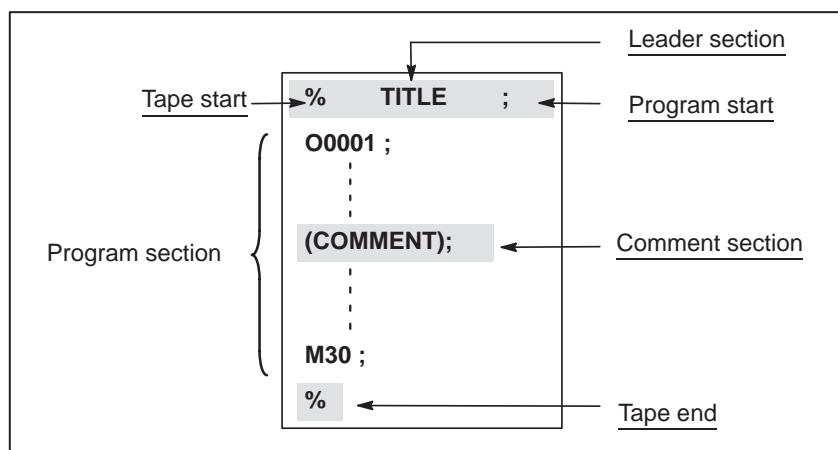


Fig. 12.1 Program configuration

Explanations

- **Tape start**

The tape start indicates the start of a file that contains NC programs. The mark is not 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 parity check is not made. So a leader section can contain any codes except the EOB code.

- **Program start**

The program start code is to be entered immediately after a leader section, that is, immediately before a program section. This code indicates the start of a program, and is always required to disable the label skip function.

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. in a comment section. 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 6 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 in memory command mode, all comment sections are ignored.

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

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. The mark is not displayed on the CRT display screen. However, when a file is output, the mark is automatically output at the end of the file.

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.

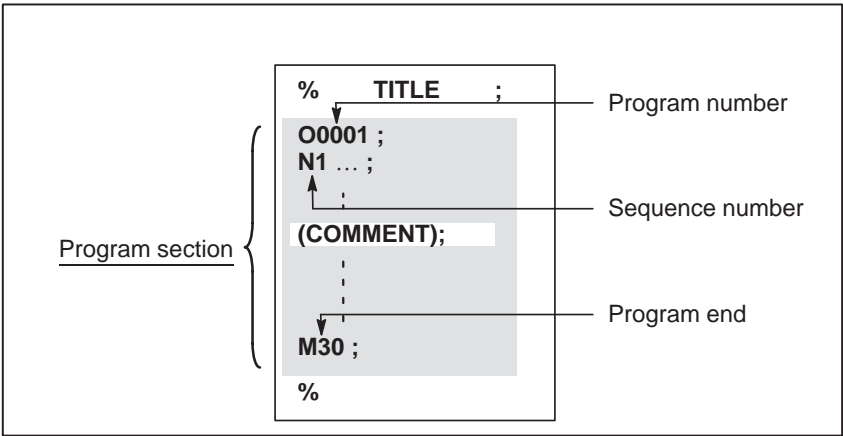


Fig. 12.2 (a) Program configuration

• 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 five 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. 11.5.1 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 (*)	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.
Dwell	P, X, U	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>Z</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 feedrate 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 to 9999	1 to 9999
Sequence number		N	1 to 9999	1 to 9999
Preparatory function		G	0 to 99	0 to 99
Dimension word	Increment system IS-B	X, Y, Z, U, V, W, A, B, C, I, J, K, R,	-99999.999 to +99999.999	-9999.9999 to +9999.9999
	Increment system IS-C		-9999.9999 to +9999.9999	-999.99999 to +999.99999
Feed per minute	Increment system IS-B	F	1 to 100000mm/min	0.01 to 4000.00 inch/min
	Increment system IS-C		1 to 12000mm/min	0.01 to 480.00 inch/min
Feed per revolution		F	0.0001 to 500.0000 mm/rev	0.000001 to 9.999999 inch/rev
Spindle speed function		S	0 to 20000	0 to 20000
Tool function		T	0 to 9999	0 to 9999
Auxiliary function		M	0 to 999	0 to 999
		B	0 to 999999999	0 to 999999999
Dwell	Increment system IS-B	P, X, U	0 to 99999.999s	0 to 99999.999s
	Increment system IS-C		0 to 9999.9999s	0 to 9999.9999s
Designation of a program number		P	1 to 9999	1 to 9999
Number of repetitions		P	1 to 9999	1 to 9999

NOTE

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

- **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 punching 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.

WARNING

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 previous item for optional block skip.)

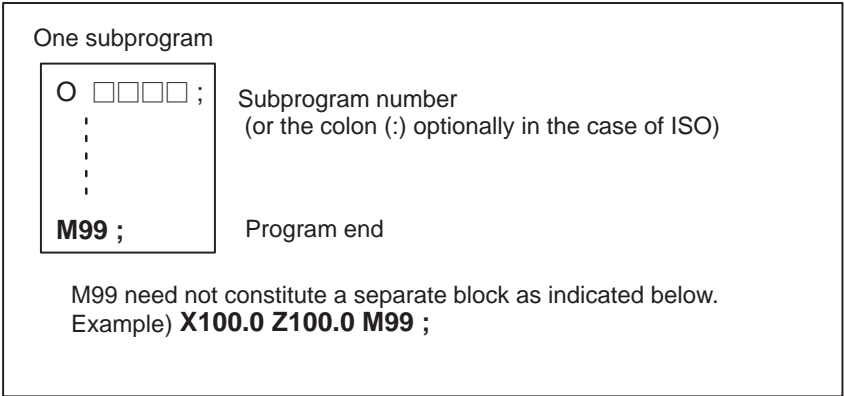
12.3

SUBPROGRAM

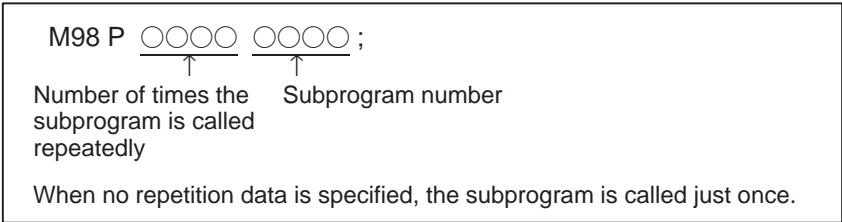
If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

- Subprogram configuration

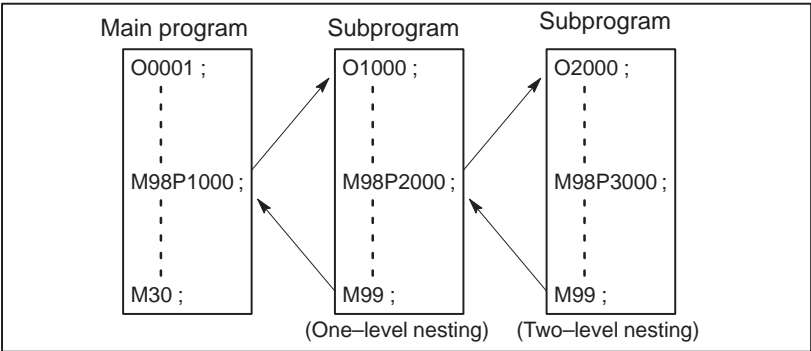


- Subprogram call



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

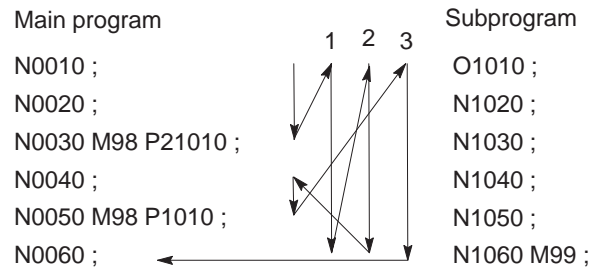
- 1 The M98 and M99 signals are not output to the machine tool.
- 2 If the subprogram number specified by address P cannot be found, an alarm (No. 078) is output.

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**

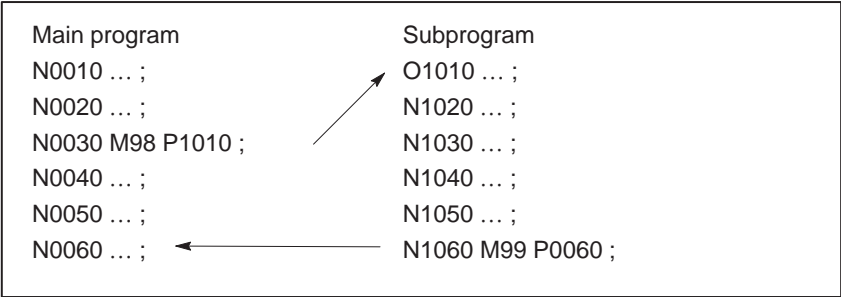
A subprogram can call another subprogram in the same way as a main program calls a subprogram.

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 much longer time than the normal return method to return to the main program.

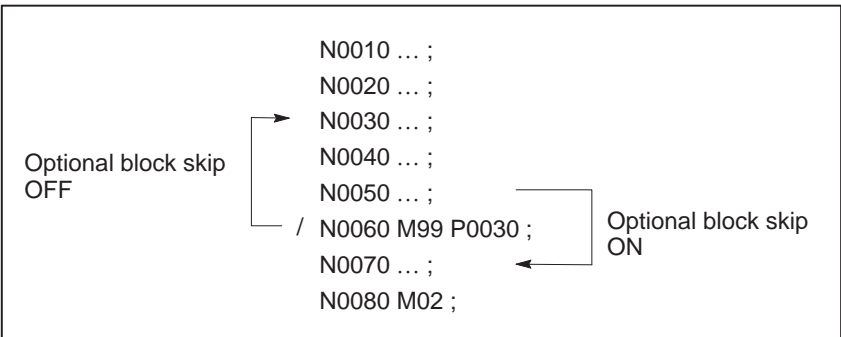


• Using M99 in the main program

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.

Execution is repeated while the optional block skip function is set to off. If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.

If /M99Pn ; is specified, control returns not to the start of the main program, but to sequence number n.

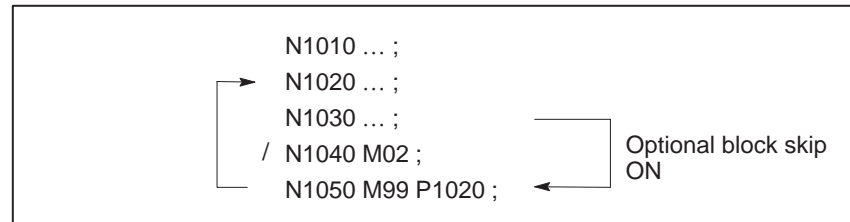


- **Using a subprogram only**

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

(See Section 9.3 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 switch 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 MULTIPLE REPETITIVE CYCLE
- 13.3 CANNED CYCLE FOR DRILLING
- 13.4 CANNED GRINDING CYCLE
- 13.5 CHAMFERING AND CORNER R
- 13.6 MIRROR IMAGE FOR DOUBLE TURRET
- 13.7 DIRECT DRAWING DIMENSIONS PROGRAMMING

NOTE

Explanatory diagrams in this chapter uses diameter programming in X axis.
In radius programming, changes U/2 with U and X/2 with X.

13.1 CANNED CYCLE (G90, G92, G94)

There are three canned cycles : the outer diameter/internal diameter cutting canned cycle (G90), the thread cutting canned cycle (G92), and the end face turning canned cycle (G94).

13.1.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

- Straight cutting cycle

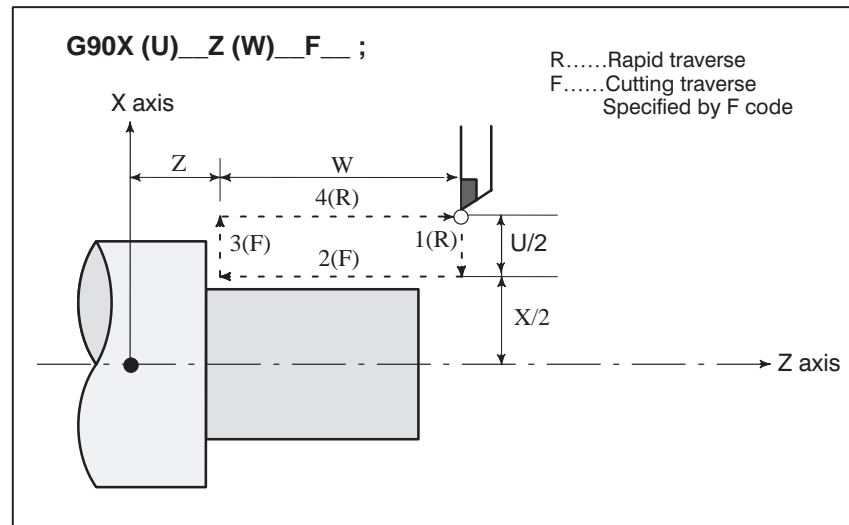


Fig. 13.1.1 (a) Straight Cutting Cycle

In incremental programming, the sign of the numbers following address U and W depends on the direction of paths 1 and 2. In the cycle of 13.1.1 (a), the signs of U and W are negative.

In single block mode, operations 1, 2, 3 and 4 are performed by pressing the cycle start button once.

● Taper cutting cycle

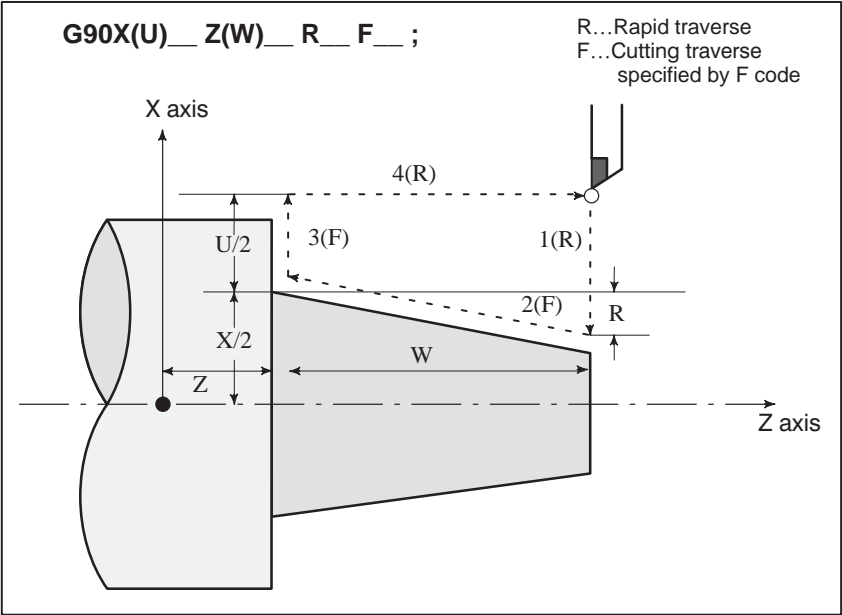


Fig. 13.1.1 (b) Taper Cutting Cycle

● Signs of numbers specified in the taper cutting cycle

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

1. $U < 0, W < 0, R < 0$	2. $U > 0, W < 0, R > 0$
3. $U < 0, W < 0, R > 0$ at $ R \leq \frac{ U }{2}$	4. $U > 0, W < 0, R < 0$ at $ R \leq \frac{ U }{2}$

13.1.2 Thread Cutting Cycle (G92)

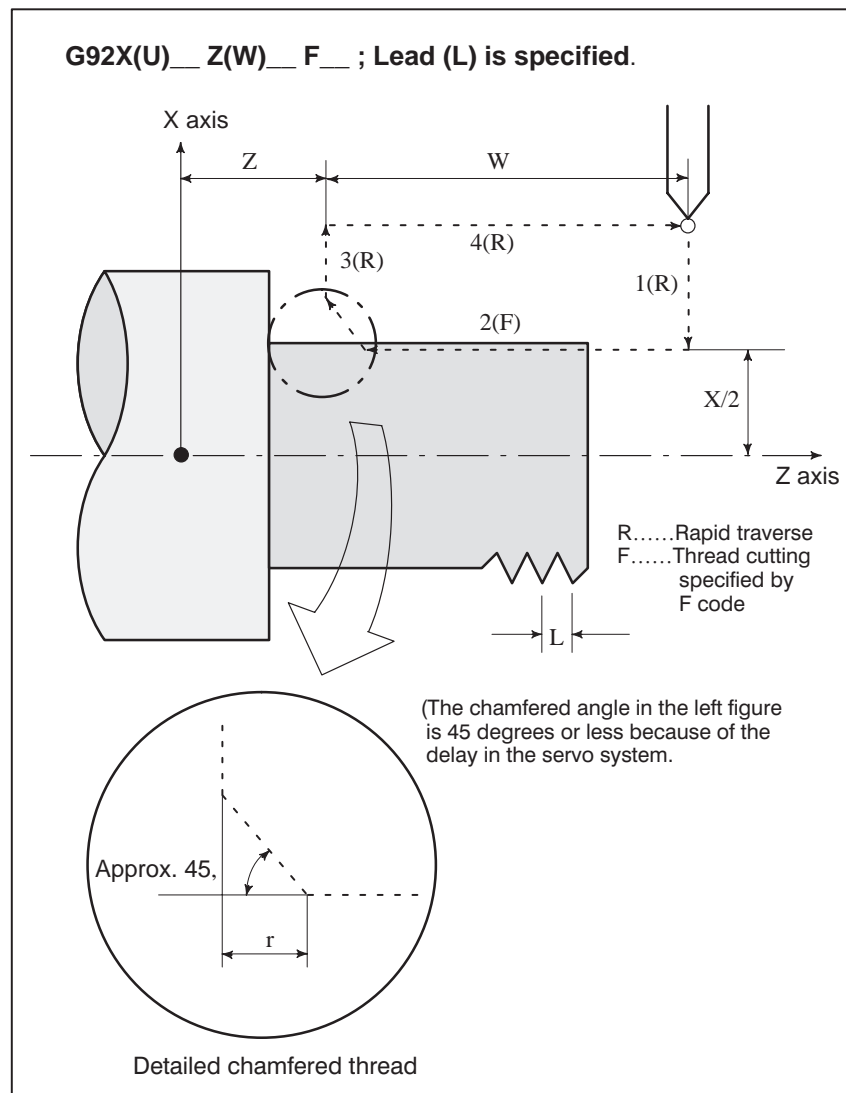


Fig. 13.1.2 (a) Straight Thread Cutting

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is, if the direction of path 1 is the negative along the X axis, the value of U is negative.

The range of thread leads, limitation of spindle speed, etc. are the same as in G32 (thread cutting). Thread chamfering can be performed in this thread cutting cycle. A signal from the machine tool, initiates thread chamfering. The chamfering distance is specified in a range from 0.1L to 12.7L in 0.1L increments by parameter (No. 5130). (In the above expression, L is the thread lead.)

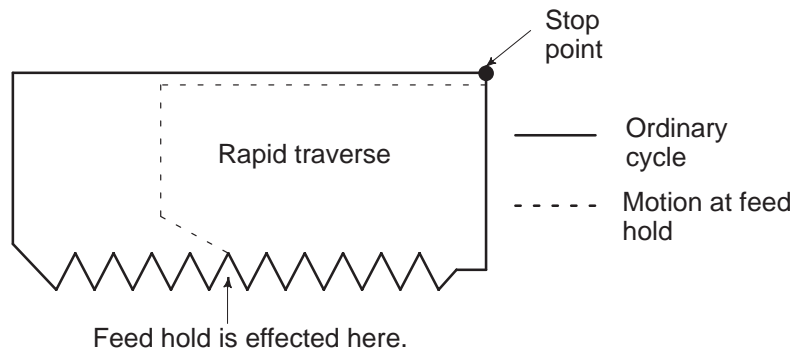
In the single block mode, operations 1, 2, 3, and 4 are performed by pressing cycle start button once.

WARNING

Notes on this thread cutting are the same as in thread cutting in G32. However, a stop by feed hold is as follows ; Stop after completion of path 3 of thread cutting cycle.

CAUTION

The tool retreats while chamfering and returns to the start point on the X axis then the Z axis, as soon as the feed hold status is entered during thread cutting (motion 2) when the "Thread Cutting Cycle retract" option is used.



Another feed hold cannot be made during retreat. The chamfered amount is the same as that at the end point.

- Taper thread cutting cycle

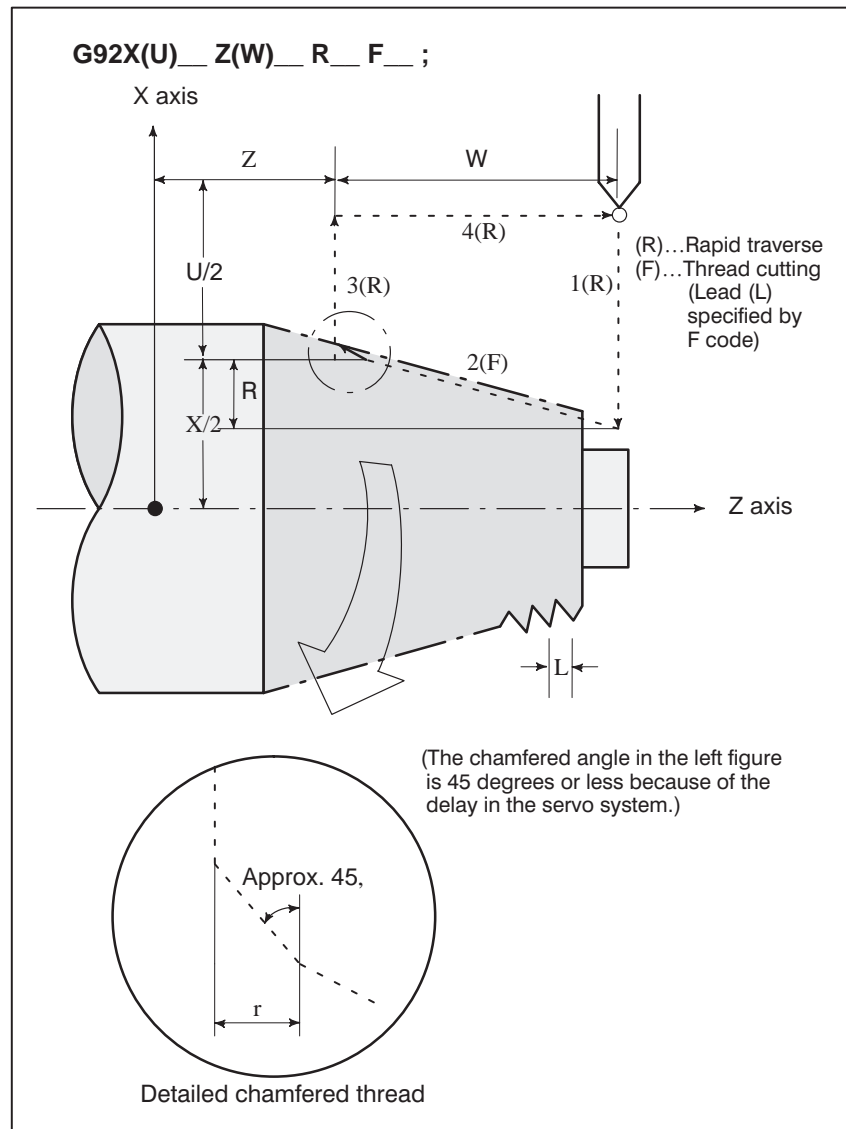


Fig. 13.1.2 (b) Taper thread cutting cycle

13.1.3 End Face Turning Cycle (G94)

- Face cutting cycle

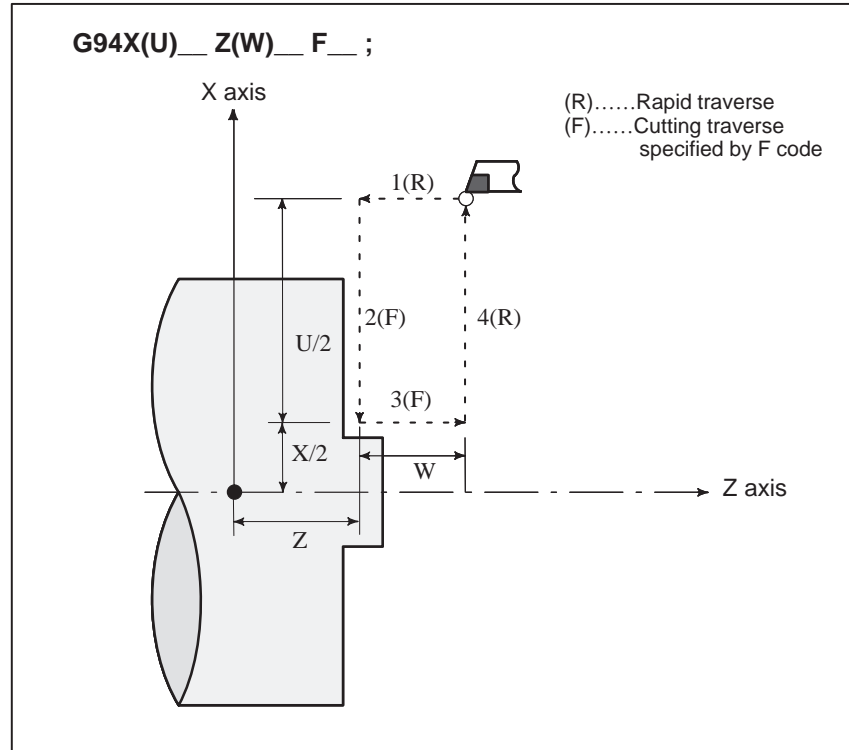


Fig. 13.1.3 (a) Face Cutting Cycle

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is, if the direction of the path is in the negative direction of the Z axis, the value of W is negative.

In single block mode, operations 1, 2, 3, and 4 are performed by pressing the cycle start button once.

- Taper face cutting cycle

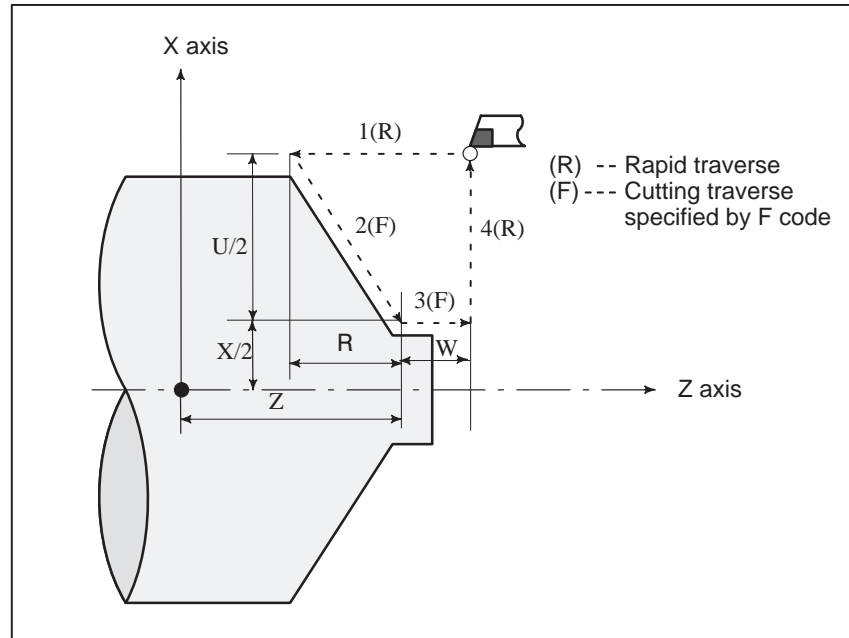


Fig. 13.1.3 (b)

- Signs of numbers specified in the taper cutting cycle

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

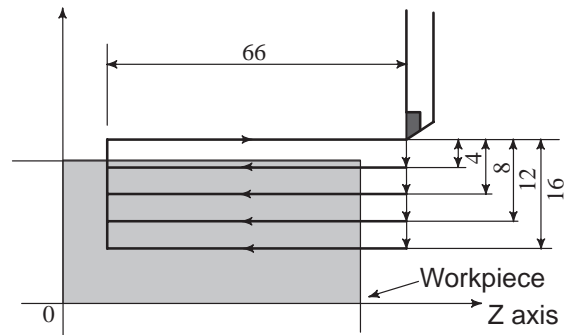
1. $U < 0, W < 0, R < 0$	2. $U > 0, W < 0, R < 0$
3. $U < 0, W < 0, R > 0$ at $ R \leq W $	4. $U > 0, W < 0, R < 0$ at $ R \leq W $

NOTE

- 1 Since data values of X (U), Z (W) and R during canned cycle are modal, if X (U), Z (W), or R is not newly commanded, the previously specified data is effective. Thus, when the Z axis movement amount does not vary as in the example below, a canned cycle can be repeated only by specifying the movement commands for the X-axis.

However, these data are cleared, if a one-shot G code expect for G04 (dwell) or a G code in the group 01 except for G90, G92, G94 is commanded.

(Example) X axis



The cycle in the above figure is executed by the following program.

```
N030 G90 U-8.0 W-66.0 F0.4 ;
N031 U-16.0 ;
N032 U-24.0 ;
N033 U-32.0 ;
```

- 2 The following three applications can be performed.
 - (1) If an EOB or zero movement commands are specified for the block following that specified with a canned cycle, the same canned cycle is repeated.
 - (2) By specifying a canned cycle in the MDI mode, and pushing the cycle start button after the block terminates, the same canned cycle as the previous one will be performed.
 - (3) If the M, S, T function is commanded during the canned cycle mode, both the canned cycle and M, S, or T function can be performed simultaneously. If this is inconvenient, cancel the canned cycle once as in the program examples below (specify G00 or G01) and execute the M, S, or T command. After the execution of M, S, or T terminates, command the canned cycle again.

(Example)

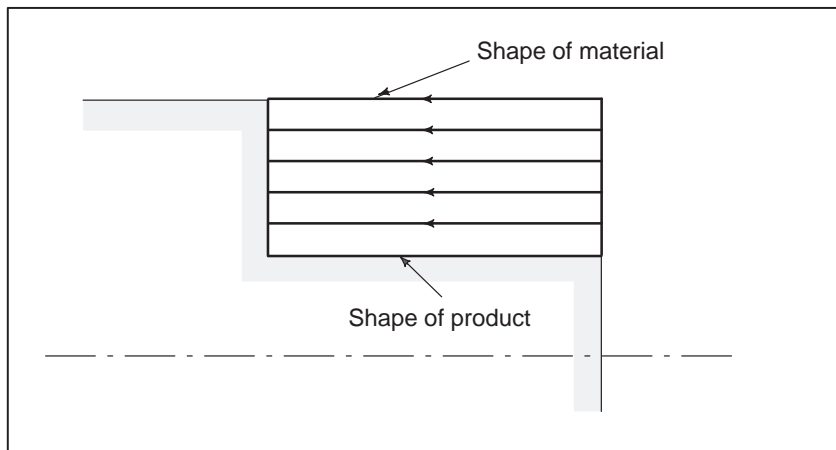
```
N003 T0101 ;
:
:
N010 G90 X20.0 Z10.0 F0.2 ;
N011 G00 T0202 ;
N012 G90 X20.5 Z10.0 ;
```

13.1.4

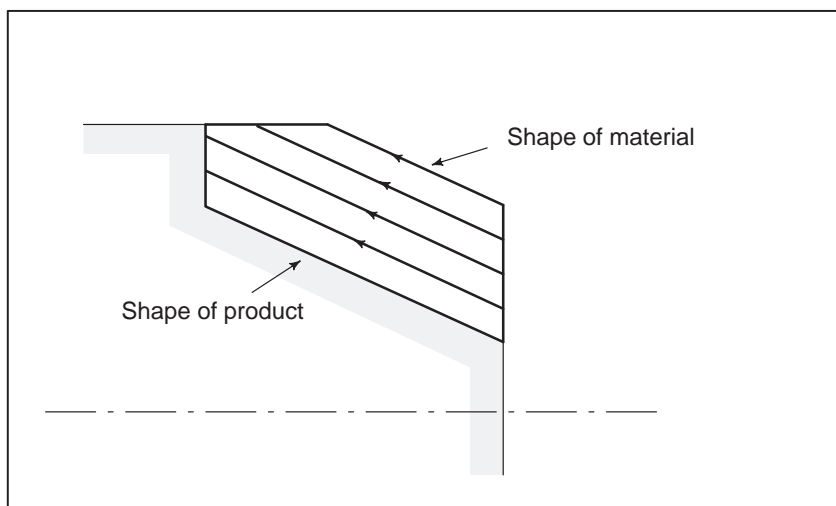
How to Use Canned Cycles (G90, G92, G94)

An appropriate canned cycle is selected according to the shape of the material and the shape of the product.

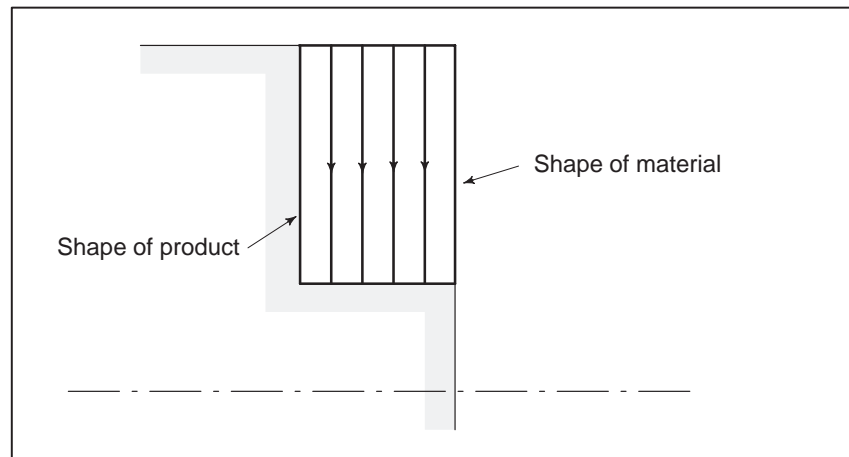
- Straight cutting cycle (G90)



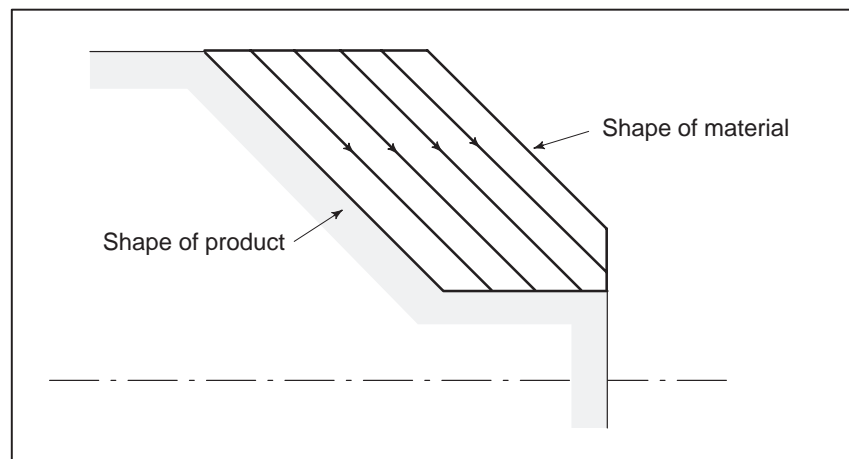
- Taper cutting cycle (G90)



- **Face cutting cycle (G94)**



- **Face taper cutting cycle (G94)**



13.2 MULTIPLE REPETITIVE CYCLE (G70 TO G76)

This option canned cycles to make CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the thread cutting is available.

13.2.1 Stock Removal in Turning (G71)

• Type I

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

If a finished shape of A to A' to B is given by a program as in the figure below, the specified area is removed by Δd (depth of cut), with finishing allowance $\Delta u/2$ and Δw left.

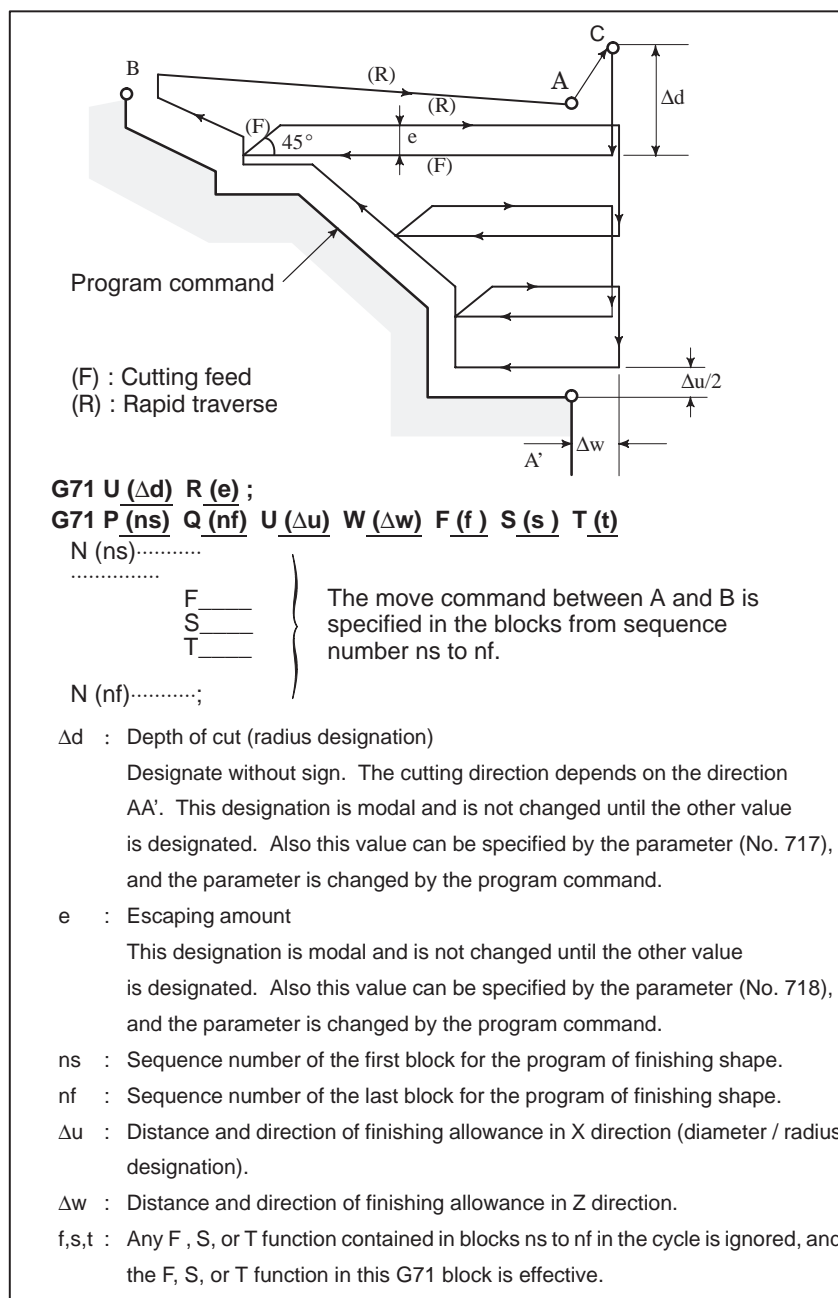


Fig. 13.2.1 (a) Cutting Path in Stock Removal in Turning (Type I)

NOTE

- 1 While both Δd and Δu , are specified by address U, the meanings of them are determined by the presence of addresses P and Q.
- 2 The cycle machining is performed by G71 command with P and Q specification. F, S, and

- **Type II**

Type II differs from type I in the following : The profile need not show monotone increase or decrease along the X axis, and it may have up to 10 concaves (pockets).

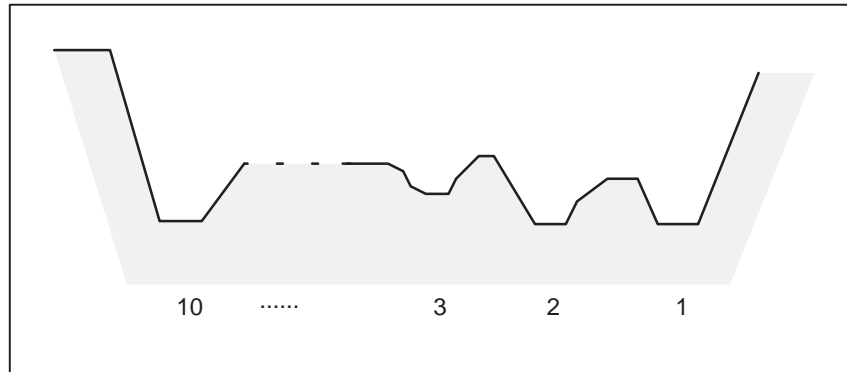


Fig. 13.2.1 (b) Number of Pockets in Stock Removal in Turning (Type II)

Note that, however, the profile must have monotone decrease or increase along the Z axis. The following profile cannot be machined:

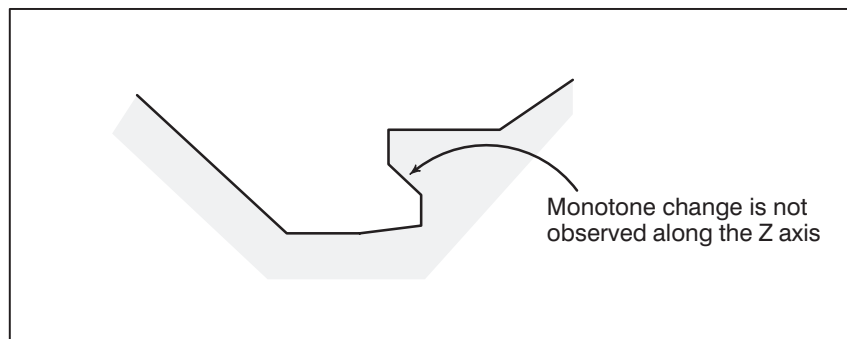


Fig. 13.2.1 (c) Figure Which Cannot Be Machined in Stock Removal in Turning (Type II)

The first cut portion need not be vertical ; any profile is permitted if monotone change is shown along the Z axis.

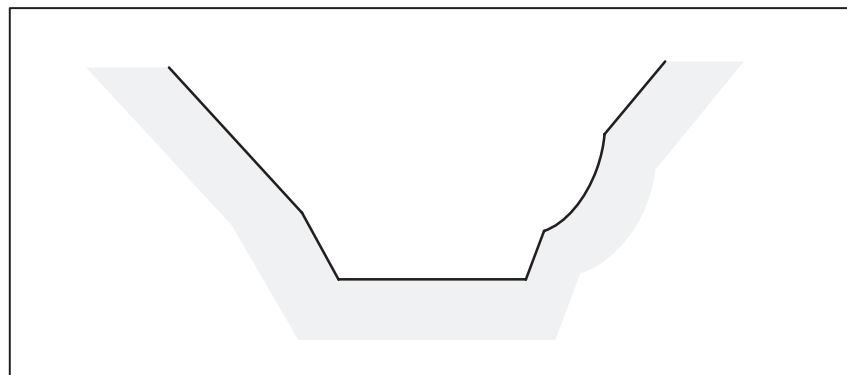


Fig. 13.2.1 (d) Figure Which Can Be Machined (Monotonic change) in Stock Removal in Turning (Type II)

After turning, a clearance is provided by cutting along the workpiece profile.

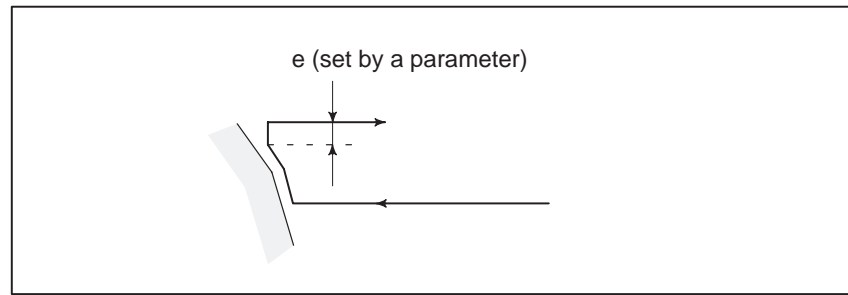


Fig. 13.2.1 (e) Chamfering in Stock Removal in Turning (Type II)

The clearance e (specified in R) to be provided after cutting can also be set in parameter No. 718.

A sample cutting path is given below:

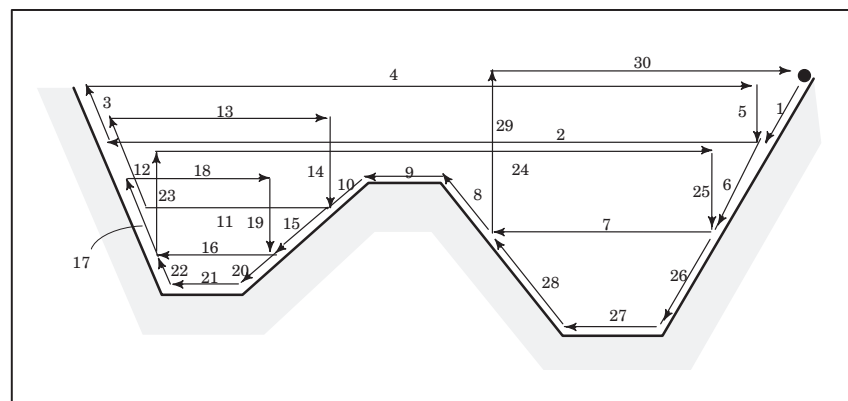


Fig. 13.2.1 (f) Cutting Path in Stock Removal in Facing

Δu and Δw . In turning, the offset of the tool tip radius is assumed to be zero.

$W=0$ must be specified ; otherwise, the tool tip may cut into one wall side. For the first block of a repetitive portion, two axes X(U) and Z (W) must be specified. When Z motion is not performed, W0 is also specified.

• Distinction between type I and type II

When only one axis is specified in the first block of a repetitive portion

-- Type I

When two axes are specified in the first block of a repetitive portion

-- Type II

When the first block does not include Z motion and type II is to be used, W0 must be specified.

(Example)

TYPE I	TYPE II
G71 U10.0 R5.0 ;	G71 U10.0 R5.0 ;
G71 P100 Q200....;	G71 P100 Q200.....;
N100X (U)___;	N100X (U)___ Z(W)___;
:	:
:	:
N200.....;	N200.....;

For the Type II, the option of multiple repetitive cycle addition is necessary.

13.2.2 Stock Removal in Facing (G72)

As shown in the figure below, this cycle is the same as G71 except that cutting is made by a operation parallel to X axis.

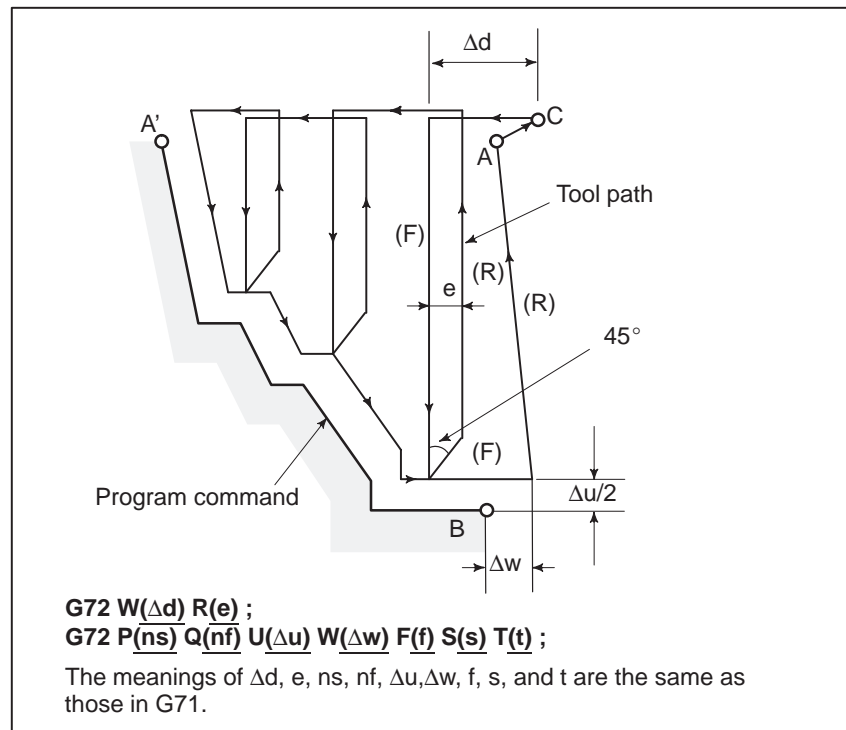


Fig. 13.2.2 (a) Cutting Path in Stock Removal in Facing

- **Signs of specified numbers**

The following four cutting patterns are considered. All of these cutting cycles are made parallel to X axis and the sign of Δu and Δw are as follows :

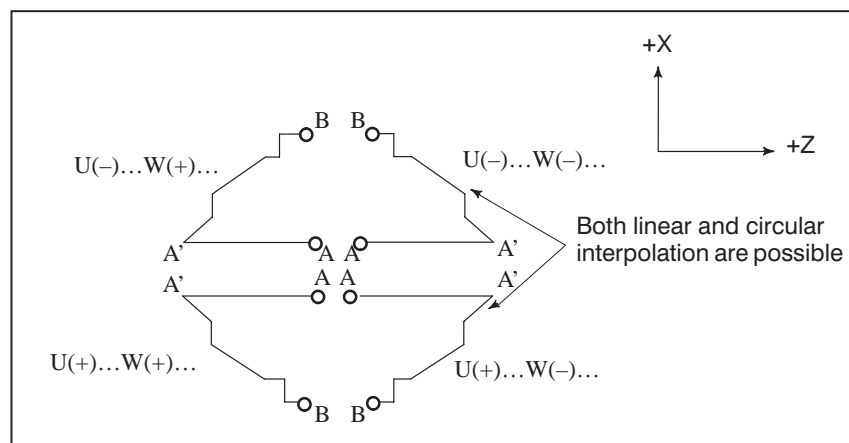
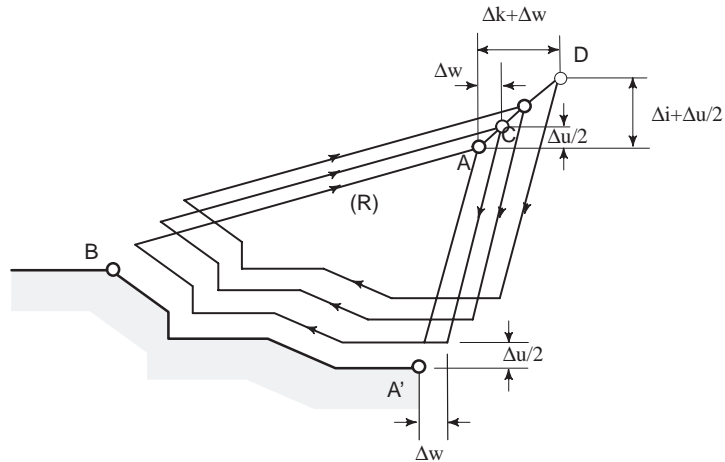


Fig. 13.2.2 (b) Signs of Numbers Specified with U and W in Stock Removal in Facing

The tool path between A and A' is specified in the block with sequence number "ns" including G00 or G01, and in this block, a move command in the X axis cannot be specified. The tool path between A' and B must be steadily increasing and decreasing pattern in both X and Z axes. Whether the cutting along AA' is G00 or G01 mode is determined by the command between A and A', as described in item 14.2.1.

13.2.3 Pattern Repeating (G73)

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



The pattern commanded in the program should be as follows.

A→A'→B

G73 U (Δi) W (Δk) R (d) ;

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

N (ns).....

.....

F _____
S _____
T _____

N (nf).....;

The move command between A and B is specified in the blocks from sequence number ns to nf.

Δi : Distance and direction of relief in the X axis direction (Radius designation).

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 719, and the parameter is changed by the program command.

Δk : Distance and direction of relief in the Z axis direction.

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 720, and the parameter is changed by the program command.

d : The number of division

This value is the same as the repetitive count for rough cutting. This designation is modal and is not changed until the other value is designated.

Also, this value can be specified by the parameter No. 721, and the parameter is changed by the program command.

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

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

Δu : Distance and direction of finishing allowance in X direction (diameter/radius designation)

Δw : Distance and direction of finishing allowance in Z direction

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

Fig. 13.2.3 Cutting path in Pattern Repeating

NOTE

- 1 While the values Δi and Δk , or Δu and Δw are specified by address U and W respectively, the meanings of them are determined by the presence of addresses P and Q in G73 block.
When P and Q are not specified in a same block, addresses U and W indicates Δi and Δk respectively. When P and Q are specified in a same block, addresses U and W indicates Δu and Δw respectively.
- 2 The cycle machining is performed by G73 command with P and Q specification. The four cutting patterns are considered. Take care of the sign of Δu , Δw , Δk , and Δi . When the machining cycle is terminated, the tool returns to point A.
- 3 Nose radius compensation is disabled during cycle operation. When the imaginary tool nose number is 0 or 9, however, a nose radius compensation value is added to U and W.

13.2.4 Finishing Cycle (G70)

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

Format

G70P (ns) Q (nf) ; U (Δu) W (Δw) ;

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

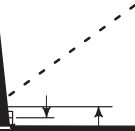
(nf) : Sequence number of the last block for the program of finishing shape.

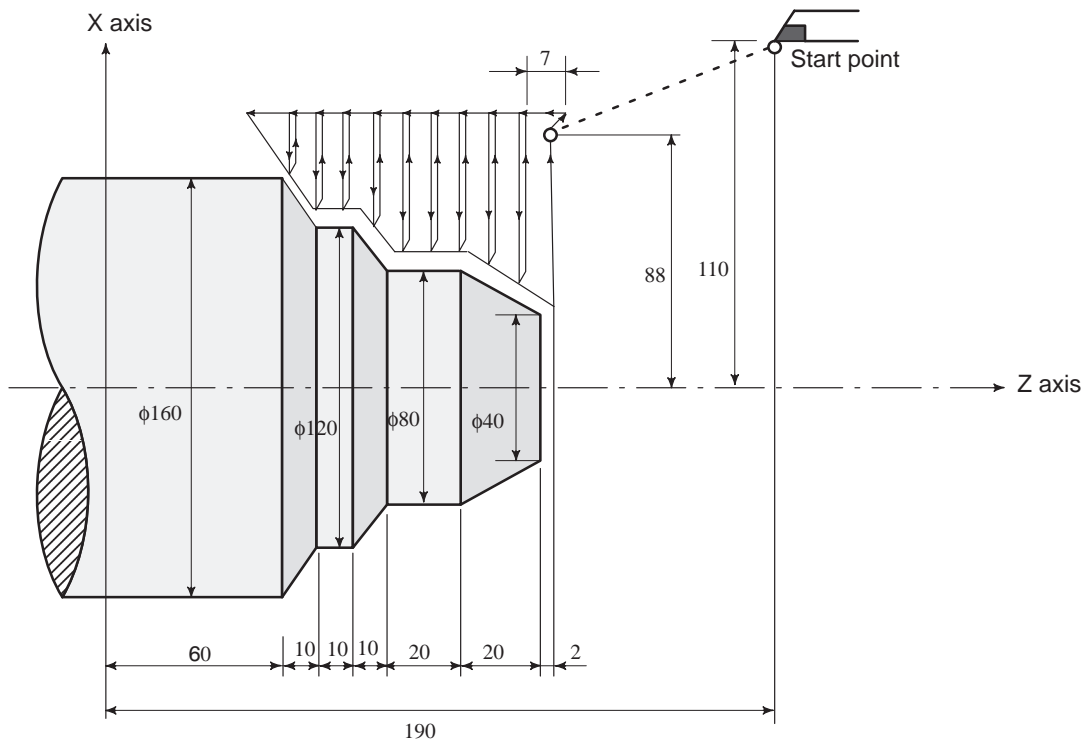
(Δu) :Cutting allowance in X direction (diameter/radius specification)

(Δw) :Cutting allowance in Z direction

NOTE

- 1 F, S, and T functions specified in the block G71, G72, G73 are not effective but those specified between sequence numbers "ns" and "nf" are effective in G70.
- 2 When the cycle machining by G70 is terminated, the tool is returned to the start point and the next block is read.
- 3 In blocks between "ns" and "nf" referred in G70 through G73, the subprogram cannot be called.
- 4 Nose radius compensation is enabled during a finishing cycle.
- 5 Specify cutting allowances U and W when performing finish machining that involves cutting a finishing allowance in several steps.



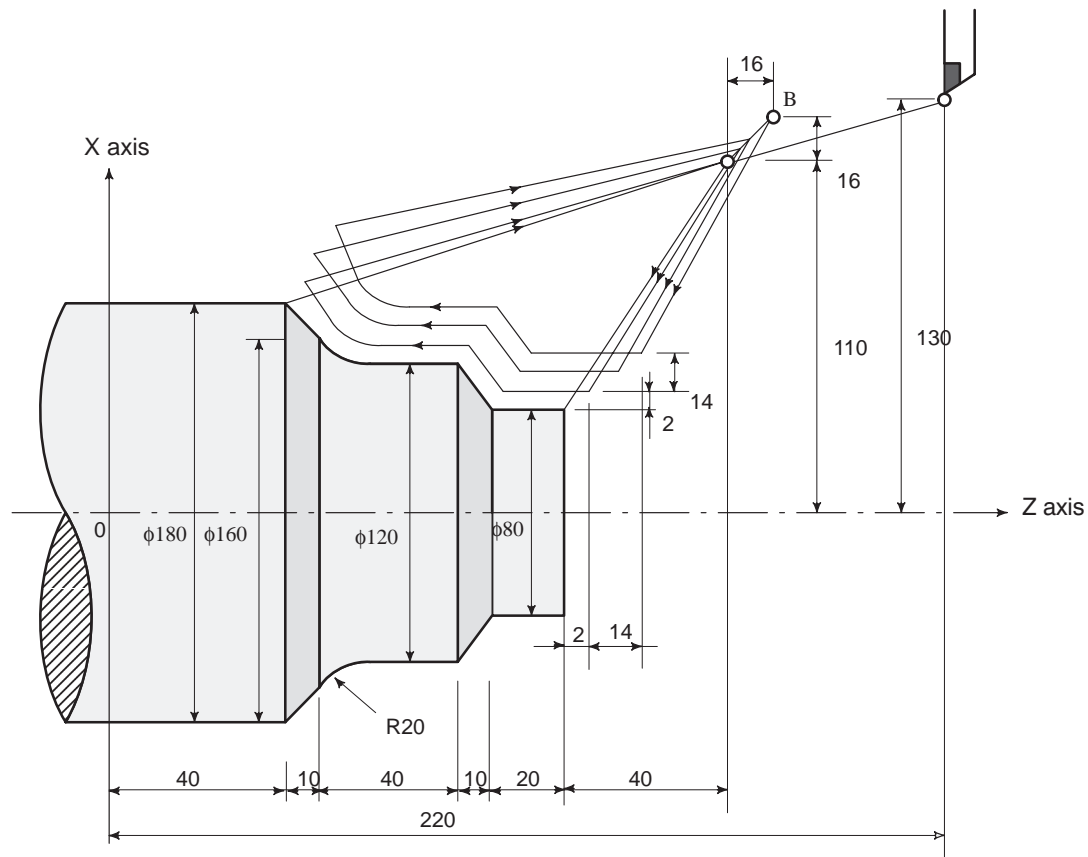
Stock Removal In Facing (G72)

(Diameter designation, metric input)

```

N010 G50 X220.0 Z190.0 ;
N011 G00 X176.0 Z132.0 ;
N012 G72 W7.0 R1.0 ;
N013 G72 P014 Q019 U4.0 W2.0 F0.3 S550 ;
N014 G00 Z58.0 S700 ;
N015 G01 X120.0 W12.0 F0.15 ;
N016     W10.0 ;
N017     X80.0 W10.0 ;
N018     W20.0 ;
N019     X36.0 W22.0 ;
N020 G70 P014 Q019 ;

```

Pattern Repeating (G73)

(Diameter designation, metric input)

```

N010 G50 X260.0 Z220.0 ;
N011 G00 X220.0 Z160.0 ;
N012 G73 U14.0 W14.0 R3 ;
N013 G73 P014 Q019 U4.0 W2.0 F0.3 S0180 ;
N014 G00 X80.0 W-40.0 ;
N015 G01 W-20.0 F0.15 S0600 ;
N016 X120.0 W-10.0 ;
N017 W-20.0 S0400 ;
N018 G02 X160.0 W-20.0 R20.0 ;
N019 G01 X180.0 W-10.0 S0280 ;
N020 G70 P014 Q019 ;

```


13.2.5 End Face Peck Drilling Cycle (G74)

The following program generates the cutting path shown in Fig. 13.2.5. Chip breaking is possible in this cycle as shown below. If X (3) and Pare omitted, operation only in the Z axis results, to be used for drilling.

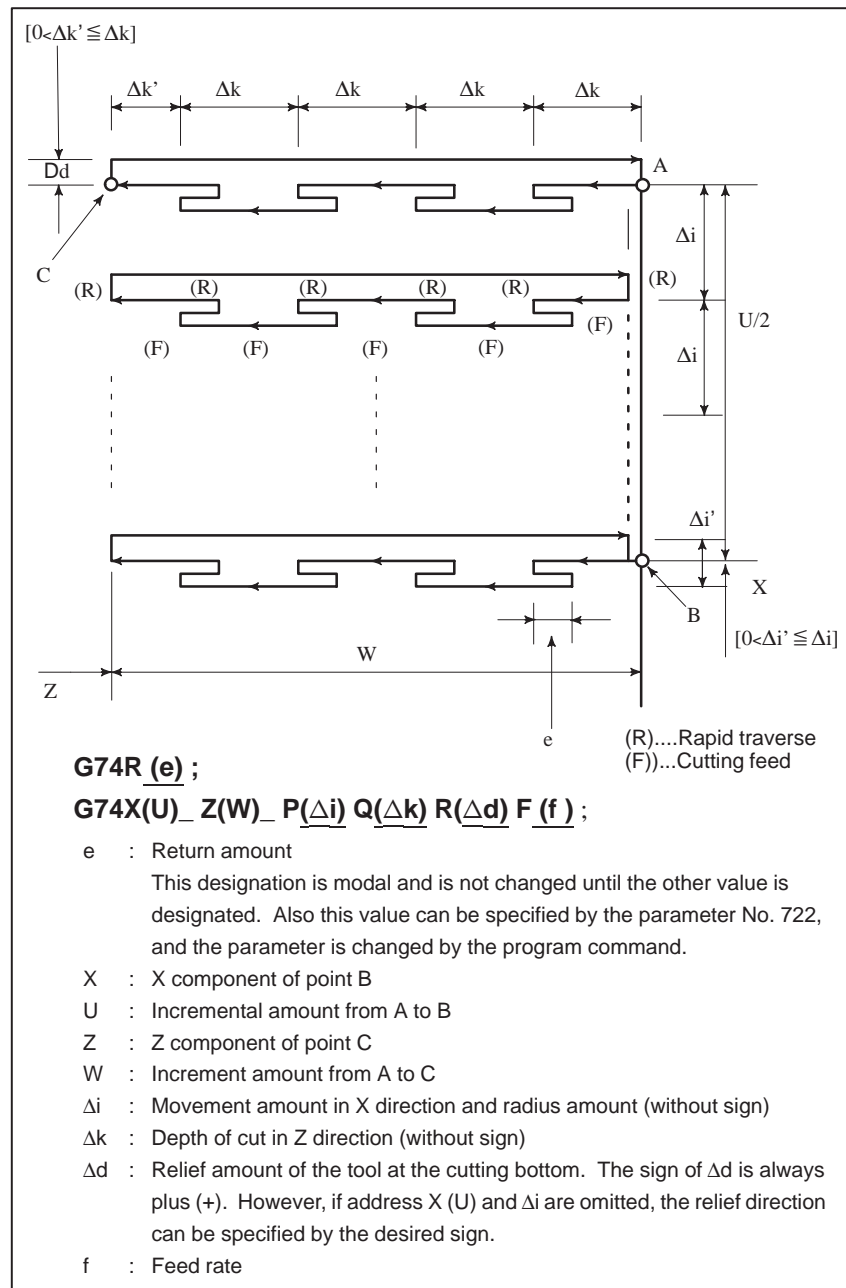


Fig. 13.2.5 Cutting Path in End Face Peck Drilling Cycle

NOTE

- 1 While both e and Δd are specified by address R, the meanings of them are determined by the present of address X (U). When X(U) is specified, Δd is used.
- 2 The cycle machining is performed by G74 command with X (U) specification.

13.2.6 Outer Diameter / Internal Diameter Drilling Cycle (G75)

The following program generates the cutting path shown in Fig. 13.2.6. This is equivalent to G74 except that X is replaced by Z. Chip breaking is possible in this cycle, and grooving in X axis and peck drilling in X axis (in this case, Z, W, and Q are omitted) are possible.

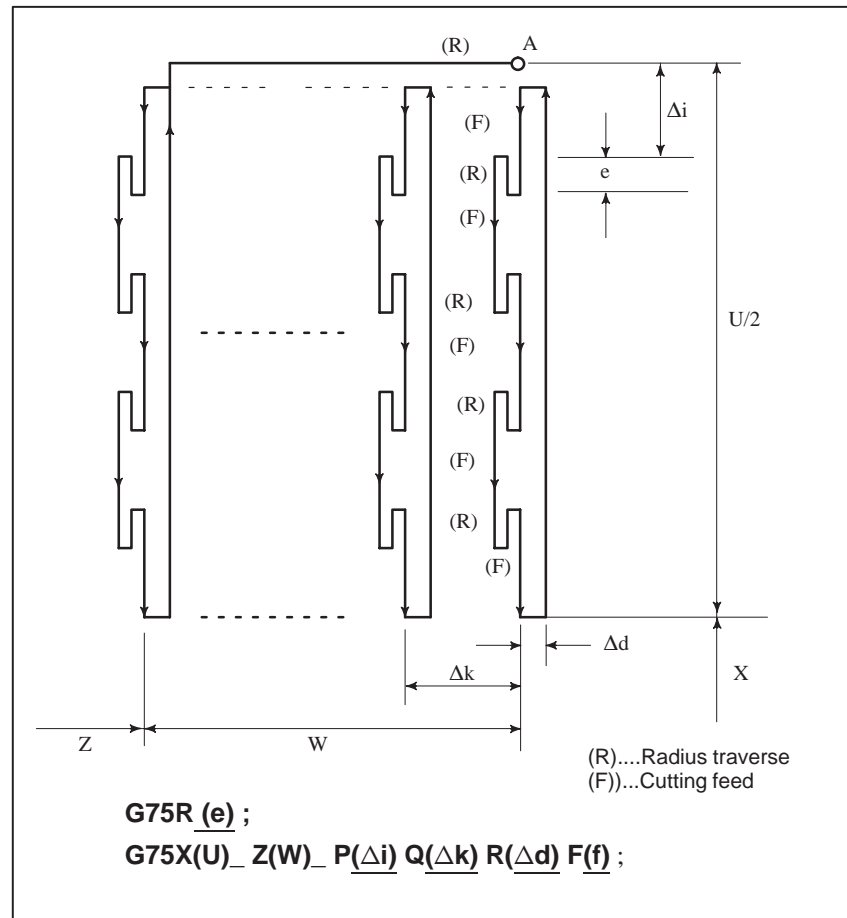


Fig. 13.2.6 Cutting Path in Outer Diameter / Internal Diameter Drilling Cycle

Both G74 and G75 are used for grooving and drilling, and permit the tool to relief automatically. Four symmetrical patterns are considered, respectively.

13.2.7 Multiple Thread Cutting Cycle (G76)

The thread cutting cycle as shown in Fig.13.2.7 (a) is programmed by the G76 command.

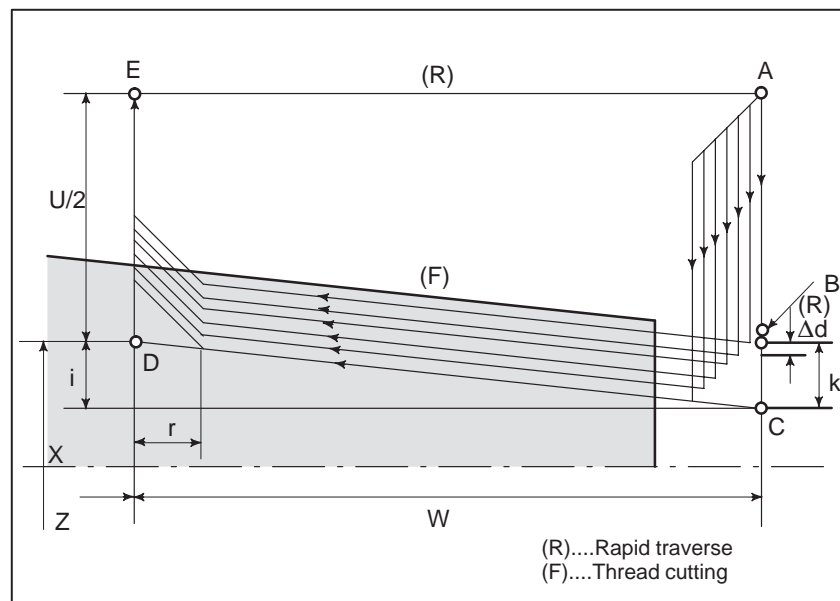
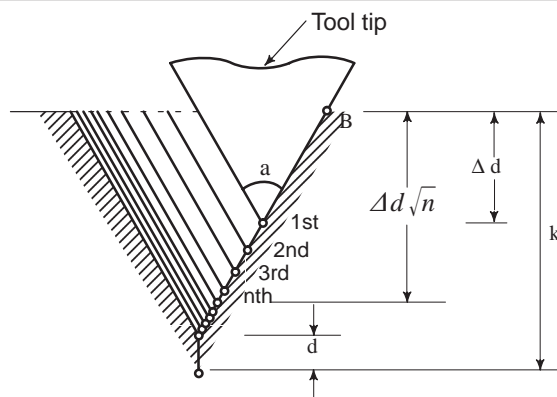


Fig. 13.2.7 (a) Cutting Path in Multiple thread cutting cycle



G76P (m) (r) (a) Q (Δd min) R(d);
G76X (u) _ Z(W) _ R(i) P(k) Q(Δd) F(L) ;

m ; Repetitive count in finishing (1 to 99)

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 723, and the parameter is changed by the program command.

r : Chamfering amount

When the thread lead is expressed by L, the value of L can be set from 0.0L to 9.9L in 0.1L increment (2-digit number from 00 to 90).

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 109, and the parameter is changed by the program command.

a : Angle of tool tip

One of six kinds of angle, 80°, 60°, 55°, 30°, 29°, and 0°, can be selected, and specified by 2-digit number.

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 724, and the parameter is changed by the program command.

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

(Example)

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

P $\frac{02}{m}$ $\frac{12}{r}$ $\frac{60}{a}$

Δdmin : Minimum cutting depth (specified by the radius value)

When the cutting depth of one cycle operation ($\Delta d - \Delta d - 1$) becomes smaller than this limit, the cutting depth is clamped at this value. This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter No. 725, and the parameter is changed by the program command.

d : Finishing allowance (Command with radius amount)

This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter No. 726, and the parameter is changed by the program command.

i : Taper value command with radius amount

If i = 0, ordinary straight thread cutting can be made.

k : Height of thread

This value is specified by the radius value.

Δd : Depth of cut in 1st cut (Command with radius amount)

L : Lead of thread (same as G32).

Fig. 13.2.7 (b) Detail of cutting

- **Thread cutting cycle retract**

When feed hold is applied during threading in the multiple thread cutting cycle (G76), the tool quickly retracts in the same way as in chamfering performed at the end of the thread cutting cycle. The tool goes back to the start point of the cycle. When cycle start is triggered, the multiple thread cutting cycle resumes.

Without this retraction function, when feed hold is applied during threading, the tool goes back to the start point of the cycle after threading is completed.

See notes in 13.1.2.

NOTE

1 The meanings of the data specified by address P, Q, and R determined by the presence of X (U) and X (W).

2 The cycle machining is performed by G76 command with X (U) and Z (W) specification.

By using this cycle, one edge cutting is performed and the load on the tool tip is reduced.

Making the cutting depth Δd for the first path, and $\Delta d_n = \Delta d \sqrt{n}$ for the nth path, cutting amount per one cycle is held constant.

Four symmetrical patterns are considered corresponding to the sign of each address.

The internal thread cutting is available. In the above figure, the feed rate between C and D is specified by address F, and in the other path, at rapid traverse. The sign of incremental dimensions for the above figure is as follows :

U, W : minus (determined by the direction of the tool path AC and CD.)

i : minus (determined by the direction of the tool path AC.)

k : plus (always)

d : plus (always)

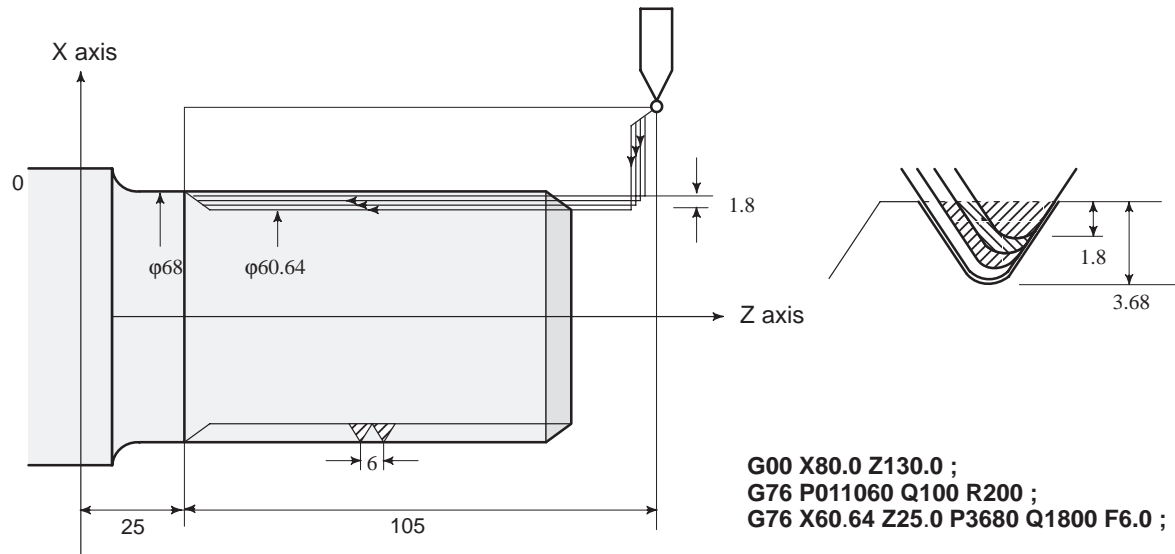
3 Notes on thread cutting are the same as those on G32 thread cutting and G92 thread cutting cycle.

4 The designation of chamfering is also effective for G92 thread cutting cycle.

5 The tool returns to the cycle start point at that time (cutting depth Δd_n) as soon as the feed hold status is entered during thread cutting when the "Thread Cutting Cycle retract" option is used.

Examples

Multiple repetitive cycle (G76)



13.2.8

Notes on Multiple Repetitive Cycle (G70 to G76)

1. In the blocks where the multiple repetitive cycle are commanded, the addresses P, Q, X, Z, U, W, and R should be specified correctly for each block.
2. In the block which is specified by address P of G71, G72 or G73, G00 or G01 group should be commanded. If it is not commanded, P/S alarm No.65 is generated.
3. In MDI mode, G70, G71, G72, or G73 cannot be commanded. If it is commanded, P/S alarm No. 67 is generated. G74, G75, and G76 can be commanded in MDI mode.
4. In the blocks in which G70, G71, G72, or G73 are commanded and between the sequence number specified by P and Q, M98 (subprogram call) and M99 (subprogram end) cannot be commanded.
5. In the blocks between the sequence number specified by P and Q, the following commands cannot be specified.
 - One shot G code except for G04 (dwell)
 - 01 group G code except for G00, G01, G02, and G03
 - 06 group G code
 - M98 / M99
6. While a multiple repetitive cycle (G70 to G76) is being executed, it is possible to stop the cycle and to perform manual operation. But, when the cycle operation is restarted, the tool should be returned to the position where the cycle operation is stopped.
If the cycle operation is restarted without returning to the stop position, the movement in manual operation is added to the absolute value, and the tool path is shifted by the movement amount in manual operation.
7. When G70, G71, G72, or G73 is executed, the sequence number specified by address P and Q should not be specified twice or more in the same program.
8. Do not program so that the final movement command of the finishing shape block group designated with P and Q for G70, G71, G72, and G73 finishes with chamfering or corner rounding. If it is specified, P/S alarm No. 69 is generated.

13.3 CANNED CYCLE FOR DRILLING (G80 TO G89)

The canned cycle for drilling simplifies the program normally by directing the machining operation commanded with a few blocks, using one block including G function.

This canned cycle conforms to JIS B 6314.

Following is the canned cycle table.

Table 13.3 (a) Canned Cycles

G code	Drilling axis	Hole machining operation (– direction)	Operation in the bottom hole position	Retraction operation (+ direction)	Applications
G80	—	—	—	—	Cancel
G83	Z axis	Cutting feed / intermittent	Dwell	Rapid traverse	Front drilling cycle
G84	Z axis	Cutting feed	Dwell→spindle CCW	Cutting feed	Front tapping cycle
G85	Z axis	Cutting feed	—	Cutting feed	Front boring cycle
G87	X axis	Cutting feed / intermittent	Dwell	Rapid traverse	Side drilling cycle
G88	X axis	Cutting feed	Dwell→Spindle CCW	Cutting feed	Side tapping cycle
G89	X axis	Cutting feed	Dwell	Cutting feed	Side boring cycle

Explanations

In general, the drilling cycle consists of the following six operation sequences.

- Operation 1 Positioning of X (Z) and C axis
C–axis clamp M code output (as required)
- Operation 2 Rapid traverse up to point R level
C–axis clamp M code output (as required)
- Operation 3 Hole machining
- Operation 4 Operation at the bottom of a hole
- Operation 5 Retraction to point R level
- Operation 6 C–axis unclamp M code output and dwell (as required)
- Operation 7 Rapid traverse to initial point

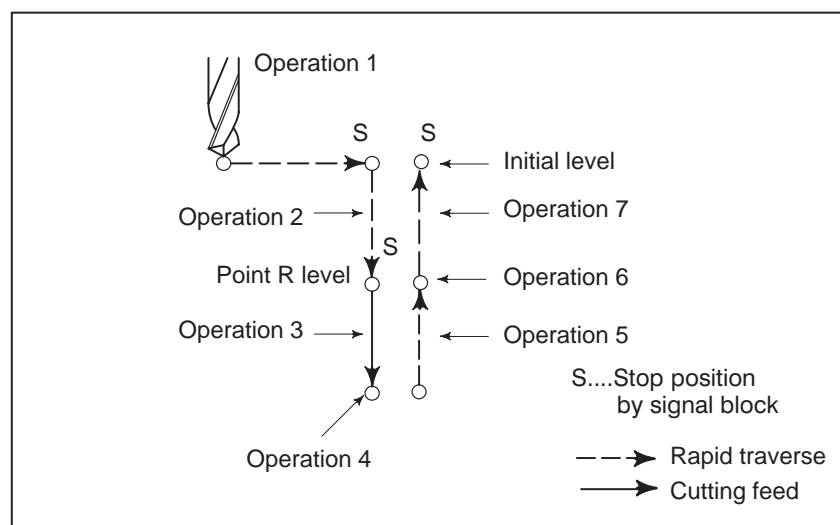


Fig. 13.3 Drilling Cycle Operation Sequence

- **Positioning axis and drilling axis**

A drilling G code specifies positioning axes and a drilling axis as shown below. The C-axis and X- or Z-axis are used as positioning axes. The X- or Z-axis, which is not used as a positioning axis, is used as a 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.

Table 13.3(b) Positioning axis and drilling axis

G code	Positioning plane	Drilling axis
G83, G84, G85	X axis, C axis, Y axis	Z axis
G87, G88, G89	Z axis, C axis, Y axis	X axis

G83 and G87, G84 and G88, and G85 and G89 have the same function respectively except for axes specified as positioning axes and a drilling axis.

- **Drilling mode**

G83 to G85 / G87 to 89 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.

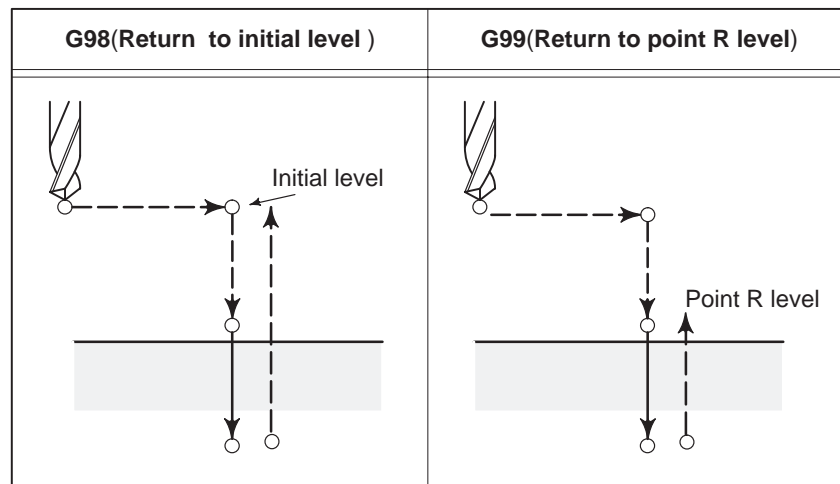
Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

- **Return point level
G98/G99**

In G code system A, the tool returns to the initial level from the bottom of a hole. In G code system B or C, specifying G98 returns the tool to the initial level from the bottom of a hole and specifying G99 returns the tool to the point-R level from the bottom of a hole.

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.



- **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.

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

Number of repeats K The maximum command value = 9999

If no specification is made for K, K = 1 is assumed.

- **M code used for C-axis clamp/unclamp**

When an M code specified in parameter No.204 for C-axis clamp / unclamp is coded in a program, the CNC issues the M code (parameter No.204) for C-axis clamp after the tool is positioned and before the tool is fed in rapid traverse to the point-R level. The CNC also issues the M code (parameter No. 204+1) for C-axis unclamp after the tool retracts to the point-R level. The tool dwells for the time specified in parameter No. 591.

If an M code other than the clamp M code set in parameter No. 204 is specified, the M code is output in operation 1.

- **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 (CW)

G03 : Circular 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)
P1	Dwell specified in the program
P2	Dwell specified in parameter No.591
Mα	Issuing the M code for C-axis clamp
Mβ	Issuing the M code for C-axis unclamp

13.3.1

Front Drilling Cycle (G83) / Side Drilling Cycle (G87)

- **High-speed peck drilling cycle (G83, G87)**
(parameter No. 031#4=0)

The peck drilling cycle or high-speed peck drilling cycle is used depending on the setting in RTCT, bit 2 of parameter No. 031#4. If depth of cut for each drilling is not specified, the normal drilling cycle is used.

This cycle performs high-speed peck drilling. The drill repeats the cycle of drilling at the cutting feedrate and retracting the specified retraction distance intermittently to the bottom of a hole. The drill draws cutting chips out of the hole when it retracts.

Format

G83 X(U)_ C(H)_ Z(W)_ R_ Q_ P_ F_ M_ ;

or

G87 Z(W)_ C(H)_ X(U)_ R_ Q_ P_ F_ M_ ;

X_ C_ or Z_ C_ : Hole position data

Z_ or X_ : Bottom of the hole

W_ or U_ : 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

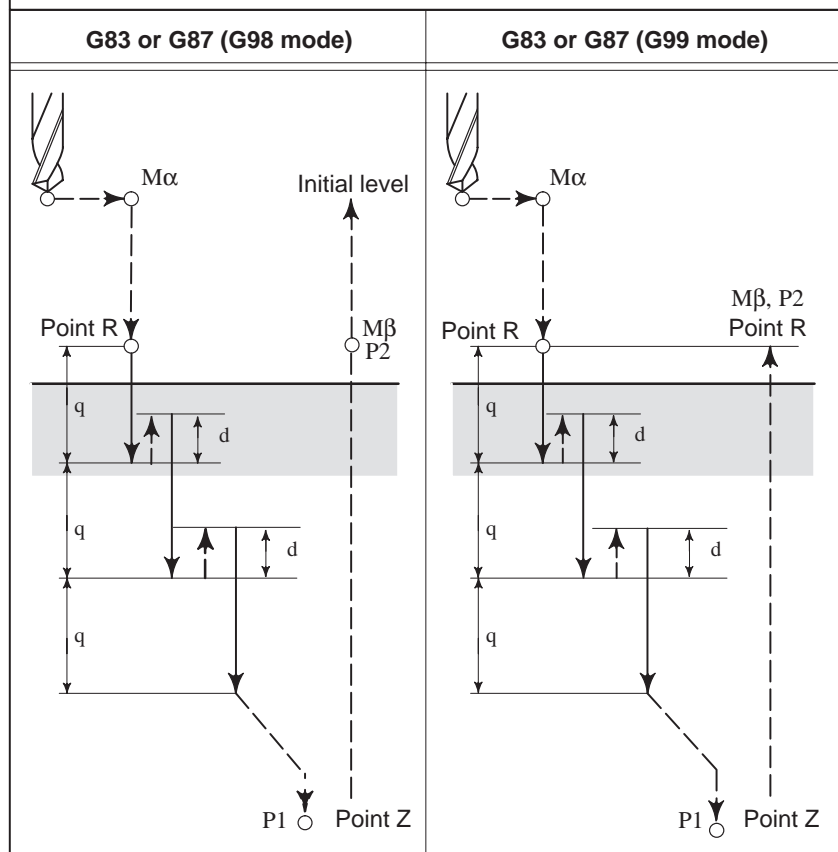
P_ : Dwell time at the bottom of a hole

F_ : Cutting feedrate

K_ : Number of repeats (When it is needed)

M_ : M code for C-axis clamp (When it is needed.)

d_ : Clearance specified with parameter No. 592



Format

G83 or G87 (G98 mode)	G83 or G87 (G99 mode)
<p>Diagram illustrating the G83 or G87 (G98 mode) drilling cycle. The cycle starts at the Initial level, moves down to Point R, then to a depth 'q' below Point R. It then moves up to a depth 'd' below Point R, and repeats this cycle. The final depth is Point Z. The diagram is labeled with $M\alpha$, Initial level, Point R, $M\beta$, P2, q, d, and Point Z.</p>	<p>Diagram illustrating the G83 or G87 (G99 mode) drilling cycle. The cycle starts at the Initial level, moves down to Point R, then to a depth 'q' below Point R. It then moves up to a depth 'd' below Point R, and repeats this cycle. The final depth is Point Z. The diagram is labeled with $M\alpha$, Point R, $M\beta$, P2, Point R, q, d, and Point Z.</p>

Setting C-axis index mode ON
Rotating the drill
Drilling hole 1
Drilling hole 2
Drilling hole 3
Drilling hole 4
Canceling the drilling cycle and
stopping drill rotation
Setting C-axis index mode off

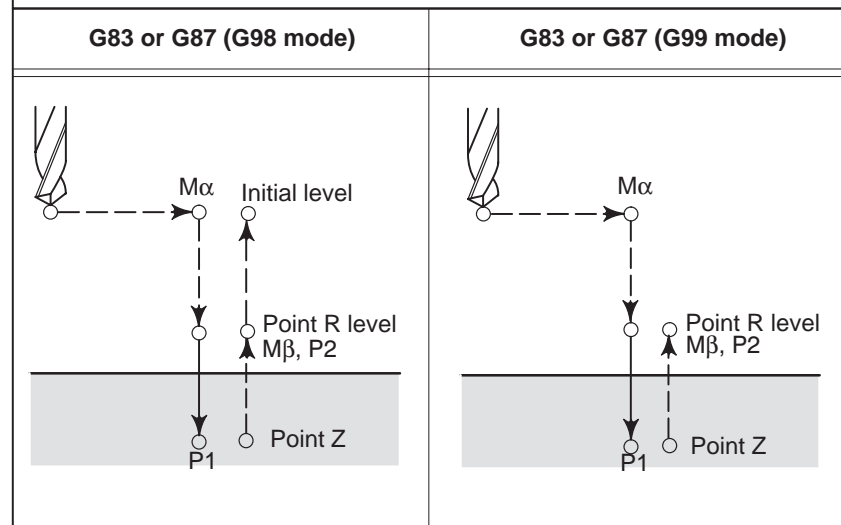
- **Drilling cycle
(G83 or G87)**

If depth of cut is not specified for each drilling, the normal drilling cycle is used. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

G83 X(U)_ C(H)_ Z(W)_ R_ P_ F_ M_ ;
or
G87 Z(W)_ C(H)_ X(U)_ R_ P_ F_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : Bottom of the hole
W_ or X_ : 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 at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C-axis clamp (When it is needed.)



Examples

M51 ;
M3 S2000 ;
G83 X50.0 C0.0 Z-40.0 R-5.0 P500 F5.0 M31 ;
C90.0 M31 ;
C180.0 M31 ;
C270.0 M31 ;
G80 M05 ;

M50 ;

Setting C-axis index mode ON

Rotating the drill

Drilling hole 1

Drilling hole 2

Drilling hole 3

Drilling hole 4

Canceling the drilling cycle and
stopping drill rotation

Setting C-axis index mode off

13.3.2

Front Tapping Cycle (G84) / Side Tapping Cycle (G88)

Format

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.

<div><div><div>G84 X(U)_ C(H)_ Z(W)_ R_ P_ F_ M_ ;</div><div>or</div><div>G88 Z(W)_ C(H)_ X(U)_ R_ P_ F_ M_ ;</div></div><div><div>X_ C_ or Z_ C_ : Hole position data</div><div>Z_ or X_ : Bottom of the hole</div><div>W_ or U_ : 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 feedrate</div><div>K_ : Number of repeats (When it is needed.)</div><div>M_ : M code for C-axis clamp (when it is needed.)</div></div></div>	
G84 or G88 (G98 mode)	G84 or G88 (G99 mode)

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.

NOTE

Bit 2 of parameter No. 031 specifies whether the spindle stop command (M05) is issued before the direction in which the spindle rotates is specified with M03 or M04. For details, refer to the operator’s manual created by the machine tool builder.

Examples

M51 ;	Setting C-axis index mode ON
M3 S2000 ;	Rotating the drill
G84 X50.0 C0.0 Z-40.0 R-5.0 P500 F5.0 M31 ;	Drilling hole 1
C90.0 M31 ;	Drilling hole 2
C180.0 M31 ;	Drilling hole 3
C270.0 M31 ;	Drilling hole 4
G80 M05 ;	Canceling the drilling cycle and stopping drill rotation
M50 ;	Setting C-axis index mode off

13.3.4 Canned Cycle for Drilling Cancel (G80)

G80 cancels canned cycle.

Format

G80 ;

Explanations

Canned cycle for drilling is canceled to perform normal operation. Point R and point Z are cleared. Other drilling data is also canceled (cleared).

Examples

M51 ;	Setting C-axis index mode ON
M3 S2000 ;	Rotating the drill
G83 X50.0 C0.0 Z-40.0 R-5.0 P500 F5.0 M31 ;	Drilling hole 1
C90.0 M31 ;	Drilling hole 2
C180.0 M31 ;	Drilling hole 3
C270.0 M31 ;	Drilling hole 4
G80 M05 ;	Canceling the drilling cycle and stopping drill rotation
M50 ;	Setting C-axis index mode off

13.3.5 Precautions to be Taken by Operator

- **Reset and emergency stop**

Even when the controller is stopped by resetting or emergency stop in the course of drilling cycle, the drilling mode and drilling data are saved ; with this mind, therefore, restart operation.

- **Single block**

When drilling cycle is performed with a single block, the operation stops at the end points of operations 1, 2, 7 in Fig. 13.3. Consequently, it follows that operation is started up 3 times to drill one hole. The operation stops at the end points of operations 1, 2 with the feed hold lamp ON. The operation stops in the feed hold conditions at the end point of operation 6 if the repeat remains, and it stops in stop conditions in other cases.

- **Feed hold**

When "Feed Hold" is applied between operations 3 and 5 by G84/G88, the feed hold lamp lights up immediately if the feed hold is applied again to operation 7.

- **Override**

During operation with G84 and G88, the feedrate override is 100%.

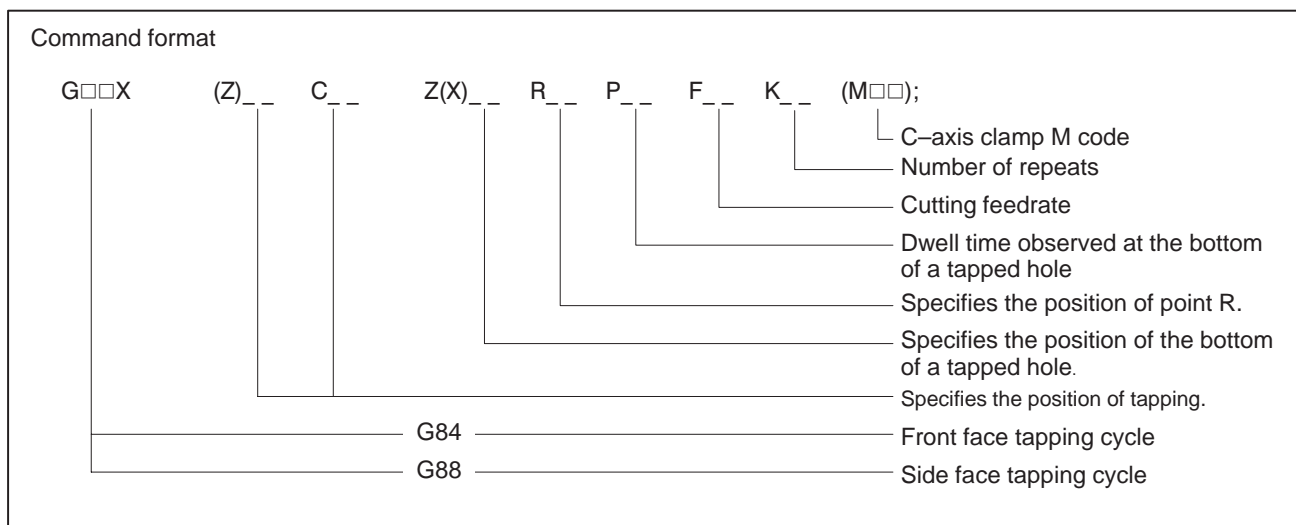
13.3.6 Rigid Tapping

Tapping cycles (G84, G88) are classified into floating tapping and rigid tapping cycles. As explained in Section 13.3.2, floating tapping is the conventional tapping method. In floating tapping, the spindle is rotated either clockwise or counterclockwise, synchronized with the motion along the Z-axis (X-axis) according to miscellaneous functions M03 (spindle CW rotation), M04 (spindle CCW rotation), and M05 (spindle stop).

In rigid tapping, the spindle motor is controlled in the same way as a control motor by distributing pulses to instigate motion along the Z-axis (X-axis) and of the spindle.

In rigid tapping, compensation is applied to motion along the Z-axis (X-axis) and spindle, so that each turn of the spindle corresponds to a certain amount of feed (screw lead) along Z-axis (X-axis). This also applies to acceleration/deceleration. This means that rigid tapping does not demand the use of special tappers as in the case of floating tapping, thus enabling high-speed, high-precision tapping.

This text assumes the use of G-code system B.



Rigid tapping can be specified in any of three ways. In the first method, M29S**** is specified before the specification of a tapping cycle. In the second method, M29S**** is specified in the block specifying a tapping cycle. The third method allows rigid tapping to be performed without specifying M29S****. When using the last method, specify S**** either before or in a G84 (G88) block.

With the specification of rigid tapping, the spindle first stops, then the specified tapping cycle performs rigid tapping. M29 is referred to as the rigid tapping miscellaneous function. By setting a desired value in parameter No. 0253, a separate M code can be assigned to the rigid tapping miscellaneous function. This manual, however, uses M29.

Rigid tapping can be canceled by executing G80;. However, a separate canned cycle G code or group 01 G code, that is, a command for canceling a tapping cycle, can also be used to cancel rigid tapping.

When a tapping cycle is terminated by executing a rigid tapping cancellation command, the spindle stops. (The spindle analog voltage changes to the same level as that set when S0 is specified.)

Rigid tapping can also be canceled by a reset (reset button or external reset). Note, however, that canned cycle mode cannot be canceled by a reset.

(1) Specifying M29 before a G84 (G88) block

```

      ;
      M29S****;
      G□□X(Z) _ C _ Z(X) _ R _ P _ F _ K _ (M□□);
      X(Z) _ C _ ;
      X(Z) _ C _ ;
      ;
      G80;
      ;
  
```

Rigid tapping mode

(2) Specifying M29 in a G84 (G88) block (In this case, M29 and M for C-axis clamping cannot be specified in the same block.)

```

      ;
      G□□X(Z) _ Z(X) _ R _ P _ F _ K _ M29S****;
      X(Z) _ C _ ;
      X(Z) _ C _ ;
      ;
      G80;
      ;
  
```

Rigid tapping mode

(3) Handling G84 (G88) as a G code for rigid tapping (Set bit 3 of parameter No. 076 to 1.)

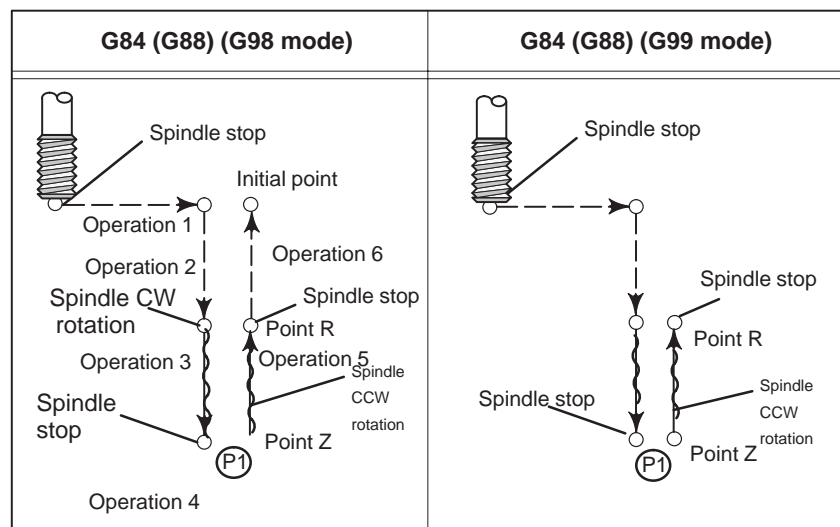
```

      ;
      G□□X(Z) _ C _ Z(X) _ R _ P _ F _ K _ S****(M□□);
      X(Z) _ C _ ;
      X(Z) _ C _ ;
      ;
      G80;
      ;
  
```

Rigid tapping mode

NOTE

- 1 In feed per minute mode, F_/_/S**** specifies a screw lead. In feed per rotation mode, F_/_ specifies a screw lead.
- 2 S**** must specify a value not exceeding those set in parameter No. 0423 to No. 0426. Otherwise, a P/S alarm is issued in a G84 (G88) block.
- 3 F_/_ must specify a value not exceeding the maximum allowable cutting feedrate (set in parameter No. 527). Otherwise, P/S alarm No. 011 is issued.
- 4 Neither S nor motion along an axis must be specified between M29 and G84 (G88). In addition, M29 must not be specified during a tapping cycle. Otherwise, a P/S alarm is issued.
- 5 When both G84 and G88 are specified in rigid tapping mode, a P/S alarm is issued.

G84, G88 (tapping cycle)**NOTE**

- 1 A feedrate override of 100% is assumed for feed by pulse distribution for motion along the Z-axis (X-axis) and the spindle. A spindle override of 100% is also assumed. For retraction (operation 5), however, a fixed override of up to 200% can be applied by setting parameter No. 0254 and bit 4 of parameter No. 063.
- 2 Single block, feed hold Even during rigid tapping, the single block and feed hold functions are enabled. By setting bit 3 of parameter No. 397, these functions can be disabled during rigid tapping, as in the case of conventional tapping.

Examples

Using a feedrate along the Z-axis of 1000 mm/min and a spindle speed of 1000 rpm, threading with a lead of 1 mm can be specified in feed per minute mode, as indicated below.

```
O0001;
G94;

M29 S1000;
G84 Z-100. R-20. F1000.;

G80;
```

In feed per rotation mode, the same threading operation can be specified under the same condition, as indicated below. (In feed per rotation mode, F specifies a screw lead.)

```
O0002;
G95;

M29 S1000;
G84 Z-100. R-20. F1.;

G80;
```

13.3.7 Counter Rigid Tapping

Format

	M29		S <u>Spindle Speed</u>	P1 ;
<input type="checkbox"/>	G84	<input type="checkbox"/>	<u>Rigid Tapping Cycle</u> ;	
<input type="checkbox"/>	G88	<input type="checkbox"/>		
	G80		;	

When P1 is commanded on the same block of M-code (M29) for rigid tapping, the rigid tapping cycle (G84/G88) thereafter is regarded as the counter rigid tapping cycle.

Namely, the spindle rotates in the opposite direction to the ordinary rigid tapping cycle. P1 is valid till the rigid tapping cancel.

NOTE

- 1 The M-code (M29) should be commanded alone.
- 2 Parameter (No.076#3) is not used for the counter rigid tapping.

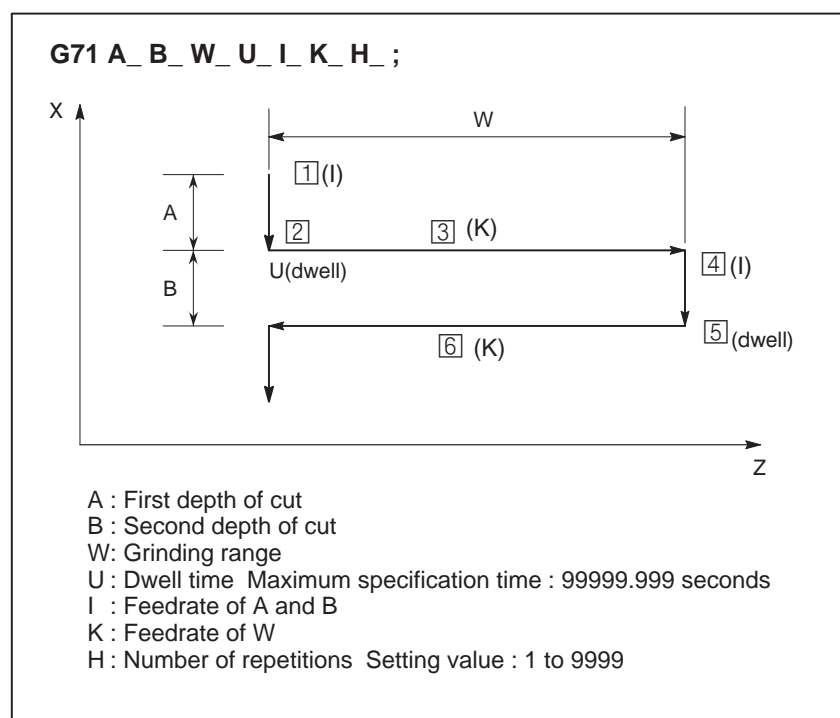
13.4 CANNED GRINDING CYCLE (0-GCC, 00-GCC, 0-GCD/II)

There are four grinding canned cycles : the traverse grinding cycle (G71), traverse direct fixed-dimension grinding cycle, oscillation grinding cycle, and oscillation direct fixed-dimension grinding cycle.

With a machine tool that allows canned cycles for grinding to be used, the multiple repetitive canned cycle for turning cannot be used.

13.4.1 Traverse Grinding Cycle (G71)

Format



Explanations

A, B, and W are to be specified in an incremental mode.

In the case of a single block, the operations 1, 2, 3, 4, 5, and 6 are performed with one cycle start operation.

A=B=0 results in a spark-out.

13.4.2

Traverse Direct Fixed-Dimension Grinding Cycle (G72)

Format

G72 P_ A_ B_ W_ U_ I_ K_ H_ ;

P : Gauge number (1 to 4)
 A : First depth of cut
 B : Second depth of cut
 W : Grinding range
 U : Dwell time Maximum specification time : 99999.999seconds
 I : Feedrate of A and B
 K : Feedrate of W
 H : Number of repetitions Setting value : 1 to 9999

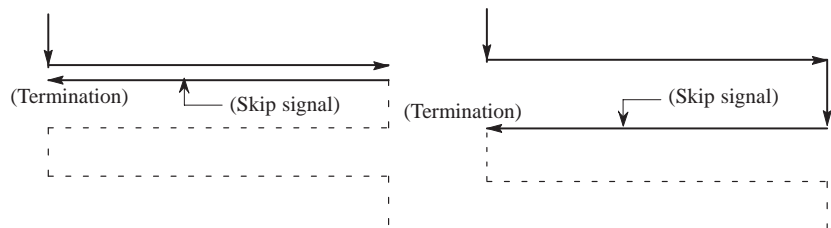
Explanations

When the multistage skip operation is used, a gauge number can be specified. The method of gauge number specification is the same as the method of multistage skip function. When the multistage skip operation is not used, the conventional skip signal is valid.

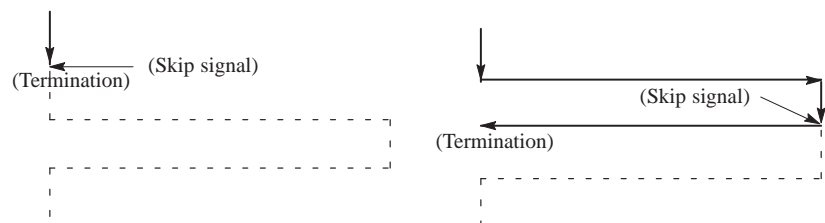
The same specifications as G71 apply except for gauge number specification.

- Operation at the time of skip signal input**

1. When the tool moves along the Z-axis to grind a workpiece, if a skip signal is input, the tool returns to the Z coordinate where the cycle started after the tool reaches the end of the specified grinding area.



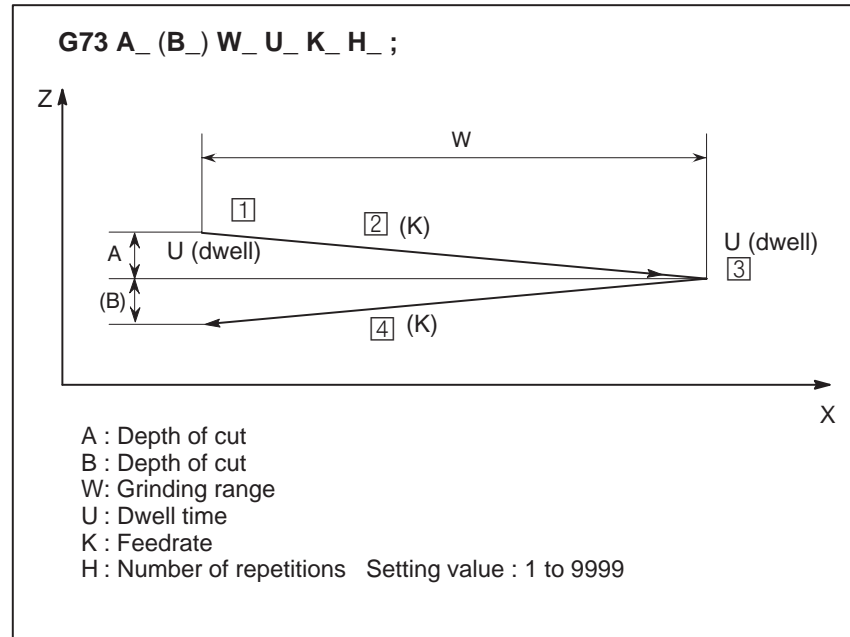
2. When the tool cuts a workpiece along the X-axis, if a skip signal is input, the tool stops cutting immediately and returns to the Z coordinate where the cycle started.



3. The skip signal is valid during dwell, without being affected by parameters DS1 to DS8 (No. 035#0 to #3). Dwell is immediately stopped for return to the Z axis coordinate where the cycle started.

13.4.3 Oscillation Grinding Cycle (G73)

Format



Explanations

A, B, and W are to be specified in an incremental mode.

In the case of a single block, the operations 1, 2, 3, and 4 are performed with one cycle start operation.

The specification of B is valid only for a specified block. This is not associated with B of the G71 or G72 cycle.

13.4.4 Oscillation Direct Fixed-Dimension Grinding Cycle (G74)

Format

G74 P_ A_ (B_) W_ U_ K_ H_ ;

P : Gauge number (1 to 4)

A : Depth of cut

B : Depth of cut

W: Grinding range

U : Dwell time

K : Feedrate of W

H : Number of repetitions Setting value : 1 to 9999

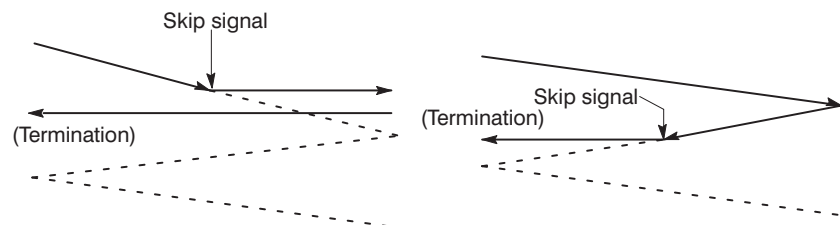
Explanations

When the multistage skip operation is used, a gauge number can be specified. The method of gauge number specification is the same as the method of multistage skip function. When the multistage skip operation is not used, the conventional skip signal is valid.

The same specifications as G73 apply to the other items.

• Operation at the time of skip signal input

1. When the tool moves along the Z-axis to grind a workpiece, if a skip signal is input, the tool returns to the Z coordinate where the cycle started after the tool reaches the end of the specified grinding area.



2. The skip signal is valid during dwell, without being affected by parameters DS1 to DS4 (No. 6035#0 to #3). Dwell is immediately stopped for return to the Z axis coordinate where the cycle started.

NOTE

- 1 The data items A, B, W, U, I, and K in a canned cycle are modal values common to G71 through G74. The data items A, B, W, U, I and K are cleared when a one-shot G code other than G04 or a 01 group G code other than G71 to G74 is specified.
- 2 No B code can be specified in the canned cycle mode.

13.5

CHAMFERING AND CORNER R

- Chamfering
Z → X

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

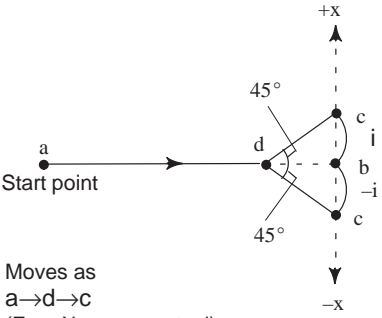
Format	Tool movement
<div>G01Z (W) I (C) $\underline{\pm i}$; Specifies movement to point b with an absolute or incremental command in the figure on the right. Whether to use address I or C to specify chamfering is set using bit 4 of parameter No. 029.</div>	<div> Moves as $a \rightarrow d \rightarrow c$ (For -X movement, -i)</div>

Fig. 13.5 (a) Chamfering (Z→X)

- Chamfering
X → Z

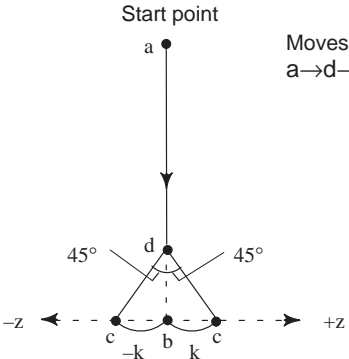
Format	Tool movement
<div>G01X (U) $\underline{\pm k}$ (C) $\underline{\pm k}$; Specifies movement to point b with an absolute or incremental command in the figure on the right. Whether to use address K or C to specify chamfering is set using bit 4 of parameter No. 029.</div>	<div> Moves as $a \rightarrow d \rightarrow c$ (For -Z movement, -k)</div>

Fig. 13.5 (b) Chamfering (X→Z)

- Corner R
Z → X

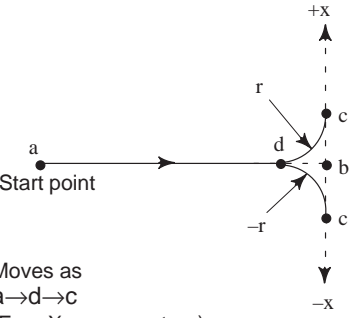
Format	Tool movement
<div>G01Z (W) $\underline{R \pm r}$; Specifies movement to point b with an absolute or incremental command in the figure on the right.</div>	<div> Moves as $a \rightarrow d \rightarrow c$ (For -X movement, -r)</div>

Fig. 13.5 (c) Corner R (Z→X)

- **Corner R**
X → Z

Format	Tool movement
G01X (U) _R \pmr ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	<p>Start point a</p> <p>(For -x movement, -r)</p> <p>Moves as a→d→c</p> <p>-r r</p> <p>-Z +Z</p> <p>c b c</p>

Fig. 13.5 (d) Corner R (X→Z)

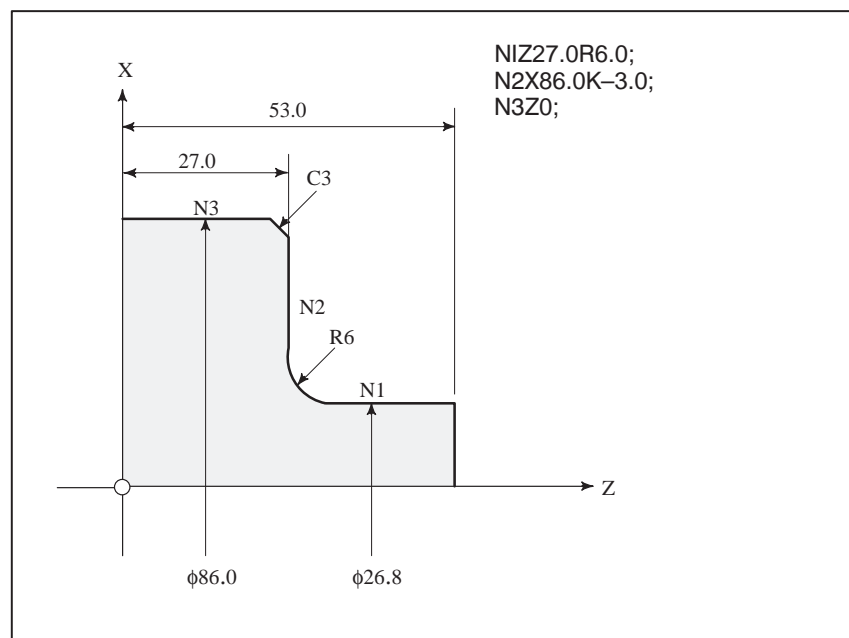
Explanations

The movement for chamfering or corner R must be a single movement along the X or Z axis in G01 mode. The next block must be a single movement along the X or Z axis perpendicular to the former block.

Specification value of chamfering or corner-R is radius value.

Note that the start point for a command specified in a block following a chamfering or corner-R block is not point c but point b shown in Figs. 13.5 (a) to (d). In incremental programming, specify a distance from point b.

Examples



NOTE

- 1 The following commands cause an alarm.
 - 1) Chamfering or corner-R is commanded when X and Z axes are specified by G01. (alarm No. 054)
 - 2) Move amount of X or Z is less than chamfering value and corner R value in the block where chamfering and corner R are specified. (alarm No. 055)
 - 3) Next block to the block where chamfering and corner R were specified, has not G01 command. (alarm No. 051, 052)
- 2 A single block stops at point c of Fig. 13.5 (a) to 13.5 (d), not at point d.
- 3 Chamfering and corner R cannot be applied to a thread cutting block.
- 4 C can be used instead of I or K as an address for chamfering on the system which does not use C as an axis name. To set I/K for an address for chamfering, fix parameter No.029#4 to 1.
- 5 If both C and R are specified with G01 in a block, the address specified last is valid.
- 6 Neither chamfering nor corner-R machining can be specified in direct drawing dimension programming.

13.6

MIRROR IMAGE FOR DOUBLE TURRET (G68, G69)

Format

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

Explanations

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

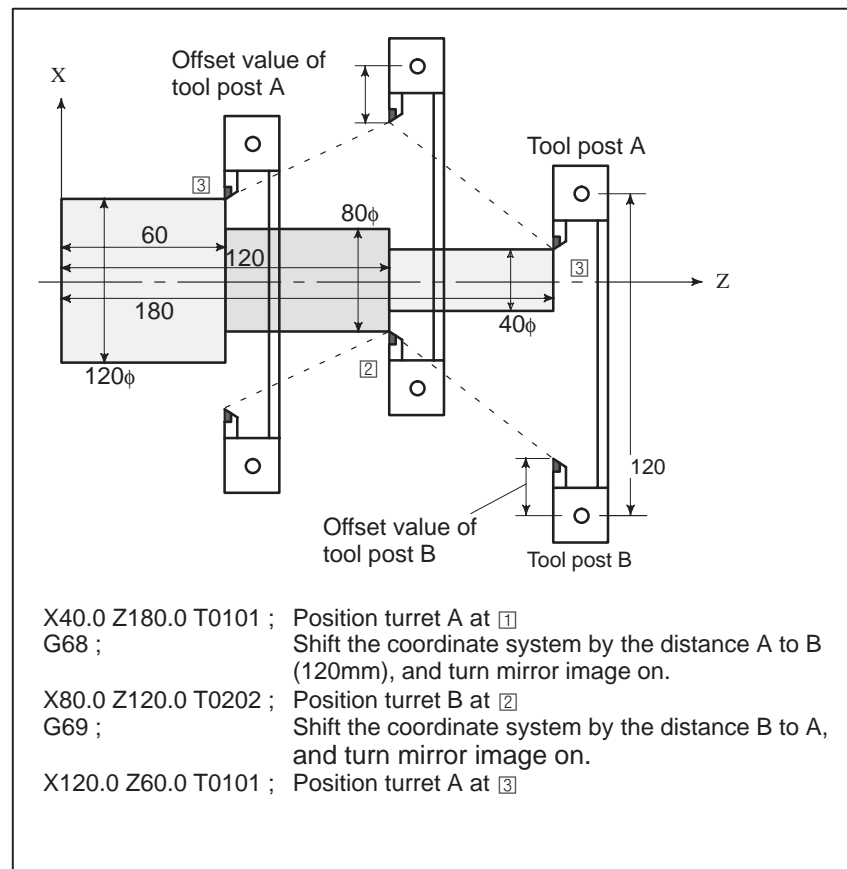
When G68 is designated, the coordinate system is shifted to the mating turret side, and the X-axis sign is reserved from the programmed command to perform symmetrical cutting. This function is referred to as mirror image of facing tool posts. The user can return to the original coordinate system by specifying G69.

A block containing G68 or G69 must not include any other codes.

To use this function, set the distance between the two turrets to a parameter (No. 730).

Examples

- Double turret programming



13.7 DIRECT DRAWING DIMENSIONS PROGRAMMING

Angles of straight lines, chamfering value, corner rounding values, and other dimensional values on machining drawings can be programmed by directly inputting these values. In addition, the chamfering and corner rounding can be inserted between straight lines having an optional angle. This programming is only valid in memory operation mode.

Format

Table 13.7 Commands table

	Commands	Movement of tool
1	$X_{2_} (Z_{2_}) A_{_};$ or $Z_{2_} A_{_};$	
2	$A_{1_};$ $X_{3_} Z_{3_} A_{2_};$	
3	$X_{2_} Z_{2_} R_{1_};$ $X_{3_} Z_{3_};$ or $A_{1_} R_{1_};$ $X_{3_} Z_{3_}, A_{2_};$	
4	$X_{2_} Z_{2_} C_{1_};$ $X_{3_} Z_{3_};$ or $A_{1_} C_{1_};$ $X_{3_} Z_{3_}, A_{2_};$	

	Commands	Movement of tool
5	$X_2_Z_2_R_1_;$ $X_3_Z_3_R_2_;$ $X_4_Z_4_;$ or $A_1_R_1_;$ $X_3_Z_3_A_2_R_2_;$ $X_4_Z_4_;$	
6	$X_2_Z_2_C_1_;$ $X_3_Z_3_C_2_;$ $X_4_Z_4_;$ or $A_1_C_1_;$ $X_3_Z_3_A_2_C_2_;$ $X_4_Z_4_;$	
7	$X_2_Z_2_R_1_;$ $X_3_Z_3_C_2_;$ $X_4_Z_4_;$ or $A_1_R_1_;$ $X_3_Z_3_A_2_C_2_;$ $X_4_Z_4_;$	
8	$X_2_Z_2_C_1_;$ $X_3_Z_3_R_2_;$ $X_4_Z_4_;$ or $A_1_C_1_;$ $X_3_Z_3_A_2_R_2_;$ $X_4_Z_4_;$	

Explanations

A program for machining along the curve shown in Fig. 13.7 is as follows
:

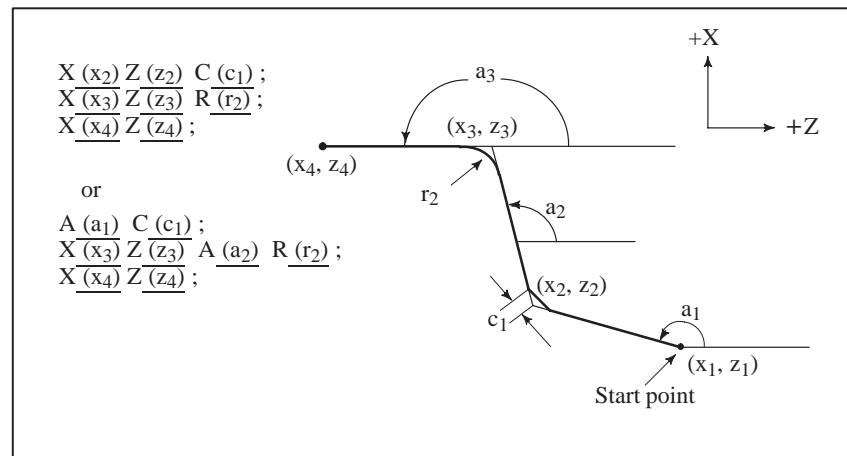


Fig. 13.7 Machining Drawing (example)

For command a straight line, specify one or two out of X, Z, and A.

If only one is specified, the straight line must be primarily defined by a command in the next block.

To command the degree of a straight line or the value of chamfering or corner R, command with following address.

- A_.....Straight line
- C_.....Chamfering
- R_.....Corner R

By specifying 1 to parameter No. 029#4 on the system which use C as an axis name, the degree of a straight line or the value of chamfering or corner R can be commanded with a comma before address.

- , A_.....Straight line
- , C_.....Chamfering
- , R_.....Corner R

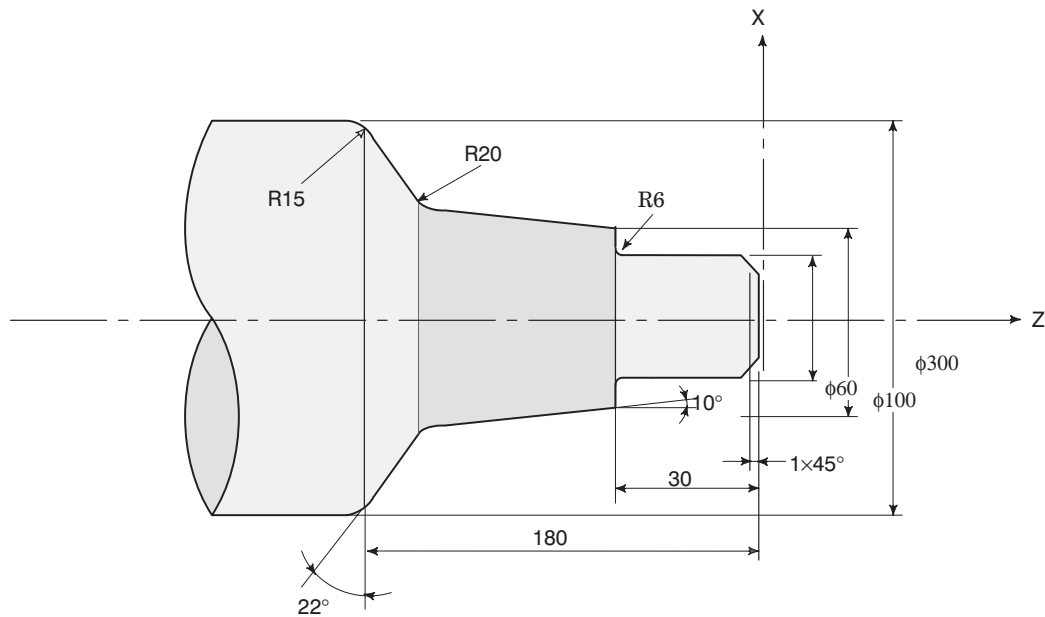
NOTE

- 1 The following G codes are not applicable to the same block as commanded by direct input of drawing dimensions or between blocks of direct input of drawing dimensions which define sequential figures.
 - 1) G codes (other than G04) in group 00.
 - 2) G02, G03, G90, G92, and G94 in group 01.
 - 3) Stock removal in turning (G71) type II
- 2 Corner rounding cannot be inserted into a threading block.
- 3 Neither chamfering and corner rounding commands specified in 13.5 nor those of direct input of drawing dimensions can be used concurrently.
- 4 When the end point of the previous block is determined in the next block according to sequential commands of direct input of drawing dimensions, the single block stop is not done, but the feed hold stop is done at the end point of the previous block.
- 5 The angle allowance in calculating the point of intersection in the program below is $\pm 1^\circ$. (Because the travel distance to be obtained in this calculation is too large.)
 - 1) $X_ , A_ ;$ (If a value within $0^\circ \pm 1^\circ$ or $180^\circ \pm 1^\circ$ is specified for the angle instruction, the alarm No.057 occurs.)
 - 2) $Z_ , A_ ;$ (If a value within $90^\circ \pm 1^\circ$ or $270^\circ \pm 1^\circ$ is specified for the angle instruction, the alarm No. 057 occurs.)
- 6 An alarm occurs if the angle made by the 2 lines is within $\pm 1^\circ$ when calculating the point of intersection.
- 7 Chamfering or corner R is ignored if the angle made by the 2 lines is within $\pm 1^\circ$.
- 8 Both a dimensional command (absolute programming) and angle instruction must be specified in the block following a block in which only the angle instruction is specified.
(Example)


```

N1 X_ A_ R_ ;
N2 A_ ;
N3 X_ Z_ A_ ;
      
```

 (In addition to the dimensional command, angle instruction must be specified in block No. 3.)



(Diameter specification, metric input)

```
N001 G50 X0.0 Z0.0 ;  
N002 G01 X60.0 A90.0 C1.0 F80 ;  
N003 Z-30.0 A180.0 R6.0 ;  
N004 X100.0 A90.0 ;  
N005 A170.0 R20.0 ;  
N006 X300.0 Z-180.0 A112.0 R15.0 ;  
N007 Z-230.0 A180.0 ;  
:  
:  
:
```

14

COMPENSATION FUNCTION



This chapter describes the following compensation functions:

14.1 TOOL OFFSET

14.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION

14.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION

14.4 TOOL COMPENSATION VALUES

14.5 AUTOMATIC TOOL OFFSET

14.1 TOOL OFFSET

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool).

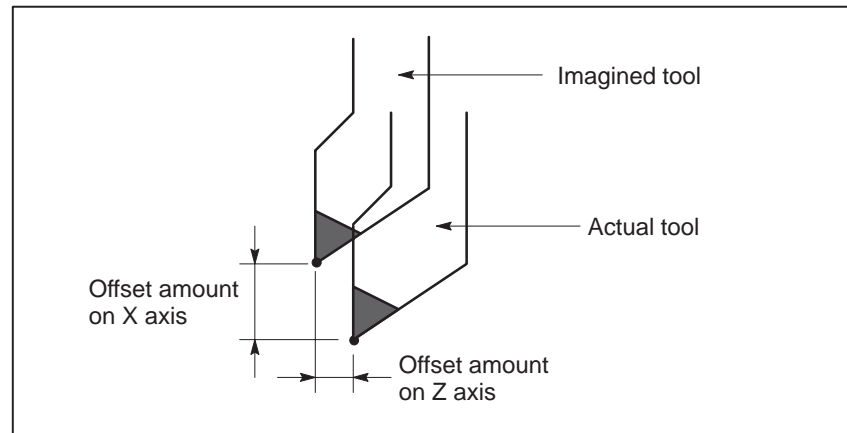


Fig. 14.1 Tool offset

14.1.1 Tool Geometry Offset and Tool Wear Offset

Tool geometry offset is used to compensate for a tool figure or tool attachment position difference. Tool wear offset is used to compensate for tool tip wear. These offset values can be set separately. When no distinction is to be made between these values, set the total of these values as the tool position offset.

NOTE

Tool geometry offset and tool wear offset are optional.

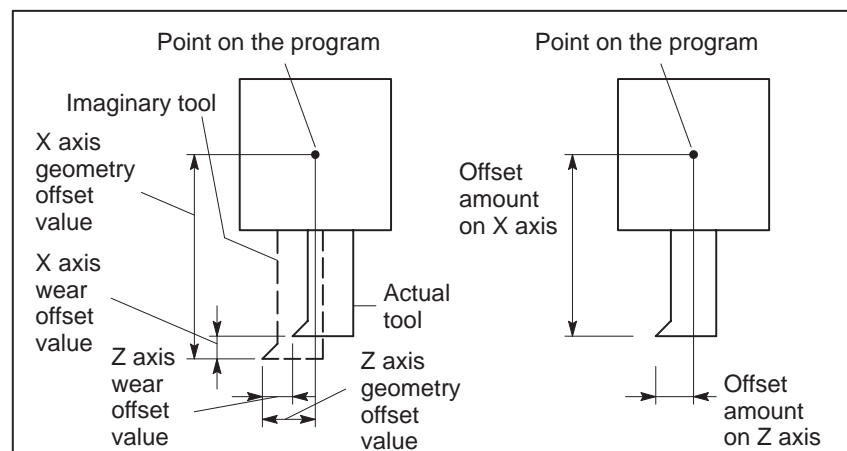


Fig. 14.1.1(a) Difference the tool geometry offset from tool wear offset

Fig. 14.1.1(b) Not difference the tool geometry offset from tool wear offset (The moving is same as tool wear offset)

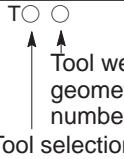
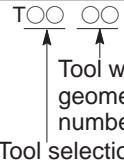
14.1.2 T code for Tool Offset

There are two methods for specifying a T code as shown in Table 14.1.2(a) and Table 14.1.2(b).

Format

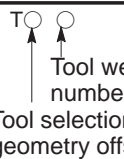
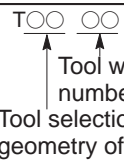
- Lower digit of T code specifies geometry and wear offset number

Table 14.1.2 (a)

Kind of T code	Meaning of T code	Parameter setting for specifying of offset No.	
2-digit command		When bit 0 of parameter No.014, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When bit 1 of parameter No.013, is set to 0, the tool geometry offset number and tool wear offset number specified for a certain tool are the same.
4-digit command		When bit 0 of parameter No.014, is set to 0, a tool wear offset number is specified with the last two digits of a T code.	

- Lower digit of T code specifies wear offset number and higher digit number specifies tool selection number and geometry offset number

Table 14.1.2(b)

Kind of T code	Meaning of T code	Parameter setting for specifying of offset No.	
2-digit command		When bit 0 of parameter No.014, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When bit 1 of parameter No.013, is set to 1, the tool geometry offset number and tool wear offset number specified for a certain tool are the same.
4-digit command		When bit 0 of parameter No.014, is set to 0, a tool wear offset number is specified with the last two digits of a T code.	

14.1.3 Tool Selection

Tool selection is made by specifying the T code corresponding to the tool number. Refer to the machine tool builder's manual for the relationship between the tool selection number and the tool.

14.1.4 Offset Number

Tool offset number has two meanings.

It specifies the offset distance corresponding to the number that is selected to begin the offset function. A tool offset number of 0 or 00 indicates that the offset amount is 0 and the offset is cancelled.

14.1.5 Offset

There are two types of offset. One is tool wear offset and the other is tool geometry offset.

Explanations

- **Tool wear offset**

The tool path is offset by the X, Y, and Z wear offset values for the programmed path. The offset distance corresponding to the number specified by the T code is added to or subtracted from the end position of each programmed block.

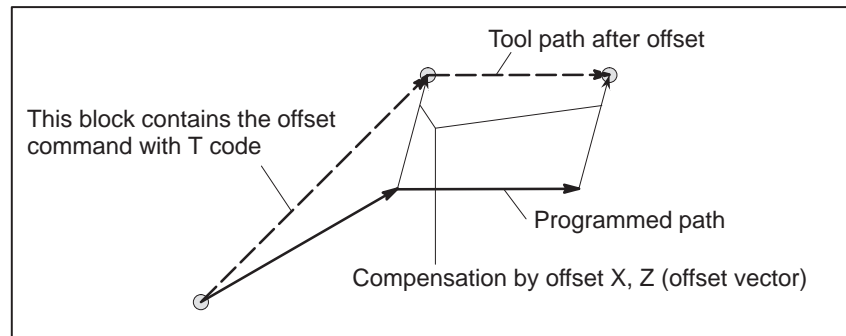


Fig. 14.1.5 (a) Movement of offset (1)

- **Offset vector**

In Fig.15.1.5(a), the vector with offset X, Y, and Z is called the offset vector. Compensation is the same as the offset vector.

- **Offset cancel**

Offset is cancelled when T code offset number 0 or 00 is selected. At the end of the cancelled block, the offset vector becomes 0.

N1 X50.0 Z100.0 T0202 ; Creates the offset vector corresponding to offset number 02

N2 X200.0 ;

N3 X100.0 Z250.0 T0200 ; Specifying offset number 00 deletes the offset vector.

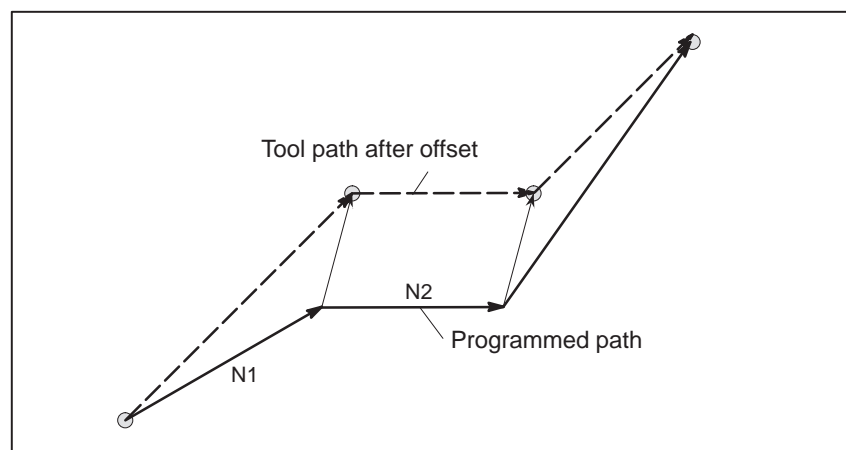


Fig. 14.1.5 (b) Movement of offset (2)

Parameter (No.0001#3) can be set so that offset will not be cancelled by pressing the reset key or by reset input.

- **Tool wear offset in only T code command**

When only a T code is specified in a block, the tool is moved by the wear offset value without a move command. The movement is performed at rapid traverse rate in the G00 mode. It is performed at feedrate in other modes.

When a T code with offset number 0 or 00 is specified by itself, movement is performed to cancel the offset.

By setting bit 4 of parameter No. 014, however, an offset movement can be made together with the axis movement specified by the next block.

WARNING

When G50 X_Z_T_ ; is specified

Tool is not moved.

The coordinate system in which the coordinate value of the tool position is (X,Z) is set. The tool position is obtained by subtracting the wear offset value corresponding to the offset number specified in the T code.

- **Tool geometry offset**

With the tool geometry offset, the work coordinate system is shifted by the X, Y, and Z geometry offset amounts. Namely, the offset amount corresponding to the number designated with the code is added to or subtracted from the current position.

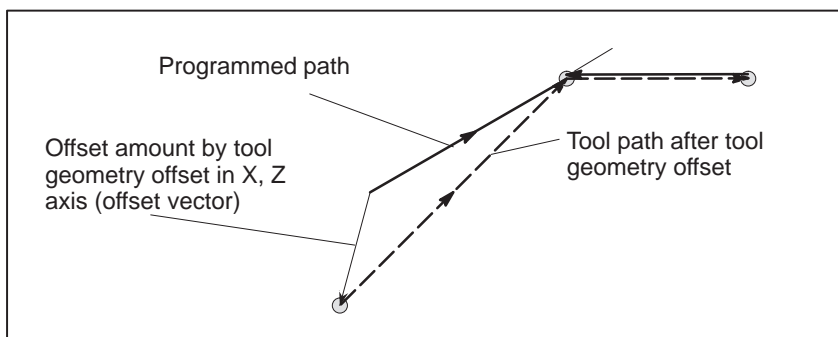


Fig. 14.1.5 (c) Movement of tool geometry offset

NOTE

As well as wear offset, the tool can be compensated by parameter setting (No.013#2) to add or subtract the programmed end point of each block.

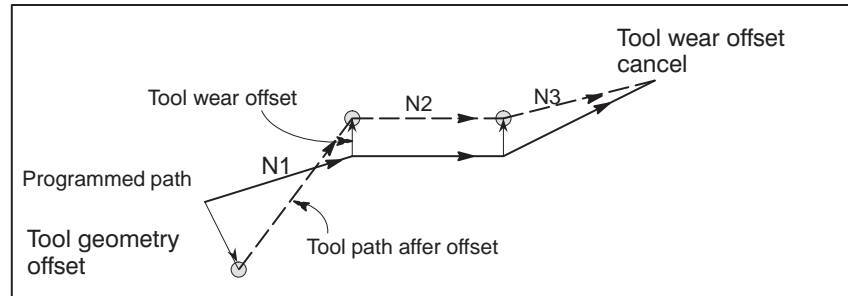
- **Tool geometry offset cancel**

Tool geometry offset is not canceled, even when offset number 0 is specified. By setting bit 3 of parameter No. 013, however, tool geometry offset can be canceled when offset number 0 is specified.

Furthermore, by setting bit 3 of parameter No. 001/bit 1 of parameter No. 014, tool geometry offset can also be canceled by a reset.

Examples

1. When a tool geometry offset number and tool wear offset number are specified with the last two digits of a T code (when bit 1 of parameter No.013, is set 0),
N1 X50.0 Z100.0 T0202 ; Specifies offset number 02
N2 Z200.0 ;
N3 X100.0 Z250.0 T0200 ; Cancels offset



14.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness. The tool nose radius compensation function compensates automatically for the above errors.

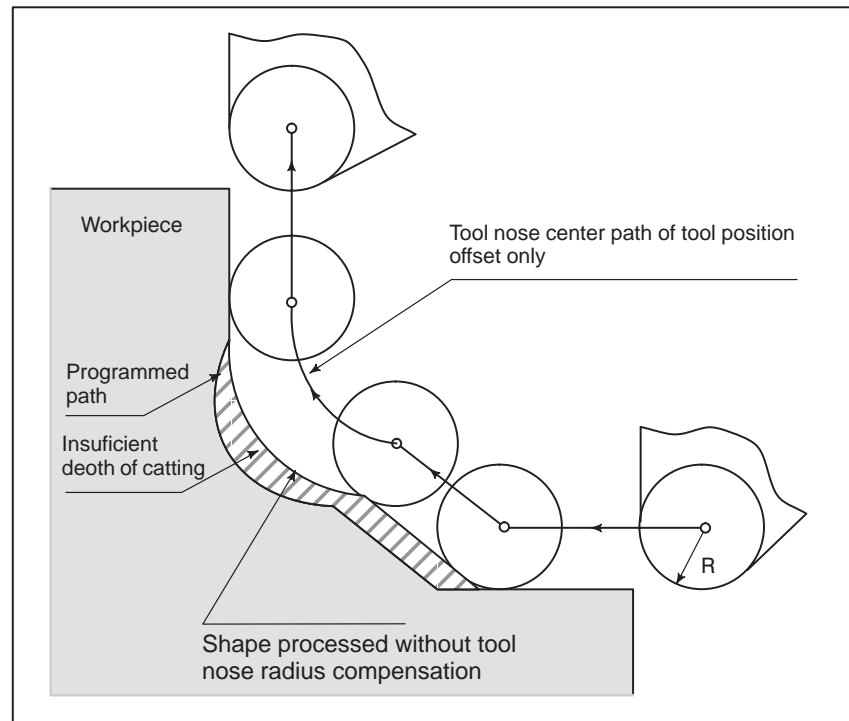


Fig. 14.2 Tool path of tool nose radius compensation

14.2.1 Imaginary Tool Nose

The tool nose at position A in following figure does not actually exist. The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start position than the imaginary tool nose (Note).

Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.

The position relationship when the tool is set to the start position is shown in the following figure.

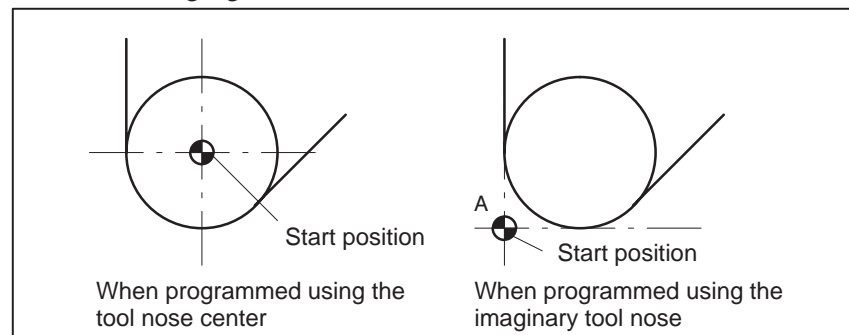


Fig. 14.2.1 (a) Tool nose radius center and imaginary tool nose

NOTE

In a machine with reference positions, a standard position like the turret center can be placed over the start position. The distance from this standard position to the nose radius center or the imaginary tool nose is set as the tool offset value.

Setting the distance from the standard position to the tool nose radius center as the offset value is the same as placing the tool nose radius center over the start position, while setting the distance from the standard position to the imaginary tool nose is the same as placing the imaginary tool nose over the standard position. To set the offset value, it is usually easier to measure the distance from the standard position to the imaginary tool nose than from the standard position to the tool nose radius center.

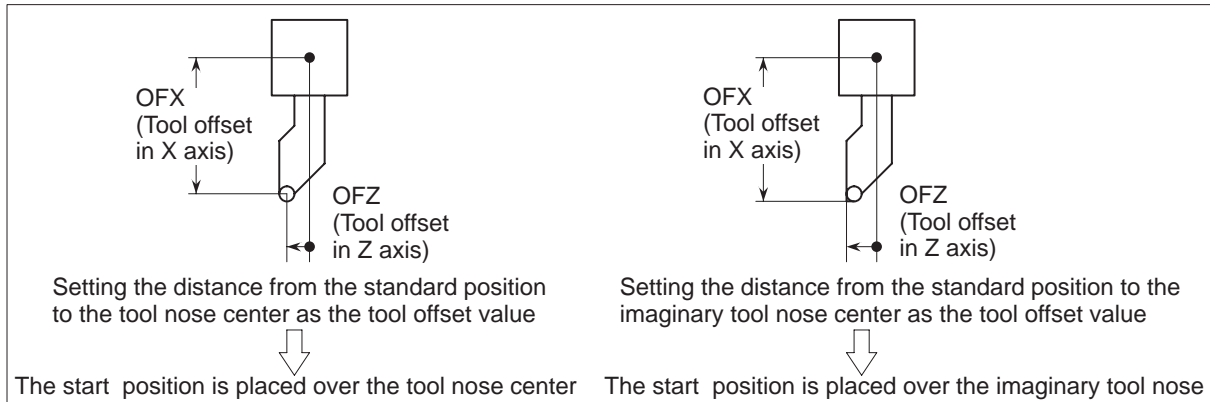
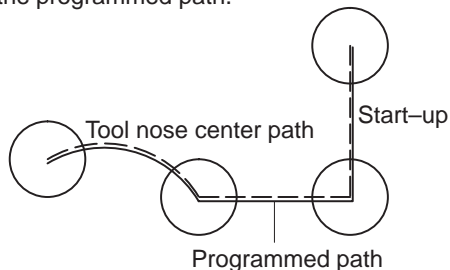


Fig. 14.2.1 (b) Tool offset value when the turret center is placed over the start position

Unless tool nose radius compensation is performed, the tool nose center path is the same as the programmed path.



If tool nose radius compensation is used, accurate cutting will be performed.

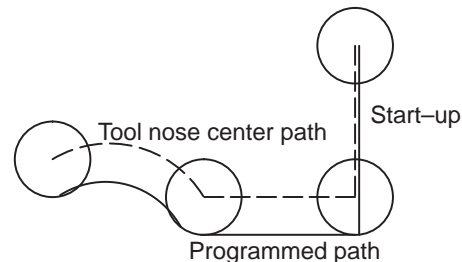
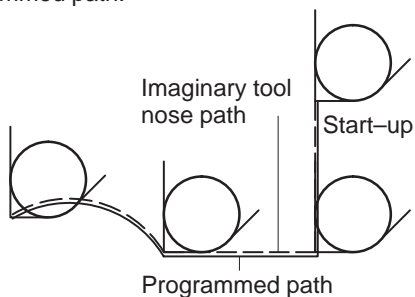


Fig. 14.2.1 (c) Tool path when programming using the tool nose center

Without tool nose radius compensation, the imaginary tool nose path is the same as the programmed path.



With tool nose radius compensation, accurate cutting will be performed.

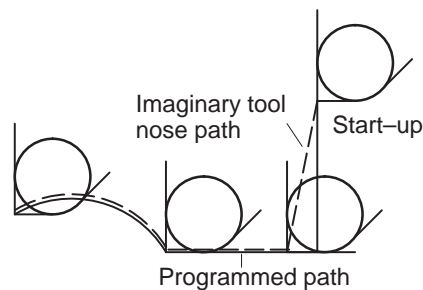


Fig. 14.2.1 (d) Tool path when programming using the imaginary tool nose

14.2.2

Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose center is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values.

The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig.14.2.2 below together with their corresponding codes.

This Fig14.2.2 illustrates the relation between the tool and the start position. The following apply when the tool geometry offset and tool wear offset option are selected.

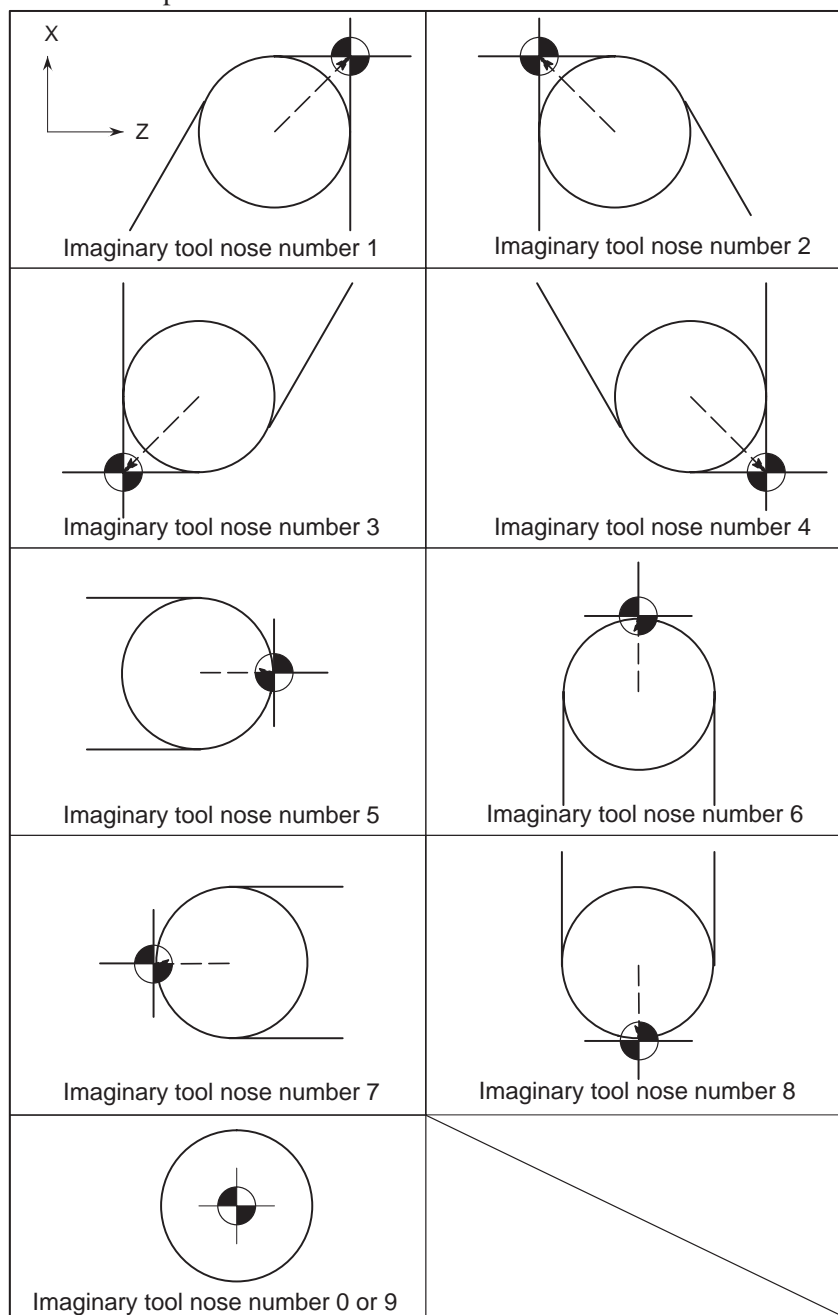


Fig. 14.2.2 Direction of imaginary tool nose

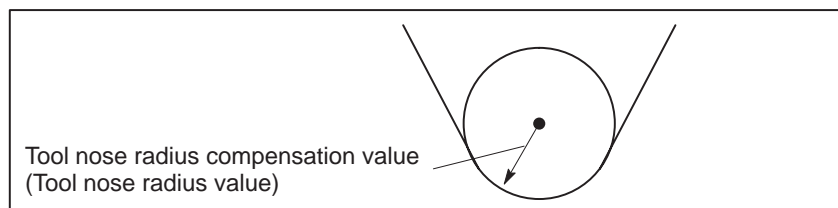
Imaginary tool nose numbers 0 and 9 are used when the tool nose center coincides with the start position. Set imaginary tool nose number to address OFT for each offset number.

14.2.3

Offset Number and Offset Value

Explanations

- Offset number and offset value



This value is set from the MDI according to the offset number.

When the options of tool geometry compensation and tool wear compensation are selected, offset values become as follows :

Table 14.2.3 (a) Offset number and offset value

Offset number Max 32 pairs	OFX (Offset value on X axis)	OFZ (Offset value on Z axis)	OFR (Tool nose radius compensation value)	OFT (Direction of imaginary tool nose)	OFY (Offset value on Y axis)
01	0.040	0.020	0.20	1	0.030
02	0.060	0.030	0.25	2	0.040
⋮	⋮	⋮	⋮	⋮	⋮
31	0.050	0.015	0.12	6	0.025
32	0.030	0.025	0.24	3	0.035

When the options of tool geometry compensation and tool wear compensation are selected, the offset values become as follows :

Table 14.2.3 (b) Tool geometry offset

Geometry offset number Max 32 pairs	OFGX (X-axis geometry offset amount)	OFGZ (Z-axis geometry offset amount)	OFGR (Tool nose radius geometry offset value)	OFT (Imaginary tool nose direction)	OFGY (Y-axis geometry offset amount)
G01	10.040	50.020	0	1	70.020
G02	20.060	30.030	0	2	90.030
G03	0	0	0.20	6	0
G04	⋮	⋮	⋮	⋮	⋮
G05	⋮	⋮	⋮	⋮	⋮
⋮	⋮	⋮	⋮	⋮	⋮

Table 14.2.3 (c) Tool wear offset

Wear offset number Max 32 pairs	OFGX (X-axis wear off-set amount)	OFGZ (Z-axis wear off-set amount)	OFGR (Tool nose radius wear off-set value)	OFT (Imaginary tool nose direction)	OFGY (Y-axis wear off-set amount)
W01	0.040	0.020	0	1	0.010
W02	0.060	0.030	0	2	0.020
W03	0	0	0.20	6	0
W04	:	:	:	:	:
W05	:	:	:	:	:
:	:	:	:	:	:

- **Tool nose radius compensation**

In this case, the tool nose radius compensation value is the sum of the geometry or the wear offset value.

$$\text{OFR} = \text{OFGR} + \text{OFWR}$$

- **Imaginary tool nose direction**

The imaginary tool nose direction may be set for either the geometry offset or the wear offset.

However, the last designated direction later is effective.

- **Command of offset value**

A offset number is specified with the same T code as that used for tool offset. For details, see 14.1.2.

NOTE

When the geometry offset number is made common to the tool selection by the parameter (No.013#1) setting and a T code for which the geometry offset and wear offset number differ from each other is designated, the imaginary tool nose direction specified by the geometry offset number is valid. By setting bit 3 of parameter No. 075, however, the imaginary tool nose direction, specified with a wear offset number, can be enabled.

Example) **T0102**
 OFR=RFGR₀₁+OFWR₀₂
 OFT=OFT₀₁

- **Setting range of offset value**

The range of the offset value is as follows :

Increment system	metric system	Inch system
IS-B	0 to ± 999.999 mm	0 to ± 99.9999 inch
IS-C	0 to ± 999.9999 mm	0 to ± 99.99999 inch

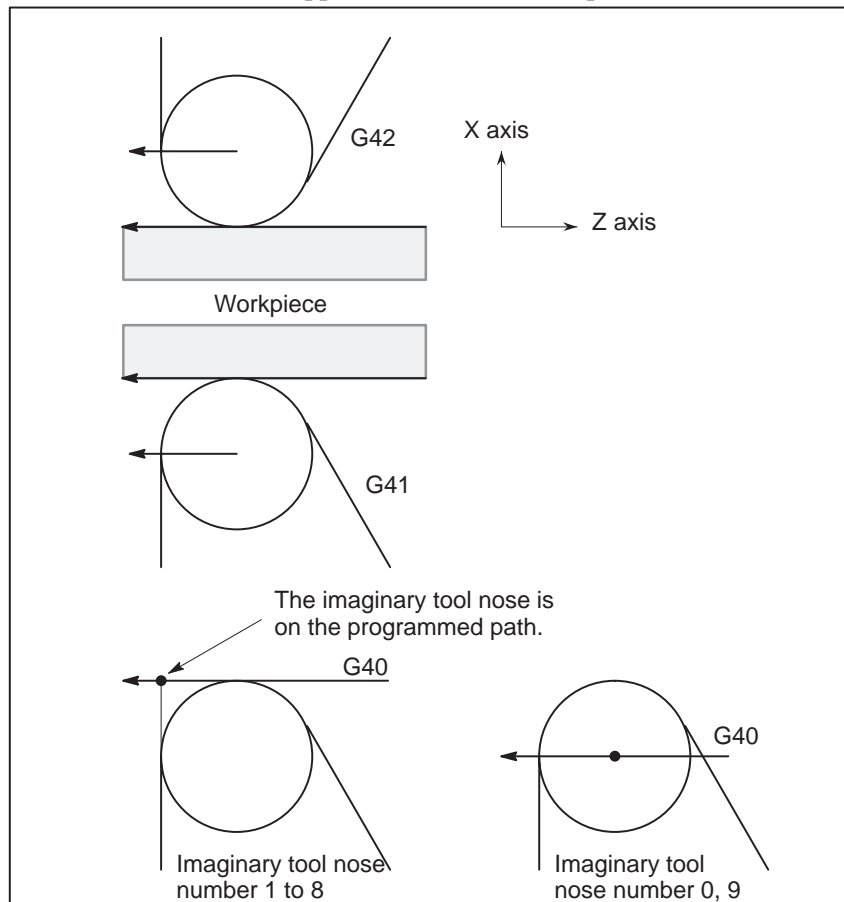
The offset value corresponding to the offset number 0 is always 0.
 No offset value can be set to offset number 0.

14.2.4 Work Position and Move Command

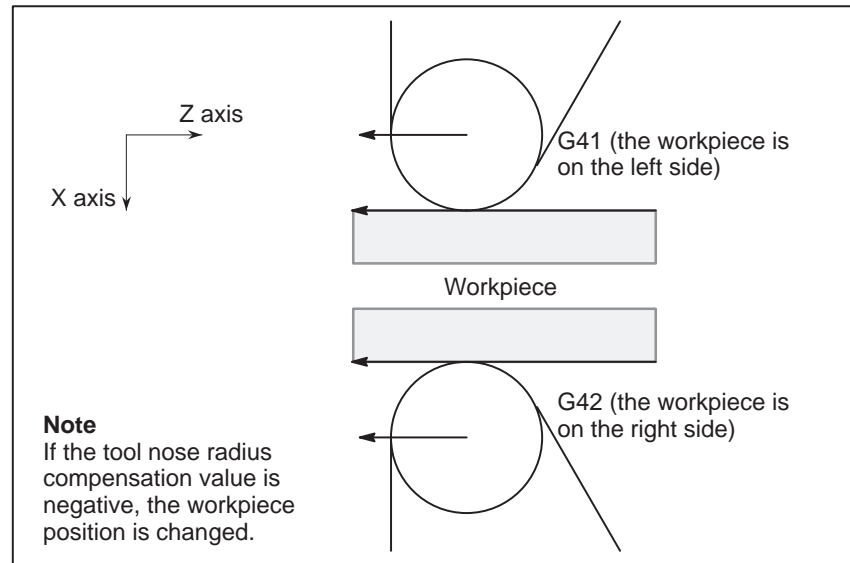
In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified.

G code	Workpiece position	Tool path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side the programmed path
G42	Left side	Moving on the right side the programmed path

The tool is offset to the opposite side of the workpiece.



The workpiece position can be changed by setting the coordinate system as shown below.



G40, G41, and, G42 are modal.

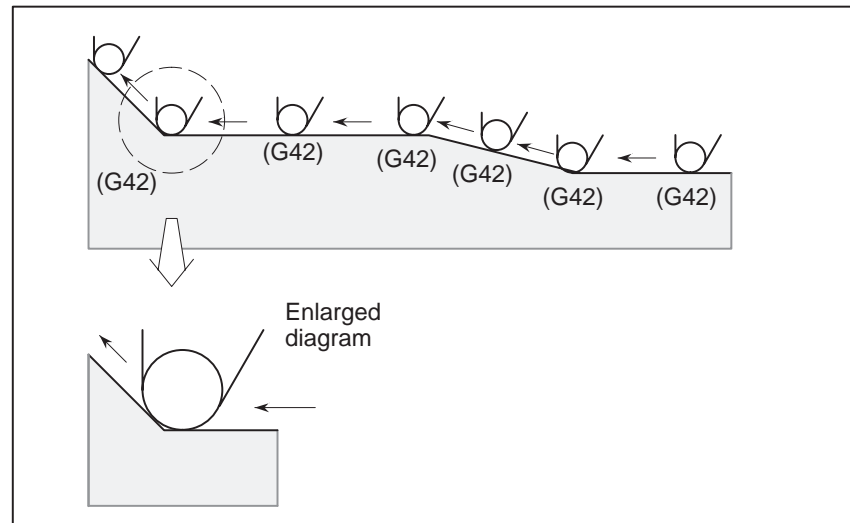
Don't specify G41 while in the G41 mode. If you do, compensation will not work properly.

Don't specify G42 while in the G42 mode for the same reason.

G41 or G42 mode blocks in which G41 or G42 are not specified are expressed by (G41) or (G42) respectively.

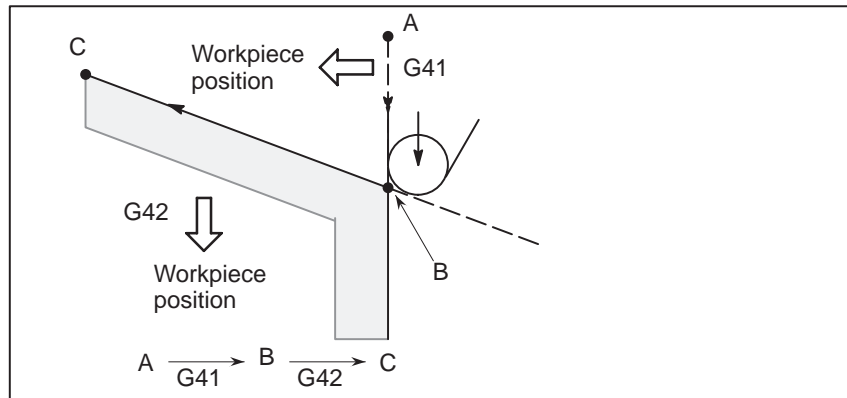
- **Tool movement when the workpiece position does not change**

When the tool is moving, the tool nose maintains contact with the workpiece.



- **Tool movement when the workpiece position changes**

The workpiece position against the tool changes at the corner of the programmed path as shown in the following figure.



Although the workpiece does not exist on the right side of the programmed path in the above case, the existence of the workpiece is assumed in the movement from A to B. The workpiece direction must not be changed in the block next to the start-up block.

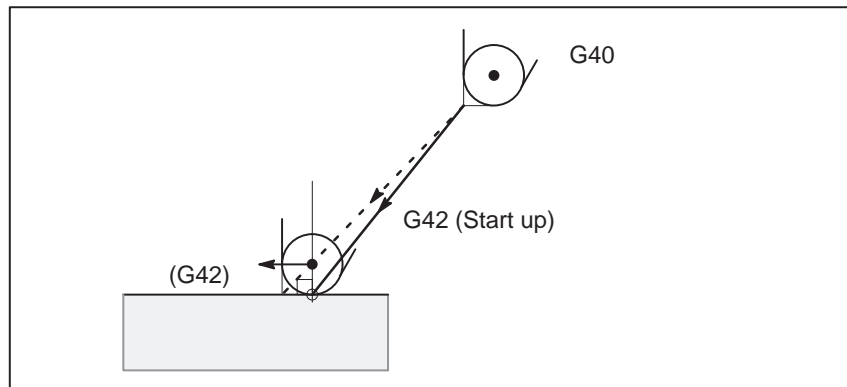
- **Start-up**

The block in which the mode changes to G41 or G42 from G40 is called the start-up block.

G40 _ ;

G41 _ ; (Start-up block)

Transient tool movements for offset are performed in the start-up block. In the block after the start-up block, the tool nose center is positioned Vertically to the programmed path of that block at the start position.



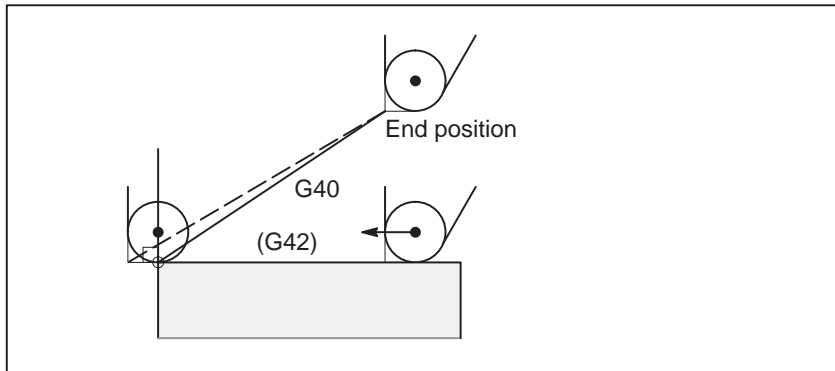
- **Offset cancel**

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block.

G41 _ ;

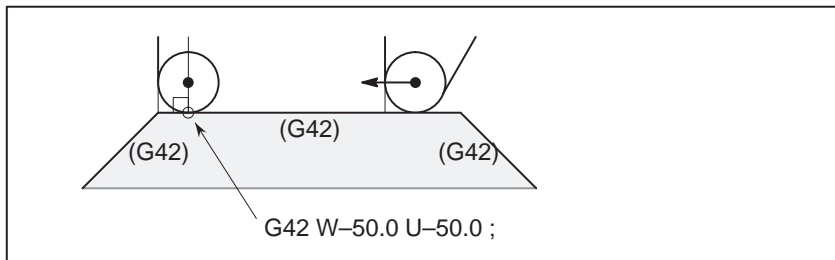
G40 _ ; (Offset cancel block)

The tool nose center moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40) as shown below.



- **Specification of G41/G42 in G41/G42 mode**

When is specified again in G41/G42 mode, the tool nose center is positioned vertical to the programmed path of the preceding block at the end position of the preceding block.

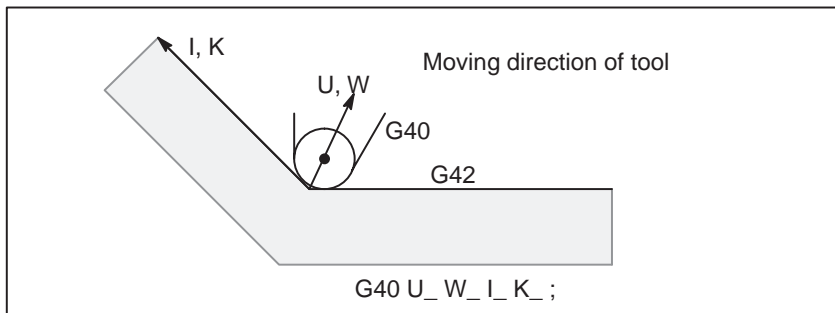


In the block that first specifies G41/G42, the above positioning of the tool nose center is not performed.

- **Tool movement when the moving direction of the tool in a block which includes a G40 command is different from the direction of the workpiece**

When you wish to retract the tool in the direction specified by X(U) and Z(W) cancelling the tool nose radius compensation at the end of machining the first block in the figure below, specify the following :

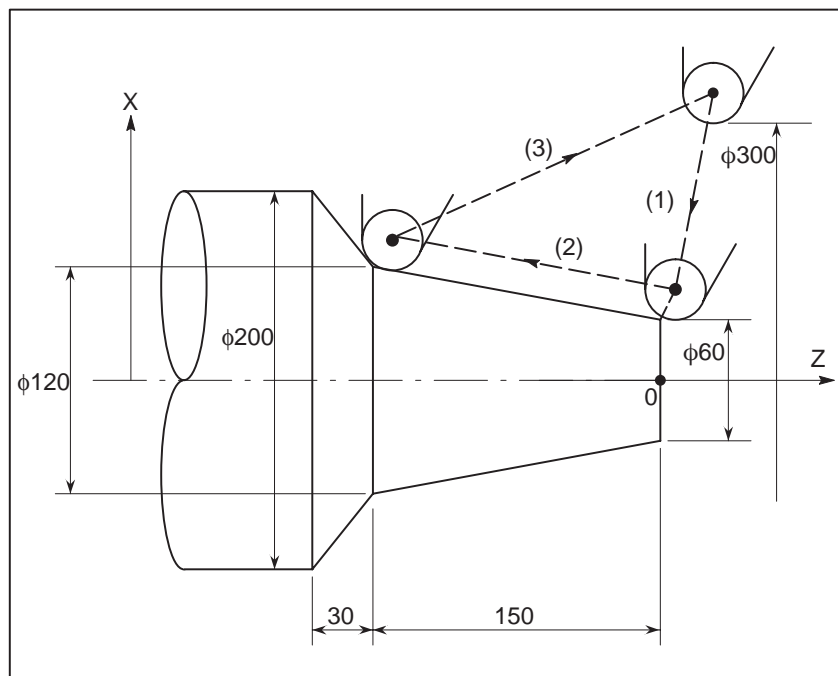
G40 X(U) _ Z(W) _ I _ K _ ;



The workpiece position specified by addresses I and K is the same as that in the preceding block.

G40 X_ Z_ I_ K_ ;	I, J of tool nose radius compensation cancel
G40 G02 X_ Z_ I_ K_ ;	I, J of circular interpolation center

Examples



(G40 mode)

1.G42 G00 X60.0 ;

2.G01 X120.0 W-150.0 F10 ;

3.G40 G00 X300.0 W150.0 I40.0 K-30.0 ;

14.2.5

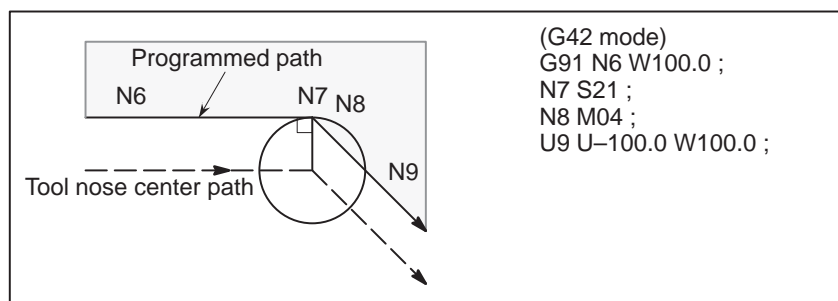
Notes on Tool Nose Radius Compensation

Explanations

- Tool movement when two or more blocks without a move command should not be programmed consecutively

1.M05 ;	M code output
2.S210 ;	S code output
3.G04 X1.0 ;	Dwell
4.G01 U0 ;	Feed distance of zero
5.G98 ;	G code only
6.G10 P01 X10.0 Z20.0 R0.5 Q2 ;	Offset change

If two or more of the above blocks are specified consecutively, the tool nose center comes to a position vertical to the programmed path of the preceding block at the end of the preceding block. However, if the no movement commands is **4** above, the above tool motion is attained only with one block.

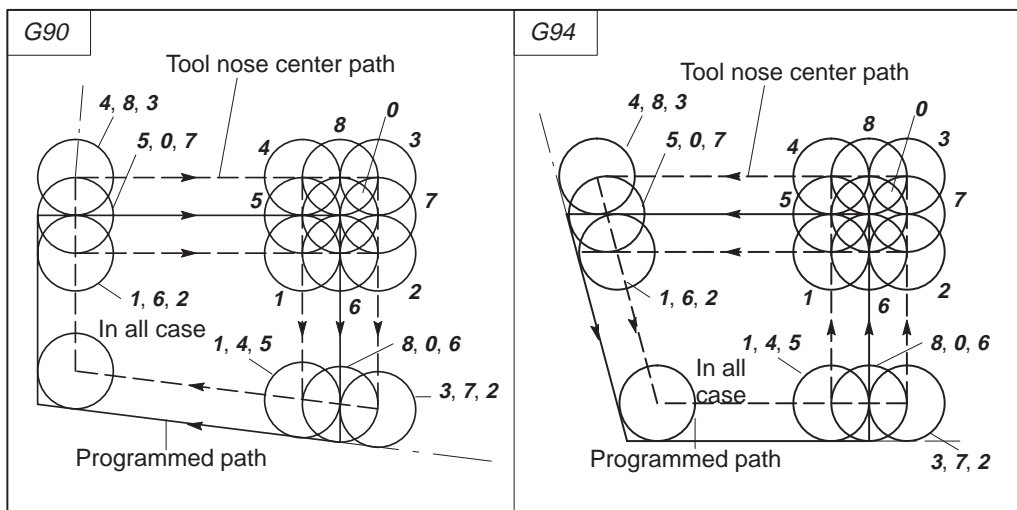


- Tool nose radius compensation with G90 or G94

Tool nose radius compensation with G90 (outer diameter/inner diameter cutting cycle) or G94 (end face turning cycle) is as follows, :

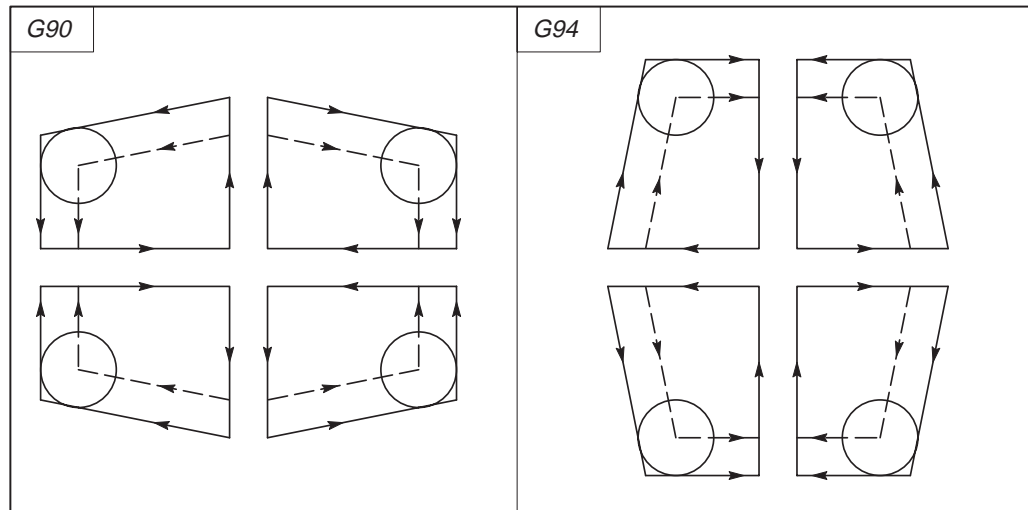
1.Motion for imaginary tool nose numbers

For each path in the cycle, the tool nose center path is generally parallel to the programmed path.



2.Direction of the offset

The offset direction is indicated in the figure below regardless of the G41/G42 mode.



- **Tool nose radius compensation with G71 to G76 or G78**

When one of following cycles is specified, the cycle deviates by a tool nose radius compensation vector. During the cycle, no intersection calculation is performed.

G71 (Stock removal in turning or traverse grinding cycle)

G72 (Stock removal in facing or traverse direct constant-dimension grinding cycle)

G73 (Pattern repeating or Oscillation grinding cycle)

When one of following cycles is specified, the tool nose radius compensation is not performed.

G74 (End face peck drilling)

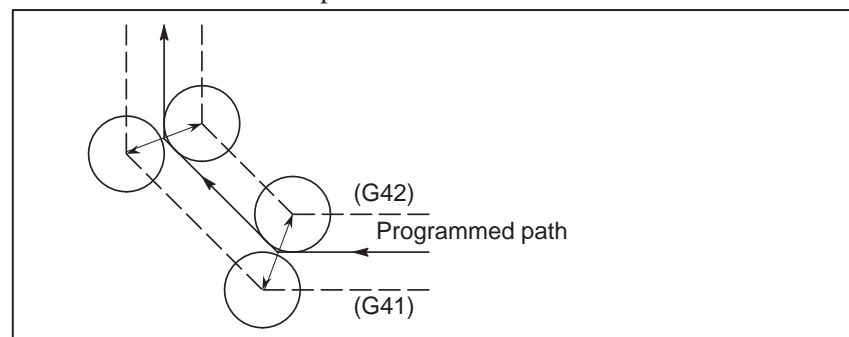
G75 (Outer diameter/internal diameter drilling)

G76 (Multiple threading cycle)

G78 (Threading cycle)

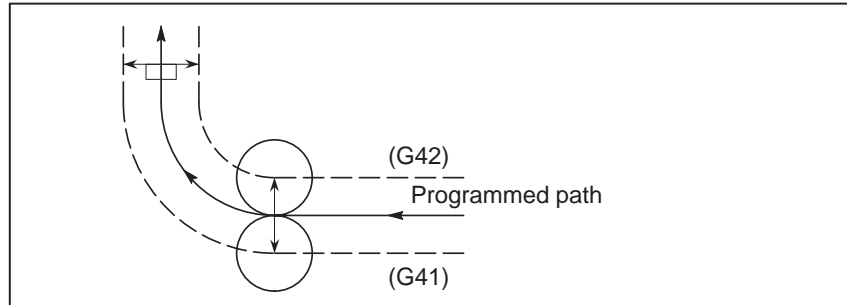
- **Tool nose radius compensation when chamfering is performed**

Movement after compensation is shown below.



- **Tool nose radius compensation when a corner arc is inserted**

Movement after compensation is shown below.



- **Tool nose radius compensation when the block is specified from the MDI**

In this case, tool nose radius compensation is not performed.

14.3

DETAILS OF TOOL NOSE RADIUS COMPENSATION

This section provides a detailed explanation of the movement of the tool for tool nose radius compensation outlined in Section 14.2.

This section consists of the following subsections:

14.3.1 General

14.3.2 Tool Movement in Start-up

14.3.3 Tool Movement in Offset Mode

14.3.4 Tool Movement in Offset Mode Cancel

14.3.5 Interference Check

14.3.6 Over cutting by Tool Nose Radius Compensation

14.3.7 Correction in Chamfering and Corner Arc

14.3.8 Input Command from MDI

14.3.9 General Precautions for Offset Operations

14.3.1

General

- **Tool nose radius center compensation vector**

The tool nose radius compensation vector is a two dimensional vector equal to the offset value specified in a T code, and the is calculated in the CNC.

Its dimension changes block by block according to tool movement.

This offset vector (simply called vector herein after) is internally crated by the control unit as required for proper offsetting and to calculate a tool path with exact offset (by tool nose compensation) from the programmed path.

This vector is deleted by resetting.

The vector always accompanies the tool as the tool advances.

Proper understanding of vector is essential to accurate programming.

Read the description below on how vectors are created carefully.

- **G40, G41, G42**

G40, G41 or G42 is used to delete or generate vectors.

These codes are used together with G00, G01, G02 or G03 to specify a mode for tool motion (Offsetting).

G code	Function	Workpiece position
G40	Tool nose radius compensation cancel	Neither
G41	Left offset along tool path	Right
G42	Right offset along tool path	Left

G41 and G42 specify an off mode, while G40 specifies cancellation of the offset.

- **Cancel mode**

The system enters the cancel mode immediately after the power is turned on, when the RESET button on the CRT/MDI panel is pushed or a program is forced to end by executing M02 or M30. (the system may not enter the cancel mode depending on the machine tool.) In the cancel mode, the vector is set to zero, and the tool path coincides with the programmed, path. A program must end in cancel mode. If it ends in the offset mode, the tool cannot be positioned at the end point, and the tool stops at a location the vector length away from the end point.

• Start-up

When a block which satisfies all the following conditions is executed in cancel mode, the system enters the offset mode. Control during this operation is called start-up.

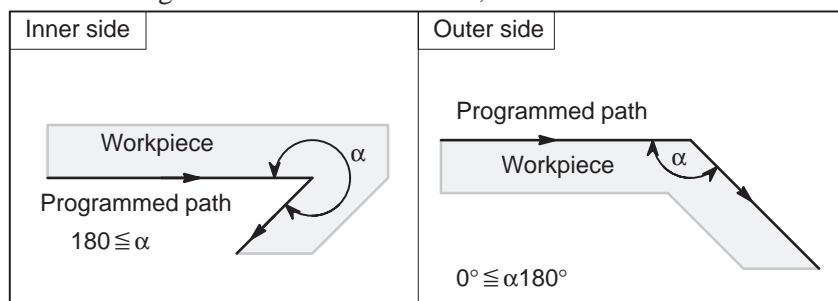
- G41 or G42 is contained in the block, or has been specified to set the system enters the offset mode. Control during this operation is called start-up.
- The offset number for tool nose radius compensation is not 00.
- X or Z moves is specified in the block and the move distance is not zero.

A circular command (G02 or G03) is not allowed in start-up.

If specified, alarm (PS34) will occur. Two blocks are read in during start-up. The first block is executed, and the second block is entered into the tool nose radius compensation buffer. In the single block mode, two blocks are read and the first block is executed, then the machine stops. In subsequent operations, two blocks are read in advance, so the CNC has the block currently being executed, and the next two blocks.

• 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:

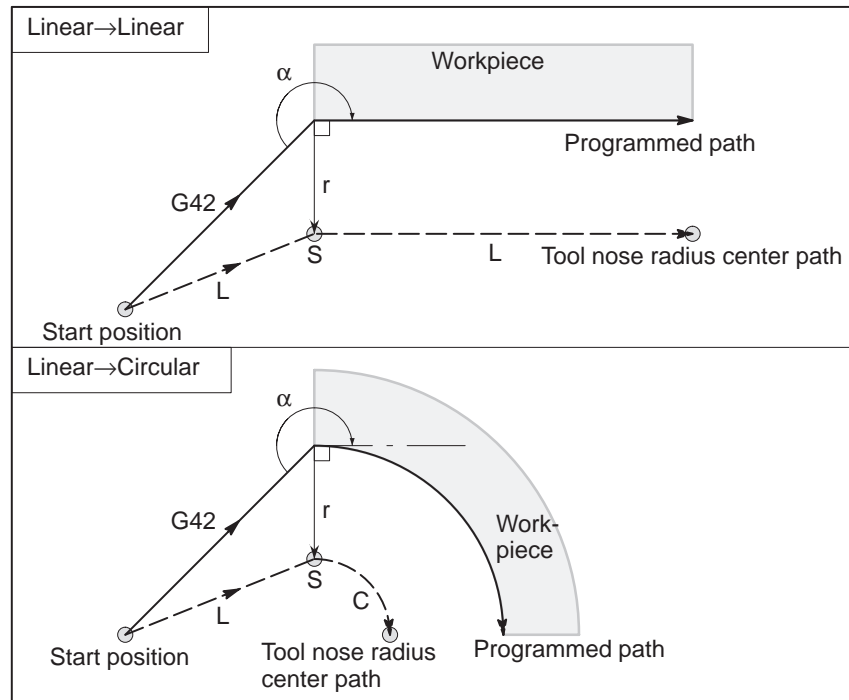
- *S* indicates a position at which a single block is executed once.
- *SS* indicates a position at which a single block is executed twice.
- *SSS* indicates a position at which a single block is executed three times.
- *L* indicates that the tool moves along a straight line.
- *C* indicates that the tool moves along an arc.
- *r* indicates the tool nose radius compensation value.
- An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by *r*.
- indicates \odot the center of the tool nose radius.

14.3.2 Tool Movement in Start-Up

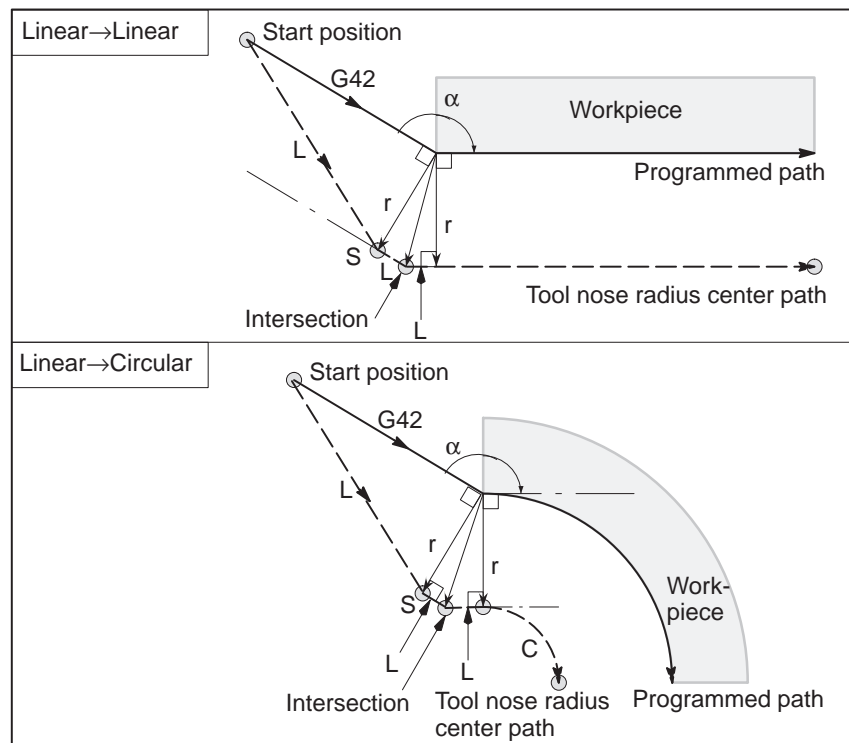
When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start-up):

Explanations

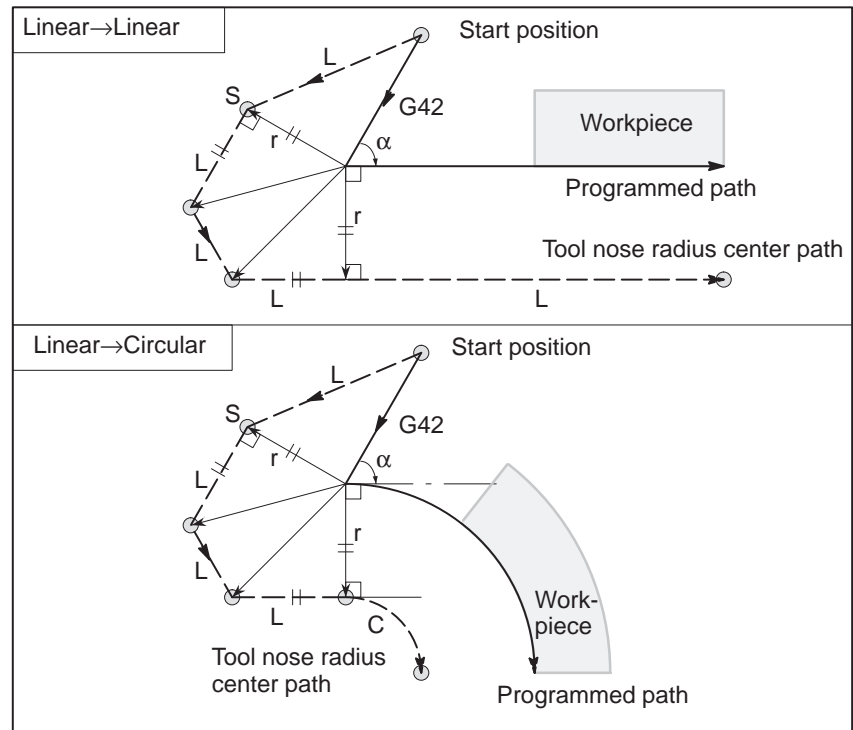
- Tool movement around an inner side of a corner ($180^\circ \cong \alpha$)



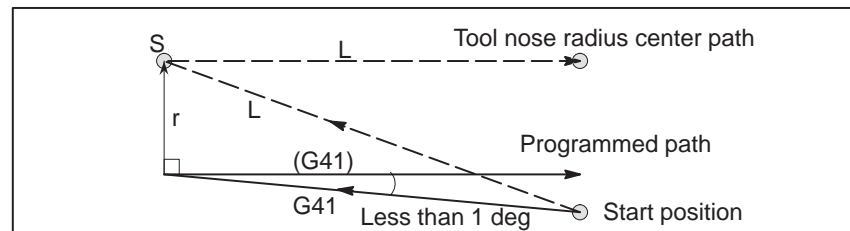
- Tool movement around the outside of a corner at an obtuse angle ($90^\circ \cong \alpha < 180^\circ$)



- Tool movement around the outside of an acute angle ($\alpha < 90^\circ$)



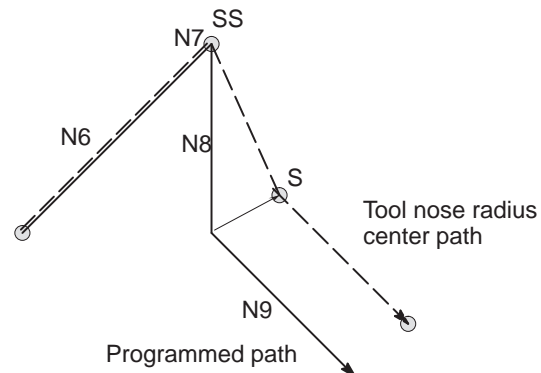
- Tool movement around the outside linear→linear at an acute angle less than 1 degree ($\alpha < 1^\circ$)



- 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 U10.0 W10.0 ;
N7 G41 U0 ;
N8 W-10.0 ;
N9 W-10.0 U10.0 ;
```



NOTE

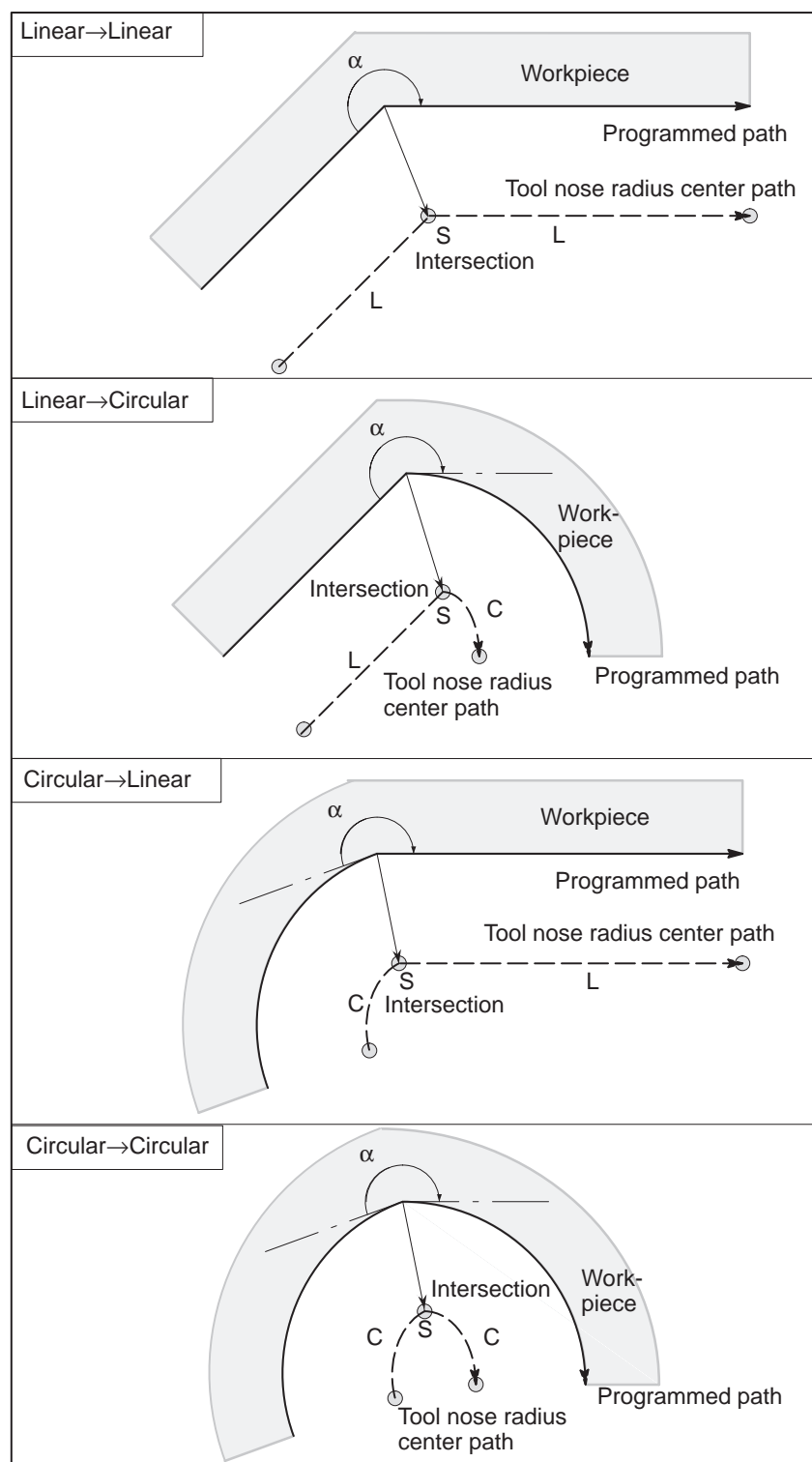
For the definition of blocks that do not move the tool, see Subsection 14.3.3.

14.3.3 Tool Movement in Offset Mode

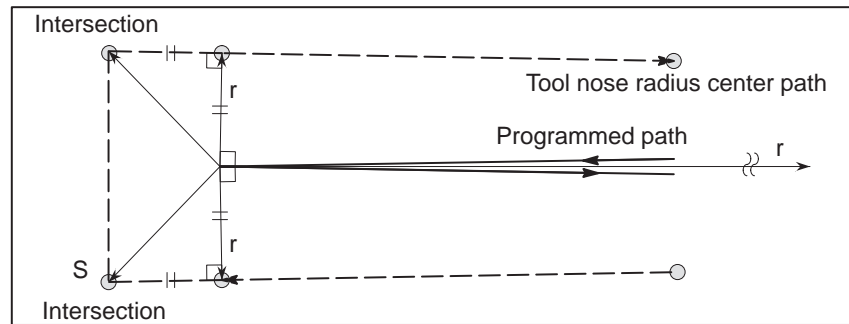
In the offset mode, the tool moves as illustrated below:

Explanations

- Tool movement around the inside of a corner ($180^\circ \cong \alpha$)

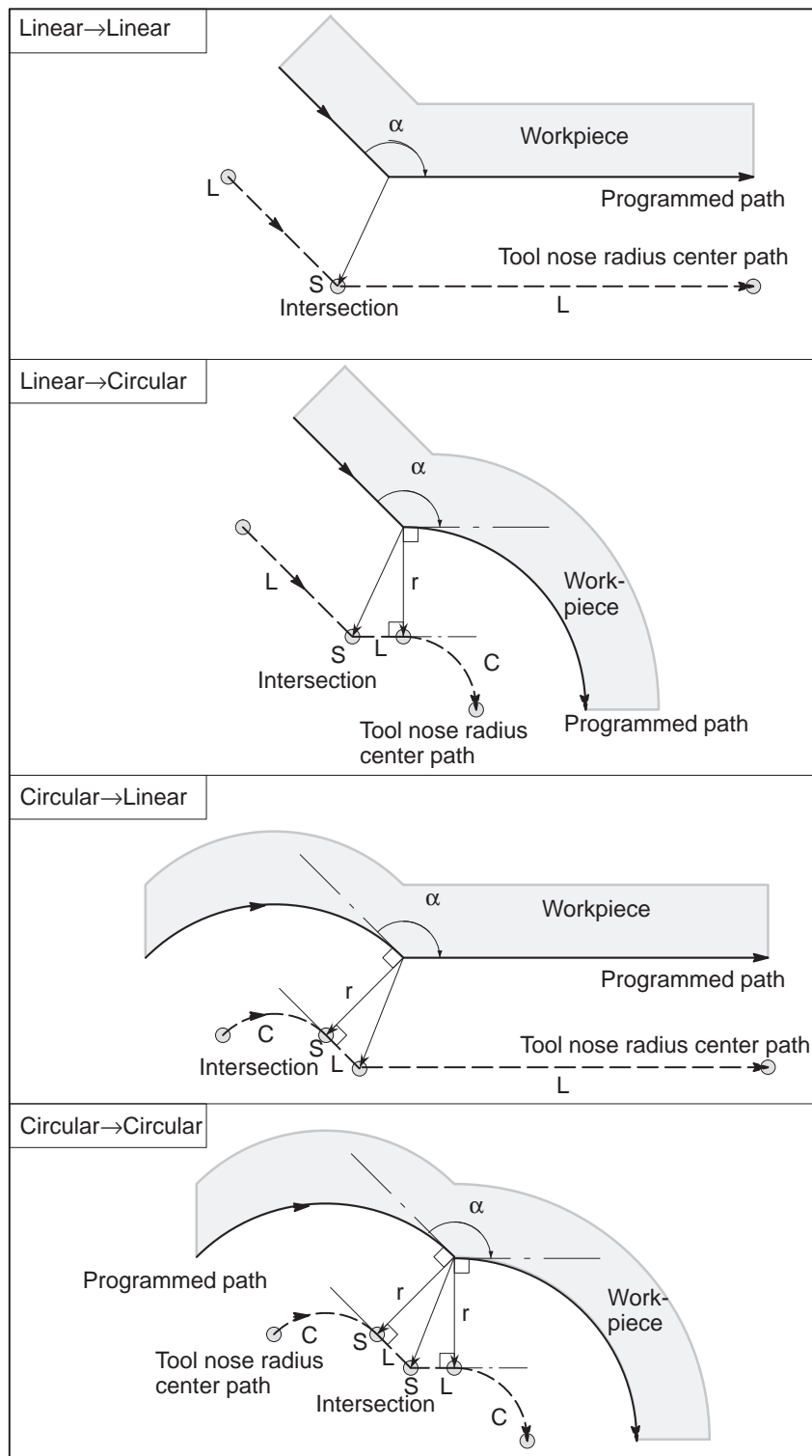


- **Tool movement around the inside ($\alpha < 1^\circ$) with an abnormally long vector, linear \rightarrow linear**

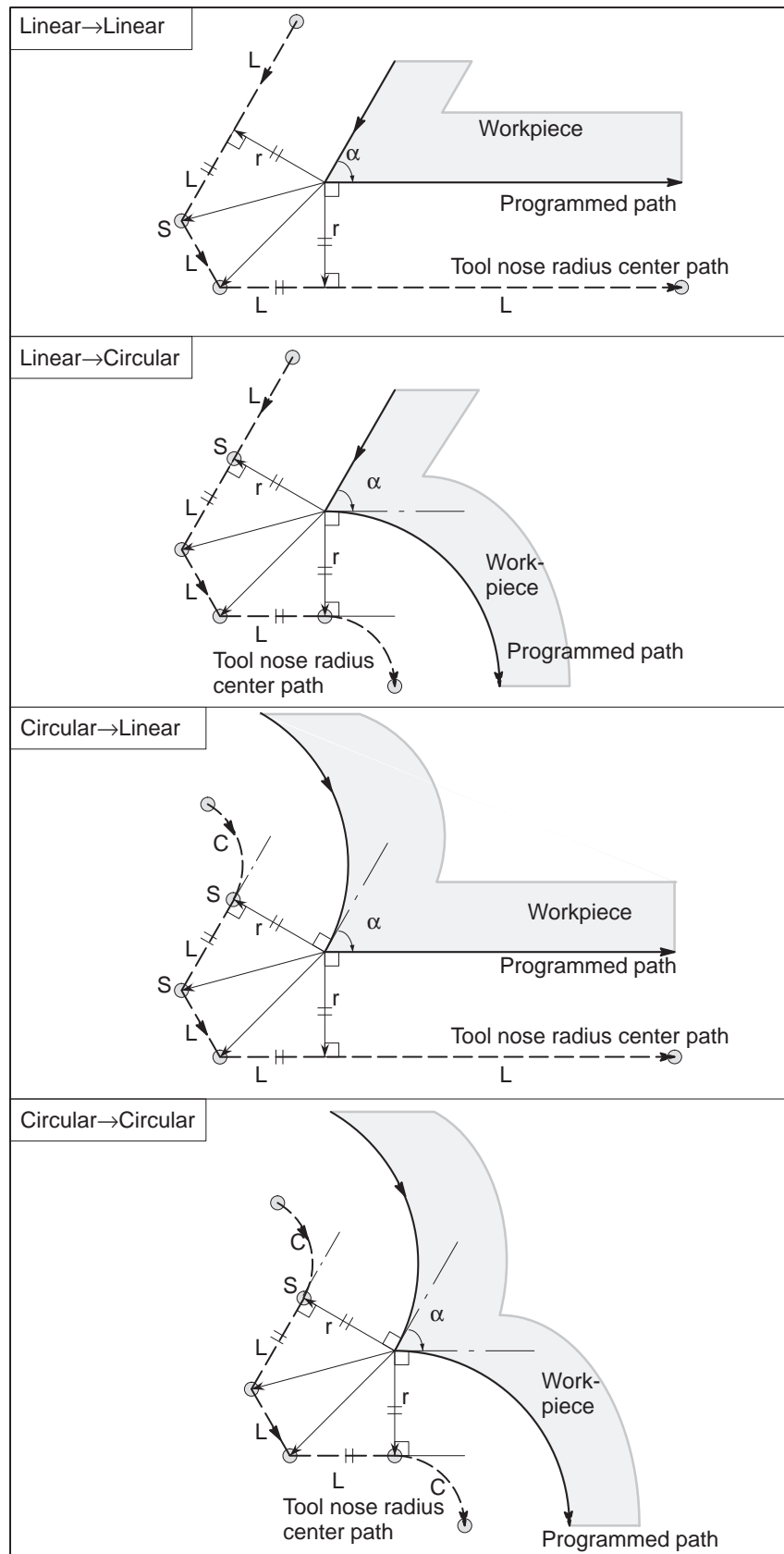


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

- Tool movement around the outside corner at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)



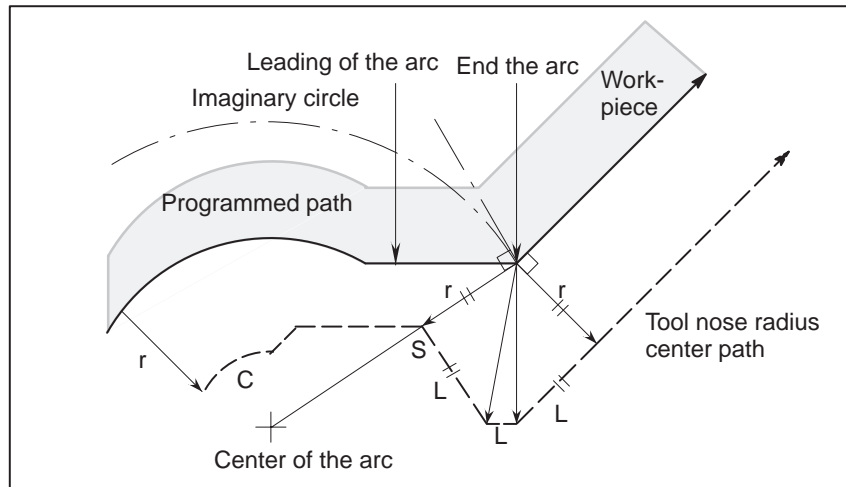
- Tool movement around the outside corner at an acute angle ($\alpha < 90^\circ$)



- When it is exceptional

End position for the arc is not on the arc

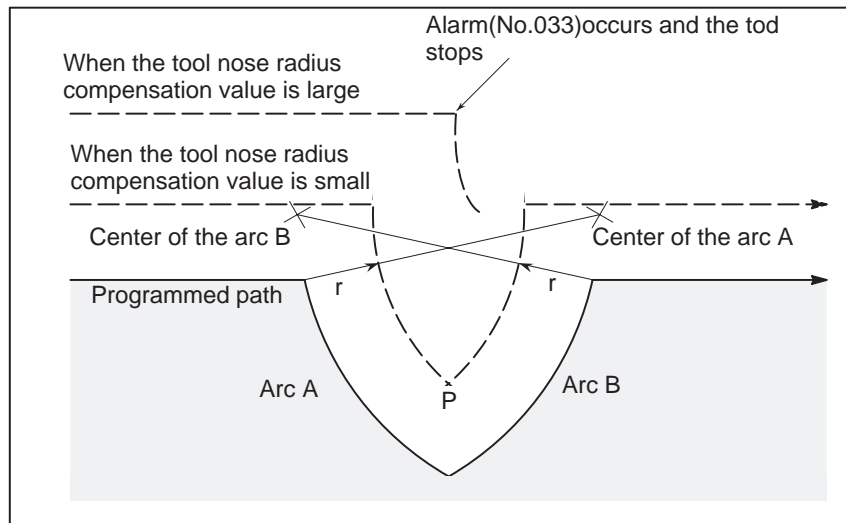
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that tool nose radius compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system creates a vector and carries out compensation. The resulting tool nose radius center path is different from that created by applying tool nose radius 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.

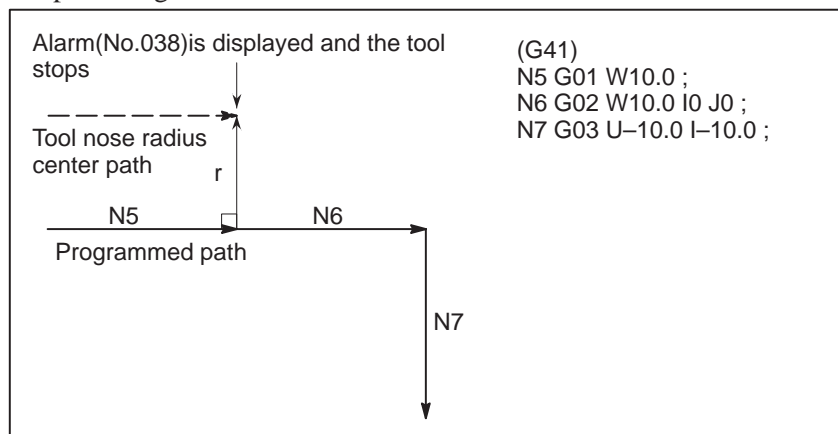
There is no inner intersection

If the tool nose radius compensation value is sufficiently small, the two circular Tool nose radius center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool nose radius 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 nose radius center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool nose radius 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.



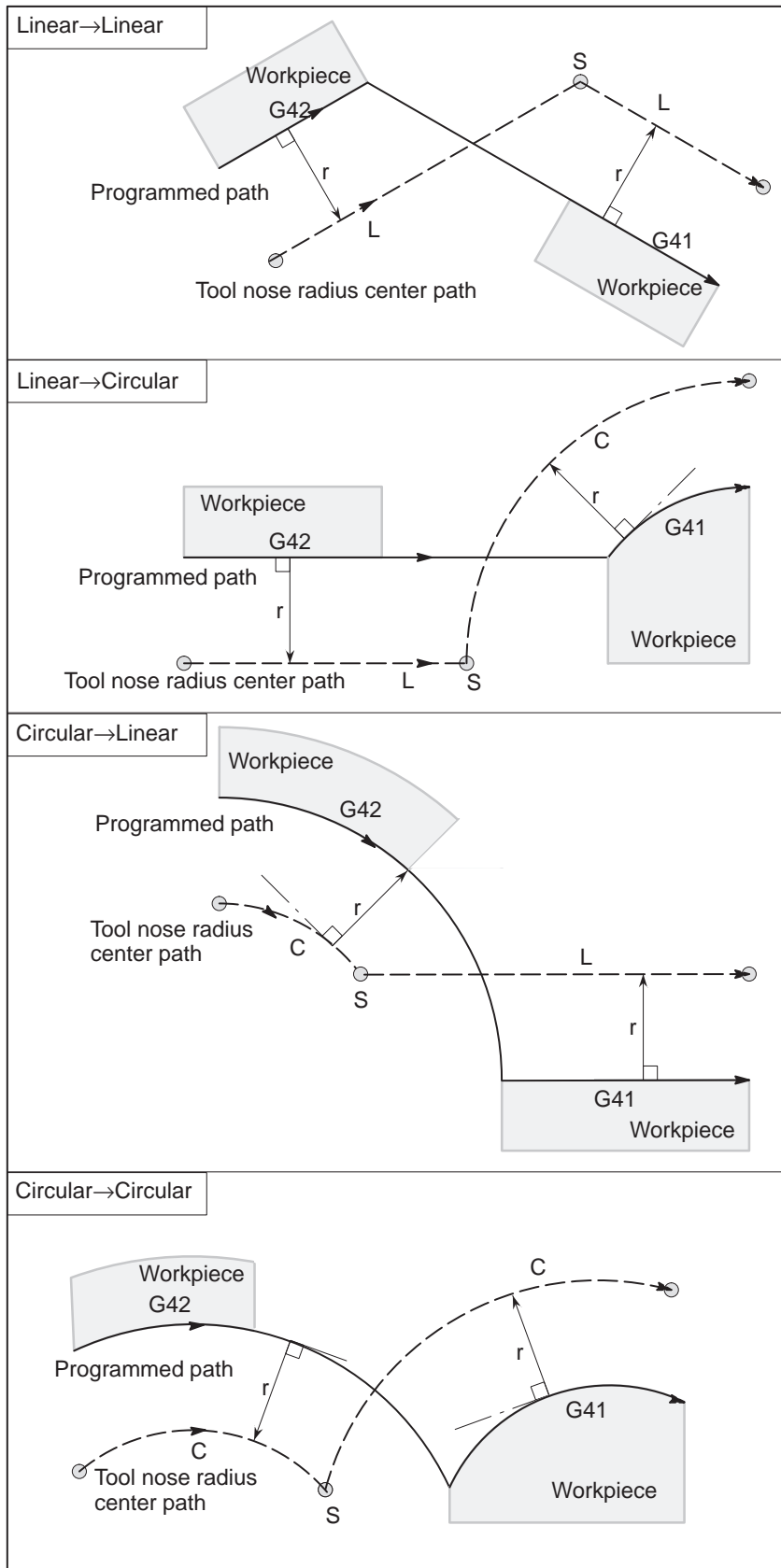
- Change in the offset direction in the offset mode**

The offset direction is decided by G codes (G41 and G42) for tool nose radius and the sign of tool nose radius compensation value as follows.

G code	Sign of offset value	
	+	-
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

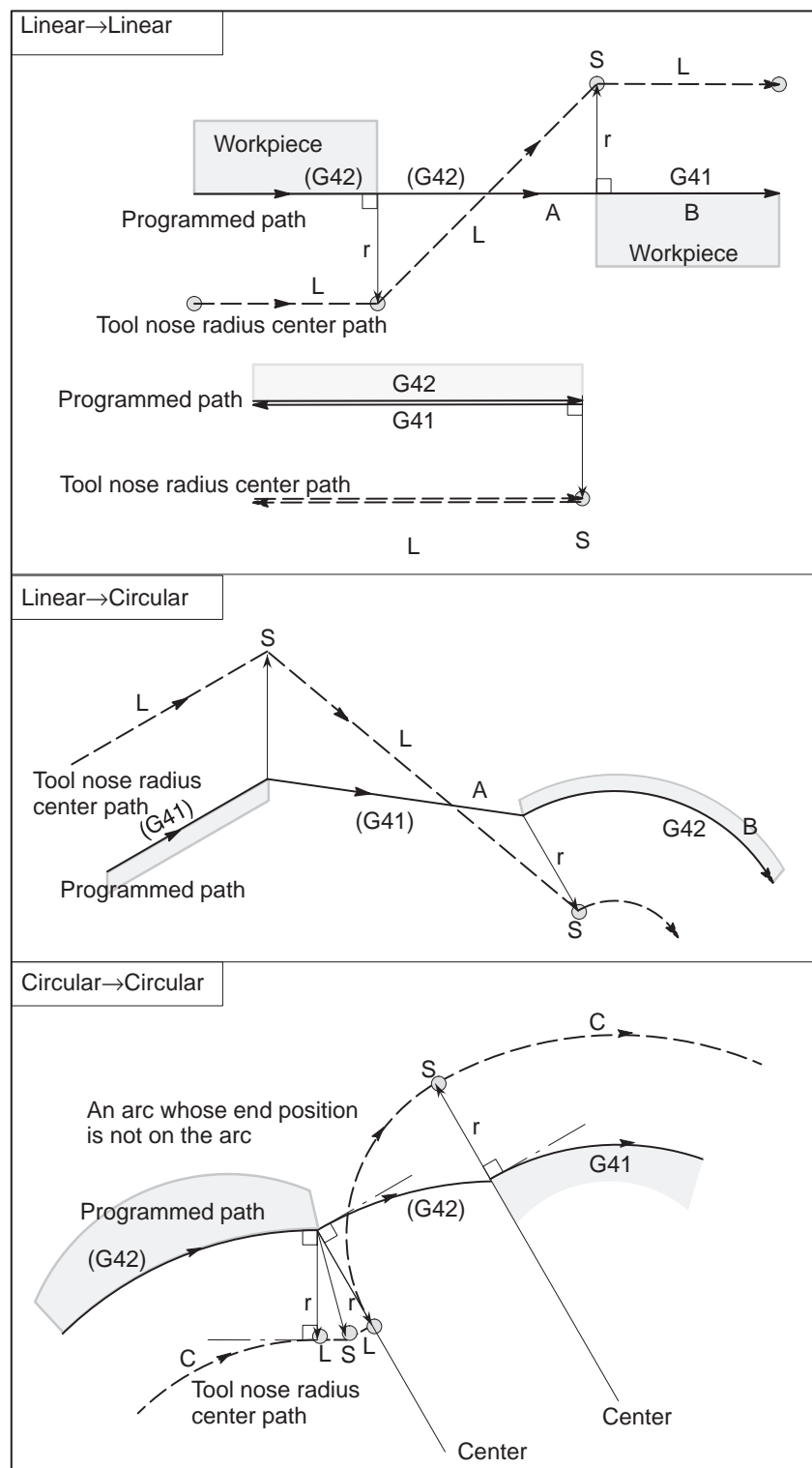
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool nose radius center path of that block and the tool nose radius center path of a preceding block. However, the change is not available in the start-up block and the block following it.

Tool nose radius center path with an intersection



Tool nose radius 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.

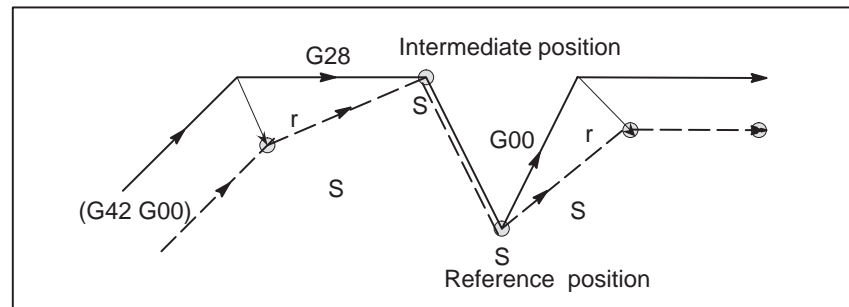


- **Temporary tool nose radius compensation cancel**

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 14.3.2 and 14.3.4.

**Specifying G28
(automatic return to the
reference position) in
the offset mode**

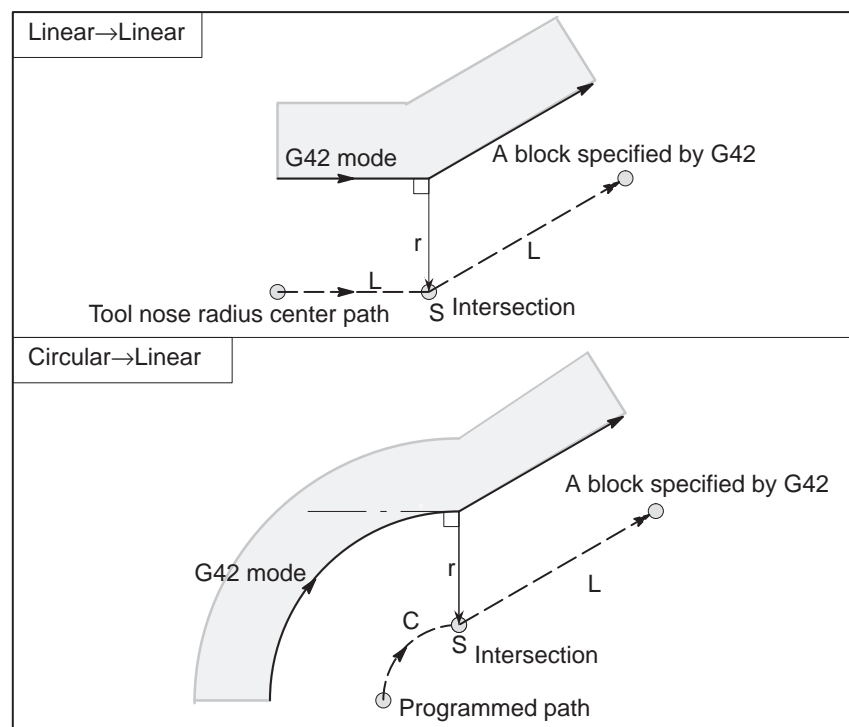
If G28 is specified in the offset mode, the offset mode is canceled at an intermediate position. Offset mode is restored automatically after the tool is returned to the reference position.



- **Tool nose radius 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 tool nose radius 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 tool nose radius compensation G code (G41, G42), refer to "Change in the offset direction in the offset mode" in Subsec.15.3.3.

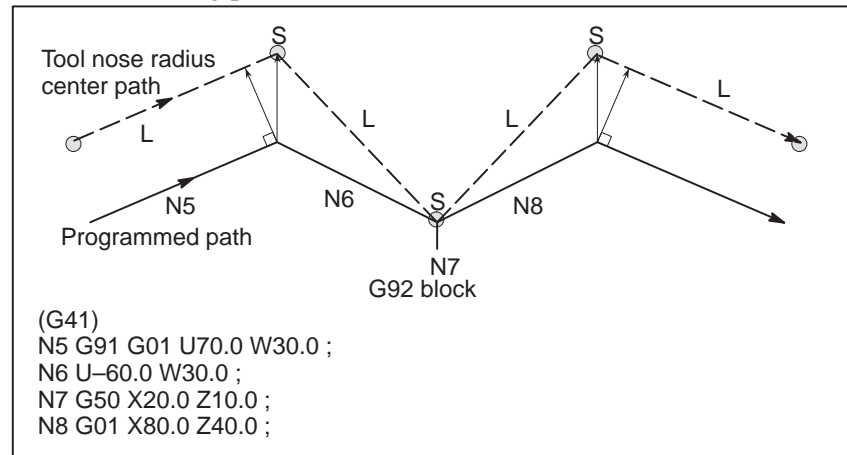


- **Command cancelling the offset vector temporarily**

During offset mode, if G92 (absolute zero point programming) is commanded, the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.

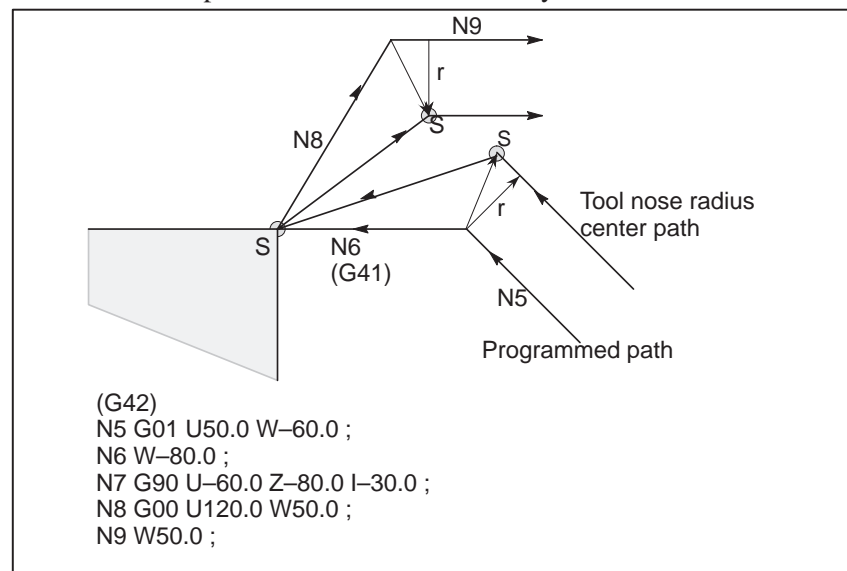
In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled. Also when restored to offset mode, the tool moves directly to the intersecting point.

- **Workpiece coordinate system setting (G50)**



- **Canned cycles (G90, G92, G94) and Multiple repetitive cycles (G71 to G76)**

See Sections 14.1 (G90, G92, G94) and 14.2 (G71 to G76) for the tool nose radius compensation is related canned cycles.



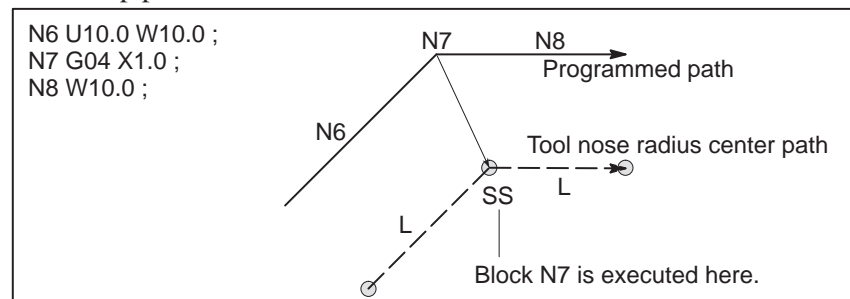
● A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if tool nose radius compensation is effected.

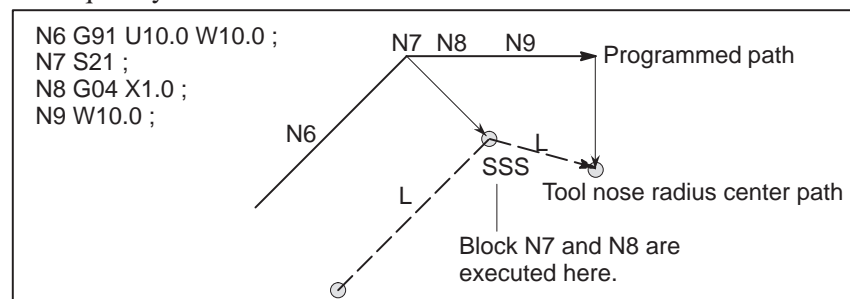
M05 ;	M code output	} Commands are no movement.
S21 ;	S code output	
G04 X1.0 ;	Dwell	
G10 P01 X10 Z20 R10.0 ;	tool nose radius compensation value setting/modification	
Y20.0 ;	Move command not included in the offset plane.	
G98 ;	G code only	
U0 ;	Move distance is zero.	

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 nose radius 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.

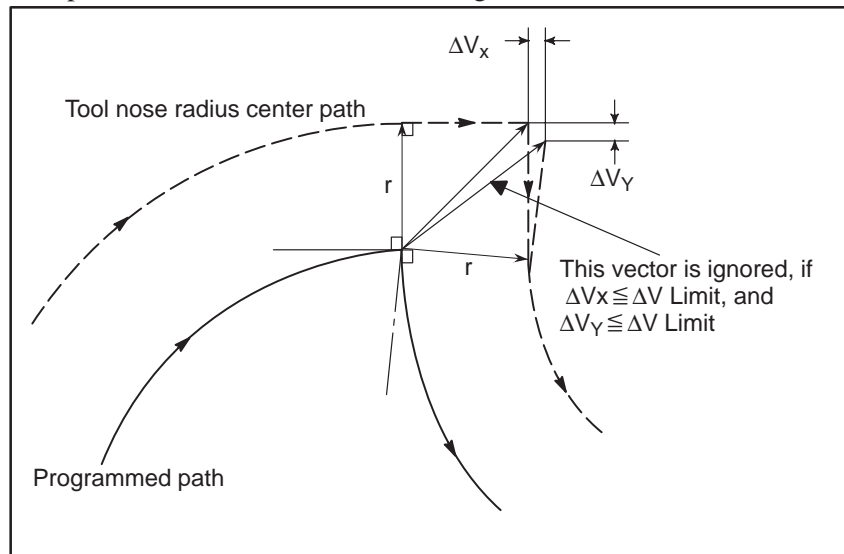


Setting bit 4 of parameter No. 395 enables the successful application of tool nose radius compensation, even when two or more blocks that do not contain tool movement are specified in succession.

- **Corner movement**

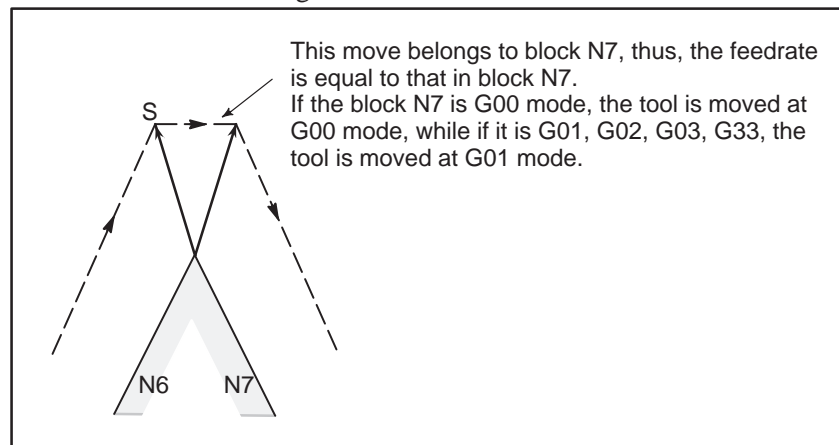
When two or more vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.

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.



- **Interruption of manual operation**

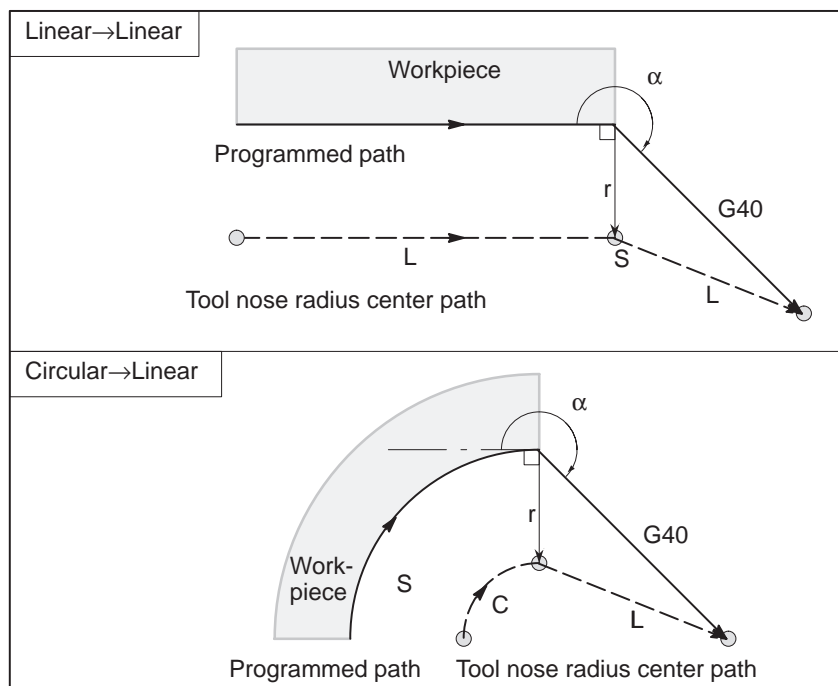
For manual operation during the tool nose radius compensation, refer to Section III-3.5, “Manual Absolute ON and OFF.”

14.3.4

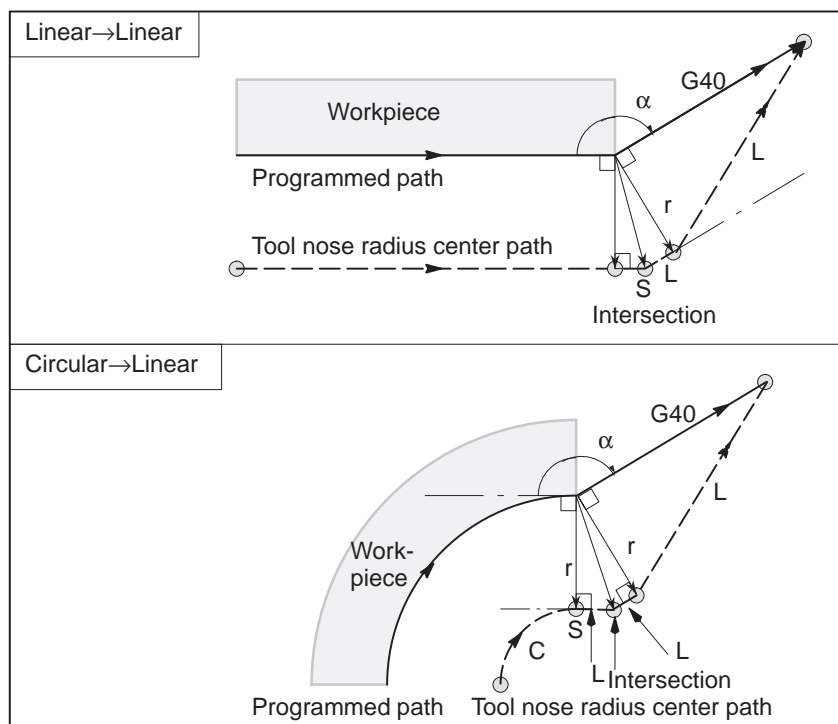
Tool Movement in Offset Mode Cancel

Explanations

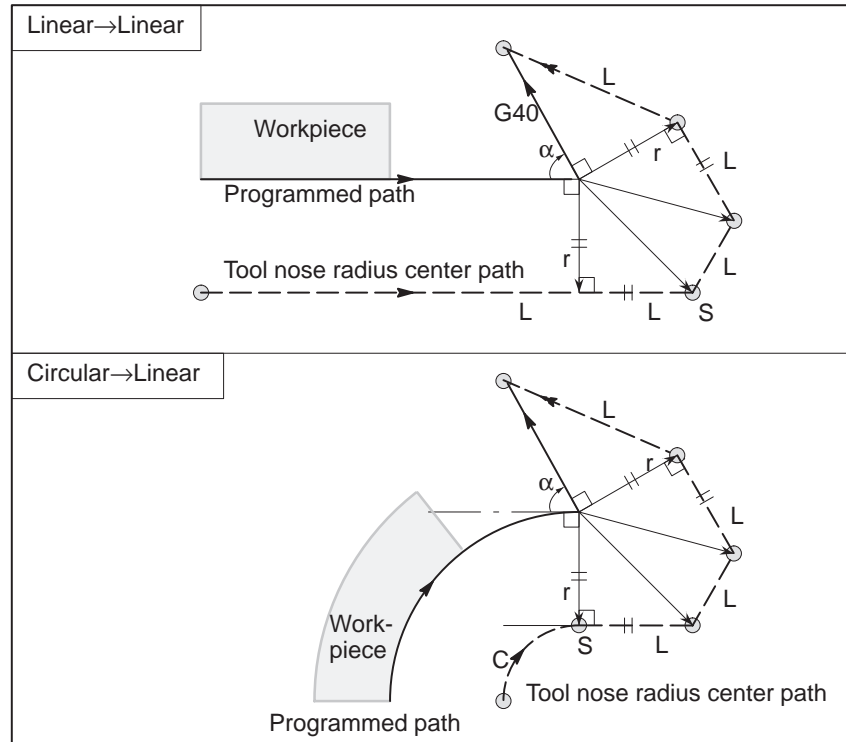
- Tool movement around an inside corner ($180^\circ \geq \alpha$)



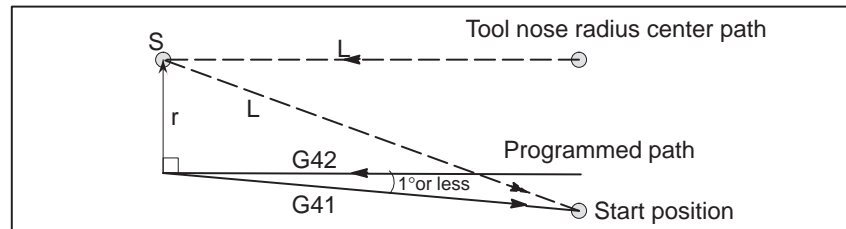
- Tool movement around an outside corner at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)



- Tool movement around an outside corner at an acute angle ($\alpha < 90^\circ$)

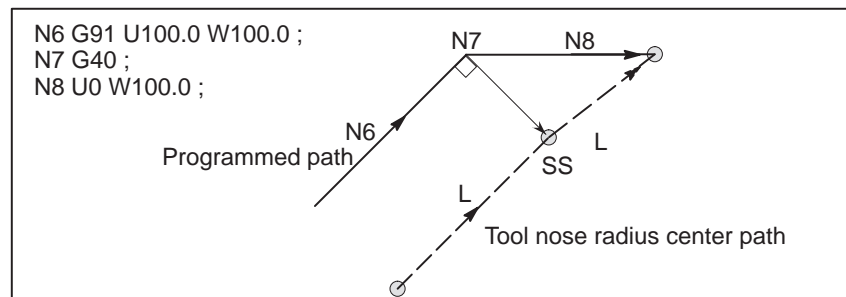


- Tool movement around the outside linear→linear at an acute angle less than 1 degree ($\alpha < 1^\circ$)



- A block without tool movement specified together with offset cancel

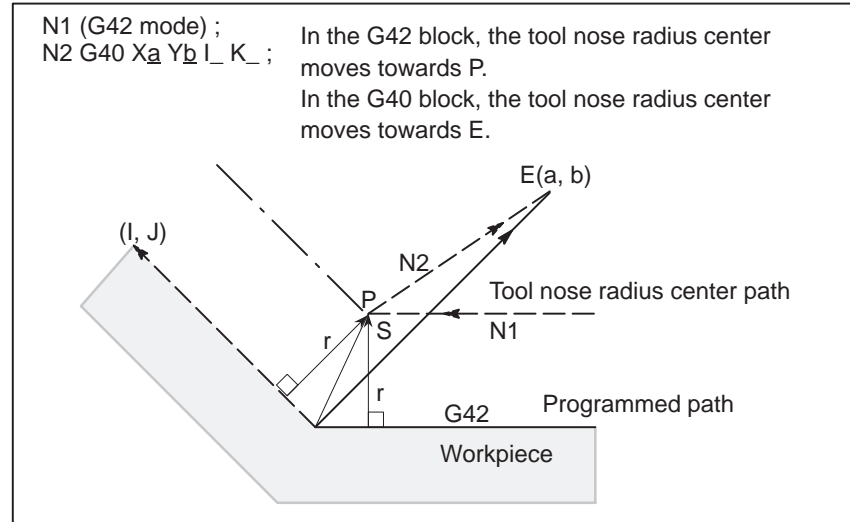
When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is cancelled in the next move command.



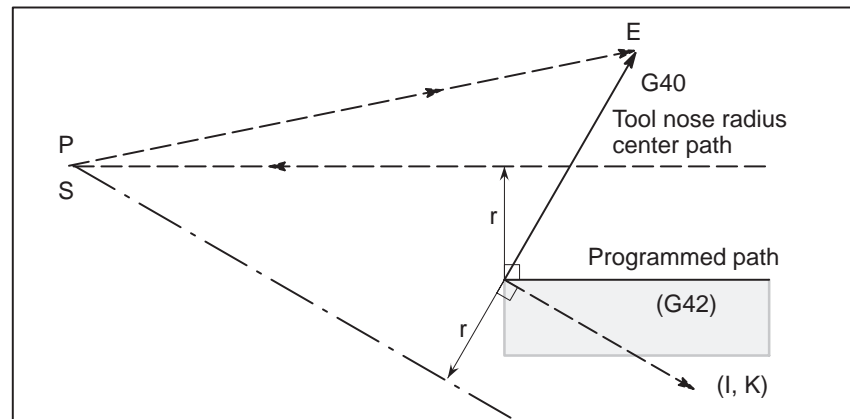
- **Block containing G40 and I_J_K_**

The previous block contains G41 or G42

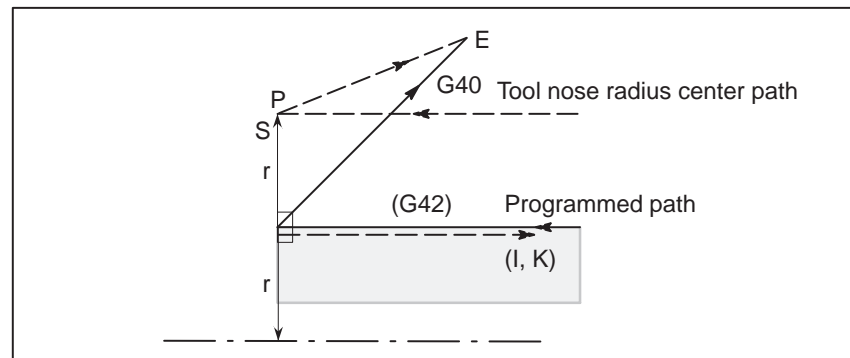
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



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.



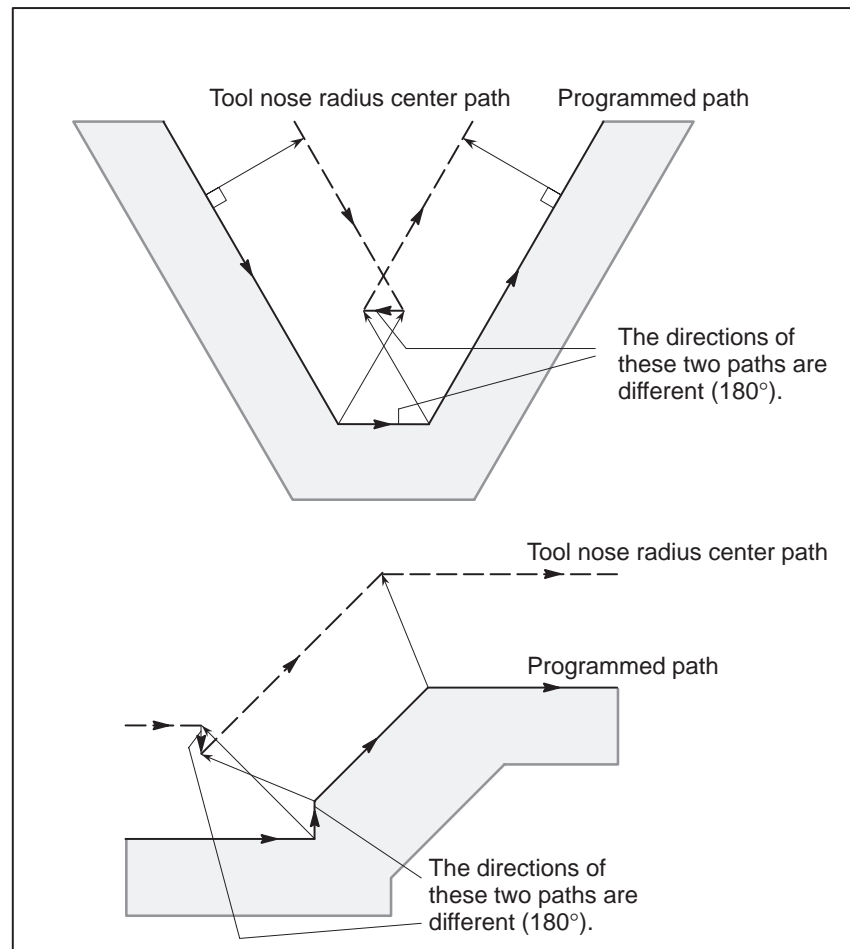
14.3.5 Interference Check

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.

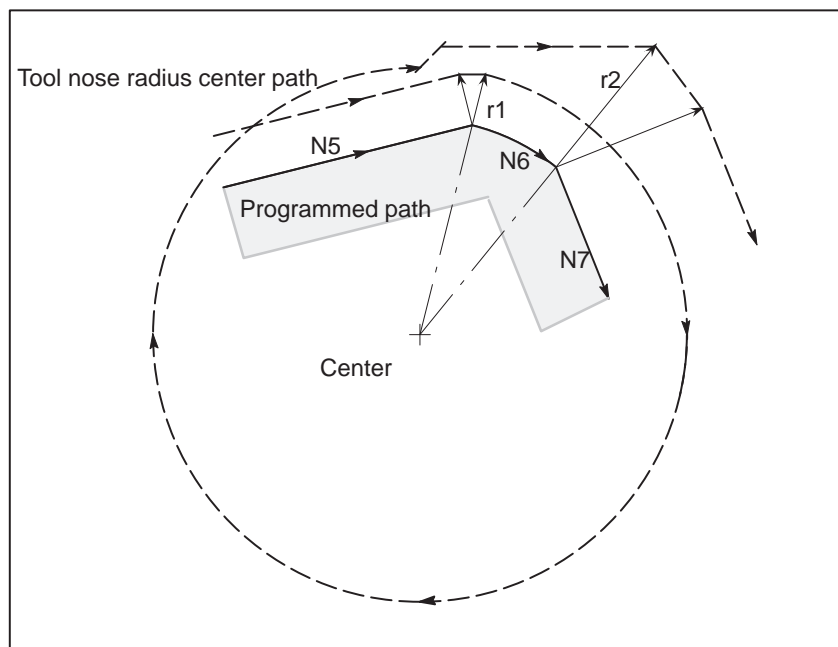
Explanations

- **Criteria for detecting interference**

- 1 The direction of the tool nose radius path is different from that of the programmed path (from 90 degrees to 270 degrees between these paths).



- ③ In addition to the condition ①, the angle between the start point and end point on the Tool nose radius 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).



(G41)

N5 G01 U20.0 W80.0 T1 ;

N6 G02 U-16.0 W32.0 I-80.0 K-20.0 T2 ;

N7 G01 U-50.0 W20.0 ;

(Tool compensation value corresponding to T1 : $r_1 = 20.0$)

(Tool compensation value corresponding to T2 : $r_2 = 60.0$)

In the above example, the arc in block N6 is placed in the one quadrant. But after tool nose radius compensation, the arc is placed in the four quadrants.

- **Correction of interference in advance**

① Removal of the vector causing the interference

When tool nose radius 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

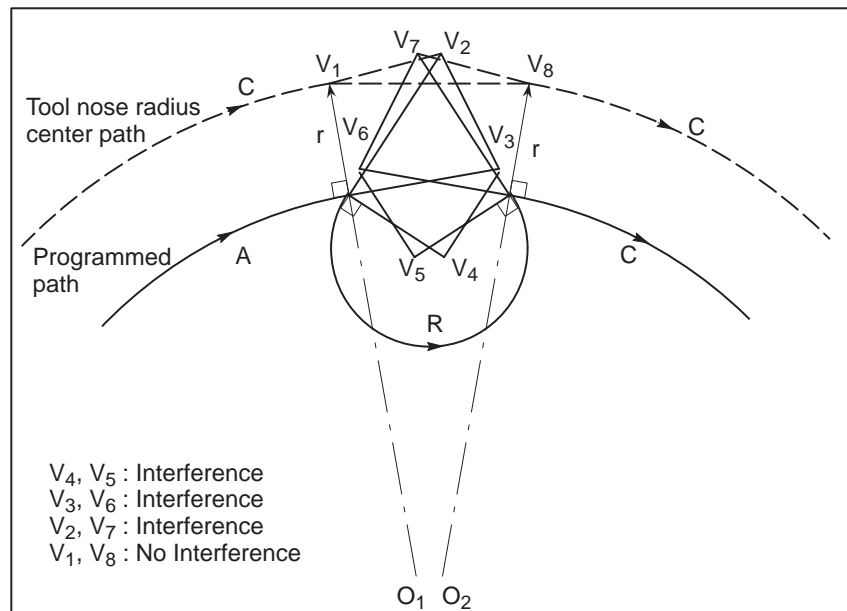
Interference V_2 and V_7 are Ignored

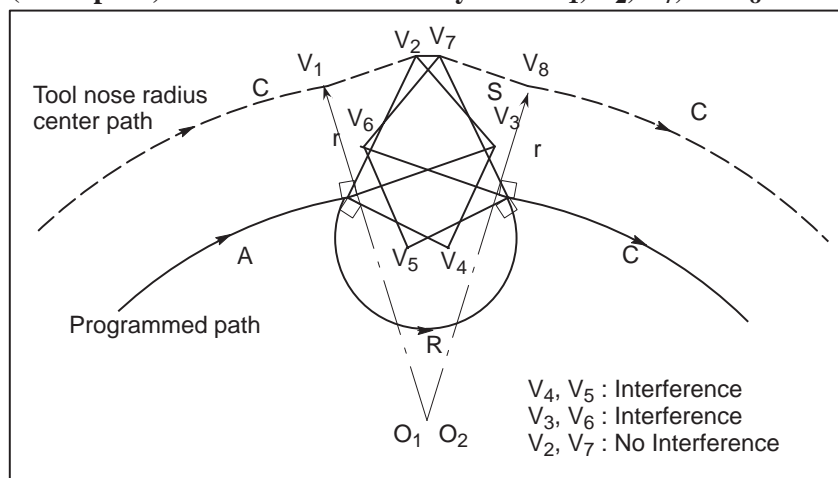
Check between V_1 and V_8

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.

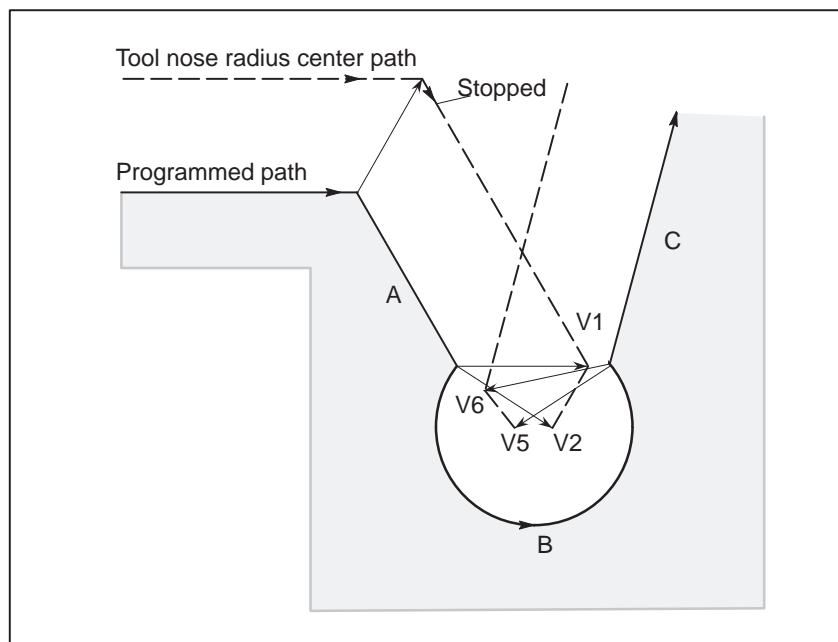
(Example 1) The tool moves linearly from V_1 to V_8



(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.

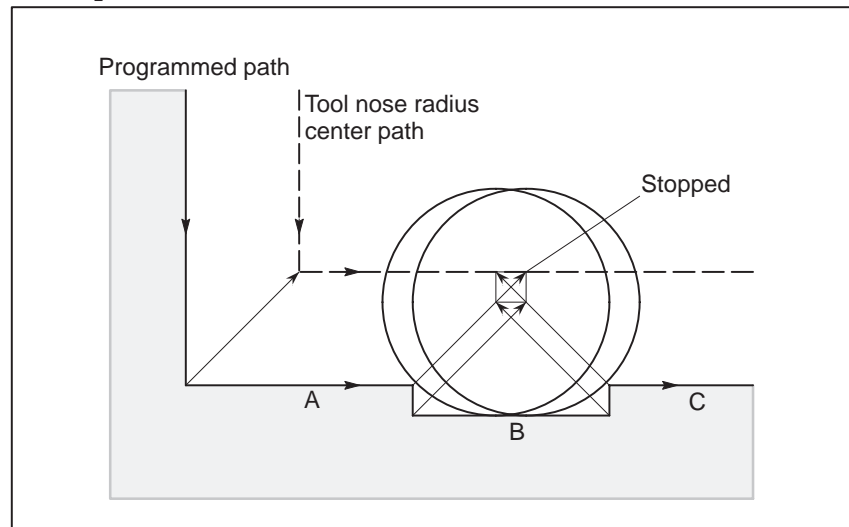
If the interference occurs after correction ① or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the alarm (No.41) is displayed and the tool is stopped immediately after execution of the preceding block. If the block is executed by the single block operation, 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.

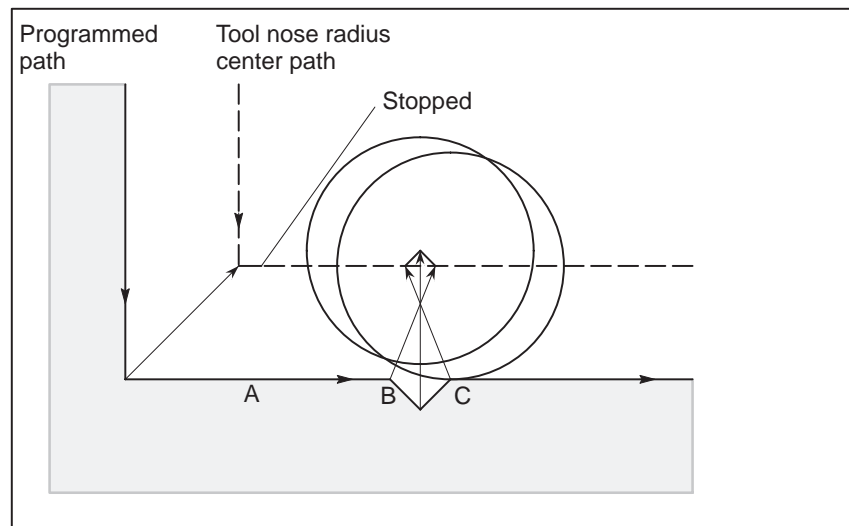
- When interference is assumed although actual interference does not occur

① Depression which is smaller than the tool nose radius compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after tool nose radius compensation the tool stops and an alarm(No.041) is displayed.

② Groove which is smaller than the tool nose radius compensation value



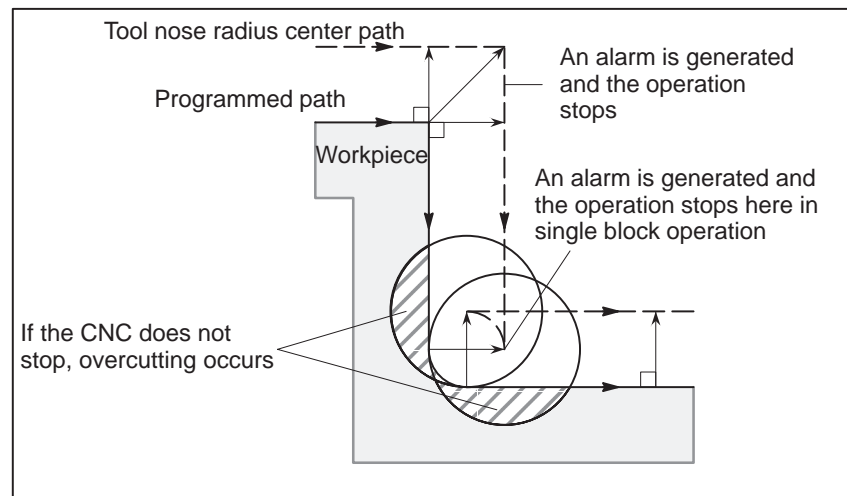
Like ①, the direction is reverse in block B.

14.3.6 Overcutting by Tool Nose Radius Compensation

Explanations

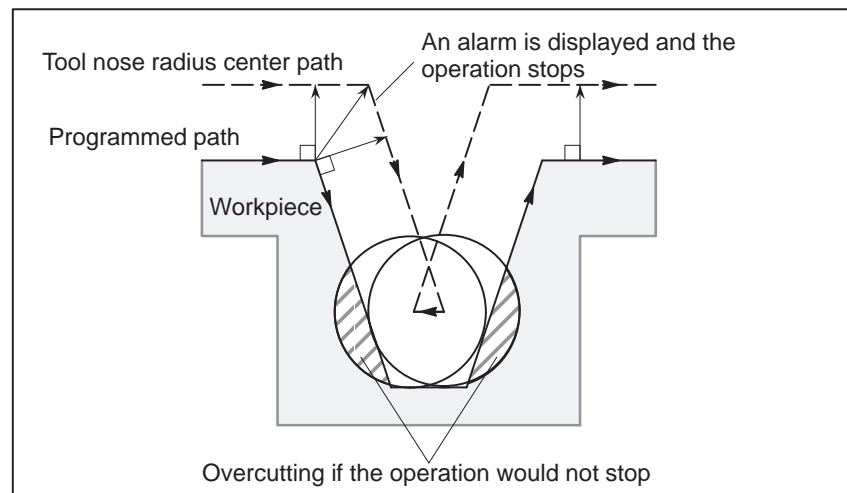
- **Machining an inside corner at a radius smaller than the tool nose radius**

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.



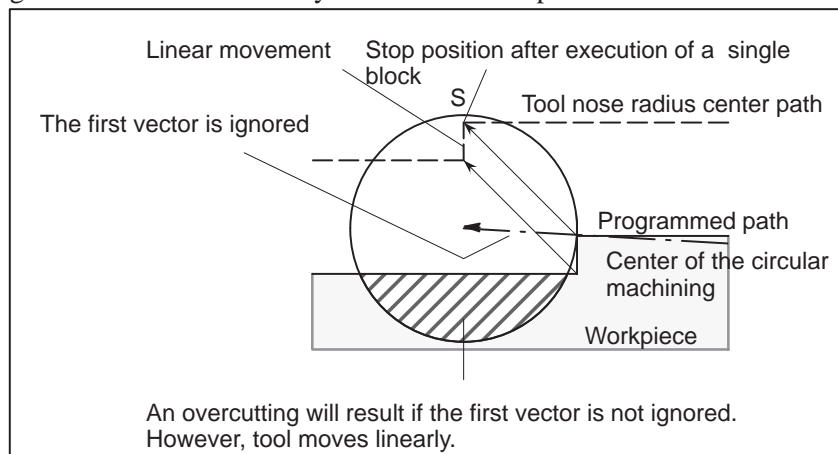
- **Machining a groove smaller than the tool nose radius**

Since the tool nose radius 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 nose radius**

When machining of the step is commanded by circular machining in the case of a program containing a step smaller than the tool nose 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.

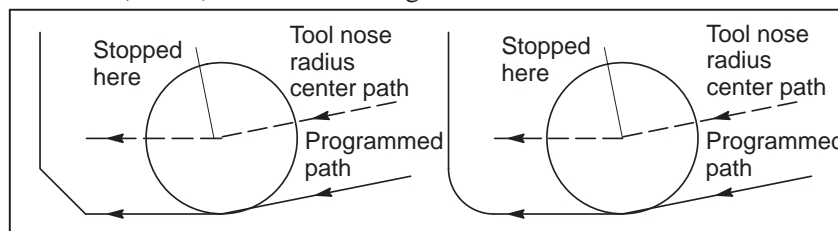


14.3.7 Correction in Chamfering and Corner Arcs

In chamfering or corner arcs, tool nose radius compensation only be performed when an ordinary intersection exists at the corner.

In offset cancel mode, a start-up block or when exchanging the offset direction, compensation cannot be performed, an alarm (No.39) is displayed and the tool is stopped.

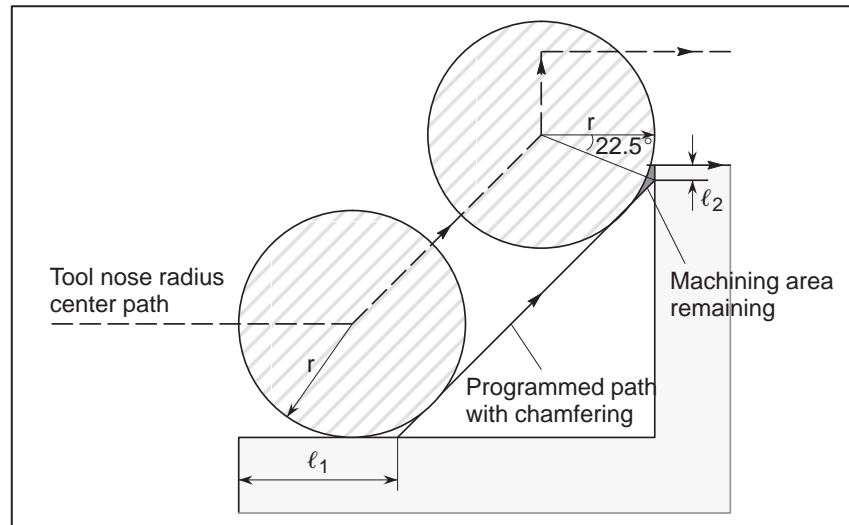
In inner chamfering or inner corner arcs, if the chamfering value or corner arc value is smaller than the tool nose radius value, the tool is stopped with an alarm (No.39) since overcutting will occur.



The valid inclination angle of the programmed path in the blocks before and after the corner is 1 degree or less so that the alarm (No.52, 54) generated by the calculating error of tool nose radius compensation does not occur.

- **When machining area remains or an alarm is generated**

The following example shows a machining area which cannot be cut sufficiently.

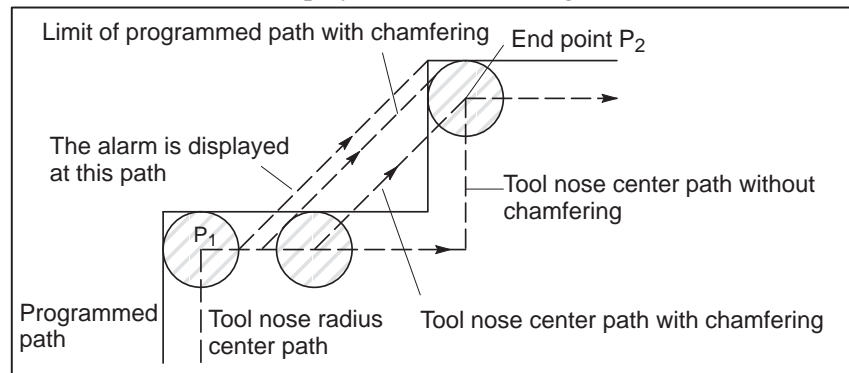


In inner chamfering, if the portion of the programmed path that is not a part of the chamfering (in the above figure l_1 or l_2) is in following range, insufficiently cut are will exist.

$$0 \leq l_1 \text{ or } l_2 < r \cdot \tan 22.5^\circ \quad (r : \text{tool nose radius})$$

Enlarged view on the remaining machining area

Alarm No.52 or 55 is displayed in the following cases :



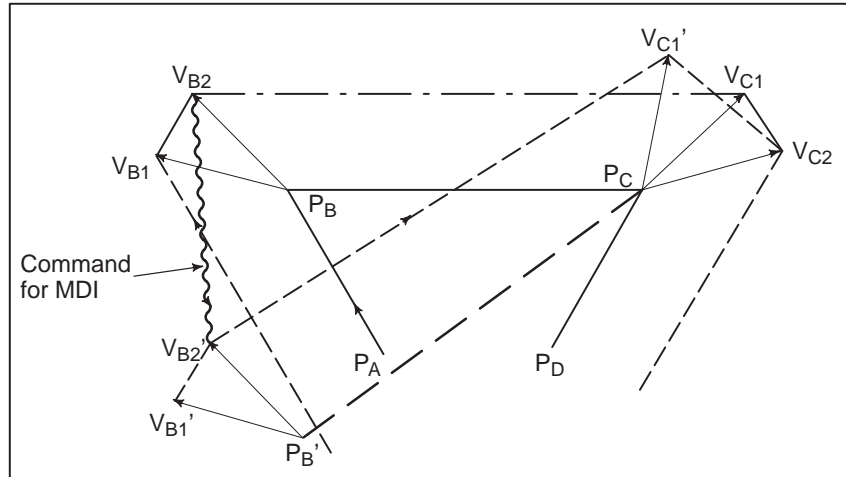
In outer chamfering with an offset, a limit is imposed on the programmed path. The path during chamfering coincides with the intersection points P_1 or P_2 without chamfering, therefore, outer chamfering is limited. In the figure above, the end point of the tool center path with chamfering coincides with the intersection point (P_2) of the next block without chamfering. If the chamfering value is more than the limit value specified, alarm No.52 or 55 will be displayed.

14.3.8 Input Command from MDI

Tool nose radius compensation is not performed for commands input from the MDI.

However, when automatic operation using the CNC tape composed of 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, tool nose radius compensation 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 .

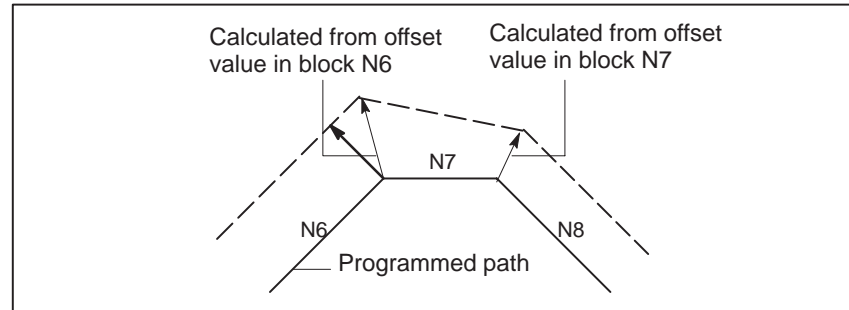
14.3.9

General Precautions for Offset Operations

- **Changing the offset value**

In general, the offset value is changed in cancel mode, or when changing tools. If the offset value is changed in offset mode, the vector at the end point of the block is calculated for the new offset value.

Additionally, the changing of hypothetical tool nose number and the changing of tool position offset are same.

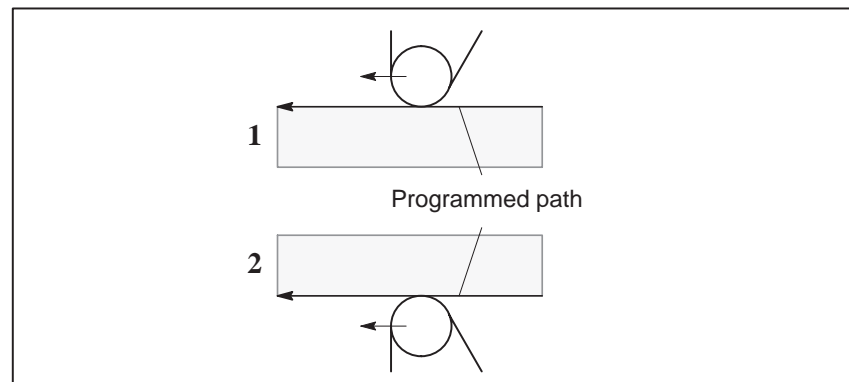


- **The polarity of the offset amount and the tool nose center path**

When a negative offset value is specified, the program is executed for the figure which is created by exchanging G41 for G42 or G42 for G41 in the process sheet.

A tool machining an inner profile will machine the outer profile, and tool machining the outer profile will machine the inner profile.

An example is shown below. In general, CNC machining is programmed assuming a positive offset value. When a program specifies a tool path as shown in **1**, the tool will move as shown in **2** if a negative offset is specified. The tool in **2** will move as shown in **1** when the sign of the offset value is reversed.



WARNING

When the sign of the offset value is reversed, the offset vector of the tool nose is reversed but the imaginary tool nose direction does not change.

Therefore, do not reverse the sign of the offset value when starting the machining meeting the imaginary tool nose to the start point.

14.4 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. 14.4 (a)).

Tool compensation can be specified without differentiating compensation for tool geometry from that for tool wear (Fig. 14.4.(b)).

In this case the operation of moving is same as the tool wear offset value

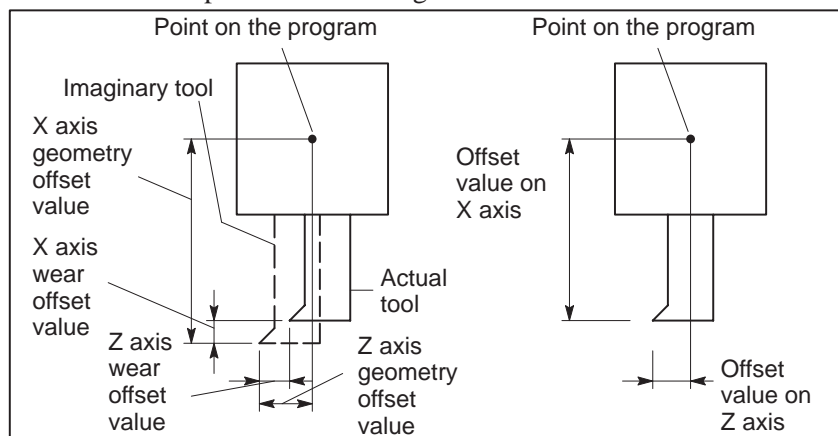


Fig. 14.4(a) Difference the tool geometry offset from tool wear offset

Fig. 14.4(b) Not difference the tool geometry offset from tool wear offset

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 T in a program.

The value is used for tool offset or tool nose radius compensation.

See subsec. 14.1.2 for details.

14.4.1 Tool Compensation and Number of Tool Compensation

- Valid range of tool compensation values

Table 14.4.1 shows the valid input range of tool compensation values.

Table 14.4.1 Valid range of tool compensation values

Increment system	Tool compensation value	
	Metric input (mm)	Inch input (inch)
IS-B	-999.999 to +999.999 mm	-99.9999 to +99.9999 inch
IS-C	-999.9999 to +999.9999 mm	-99.99999 to +99.99999 inch

The maximum tool wear compensation can be changed by setting parameter No.0729.

- Number of tool compensation

The memory can hold 16 or 32 tool compensation values.

NOTE

In the Series 0-TTC, the number of specified tool compensation values equals the number of tool compensations for each tool post.

14.4.2

Changing of Tool Offset Value

Offset values can be input by a program using the following command :

Format

G10 P_ X_ Z_ Y_ R_ Q_ ;

or

G10 P_ U_ W_ V_ C_ Q_ ;

P : Offset number

0 : Command of work coordinate system shift value

1–32 : Command of tool wear offset value
Command value is offset number

1000+(1–32) : Command of tool geometry offset value
(1–32) : Offset number

X : Offset value on X axis (absolute)

Z : Offset value on Z axis (absolute)

Y : Offset value on Y axis (absolute)

U : Offset value on X axis (incremental)

W : Offset value on Z axis (incremental)

V : Offset value on Y axis (incremental)

R : Tool nose radius offset value (absolute)

C : Tool nose radius offset value (incremental)

Q : Imaginary tool nose number

In an absolute command, the values specified in addresses X, Y, Z, and R are set as the offset value corresponding to the offset number specified by address P. In an incremental command, the value specified in addresses U, W, V, and C is added to the current offset value corresponding to the offset number.

NOTE

- 1 Addresses X, Z, Y and U, W, V can be specified in the same block.
- 2 Use of this command in a program allows the tool to advance little by little. This command can also be used input offset values one at a time from a tape by specifying this command successively instead of inputting these values one at a time from the MDI unit.
- 3 With P0 (workpiece coordinate system shift amount specification), only an X-/Z-axis shift can be entered.

14.5

AUTOMATIC TOOL OFFSET (G36, G37)

When a tool is moved to the measurement position by execution of a command given to the CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the command measurement position and uses it as the offset value for the tool. When the tool has been already offset, it is moved to the measurement position with that offset value. If the CNC judges that further offset is needed after calculating the difference between the coordinate values of the measurement position and the commanded coordinate values, the current offset value is further offset.

Refer to the instruction manuals of the machine tool builder for details.

Explanations

- **Coordinate system**

When moving the tool to a position for measurement, the coordinate system must be set in advance. (The work coordinate system for programming is used in common.)

- **Movement to measurement position**

A movement to a measurement position is performed by specifying as follows in the MDI, or AUTO mode :

G36 Xx_a ; or G37 Zz_a ;

In this case, the measurement position should be x_a or z_a (absolute command).

Execution of this command moves the tool at the rapid traverse rate toward the measurement position, lowers the feedrate halfway, then continues to move it until the approach end signal from the measuring instrument is issued. When the tool tip reaches the measurement position, the measuring instrument outputs the measurement position reach signal to the CNC which stops the tool.

- **Offset**

The current tool offset value is further offset by the difference between the coordinate value (α or β) when the tool has reached the measurement position and the value of x_a or z_a specified in G36Xx_a or G37Zz_a.

Offset value x = Current offset value x + (α - x_a)

Offset value z = Current offset value z + (β - z_a)

x_a : Programmed X-axis measurement point

z_a : Programmed Z-axis measurement point

These offset values can also be altered from the MDI keyboard.

• Feedrate and alarm

The tool, when moving from the starting position toward the measurement position predicted by x_a or z_a in G36 or G37, is fed at the rapid traverse rate across area **A**. Then the tool stops at point T ($x_a - \gamma_x$ or $z_a - \gamma_z$) and moves at the measurement feedrate set by parameter (No.558) across areas **B**, **C**, and **D**. If the approach end signal turns on during movement across area B, alarm is generated. If the approach end signal does not turn on before point V, and tool stops at point V and alarm (No.080) is generated.

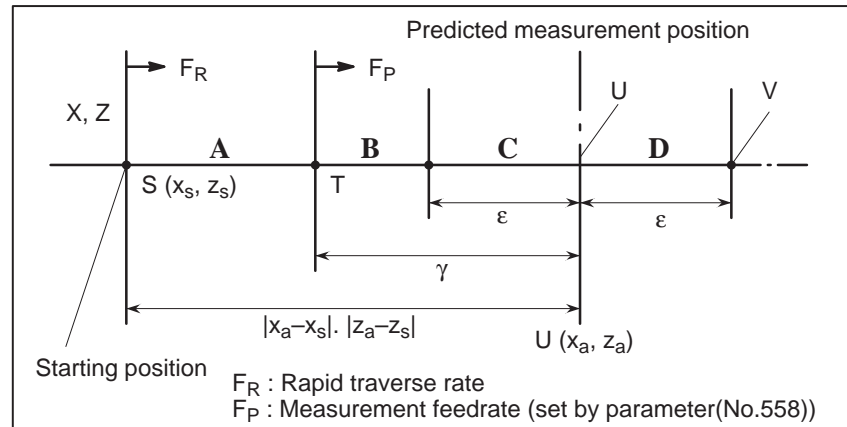
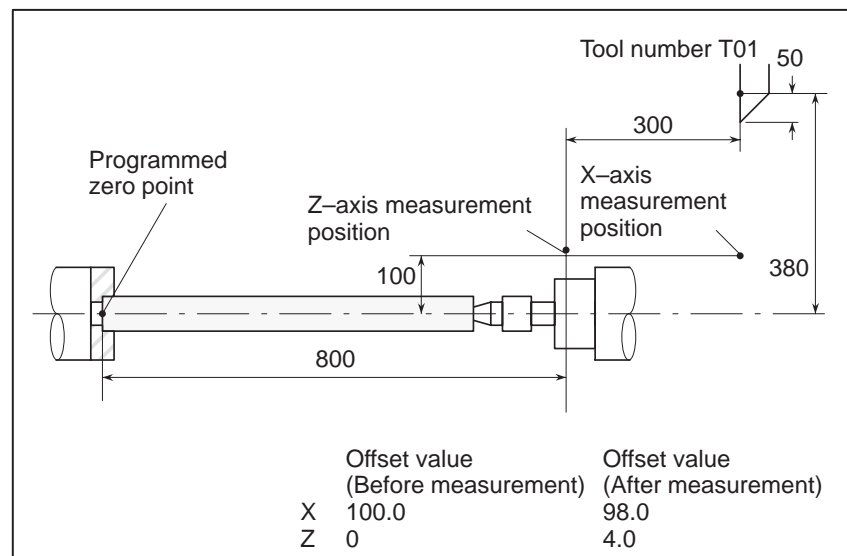


Fig. 14.5 Feedrate and Alarm

Examples



G50 X760.0 Z1100.0 ; Programming of absolute zero point
(Coordinate system setting)

S01 M03 T0101 ; Specifies tool T1, offset number 1, and spindle revolution

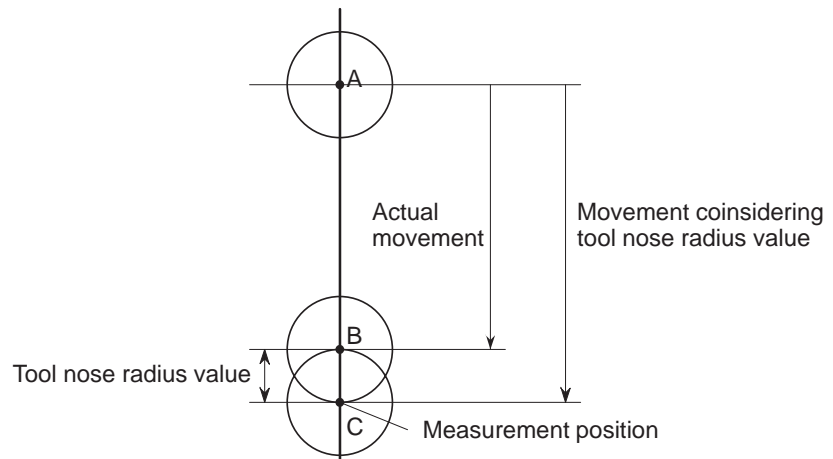
G36 X200.0 ; Moves to the measurement position
If the tool has reached the measurement position at X198.0 ; since the correct measurement position is 200 mm, the offset value is altered by $198.0 - 200.0 = -2.0\text{mm}$.

G00 X204.0 ;	Refracts a little along the X axis.
G37 Z800.0 ;	Moves to the Z-axis measurement position. If the tool has reached the measurement position at X804.0, the offset value is altered by 804.0–800.0=4.0mm.
T0101 ;	Further offsets by the difference. The new offset value becomes valid when the T code is specified again.

WARNING

- 1 Measurement speed(Fp), γ , and ε are set as parameters (Fp : No.558, γ : No.731, 732, ε : No.133, 134) by machine tool builder. ε must be positive numbers so that $\gamma > \varepsilon$.
- 2 Cancel the tool nose radius compensation before G36, G37.
- 3 When a manual movement is inserted into a movement at a measurement feedrate, return the tool to the position before the inserted manual movement for restart.
- 4 When using the optional tool nose radius compensation function, the tool offset amount is determined considering the value of tool nose R. Make sure that tool nose radius value is set correctly.

Example) When the tool nose center coincides with the start point.



The tool actually moves from point A to point B, but the tool offset value is determined assuming that the tool moves to point C considering the tool nose radius value.

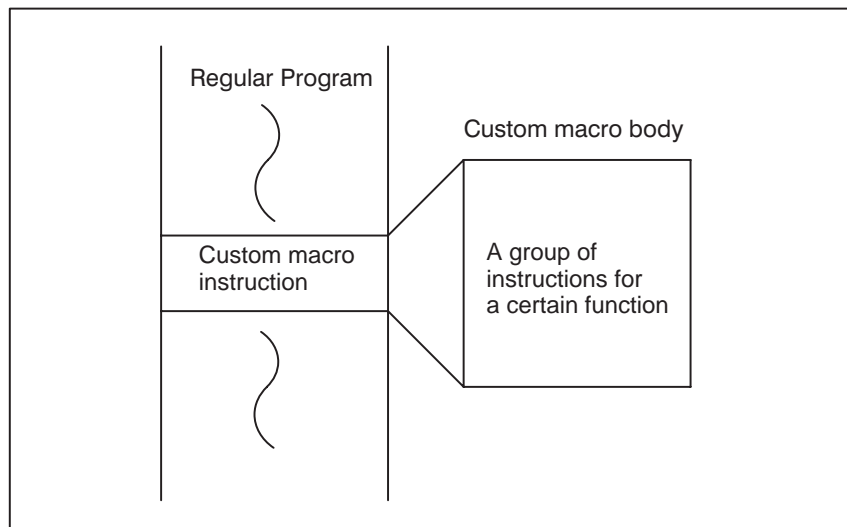
NOTE

- 1 When there is no T code command before G36 or G37, alarm No. 81 is generated.
- 2 When a T code is specified in the same block as G36 or G37, alarm No.82 is generated.

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.

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. 040 #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.

15.2 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.

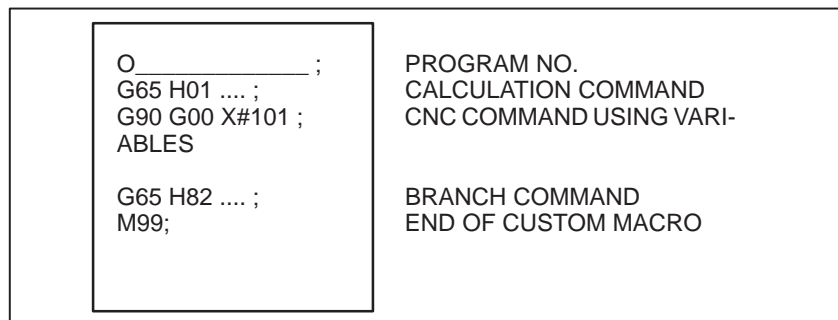


Fig. 15.2 Construction of the custom macro body

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 = 100, 101

(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#103 ... F15 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.

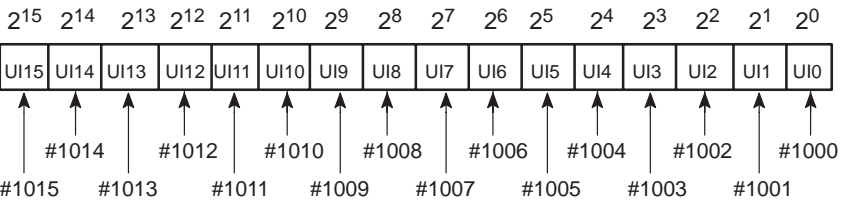
- (3) Display and setting of variable values
- Variable values can be displayed on the CRT screen, and a value can be set in a variable by using the MDI keys.

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) 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.

$$\#1032 = \sum_{i=0}^{15} \#(1000+i) \times 2^i$$

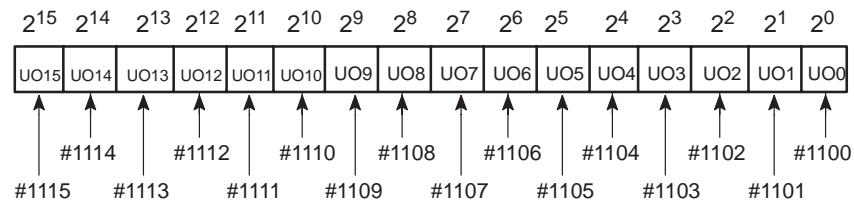
- NOTE
- 1 No value can be substituted into system variables #1000 to #1032.

2 System variables #1000 to #1015 can be displayed by diagnostic function.
No.130 U10 to U17
No.131 U18 to U15

3 System variables #1000 to #1032 can be used only when PMC is combined.

(b) Interface output signals #1100 to #1115, #1132, #1133

A value can be substituted into system variables #1100 to #1115 for sending the interface signals.



Output signal	Variable value
Contact closed	1
Contact open	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.

NOTE

- 1

If any other number than '0' or '1' is substituted into system variables #1100 to #1115, it is treated as '1'.
- 2

It is possible to read the values of system variables #1100 to #1133.
- 3

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
- 4

System variables #1100 to #1133 can be used only when PMC is combined.

(c) Tool offset values #2001 to #2932

Offset values can be checked from the values of system variables #2001 to #2932, used to hold tool offset values. By assigning a value to system variable #i, an offset value can be modified.

A tool position offset for the X-axis is handled as a radius when the offset is determined from a specified radius. A tool position offset for the X-axis is handled as a diameter when the offset is determined from a specified diameter.

	Tool offset number	Tool offset value	Wear offset value	Geometry offset value
X	1 to 32	#2001 to #2032	#2001 to #2032	#2701 to #2732
Z	1 to 32	#2101 to #2132	#2101 to #2132	#2801 to #2832
R	1 to 32	#2201 to #2232	#2201 to #2232	#2901 to #2932
T	1 to 32	#2301 to #2332	#2301 to #2332	#2301 to #2332
Y	1 to 32	#2401 to #2432	#2401 to #2432	#2401 to #2432

(Example) #103=#2005

The X-axis tool position offset value of offset number 5 is assigned to variable #103. When the offset is 1.5 mm, the value of #103 is 1500.

(d) Workpiece coordinate system shift amounts #2501, #2601

The amounts of workpiece coordinate system shift can be determined by reading system variables #2501 and #2601. The amounts of workpiece coordinate system shift can also be modified by assigning values to system variables #i.

Controlled axis	Workpiece coordinate system shift amount
X-axis	#2501
Z-axis	#2601

(e) Clock information #3011, 3012 (option)

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	#3012

(Example) When it is May 20, 1997 4:17 5" PM

#3012=19970520,

#3012=161705

- (f) Number of necessary parts, number of machined parts
By using system variables, the number of parts required and the number of parts machined can be read and assigned.

Kind	System variable
Number of machined parts	#3901
Number of necessary parts	#3902

- (g) 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.

NOTE

Do not substitute a negative value.

Variables	Modal information
#4001	G CODE (GROUP 01)
#4002	G CODE (GROUP 02)
#4003	G CODE (GROUP 03)
⋮	
#4022	G CODE (GROUP 22)
#4109	F 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.

(h) Position information #5001 to #5124

The position information can be known by reading system variables #5001 to #5124. 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 Block end point position of Z axis Block end point position of 3rd axis (C axis) Block end point position of 4th axis (Y axis)	Possible	Not considered. Program command position
#5021 #5022 #5023 #5024	X axis coordinate position Z axis coordinate position 3rd axis (Cf axis) coordinate position 4th axis (Y axis) coordinate position	Impossible	Considered. Position of tool reference point (Machine coordinate)
#5041 #5042 #5043 #5044	Present position of X axis Present position of Z axis Present position of 3rd axis (Cf axis) Present position of 4th axis (Y axis)	Impossible	Considered. Position of tool reference point
#5061 #5062 #5063 #5064	Skip signal position of X axis Skip signal position of Z axis Skip signal position of 3rd axis (Cf axis) Skip signal position of 4th axis (Y axis)	Possible	Considered. Position of tool reference point
#5081 #5082 #5084	Tool position offset amount or wear offset amount of X axis Tool position offset amount or wear offset amount of Z axis Tool position offset amount or wear offset amount of 4th axis (Y axis)	impossible	
#5121 #5122 #5124	Graphic offset amount of X axis Graphic offset amount of Z axis Graphic offset amount of 4th axis (Y axis)	impossible	

15.2.3 Operation Instruction and Branch Instruction (G65)

General format:

G65HmP#i Q#j R#k ;

m : 01 to 99. An operation instruction or branch instruction function is represented.

#i : Name of variable used to hold the result of an operation

#j : Name of variable on which an operation is to be performed.

(A constant can also be specified.)

#k : Name of variable on which an operation is to be performed.

(A constant can also be specified.)

(Meaning) $\#i = \#j \oplus \#k$

↑
Specified using the Hm operator

Example

P#100 Q#101 R#102 ... #100=#101 \oplus #102

P#100 Q#101 R15 ... #100=#101 \oplus 15

P#100 Q-100 R#102 ... #100=-100 \oplus #102

P#100 Q120 R-50 ... #100=120 \oplus -50

P#100 Q-#101 R#102 .. #100=-#101 \oplus #012

NOTE

- 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 input increment is 1/1000 degree.
(Example) 100 -- 0.1°

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 $

Table 15.2.3

G code	H code	Function	Definition
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	$\#i = (\#i \times \#j) \div \#k$
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	GOTOn
G65	H81	Conditional divergence 1	IF#j = #k, GOTOn
G65	H82	Conditional divergence 2	IF#j \neq #k, GOTOn
G65	H83	Conditional divergence 3	IF#j > #k, GOTOn
G65	H84	Conditional divergence 4	IF#j < #k, GOTOn
G65	H85	Conditional divergence 5	IF#j \geq #k, GOTOn
G65	H86	Conditional divergence 6	IF#j \leq #k, GOTOn
G65	H99	P/S alarm occurrence	P/S alarm number 500 +n occurrence

● **Variable arithmetic command**

(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×#103)

- (e) Division $\#i = \#j \div \#k$
 G65 H05 P#i Q#j R#k;
 [Example] G65 H05 P#101 Q#102 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 Q#102 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 $\#i = \#j \times \text{SIN} (\#k)$ (degree unit)
 G65 H31 P#i Q#j R#k ;
 [Example] G65 H31 P#101 Q#102 R#103 ; ($\#101 = \#102 \times \text{SIN} (\#103)$)

- (r) Cosine $\#i = \#j \times \text{COS} (\#k)$ (degree unit)
 G65 H32 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H32 P#101 Q#102 R#103 ;
 ($\#101 = \#102 \times \text{COS} (\#103)$)
- (s) Tangent $\#i = \#j \times \text{TAN} (\#k)$ (degree unit)
 G65 H33 P $\#i$ Q $\#j$ R $\#k$;
 [Example] G65 H33 P#101 Q#102 R#103 ;
 ($\#101 = \#102 \times \text{TAN} (\#103)$)
- (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 ;
 ($\#101 = \text{ATAN} (\#102 / \#103)$)

NOTE

- 1 Angle in (q) to (t) must be indicated by degree and the least input increment is 1/1000 degree.
- 2 If either Q or R necessary for each arithmetic operation was not indicated, its value is calculated as '0'.
- 3 All figures below decimal point are truncated if each arithmetic result includes decimal point.

- **Branch instruction**

- (a) Unconditional branch
 G65 H80 P n ; n : Sequence number
 [Example] G65 H80 P120 ; (Diverge to N120)
- (b) Conditional divergence 1 ($\#j = \#k$)
 G65 H81 P n Q $\#j$ R $\#k$; n : Sequence number
 [Example] G65 H81 P1000 Q#101 R#102 ;
 $\#101 = \#102$, go to N1000
 $\#101 \neq \#102$, go to next
- (c) Conditional divergence 2 ($\#j \neq \#k$)
 G65 H82 P n Q $\#j$ R $\#k$; n : Sequence number
 [Example] G65 H82 P1000 Q#101 R#102 ;
 $\#101 \neq \#102$, go to N1000
 $\#101 = \#102$, go to
- (d) Conditional divergence 3 ($\#j > \#k$)
 G65 H83 P n Q $\#j$ R $\#k$; n : Sequence number
 [Example] G65 H83 P1000 Q#101 R#102 ;
 $\#101 > \#102$, go to N1000
 $\#101 \leq \#102$, go to next
- (e) Conditional divergence 4 ($\#j < \#k$)
 G65 H84 P n Q $\#j$ R $\#k$; n : Sequence number
 [Example] G65 H84 P1000 Q#101 R#102 ;
 $\#101 < \#102$, go to N1000
 $\#101 \geq \#102$, go to next
- (f) Conditional divergence 5 ($\#j \geq \#k$)
 G65 H85 P n Q $\#j$ R $\#k$; n : Sequence number
 [Example] G65 H85 P1000 Q#101 R#102 ;
 $\#101 \geq \#102$, go to N1000
 $\#101 < \#102$, go to next

(g) Conditional divergence 6 ($\#j \leq \#k$)

G65 H86 Pn Q#j R#k ; n : Sequence number

[Example] G65 H86 P1000 Q#101 R#102 ;

#101 \leq #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

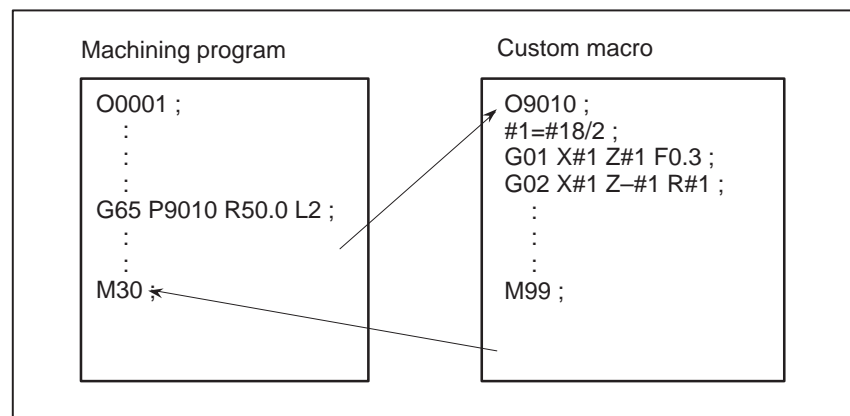
Notes on Custom Macro

- 1) How to input "##"
When " /# EOB" key is depressed after address, # 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 The operation and branch instructions 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 The operation and branch instructions block does not stop even if single block stop is turned on. However, by setting parameter No. 011#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 F0.3 ;
```

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 16.1Types 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#1Z#2; results in G00X0;.

- **Custom macro variables common to tool posts (0–TTC)**

In the 0–TTC macro variables are provided for each tool post. Specifying bit 5 or 6 of parameter Nos.047 and 218 allows some of the common variables to be used for all tool posts.

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#3Z200.0;

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 to #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 to #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.

- **Tool compensation values**

When the system does not differentiate tool geometry compensation from tool wear compensation, use variable numbers for wear compensation.

Table 16.2 (b) System variables for tool compensation memory C

Compensation number	X axis tool position		Z axis tool position		Tool nose radius compensation value		Imaginary tool nose position T	Y axis current position offset	
	Wear	Geometry	Wear	Geometry	Wear	Geometry		Wear	Geometry
1 : 32	#2001 : #2032	#2701 : #2732	#2101 : #2132	#2801 : #2832	#2201 : #2232	#2901 : #2932	#2301 : #2332	#2401 : #2432	#2451 : #2482

- **Macro alarms**

Table 16.2 (c) System variable for macro alarms

Variable number	Function
#3000	When a value from 0 to 200 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 "501 TOOL NOT FOUND."

- **Time information**

Time information can be read and written.

Table 16.2 (d) System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 1-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) (This is option). Year/month/day information is converted to an apparent decimal number. For example, March 28, 1997 is represented as 19970328.
#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 (e) 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 (f) System variable (#3004) for automatic operation control

#3004	Feed hold	Feedrate Override
0	Enabled	Enabled
1	Disabled	Enabled
2	Enabled	Disabled
3	Disabled	Disabled

- When the power is turned on, the value of this variable is 0.
- When feed hold is disabled:
 - ① 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.
 - ② 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.

• Settings

Settings can be read and written. Binary values are converted to decimals.

#3005								
Setting	#15	#14	#13	#12	#11	#10	#9	#8
							TAPEF	
Setting	#7	#6	#5	#4	#3	#2	#1	#0
				SEQ		INCH	ISO	TVON
#9 (TAPEF) : Whether to use the FS15 tape format conversion capability								
#5 (SEQ) : Whether to automatically insert sequence numbers								
#2 (INCH) : Millimeter input or inch input								
#1 (ISO) : Whether to use EIA or ISO as the output code								
#0 (TVON) : Whether to make a TV check								

• 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 (g) 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 (h) System variables for modal information

Variable number	Function
#4001	G00, G01, G02, G03, G33, G34 (Group 01)
#4002	G96, G97 (Group 02)
#4003	(Group 03)
#4004	G68, G69 (Group 04)
#4005	G98, G99 (Group 05)
#4006	G20, G21 (Group 06)
#4007	G40, G41, G42 (Group 07)
#4008	G25, G26 (Group 08)
#4009	G22, G23 (Group 09)
#4010	G80 – G89 (Group 10)
#4011	(Group 11)
#4012	G66, G67 (Group 12)
#4014	G54–G59 (Group 14)
#4015	(Group 15)
#4016	G17 – G19 (Group 16)
:	:
#4022	(Group 22)
#4109	F 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, 33 or 34.

- **Current position**

Position information cannot be written but can be read.

Table 16.2 (i) System variables for position information

Variable number	Position information	Coordinate system	Tool compensation value	Read operation during movement
#5001 to #5004	Block end point	Workpiece coordinate system	Not included	Enabled
#5021 to #5026	Current position	Machine coordinate system	Included	Disabled
#5041 to #5046	Current position	Workpiece coordinate system		
#5061 to #5064	Skip signal position			Enabled
#5081 to #5084	Tool offset value			Disabled
#5101 to #5104	Deviated servo position			

· The first digit (from 1 to 8) represents an axis number.

- Tool position offset information represents the value being used for execution, not the previous value.
- Skip signal position information represents the position where the skip signal is turned on in a G31 (skip function) block.
If the skip signal is not turned on in a G31 block, skip signal position information represents the end point of the specified block.
- 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 origin offset values)**

Workpiece origin offset values can be read and written.

Table 16.2 (j) System variables for workpiece origin offset values

Variable number	Function
#2550	X-axis external workpiece origin offset value
#2551	X-axis workpiece origin offset value 1 (G54)
⋮	⋮
#2556	X-axis workpiece origin offset value 6 (G59)
#2650	Z-axis external workpiece origin offset value
#2651	Z-axis workpiece origin offset value 1 (G54)
⋮	⋮
#2656	Z-axis workpiece origin offset value 6 (G59)
#2750	C-axis external workpiece origin offset value
#2751	C-axis workpiece origin offset value 1 (G54)
⋮	⋮
#2756	C-axis workpiece origin offset value 6 (G59)
#2850	Y-axis external workpiece origin offset value
#2851	Y-axis workpiece origin offset value 1 (G54)
⋮	⋮
#2856	Y-axis workpiece origin offset value 6 (G59)

NOTE

Variables #2550 to #2856 are optional variables for the 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-[ROUND[#1]+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

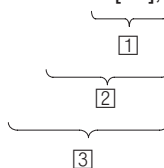
- **Priority of operations**

① Functions

② Operations such as multiplication and division (*, /, AND)

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{\epsilon}{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	Min $\left \frac{\epsilon}{b} \right $, $\left \frac{\epsilon}{c} \right $ (*2)
a = SIN [b] a = COS [b]	5.0×10 ⁻⁹	1.0×10 ⁻⁸	Absolute error(*3) $\left \epsilon \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.

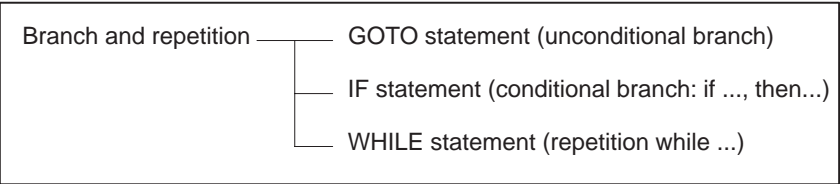
Explanations

- **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 No.061#5 is 1.
 - Macro blocks are not regarded as blocks that involve no movement in the tool nose radius compensation mode (see Section 16.7).
- **NC statements that have the same property as macro statements**
 - NC statements that include a subprogram call command (such as subprogram calls by M98 or other M codes, or by T codes) and also include an O, N, P, or L address 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 99999 is specified, alarm No. 128 occurs. A sequence number can also be specified using an expression.

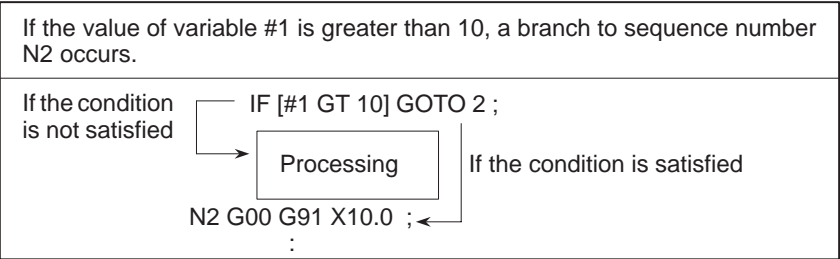
```
GOTO n ;      n: Sequence number (1 to 99999)
```

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(>)
GE	Greater than or equal to(\geq)
LT	Less (<)
LE	Less than or equal to(\leq)

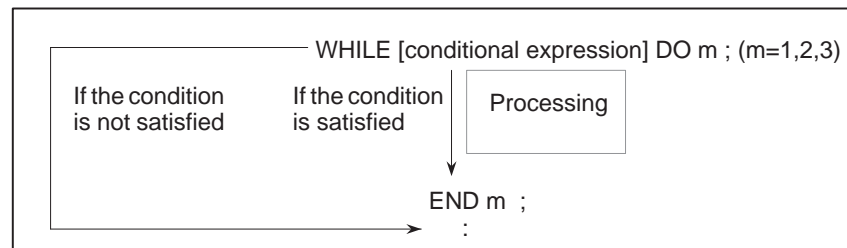
Sample program

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.

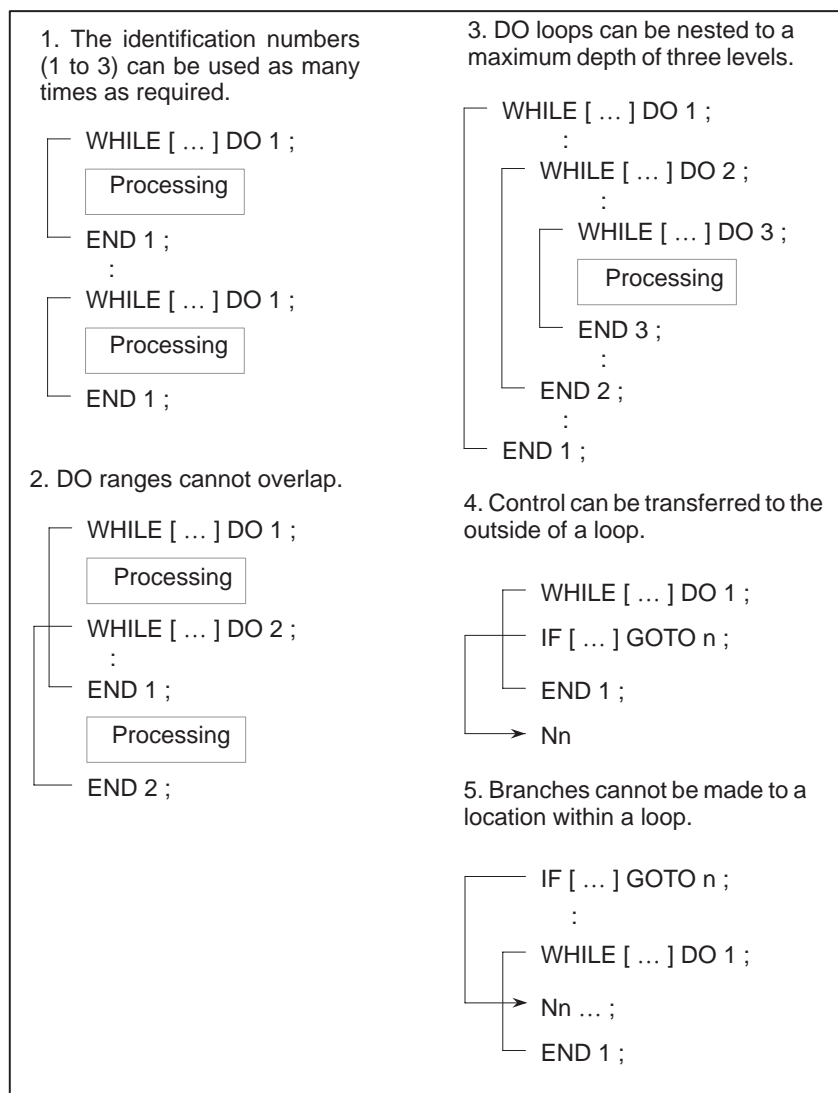


Explanations

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.

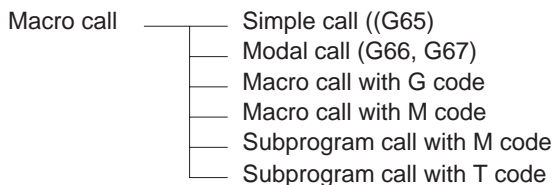
Sample program

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:



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

O0001 ; : G65 P9010 L2 A1.0 B2.0 ; : M30 ;	O9100 ; #3=#1+#2 ; IF [#3 GT 360] GOTO 9 ; G00 X#3 ; N9 M99 ;
--------------------------------------------------------	---------------------------------------------------------------------------

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**
- **Mixture of argument specifications I and II**
- **Position of the decimal point**

G65 must be specified before any argument.

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.

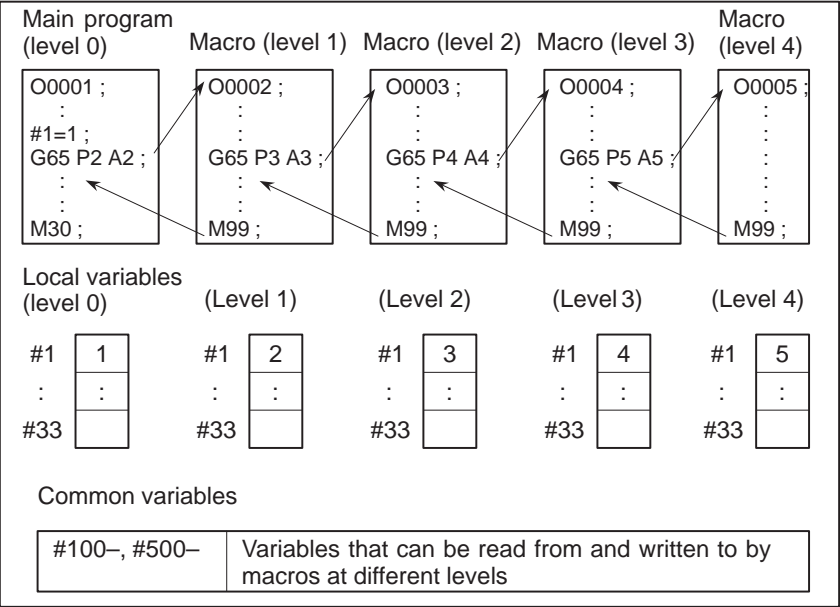
The units used for argument data passed without a decimal point correspond to the least input increment of each address. The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

• **Call nesting**

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).

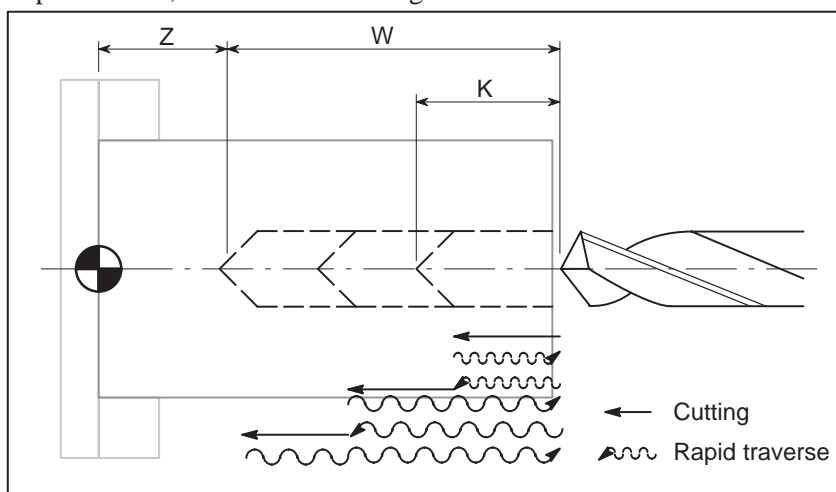
• **Local variable levels**

- 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 (Drill cycle)

Move the tool beforehand along the X- and Z-axes to the position where a drilling cycle starts. Specify Z or W for the depth of a hole, K for the depth of a cut, and F for the cutting feedrate to drill the hole.



• Calling format

```
G65 P9100 {  $\begin{matrix} Zz \\ Ww \end{matrix} \} Kk Ff ;$ 
```

Z : Hole depth (absolute specification)

W : Hole depth (incremental specification)

K : Cutting amount per cycle

F : Cutting feedrate

• Program calling a macro program

```
O0002;
G50 X100.0 Z200.0 ;
G00 X0 Z102.0 S1000 M03 ;
G65 P9100 Z50.0 K20.0 F0.3 ;
G00 X100.0 Z200.0 M05 ;
M30 ;
```

• Macro program (called program)

```

O9100;
#1=0; ..... Clear the data for the depth of the current hole.
#2=0; ..... Clear the data for the depth of the preceding hole.
IF [#23 NE #0] GOTO 1; . If incremental programming, specifies the jump to
N1.
IF [#26 EQ #0] GOTO 8; If neither Z nor W is specified, an error occurs.
#23=#5002-#26; ..... Calculates the depth of a hole.
N1 #1=#1+#6; ..... Calculates the depth of the current hole.
IF [#1 LE #23] GOTO 2; Determines whether the hole to be cut is too
deep.?
#1=#23; ..... Clamps at the depth of the current hole.
N2 G00 W-#2; ..... Moves the tool to the depth of the preceding hole
at the cutting feedrate.
G01 W- [#1-#2] F#9; .. Drills the hole.
G00 W#1; ..... Moves the tool to the drilling start point.
IF [#1 GE #23] GOTO 9; Checks whether drilling is completed.
#2=#1; ..... Stores the depth of the current hole.
GOTO 1;
N9 M99;
N8 #3000=1 (NOT Z OR U COMMAND)

```

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.

```

G66 P p L ℓ <argument-specification>;
P      : Number of the program to call
ℓ      : Repetition count (1 by default)
Argument : Data passed to the macro

```

```

O0001;
:
G66 P9100 L2 A1.0 B2.0;
G00 G90 X100.0;
X125.0;
X150.0;
G67;
:
M30;

```

```

O9100;
:
G00 Z-#1;
G01 Z-#2 F0.3;
:
:
:
M99;

```

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**

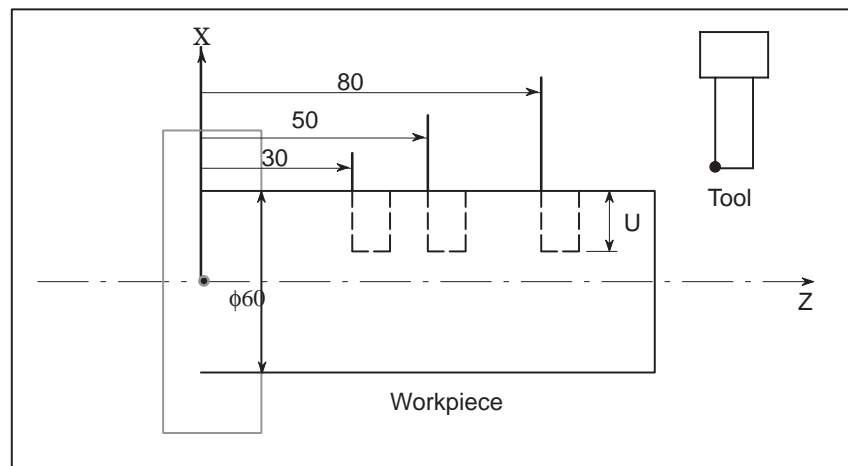
Modal calls can be nested by specifying another G66 code during a modal call.

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

This program makes a groove at a specified position.



- **Calling format**

```
G66 P9110 Uu Ff ;
```

U: Groove depth (incremental specification)

F: Cutting feed of grooving

- **Program that calls a macro program**

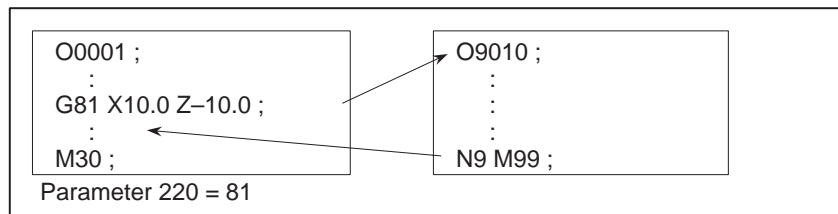
```
O0003 ;
G50 X100.0 Z200.0 ;
S1000 M03 ;
G66 P9110 U5.0 F0.5 ;
G00 X60.0 Z80.0 ;
Z50.0 ;
Z30.0 ;
G67 ;
G00 X00.0 Z200.0 M05 ;
M30;
```

- **Macro program (program called)**

```
O9110 ;
G01 U-#21 F#9 ; ..... Cuts the workpiece.
G00 U#21 ; ..... Retracts the tool.
M99 ;
```

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.

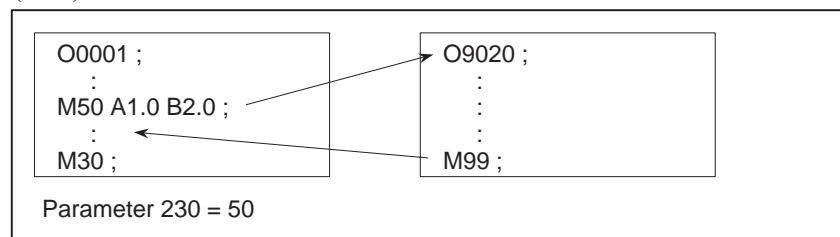
Limitations

- **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.

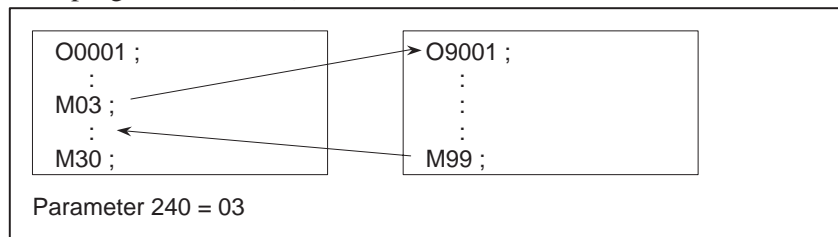
As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

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 custom macro program (9001 to 9003) 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.

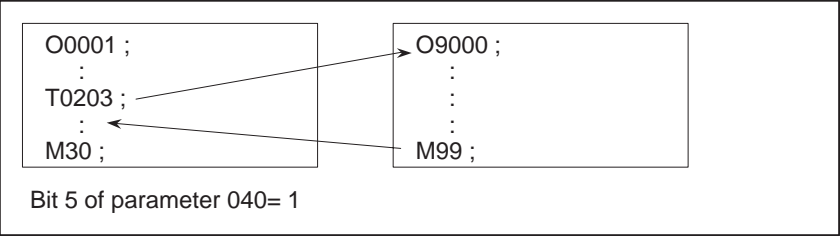
Limitations

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 tool numbers 1 to 5 is measured. The time is not measured for tools whose number is 6 or more.
- 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;
T0100 M06;
M03;
:
M05; ..... Changes #501.
T0200 M06;
M03;
:
M05; ..... Changes #502.
T0300 M06;
M03;
:
M05; ..... Changes #503.
T0400 M06;
M03;
:
M05; ..... Changes #504.
T0500 M06;
M03;
:
M05; ..... Changes #505.
M30;
```

**Macro program
(program called)**

```
O9001(M03) ; ..... Macro to start counting
  IF[FIX[#4120/100] EQ 0]GOTO 9; ..... No tool specified
  IF[FIX[#4120/100] 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[FIX[#4120/100] EQ 0]GOTO 9; ..... No tool specified
  IF[FIX[#4120/100] GT 5]GOTO 9; ..... Out-of-range tool
                                     number
  #[500+FIX[#4120/100]]=#3002+#[500+FIX[#4120/100]];
  ..... Calculates cumulative time.

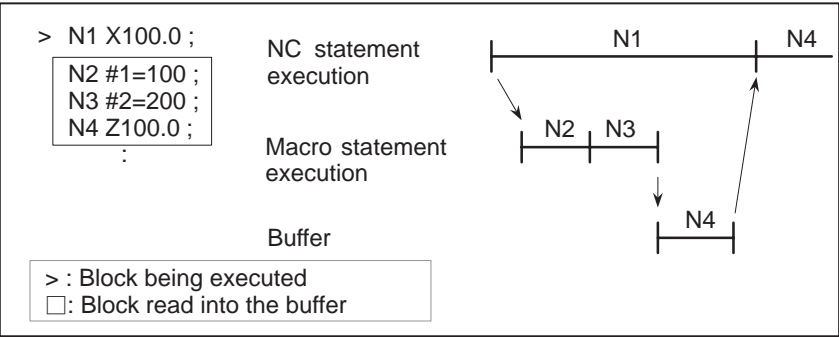
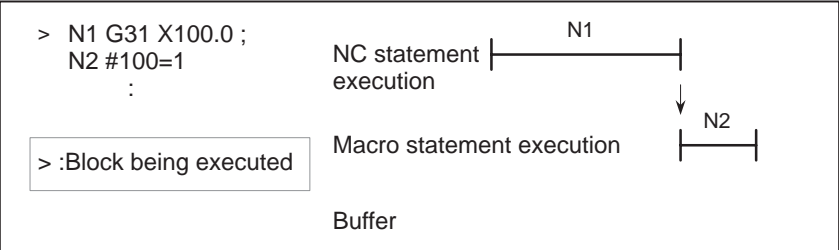
N9 M05; ..... Stops the spindle.
  M99;
```

16.7
PROCESSING
MACRO
STATEMENTS

For smooth machining, the NC prereads the NC statement to be performed next. This operation is referred to as buffering. In tool nose radius compensation mode (G41, G42), the NC prereads 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 preread.

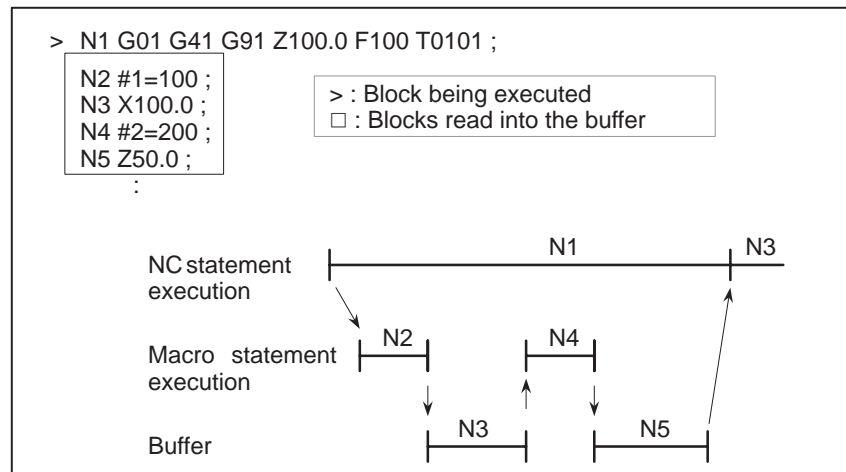
Explanations

- When the next block is not buffered (M codes that are not buffered, G31, etc.)
- Buffering the next block in other than tool nose radius compensation mode (G41, G42) (normally prereading one block)



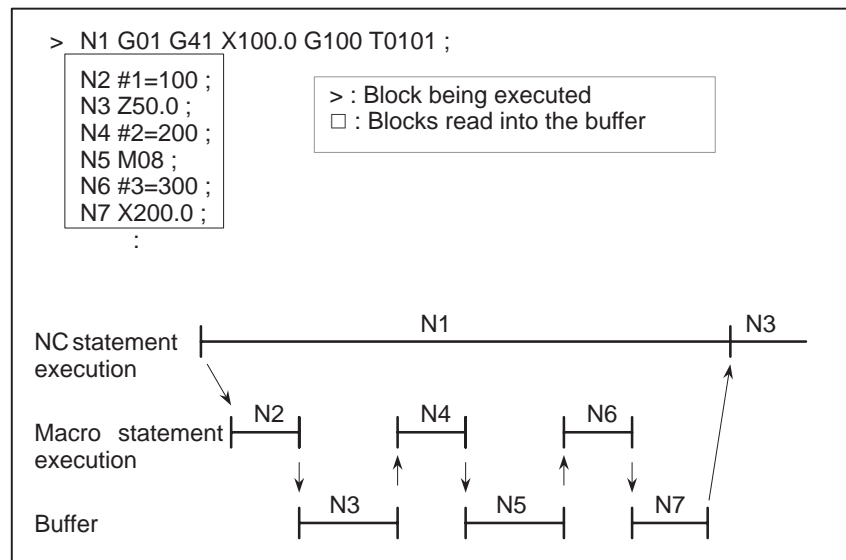
When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- **Buffering the next block in tool nose radius 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 tool nose radius 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**

When it is MDI operation B, the macro call command can be specified in MDI mode too. During automatic operation, however, it is impossible to switch to the MDI mode for a macro program call.
- **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 SBKM (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 tool nose radius compensation mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)
- **Optional block skip**

A / appearing in the middle of an <expression> (enclosed in brackets [] on the right-hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.
- **Operation in EDIT mode**

Registered custom macro programs and subprograms should be protected from being destroyed by accident. By setting (bit 4 of parameter 010) to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 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**

When memory is cleared with a reset operation, local variables and common variables #100 to #149 are cleared to null values. They can be prevented from being cleared by setting, (bits 7 and 6 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 standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- **BPRNT**
- **DPRNT**
- **POPEN**
- **PCLOS**

These commands are provided to output variable values and characters through the reader/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 ISO 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.

Example)

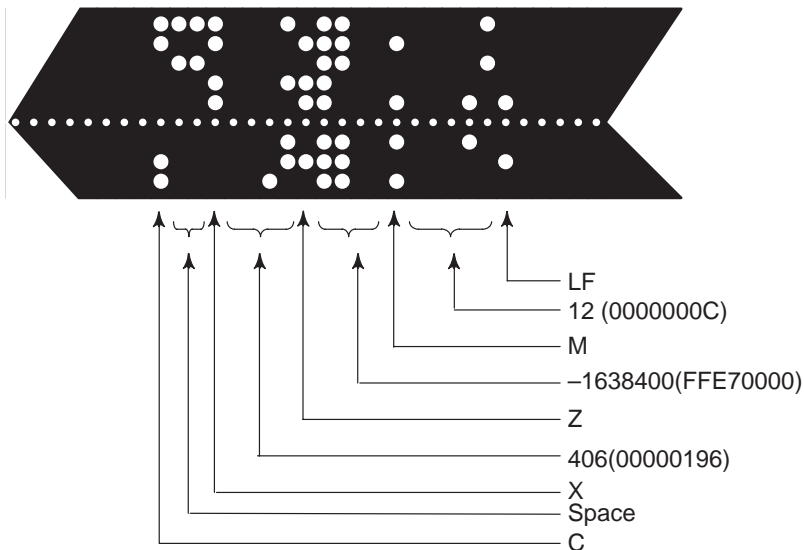
BPRINT [C X#100 [3] Z#101 [3] M#10 [0]]**

Variable value

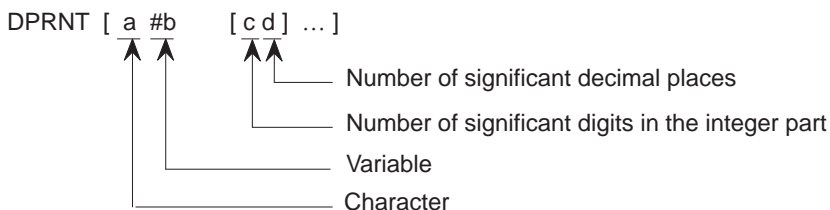
#100=0.40596

#101=-1638.4

#10=12.34



- **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.

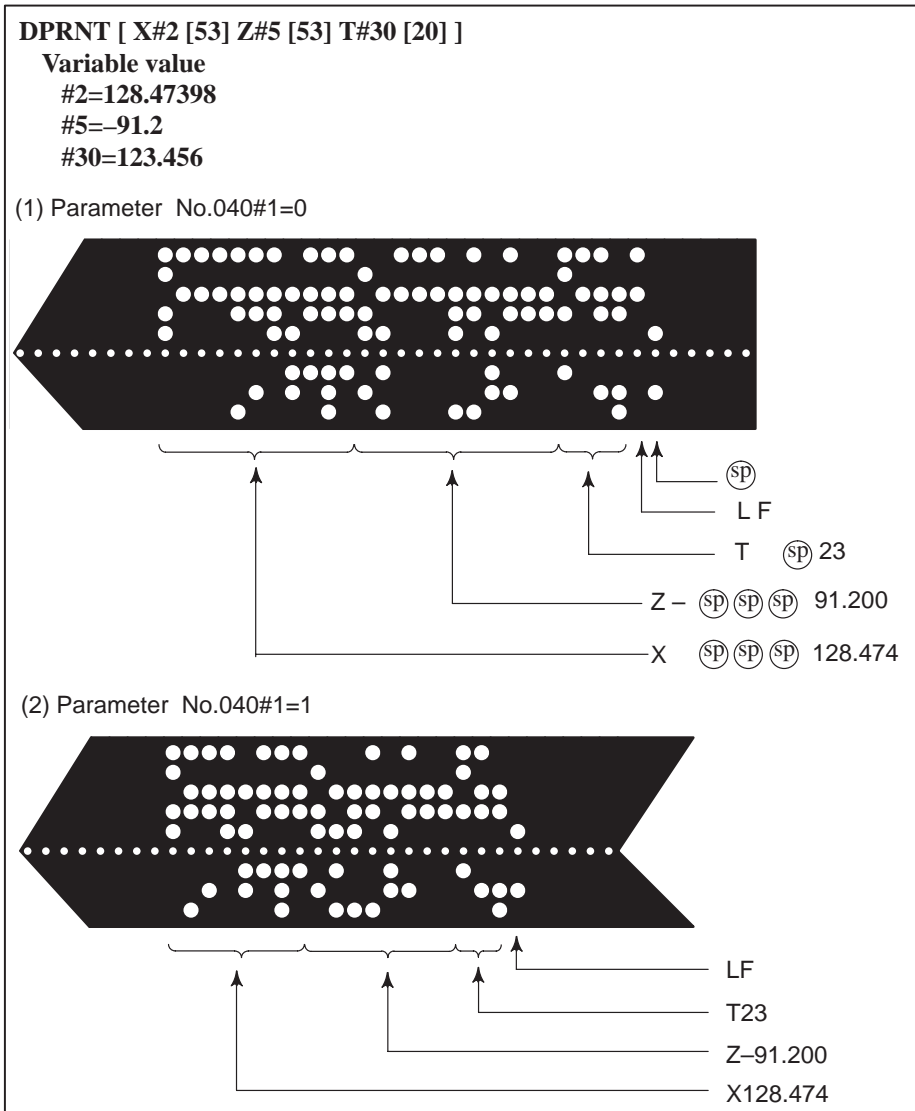
(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.

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.

Example)

- **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 setting I/O. According to the specification of this parameter, set data items (such as the baud rate) for the reader/punch interface.

I/O channel 0 : bit 0, 2, 7 of parameter No.002. Parameters No.552

I/O channel 1 : bit 0, 2, 7 of parameter No.012. Parameters No.553

I/O channel 2 : bit 0, 2, 7 of parameter No.050. Parameters No.250

Specify parameter so 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 .

Notes**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.
- 4 Abbreviated macro words enclosed in brackets [] remains unchanged. However, note that when the characters in brackets are divided and input several times, the second and subsequent abbreviations are converted and input.
- 5 O can be specified in brackets []. Note that when the characters in brackets [] are divided and input several times, O is omitted in the second and subsequent inputs.

16.11

INTERRUPTION TYPE
CUSTOM MACRO

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.

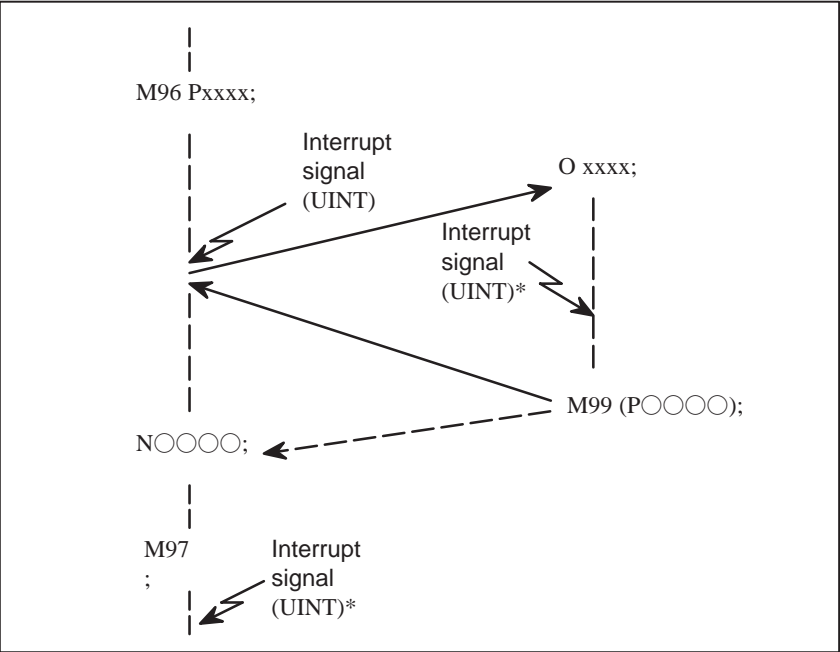


Fig. 16.11 Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx.

When the interrupt signal (UINT, 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

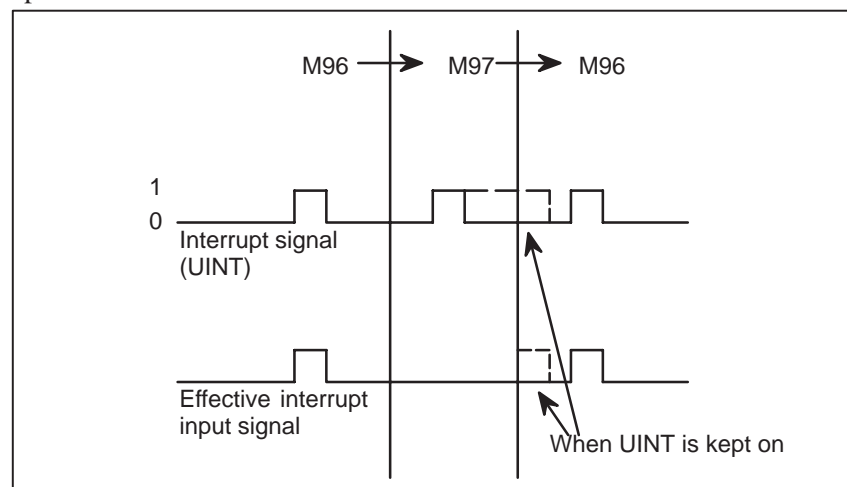
• Interrupt conditions

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

- **When memory operation or MDI operation 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 MSB (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.

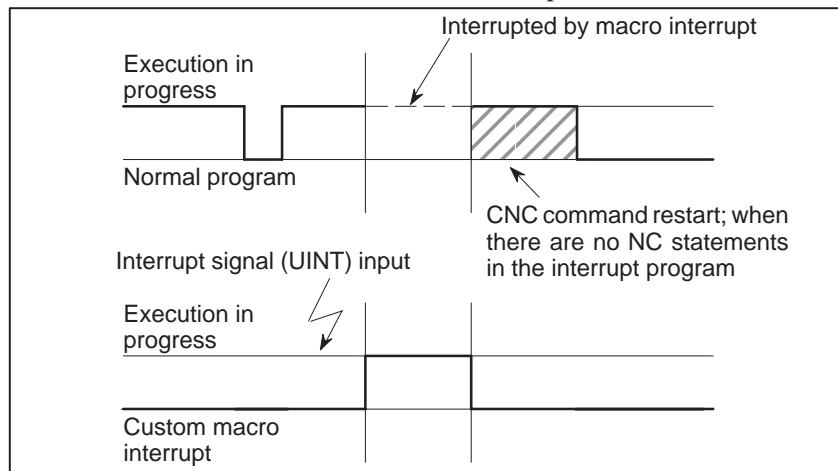
The M codes used for custom macro interrupt control are processed internally (they are not output to external units).

- **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.

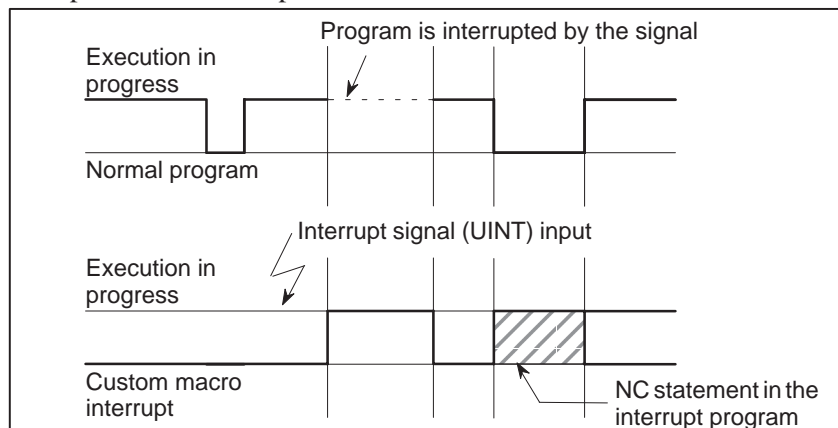
Type I
(when an interrupt is
performed even in the
middle of a block)

- (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.

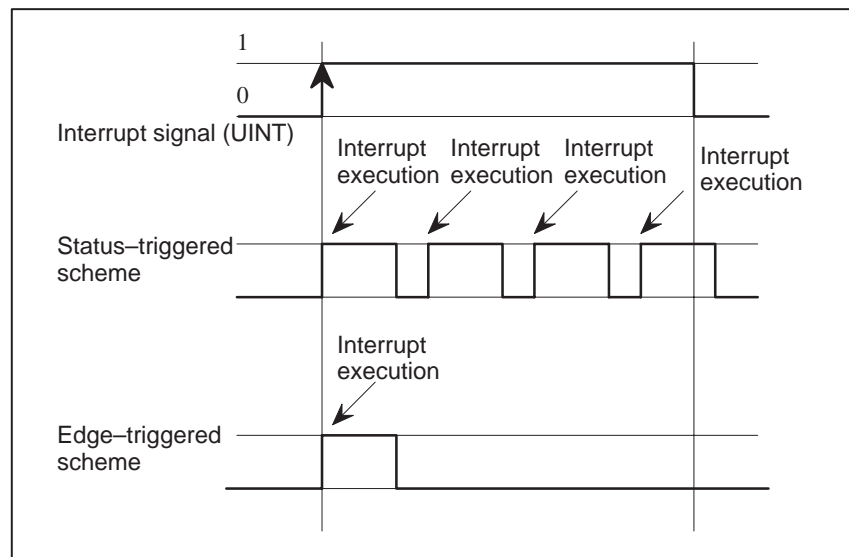
- **Custom macro interrupt signal (UINT)**

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 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.



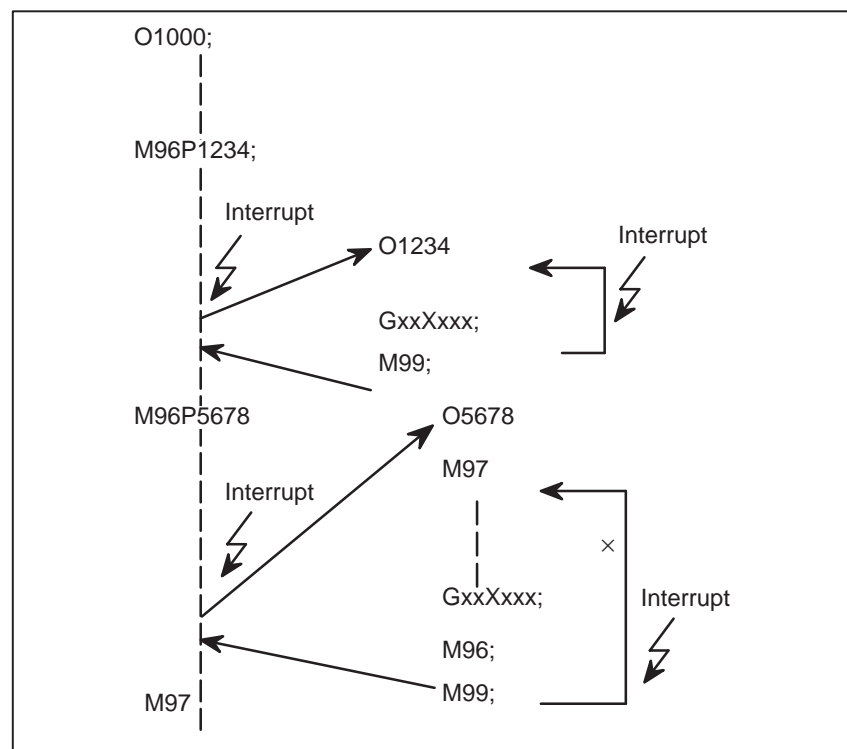
In the above example, an interrupt is executed four times when the status triggered scheme is used; when the edge-triggered scheme is used, the interrupt is executed just once.

- **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. An interrupt is not enabled in Gxx block of O5678 because of M97, and it is 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 ;

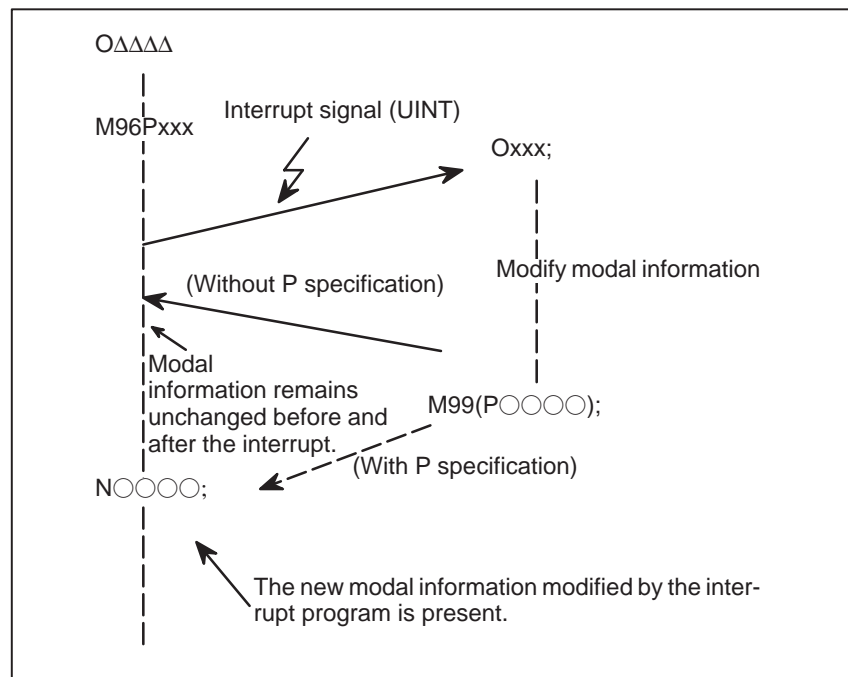
- **Custom macro interrupt and modal information**

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.

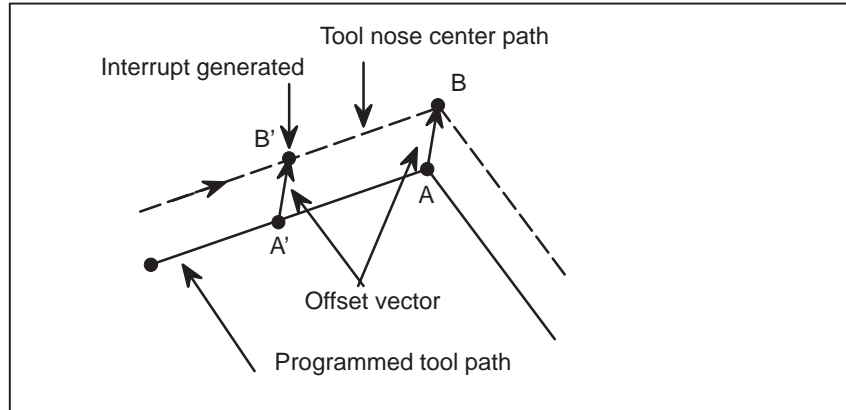
Modal information when control is returned by M99 P○○○○○

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] [] [WKSFT] [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**

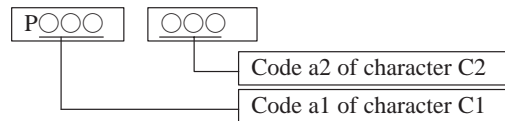
Menu title : $C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 C_9 C_{10} C_{11} C_{12}$
 C_1, C_2, \dots, C_{12} : Characters in the menu title (12 characters)

Macro instruction

$G65 H90 P_p Q_q R_i J_j K_k$:

H90: Specifies the menu title

p : Assume a_1 and a_2 to be the codes of characters C_1 and C_2 . Then,



q : Assume a_3 and a_4 to be the codes of characters C_3 and C_4 . Then,
 $q = a_3 10^3 + a_4$

r : Assume a_5 and a_6 to be the codes of characters C_5 and C_6 . Then,
 $r = a_5 10^3 + a_6$

i : Assume a_7 and a_8 to be the codes of characters C_7 and C_8 . Then,
 $i = a_7 10^3 + a_8$

j : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10} . Then,
 $j = a_9 10^3 + a_{10}$

k : Assume a_{11} and a_{12} to be the codes of characters C_{11} and C_{12} . Then,
 $k = a_{11} 10^3 + a_{12}$

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] [      ] [WKSFT] [ 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)			
U	0.000	W	0.000
H	0.000	V	0.000
NO. 500 =		S	0 T
10:00:48			
[OFFSET] [] [WKSFT] [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

CAUTION

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)			
U	0.000	W	0.000
H	0.000	V	0.000
NO. 500 =		S	0 T
10:00:48			
[OFFSET] [] [WKSFT] [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 program. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.

Format

Format	
G10L50;	Parameter entry mode setting
N_P_;	For parameters entry
.....	
G11;	Parameter entry mode cancel
Meaning of command	
N_:	Parameter No. (4digits)
P_:	Parameter setting value (Leading zeros can be omitted.)

Explanations

- **Parameter setting value (P_)**

Do 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. When not cancelled, the drilling motion will be activated.

NOTE

- 1 Other NC statements cannot be specified while in parameter input mode.
- 2 Parameter input (G10) cannot be specified during axis movement according to PMC axis control.

19

MEMORY OPERATION BY SERIES 10/11 TAPE FORMAT

Programs in the Series 10/11 tape format can be operated by setting parameter TAPEF. Operation are possible for the functions which use the same tape format as that for the Series 10/11 as well as for the following functions which use a different tape format:

- Equal-lead threading
- Subprogram calling
- Canned cycle
- Multiple repetitive canned cycle

NOTE

Registration to memory and memory operation are possible only for the functions available in the Series 0.

19.1 ADDRESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 10/11 TAPE FORMAT

Some addresses which cannot be used for the Series 0 can be used in the Series 10/11 tape format. The specifiable value range for the Series 10/11 tape format is basically the same as that for the Series 0. Sections 19.2 to 19.5 describe the addresses with a different specifiable value range. If a value out of the specifiable value range is specified, an alarm is issued.

19.2 EQUAL-LEAD THREADING

Format

G32IP_F_Q_;

or

G32IP_E_Q_;

IP: Combination of axis addresses

F: Lead along the longitudinal axis

E: Lead along the longitudinal axis

Q: Shift of the threading start angle

Explanations

- **Address**

Although the Series 10/11 allows the operator to specify the number of threads per inch with address E, the Series 10/11 tape format does not. Addresses E and F are used in the same way for specifying the lead along the longitudinal axis. The thread lead specified with address E is therefore also assumed as a continuous-state value for address F.

Specify Q for the shift angle of the threading start angle in steps of 0.001 degrees. A decimal point must not be specified.

- **Specifiable value range for the thread lead**

Address for thread lead		mm input	inch input
E		0.0001 to 500.0000mm	0.000001 to 9.999999inch
F	Command with a decimal point	0.0001 to 500.0000mm	0.000001 to 9.999999inch
	Command without a decimal point	0.01 to 500.00mm	0.0001 to 9.9999inch

- **Specifiable value range for the feedrate**

Address for feedrate			mm input	inch input
F	Feed per minute	Increment system (IS-B)	1 to 100000 mm/min	0.01 to 4000.00 inch/min
		Increment system (IS-C)	1 to 12000 mm/min	0.01 to 480.00 inch/min
	Feed per rotation		0.01 to 500.00 mm/rev	0.0001 to 9.9999 inch/rev

WARNING

Specify the feedrate one more time when switching between feed per minute and feed per rotation.

19.3 SUBPROGRAM CALLING

Format

```
M98P○○○○L○○○;  
P : Subprogram number  
L : Repetition count
```

Explanation

- **Address**
Address L cannot be used in the Series 0 tape format but can be used in the Series 10/11 tape format.
- **Subprogram number**
The specifiable value range is the same as that for the Series 0 (1 to 9999)
- **Repetition count**
The repetition count L can be specified in the range from 1 to 9999. If no repetition count is specified, 1 is assumed.

19.4 CANNED CYCLE

Format

Outer / inner surface turning cycle (straight cutting cycle)
G90X_Z_F_;
Outer / inner surface turning cycle (taper cutting cycle)
G90X_Z_I_F_;
I:Length of the taper section along the X-axis (radius)
Threading cycle (straight threading cycle)
G92X_Z_F_Q_;
F:Thread lead
Q:Shift of the threading start angle (ignored if specified)
Threading cycle (taper threading cycle)
G92X_Z_I_F_;
I:Length of the taper section along the X-axis (radius)
End surface turning cycle (front taper cutting cycle)
G94X_Z_F_;
End surface turning cycle (front taper cutting cycle)
G94X_Z_K_F_;
K:Length of the taper section along the Z-axis

- **Address**

Addresses I and K cannot be used for a canned cycle in the Series 0 tape format but can be used in the Series 10/11 tape format.

- **Specifiable value range for the feedrate**

Same as that for equal-lead threading in section 19.2 See section 19.2.

19.5

MULTIPLE REPETITIVE CANNED TURNING CYCLE

Format

Outer / inner surface turning cycle

G71P_Q_U_W_I_K_D_F_S_T_;

I : Length and direction of cutting allowance for finishing the rough machining cycle along the X-axis (ignored if specified)

K : Length and direction of cutting allowance for finishing the rough machining cycle along the Z-axis (ignored if specified)

D : Depth of cut

End surface rough machining cycle

G72P_Q_U_W_I_K_D_F_S_T_;

I : Length and direction of cutting allowance for finishing the rough machining cycle along the X-axis (ignored if specified)

K : Length and direction of cutting allowance for finishing the rough machining cycle along the Z-axis (ignored if specified)

D : Depth of cut

Closed-loop turning cycle

G73P_Q_U_W_I_K_D_F_S_T_;

I : Length and direction of clearance along the X-axis (radius)

K : Length and direction of clearance along the Z-axis

D : Number of divisions

End surface cutting-off cycle

G74X_Z_I_K_F_D_;

or

G74U_W_I_K_F_D_;

I : Distance to be traveled along the X-axis

K : Depth of cut along the Z-axis

D : Clearance of the tool at the end of the cutting path

Outer / inner surface cutting-off cycle

G75X_Z_I_K_F_D_;

or

G75U_W_I_K_F_D_;

I : Distance to be traveled along the X-axis

K : Depth of cut along the Z-axis

D : Clearance of the tool at the end of the cutting path

Multiple repetitive threading cycle

G76X_Z_I_K_D_F_A_P_Q_;

I : Difference of radiuses at threads

K : Height of thread crest (radius)

D : Depth of the first cut (radius)

A : Angle of the tool tip (angle of ridges)

P : Method of cutting

Q : Shift angle of thread cutting start angle

- **Addresses and specifiable value range**

If the following addresses are specified in the Series 10/11 tape format, they are ignored.

·I and K for the outer/inner surface rough machining cycle (G71)

·I and K for the end surface rough machining cycle (G72)

Address P for specifying the method of cutting for the multiple repetitive threading cycle (G76) is always P1 (constant depth of cut with a single edge), P2 (constant amount of cut, zigzag cutting). Before P2 can be specified, however, the optional function for multiple repetitive turning canned cycle addition must be selected. Address A for specifying the angle of the tool tip can be specified only with 0,29,30,55,60, and 80 degrees. If other values are specified, alarm 062 is issued.

Table 19.5 (a) lists the specifiable value range for address D (depth of cut and clearance).

Table 19.5 (a) Specifiable value range for address D
(depth of cut and clearance)

Increment system	mm input	inch input
IS-B	– 99999.999 to 99999.999mm	– 9999.9999 to 9999.9999inch
IS-C	– 9999.9999 to 9999.9999mm	– 999.99999 to 999.99999inch

The specifiable value range for the feedrate is the same as that for equal-lead threading. See section **19.2**.

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 L000 ;

P10000 is number of the machining cycle to be called first:

P10001 to P10999

L000 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.).

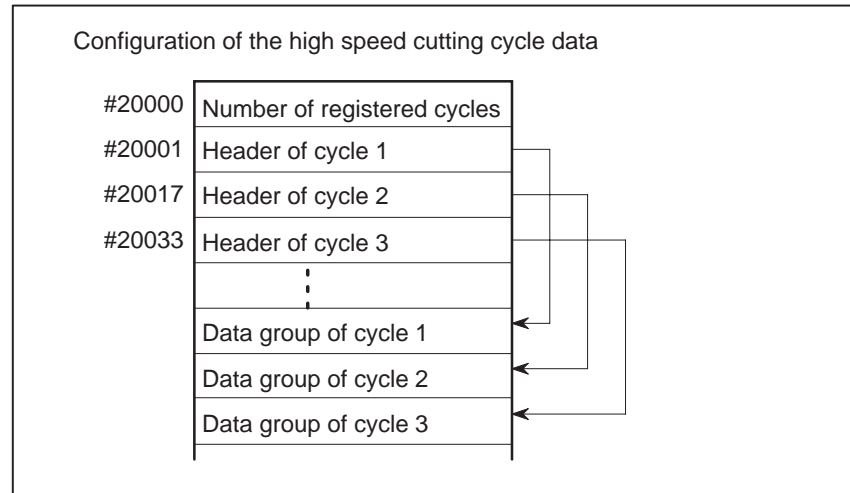
20.2 PULSE DISTRIBUTION

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..	Empty
#20010/20026/20042..	Empty
#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..	Empty
#20016/20032/20048..	Empty

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: 1, t2 and t1: 0)

- **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, expanded RAM is required, 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 Set the total number of distribution data items for one cycle to a multiple of the following values, according to the distribution cycle. 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	The contents of the header are invalid. This alarm is issued in the following cases. <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

POLYGONAL TURNING

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

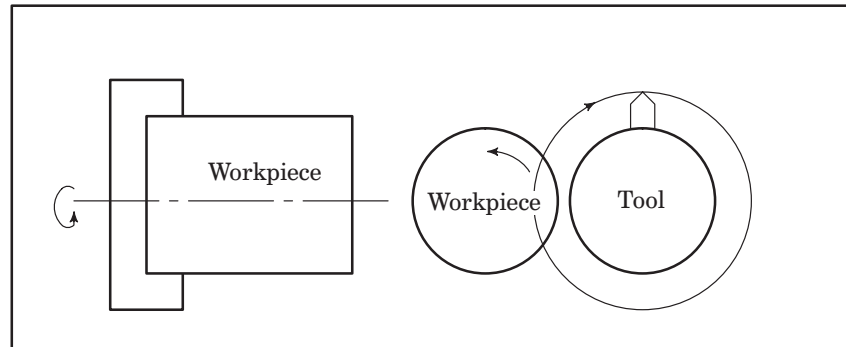


Fig. 21 (a) Polygonal turning

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

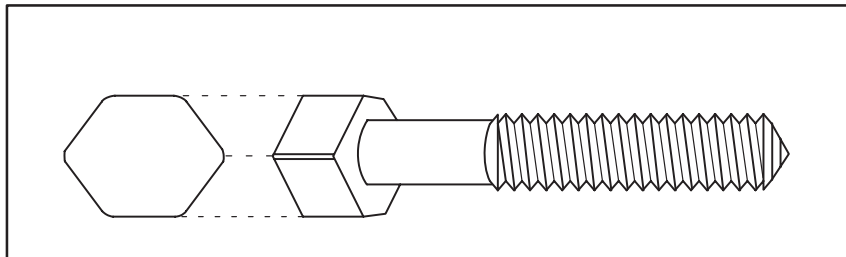


Fig. 21 (b) Hexagon bolt

Format

G251	P_Q_;	
	P,Q:	Rotation ratio of spindle and Y axis Specify range:Interfer 1 to 9 for both P and Q When Q is a positive value, Y axis makes positive rotation. When Q is a negative value, Y axis makes negative rotation.
G250		Cancel of polygonal turning mode.

Explanations

Tool rotation for polygonal turning is controlled by CNC controlled axis. This rotary axis of tool is called Y axis in the following description.

The Y axis is controlled by G251 command, so that the rotation speeds of the workpiece mounted on the spindle (previously specified by S-command) and the tool become the specified ratio.

(Example) Rotation ratio of workpiece (spindle) to Y axis is 1:2, and the Y axis makes positive rotation.

G251P1Q2;

When simultaneous start is specified by G251, the one-rotation signal sent from the position codes set on the spindle is detected. After this detection, the Y axis rotation is controlled according to the rotation ratio (P:Q) while synchronizing with the spindle speed. Namely, the Y axis rotation is controlled so that the spindle and Y axis stand in a relation of P:Q. This relation will be maintained until the polygonal turning cancel command is executed (G250 or reset operation). The direction of Y axis rotation is determined by the code

Q and not affected by the direction of the position coder rotation.

Synchronization of the spindle and Y axis is canceled by the following command:

G250;

When G250 is specified, synchronization of the spindle and Y axis is canceled and the Y axis stops.

This synchronization is also canceled in the following cases:

- i) Power off
- ii) Emergency stop
- iii) Servo alarm
- iv) Reset (external reset signal ERS, reset/rewind signal RRW, and RESET key on the CRT/MDI panel)
- v) Occurrence of P/S alarms 217 to 221

Example

```

G00X100.0Z20.0S1000.0M03 ; Workpiece rotation speed 1000rpm
G251P1 Q2 ; ..... Tool rotation start (tool rotation speed
                                     2000rpm)
G01X80.0 F10.0 ; ..... X axis infeed
G04X2. ;
G00X100.0 ; ..... X axis escape
G250 ; ..... Tool rotation stop
M05 ; ..... Spindle stop
Specify G250 and G251 always in a single block.

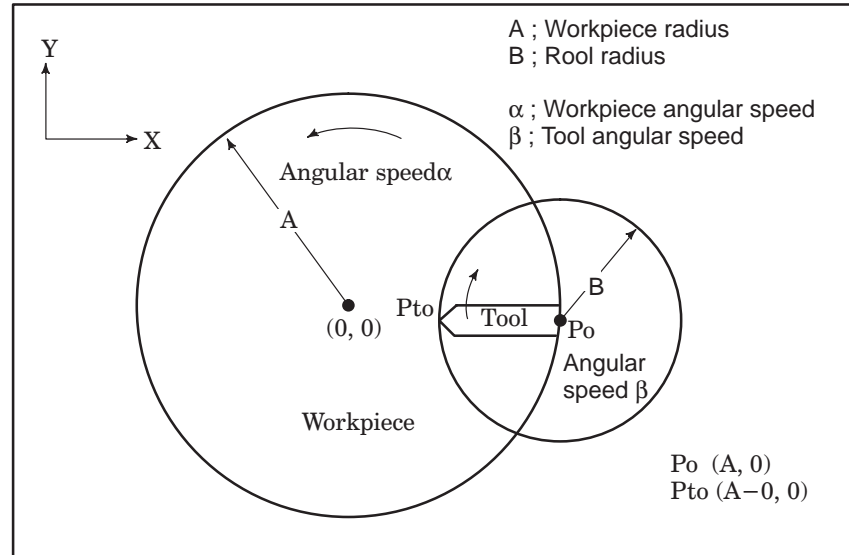
```

- **Principle of PolygonalTurning**

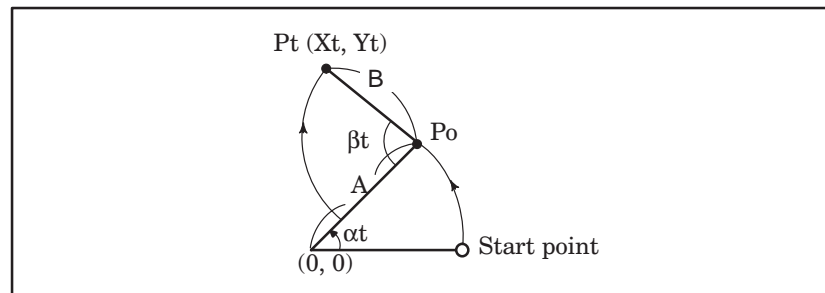
The principle of polygonal turning is explained below. In the figure below the radius of tool and workpiece are A and B , and the angular speeds of tool and workpiece are α and β . The origin of XY cartesian coordinates is assumed to be the center of the workpiece.

Simplifying the explanation, consider that the tool center exists at the position

$P_o (A,0)$ on the workpiece periphery, and the tool nose starts from position $P_{to}(A-B, 0)$.



In this case, the tool nose position $P_t (X_t, Y_t)$ after time t is expressed by equation 1:



$$X_t = A \cos \alpha t - B \cos(\beta - \alpha) t$$

(Equation 1)

$$Y_t = A \sin \alpha t + B \sin(\beta - \alpha) t$$

Assuming that the rotation ration of workpiece to tool is 1:2, namely, $\beta=2\alpha$, equation 1 is modified as follows

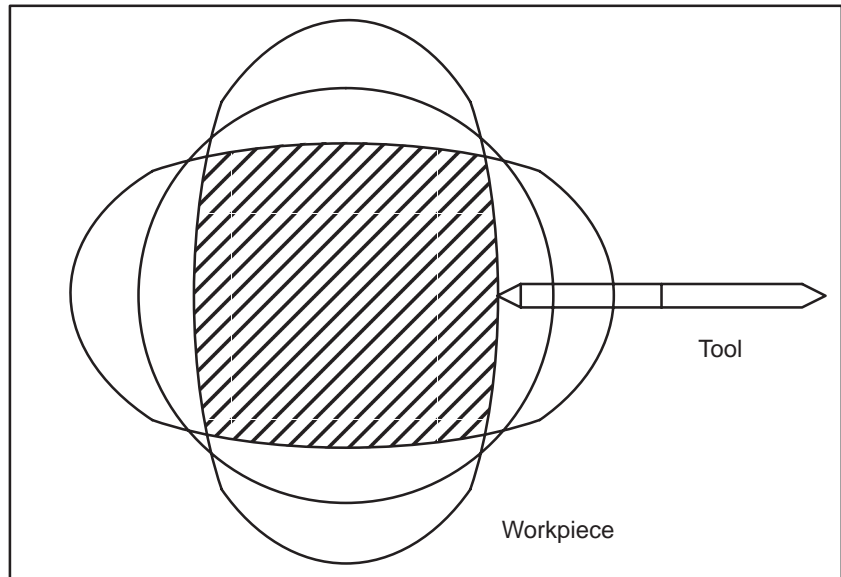
$$X_t = A \cos at - B \cos at = (A - B) \cos at$$

(Equation 2)

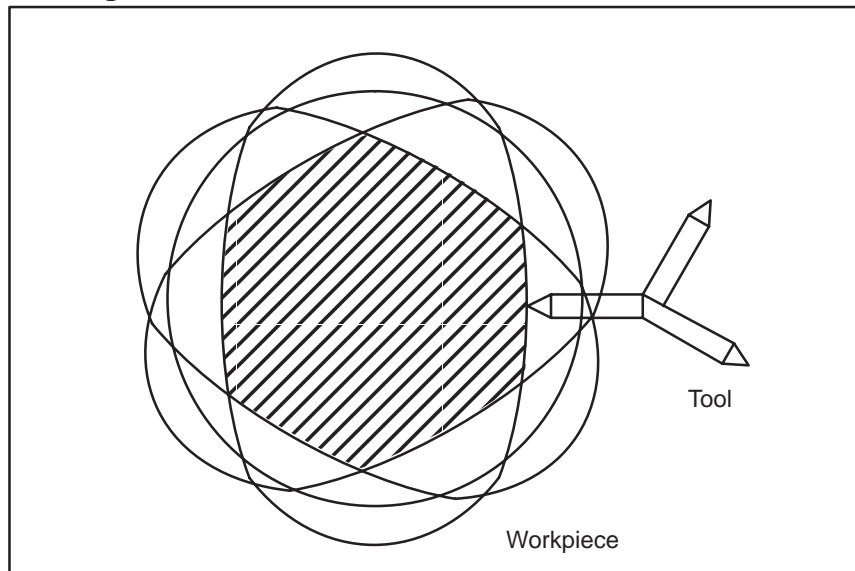
$$Y_t = A \sin at + B \sin at = (A + B) \sin at$$

Equation 2 indicates that the tool nose path draws an ellipse with longer diameter $A+B$ and shorter diameter $A-B$.

Then consider the case when one tool is set at 180° symmetrical positions, for a total of two. It is seen that a square can be machined with these tools as shown below.



If three tools are set at every 120° the machining figure will be a hexagon as shown below.



WARNING

- 1 For the maximum rotation speed of tool, refer to the manual published by the MTB. Do not specify a spindle speed or ratio to spindle speed exceeding the maximum rotation speed of the tool.
- 2 An absolute position detector cannot be set on the Y axis.
- 3 The following signals become either valid or invalid in relation to the Y axis in polygonal turning.
Valid signals in relation to Y axis:
 machine lock
 servo off
Invalid signals in relation to Y axis:
 feed hold
 interlock
 override
 dry run
(During a dry run, however, there is no wait for a revolution signal in the G251 block.)
- 4 Jogging or handle feeding is invalid when the Y axis is in polygonal turning.
- 5 The starting point of the threading process becomes inconsistent when performed during polygonal turning.
Cancel the polygonal turning by executing G250 when threading.
- 6 The Y axis in polygonal turning is not included in the number of axis controlled simultaneously.

CAUTION

Unlike the other controlled axes, the least command increment of the Y axis is not 0.001 degree since an axis move command is unnecessary for the Y axis. The least command increment of the Y axis is related to parameters such as feedrate.

Therefore, pay attention to these parameters upon machine adjustment.

L: Move distance (deg) per motor rotation

Q: Number of pulses per pulse coder rotation

CMR: Command multiply

DMR: Detection multiply

$$\text{Least command increment} = \frac{L \times \text{CMR}}{Q \times \text{DMR}}$$

$$\begin{aligned} \text{Detection unit} &= \frac{\text{Least command increment}}{\text{CMR}} \\ &= \frac{L}{Q \times \text{DMR}} \end{aligned}$$

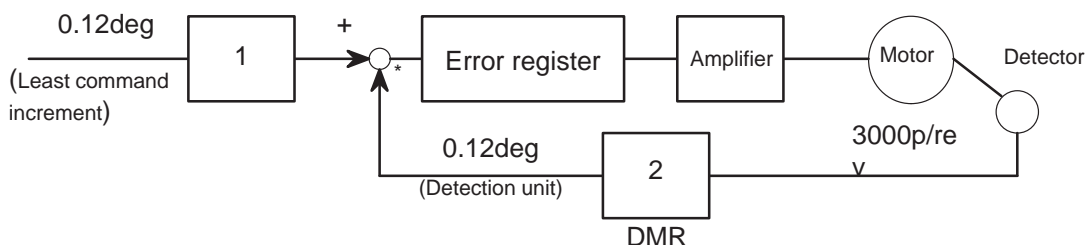
$$\text{Move distance per Y axis rotation} = \frac{360}{\text{Least command increment}} \quad (\text{Parameter No.778})$$

For example, in case of L=720 deg, Q=3000 pulses, CMR=2, the move distance is as follows;

Least command increment :0.12deg

Detection unit:0.12deg

Move distance per Y axis rotation:3000



When the least command increment of the Y axis is 0.12 deg, the parameters concerning the feedrate of the Y axis should be set in unit of 120 deg/min (Thousandfold of least command increment), not in 1 deg/min.

NOTE

- 1 The Y axis, unlike the other controlled axes, cannot be specified a move command as Y—. That is, an axis move command is unnecessary for the Y axis. Because, when G251 (polygonal turning mode) is specified, it is only necessary to control the Y axis so that the tool rotates at a certain ratio to the spindle rotation speed. However, only the reference point return command (G28V0;) can be specified since the Y axis rotation is stopped at the unstable position when G250 (polygonal turning mode cancel command) is specified. If the tool rotation start position is unstable, a problem may occur, for example, when the same figure is machined with a finishing tool after once machined with a roughing tool. Specification of G28V0; for Y axis is equal to the orientation command for the spindle. In the other axes, unlike the manual reference point return, G28 usually makes reference point return without detecting the deceleration limit. However, with G28V0; for the Y axis, reference point return is executed by detecting the deceleration limit, like manual reference point return.
To machine a workpiece into the same figure as the previous one, the tool and the spindle must be in the same position as the previous time when the tool starts rotating. The tool is set start rotation when the one-rotation signal of the position coder set on the spindle is detected.
- 2 The Y axis used to control tool rotation for polygonal turning uses the 4th axis. However, by setting parameters, (No.069#1) the 3rd axis may also be used. In this case, that axis must be named C axis.
- 3 Among the position display of the Y axis, the display for the machine coordinate value (MECHINE) will change from a range of 0 to the parameter setting (the amount of movement per revolution) as the Y axis moves. Absolute or relative coordinate values are not renewed.

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 388 to 1.

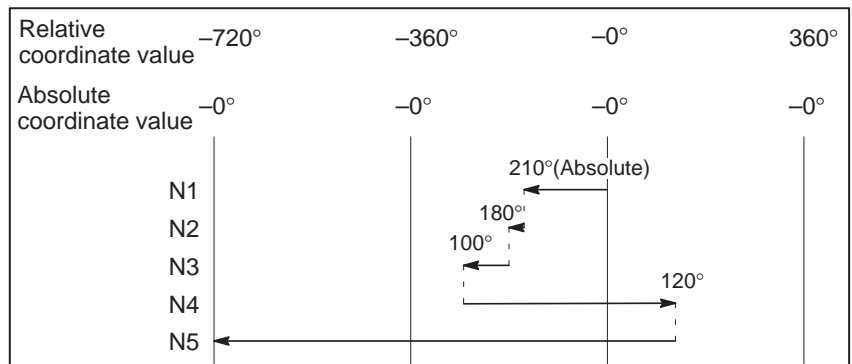
Explanations

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.788, 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.388 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.388 is set to 1.

Examples

Assume that axis C is the rotating axis and that the amount of movement per rotation is 360.000 (parameter No.788 = 360000). When the following program is executed using the roll-over function of the rotating axis, the axis moves as shown below.

C0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 C-150.0 ;	N1	-150	210
N2 C540.0 ;	N2	-30	180
N3 C-620.0 ;	N3	-80	100
N4 H380.0 ;	N4	+380	120
N5 H-840.0 ;	N5	-840	0

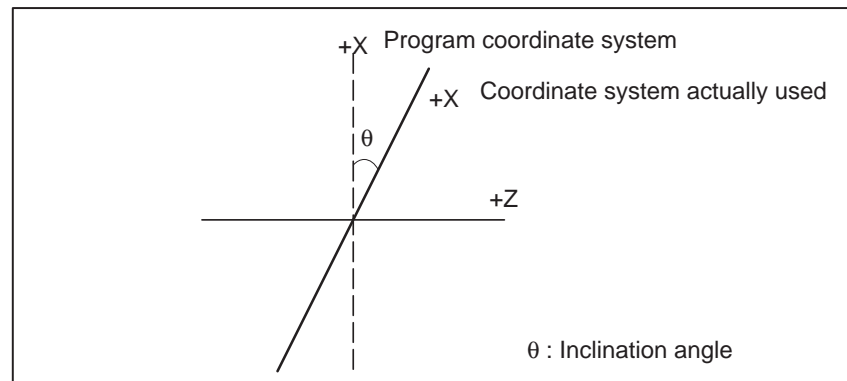


23

ANGULAR AXIS CONTROL
(0–GCC, 00–GCC, 0–GCD/II)

General

When the X-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 X-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 to be traveled along the X-axis is determined by the following formula when it is specified with a diameter :

$$X = \frac{Xp}{2 \cos \theta}$$

The distance traveled along the Z-axis is corrected by the inclination of the X-axis, and is determined by the following formula:

$$Za = Zp - \frac{1}{2} Xp \tan \theta$$

The feedrate is determined as described below. The speed component along the X-axis is determined by the following formula:

$$Fa = \frac{Fp}{\cos \theta}$$

Xa, Za, Fa:Actual distance and speed

Xp, Zp, Fp:Programmed distance and speed

- Method of use

Parameter (No.036#0) 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.755).

Parameter (No.036#2) enables X-axis manual reference point return only with a distance along the X-axis.

- **Absolute and relative position display**
- **Machine position display**

An absolute and a relative position are indicated in the programmed Cartesian coordinate system. Machine position display

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle.

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 X-axis manual reference point return operation, be sure to perform reference point return operation starting with the X-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 $\pm 60^\circ$ should be used.
- 4 Before a Z-axis reference point return check (G37) can be made, X-axis reference point return operation must be completed.

24

2 SYSTEMS CONTROL FUNCTION (0–TTC)



24.1 GENERAL

- **Application to lathes with one spindle and two tool posts**

Series 0-TTC is CNC system that can control two systems ; these series are designed for those lathes that operate two tool posts independently of each other to enable simultaneous cutting processing with the two tool posts.

Series 0-TTC can be used for a lathe that machines one workpiece attached to one spindle with two tool posts simultaneously.

For example, while one tool post is performing outer surface machining, the other tool post can perform inner surface machining, thus reducing machining time dramatically.

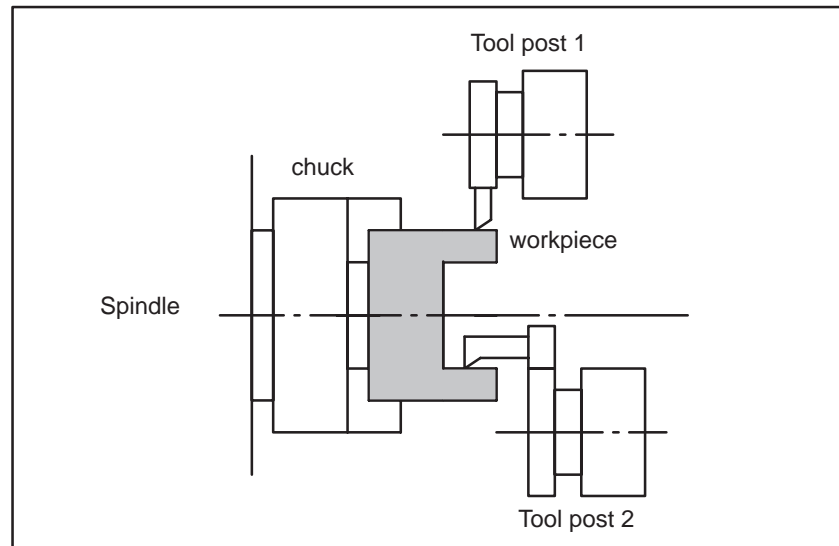


Fig. 24.1 (a) Application to lathes with one spindle and tow tool posts

- **Application to lathes with two spindles and two tool posts**

Series 0-TTC can be used for a lathe that machines a workpiece attached to each of two spindles with two tool posts simultaneously. In this case, each tool post operates independently of each other as if two lathes were used, thus improving productivity.

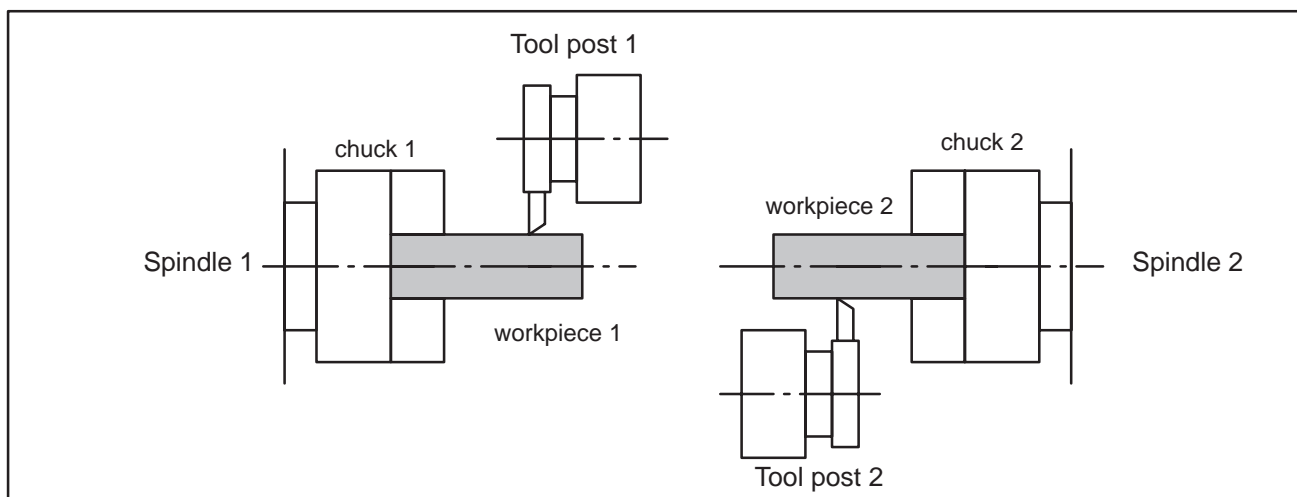


Fig. 24.1 (b) Application to lathes with two spindles and two tool posts

- **Controlling two tool posts independently at the same time**

The operations of two tool posts are programmed independently of each other, and each program is stored in program memory for each tool post. When automatic operation is to be performed, each tool post is activated after selecting a program for machining with tool post 1 and a program for machining with tool post 2 from the programs stored in program memory for each tool post. Then the programs selected for the tool posts are executed independently at the same time. When tool post 1 and tool post 2 need to wait for each other during machining, the waiting function is available (Section 23.2)

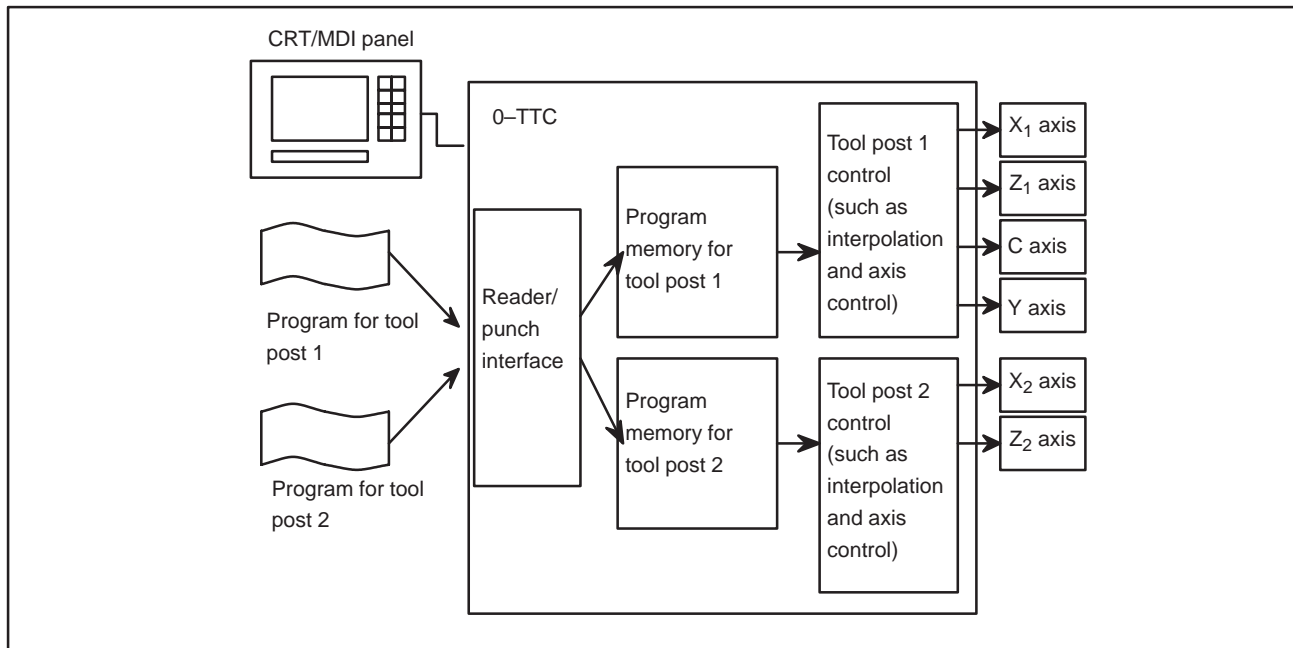


Fig. 24.1 (c) Controlling two tool posts independently at the same time

Just one CRT/MDI panel is provided for the two tool posts. Before operation and display on the CRT/MDI panel, the tool post selection signal is used to switch between the two tool posts.

24.2

WAITING FOR TOOL POSTS

Explanations

Control based on M codes is used to cause one tool post to wait for the other during machining. By specifying an M code in a machining program for each tool post, the two tool posts can wait for each other at a specified block. When an M code for waiting is specified in a block for one tool post during automatic operation, the other tool post waits for the same M code to be specified before starting the execution of the next block. This function is called the tool post waiting function.

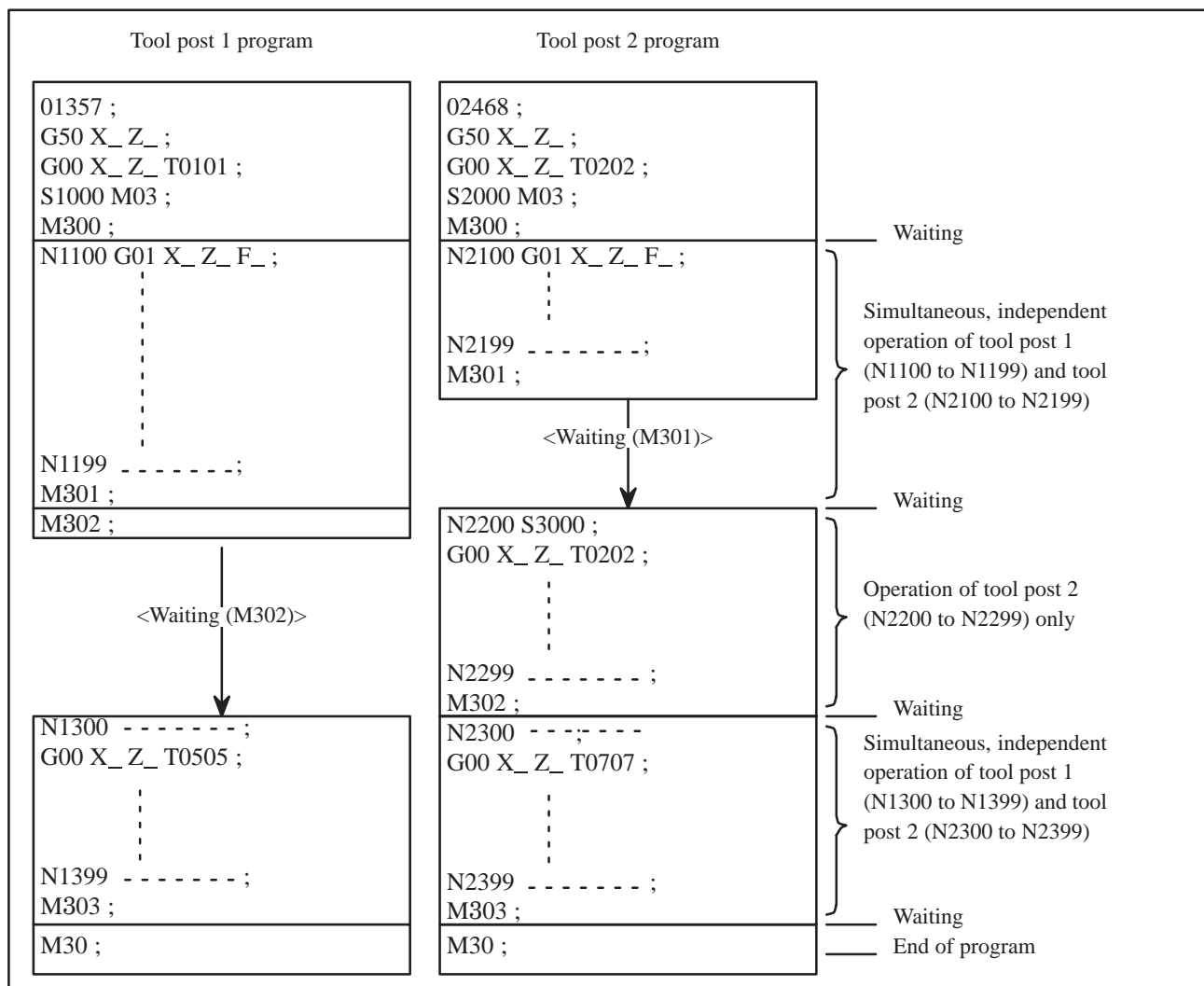
A range of M codes used as M codes for waiting is to be set in the parameters (No.047#4 and 243) before hand.

Example

M300 to M399 are used as M codes for waiting.

Parameter setting: No.047#4=0

No.243=3



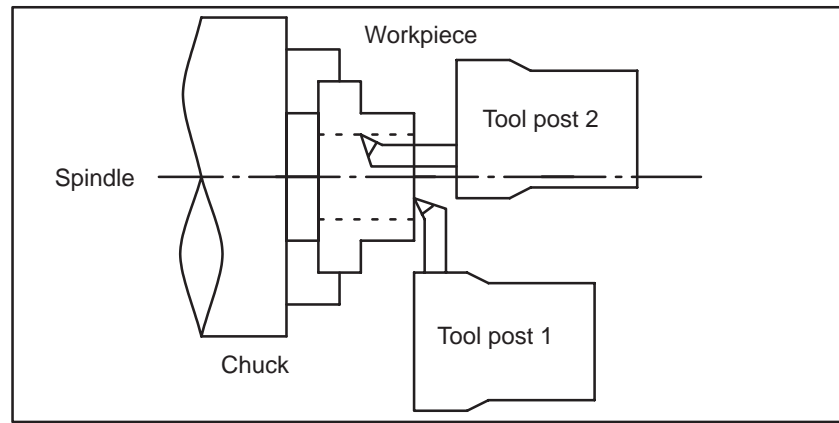
NOTE

- 1 An M code for waiting must always be specified in a single block.
- 2 If one tool post is waiting because of an M code for waiting specified, and a different M code for waiting is specified with the other tool post, an alarm (No. 160) is raised. In this case, both tool posts stop operation.
- 3 PMC–CNC interface
 - M codes for waiting, unlike other M codes, do not have a code signal and strobe signal output.
 - Waiting ignore signal NOWT#1 (G133#1), NOWT#2 (G1333#1) By using the waiting ignore signal, an M code for waiting specified in a machining program can be ignored. The waiting ignore signal is used to operate only one tool post.
 - Waiting–in–progress signals WATO#1 (F160#6), WATO#2 (F1360#6) While waiting is in progress (from specification of an M code for waiting with one tool post until specification of the same M code for waiting with the other tool post), the waiting tool post outputs the waiting–in–progress signal.
 - Refer to the "FANUC Series 0/00/0–Mate CONNECTION MANUAL (FUNCTION) (B–61393E–2)" for detailed information about each signal.

24.3 TOOL POST INTERFACE CHECK

24.3.1 General

When two tool posts machine the same workpiece simultaneously, the tool posts can approach each other very closely. If the two tool posts interfere with each other due to a program error or any other setting error, a serious damage such as a tool or machine destruction can occur. The function "tool post interference check" is available which can decelerate and stop the two tool posts before the tool posts interfere with each other due to an incorrect command.



The contours of the two tool posts are checked to determine whether or not an interference occurs.

24.3.2 Data Setting for the Tool Post Interference Check Function

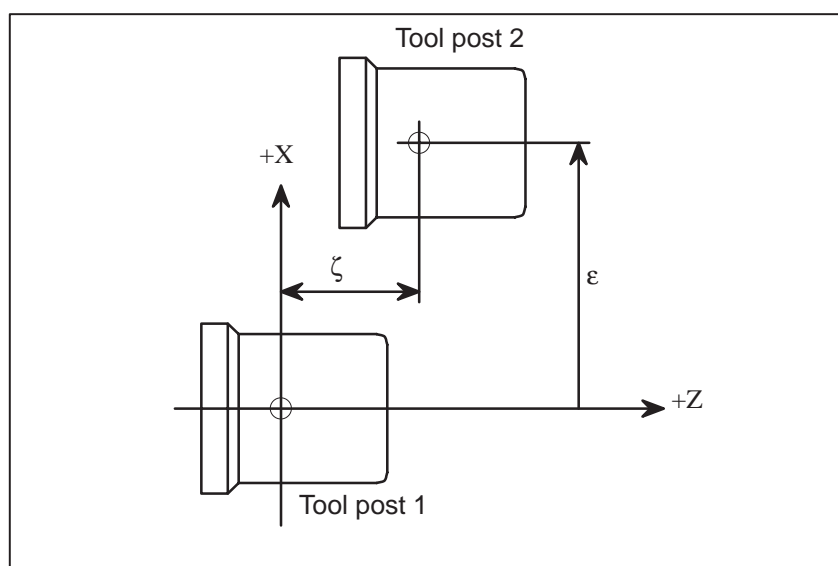
To make a tool post interference check, data including the relationships between the two tool posts and interference forbidden areas (that is, tool shapes) needs to be set. The method of such data setting is described below.

With the tool post interference check function, whether or not the two tool posts interfere with each other is determined by checking if the interference forbidden areas (based on the interference forbidden areas of the currently selected tools) of the tool posts overlap each other after the movement of the tool posts.

Explanations

- **Position setting for reference points of two tool posts**

When reference point return operation is completed with all axes (X1,Z1, X2, Z2), the reference point of tool post 1 is set at the origin of the ZX plane coordinate system. At this time, the position of the reference point of tool post 2 is set in a parameter. The next item describes the reference points.



In the ZX plane coordinate system at the origin of which the reference point of tool post 1 is set, set the X coordinate (ϵ) of the reference point of tool post 2 in parameter No.768, and its Z coordinate (ζ) in parameter No.769.

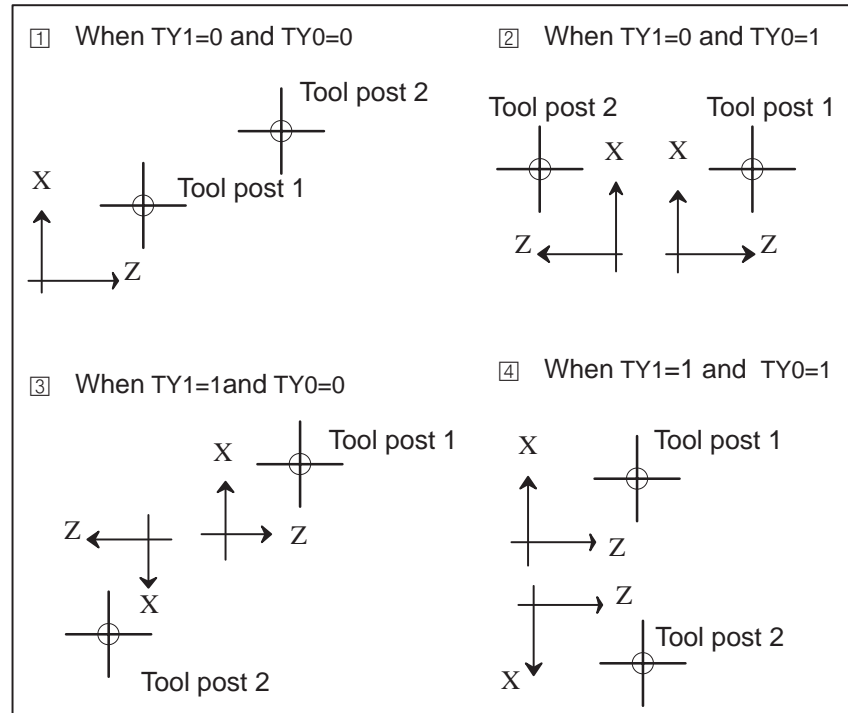
The unit of setting is the least command increment. For an axis subject to diameter specification, a diameter value is to be specified.

Measure (ϵ) and (ζ) when reference point return operation is completed with the four axes (X1, Z1, X2, Z2). When the relative coordinate parameters (Nos.768 and 769) of the two tool posts are to be updated, reference point return operation must always be completed with the four axes beforehand. Otherwise, the internally memorized relational positions of the tool posts are not updated to new parameter values.

- **Set the relationship between the coordinate systems of the two tool posts**

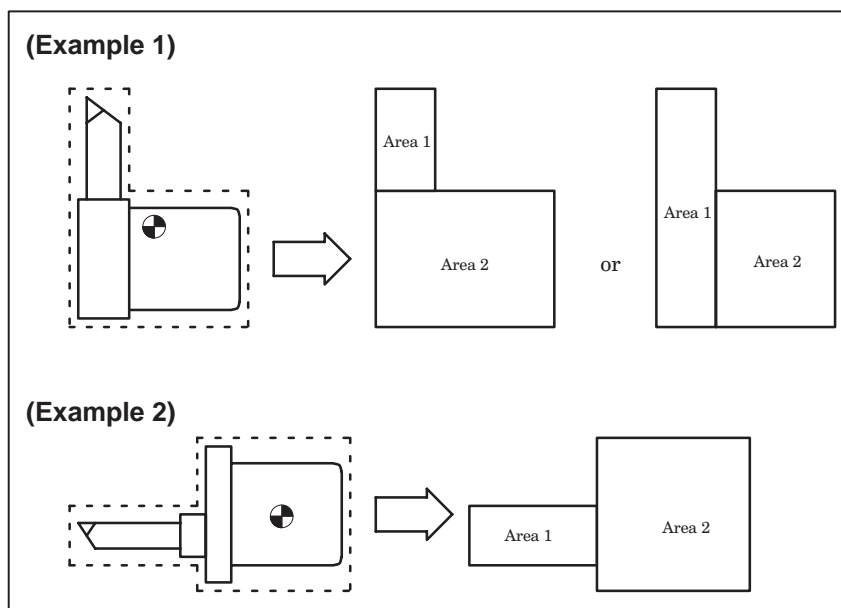
	#7	#6	#5	#4	#3	#2	#1	#0
048							TY1	TY0

TY0, TY1: Set the relationship between the coordinate systems of the two tool posts, with tool post 1 used as the reference.

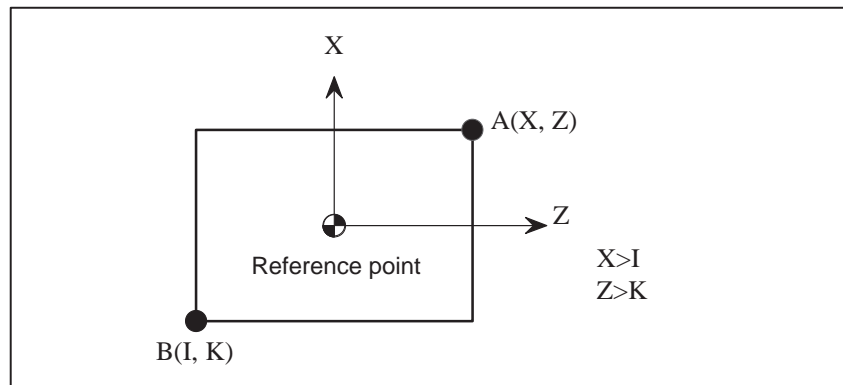


- **Setting of interference forbidden area**

An interference forbidden area is set using a combination of two rectangular areas. Some examples are shown below. The dashed lines indicate interference forbidden areas.



The coordinates of the upper and lower ends (points A and B shown below) of each of two rectangles are set, with the reference point of the tool post set as the origin.






See Section 23.2.3 for information about the coordinate setting procedure.

24.3.3 Setting and Display of Interference Forbidden Areas for Tool Post Interference Checking

Explanations

Display and set tool shape data (interference forbidden areas) according to the procedure below.

- ① Press the  function key.
- ② Press the continuation menu key () , then press the [TOOLFM] soft key.
- ③ With the tool post selection signal, select a tool post for which interference forbidden areas for tool post interference checking are to be displayed and set.
- ④ Display the screen including a tool number for which data is to be set.
Method 1: Select the screen by using the page keys and cursor keys.
Method 2: Press the  key, then enter the desired tool number.

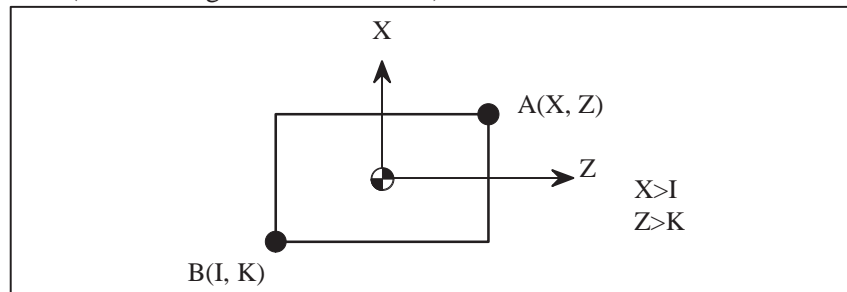
Then, press the  key.


```

TOOL FORM DATA      HEAD1 :O0000 N0000
OFFSET NO. = 01
  AREA1
    X = 20.000
    Z = 70.000
    I = -10.000
    K = -50.000
  AREA2
    X = 40.000
    Z = 70.000
    I = 20.000
    K = 30.000
OFFSET NO. = 02
  AREA1
    X = 160.000
    Z = 170.000
    I = -200.000
    K = -120.000
  AREA2
    X = -200.000
    Z = -60.000
    I = -280.000
    K = -120.000
ADRS.                S 0 T
                    MDI
[ WEAR ][ GEOM ][W.SHFT][MACRO ][      ]

```

- ⑤ With the numeric keys, enter the coordinates of point A or B.
(Fraction digits can be entered.)



- ⑥ By pressing the soft key INPUT, the entered coordinates are set.

NOTE

Tool number

The tool geometry data must be set for each tool number. The tool number here refers to the offset number. When both tool geometry offset and tool wear offset are used, the tool number corresponds to the wear offset number. To use two or more offset numbers for the same tool, the same data for the tool must be set two or more times in the tool geometry data.

24.3.4

Conditions for Making a Tool Post Interference Check

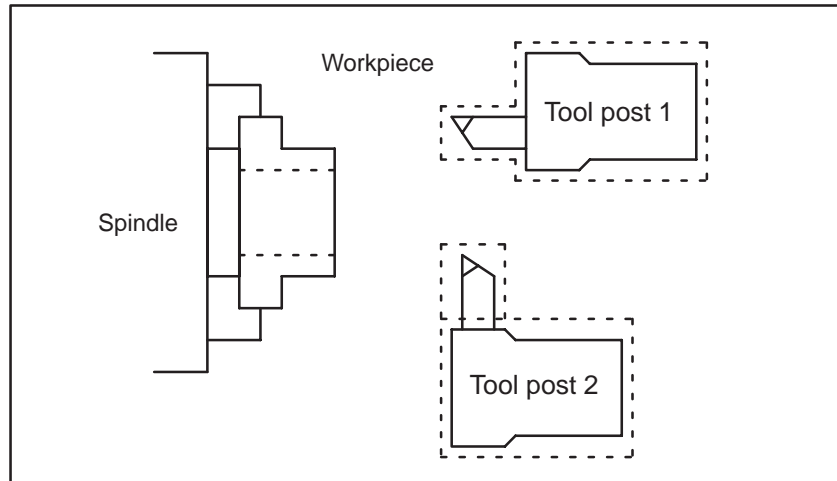
A tool post interference check is made when all conditions listed below are satisfied.

- (1) Parameter (No.048#4) for enabling the tool post interference check function is set to 0.
- (2) After power is turned on, reference point return operation is completed with all axes (X1,Z1, X2, Z2).(When an absolute-position detector is used, the matching between a machine position and absolute-position detector position must be completed.)
- (3) Offset numbers other than 0 are specified using T codes for two tool posts.
- (4) When manual mode is used,parameter No.048#3 for enabling the tool post interference check function in manual mode is set to 1.
When all conditions for making a tool post interference check are satisfied, the tool post interference check in progress signal is output to the PMC.

24.3.5 Execution of Tool Post Interference Checking

when all conditions described in Section 24.3.4 are satisfied, a tool post interference check is started. When a tool post interference check is made, an interference forbidden area is set for the two tool posts by using the offset shape data corresponding to the currently selected offset numbers. Then whether the areas interfere with each other is checked.

Explanations



When interface forbidden areas (tool shapes) as indicated by dashed lines are set for tool posts 1 and 2 as shown above, a check is made by determining whether the two interference forbidden areas indicated by dashed lines overlap each other after the movement of the tool posts.

If the two areas interfere with each other an alarm (No. 508 or No. 509) is raised; the two tool posts are decelerated and stopped.

If an interference alarm is raised, a tool post interference alarm signal is output to the PMC.

If an interference alarm is raised by the interference of the two tool posts during automatic operation, switch to manual mode to move the tool posts out of the interference state. Then release the alarm by a reset.

The interference check function can be enabled even in manual mode by setting the parameter (No. 048#3) to 1. This allows the tool posts interfering with each other to be moved along the axes only in such directions that clear the interference. With this capability, the two tool posts interfering with each other in automatic operation cannot be manually moved by mistake further into the interference forbidden areas after the mode is switched to manual mode to clear the interference, thus providing safety.

WARNING

When an alarm is raised, the CNC system and machine system stop with some delay in time. So an actual stop position can be closer to the other tool post beyond an interference forbidden position specified using tool shape data. So, for safety, tool shape data a little larger than the actual shape should be set. The extra distance, L, required for this purpose is calculated from a rapid traverse feedrate as follows

$$L = (\text{Rapid traverse feedrate}) \times \frac{1}{7500}$$

For example, when a rapid traverse feedrate of 15 m/min is used, L=2mm.

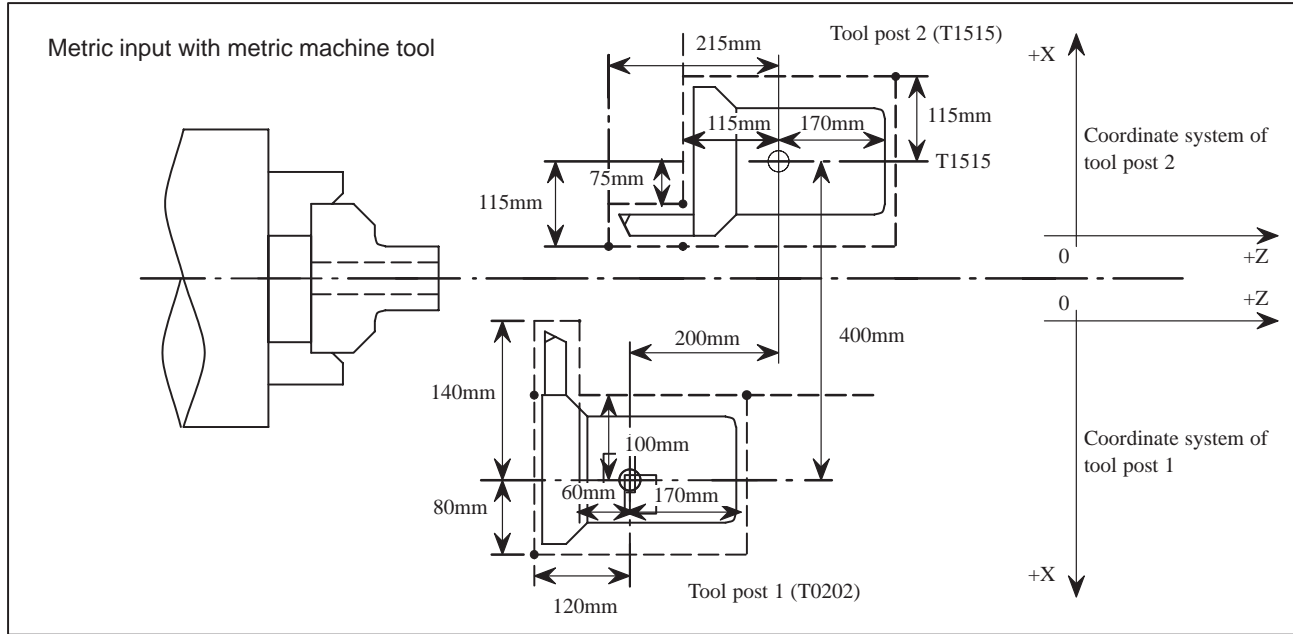
CAUTION

When parameters and interface forbidden areas (Subsec 24.3.2) are set to use the interference check function, be sure to check that correct interference forbidden areas are set. For this purpose, set manual mode, and cause the tool posts to interfere with each other in various directions.

24.3.6

Example of Making a Tool Post Interference Check

Explanations



The coordinate systems shown on the right of the figure above are the ZX plane coordinate systems of tool posts 1 and 2. For clarity, the coordinate systems are shifted; actually, the origins of the coordinate systems must match the machine zero points.

Assume the machine configuration shown above. Assume also that offset number 02 is assigned to tool post 1, and offset number 15 is assigned to tool post 2.

Suppose that the figure represents the state of reference point return operation completed with all axes (X1,Z1, X2, Z2). Then set -800 mm(diameter) and -200 mm in parameter Nos. 768 and 769, respectively. The positional relationship of the two tool posts matches type [4] indicated in item “Set the relationship between the coordinate systems of the two tool posts” in Subsec. 24.3.2. So set parameters No.048#1 and No.048#0 as follows:

Parameter No.048#1=1

Parameter No.048#0=1

Then set tool shape data (interference forbidden area) for each tool post.

The figures below show the setting of data for tool number 02 assigned to tool post 1 and for tool number 15 assigned to tool post 2.

TOOL FORM DATA		O0001 N0001
OFFSET NO. =01		
AREA 1		AREA 2
X=	20.000	X= 40.000
Z=	70.000	Z= 70.000
I=	-10.000	I= 20.000
K=	-50.000	K= 30.000
OFFSET NO. =02		
AREA 1		AREA 2
X=	80.000	X= -100.000
Z=	170.000	Z= -60.000
I=	-100.000	J= -140.000
K=	-120.000	K= -120.000
>_		S 0 T0000
MEM	**** *	12:02:08 HEAD 1
〔NO. SRH〕〔 〕〔 〕〔+INPUT〕〔 INPUT〕		

TOOL FORM DATA		O0001 N0001
OFFSET NO. =15		
AREA 1		AREA 2
X=	115.000	X= -75.000
Z=	170.000	Z= -115.000
I=	-115.000	I= -115.000
K=	-115.000	K= -215.000
OFFSET NO. =16		
AREA 1		AREA 2
X=	0.000	X= 0.000
Z=	0.000	Z= 0.000
I=	0.000	I= 0.000
K=	0.000	K= 0.000
>_		S 0 T0000
MEM	**** *	12:02:36 HEAD 2
〔NO. SRH〕〔 〕〔 〕〔+INPUT〕〔 INPUT〕		

Set data for other tools similarly. A preparation for an interference check is completed when data has been set for all tools. Turn on power. Then, an interference check is started when a T code is specified with each tool post after reference point return operation is completed with all of the four axes (X1, Z1, X2, Z2).

24.4

BALANCE CUT (G68, G69)

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

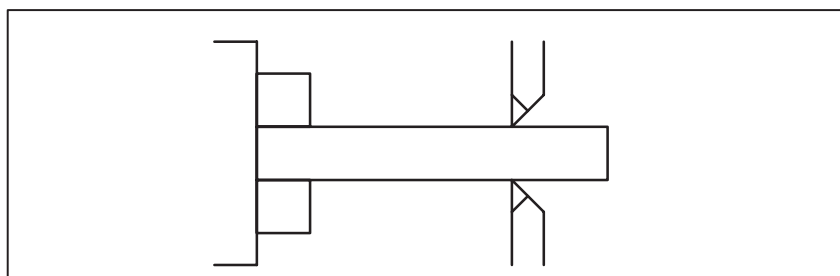


Fig. 24.4 (a) Balance cut

Explanations

When G68 is specified in the programs for both tool post 1 and tool post 2, the pulse distribution of tool post 1 is synchronized with that of tool post 2 to start balance cutting. Thus the two tool posts can move exactly at the same time to allow balance cutting.

G code	Meaning
G68	Balance cut mode
G69	Balance cut mode cancel

In the balance cut mode, balance cutting is performed only when a move command is specified for both tool posts. Balance cutting is performed even when different axes are specified for each tool post or an offset move command is specified. G68 or G69 must be specified in a single block. (Otherwise, a P/S alarm (No. 163) is raised. When G68 or G69 is specified with one tool post, the tool post does not move until the execution for the other tool post proceeds to G68 or G69. And if cutting is specified with one tool post in the balance cut mode, the tool post does not move until the execution of the other tool post proceeds to a cutting command.

CAUTION

Balance cut only starts cutting feed on both tool posts at the same time; it does not maintain synchronization thereafter. To synchronize all the movements of both tool posts, the setting for both tool posts, such as the travel distance and feedrate, must be the same.

Example

Tool post 1 program

```

:
G68 ;
G01Z100.0 ;
Z0 ;
G69 ;
:

```

Tool post 2 program

```

:
G68 ; ← Balance cut mode
G01Z100.0 ; ← Balance cut
Z0 ; ← Balance cut
G69 ; ← Balance cut mode
:      cancel

```

CAUTION

- 1 Balance cutting is not performed in dry run or machine lock state.
- 2 When rapid traverse operation is specified, balance cut processing is not performed.
- 3 A workpiece for which thread cutting has been performed in the balance cut mode cannot be subjected to thread cutting in the cancel mode. Thread cutting starts at a different position.

NOTE

- 1 Time delay before the pulse distribution of both tool posts is started is 2 msec or shorter.
- 2 In the balance cut mode, synchronization is established at the start of a move block, so movement can momentarily stop.
- 3 If feed hold operation is performed during balance cutting using both tool posts, balance cut processing is not performed at restart time, it is performed when the next move command is specified for both tool posts.
- 4 The cancel mode (G69) is set by a reset.
- 5 When the option "mirror image for double turrets" is selected, the balance cut function cannot be used.

24.5

CUSTOM MACRO VARIABLES COMMON TO TOOL POSTS

Some or all of the custom macro common variables (#100 to #149, #500 to #531) can be made commonly usable by both tool post 1 and tool post 2; such variables can be written to or read from either tool post. See Item "Types of variables" in Section 16.1.

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 which is machine's own.

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.

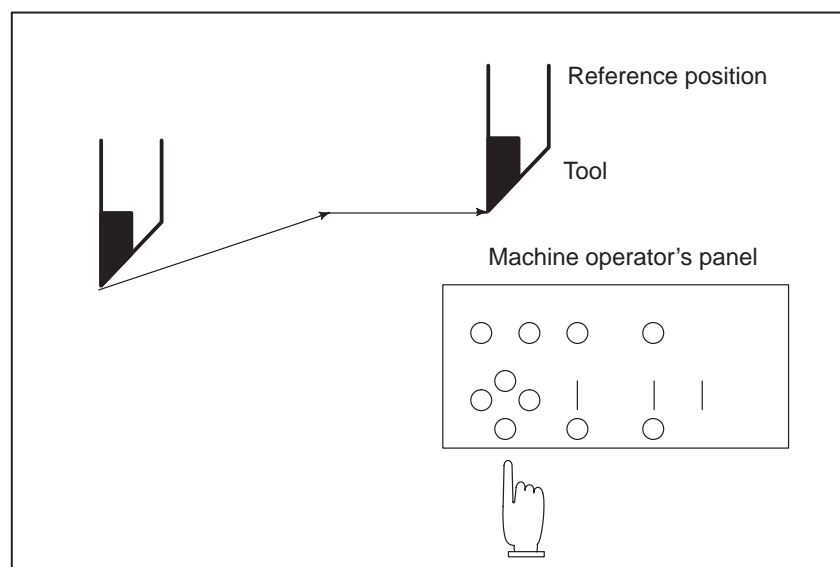


Fig. 1.1 (a) Manual reference position return

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, push buttons, or the manual handle, the tool can be moved along each axis.

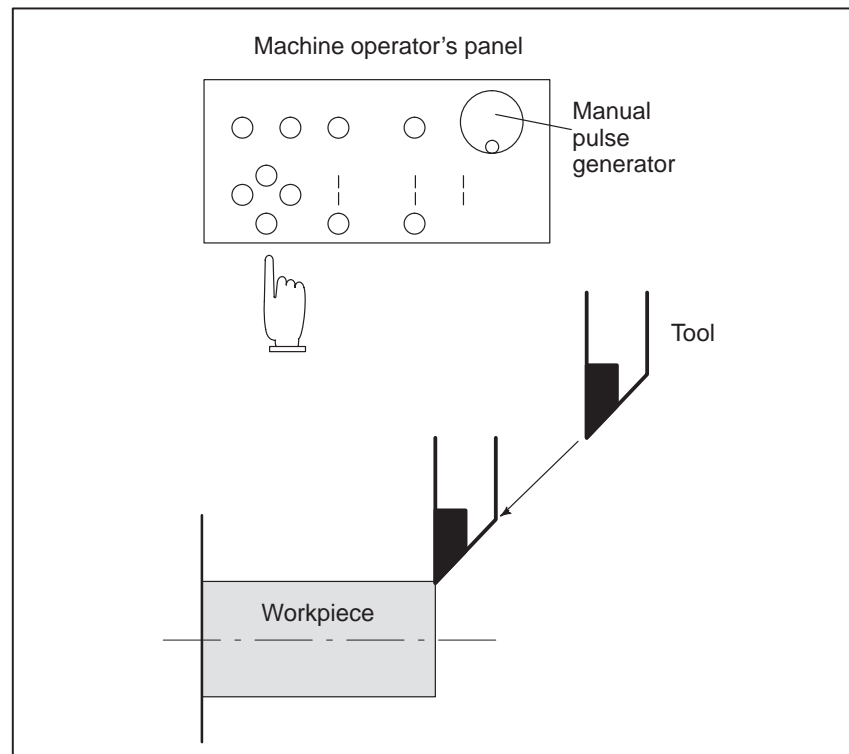


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

- (i) Jog feed (See Section III-3.2)
The tool moves continuously while a pushbutton remains pressed.
- (ii) Incremental feed (See Section III-3.3)
The tool moves by the predetermined distance each time a button is pressed.
- (iii) Manual handle feed (See Section III-3.4)
By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, DNC and MDI operations. (See Section III-4).

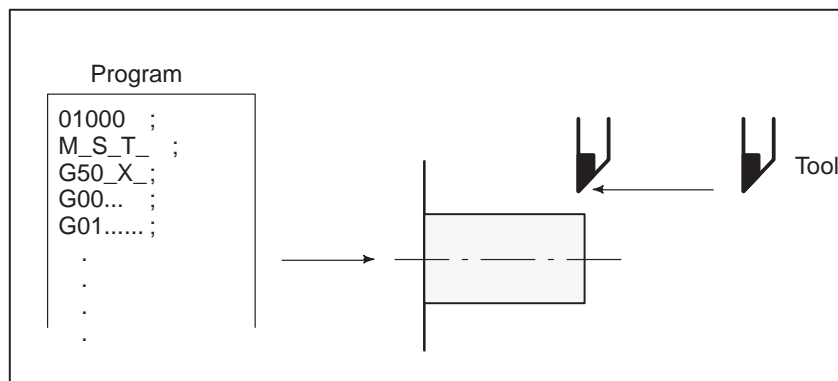


Fig. 1.2 (a) Tool Movement by Programming

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.

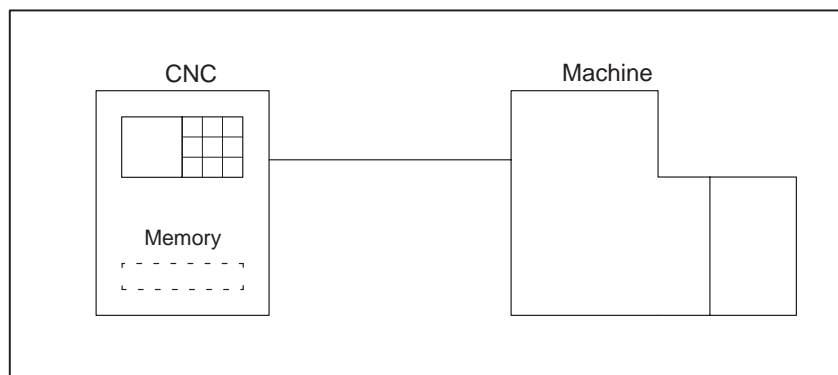


Fig. 1.2 (b) Memory Operation

- **DNC operation**

The machine can operate by reading a program directly from a connected I/O device, without registering the program into CNC memory. This function is useful when a program is too large to be registered into CNC memory. This function can also be used when remote buffers are used for high-speed machining.

- **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.

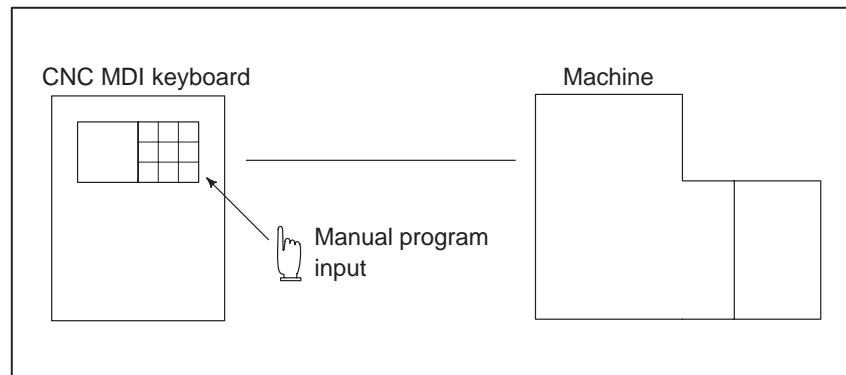


Fig. 1.2 (c) MDI operation

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).

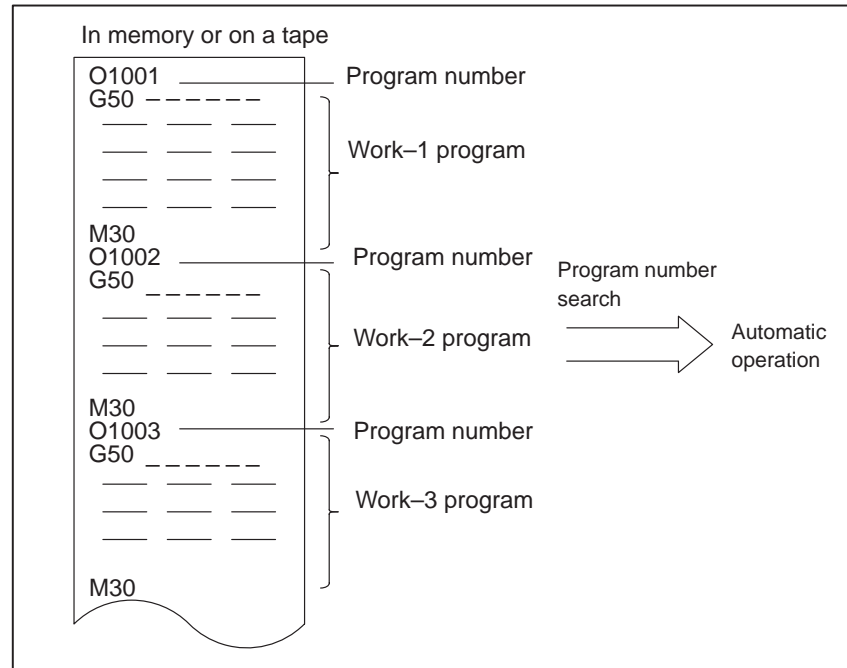


Fig. 1.3 (a) Program Selection for Automatic Operation

- **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.

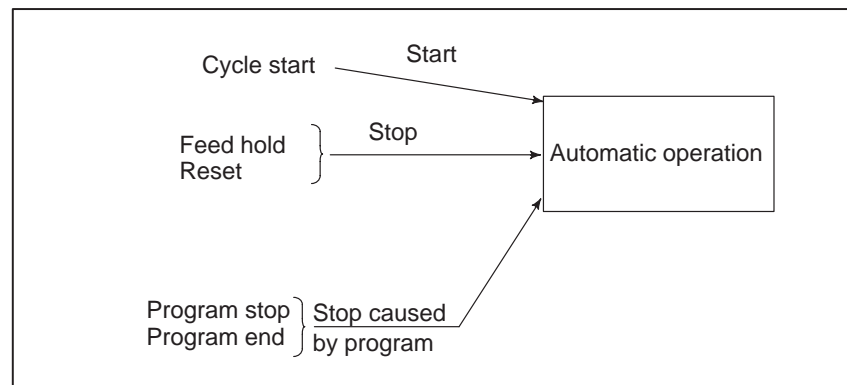


Fig. 1.3 (b) Start and Stop for Automatic Operation

- **Handle interruption (See Section III-4.6)**

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle.

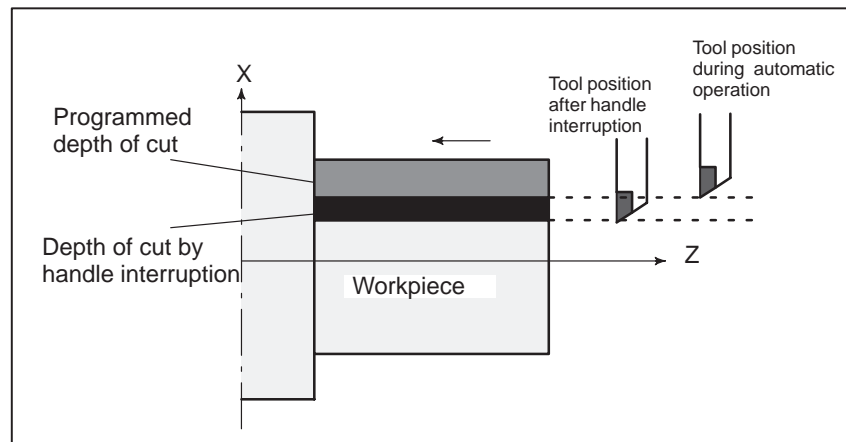


Fig. 1.3 (c) Handle Interruption for Automatic Operation

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.

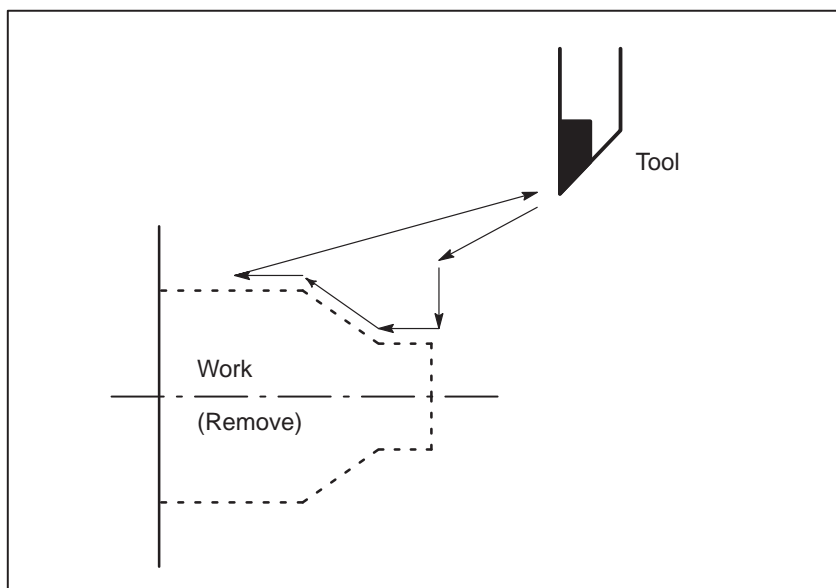


Fig. 1.4.1 (a) Dry run

- **Feedrate override**
(See Section III-5.2)

Check the program by changing the program of feed rate command.

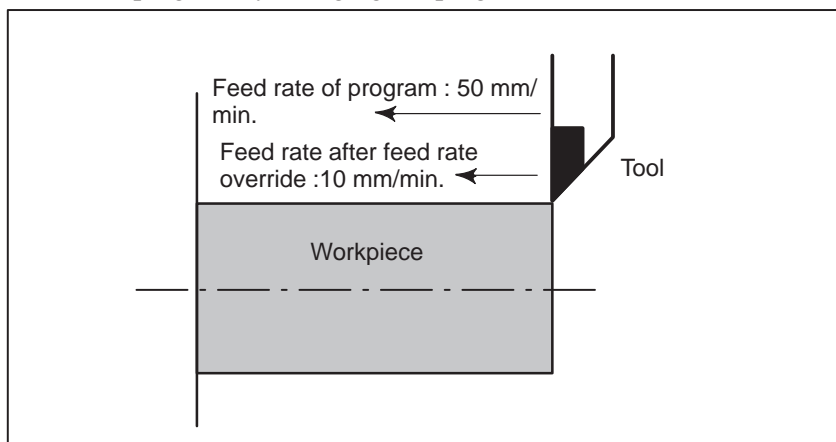


Fig. 1.4.1 (b) Feedrate Override

- **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.

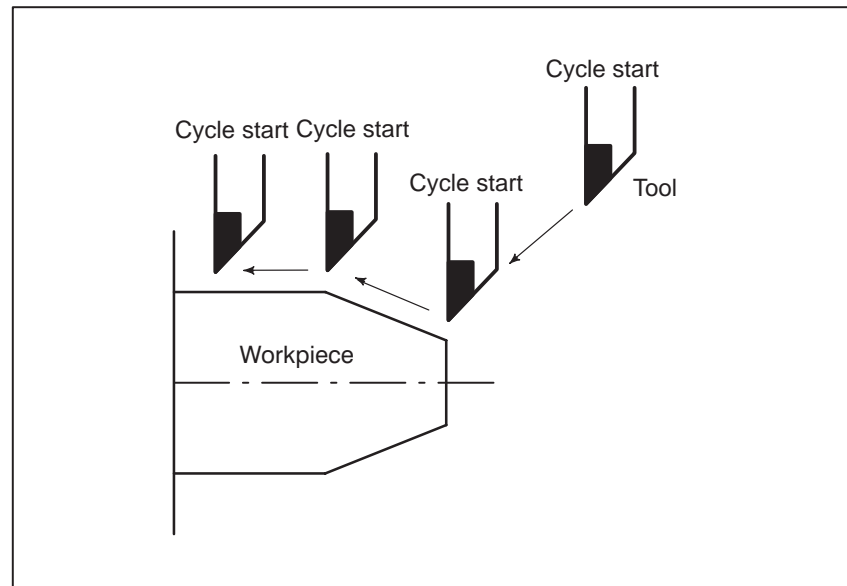


Fig. 1.4.1 (c) Single Block

1.4.2

How to View the Position Display Change without Running the Machine

Explanations

- **Machine lock**
(See Sections III-5.1)

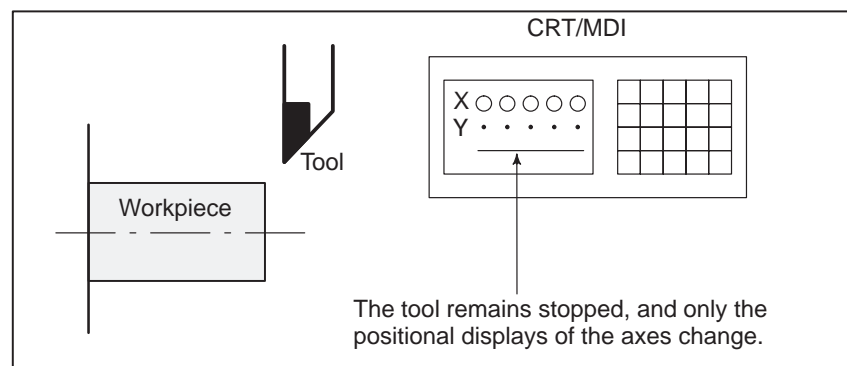


Fig. 1.4.2 Machine Lock

- **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.

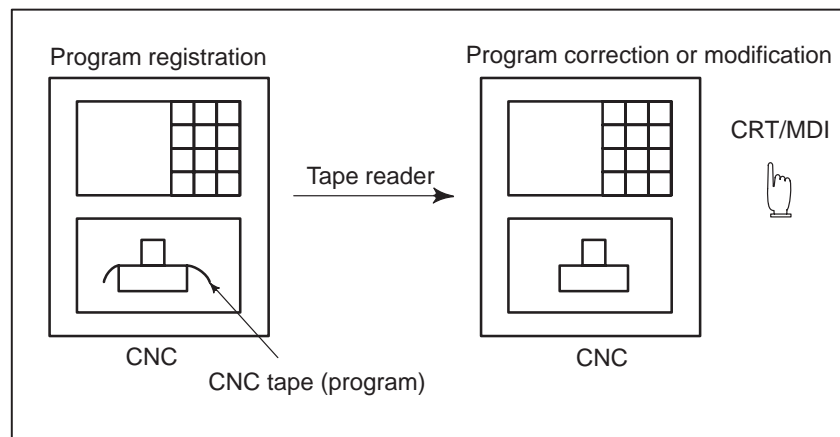


Fig. 1.5 Part Program Editing

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).

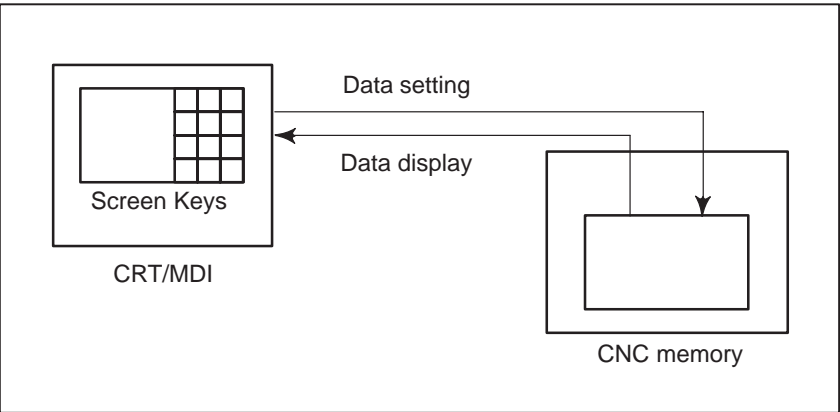


Fig. 1.6 (a) Displaying and Setting Data

Explanations

- Offset value

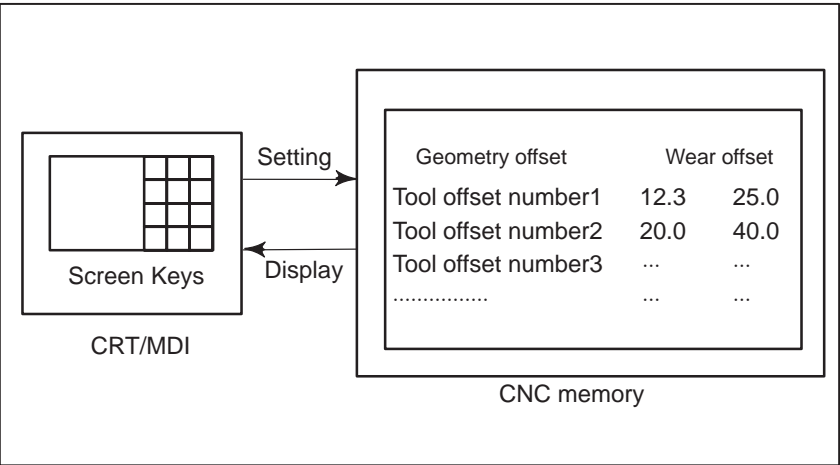


Fig. 1.6 (b) Displaying and Setting Offset Values

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).

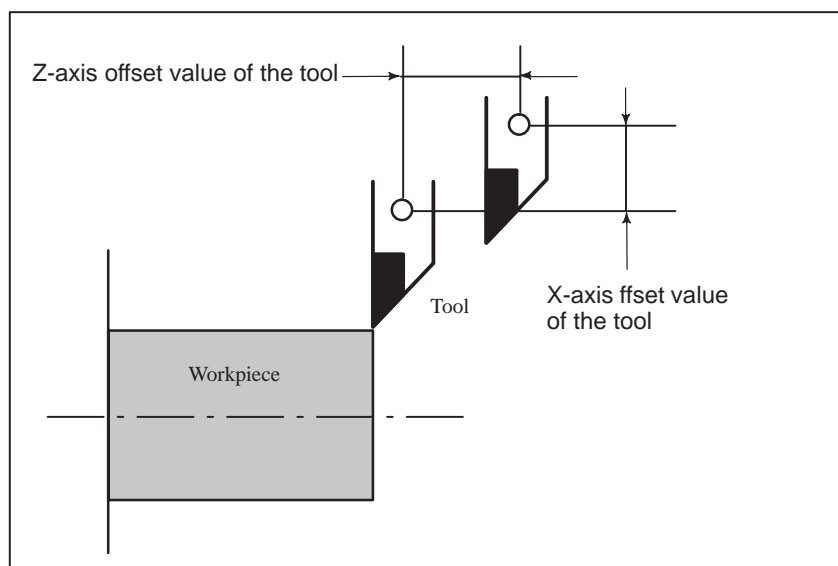


Fig. 1.6 (c) Offset Value

- **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

The above data is called setting data (See Section III-11.4.7).

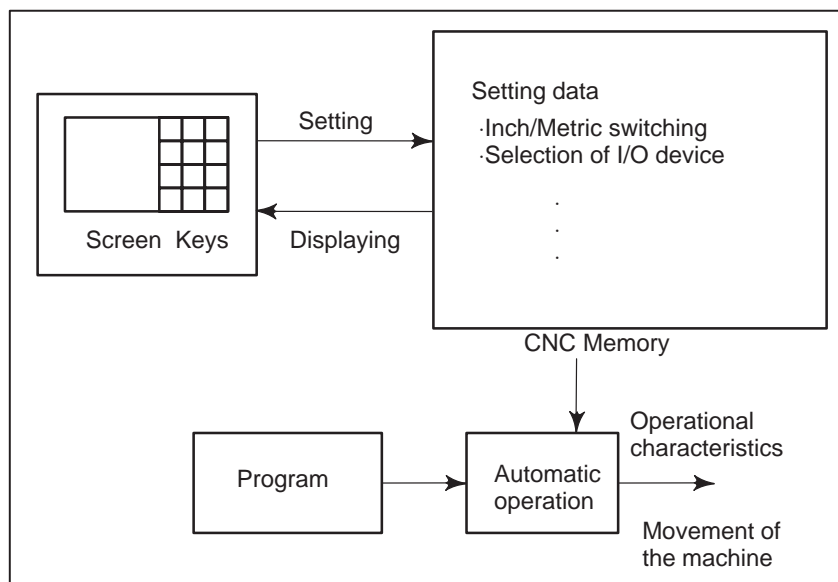


Fig. 1.6 (d) Displaying and Setting Operator's setting data

• 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.

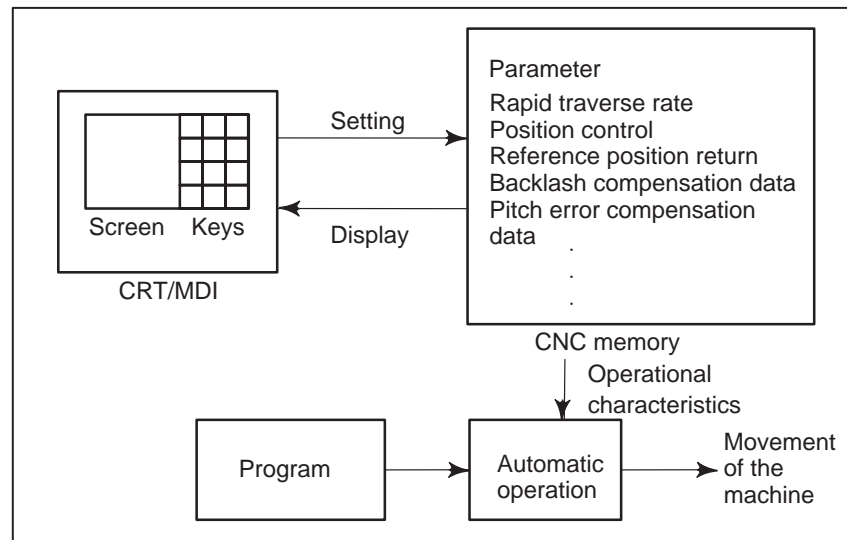


Fig. 1.6 (e) Displaying and setting parameters

• 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).

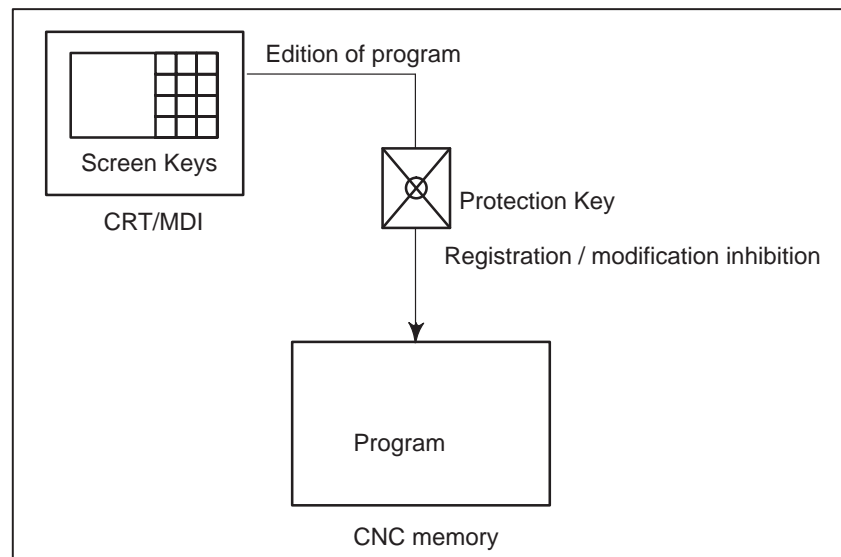


Fig. 1.6 (f) Data Protection Key

1.7

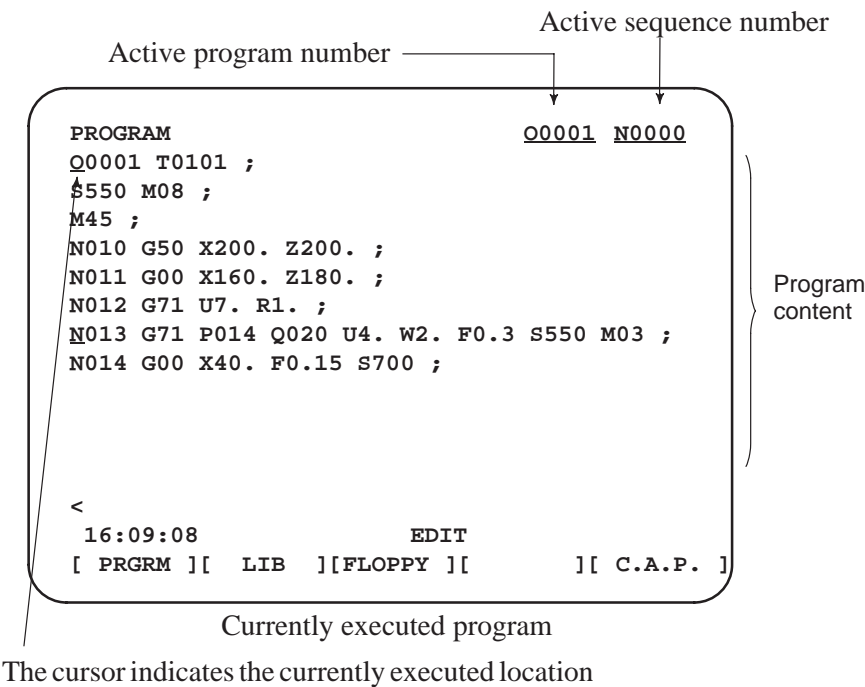
DISPLAY

1.7.1

Program Display

(See Section III-11.2.1)

The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed.

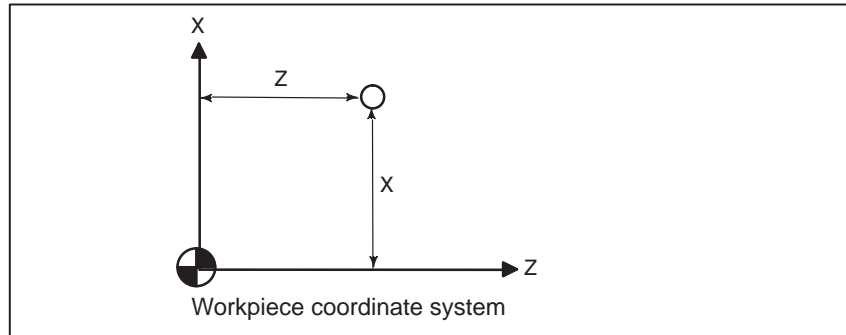


```
PROGRAM                                00001  N0000
SYSTEM SDITION                        0666 - 21
PROGRAM NO, USED :      24 FREE :    39
MEMORY AREA USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
00021 00041 00615 00651 00601 00645
00613 00021 01041 01051 00010 02011
02505 00011 03511 03148 03153 04011
04048 05221 05111 05766 06032 00001

<
16:11:12                                EDIT
[ PRGRM ][CONDNS ][           ][ C.A.P. ]
```

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)		O0001 N0023
X	200.000	
Z	220.000	
C	0.000	
Y	0.000	
PART COUNT 1786		
RUN TIME 2H47M	CYCLE TIME 0H 1M47S	
ACT.F 3000 MM/M		
16:14:02	AUTO	
[ABS]	[REL]	[ALL]
[HNDL]		

1.7.3 Alarm Display (See Section III-7.1)

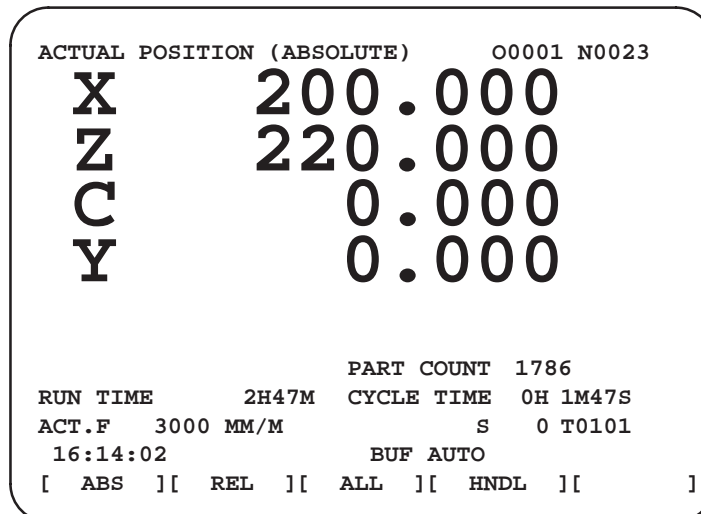
When a trouble occurs during operation, error code and alarm message are displayed on CRT screen. See APPENDIX 7 for the list of error codes and their meanings.

ALARM MESSAGE		O0001 N0011
010 P/S ALARM		
16:20:30 ALARM		S 0 T0101
BUF MDI		
[ALARM]	[OPR]	[MSG]

1.7.4

Parts Count Display, Run Time Display (See Section III-11.5.3)

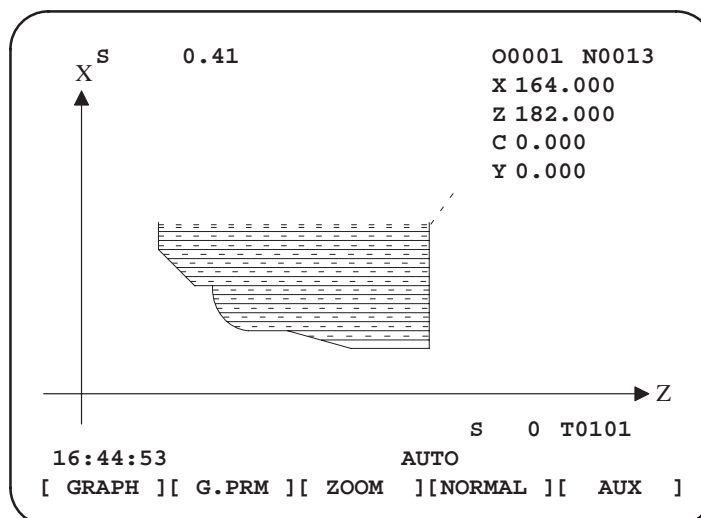
When option is selected, two types of run time and number of parts are displayed on the screen.



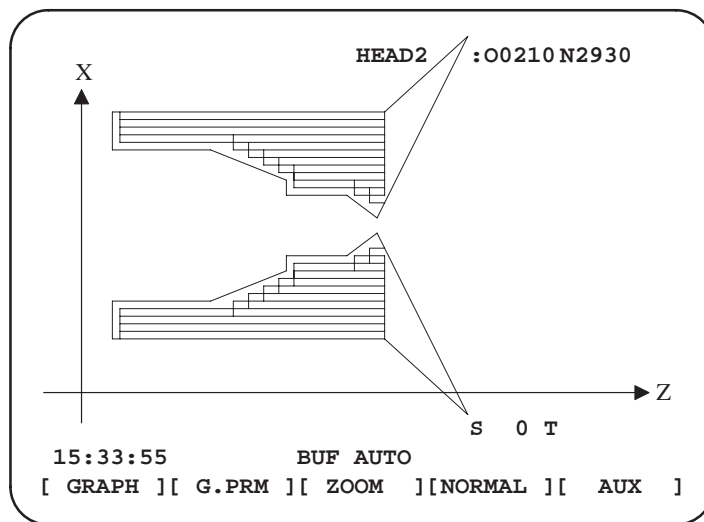
1.7.5

Graphic Display (See Section III-12)

The graphic can be used to draw a tool path for automatic operation and manual operation, thereby indicating the progress of cutting and the position of the tool.



0-TC



0-TTC

1.8 DATA OUTPUT

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.

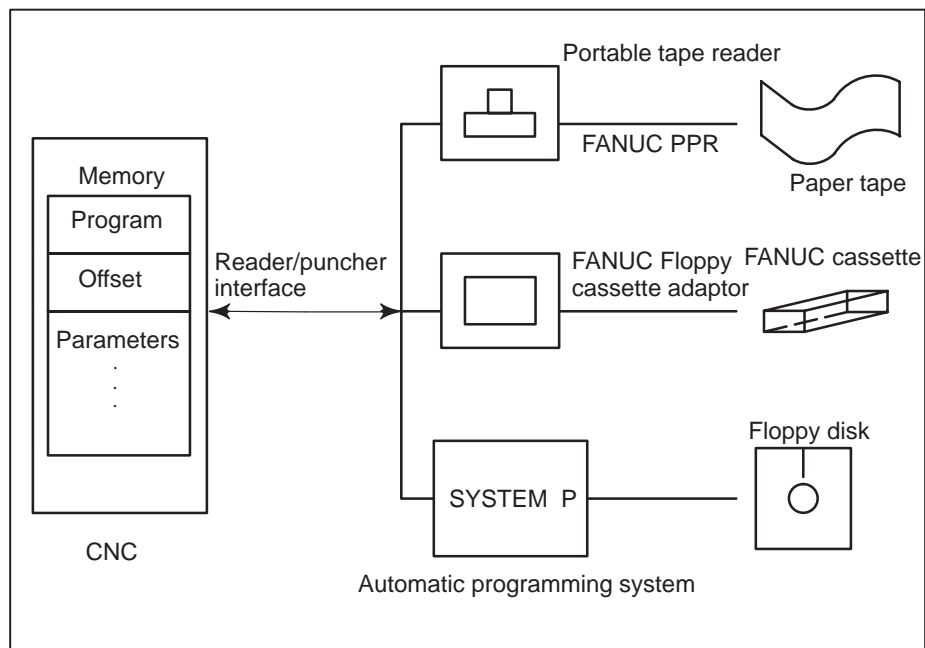


Fig. 1.8 Data Output

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

Figs. 2.1 (a) to 2.1 (f) show the CRT/MDI.

9" small monochrome CRT/MDI (with soft key) . . . Fig.2.1(a)

9" full key monochrome CRT/MDI (with soft key) . . Fig.2.1(b)

External view

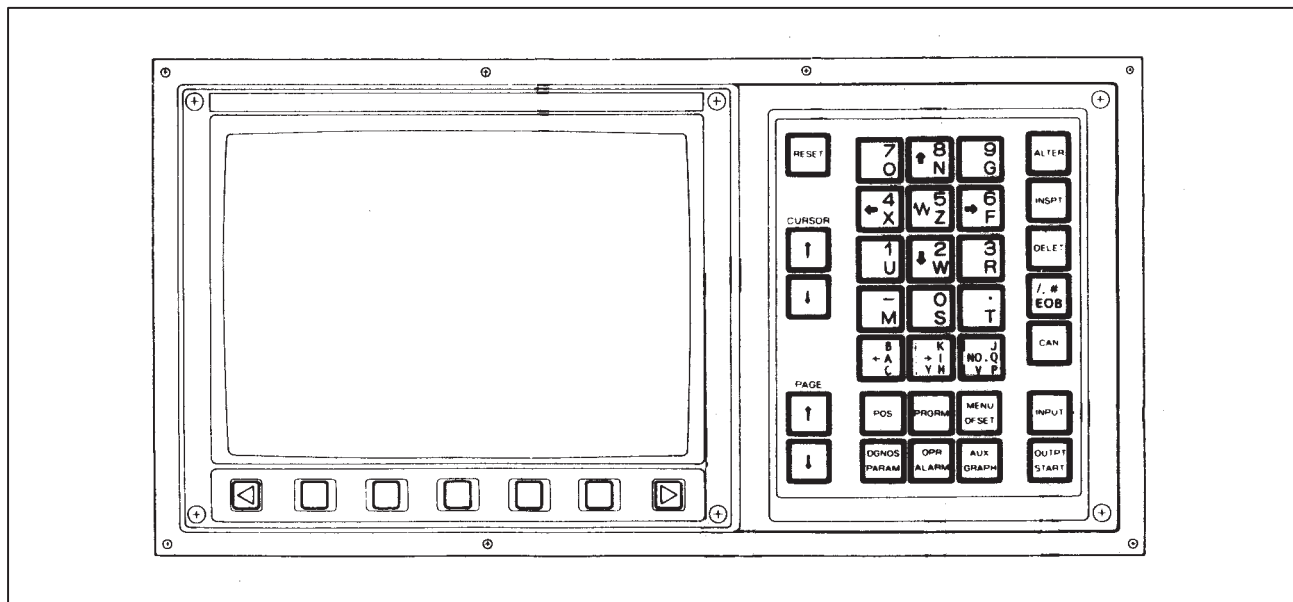


Fig. 2.1 (a) 9" small monochrome CRT/MDI (with soft key)

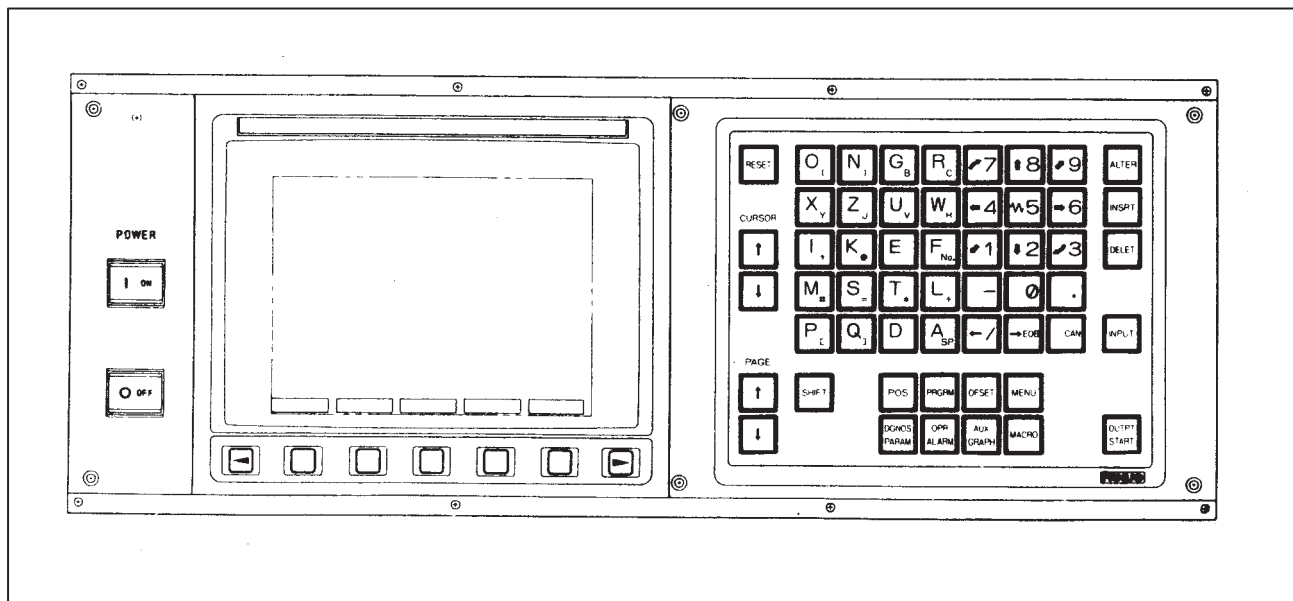


Fig. 2.1 (b) 9" full key monochrome CRT/MDI (with soft key)

Explanation of the keyboard

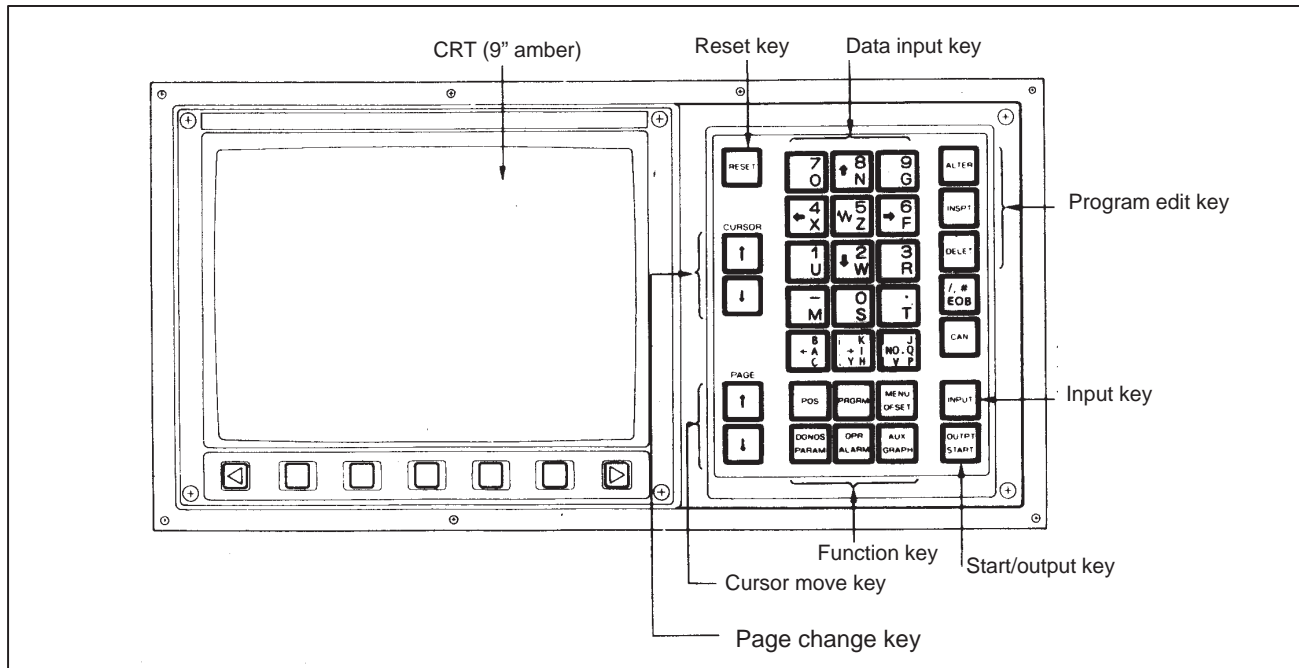


Fig. 2.1 (c) 9" small monochrome CRT/MDI panel (with soft key)

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	HELP key, START key 	Used to start MDI or automatic operation. The use of this key depends on the machine tool builder. Refer to the manual of the machine tool builder. This key is used also to output data to an I/O 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 \hat{E} is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered. (This key is not on standard type MDI keyboard)
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 equivalent to the [INPUT] key of the soft keys. This operation is same as using I/O devices.

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. When the key input buffer displays
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.  : This key is used to move the cursor in an upward or reverse direction.
11	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.

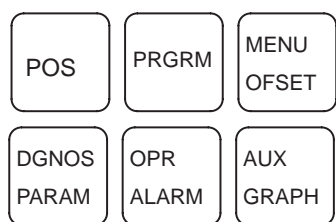
Explanations

● Keyboard operation in the Series 0-TTC

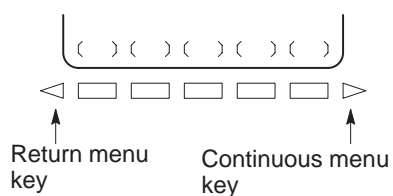
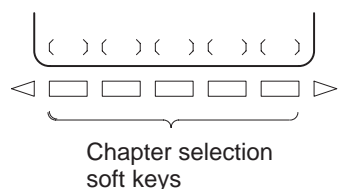
In the Series 0-TTC, be sure to select the tool post for which data is specified, using the tool-post selection switch on the machine operator's panel. Then, perform keyboard operation, such as displaying or specifying various data items, and editing a program.

2.2 FUNCTION KEYS AND SOFT KEYS

2.2.1 General Screen Operations



Function keys



- 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 chapter selection 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 chapter selection 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 soft keys are not provided, press the key with the identical function to select a desired chapter. The description of the soft keys is automatically displayed according to the optional configuration. The description of the soft keys can be displayed regardless of the optional configuration if bit 7 of parameter 048 is set 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 and panels:



Press this key to display the **position screen**.



Press this key to display the **program screen**.



Press this key to display the **offset/screen**.



Press this key to display the **setting screen / parameter screen / diagnosis screen**.



Press this key to display the **alarm screen/operator message screen**.



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.

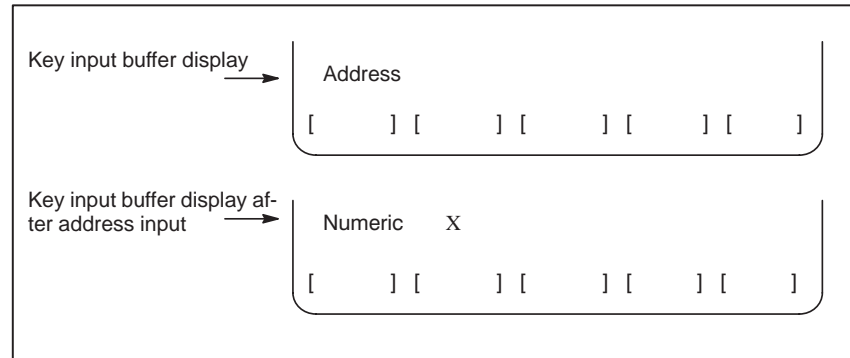
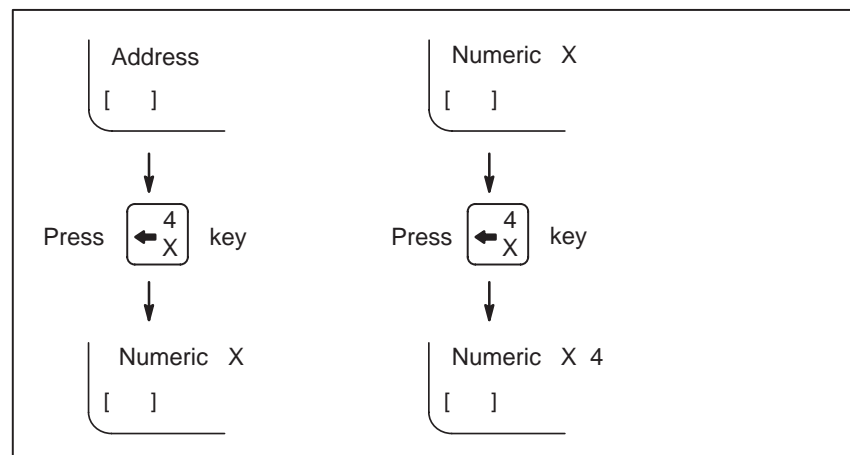


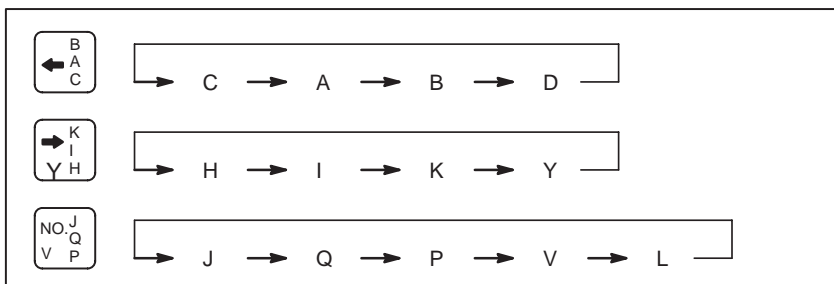
Fig. 2.2.3 Key input buffer display

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.

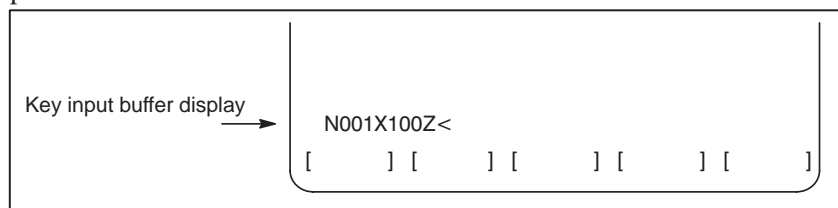


Fig. 2.2.3 (b) Key input buffer display

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 " ^ ". 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.

2.3 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 (a) External I/O device

Device name	Usage	Max. storage capacity	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

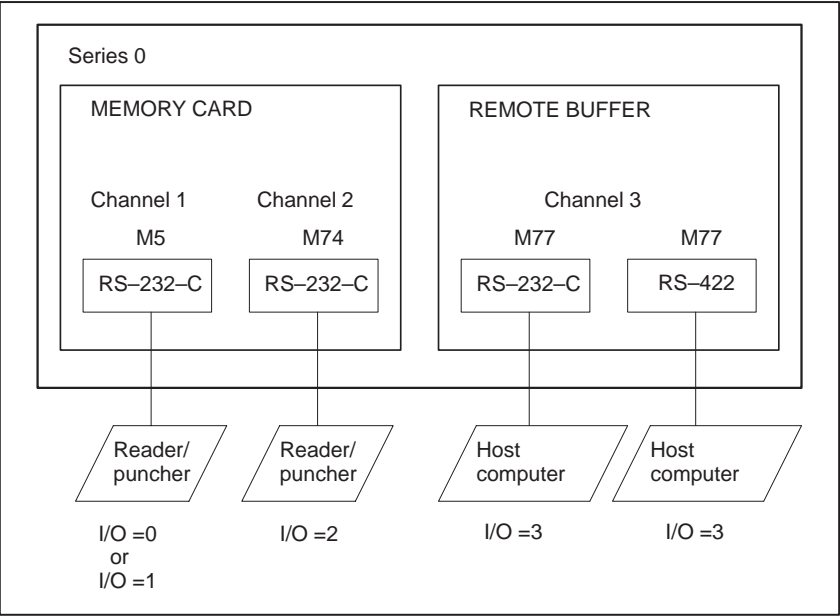
The following data can be input/output to or from external input/output devices:

- 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.

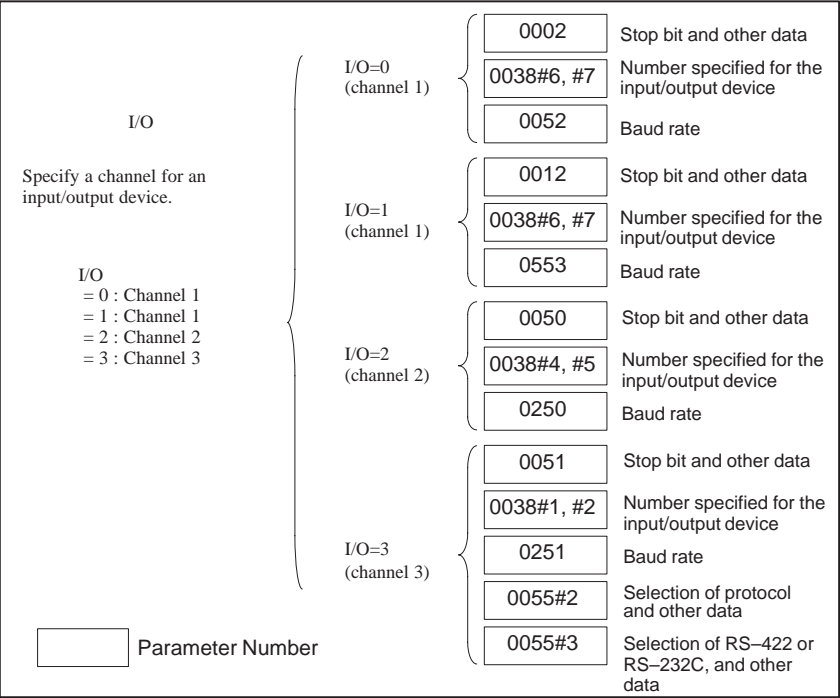


Series 16/18 has three channels of reader/punch interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting parameter I/O.

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.

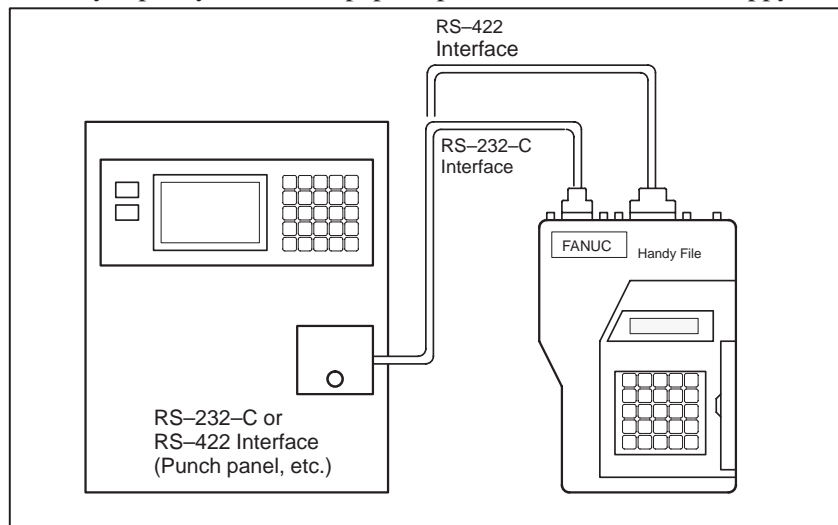


2.3.1 FANUC Handy File

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.

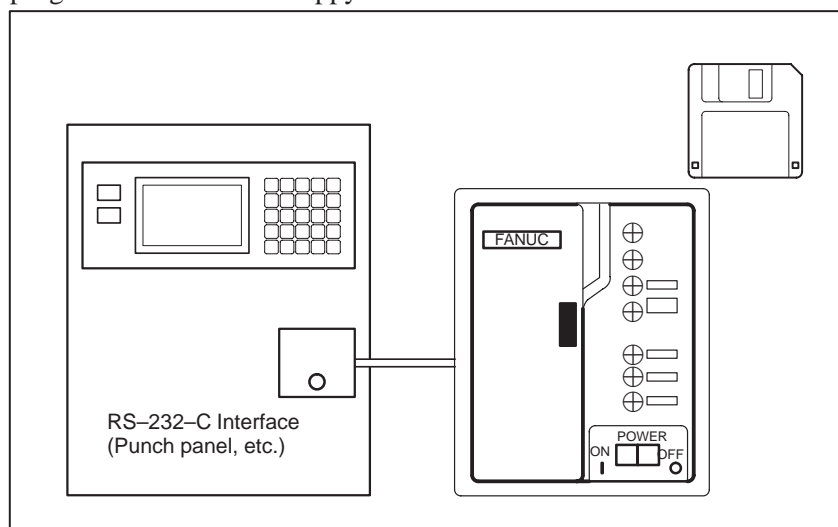
The Handy File uses 3.5-inch floppy disks, which do not have the problems of paper tape (i.e., noisy during input/output, easily broken, and bulky).

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

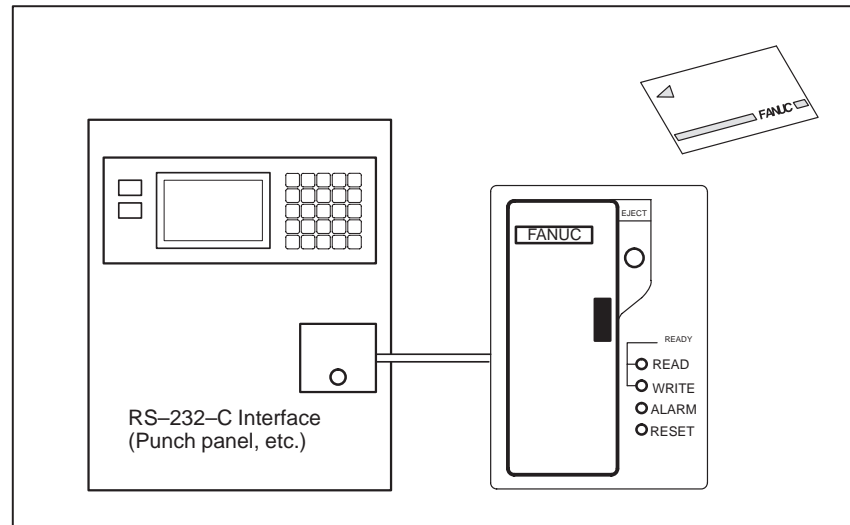
When the Floppy Cassette is connected to the NC, machining programs stored in the NC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the NC.



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.

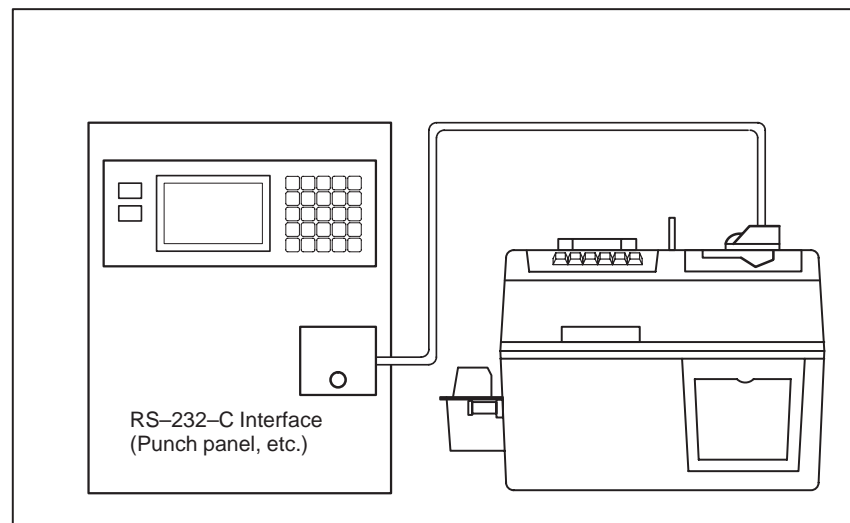
When an FA Card is connected to the CNC via the card adapter, NC machining programs stored in the CNC can be transferred to and saved in an FA Card. Machining programs stored on an FA Card can also be transferred to the CNC.



2.3.4 FANUC PPR

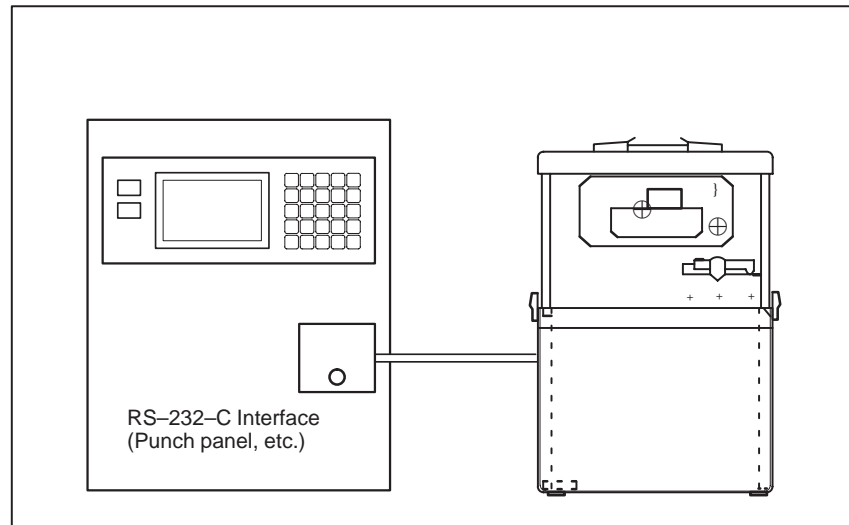
The FANUC PPR consists of three units: A printer, paper tape punch, and paper tape reader.

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

- 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 screen shown in Section 2.4.2 is displayed, a system failure may have occurred.

If the machine tool is in the emergency stop state, the software configuration screen, shown in 2.4.2, appears.

Press the  function key on the CRT/MDI panel, or release the machine from emergency stop.

The position screen then appears. If the position screen is not displayed, a system failure may have occurred.

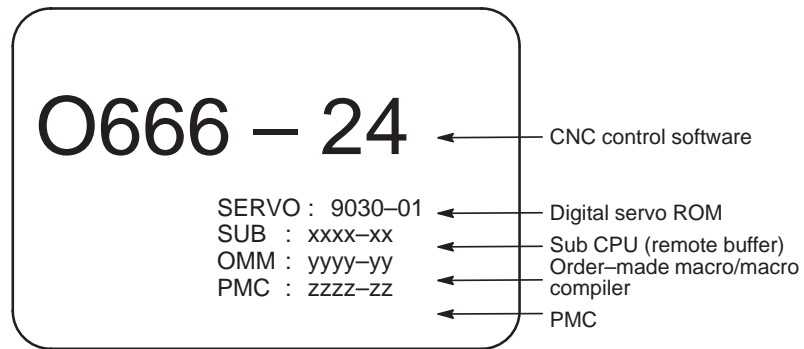
ACTUAL POSITION (ABSOLUTE)		O0001 N0023	
X	200.000		
Z	220.000		
C	0.000		
Y	0.000		
PART COUNT 1786			
RUN TIME 2H47M	CYCLE TIME 0H 1M47S		
ACT.F 3000 MM/M	S 0 T0101		
16:14:02	BUF AUTO		
[ABS]	[REL]	[ALL]	[HNDL]

- 4 Check that the fan motor is rotating.

WARNING

When pressing the <POWER ON> key. Until the positional or alarm screen is displayed, do not touch keys of CRT/MDI panel. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

Display of software configuration



2.4.2 Power Disconnection

Procedure for Power Disconnection


- 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 four kinds as follows :

1. **Manual reference position return**
2. **Jog feed**
3. **Incremental feed**
4. **Manual handle feed**
5. **Manual absolute on/off**

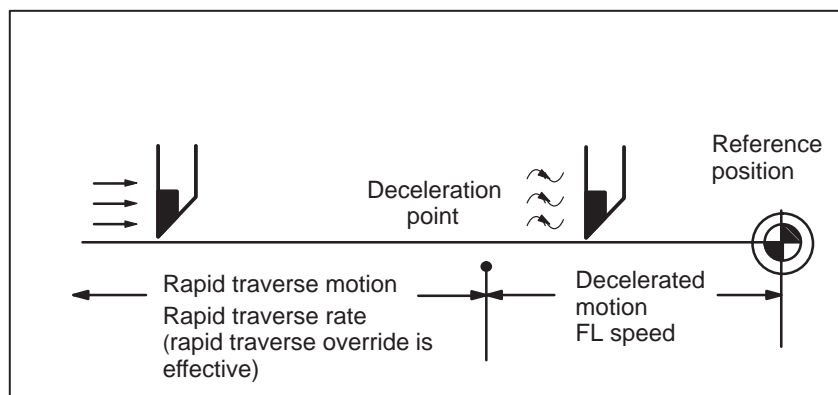
3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :

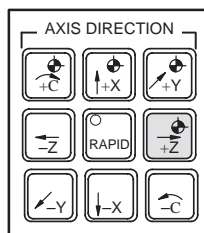
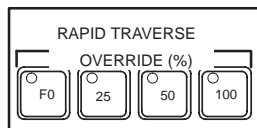
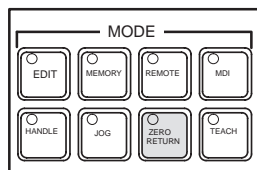
The tool is moved in the direction specified in parameter (bit0 to 3 of No.0003) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 0518 to 0521, 0533, 0544).

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 (bit4 of No.0049).



Procedure for Manual Reference Position Return Operation



- 1 Press the reference position return switch, one of the mode selection switches.
- 2 To decrease the feedrate, press a rapid traverse override switch.
- 3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter.
When the tool has returned to the reference position, the reference position return completion LED goes on.
- 4 Perform the same operations for other axes, if necessary.
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

ZERO POSITION				PRO-GRAM STOP	M02/ M30	MANU ABS				
X	Z	C	Y							
<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
TOOL NUMBER										
1	2	3	4	5	6	7	8	NC?	MC?	
<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Explanation

- **Automatically setting the coordinate system**

Bit7 of parameter No.0010 is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.

When α and γ are set in parameter 0708 to 0711, the workpiece coordinate system is determined so that the reference point on the tool holder or the position of the tip of the reference tool is $X=\alpha, Z=\gamma$ when reference position return is performed.

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:

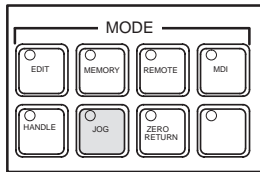
- Moving from the reference position.
- Entering an emergency stop state.

- **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.

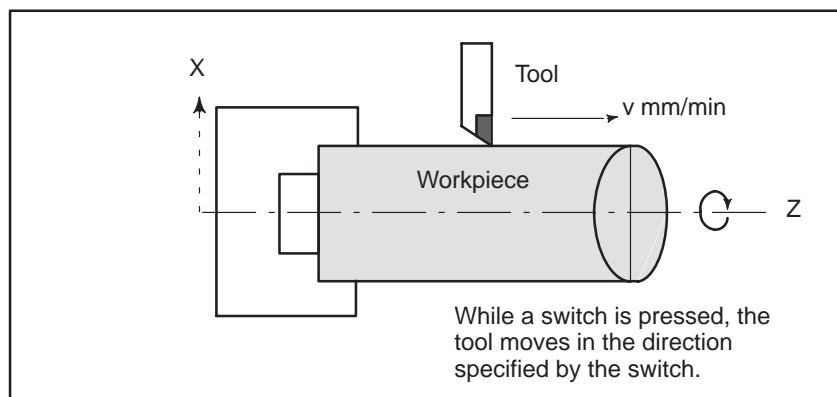
The jog feedrate is described following table 3.2.

Table 3.2 Jog Feedrate

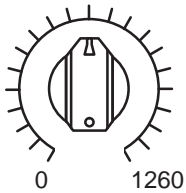
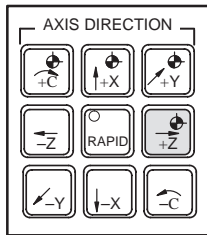
Rotary switch position	Feedrate		Rotary switch position	Feedrate	
	Metric input (mm/min)	Inch input (inch/min)		Metric input (mm/min)	Inch input (inch/min)
0	0	0	8	50	2.0
1	2.0	0.08	9	79	3.0
2	3.2	0.12	10	126	5.0
3	5.0	0.2	11	200	8.0
4	7.9	0.3	12	320	12
5	12.6	0.5	13	500	20
6	20	0.8	14	790	30
7	32	1.2	15	1260	50

Note The feedrate error (about $\pm 3\%$) affects on the feedrate in the table above.

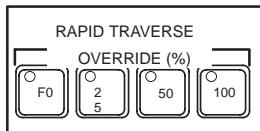
The jog feedrate can be adjusted with the jog feedrate override dial. Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate regardless of the position of the jog feedrate override dial. Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.0049#4).



Procedure for Jog Feed Operation



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 described table 3.2. The tool stops when the switch is released.
- 3 The jog feedrate can be adjusted with the jog feedrate override switch.
- 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**

Acceleration/deceleration method and time constant for rapid traverse are the same as G00 in programmed command.

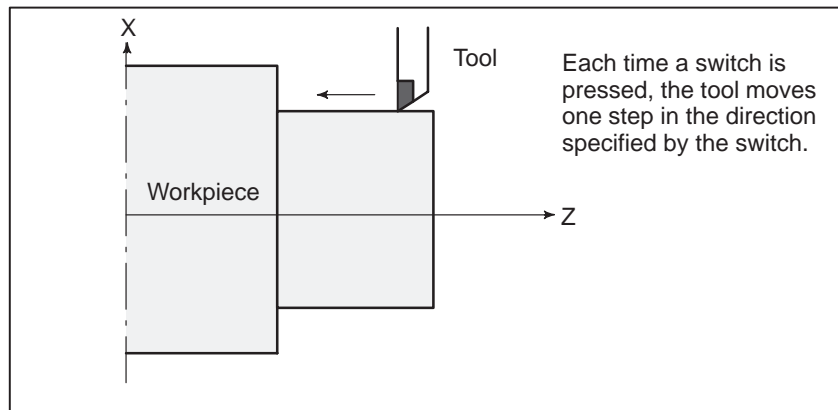
Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

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.0010#0).

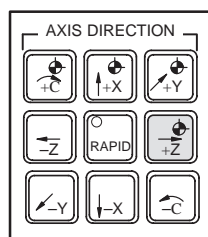
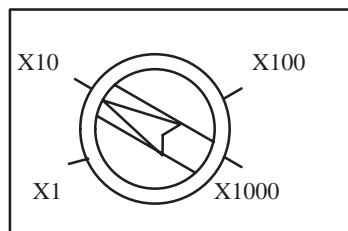
3.3 INCREMENTAL FEED

In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.

This mode is effective when a manual pulse generator is not connected.



Procedure for Incremental Feed Operation



- 1 Press the INC switch, one of the mode selection switches.
- 2 Select the distance to be moved for each step with the magnification dial.
- 3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.
- 4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate. Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

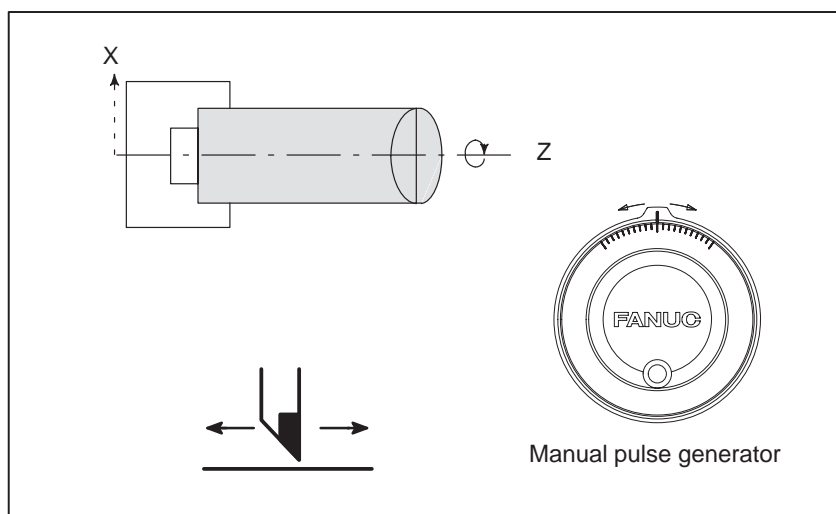
- **Travel distance specified with a diameter**

The distance the tool travels along the X-axis can be specified with a diameter.

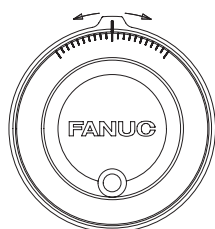
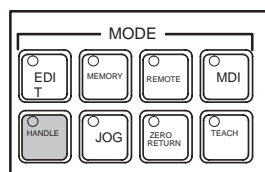
3.4 MANUAL HANDLE FEED

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. 0121 and 0699).



Procedur for Manual Handle Feed Operation



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.

Explanation

- **Availability of manual handle feed in Jog mode**

Parameter (bit 0 of No. 0013) enables or disables the manual handle feed in the JOG mode.

When the parameter (bit 0 of No. 0013) is set 1, both manual handle feed and incremental feed are enabled.

- **Availability of manual handle feed in TEACH IN JOG mode**

Parameter (bit 6 of No. 0002) enables or disables the manual handle feed in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate**

Parameter (bit 4 of No. 0060) 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.)

- **Movement direction of an axis to the rotation of MPG**

Parameter (No.0386#0 to 3) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Restrictions

Number of MPGs

Up to two manual pulse generators can be connected, one for each axis. The two 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.

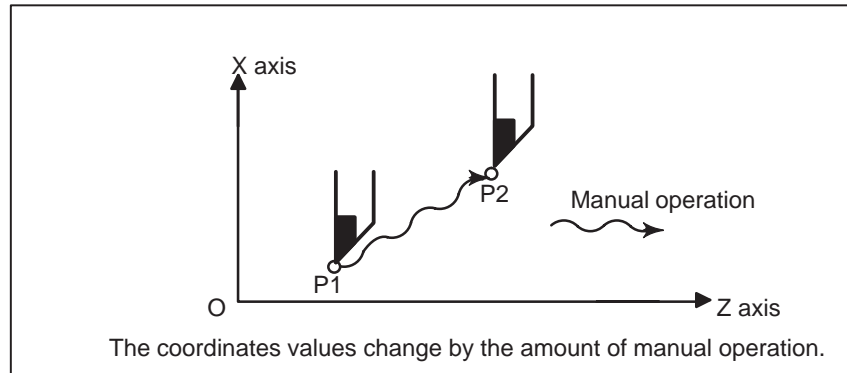


Fig. 3.5 (a) Coordinates with the switch ON

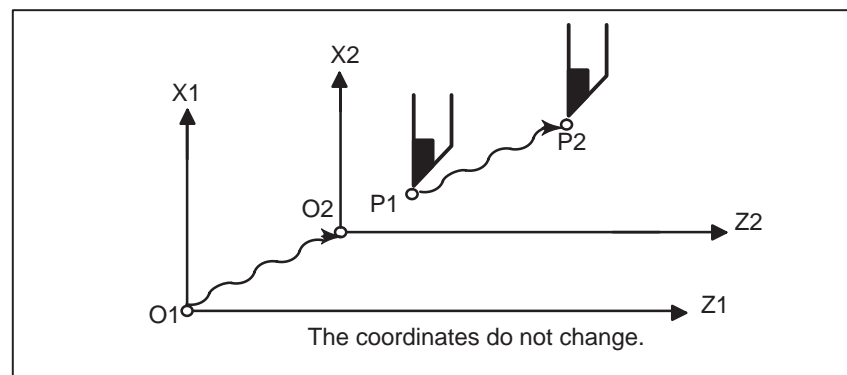




Fig. 3.5 (b) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01    X100.0Z100.0F010 ;    [1]
        X200.0Z150.0    ;    [2]
        X300.0Z200.0    ;    [3]
```

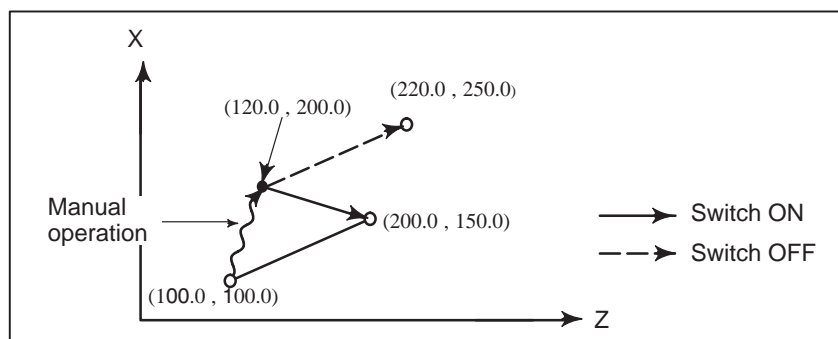
The subsequent figures use the following notation:

 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.

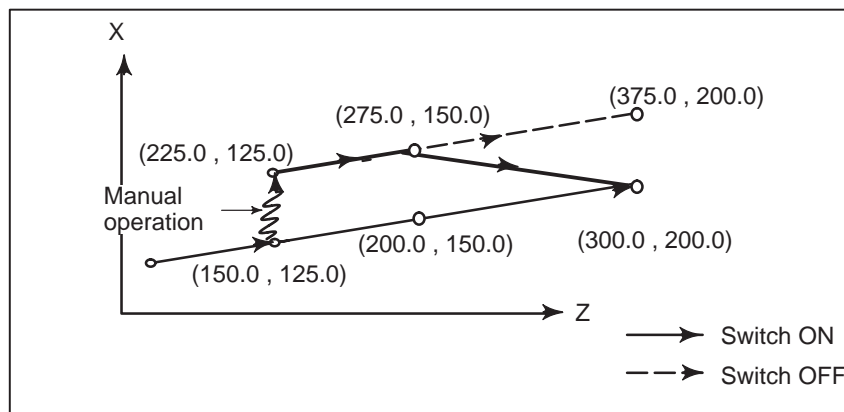
- Manual operation after the end of block

Coordinates when block [2] has been executed after manual operation (X-axis +20.0, Z-axis +100.0) at the end of movement of block [1].



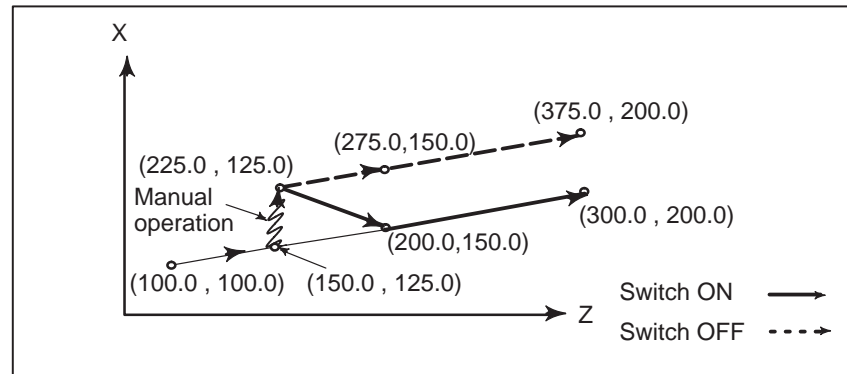
- Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block [2] is being executed, manual operation (X-axis + 75.0) is performed, and the cycle start button is pressed and released



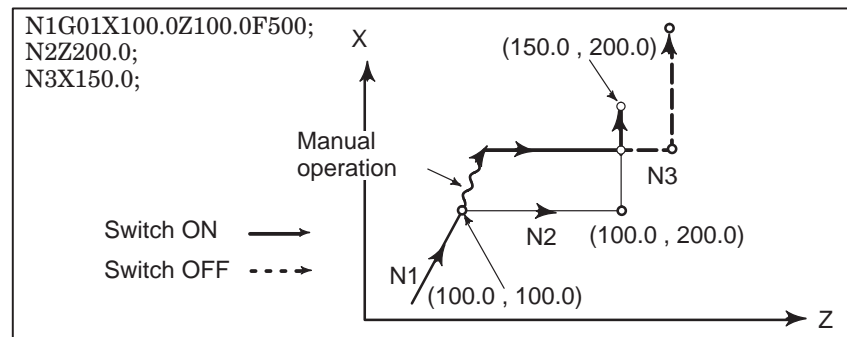
- **When reset after a manual operation following a feed hold**

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.

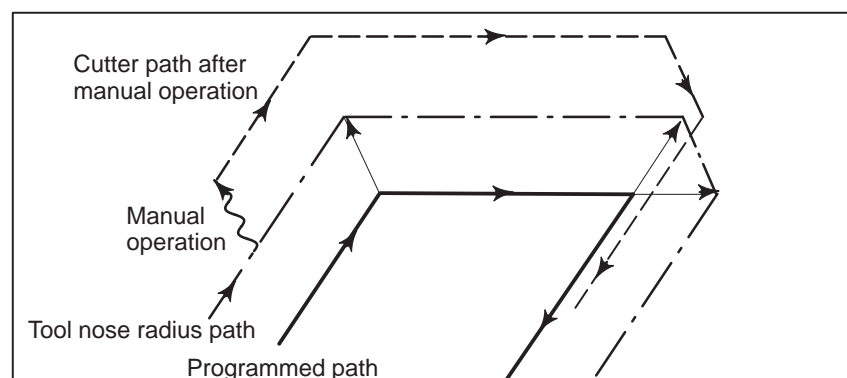


- **When the next move block is an incremental**
- **Manual operation during tool nose radius compensation**

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 tool nose radius compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed. The amount of separation equals to the amount that was performed manually.

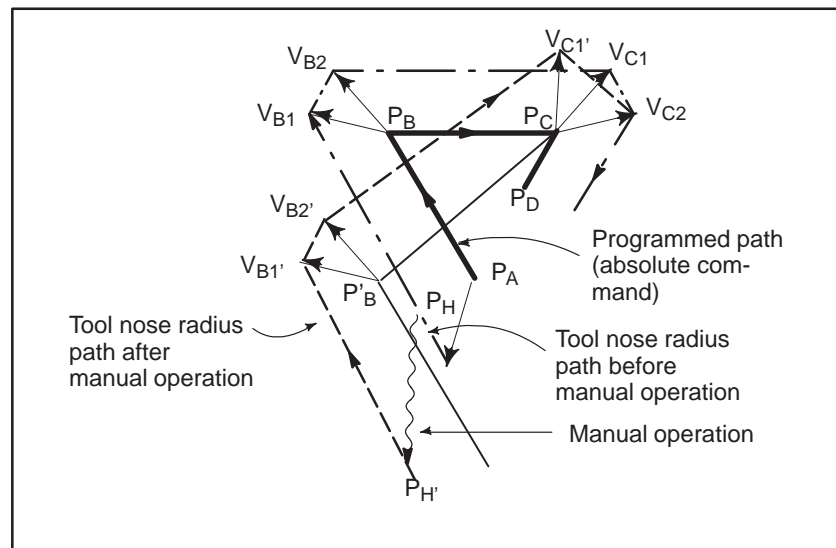


When the switch is ON during tool nose radius compensation

Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the tool nose radius compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

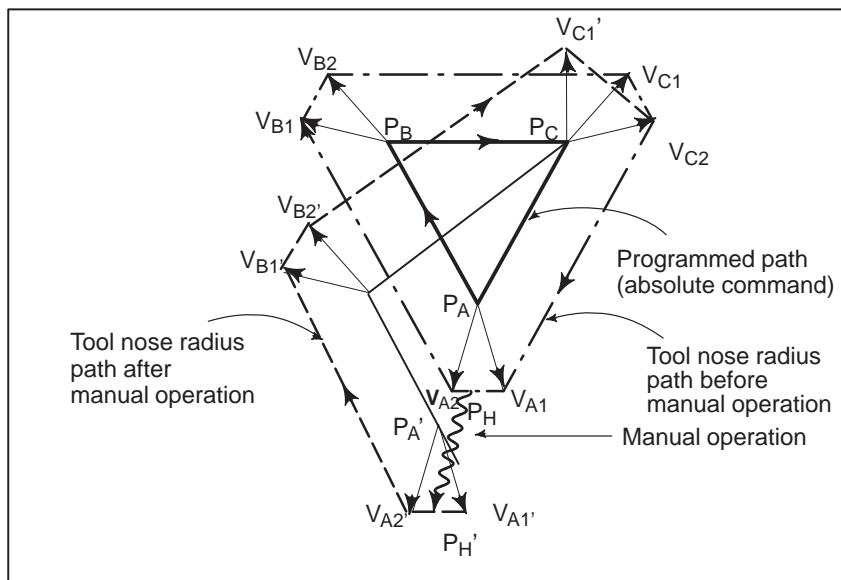
Manual operation performed in other than cornering

Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A , P_B , and P_C and that the tool was manually moved to $P_{H'}$. 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

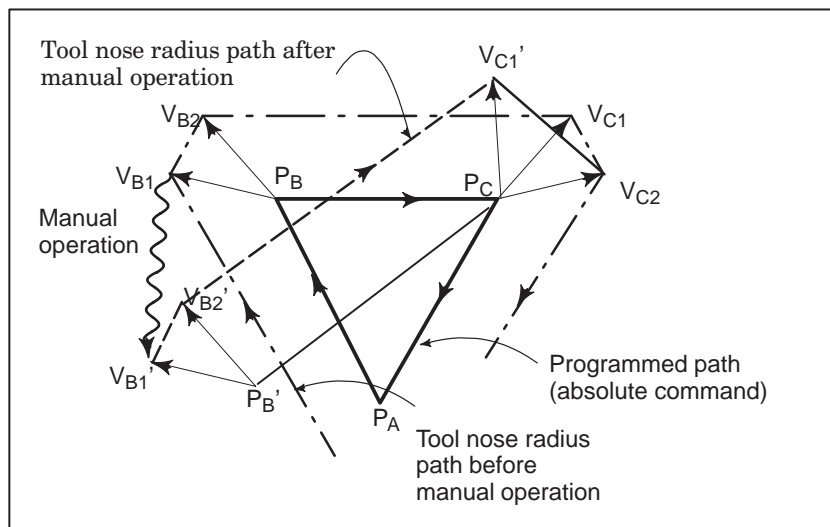
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2} , V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2} . Then correct tool nose radius compensation is performed for the blocks following P_C .



Manual operation after single block stop

Manual operation was performed when execution of a block was terminated by single block stop.

Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub-sequent processing is the same as case a described above. An MDI operation can also be 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 the machine by reading a program from the I/O unit

- **Search for a sequence number**

Function for starting operation from a sequence number in the middle of a program

- **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

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.




When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.

When the reset switch on the CRT/MDI panel is pressed, automatic operation terminates and the reset state is entered.

For the 0-TTC, the programs for the two tool posts can be executed simultaneously so the two tool posts can operate independently at the same time.

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Procedure for Memory Operation

- 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  soft key.

For the 0-TTC, select the program for the tool post to be operated. When operating the two tool posts at the same time, select a program for each tool post.
- 3 For the 0-TTC, select the tool post to be operated with the tool post selection switch on the machine operator's panel.
- 4 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.
- 5 To stop or cancel memory operation midway through, follow the steps below.
 - a. Stopping memory operation

Press the feed hold switch on the machine operator's panel. The feed hold 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) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.

When the cycle start switch on the machine operator's panel is pressed while the feed hold lamp is on, machine operation restarts.

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.

Explanation

Memory operation

After memory operation is started, the following are executed:

- ① A one-block command is read from the specified program.
- ② The block command is decoded.
- ③ The command execution is started.
- ④ The command in the next block is read.
- ⑤ Buffering is executed. That is, the command is decoded to allow immediate execution.
- ⑥ Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.
- ⑦ Hereafter, memory operation can be executed by repeating the steps ④ to ⑥.

Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

- The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
- There are two keys to stop memory operation: The feed hold key and reset key.

● Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

● Optional stop (M01)


Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

● Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered. In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

- **Feed hold**

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.
- **Reset**

Automatic operation can be stopped and the system can be made to the reset state by using  key on the 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.
- **Cycle start for the 0-TTC**

For the 0-TTC, a cycle start switch is provided for each tool post. This allows the operator to activate a single tool posts to operate them at the same time in memory operation or MDI operation. In general, select the tool post to be operated with the tool post selection switch on the machine operator's panel and then press the cycle start button to activate the selected tool post. (The procedure may vary with the machine tool builder. Refer to the appropriate manual issued by the machine tool builder.)
- **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 Z200.5;

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	O0001 N0020
(MDI)	(MODAL)
	G00 F 0.3000
	G97 M 003
	G69 S 00550
	G99 T 0101
	G21
	G40 WX 0.000
	G25 WZ 0.000
	G22 SRPM550
	G54 SSPM0
	SMAX 32767
	SACT 0
ADRS.	S 0 T0101
16:49:30	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 "Z 200.5" by address/numeric key.
- 7 Press  key.

The data, Z200.5 is input and displayed. If you pressed wrong number keys, correct the operation following the instruction described above.





PROGRAM		O0001 N0020
(MDI)		(MODAL)
.X 10.500		G00 F 0.3000
.Y 200.500		G97 M 003
		G69 S 00550
		G99 T 0101
		G21
		G40 WX 0.000
		G25 WZ 0.000
		G22 SRPM550
		G54 SSPM0
		SMAX 32767
		SACT 0
ADRS.		S 0 T0101
16:51:13		MDI
[PRGRM]	[CURRNT]	[NEXT]
		[MDI]
		[RSTR]

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 X10.5 Z200.5 to X10.5, cancel Z200.5, by following the steps described below:

- 1 Press    keys.
- 2 Press the  or the cycle start button on the machine operator's panel.


NOTE

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.
- The input block is cleared when the MDI operation is completed or when a reset is specified.

Procedure for MDI Operation – B

- 1 Press the **MDI** mode selection switch.
For the 0-TTC, select the tool post for which a program is to be created with the tool post selection switch. Create a separate program for each tool post.
- 2 Press the  function key on the CRT/MDI panel to select the program screen. The following screen appears:




```

PROGRAM                                O0001 N0020
Q0000
%

(MODAL)
G01 G69 G21 G25
G97 G99 G40 G22
F 0.1500 S00700
M003 T0101
< S 0 T0101
16:54:29 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 0057 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. (For the 0-TTC, select the tool post to be operated with the tool post selection switch on the machine operator's panel beforehand.) 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)                                O0001 N0001
Q0000 G00 X10. Z200. ;
N10 M03 ;
N20 G01 Z120. F500 ;
N30 M93 P9010 ;
N40 G00 Z0 ;
%

(MODAL)
G01 G69 G21 G25
G97 G99 G40 G22
F 0.1500 S00700
M003 T0101
< S 0 T0101
16:54:29 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.

The machine responds as follows:

- (i) When the machine was moving, feed operation decelerates and stops.
- (ii) When dwell was being performed, dwell is stopped.
- (iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.

When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b. Terminating MDI operation

Press the  key 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.

Explanation

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program, M99 performs this function.

• Erasing the program

Programs prepared in the **MDI** mode will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
- 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.

Restrictions

- **Program registration**
- **Number of lines in a program**
- **Subprogram nesting**

Programs created in MDI mode cannot be registered.

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.0028 #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).

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).

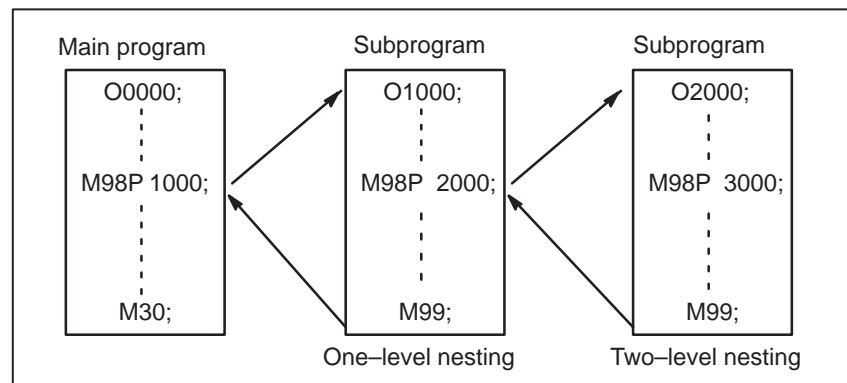


Fig. 4.2 (a) Nesting Level of Subprograms Called from the MDI Program

- **Macro call**
- **Memory area**

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.

When a program is created in the **MDI** mode, an empty area in program memory is used. If program memory is full, no programs can be created in the **MDI** mode.

4.3 DNC OPERATION

DNC operation enables machine operation by reading a program directly from the connected I/O unit. The program is not registered in CNC memory. This method is useful when a program is too large to be registered in CNC memory. This operation is also used for high-speed machining with a remote buffer. For the DNC operation with the Floppy Cassette directory display function (the Floppy Cassette directory display function is incorporated and the Floppy Cassette is specified as the I/O unit), see Chapter 9.

Operating procedure for DNC operation

- 1 Select the MDI mode and set the channel of the connected I/O unit in the I/O field on the setting screen.
- 2 Select the **AUTO** mode.
- 3 Call the beginning of the program in the I/O unit.
- 4 Input the DNCI signal. (For the actual operation, refer to the manual of the machine tool builder.)
- 5 Press the **CYCLE START** button.

DNC operation is started. Operation can be started and resumed in the same manner described for memory operation.

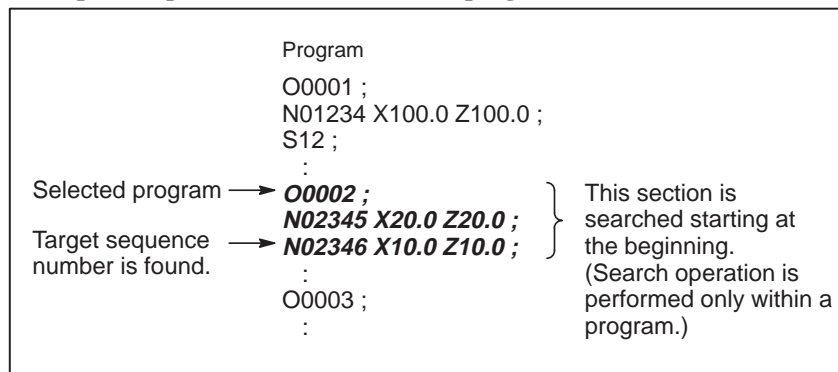
Explanations

- The program running under DNC operation can call a subprogram from memory.
- The program running under DNC operation can specify a custom macro. However, repeat and branch instructions cannot be programmed.
- The program running under DNC operation can call a macro program from memory.
- In DNC operation, no sequence number can be specified in the command for returning control from the called subprogram or macro program to the calling program (M99P****).
- In DNC operation with I/O channel 1 or 2, the DC3 code is output and reading is stopped at the end of each block (each time EOB is read). To read blocks continuously, specify bit 7 of parameter 0390 accordingly.
- To enable a program to be resumed in DNC operation, set bit 7 of parameter 0387 accordingly.
- During DNC operation, no program can be displayed; only the current block and next block can be displayed.
- During DNC operation, F is displayed as the address of the program number at the top right corner of the CRT screen.
- With the Series 0-TTC, DNC operation cannot be simultaneously executed on both tool posts.




4.4 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example) Sequence number 02346 in a program (O0002) is searched for.



Procedure for sequence number search

- 1 Select **AUTO** mode.
- 2 Press .
- 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, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC states at that point.

- **Checking during search**

During search operation, the following checks are made:

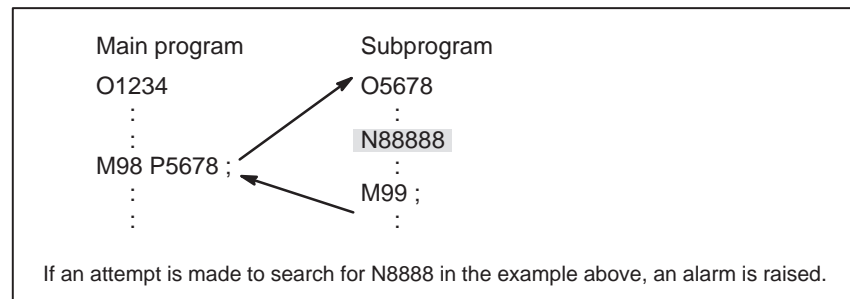
- Optional block skip
- P/S alarm (No. 003 to 010)

Limitations

To ignore a P/S alarm (Nos. 003 to 010) during a search for a sequence number, set bit 1 of parameter 0051 accordingly.

- **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

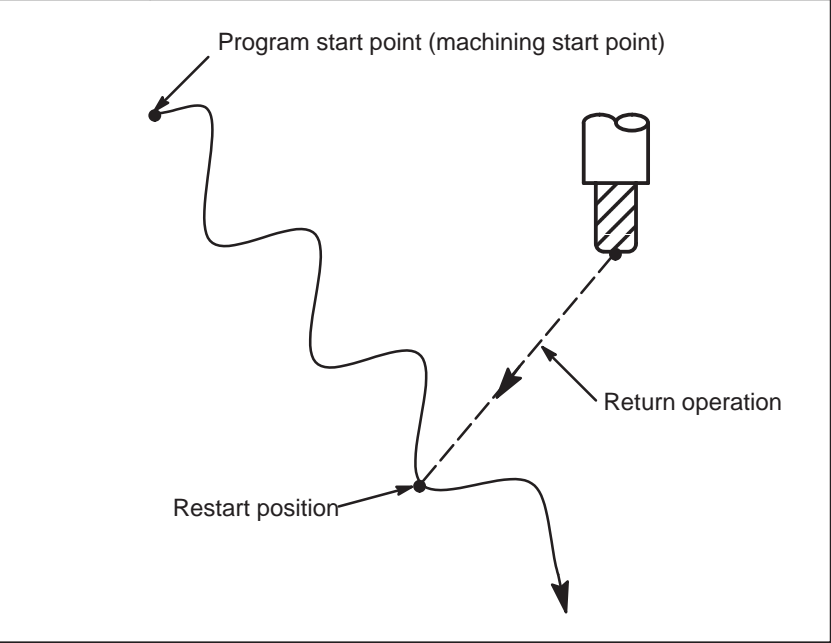
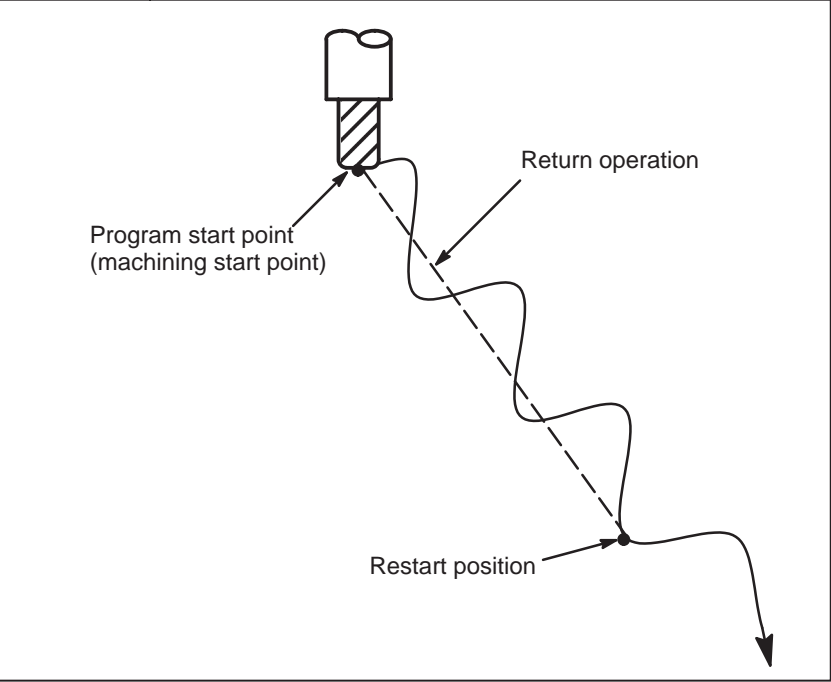
Number	Contents
60	Command sequence number was not found in the sequence number search.

4.5

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) at a 'Restart position'. A dashed line labeled 'Return operation' points from the restart position back to the 'Program start point (machining start point)'. A solid line with an arrow shows the path from the restart position, following the original program path, and then continuing from the start point.</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 at a 'Restart position'. A dashed line labeled 'Return operation' points from the restart position back to the 'Program start point (machining start point)'. A solid line with an arrow shows the path from the restart position, following the original program path, and then continuing from the start point.</p>	

Procedure for Program restart operation

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.

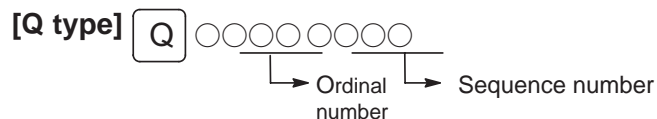
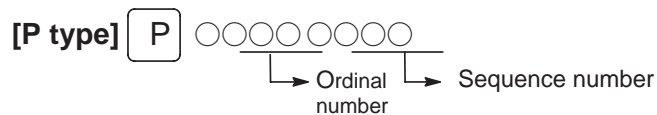
4 [P type]

Input address  and the sequence number of the block to be resumed in that order. Then, press the  cursor key.

[Q type]

Input address  and the sequence number of the block to be resumed in that order. Then, press the  cursor key.

If two or more identical sequence numbers are used, specify the sequence number of the block from which the program is resumed. To do this, specify the position of that number prior to the sequence number.



- 5 The sequence number is searched for, and the program restart screen appears on the CRT display.

```

PROGRAM RESTART                                O0001 N0013

  (DESTINATION)    M008 045 *** *** ***
    X   160.000    *** *** *** *** ***
    Z   180.000    *** *** *** *** ***
    C    0.000    *** *** *** *** ***
    Y    0.000    *** *** *** *** ***
                    *** *** *** *** ***
  (DISTANCE TO GO) *** *** *** *** ***
4 X   -40.000
3 Z   -40.000    T   0101 ****
2 C    0.000    S   00550
1 Y    0.000

                                  S   0 T0101
17:05:48                        BUF AUTO  RSTR
[ PRGRM ][CURRNT ][ NEXT  ][ CHECK ][ 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.0124 to 0127)) along which the tool moves to the restart position.
 M: Tirty-fifth most recently specified M codes
 T: Two most recently specified T codes
 S: Most recently specified S 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, and T codes to be executed. If they are found, enter the **MDI** mode, then execute the M, S and T functions. 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.

Number	Contents
094	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.)
095	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 cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.)
097	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P/S alarm 94 to 97 were reset, no automatic operation was performed.)
098	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	After completion of search in program restart, a move command is given with MDI.

WARNING

As a rule, the tool cannot be returned to a correct position under the following conditions.

Special care must be taken in the following cases since none of them cause an alarm:

- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked.
- When the mirror image is used.
- When manual operation is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
- When program restart specified for an intermediate block for a multiple repetitive canned cycle

4.6 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 when the function for displaying the directory of the Floppy Cassette is incorporated and Floppy Cassette is set as the I/O 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 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		O0001 N0013
CURRENT SELECTED:SCHEDULE		
NO.	FILE NAME	(METER) VOL
0000	SCHEDULE	
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
		S 0 T0101
17:08:08		AUTO
[SELECT]	[]	[] [SCHEDUL]

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 [EXEC] soft keys. The file for the entered file number is selected, and the file name is indicated after "CURRENT SELECTED:".

FILE DIRECTORY		O0001 N0013
CURRENT SELECTED:O0001		
NO.	FILE NAME	(METER) VOL
0000	SCHEDULE	
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
		S 0 T0101
17:09:45		AUTO
[SELECT]	[]	[] [SCHEDUL]

Screen No.2

- 4 Input DNC signal, then press the cycle start switch. The selected file is executed. For details on the DNC signal, 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		O0003 N0013	
CURRENT SELECTED:O0001			
17:12:19		S 0 T0101	
[SELECT][] [AUTO] [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 keys to display "SELECT FILE NO."
- 3 Enter file number 0, and press the **[EXEC]** soft keys. "SCHEDULE" is indicated after "CURRENT SELECTED:".
- 4 Press the **[SCHEDUL]** soft key. Screen No. 4 appears.

FILE DIRECTORY		O0001 N0013	
ORDER	FILE NO.	REQ.REP	CUR.REP
01			
02			
03			
04			
05			
06			
07			
08			
09			
10			
NUM.		S 0 T0101	
17:15:43		AUTO	
[][][CLEAR][][DIR]

Screen No.4

- 5 Input the DNC signal, 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		O0001 N0013	
ORDER	FILE NO.	REQ.REP	CUR.REP
01	_0003	5	0
02	0007	12	0
03	0010	LOOP	0
04			
05			
06			
07			
08			
09			
10			
NUM.		S	0 T0101
17:14:42		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 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 soft key of left side (return menu key) is pressed on screen No. 1, 2, 3, 4, or 5, the program screen is displayed.

Restrictions

- **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.

- **Scheduling function for the 0-TTC**

The scheduling function can be used only for a single tool post.

Alarm

Alarm No.	Description
086	An attempt was made to execute a file that was not registered in the floppy disk.
210	M198 and M099 were executed during scheduled operation, or M198 was executed during DNC operation.

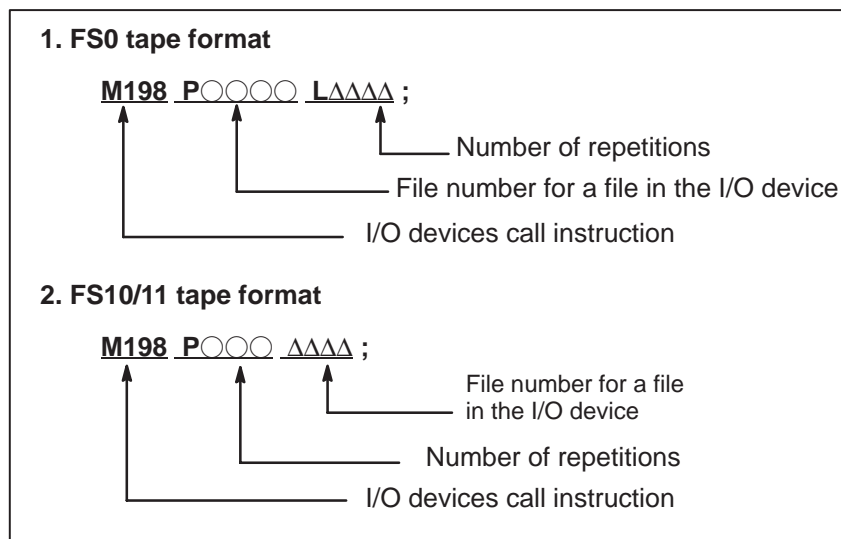
4.7 SUBPROGRAM CALL FUNCTION

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 optional function of directory display for floppy cassette is need.

Format



Explanation

The subprogram call function is enabled when the input/output device is set to FLOPPY CASSETTE, PROGRAM File Mate. When the custom macro 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.0248. In this case, M198 is executed as a normal M code. The file number is specified at address P. Depend on the bit (bit 2) of parameter No.0063, a program number can be specified. When a file number is specified at address P, Fxxxx is indicated instead of Oxxxx at upper right of CRT screen.

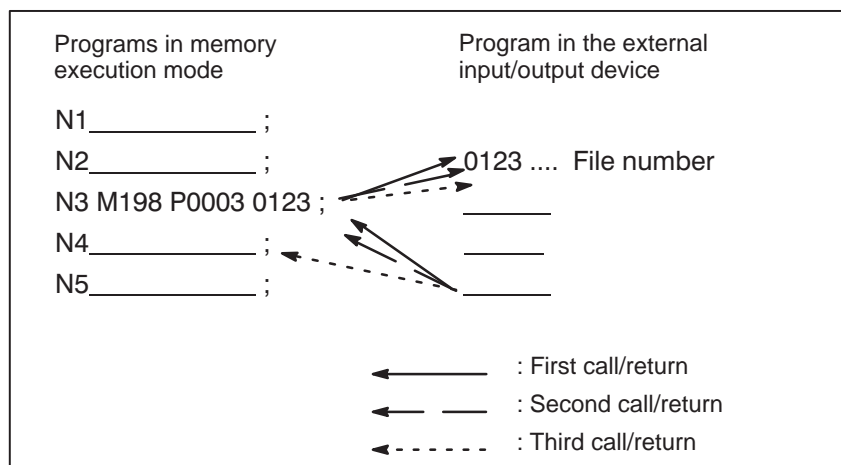


Fig. 4.7 Program Flow When M198 is Specified

Restrictions

For the 0-TTC, subprograms in a floppy cassette cannot be called for the two tool posts at the same time.

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. When the reset operation is done in the MDI mode after M198 is commanded in the AUTO mode, it does not influence on the memory operation and the operation is continued by restarting it in the AUTO mode.

4.8 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.

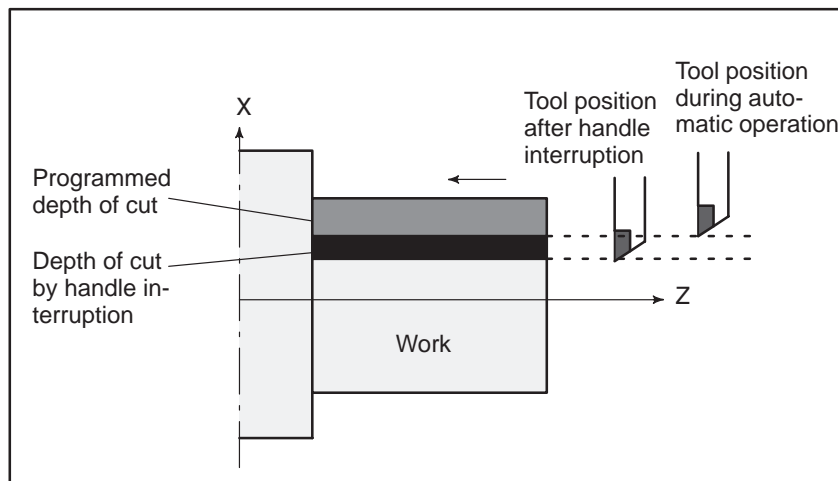


Fig. 4.8 Manual Handle Interruption

·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).

Since this movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.

NOTE

The travel distance caused by a handle interrupt has the same units as for output.

To change this distance to input units, set bit 2 of parameter 0052 accordingly. In this case, acceleration/deceleration is also performed for travel caused by a 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		O0001 N0013	
(INPUT UNIT)		(OUTPUT UNIT)	
X	0.228	X	0.228
Z	-0.177Z	Z	-0.177
C	0.000	C	0.000
Y	0.102	Y	0.102
(RELATIVE)		(DISTANCE TO GO)	
U	220.000	X	0.000
W	220.000	Z	0.000
H	0.000	C	0.000
V	0.000	Y	0.000
PART COUNT 1796			
RUN TIME	3H12M	CYCLE TIME	0H 0M1S
ACT.F	0 MM/M	S	0T0101
17:21:01		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 manual reference
position return or the first reference position return by G28 when power
is turned on ends every axis.

4.9 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.

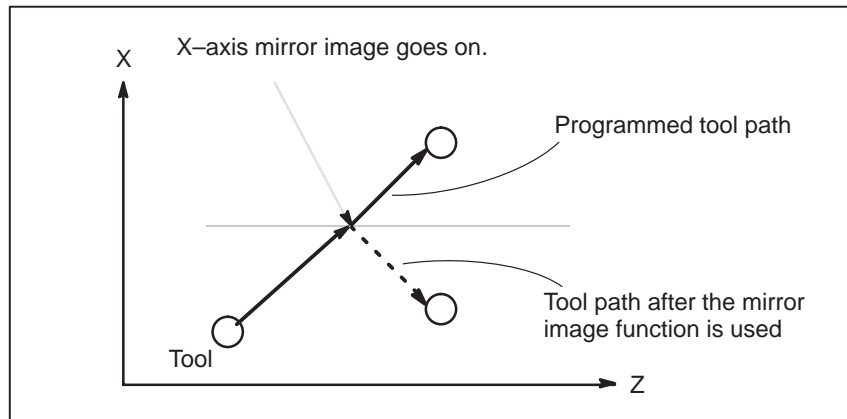


Fig. 4.9 Mirror Image

Procedure

For operation procedure, refer to the manual supplied by the machine tool builder.

Restrictions

The mirror image function is not effective in the of movement during manual operation, the movement from an intermediate point to the reference position during automatic reference position return.

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.

In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

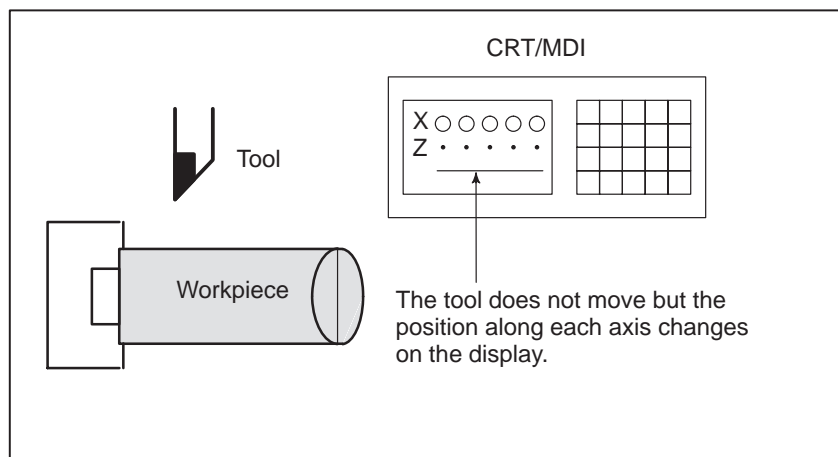


Fig. 5.1 Machine lock

Procedure for Machine Lock and Auxiliary Function Lock Operation

- **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.

After an automatic operation is executed with the machine lock function, the relationship in position between workpiece coordinates and machine coordinates prior to the automatic operation may be changed. If the relationship is changed, reset the workpiece coordinate system by specifying the command for setting the coordinate system or by performing a manual reference position return. Refer to the appropriate manual provided by the machine tool builder for machine lock.

- **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.

Restrictions

- **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 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.

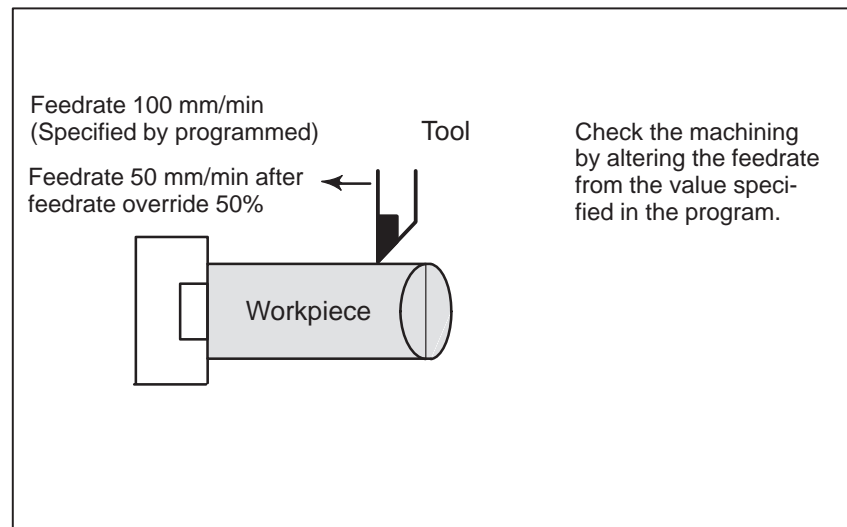
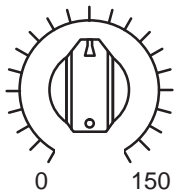


Fig. 5.2 Feedrate override

Procedure for Feedrate Override Operation



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.

Restrictions

- **Override Range**
- **Override during thread**

The override that can be specified ranges from 0 to 150% (10% step). 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. 0533).

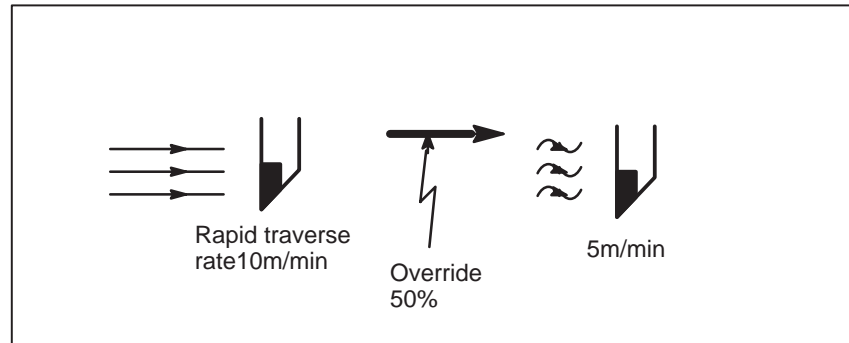
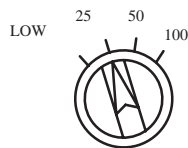


Fig. 5.3 Rapid traverse override

Procedure for Rapid Traverse Override Operation



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.

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

- 1) Rapid traverse by G00.
- 2) Rapid traverse during a canned cycle.
- 3) Rapid traverse in G27, G28 and G30.
- 4) Manual rapid traverse.
- 5) Rapid traverse of manual reference position return

5.4 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

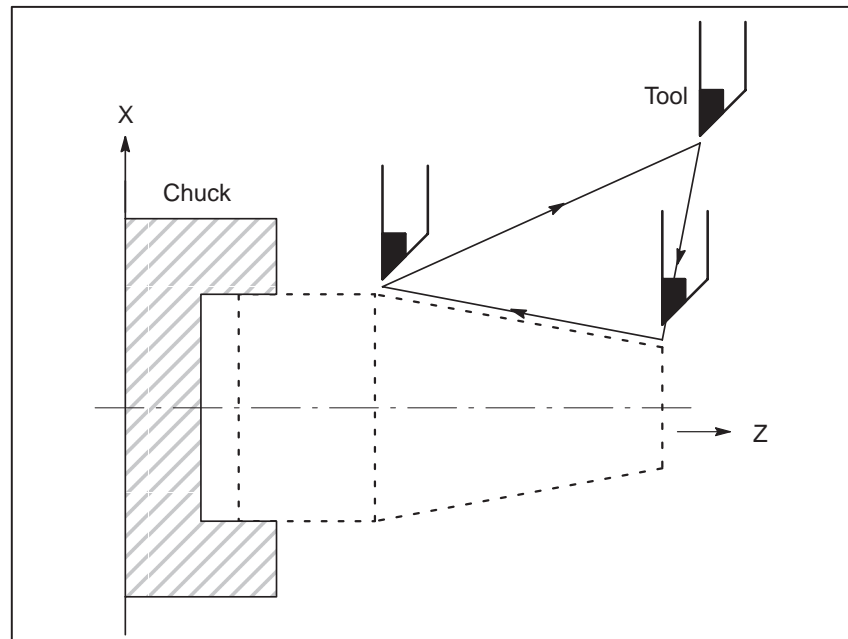


Fig. 5.4 Dry run

Procedure for Dry Run Operation

Press the dry run switch on the machine operator's panel during automatic operation.

The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate.

Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

• Dry run feedrate



The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	Rapid traverse rate	Max. cutting feedrate
OFF	Jog feed rate or rapid traverse rate *1)	Jog feed rate

*1: Job feed rate when parameter (bit 6 of No. 0001) is 1.
Rapid traverse rate when 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.

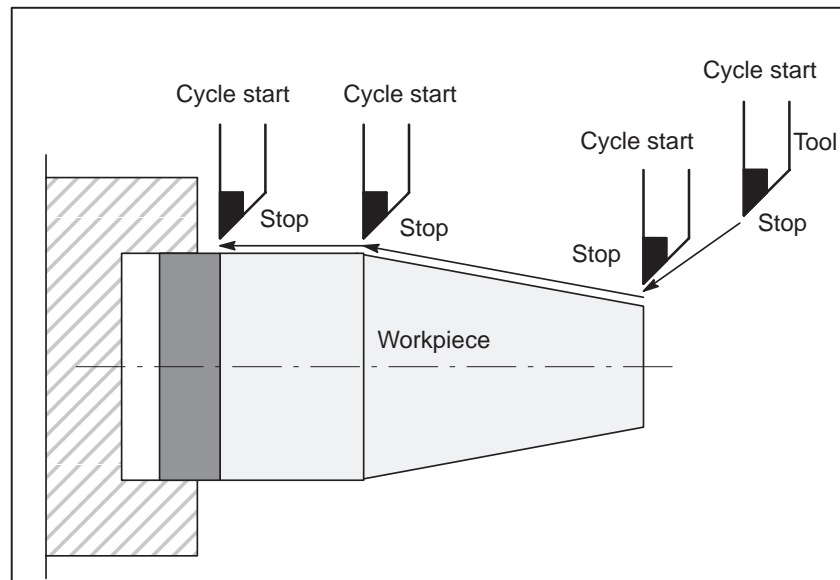


Fig. 5.5 Single block

Procedure for Single Block Operation

- 1 Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed.
- 2 Press the cycle start button to execute the next block. The tool stops after the block is executed.

Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

- **Reference position return and single block**

If G28 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 as follows.

		<div> <div>— — ➔</div> <div>— ➔</div> </div> <div> <div>Rapid traverse</div> <div>Cutting feed</div> </div>	
	S : Single block		
	Tool path	Explanation	
☆G90 (Outer/inner turning cycle)	Straight cutting cycle 	Taper cutting cycle 	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G92 (Threading cycle)	Straight threading cycle 	Taper threading cycle 	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G94 (End surface turning cycle)	Straight end surface cutting cycle 	Taper end surface cutting cycle 	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G70 (Finishing cycle)		Tool path 1 to 7 is assumed as one cycle. After 7 is finished, a stop is made.	
☆G71 (Outer surface rough machining cycle) G72 (End surface rough machining cycle)	<p>This figure shows the case for G71. G72 is the same.</p>	Each tool path 1 to 4, 5 to 8, 9 to 12, 13 to 16 and 17 to 20 is assumed as one cycle. After each cycle is finished, a stop is made.	

Fig. 5.5 (a) Single block during canned cycle (1/2)

— — —> Rapid traverse

S : Single-block stop

————> Cutting feed

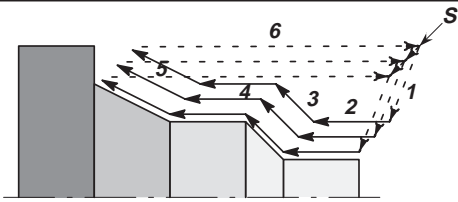
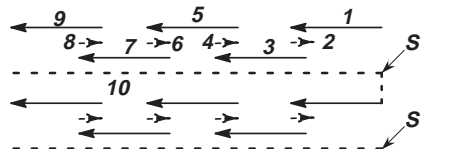
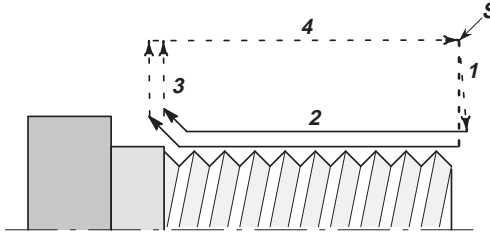
	Tool path	Explanation
☆G73 (Closed-loop cutting cycle)		Tool path 1 to 6 is assumed as one cycle. After 10 is finished, a stop is made.
☆G74 (End surface cutting-off cycle) G75 (Outer/inner surface cutting-off cycle)	 <p>This figure shows the case for G74. G75 is the same.</p>	Tool path 1 to 10 is assumed as one cycle. After 10 is finished, a stop is made.
☆G76 (Multiple repetitive threading cycle)		Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.

Fig. 5.5 (b) Single block during canned cycle (2/2)

● Subprogram call and single block

Single block stop is not performed in a block containing M98P_.; or G65. However, single block stop is even performed in a block with M98P_ 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.

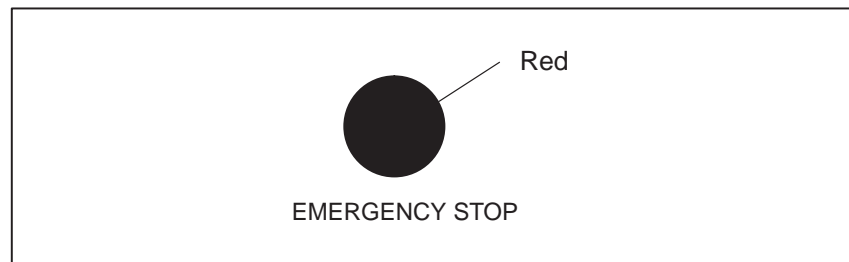


Fig. 6.1 Emergency stop

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.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the Z axis direction tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

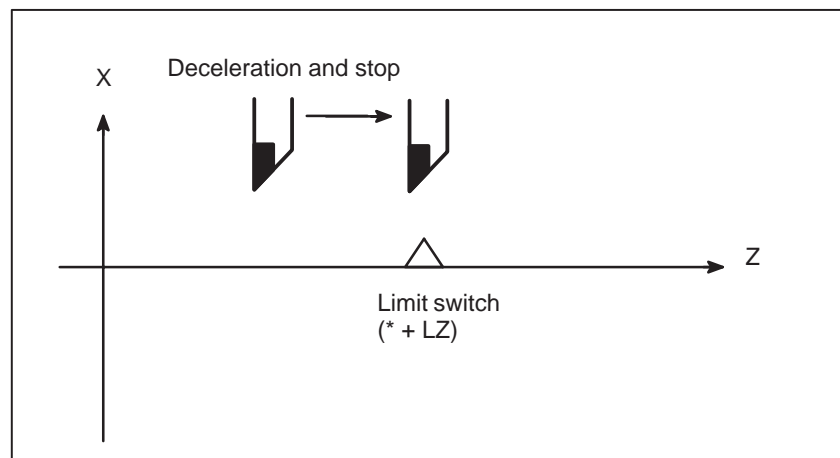


Fig. 6.2 Overtravel

Explanation

- **Releasing overtravel**
- **Alarm**

Press the reset button to reset the alarm after moving the tool to the Z axis minus direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

No.	Message	Description
520	Overtravel: +Z	The tool has exceeded the hardware-specified overtravel limit along the positive zth axis.

The overtravel limit signal (*+LZ) can be enabled or disabled, depending on bit 2 of parameter 0015. For details, refer to the manual of the machine tool builder.

6.3 STROKE CHECK

There areas which the tool cannot enter can be specified with stored stroke check 1/2, stored stroke check 3, and stored stroke check 4.

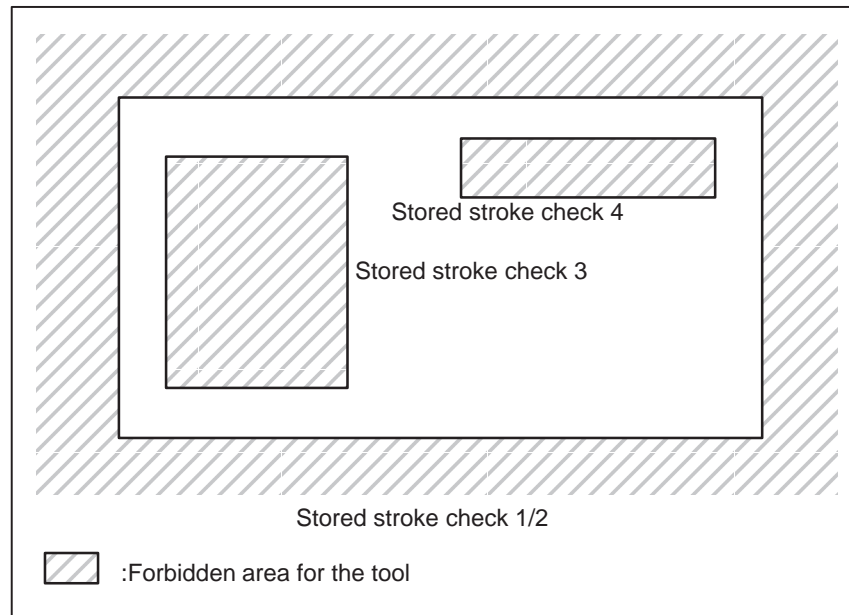


Fig. 6.3 (a) Stroke check

When the tool exceeds a stored stroke check, an alarm is displayed and the tool is decelerated and stopped.

When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

- **Stored stroke check 1/2**
- **Stored stroke check 3
(Extended stored stroke check)**

Parameters (Nos.0700 to 0707 or Nos.0770 to 0777) 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. The parameter used depends on the signal from the machine.

Parameters (Nos.0747 to 0754) 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:

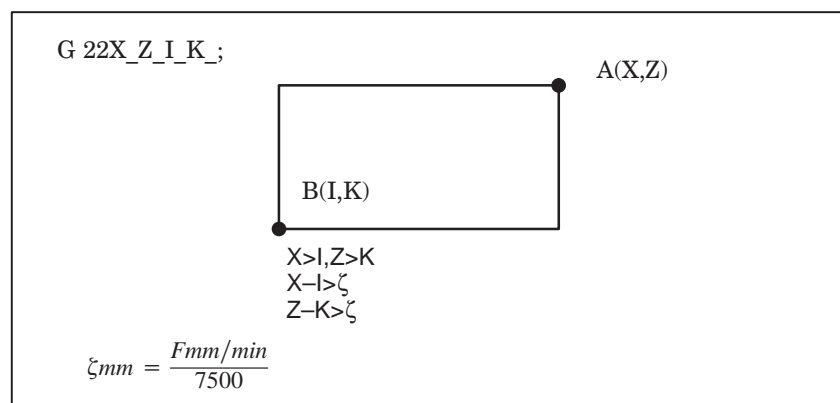


Fig. 6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

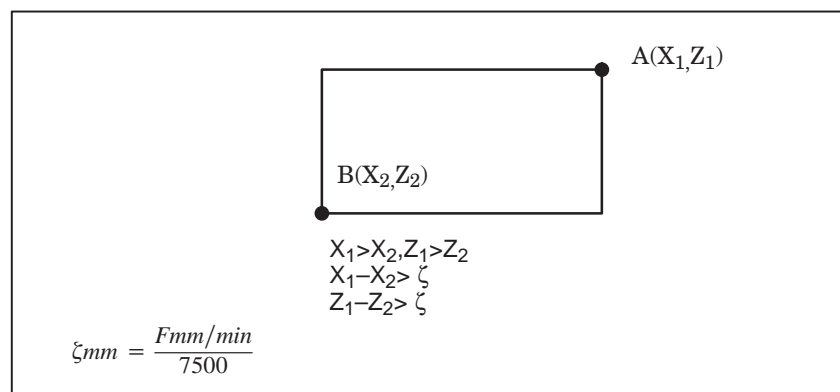


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

In check 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 $X_1, Z_1, X_2,$ and Z_2 through parameters (Nos.0747 to 0754), the data should be specified by the distance from the reference position in the least command increment. (Output increment)

If set the forbidden area XZIK 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.

When power is on, it becomes G22 mode.

• Stored stroke check 4

Set the boundary with parameters No.0760 to 0767. The area inside the boundary becomes the forbidden area.

Option of extended stored stroke check is required.

- **Checkpoint for the forbidden area**

The parameter setting or programmed value (XZIK) depends on which part of the tool or tool holder is checked for entering the forbidden area. Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.

If point C (The top of the tool) is checked in Fig. 6.3 (d), the distance “c” should be set as the data for the stored stroke limit function. If point D (The tool chuck) is checked, the distance “d” must be set.

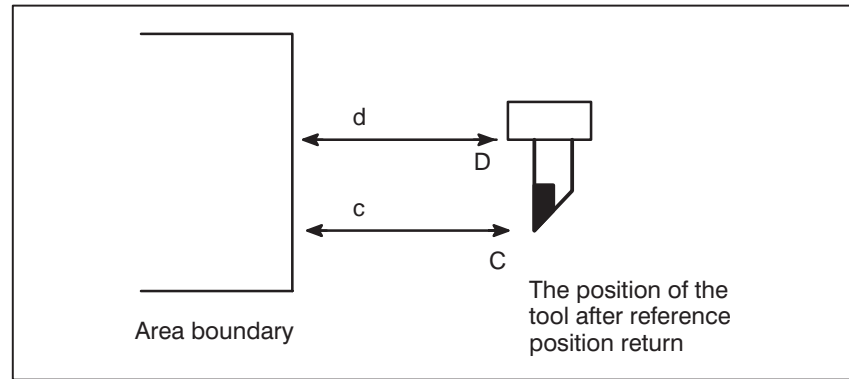


Fig. 6.3 (d) Setting the forbidden area

- **Forbidden area over-lapping**

Area can be set in piles.

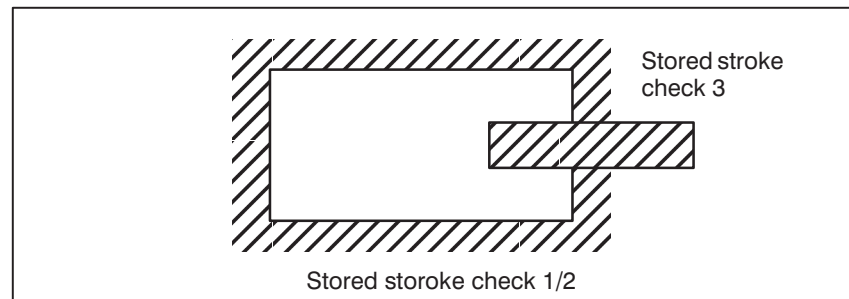


Fig. 6.3 (e) Setting the forbidden area over lapping

- **Amount of overrun relative to stored stroke check**

Unnecessary limits should be set beyond the machine stroke.

The maximum amount of overrun L (mm), relative to the stored stroke check, is calculated using the following formula, where F (mm/min) is the maximum rapid traverse rate:

$$L = F/7500$$

When L is set, the tool can enter a previously set inhibited area by up to L mm. You can also set bit 7 of parameter No. 0076 to stop the tool up to L mm outside the set inhibited area. In the latter case, the tool does not enter the inhibited area.

- **Effective time for a forbidden area**

Each limit becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed.

After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke check 3).

- **Releasing the alarms**

When a stroke check alarm is issued, manually move the tool out of the inhibited area, in the direction opposite to that indicated in the alarm message. Then, press the reset key to release the alarm. If the tool has entered two inhibited areas at the same time, such that it cannot be manually moved out of either area, apply emergency stop, change the stroke check parameters to release the alarm, then manually move the tool out of the inhibited areas.

- **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.

- **Setting the forbidden area for the 0-TTC**

For the 0-TTC, set a forbidden area for each tool post.

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 check 1/2, all areas are forbidden areas.
2. When the forbidden area is check 3 or check 4 all areas are movable areas.

Alarms

Number	Message	Contents
5n0	OVER TRAVEL: +n	Exceeded the n-th axis) + side stored stroke check 1/2.
5n1	OVER TRAVEL: -n	Exceeded the n-th axis - side stored stroke check 1/2.
5n2	OVER TRAVEL: +n	Exceeded the n-th axis (1 to 8) + side stored stroke check 2.
5n3	OVER TRAVEL: -n	Exceeded the n-th axis (1 to 8) - side stored stroke check 2.
5n4	OVER TRAVEL: +n	Exceeded the n-th axis (1 to 8) + side stored stroke check 3.
5n5	OVER TRAVEL: -n	Exceeded the n-th axis (1 to 8) - side stored stroke check 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		O0001 N0011	
511 OVER TRAVEL : -X			
18:46:03 ALARM		S 0 T0101	
MDI			
[ALARM]	[OPR]	[MSG]	[]


- Another method for alarm displays

In some cases, the alarm screen does not appear, but an ALM blinks at the bottom of the screen.

PARAMETER		O0001 N0013	
(SETTING 1)			
TVON= 0			
ISO = 1 (0:EIA 1:ISO)			
INCH= 0 (0:MM 1:INCH)			
I/O = 0			
SEQ = 1			
NO. TVON		CLOCK 94/03/17	
18:46:36 ARARM		18:46:45	
MDI		S 0 T0101	
[PARAM]	[DGNOS]	[SV-PRM]	[]

In this case, display the alarm screen as follows:

Procedure for displaying the alarm screen

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 998: 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 number of control axis.




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 Diagnostic

- 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  key.
 - Key input the number of the diagnostic data to be displayed.
 - Press  key.

DIAGNOSTIC		O0001 N0011	
NO.	DATA	NO.	DATA
0000	00000001	0010	00000000
0001	00000000	0011	00000000
0002	00000000	0012	00000000
0003	00000000	0013	00000000
0004	10000000	0014	00000000
0005	00000000	0015	00000000
0006	00000000	0016	00100010
0007	00000000	0017	00100000
0008	00000000	0018	10100000
0009	00000000	0019	00000000
NO. 0000			
18:48:34			
AUTO			
[PARAM][DGNOS][SV-PRM][]			

For the 0-TTC, the diagnostic screen for the tool post selected with the tool post selection switch is displayed. When displaying the diagnostic screen for the other tool post, specify the tool post with the tool post selection switch.

Data of self-Diagnosis

	#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

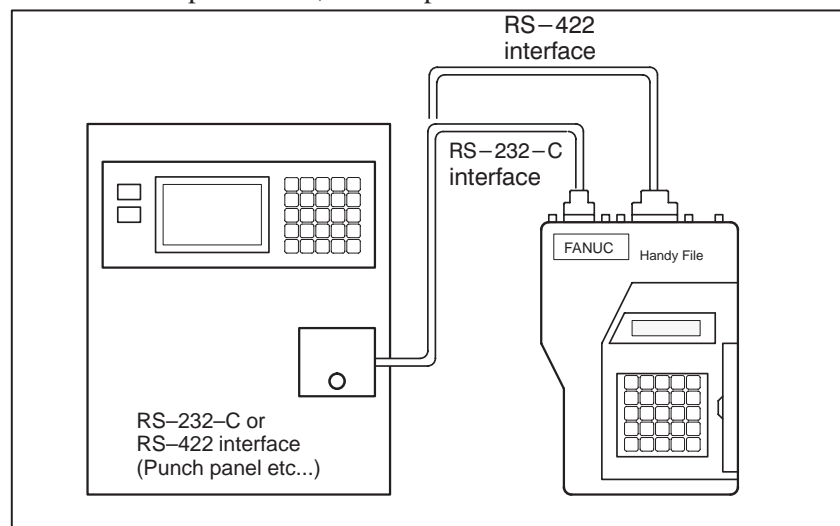
NC data is transferred between the NC and external input/output devices such as the Handy File.

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.

For how to set parameters, see Chapter 2 **OPERATIONAL DEVICES**.



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.

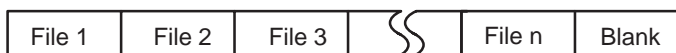
Input/output is possible with data extending over more than one floppy disk.

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.

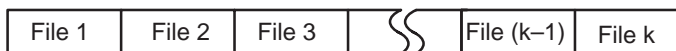


• Request for floppy replacement

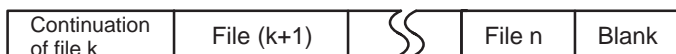
When one file has been entered over two floppies, LEDs on the adaptor flash alternately on completion of data input/output between the first floppy and the CNC, prompting floppy replacement. In this case, take the first floppy out of the adaptor and insert a second floppy in its place. Then, data input/output 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



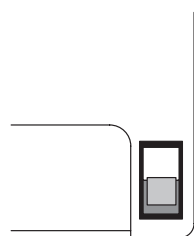
Since floppy replacement is processed by the input/output device, no special operation is required. The CNC will interrupt data input/output operation until the next floppy is inserted into the adaptor.

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.

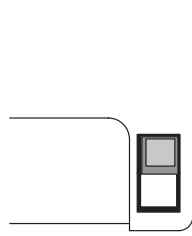
• Protect switch

The floppy is provided with the write protect switch. Set the switch to the write enable state. Then, start output operation.

Write protect switch of a cassette

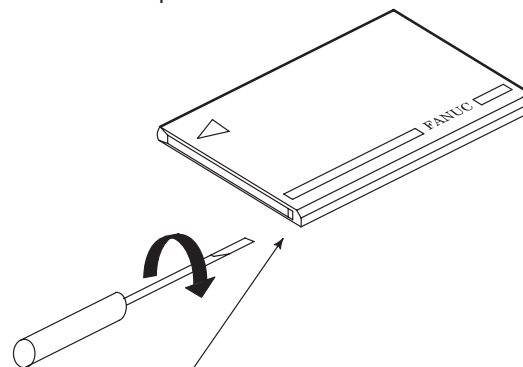


(1) Write-protected
(Only reading is possible.)

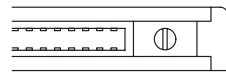


(2) Write-enabled (Reading, writing, and deletion are possible.)

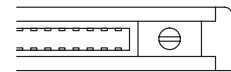
Write protect switch of a card



Write protect switch



(1) Write-protected
(Only reading is possible.)



(2) Write-enabled (Reading, writing, and deletion are possible.)

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.

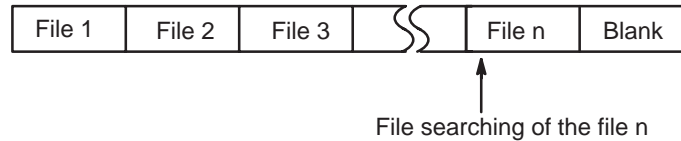
(Entry example on MEMO)

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 Operation

- 1 Press the EDIT or **AUTO** switch on the machine operator's panel.
- 2 Press function key  .
- 3 Enter address  .
- 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  .

Explanation

- **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	The ready signal (DR) of an input/output device is off. 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). 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.)

8.3 FILE DELETION

Files stored on a floppy can be deleted file by file as required.

Procedure for File Deletion

- 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 key . Display program screen.
- 4 Enter address .
- 5 Enter the number (from 1 to 9999) of the file to delete.
- 6 Press soft key .

The file specified in step 5 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

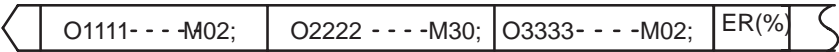
Inputting a Program

This section describes how to load a program into the CNC from a floppy or NC tape.

Procedure for Inputting a Program

- 1
- Make sure the input device is ready for reading.
For the 0-TTC, select the tool post for which a program to be input is used with the tool post selection switch.
- 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 
- 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 stop input operation halfway, 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 I/O device.

Procedure for program input 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 **[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                                00615 N0000
  SYSTEM EDITION 0671 - 04
PROGRAM NO.  USED :    24 FREE : 39
MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
00001 00010 00011 00021 00041 00601
00613 00615 00645 00651 01021 01041
01051 02011 02505 03148 03153 03511
04011 04048 05111 05221 05766 06032

<
                                EDIT
[ PUNCH ][ READ ][ CAN ][          ][ STOP ]
```

- **Program input in background edit mode**

Program input is identical to that in foreground edit mode. If the  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

- 1 Make sure the output device is ready for output.
For the 0-TTC, select the tool post for which a program to be output is used with the tool post selection switch.
- 2 To output to an NC tape, specify the punch code system (ISO or EIA) using a parameter. To output to floppy, specify ISO.
- 3 Press the EDIT switch on the machine operator's panel.
- 4 Press function key  .
- 5 Enter address  .
- 6 Enter a program number. If -9999 is entered, all programs stored in memory are output.
To output multiple programs at one time with full MDI key, 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
- An alarm while a program is output
- Outputting a program after file heading
- Efficient use of memory
- On the memo record

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.

When P/S alarm 86 occurs during program output, the floppy is restored to the condition before the output.

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.

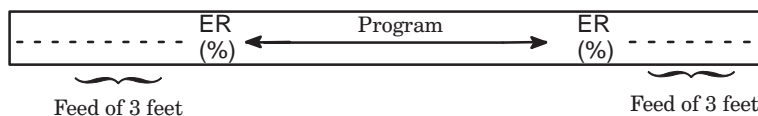
To efficiently use the memory in the cassette or card, output the program by setting parameter (No.0002#7, 0012#7, 0050#7, or 0051#7) to 1. This parameter makes the feed is not output, utilizing the memory efficiently.

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.

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 CAN key during feed punching to cancel the subsequent feed punching.

A space code for TV check is automatically punched.

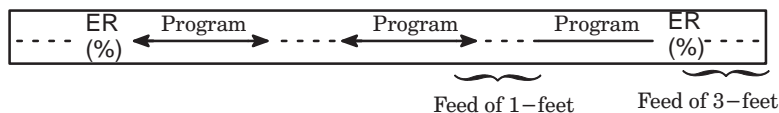
When a program is punched in ISO code, two CR codes are punched after an LF code.

----- LF CR CR

To set LF code only, set parameter bit 7 of No.0070.

Press the RESET key to stop punch operation.

All programs are output to paper tape in the following format.



The sequence of the programs punched is undefined.

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 I/O device.

- TV check
- ISO code
- Stopping the punch
- Punching all programs
- Output with the soft keys

Procedure for program output with the soft keys

- 1 Display the program screen in EDIT mode.
- 2 Press the **[I/O]** soft key.
- 3 Input address **[O]**, 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:
O△△△△, O□□□□
Programs No. △△△△ 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                                00615 N0000
  SYSTEM EDITION 0671 - 04
  PROGRAM NO.   USED :    24 FREE : 39
  MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
00001 00010 00011 00021 00041 00601
00613 00615 00645 00651 01021 01041
01051 02011 02505 03148 03153 03511
04011 04048 05111 05221 05766 06032

<
                                EDIT
[ PUNCH ][ READ ][ CAN ][          ][ STOP ]

```

- **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 **[OUTPT START]** 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.)

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

- 1 Make sure the input device is ready for reading
For the 0-TTC, select the tool post for which offset data to be input is used with the tool post selection switch.
- 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 offset screen.
- 5 Press soft key .
- 6 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

- 1 Make sure the output device is ready for output.
For the 0-TTC, select the tool post for which offset data to be output is used with the tool post selection switch.
- 2 Specify the punch code system (ISO or EIA) using a parameter.
To output to floppy, specify ISO.
- 3 Press the EDIT switch on the machine operator's panel.
- 4 Press function key , and display offset screen.
- 5 Press soft key .

Offset data is output in the output format described below.

Explanations

• Output format

Output format is as follows:

Format

G10P_X_Y_Z_R_Q;

P: Offset number

:P=0 Work sheet

:P=Wear offset number For wear offset amount

:p=10000+geometry offset number .. For geometry offset amount

X:Offset value on X axis

Y: Offset value on Y axis

Z:Offset value on Z axis

Q:Imaginary tool nose number

R:Tool nose radius offset value

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

Pitch error compensation data is part of the parameter data. The same input/output operation as for other parameters can be used for pitch error compensation data. This section describes the method of parameter input/output operation.

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.

Procedure for Inputting Parameters

- 1 Make sure the input device is ready for reading.
For the 0-TTC, select the tool post for which parameters to be input are used with the tool post selection switch.
- 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 parameter screen.
- 5 Enter 1 in response to the prompt for PWE (writing parameters). Alarm P/S100 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 PWE (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.

Procedure for Outputting Parameters

- 1 Make sure the output device is ready for output.
For the 0-TTC, select the tool post for which parameters to be input are used with the tool post selection switch.
- 2 Specify the punch code system (ISO or EIA) using a parameter.
To output a floppy, specify ISO.
- 3 Press the EDIT switch on the machine operator's panel.
- 4 Press function key , and display parameter screen.
- 5 Press soft key  .
All parameters (pitch error compensation is included) 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 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.

Procedure for Inputting Custom Macro B Common Variables

- 1 Input the program according to the procedure in Subsection 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.

Procedure for Outputting Custom Macro Common Variable

- 1 Make sure the output device is ready for output.
- 2 Specify the punch code system (ISO or EIA) using a parameter.
To output a floppy, specify ISO.
- 3 Press the **EDIT** switch on the machine operator's panel.
- 4 Press function key , and display macro variable screen.
- 5 Press soft 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 **VMACRO VARW**.

• 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.




8.8.1

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)		O0001 N0000
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
		S 0 T0101
19:02:33		EDIT
[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).

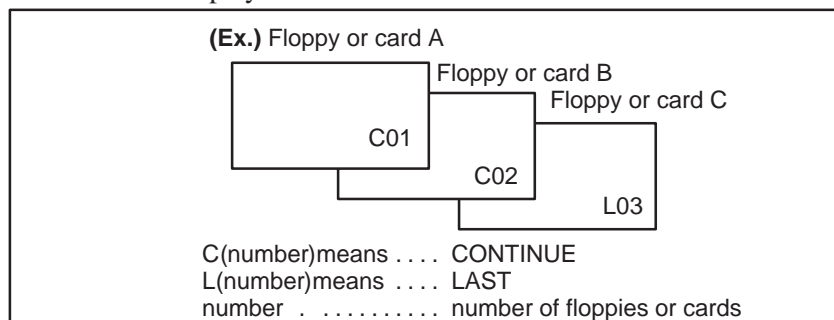
DIRECTORY (FLOPPY)		O0001 N0000
NO.	FILE NAME	(METER) VOL
0010	O0613	5.2
0011	O1021	3.9
0012	O1041	3.2
0013	O1051	3.2
0014	O0010	2.6
0015	O2011	3.9
0016	O2505	7.1
0017	O0011	6.5
0018	O3511	3.2
SEARCH		
_FILE NO.= 10		
NUM		S 0 T0101
19:04:05	EDIT	
[EXEC]	[CAN]	[STOP]

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].

```

DIRECTORY ( FLOPPY )                                00001  N0000
NO.  FILE NAME                                     ( METER ) VOL
0010  00613                                         5.2
0011  01021                                         3.9
0012  01041                                         3.2
0013  01051                                         3.2
0014  00010                                         2.6
0015  02011                                         3.9
0016  02505                                         7.1
0017  00011                                         6.5
0018  03511                                         3.2
SEARCH
 FILE NO.=      10  PROGRAM NO.=
NUM                               S    0  T0101
19:06:32                        EDIT
[ EXEC ] [ CAN ] [           ] [ STOP ]

```


- 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).

8.8.4 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].

```



DIRECTORY(FLOPPY)                O0001 N0000
  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
DELETE
  FILE NO.=      3
  NUM
19:22:01                      EDIT
[ EXEC ] [ CAN ] [          ] [ STOP ]

```

- 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.)

- **Changing a file name**

You can change the name of a file having a specified file number.

Procedure for changing the file name

Procedure

- 1 Press the EDIT switch on the machine operator's panel.
- 2 Press the  function key.
- 3 Press the **[FLOPPY]** soft key.
- 4 Press the **[RENAME]** soft key.
- 5 Position the cursor to "FILE NO.", enter the number of the file whose name will be changed, then press the  key.
- 6 Position the cursor to "NAME", enter a new file name, then press the  key.
- 7 Press the **[EXEC]** soft key.
- 8 To return to the previously displayed screen without changing the name, press the **[CAN]** soft key.

FILE DIRECTORY

NO. FILE NAME

O0001 N0000

(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)

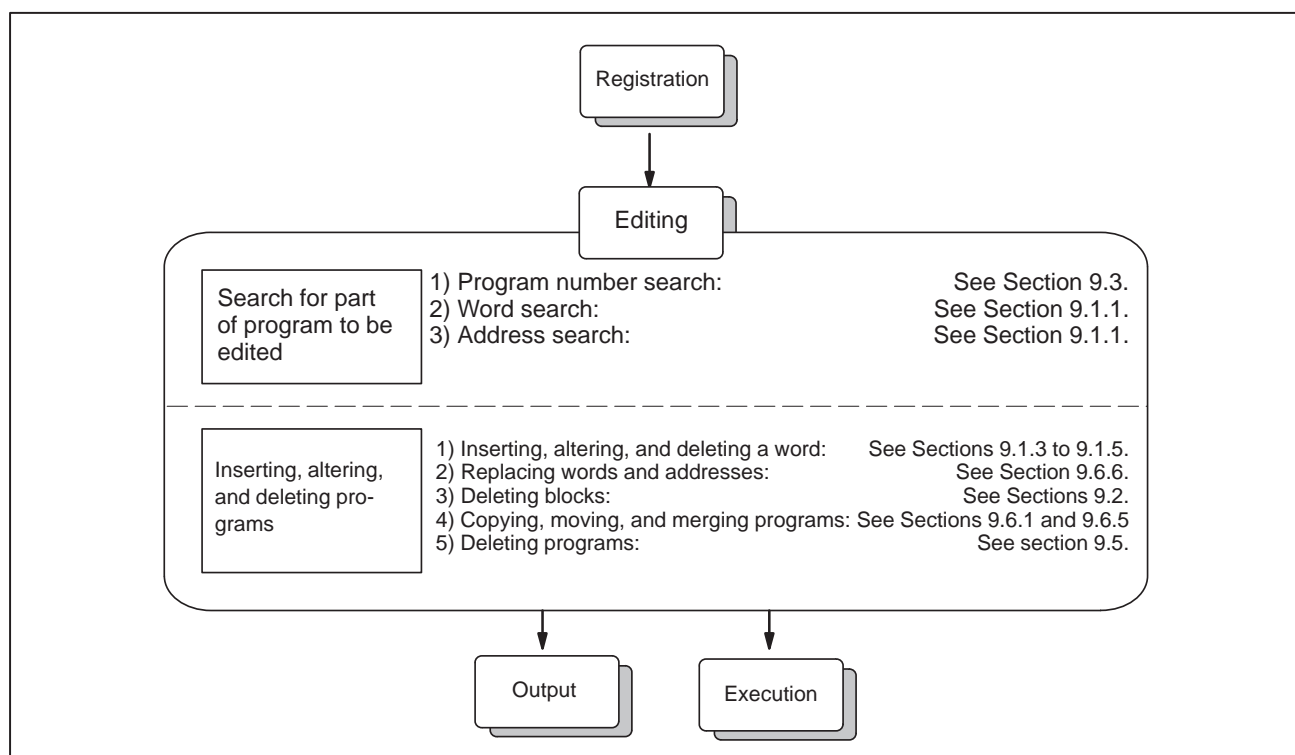
Limitations

- Use channel 1 or 2 for an I/O device. Set "INTERFACE" for the device. Note that some I/O devices cannot be used even if connected.
- When using the standard MDI keys, enter a file name as "address" + "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, 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

- 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.

Explanation

- **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

- 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.

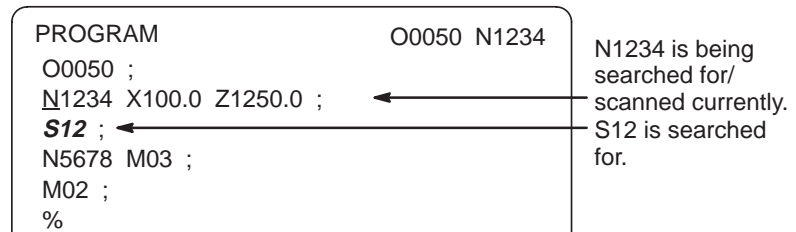
Example) When Z1250.0 is scanned

```
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

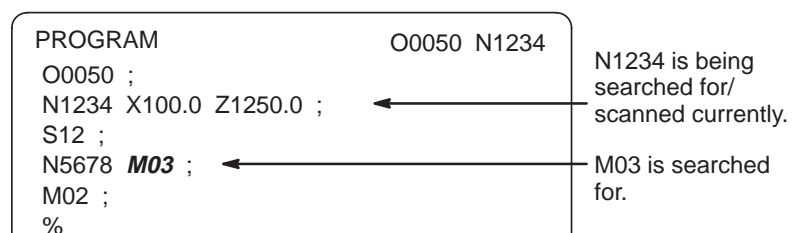
Example) of Searching for S12



- 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 <u>Z</u> 1250.0 ;	← Z1250.0 is searched for/ scanned.
S12 ;	
N5678 M03 ;	
M02 ;	
%	

- 2 Key in    .


- 3 Press the  key.

Program	O0050 N1234
O0050 ;	
N1234 X100.0 Z1250.0 <u>T15</u> ;	← T15 is inserted.
S12 ;	
N5678 M03 ;	
M02 ;	
%	

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 <u>T</u> 15 ;	← T15 is searched for/scanned.
S12 ;	
N5678 M03 ;	
M02 ;	
%	

- 2 Key in    .


- 3 Press the  key.

Program	O0050 N1234
O0050 ;	
N1234 X100.0 Z1250.0 <u>M</u> 15 ;	← T15 is changed to M15.
S12 ;	
N5678 M03 ;	
M02 ;	
%	

9.1.5

Deleting a Word

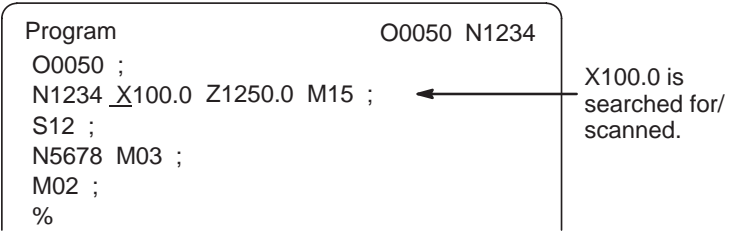
Procedure for deleting a word

- 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 .
- 3 Press the  key.

Example of deleting a block of No.1234

Procedure

- 1 Search for or scan N1234.

Program	O0050 N1234
O0050 ;	
<u>N</u> 1234 Z1250.0 M15 ;	← N1234 is searched for/ scanned.
S12 ;	
N5678 M03 ;	
M02 ;	
%	

- 2 Key in .
- 3 Press the  key.

Program	O0050 N1234
O0050 ;	← Block containing N1234 has been deleted.
<u>S</u> 12 ;	
N56789 M03 ;	
M02 ;	
%	

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.

<pre> Program O0050 N1234 O0050 ; <u>N1234</u> Z1250.0 M15 ; S12 ; N56789 M03 ; M02 ; %</pre>	<p>← N1234 is searched for/ scanned.</p>
------------------------------------------------------------------------------------------------------------------------------	--------------------------------------------------

- 2 Key in N 5 6 7 8 .

<pre> Program O0050 N1234 O0050 ; <u>N1234</u> Z1250.0 M15 ; S12 ; N5678 M03 ; M02 ; %</pre>	<p>← Underlined part is de- leted.</p>
-----------------------------------------------------------------------------------------------------------------------------	------------------------------------------------

- 3 Press the DELET key.




<pre> Program O0050 N1234 O0050 ; <u>M02</u> ; %</pre>	<p>← Blocks from block containing N1234 to block contain- ing N5678 have been deleted.</p>
---------------------------------------------------------------------------------------	------------------------------------------------------------------------------------------------------------

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 .

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




- 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  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.

Restrictions

This function is available when MDI full key is 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.

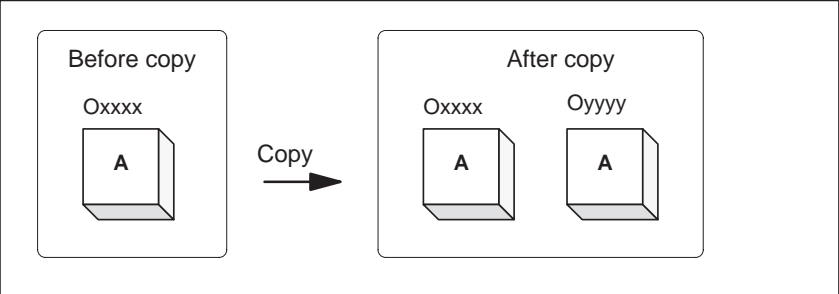



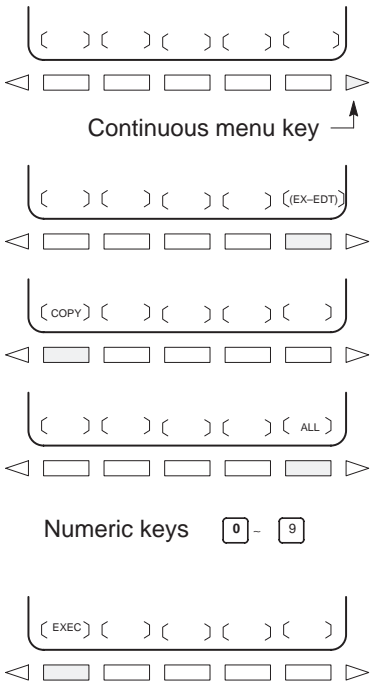


Fig. 9.5.1 Copying an Entire Program

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.

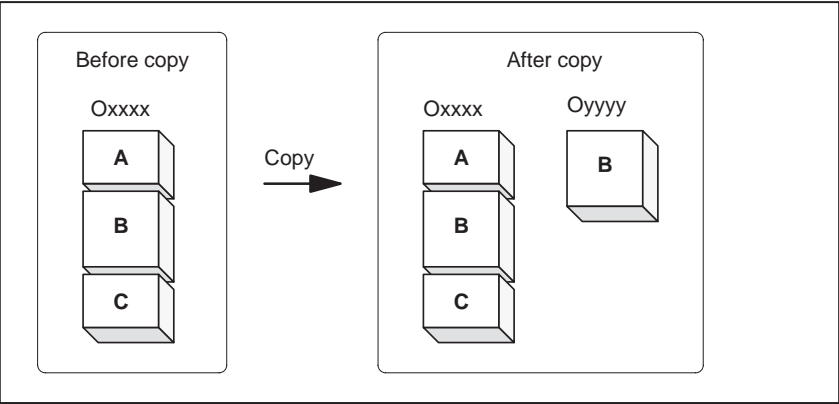
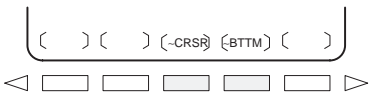
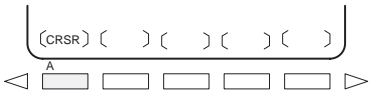


Fig. 9.5.2 Copying Part of a Program

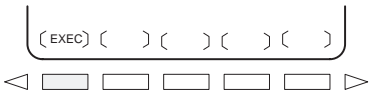
In Fig. 9.5.2, part B of the program with program number xxxx is copied to a newly created program with program number yyyy. The program which No.is xxxx remains unchanged after copy operation.

Procedure for copying part of a program

- 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 [INPUT] key.
- 5 Press soft key [EXEC].



Numeric keys [0] - [9]



9.5.3

Moving Part of a Program

A new program can be created by moving part of a program.

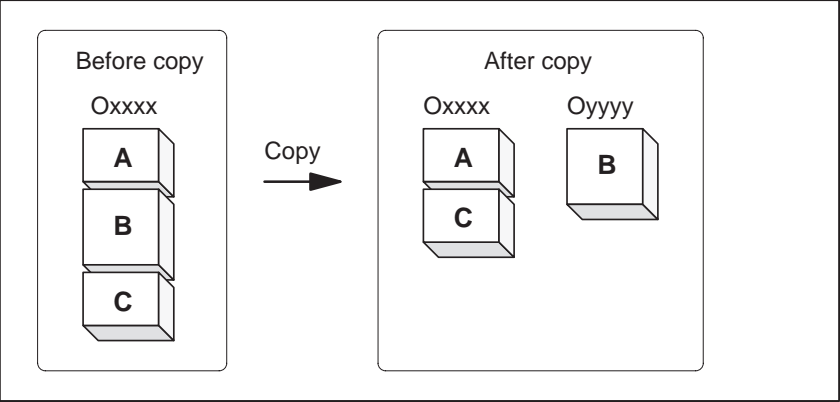
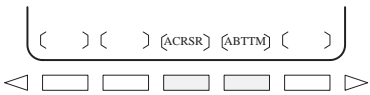
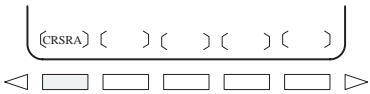
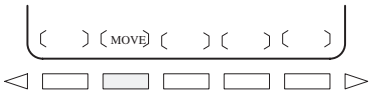


Fig. 9.5.3 Moving Part of a Program

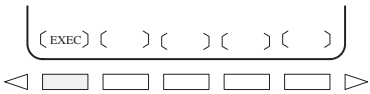
In Fig. 9.5.3, part B of the program with program number xxxx is moved to a newly created program with program number yyyy; part B is deleted from the program with program number xxxx.

Procedure for moving part of a program

- 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]**.



Numeric keys -



9.5.4 Merging a Program

Another program can be inserted at an arbitrary position in the current program.

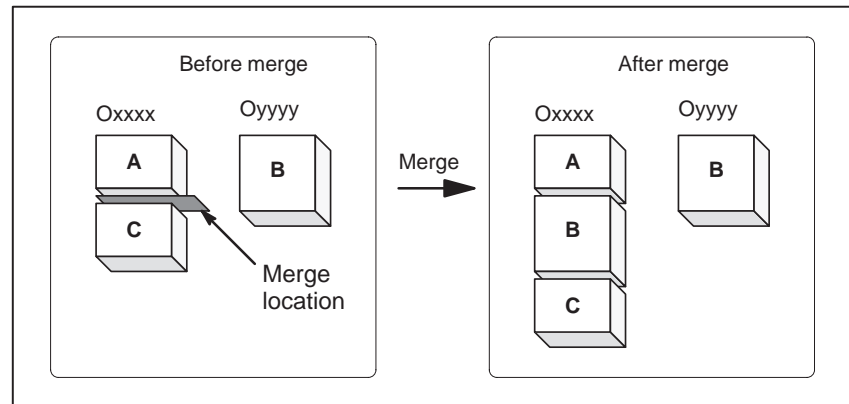
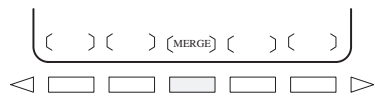


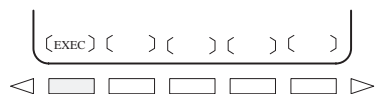
Fig. 9.5.4 Merging a program at a specified location

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



Numeric keys 0 - 9



- 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 **[~BTM']** (in the latter case, the end of the current program is displayed).
- 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 editing range must be reset starting with a start point.

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 re-registered. (In merge operation, the previous information is not deleted.) However, the program, when selected for foreground operation, cannot be re-registered in the background. (A BP/S140 alarm is raised.) When the program is re-registered, a free area is produced. Delete such a free area with the

 key.

- 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	<p>The power was interrupted during copying, moving, or inserting a program and memory used for editing must be cleared. When this alarm occurs, press the key  while pressing function key .</p> <p>Only the program being edited is deleted.</p>

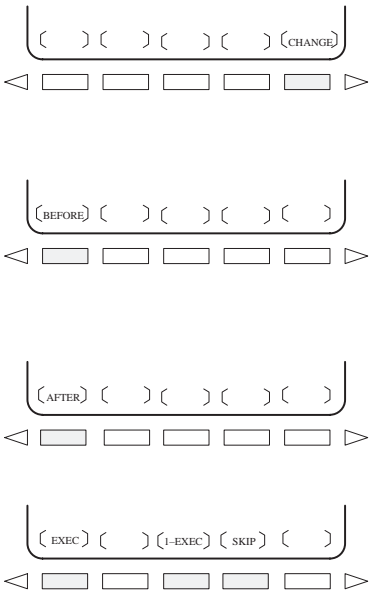
9.5.6

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.

Examples

- Replace X100 with Z200

[CHANGE] X 1 0 0 [BEFORE] Z 2 0 0
[AFTER][EXEC]

- Replace X100 Z200 with X30

[CHANGE] X 1 0 0 Z 2 0 0
[BEFORE] X 3 0 [AFTER][EXEC]

- Replace IF with WHILE

[CHANGE] I F [BEFORE] W H I L E [AFT
ER] [EXEC]

- Replace X with ,C10

[CHANGE] X [BEFOR] , C 1 0 [AFTER][EXEC]

Explanation

- 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 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 MACROS B

Unlike ordinary programs, custom macro B programs are modified, inserted, or deleted based on editing units.

Custom macro B words can be entered in abbreviated form.

Full key is required for edition of custom macros B.

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 macroB word is altered or inserted, the first two characters or more can replace the entire word.

Namely,

WHILE → WH	GOTO → GO	XOR → XO	AND → AN
SIN → SI	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 , display program screen.
- 2 Press soft key  at the right side, and 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-EDT]**.

Explanation

- **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.

CAUTION

- 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.

CAUTION

- 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
- CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION
- AUTOMATIC PROGRAM PREPARATION DEVICE (FANUC SYSTEM P)




This chapter describes creating programs using the MDI panel, TEACH IN mode, 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, and display program screen.
- 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.




Explanation

• Comments in a program


Comments can be written by full key 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 III-9.7).
- 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 0550.

Procedure for automatic insertion of sequence numbers

Procedure

- 1 Set 1 for SEQUENCE (see subsection 11.4.3).
- 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 .
- 7 Enter each word of a block.
- 8 Press .
- 9 Press . 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 10, N20 inserted and

displayed below the line where a new block is specified.

```

PROGRAM                                O0040 N0020
O0040 ;
N10 G50 X0 Z0 ;
N20
%

<                                     S    0 T0101
19:24:50                             EDIT
[ PRGRM ][ LIB ][FLOPPY ][          ][C.A.P. ]

```

- 10** · In the example above, if N20 is not necessary in the next block, pressing the key after N20 is displayed deletes N20.
- To insert N100 in the next block instead of N20, enter N100 and press after N20 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, Z, and Y axes obtained by manual operation is stored in memory as a program position to create a program.





The words other than X, Z, and Y, 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

The procedure described below can be used to store a machine position along the X, Z, and Y axes.

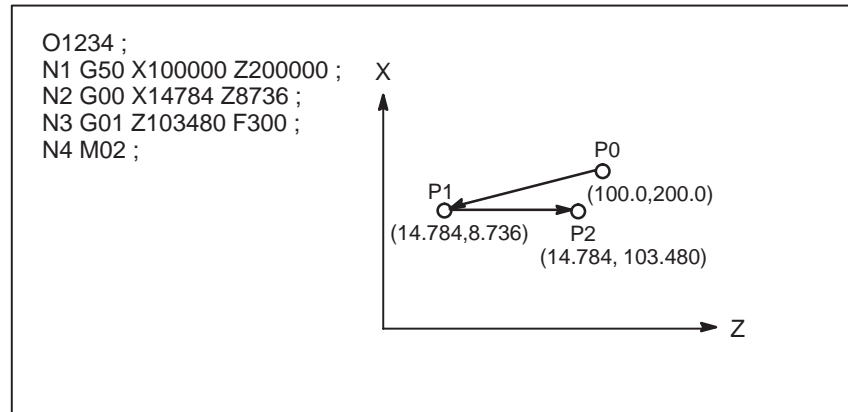
- 1 Select the **TEACH IN JOG** mode or **TEACH IN HANDLE** mode.
- 2 Move the tool to the desired position with jog or handle.
- 3 Press  key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the machine position along 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 Z axis is stored in memory. Further, key in  , then press the  key. Then a machine position along the Y axis is stored in memory.

All coordinates stored using this method are absolute coordinates.

Examples



- 1 Set the setting data **SEQUENCE NO.** to 1 (on). (The incremental value parameter (No. 0550) 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 5 0 INSRT X INSRT Z INSRT EOB INSRT

This operation registers G50 X100000 Z200000 ; 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 X INSRT Z INSRT EOB INSRT

This operation registers G00 X14784 Z8736; 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:

G

0

1

INSRT

Z

INSRT

F

3

0

0

INSRT

EOB

INSRT

This operation registers G01 Z103480 F300; in memory.
The automatic sequence number insertion function registers N4 of the fourth block in memory.

11 Register M02; in memory as follows:

M

0

2

INSRT

EOB

INSRT

N5 indicating the fifth block is stored in memory using the automatic sequence number insertion function. Press the

DELET

 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.

PROGRAM

O0040 N0020

(RELATIVE)

(ABSOLUTE)

U14.784X14.784

W103.408Z103.408

H0.000C0.000

V0.000Y0.000

O1234 ;

N10 G50 X100. Z200. ;

N20 G00 X14784 Z8736 ;

N30 G01 Z103408 F300 ;

N40 M02 ;

%

ADRS.

S0 T0101

19:35:36

TJOG

[PRGRM][LIB][][][]

- **Registering a position with compensation**

When a value is keyed in after keying in address

X

 ,

Z

 , or

Y

 , then the

INSRT

 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                                00040 N0020
O100 ;
N10 G50 X100. Z0 ;
N20 G00 X25.0 Z-5.0 ;
N30
%

G00 : POSITIONING
G01 : LINEAR IPL
G02 : CIRCULAR IPL CW
G03 : CIRCULAR IPL CCW
G04 : DWELL
G <                                S 0 T0101
19:47:26                            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                                O0040 N0020
O100 ;
N10 G50 X100. Z0 ;
N20 G00 X125. Z-5. ;
N30 G01
%

G01   X(U) --- Z(W)--- F--- M--- S--- T---

<                                     S   0 T0101
19:47:57                             EDIT
[ 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                                O0040 N0020
N10 G50 X100. Z0 ;
N20 G00 X25. Z-5. ;
N30 G01 X30. Z-60. F500 T12 ;
N40
%

<                                     S   0 T0101
19:49:38                             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.

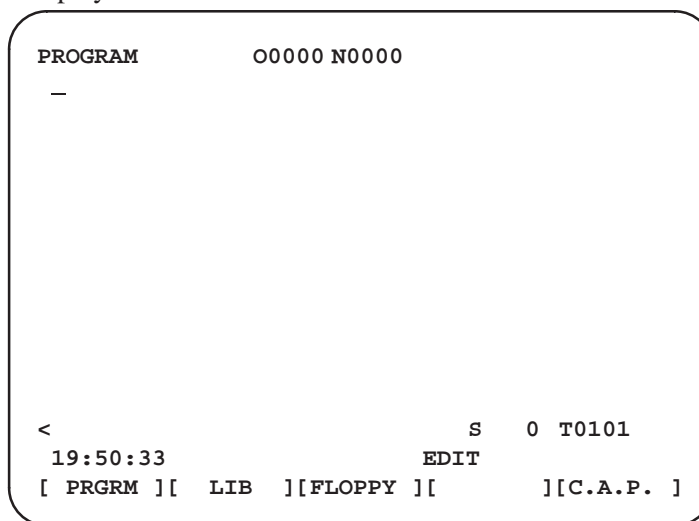
Blocks in a program can be modified, inserted, or deleted using the G code menu and conversational screen.








Procedure for Conversational Programming with Graphic Function

Procedure 1 Creating a program

- 1 Enter the **EDIT** mode.

- 2 Press . 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 100 is to be registered, key in    , then press . This registers a new program O0100.
- 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                                00100 N0000
G00 : POSITIONING
G01 : LINEAR IPL.
G02 : CIRCULAR IPL. CW
G03 : CIRCULAR IPL. CCW
G04 : DWELL
G10 : OFFSET VALUE SETTING    <0>
G20 : INCH
G21 : METRIC
G22 : STORED STROKE CHECK ON  <0>
G23 : STORED STROKE CHECK OFF <0>
G25 : SPINDLE SPEED DETECT OFF
G26 : SPINDLE SPEED DETECT ON
ADRS.
19:52:32                                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.

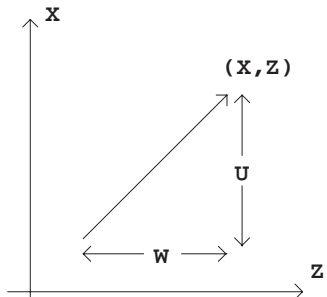
```

PROGRAM                                00100 N0000
G00 : POSITIONING

G00  G      G      G
_X      U
_Z      W
_C      H
_M
_S
_T
_B
;

19:56:25                                EDIT
[G.MENU] [ ] [ ] [ ] [ ]

```



When no keys are pressed, the standard details screen is displayed.

PROGRAM				00100 N0000	
STANDARD FORMAT					
<u>G</u>	G	G	G		
X			U		
Z			W		
A			C		
F			H		
I			K		
P			Q		
R			M		
S			T		
;					
19:57:38			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 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 key (Return menu key) at the left side. The registered programs are converted to the conversational format and displayed.
- 12 Press the key to return to the program head.


Procedure2

Modifying a block

- 1 Move the cursor to the block to be modified on the program screen and press the **[C.A.P]** soft key. Or, press the **[C.A.P]** soft key first to display the conversational screen, then press the or page key until the block to be modified is displayed.
- 2 When data other than a G code is to be altered, just move the cursor to the data and key in a desired value, then press key.
- 3 When a G code is to be altered, press 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.


Procedure3

Inserting a block

- 1 On the conversational screen, display the block immediately before a new block is to be inserted, by using the page keys. On the program screen, move the cursor with the page keys and cursor keys to immediately before the point where a new block is to be inserted.
- 2 Press the soft key **[G.MENU]** to display the G code menu. Then enter new block data.
- 3 When input of one block of data is completed in step 2, press the  key. This operation inserts a block of data.

Procedure4

Deleting a block

- 1 On the conversational screen, display the contents of a block to be deleted, then press the  key.
- 2 The contents of the block displayed are deleted from program memory. Then the contents of the next block are displayed on the conversational screen.

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



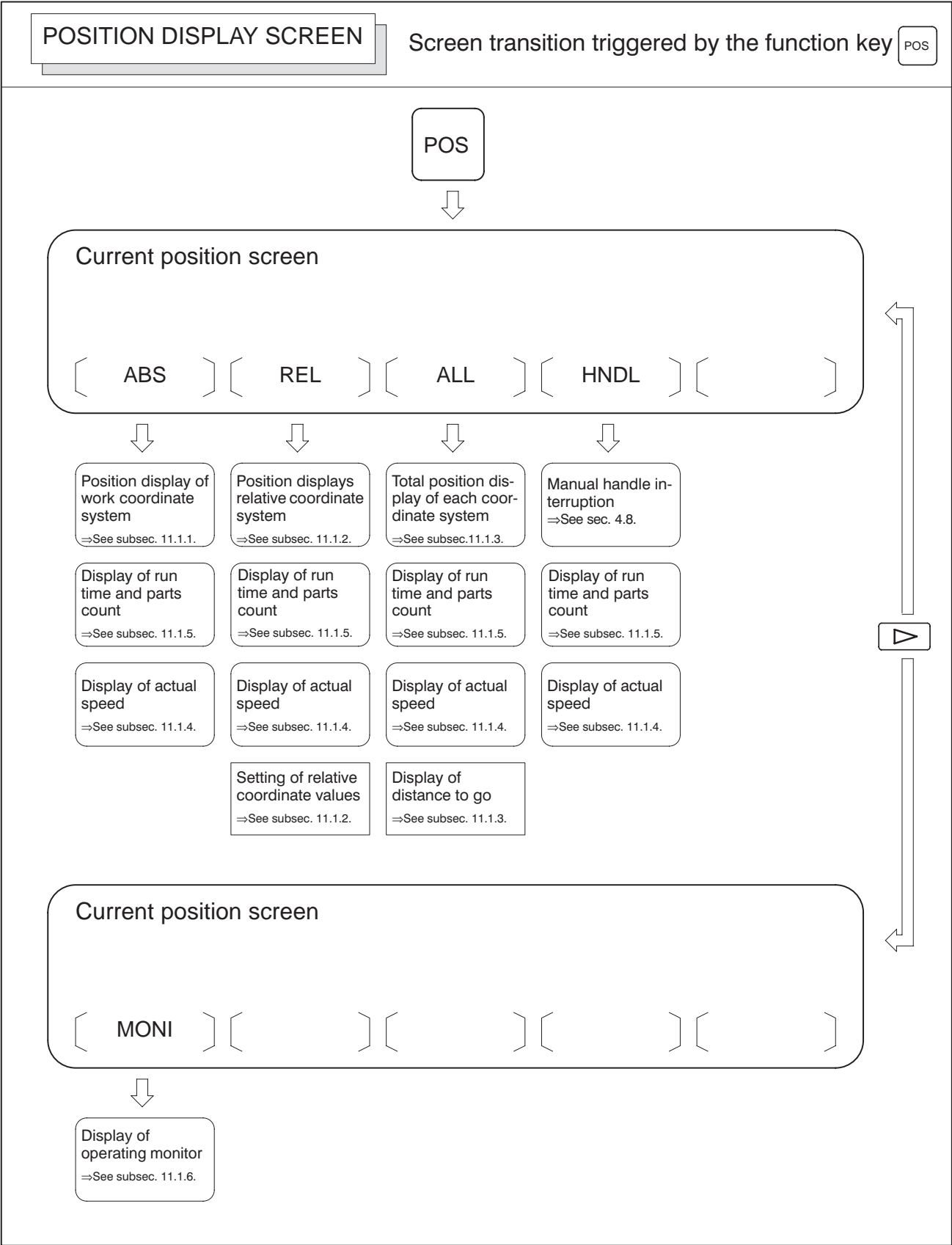
MDI function keys
(Shaded keys () are described in this chapter.)

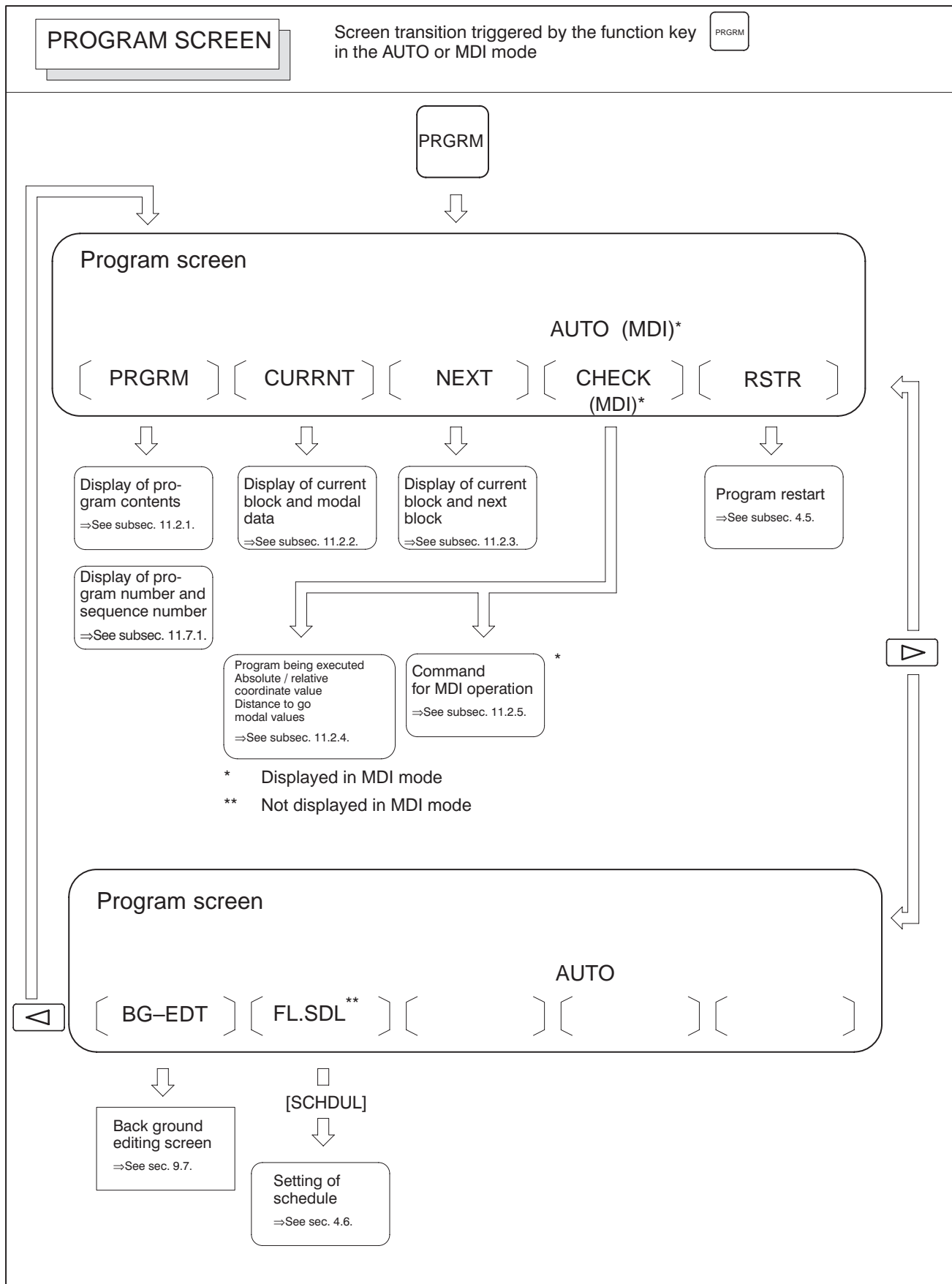
The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

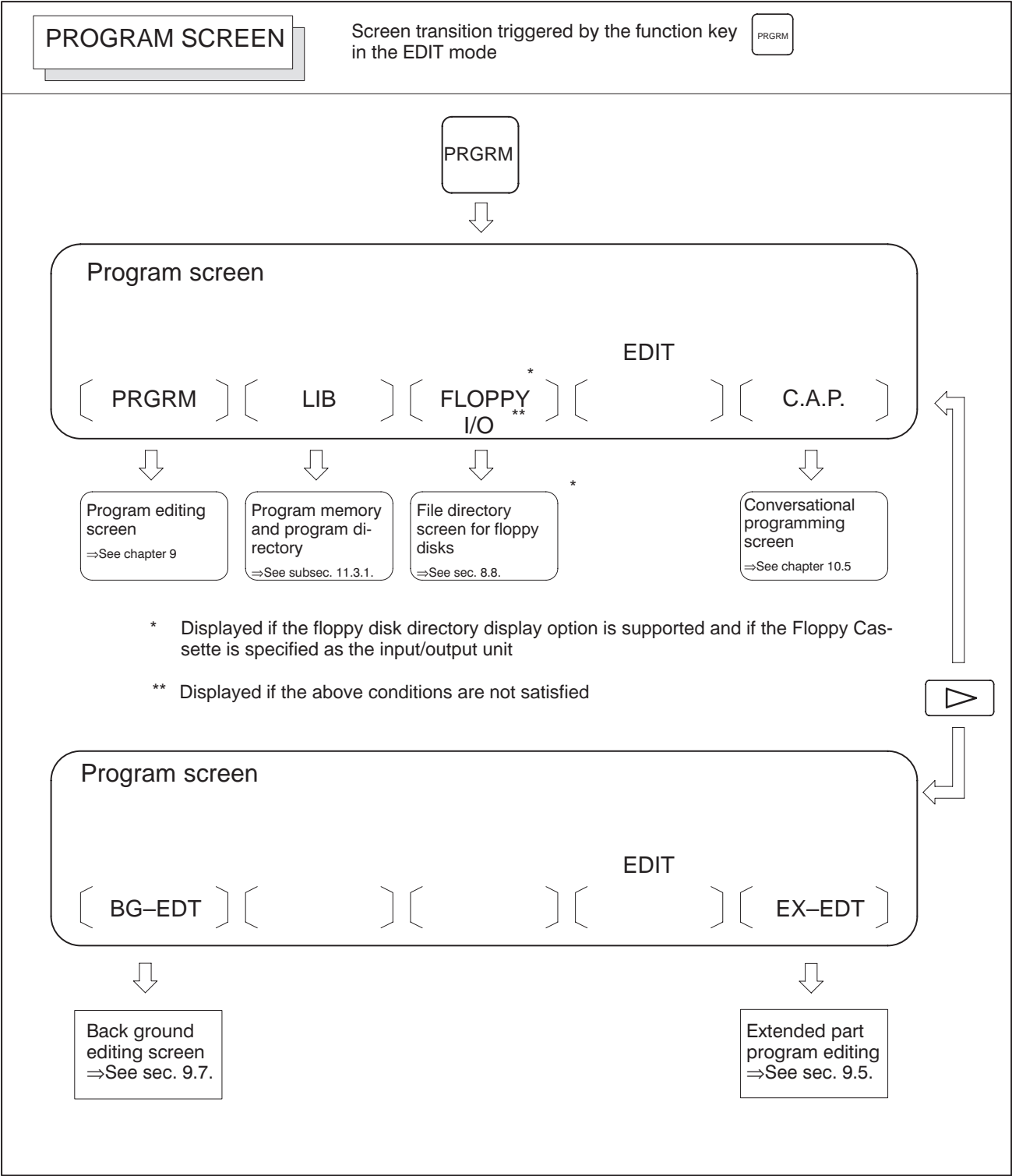
See Chapter 12 for the screen that appears when function key  is pressed.

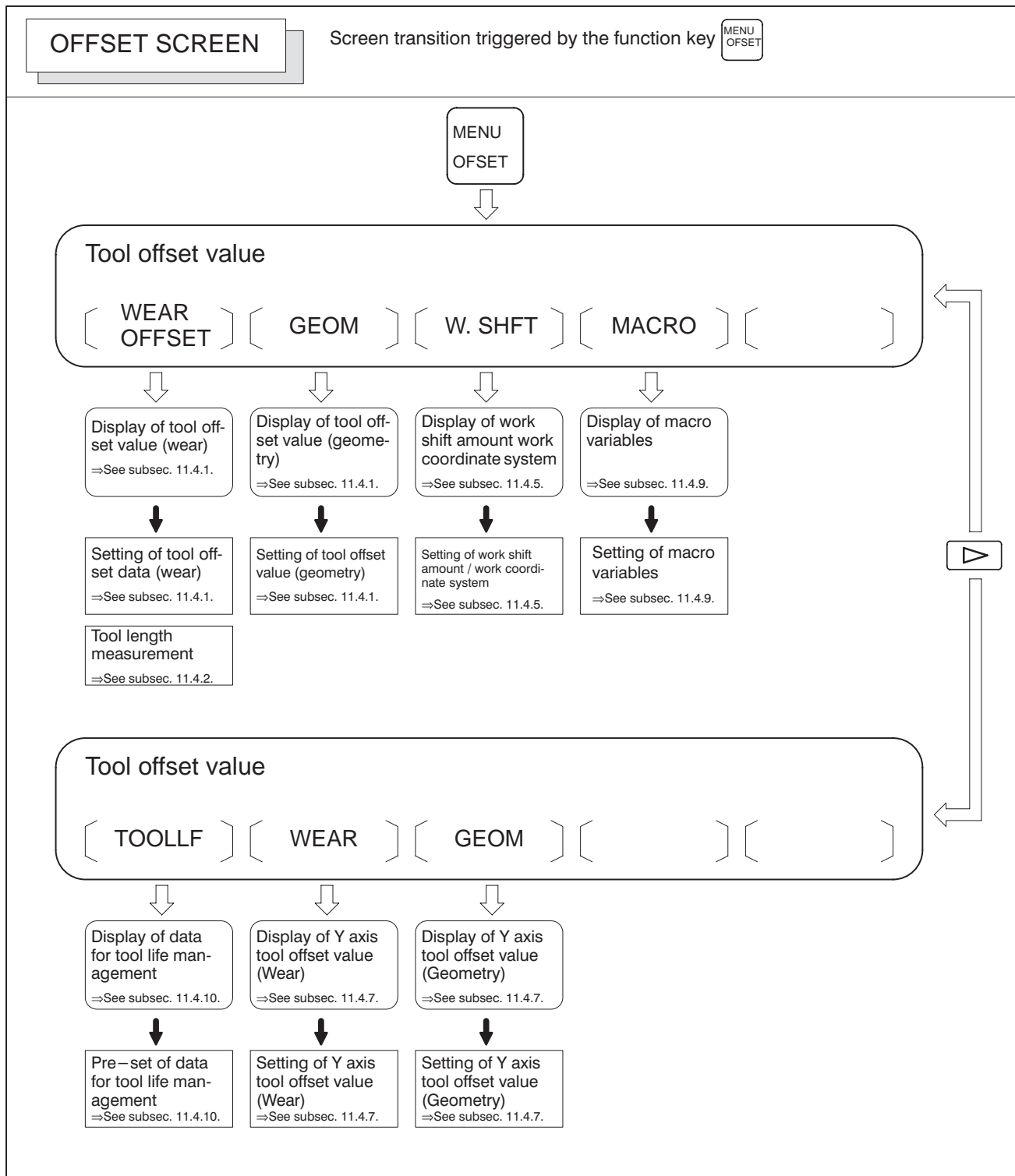
• Data protection key

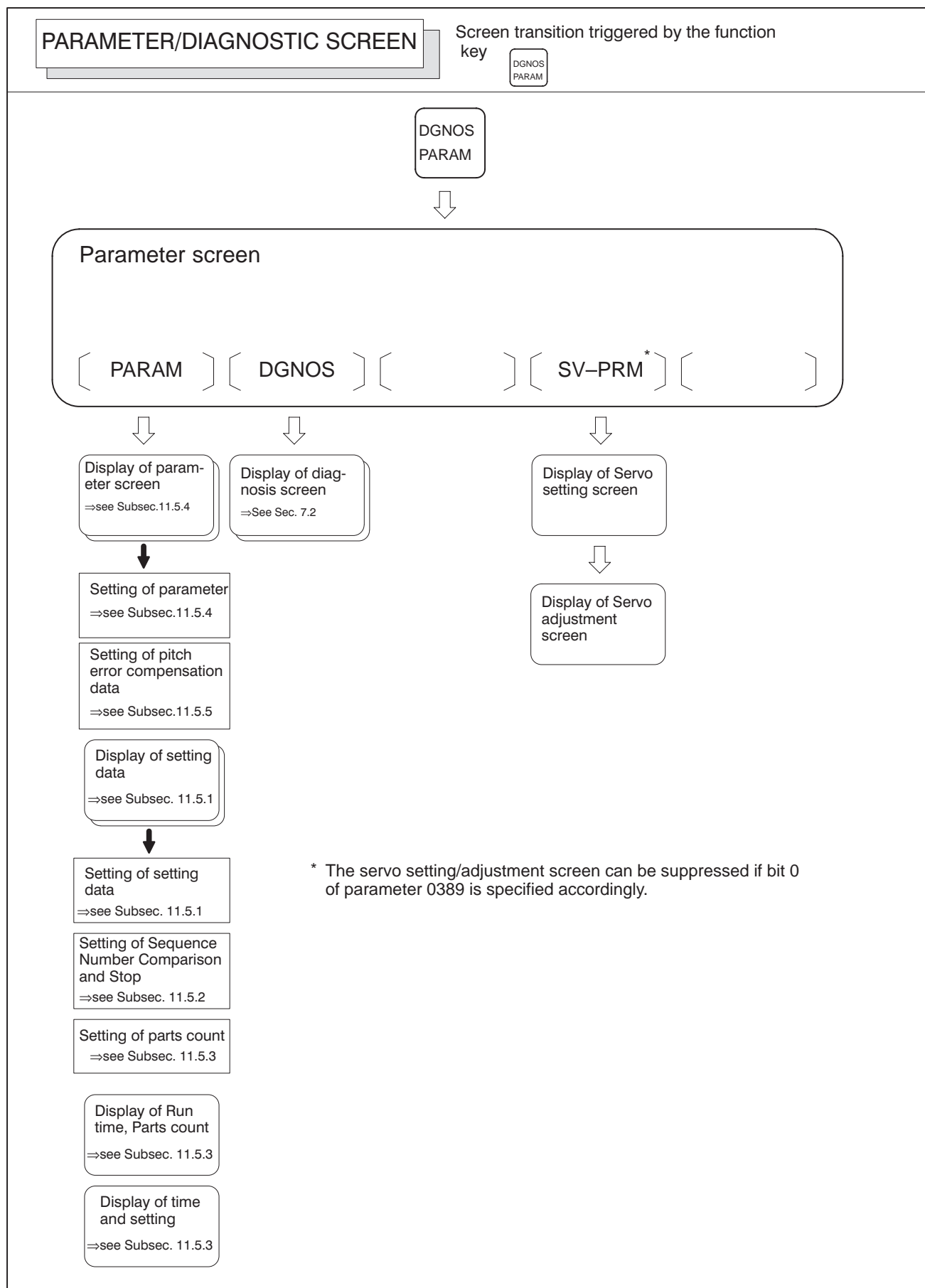
The machine may have a data protection key to protect part programs. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

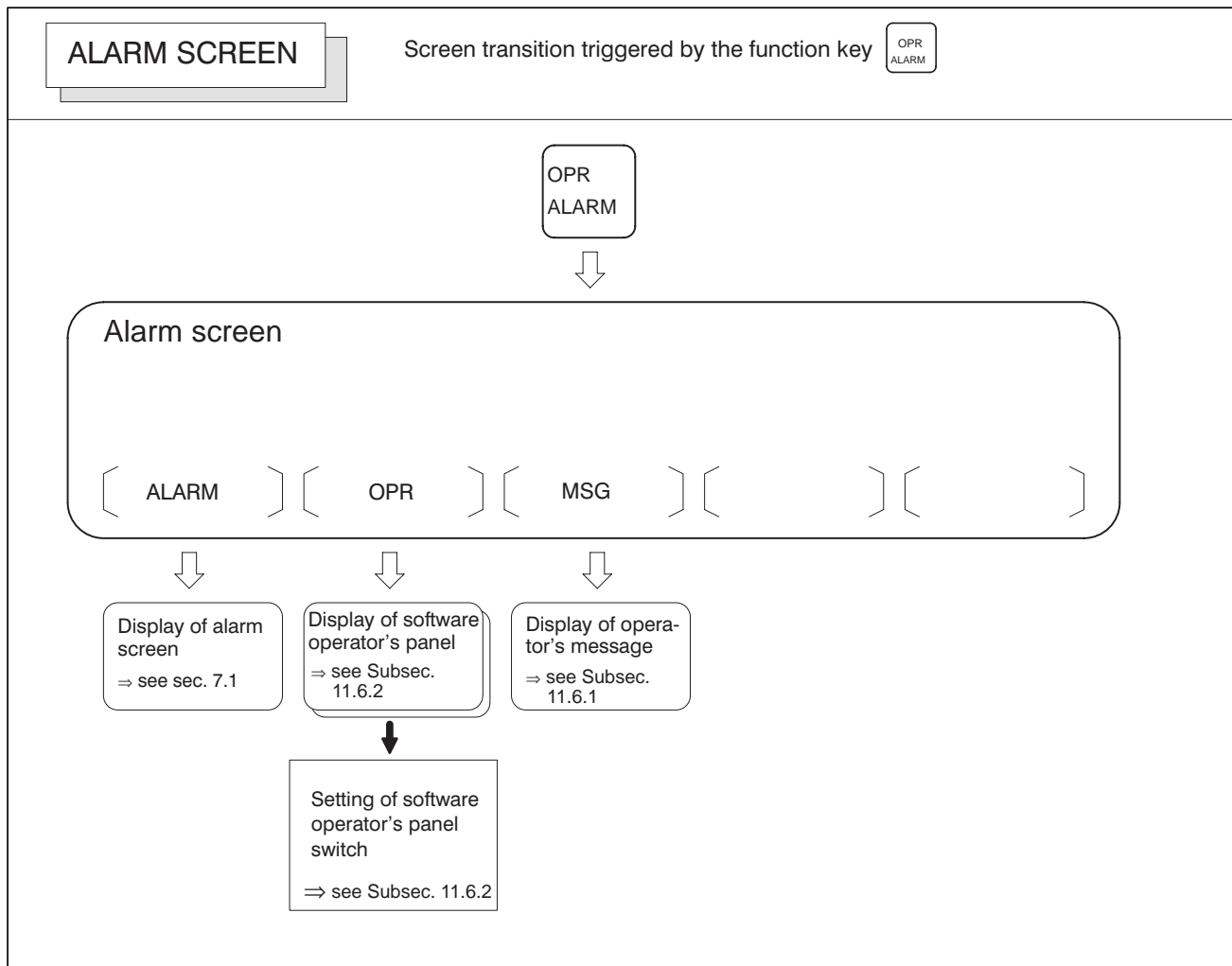












- **Setting screens**

The table below lists the data set on each screen.

Table 11 Setting screens and data on them

No.	Setting screen	Contents of setting	Reference item
1	Tool offset value	Tool offset value Tool nose radius compensation value	Subsec. 11.4.1
		Direct input of tool offset value	Subsec. 11.4.2
		Direct input of tool offset value measured B	Subsec. 11.4.3
		Counter input of offset value	Subsec. 11.4.4
		Y axis offset	Subsec. 11.4.7
2	Workpiece coordinate system setting	Workpiece coordinate system shift value	Subsec. 11.4.5
		Workpiece origin offset value	Subsec. 11.4.8
3	Setting data (handy)	Parameter write TV check Punch code (EIA/ISO) Input unit (mm/inch) I/O channel Automatic insert of Sequence No. Conversion of tape format	Subsec. 11.5.1
		Sequence number comparison and stop	Subsec. 11.5.2
4	Setting data (mirror image)	Mirror image	Subsec. 11.5.1
5	Setting data (timer)	Parts required	Subsec. 11.5.3
6	Macro variables	Custom macro common variables (#100 to #149) or (#100 to #199) (#500 to #531) or (#500 to #599)	Subsec. 11.4.9
7	Parameter	Parameter	Subsec. 11.5.4
		Pitch error compensation data	Subsec. 11.5.5

Table 11 Setting screens and data on them

No.	Setting screen	Contents of setting	Reference item
8	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
9	Tool life data (Tool life management)	Life count	Subsec. 11.4.10
10	Current position display screen	Reset of relative coordinate value.	Subsec. 11.1.2


11.1 SCREENS DISPLAYED BY FUNCTION KEY

A small rectangular icon with the text "POS" inside, representing a 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 4.8 for details on this screen.

11.1.1 Position Display in the Workpiece Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate system

- 1 Press function key  .
- 2 Press soft key [ABS].

• Display with 0-TC

ACTUAL POSITION (ABSOLUTE)		O0001 N0023
X	200.000	
Z	220.000	
C	0.000	
Y	0.000	
PART COUNT 1786		
RUN TIME 2H47M	CYCLE TIME 0H 1M47S	
ACT.F 3000 MM/M		
16:14:02	BUF AUTO	
[ABS]	[REL]	[ALL] [HNDL]

• Display with 0-TTC

ACTUAL POSITION (ABSOLUTE)		HEAD1
X ₁	163.710	O0555
Z ₁	295.868	N5555
C ₁	0.000	
Y ₁	0.000	
X ₂	211.680	O0222
Z ₂	185.247	N0222
ACT.F 0 MM/M		
14:57:19	AUTO	
[ABS]	[REL]	[ALL] [HNDL]

Explanations

• Display including compensation values

Tool compensation values and other values are displayed on the screen displaying a position in the workpiece coordinate system. The programmed point (position of the tool tip) is also displayed.

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.

Display procedure for the current position screen with the relative coordinate system

1 Press function key POS .

2 Press soft key [REL].

• Display with 0-TC

ACTUAL POSITION (RELATIVE)		O0001 N0023
U	200.000	
W	220.000	
H	0.000	
V	0.000	
PART COUNT 23		
RUNTIME 3H30M	CYCLE TIME 0H 2M14S	
ACT.F 3000 MM/M		
20:03:21	AUTO	
[ABS]	[REL]	[ALL] [HNDL]

• Display with 0-TTC


ACTUAL POSITION (RELATIVE)		HEAD1
U ₁	37.374	O0555
W ₁	145.341	N5555
H ₁	0.000	
V ₁	0.000	
U ₂	-47.040	O0222
W ₂	166.990	N0222
ACT.F 0 MM/M	JOG	
14:59:02		
[ABS]	[REL]	[ALL] [HNDL]

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 G50 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].

• Display with 0-TC

ACTUAL POSITION		O0100 N0000
(RELATIVE)		(ABSOLUTE)
U	0.000	X 200.179
W	0.000	Z 220.000
H	0.000	C 0.000
V	0.000	Y 0.000
(MACHINE)		(DISTANCE TO GO)
X	-118.170	X 0.000
Z	-21.470	Z 0.000
C	0.676	C 0.000
Y	0.046	Y 0.000
RUN TIME 3H30M		PART COUNT 23
ACT.F 0 MM/M		CYCLE TIME 0H2M14S
20:04:37		AUTO
[ABS][REL][ALL][HNDL][]		

• Display with 0-TTC

ACTUAL POSITION		HEAD1 :00555 N5555
(RELATIVE)		(ABSOLUTE)
U1	0.000	X1 200.179
W1	0.000	Z1 220.000
H1	0.000	C1 0.000
V1	0.000	Y1 0.000
(MACHINE)		
X1	-118.170	
Z1	-21.470	
C1	0.676	
Y1	0.046	
RUN TIME 3H30M		PART COUNT 23
ACT.F 0 MM/M		CYCLE TIME 0H2M14S
15:00:48		AUTO
[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 also be used by setting bit 0 of parameter 0063.

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 0028.

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)		O0001 N0023
X	200.000	
Z	220.000	
C	0.000	
Y	0.000	
PART COUNT 1786		
RUN TIME 2H47M	CYCLE TIME 0H 1M47S	
ACT.F 3000 MM/M		
16:14:02	BUF AUTO	
[ABS] [REL] [ALL] [HNDL] []		

Actual feedrate is displayed after ACT.F.

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.

- **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 (ABSOLUTE)		O0001 N0023
X	200.000	
Z	220.000	
C	0.000	
Y	0.000	
PART COUNT 1786		
RUN TIME 2H47M		CYCLE TIME 0H 1M47S
ACT.F 3000 MM/M		
16:14:02		BUF AUTO
[ABS][REL][ALL][HNDL][]		

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 0219 is executed. The number is reset to 0 when CAN key is pressed after address P key is pressed.

• RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time. The number is reset to 0 when CAN key is pressed after address R key is pressed.

• 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 Subsec 11.5.3.

• Parameter setting

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 3 (PCM) of parameter 0040 is used to specify whether the number of machined parts is incremented each time M02, M03, or an M code specified by parameter 0219 is executed, or only each time an M code specified by parameter 0219 is executed.

11.1.6 Operating Monitor Display

Explanations

• Operation

This function displays the load of basic feed axes and 1st spindle with serial interface. And also, it is possible to display the speed of 1st spindle with serial interface.

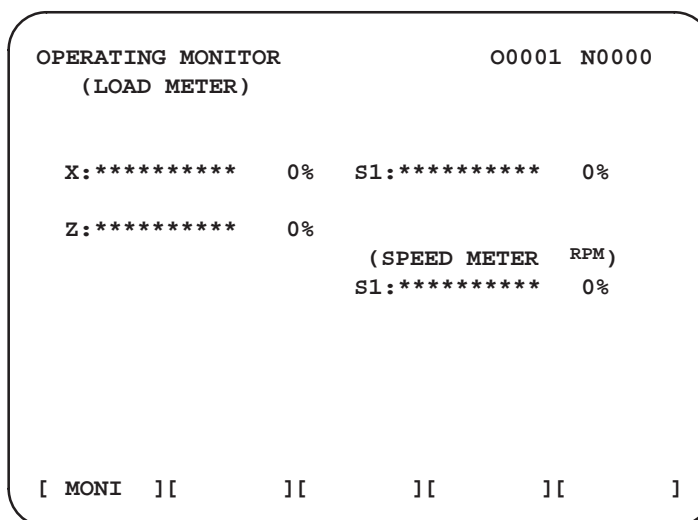
This function is basic.

- 1 The position screen is selected by pressing the function key POS.
- 2 The rightmost key > on the soft-keys is pressed.
- 3 The soft-key **[MONI]** is pressed.

By the above operation, the operating monitor screen is displayed.

Pressing the page keys ↑ ↓ can display the screen instead of the above operation (2) and (3).

• Operating Monitor Screen



(1) LOAD METER (X to Z)

The load of the axis is displayed by percentages to the rated torque of the axis.

One "*" marks on the bar graph denotes 10%.

(2) LOAD METER (S1)

The load of 1st spindle is displayed by percentages to the rated power of the spindle.

One "*" marks on the bar graph denotes 20%.


(3) SPEED METER (S1)

The speed of 1st spindle is displayed by RPM. One "*" marks on the bar graph denotes 10% of the maximum speed of the spindle.


NOTE

- 1 The loads of only the basic axes are displayed.
- 2 The load and speed of only 1st spindle with serial interface are displayed. Those for the 2nd serial spindle and analog interface spindle are not displayed.
- 3 This function is enabled by setting bit 5 of parameter No. 060 to 1. This function cannot be used with the 0-TTC.

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 4.5 for the program restart screen.
See Section 4.6 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

- 1 Press function key  to display a program screen.
- 2 Press chapter selection soft key **[PRGRM]**.
The cursor is positioned at the block currently being executed.

```

PROGRAM                                00100 N0013
S550 M08 ;
M45 ;
N010 G50 X200. Z200. ;
N011 G00 X160. Z180. ;
N012 G71 U7. R1. ;
N013 G71 P014 Q020 U4. W2. F0.3 S550 M03
;
N014 G00 X40. F0.15 S700 ;

```

```


20:06:48                                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.

PROGRAM		O0001 N0013	
(CURRENT)		(MODAL)	
G71 U	4.000	G00 F	0.3000
W	2.000	G97 M	003
F	0.3000	G69 S	00550
M	003	G99 T	0101
S	00550	G21	
P	14	G40 WX	0.000
Q	20	G25 WZ	0.000
		G22 SRPM	550
		G54 SSPM	0
		SMAX	32767
		SACT	0
20:07:48		AUTO	
[PRGRM][CURRNT][NEXT][CHECK][RSTR]			

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 chapter selection soft key **[NEXT]**.
The block currently being executed and the block to be executed next are displayed.

PROGRAM		O0100 N0000	
(CURRENT)		(NEXT)	
G71 U	4.000	G71.X	140.000
W	2.000	G97 M	-20.000
F	0.3000		
M	003		
S	00550		
P	14		
Q	20		

20:10:39


AUTO

[PRGRM][CURRNT][NEXT][CHECK][RSTR]

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.

```

PROGRAM                                O0100 N0000
N013 G71 P014 Q020 U4. W2.   F0.3 S550 M03
;
N014 G00 X40. F0.15 S700 ;
N015 G01 W-40. ;
      (RELATIVE)   (DIST TO GO)      (G)
U   -72.000 X      0.000 G00 G99 G25
W   -76.000 Z      0.000 G97 G21 G22
H    0.000 C      0.000 G69 G40 G54
V    0.000 Y      0.000 SPRM   550
                        SSPM    0
F    0.3000 S00550      SMAX  32767
M003          T0101      SACT    0
ACT.F          0 MM/M
20:11:37                BUF AUTO
[ PRGRM ][CURRNT ][ NEXT  ][ CHECK ][ RSTR  ]

```

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.0028#0).


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

Procedure

- 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

O0001 N0020

(MDI)

(MODAL)

G00.X 100.000

G00 F 0.3000

G80.Z 200.000

G97 M 003

G69 S 00550

G99 T 0101

G21

G40 WX 0.000

G25 WZ 0.000

G22 SRPM 550

G54 SSPM 0

SMAX 32767

SACT 0

ADRS.

20:17:19

MDI

[PRGRM][CURRNT][NEXT][MDI][RSTR]

(2) MDI operation – B

PROGRAM (MDI)

O0100 N0001

O0000 G00 X100. Z200. ;

N10 M03 ;

N20 G01 Z120. F500 ;

N30 M98 P9010 ;

N40 G00 Z0 ;

%

(MODAL)

G00 G69 G21 G25

G97 G99 G40 G22

F 0.3000 S00550

M003 T0101

<

20:14:17




MDI

[PRGRM][CURRNT][NEXT][MDI][RSTR]

Explanations

- MDI operation
- See Section 4.2 for MDI operation.
- Modal information
- The modal data on the MDI operation B is displayed when bit 3 of parameter 0028 is set to 1.


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.
For the 0-TTC, select the tool post for which a program is to be displayed with the tool post selection switch.
- 2 Press function key .
- 3 Press soft key **[LIB]**.

```

PROGRAM                                00615 N0000
  SYSTEM EDITION 0666 - 24
  PROGRAM NO.   USED :    24 FREE : 39
  MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
00001 00010 00011 00021 00041 00601
00613 00615 00645 00651 01021 01041
01051 02011 02505 03148 03153 03511
04011 04048 05111 05221 05766 06032

<                                     S    0 T0101
20:28:10                             EDIT
[ PRGRM ][CONDNS ][                ][C.A.P. ]

```


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. 0040#0) to 1.

PROGRAM

O0615 N0000

STSTEM EDITION

0666 - 24

PROGRAM NO. USED :

24 FREE :39

MEMORY AREA USED :

24960 FREE : 97920

PROGRAM LIBRARY LIST

O0001 (TEST-PRO)

O0010 (MILLING)

O0011 (THREADING)

O0021 (TURNING-ROUGH-1)

O0041 (TURNING-ROUGH-2)

O0601 (GROOVING)

O0613 (TURNING-FINE-1)

O0615 (TURNING-FINE-2)

<

S

0

T0101

20:26:56

EDIT

[PRGRM]

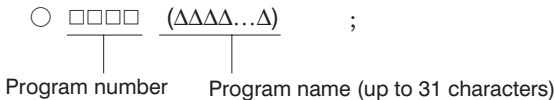
[CONDNS]

][

][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 0040 is set to 1, programs are displayed in the order of program number starting from the smallest one.

- **Order in which programs are registered**

Immediately after all programs are cleared (by turning on the power while pressing the  key), each program is registered after the last program in the list.

If some programs in the list were deleted, then a new program is registered, the new program is inserted in the empty location in the list created by the deleted programs.

Example) When bit 4 (SOR) of parameter 3107 is 0


1. After clearing all programs, register programs O0001, O0002, O0003, O0004, and O0005 in this order. The program library list displays the programs in the following order:
O0001, O0002, O0003, O0004, O0005
2. Delete O0002 and O0004. The program library list displays the programs in the following order:
O0001, O0003, O0005
3. Register O0009. The program library list displays the programs in the following order:
O0001, O0009, O0003, O0005

- **Free area**

Unused areas resulting from background editing are included when the number of programs or memory areas used is counted. After memory is reorganized, the unused area becomes a free area.

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 or workpiece coordinate system shift value
3. Custom macro common variables
4. Software operator's panel
5. Tool life management data

This section also describes following functions.

- Direct input of tool offset value
- Direct input of tool offset value measured B
- Counter input of offset value
- Direct input of workpiece coordinate system shift
- Y axis offset

The following functions depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

- Direct input of tool offset value
- Direct input of tool offset value measured B
- Software operator's panel
- Tool life management data

11.4.1 Setting and Displaying the Tool Offset Value

Dedicated screens are provided for displaying and setting tool offset values and tool nose radius compensation values.

Procedure for setting and displaying the tool offset value and the tool nose radius compensation value

- 1 Press function key  .

For the 0-TTC, select the tool post for which tool compensation values are to be displayed with the tool post selection switch.

- 2 Press soft key **[OFFSET]**, **[WEAR]**, **[GEOM]**.

Different screens are displayed depending on whether tool geometry offset, wear offset, or neither is applied.

```

OFFSET                                O0001 N0013
NO.      X          Z          R T
01        1.350      5.230      0.0000
02       -2.580     13.540      0.0000
03        5.843     -10.256      0.0000
04       50.245     100.235      0.0000
05       -3.124      36.520      0.3003
06      -16.369      53.248      0.2504
07      -10.587       0.000      0.0000
08       -0.258     -12.354      0.0000
ACTUAL POSITION (RELATIVE)
  U        0.000          W        0.000
  H        0.000          V        0.000
ADRS.                                S  0 T0101
21:02:14                                MDI
[OFFSET ][          ][W.SHFT ][ MACRO ][          ]

```

Without tool geometry/wear offset

```

OFFSET / GEOMETRY                    O0001 N0001
NO.      X          Z          R T
G_01     30.500      50.300      0.0000
G 02    -23.580    -100.300      0.0000
G 03     123.850      10.200      0.0000
G 04      55.300    -150.600      0.0000
G 05    -56.800      25.700      0.3003
G 06   -148.300      35.700      0.2504
G 07      45.800     200.500      0.0000
G 08   -159.600       0.400      0.0000
ACTUAL POSITION (RELATIVE)
  U        0.000          W        0.000
  H        0.000          V        0.000
ADRS.                                S  0 T0101
20:55:34                                MDI
[ WEAR ][ GEOM ][W.SHFT ][ MACRO ][          ]

```

With tool geometry offset

OFFSET / WEAR			O0001 N0001
NO.	X	Z	R T
W 01	1.350	5.230	0.0000
W 02	-2.580	3.542	0.0000
W 03	0.843	-0.542	0.0000
W 04	0.245	0.235	0.0000
W 05	-3.124	1.520	0.0353
W 06	-6.369	3.248	0.0854
W 07	-0.587	0.000	0.0000
W 08	-0.258	-0.354	0.0000
ACTUAL POSITION (RELATIVE)			
U	0.000	W	0.000
H	0.000	V	0.000
ADRS.		S	0 T0101
20:54:16		MDI	
[WEAR]	[GEOM]	[W.SHFT]	[MACRO] []

With tool wear offset

- Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or press key and enter the compensation number for the compensation value to be set or changed and press key.
- To set a compensation value, enter a value and press soft key.
TIP is the number of the virtual tool tip (see Programming).
Number of the virtual tool tip may be specified on the geometry compensation screen or on the wear compensation screen.

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 8.

Tool length compensation values can be set using the following functions described in subsequent subsections: direct input of tool offset value, direct-input function B for tool offset measured, and counter input of offset value.

- **Tool offset memory**

16 groups are provided for tool compensation. The number of groups can be optionally extended to 32. For the 0-TTC, the above number of groups can be used for each tool post. Tool geometry compensation or wear compensation can be selected for each group.

- **Disabling entry of compensation values**

In some cases, tool wear compensation or tool geometry compensation values can be prohibited cannot be input because of the settings in bits 0 and 1 of parameter 0078.

- **Displaying radius and number of the virtual tool tip**

The radius and TIP are not displayed if the tool tip radius compensation option is not displayed. Specify the radius using address R and the number of the virtual tool tip using address T.

- **Changing offset values during automatic operation**

When offset values have been changed during automatic operation, bit 2 and of parameter 0013 and bit 4 of parameter 0014 can be used for specifying whether new offset values become valid in the next move command or in the next T code command.

0013 #2	0014 #4	When geometry compensation values and wear compensation values are separately specified	When geometry compensation values and wear compensation values are not separately specified
0	0	Become valid in the next T code block	Become valid in the next T code block
1	0	Become valid in the next T code block	Become valid in the next T code block
0	1	Become valid in the next T code block	Become valid in the next move command
1	1	Become valid in the next move command	Become valid in the next move command

- **Incremental input**

To enable the setting of compensation values, press an incremental command address key (such as u) to set incremental input mode. In this mode, the previous compensation value and the entered value are summed to give the new compensation value.

11.4.2

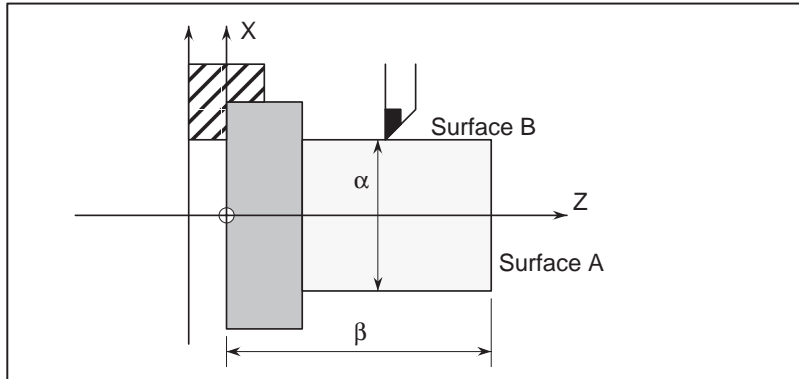
Direct Input of Tool Offset Value

To set the difference between the tool reference position used in programming (the nose of the standard tool, turret center, etc.) and the tool tip position of a tool actually used as an offset value

Procedure for direct input of tool offset value


• Setting of Z axis offset value

- 1 Cut surface A in manual mode with an actual tool.
Suppose that a workpiece coordinate system has been set.



- 2 Release the tool in X axis direction only, without moving Z axis and stop the spindle.
- 3 Measure distance β from the zero point in the workpiece coordinate system to surface A.
Set this value as the measured value along the Z-axis for the desired offset number, using the following procedure:

OFFSET			O0001 N0013	
NO.	X	Z	R	T
01	1.350	5.230	0.0000	
02	-2.580	13.540	0.0000	
03	5.843	-10.256	0.0000	
04	50.245	100.235	0.0000	
05	-3.124	36.520	0.3003	
06	-16.369	53.248	0.2504	
07	-10.587	0.000	0.0000	
08	-0.258	-12.354	0.0000	
ACTUAL POSITION (RELATIVE)				
U	0.000	W	0.000	
H	0.000	V	0.000	
ADRS.			S	0 T0101
21:02:14			MDI	
[OFFSET][][W.SHFT][MACRO][

- 3-1 Press the function key  or the soft key **[OFFSET]**, **[WEAR]**, or **[GEOM]** to display the tool compensation screen.
- 3-2 Move the cursor to the set offset number using cursor keys.

3-3 Press address key and press the address key to be set.

3-4 Key in the measured value (β).

3-5 Press the key.

The difference between measured value β and the absolute coordinate is set as the offset value.

- **Setting of X axis offset value**

4 Cut surface B in manual mode.

5 Release the tool in the Z-axis direction without moving the X-axis and stop the spindle.

6 Measure the diameter α of surface B.
Set this value as the measured value along the X-axis for the desired offset number in the same way as when setting the value along the Z-axis.

7 Repeat above procedure the same time as the number of the necessary tools. The offset value is automatically calculated and set.
For example, in case $\alpha=69.0$ when the coordinate value of surface B in the diagram above is 70.0, set 69.0 at offset No.2.
In this case, 1.0 is set as the X-axis offset value to offset No.2.

Explanations

- **Compensation values for a program created in X axis**

Enter diameter values for the compensation values for X axis.

- **Tool geometry offset value and tool wear offset value**

If measured values are set on the tool geometry compensation screen, all compensation values become geometry compensation values and all wear compensation values are set to 0. If measured values are set on the tool wear compensation screen, the differences between the measured compensation values and the current geometry compensation values become the new compensation values.

- **Retracting along two axes**

If a record button is provided on the machine, after the tool retracts along two axes offset setting can be used when bit 4 of parameter 0015 is set and the record signal is used. Refer to the appropriate manual issued by the machine tool builder.

- **Enabling or disabling the direct input of a tool offset value**

To enable the direct input of a tool offset value, set bit 5 of parameter 010 accordingly.

11.4.3

Direct Input of Tool Offset measured B

The direct input function B for tool offset measured is used to set tool compensation values and workpiece coordinate system shift values.

Procedure for setting the tool offset value

Tool position offset values can be automatically set by manually moving the tool until it touches the sensor.

Refer to the appropriate manual issued by the machine tool builder for actual operation.

- 1 Execute manual reference position return.
By executing manual reference position return, a machine coordinate system is established.
The tool offset value is computed on the machine coordinate system.
- 2 Set the offset writing mode signal to 1.
(Refer to the appropriate manual issued by the machine tool builder for actual operation.)
The CRT display is automatically changed to the tool offset screen (geometry), and the "OFST" indicator starts blinking in the status indication area in the bottom of the screen, which informs that the offset writing mode is ready.
- 3 Select a tool to be measured.
- 4 When the cursor is not coincided with the tool offset number desired to be set, move the cursor to the desired offset number by page key and cursor key.
Besides the cursor can also be coincided with the tool offset number desired to set automatically by the tool offset number input signals (when parameter (No.0024#6)=1).
In this case, the position of the cursor cannot be changed on the tool compensation screen using page keys or cursor keys.
- 5 Near the tool to the sensor by manual operation.
- 6 Place the tool edge to a contacting surface of the sensor by manual handle feed.
Bring the tool edge in contact with the sensor. This causes the offset writing signals (+MIT1, -MIT1, +MIT2 or -MIT2) to input to CNC.
The offset writing signal is set to 1, and the :
 - The axis is interlocked in this direction and its feeding is stopped.
 - The tool offset value extracted by the tool offset memory (tool geometry offset value) which corresponds to the offset number shown by the cursor is set up.
- 7 For both X-axis and Z-axis, their offset value are set by the operations 5 and 6.
- 8 Repeat operations 3 to 7 for necessary tools.
- 9 Set the offset writing signal mode to 0.
The writing mode is canceled and the blinking "OFST" indicator light goes off.

Procedure for setting the work coordinate system shift amount

Tool position offset values can be automatically set by manually moving the tool until it touches the sensor.

Refer to the appropriate manual issued by the machine tool builder for actual operation.

- 1 The tool compensation values are then calculated based on the machine coordinates of the tool.
- 2 Execute manual reference position return.
By executing manual reference position return, the machine coordinate system is established.
The workpiece coordinate system shifting amount is computed based on the machine coordinate system of the tool.
- 3 Set the workpiece coordinate system shifting amount writing signal mode to 1.
(Refer to the appropriate manual issued by the machine tool builder for actual operation.)
The CRT display is automatically switches to the workpiece shifting screen, the "WFST" indicator starts blinking at the status indicator area in the bottom of the screen, which inform that the workpiece coordinate system shifting amount writing mode is ready.
- 4 Select a tool to be measured.
- 5 Check tool offset numbers.
The tool offset number corresponding to the tool required for measurement, shall be set in the parameter (No.122) in advance.
Besides the tool offset number can be set automatically by setting the tool offset number input signal (with parameter (No.0024#6)=1).
Refer to the appropriate manual issued by the machine tool builder for details.
- 6 Manually approach the tool to an end face of the workpiece.
- 7 Place the tool edge to the end face (sensor) of the workpiece by manual handle feed.
The workpiece coordinate system shifting amount on the Z-axis is automatically set.
- 8 Feed the tool.
- 9 Set the workpiece coordinate system shifting amount writing signal mode to 0.
The writing mode is canceled and the blinking "WSFT" indicator light goes off.
(Refer to the appropriate manual issued by the machine tool builder for actual operation.)

11.4.4

Counter Input of Offset value

By moving the tool until it reaches the desired reference position, the corresponding tool offset value can be set.

Procedure for counter input of offset value

- 1 Manually move the reference tool to the reference position.
- 2 Reset the relative coordinates along the axes to 0 (see subsec. 11.1.2).
- 3 Move the tool for which offset values are to be set to the reference position.
- 4 Select the tool compensation screen. Move the cursor to the offset value to be set using cursor keys.

OFFSET			O0001 N0013
NO.	X	Z	R T
01	1.350	5.230	0.0000
02	-2.580	13.540	0.0000
03	5.843	-10.256	0.0000
04	50.245	100.235	0.0000
05	-3.124	36.520	0.3003
06	-16.369	53.248	0.2504
07	-10.587	0.000	0.0000
08	-0.258	-12.354	0.0000
ACTUAL POSITION (RELATIVE)			
U	0.000	W	0.000
H	0.000	V	0.000
ADRS.	S 0 T0101		
21:02:14	MDI		
[OFFSET]	[W.SHFT]	[MACRO]]

- 5 Press address key **X** (or **Z**) and the **INPUT** key.

Explanations

- Geometry offset and wear offset

When the above operations are performed on the tool geometry compensation screen, tool geometry compensation values are input and tool wear compensation values do not change.

When the above operations are performed on the tool wear compensation screen, tool wear compensation values are input and tool geometry compensation values do not change.

- Enabling/disabling the function

Bit 6 of parameter No. 008 is used to enable/disable this function.

11.4.5 Setting the Workpiece Coordinate System Shifting Amount

The set coordinate system can be shifted when the coordinate system which has been set by a G50 command (or G92 command for G code system B or C) or automatic coordinate system setting is different from the workpiece coordinate system assumed at programming.

Procedure for setting the workpiece coordinate system shifting amount

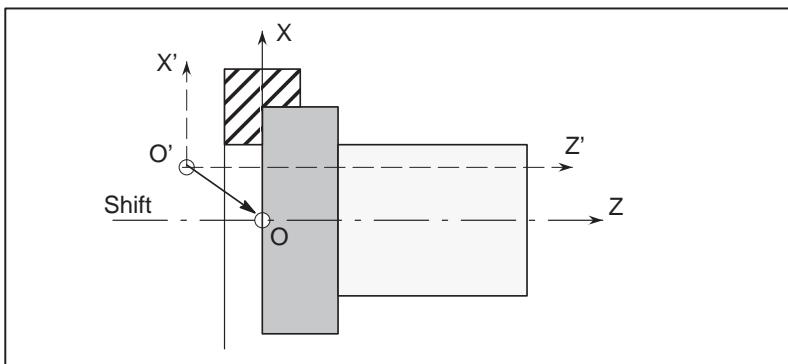
1 Press function key  .

2 Press soft key [WK.SHFT].

WORK SHIFT		O0001 N00013	
(SHIFT VALUE)		(MEASUREMENT)	
X	-2.605	X	63.587
Z	-79.582	Z	120.946
C	0.000	C	0.000
Y	0.000	Y	0.000
ACTUAL POSITION (RELATIVE)			
U	66.192	W	119.448
H	0.000	V	0.000
ADRS.		S 0 T0101	
21:12:37		JOG	
[WEAR][GEOM][WORK][MACRO][]			

3 Move the cursor using cursor keys to the axis along which the coordinate system is to be shifted.

4 Enter the shift value and press  key.



Explanations

- When shift values become valid**

Shift values become valid immediately after they are set.
- Shift values and coordinate system setting command**

Setting a command (G50 or G92) for setting a coordinate system disables the set shift values.

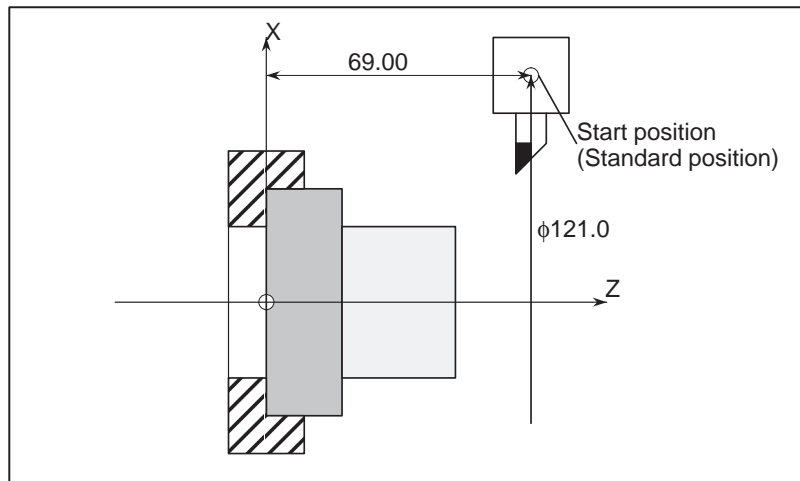
Example When G50 X100.0 Z80.0; is specified, the coordinate system is set so that the current tool reference position is $X = 100.0$, $Z = 80.0$ regardless of the shift values.
- Shift values and coordinate system setting**

If the automatic coordinate system setting is performed by manual reference position return after shift amount setting, the coordinate system is shifted instantly.
- Diameter or radius value**

Whether the shift amount on the X axis is diameter or radius value depends on that specified in program.

Examples

When the actual position of the reference point is $X = \phi 121.0$ (diameter), $Z = 69.0$ with respect to the workpiece origin but it should be $X = \phi 120.0$, $Z = 70.0$, set the following shift values:
 $X=1.0$, $Z=-1.0$



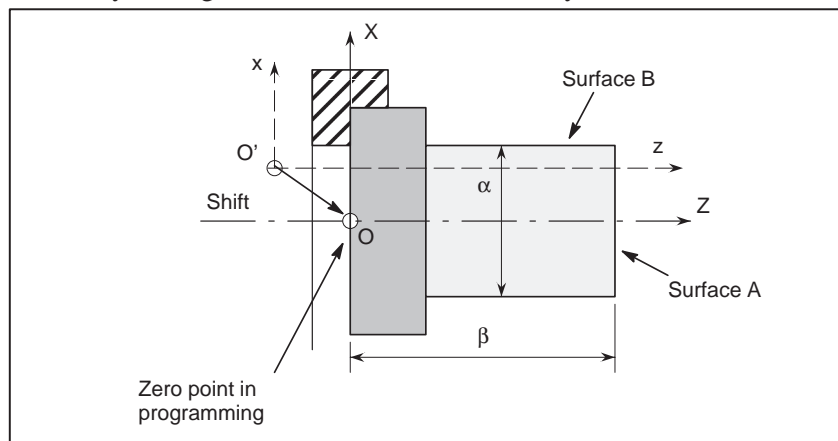
- Incremental input**

Press an incremental command address key, such as the address key for the axis for which the shift is to be set, to set incremental input mode. In this mode, the previous shift and the entered value are summed to give the new shift.

11.4.6

Direct Measured Value Input for Work Coordinate System Shift

When the work coordinate system set with a G50 command or the automatic coordinate system setting function is different from the coordinate system used in programming, the coordinate system can be shifted by storing the measured distance directly as follows.



- 1 Cut the workpiece along surface A using the standard tool in manual operation.
- 2 Release the tool only in X axis direction without moving Z axis and stop the spindle.
- 3 Measure distance “ β ”, distance between zero point to surface A, in the figure above. And store this value in the work coordinate system shift memory.
 - 3-1 Push function key MENU
OFFSET, and soft key **[W. SHT]** then display the WORK SHIFT screen.
 - 3-2 Push address key M to input measured distance.
 - 3-3 Push address key Z.
 - 3-4 Input measured distance (β)

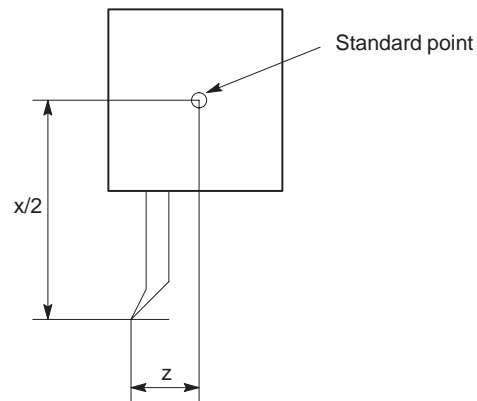
WORK SHIFT		O0001 N00013	
(SHIFT VALUE)		(MEASUREMENT)	
X	-2.605	X	63.587
Z	-79.582	Z	120.946
C	0.000	C	0.000
Y	0.000	Y	0.000
ACTUAL POSITION (RELATIVE)			
U	66.192	W	119.448
H	0.000	V	0.000
ADRS.			
21:12:37		JOG	
[WEAR][GEOM][WORK][MACRO][]

3-5 Push button.

- 4 Cut surface B in manual mode.
- 5 Release the tool in Z axis direction without moving X axis and stop the spindle.
- 6 Measure the diameter a at surface B. And input this distance as X value in the work coordinate system memory.
The shift amount, $0'$ to 0 , is automatically set to the work coordinate system memory. The work coordinate system is shifted immediately and coincides with the coordinate system expected at programming. If the offset amount of the standard tool is zero, it is said that a work coordinate system is set, where the coordinate values of the tool tip is $X=0.0$ and $Z=0.0$ when the tip of the standard tool is positioned at the zero point of the coordinate system.

NOTE

when the tool offset amount (x, z) has already been set on the standard tool as in the figure above and the tool offset function is effective and the direct measured value input for the work coordinate system shift is performed, a work coordinate system is set in which the coordinate values of the standard point are $X=0.0$ and $Z=0.0$ when the standard point is positioned at the zero point in the work coordinate system.





11.4.7 Y Axis Offset

Tool position offset values along the Y-axis can be set. Counter input of offset values is also possible.

Direct input of tool offset value and direct input function B for tool offset measured are not available for the Y-axis.

Procedure for setting the tool offset value of the Y axis

- 1 Press function key .
- 2 Press the continuous menu key , and switch soft keys.
- 3 Press soft key **[OFST]**, **[WEAR]**, or **[GEOM]**.
The Y axis offset screen is displayed.
When **[OFST]** is spressed ;

OFFSET		O0001 N0013	
NO.	Y		
01	1.350		
02	-2.580		
03	5.843		
04	50.245		
05	-3.124		
06	-16.369		
07	-10.587		
08	-0.258		
ACTUAL POSITION (RELATIVE)			
U	0.000	W	0.000
H	0.000	V	0.000
ADRS.			
21:04:41		MDI	
[TOOLLF]	[OFFSET]][][]

When soft key **[WEAR]** or **[GEOM]** is pressed ;

OFFSET /WEAR		O0001 N0001	
NO.	Y		
01	1.350		
02	-2.580		
03	5.843		
04	50.245		
05	-3.124		
06	-16.369		
07	-10.587		
08	-0.258		
ACTUAL POSITION (RELATIVE)			
U	0.000	W	0.000
H	0.000	V	0.000
ADRS.			
20:57:16		MDI	
[TOOLLF][WEAR][GEOM][]

When soft key **[GEOM]** is pressed ;

OFFSET / GEOMETRY		O0001 N0001	
NO.	Y		
01	30.500		
02	-23.580		
03	123.850		
04	55.300		
05	-56.800		
06	-148.300		
07	45.800		
08	-159.600		
ACTUAL POSITION (RELATIVE)			
U	0.000	W	0.000
H	0.000	V	0.000
ADRS.			
20:58:05		MDI	
[TOOLLF]	[WEAR]	[GEOM]	[]

- 4 Position the cursor at the offset number to be changed by using either of the following methods:

- Move the cursor to the offset number to be changed using page keys and cursor keys.

- Press key and type the offset number and press key.

- 5 Type the offset value.

- 6 Press key. The offset value is set and displayed.

Procedure for counter input of the offset value

To set relative coordinates along the Y-axis as offset values:

- 1 Move the reference tool to the reference point.
- 2 Reset relative coordinate Y to 0 (see subsec. 11.1.2).
- 3 Move the tool for which offset values are to be set to the reference point.
- 4 Move the cursor to the value for the offset number to be set, press , .


Relative coordinate Y (or V) is now set as the offset value.

11.4.8






Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59) and external workpiece origin offset. The workpiece origin offset 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 chapter selection soft key **[W.SHIFT]** and **[WORK]**.
The workpiece coordinate system setting screen is displayed.

WORK COORDINATES				O0001 N0013			
NO.		(SHIFT)		NO.		(G55)	
00	X	-2.605		02	X	-153.480	
	Z	-79.582			Z	-304.600	
	C	0.000			C	0.000	
	Y	0.000			Y	0.000	
NO.		(G54)		NO.		(G56)	
01	X	-125.300		03	X	-211.350	
	Z	-251.200			Z	-295.470	
	C	0.000			C	0.000	
	Y	0.000			Y	0.000	
ADRS.							
21:17:11				MDI			
[WEAR] [GEOM] [W.SHFT] [MACRO] []							

- 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) and press  key.
- 4 Move the cursor to the workpiece origin offset to be changed.
- 5 Press address key to set and enter a desired value by pressing numeric keys, then press  key. The entered value is specified in the workpiece origin offset value.
- 6 Repeat 4 and 5 to change other offset values.

11.4.9 Displaying and Setting Custom Macro Common Variables

Displays common variables on the CRT. 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 Soft key **[MACRO]**. The following screen is displayed:

VARIABLE		O0001 N0013	
NO.	DATA	NO.	DATA
100	01000.000	108	
101	0.000	109	40000.000
102	-50000.000	110	00.150846
103		111	00001.000
104	1238501.0	112	02000.000
105	0.000	113	00071.232
106	0.000	114	00024.864
107		115	
ACTUAL POSITION (RELATIVE)			
U	71.232	W	24.864
H	0.000	V	0.000
NO. 144=		S	0 T0101
21:19:59		MDI	
[WEAR]	[GEOM]	[W.SHFT]	[MACRO] []

3 Move the cursor to the variable number to set using either of the following methods:

- Press  key and enter the variable number and 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  with pressing  key, then press  key.

6 To set a blank in a variable. Press  key after pressing  key. The value field for the variable becomes blank.

11.4.10 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. Tool data (execution data) can be reset or cleared. 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 the continuous menu key  to display chapter selection soft key [TOOLLF].
- 3 Press soft key [TOOLLF].
- 4 One page displays data on two groups. Pressing page key  or  successively displays data on the following groups.

```

TOOL LIFE DATA :                                00001 N0001
                                           SELECTED GROUP 001
GROUP 001 :   LIFE  9800   COUNT 6501
*0034   *0078   *0012   *0056
*0090   *0076   *0032   *0098
@0054   0010   0099   0087
0077   0065   0043   0096
GROUP 002 :   LIFE  0540   COUNT 0200
*0011   *0022   *0201   *0144
*0155   #0066   0176   0188
0019   0234   0007   0112
0156   0090   0016   0232
TO BE CHANGED :  006 012 013 014 --->
NUM.                                S  0 T0101
21:28:37                                MDI
[ WEAR  ][ GEOM  ][W.SHFT ][ MACRO  ][      ]

```

- 5 To display the page containing the data for a group, Press  key and enter the group number and press  key. The cursor can be moved to an arbitrary group by pressing cursor key  or .
- 6 To change the value in the life counter for a group, move the cursor to the group, enter a new value (four digits), and press . The life counter for the group indicated by the cursor is preset to the entered value. Other data for the group is not changed.

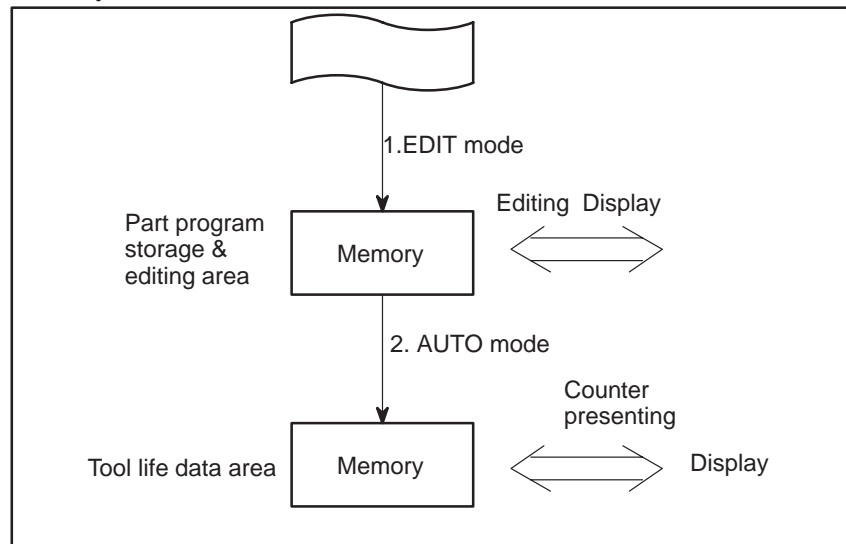
- 7 To reset the tool data, move the cursor on the group to reset, then press the -9999 keys in this order.

All execution data for the group indicated by the cursor is cleared together with the marks (@, #, or *).

Explanations

• Registering tool life management data

The tool life management data must be executed to register it in the CNC memory.



- 1 Load the program for tool life management in the EDIT mode, as with an ordinary CNC tape.
The program will be registered in the part program memory and will be made ready for display and editing.
- 2 Perform a cycle start operation in the AUTO mode to run the program.
The data will be stored in the tool life data area of the memory; at the same time, the already existing tool life data of all groups will be cancelled and the life counters will be cleared. Data once stored is not erased by turning the power off.

• Display contents

```

TOOL LIFE DATA :                                O0001 N0001
                                           SELECTED GROUP 001
GROUP 001 :   LIFE  9800   COUNT 6501
  *0034   *0078   *0012   *0056
  *0090   *0076   *0032   *0098
  @0054    0010    0099    0087
    0077    0065    0043    0096
GROUP 002 :   LIFE  0540   COUNT 0200
  *0011   *0022   *0201   *0144
  *0155   #0066    0176    0188
    0019    0234    0007    0112
    0156    0090    0016    0232
  TO BE CHANGED :  006 012 013 014 --->
NUM.                                S  0 T0101
21:28:37                            MDI
[ WEAR  ][ GEOM  ][W.SHFT ][ MACRO  ][      ]

```

- 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.0039#2) as either minutes(or hours) or number of times used.
In lines 4 to 7, tool numbers are displayed. In this case, the tool is selected in the order, 0034 → 0078 → 0012 → 056 → 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



When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.

This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the Handy File (see Chapter 9).

In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed. The setting data for operating the machine can also be displayed and set.

See Chapter 7 for the diagnostic screens displayed by pressing function

key  .

11.5.1 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.

See Chapter 9 for automatic insertion of sequence numbers.

See subsec 11.5.2 for the sequence number comparison and stop function. This subsection describes how to set data.

Procedure for setting the setting data

1 Select the **MDI** mode.

2 Press function key .

3 Press soft key **[W.PRM]**.

This screen consists of several pages.

4 Press  key, and press  key,  key.

To switch each screen of setting data, press page key  or .

PARAMETER		O0001 N0001	
(SETTING 1)			
TVON	= 1		
ISO	= 1	(0:EIA 1:ISO)	
INCH	= 0	(0:MM 1:INCH)	
I/O	= 0		
SEQ	= 1		
		CLOCK	97/05/16
			12:11:52
NO. TVON			
21:35:25		MDI	
[PARAM]	[DGNOS]	[SV-PRM]	[]

PARAMETER		O0001 N0001	
(SETTING 2)			
_PWE	= 0	(0:DISABLE 1:ENABLE)	
TAPEF	= 0		
(SEQUENCE STOP)			
PRGNO	= 0		
SEQNO	= 0		
PART TOTAL	=	23	
PART REQUIRED	=	0	
PART COUNT	=	23	
RUN TIME	3H36M	CYCLE TOME	0H 0M 0S
NO. PWE			
21:36:25		MDI	
[PARAM]	[DGNOS]	[SV-PRM]	[]

- 5 Move the cursor to the item to be changed by pressing cursor keys



- 6 Enter a new value and press key.

Contents of settings

• TV ON

Setting to perform TV check.

0 : No TV check

1 : Perform TV check

• ISO

Setting code when data is output through reader puncher interface.

0 : EIA code output

1 : ISO code output (To output to floppy cassette, be sure to set to 1.)

• INCH

Setting a program input unit, inch or metric system

0 : Metric

1 : Inch

• I/O

Using channel of reader/puncher interface.

0 : Channel 1

1 : Channel 1

2 : Channel 2

3 : Channel 3

• SEQUENCE STOP

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.

• PWE

Setting whether parameter writing is enabled or disabled.

0 : Disabled

1 : Enabled (P/S alarm No.100 is displayed.)

- **TAPEF**


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.






- **Time**

To set the time, move the cursor to date/time, enter desired data, then press the  key. The date is displayed in the year/month/day format. The time is displayed in the hour:minute:second format (24-hour clock format).

11.5.2 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

- 1 Select the **MDI** mode.
- 2 Press function key  .
- 3 Press chapter selection soft key **[PARAM]**.
- 4 Press  key and press  key,  key.
- 5 Press page key  to display the following screen.

PARAMETER		O0001 N0001	
(SETTING 2)			
_PWE	= 0	(0:DISABLE 1:ENABLE)	
TAPEF	= 0		
(SEQUENCE STOP)			
PRGNO	= 0		
SEQNO	= 0		
PART TOTAL	=	23	
PART REQUIRED	=	0	
PART COUNT	=	23	
RUN TIME	3H36M	CYCLE TOME	0H 0M 0S
NO. PWE			
21:36:25		MDI	
[PARAM]	[DGNOS]	[SV-PRM]	[]

- 6 Enter in (PRGNO) for SEQUENCE STOP the number (1 to 9999) of the program containing the sequence number with which operation stops.
- 7 Enter in (SEQNO) for SEQUENCE STOP (with four or less digits) the sequence number with which operation is stopped.
- 8 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 block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

Example

```

N1 #1=1 ;
N2 IF [#1 EQ 1] GOTO 08 ;
N3 GOTO 09 ;
N4 M98 P1000 ;
N5 M99 ;

```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.
- **Stop in the canned cycle**

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.

11.5.3 Displaying and Setting Run Time, Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed. This data can be set by parameters or on this screen.

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  to display the following screen.

PARAMETER
O0001 N0001

(SETTING 2)

_PWE = 0 (0:DISABLE 1:ENABLE)

TAPEF = 0

(SEQUENCE STOP)

PRGNO = 0

SEQNO = 0

PART TOTAL = 23

PART REQUIRED = 0



PART COUNT = 23

RUN TIME 3H36M CYCLE TOME 0H 0M 0S

NO. PWE

21:36:25 MDI

[PARAM][DGNOS][][SV-PRM][]

- 5 To set the number of parts required and parts count, move the cursor to PARTS REQUIRED or PARTS COUNT and enter the number of parts, and press  key.
- 6 To set the operationg time or cutting time, move the cursor to DATE or TIME, enter a new hour/minute/second, 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 0779.

• 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. 0600).

• 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.
- **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.

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. 0040#3) is set to 1). For details, see the manual issued by machine tool builders.

11.5.4 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.

Normally, the user need not change parameter setting.









Procedure for displaying and setting parameters

- 1 Press function key  .
- 2 Press chapter selection soft key **[PARAM]** to display the parameter screen.

PARAMETER		O0001 N0013	
NO.	DATA	NO.	DATA
0001	00000000	0011	00000000
0002	00000001	0012	00000000
0003	00000000	0013	00000000
0004	01110111	0014	00000100
0005	01110111	0015	00000000
0006	01110111	0016	00000000
0007	01110111	0017	11111111
0008	00000000	0018	00000000
0009	00000000	0019	10000000
0010	11100000	0020	00000000

NO. 0001 =
21:39:31

MDI
[PARAM][DGNOS][SV-PRM]

- 3 Move the cursor to the parameter number to be set or displayed in either of the following ways:
 - Press  key and enter the parameter number and press  key.
 - Move the cursor to the parameter number using the page keys,  and  , and cursor keys,  ,  .
- 4 To set the parameter set 1 for **PWE (PARAMETER WRITE)** to enable writing. See the procedure for setting data writing described below.
- 5 Enter a new value with numeric keys and press  key in the MDI mode. The parameter is set to the entered value and the value is displayed.
- 6 Set 0 for **PWE (PARAMETER WRITE)** to disable writing.
- 7 To release the alarm, press the  key.

Explanations

- **Setting parameters with external input/output devices**
See Chapter 8 for setting parameters with external input/output devices such as the Handy File.
- **Parameters that require turning off the power**
Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm 000. In this case, turn off the power, then turn it on again.

11.5.5 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.

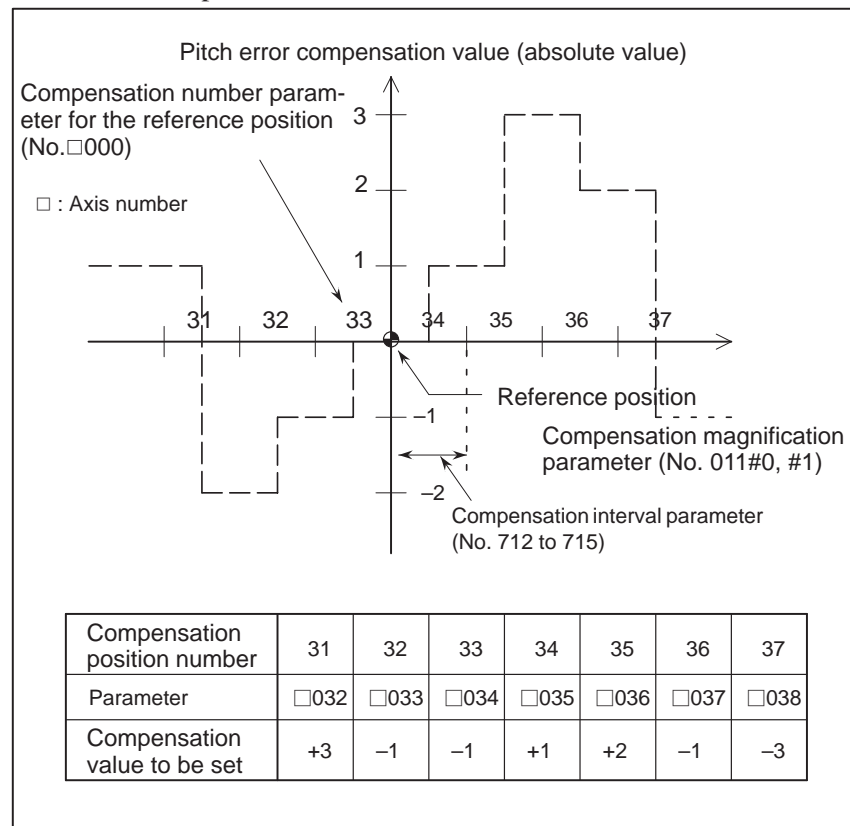
The pitch error compensation data is set according to the characteristics of the machine. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced.

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 8). Compensation data can also be written directly with the MDI panel.

The following parameters must be set for pitch error compensation. Set the pitch error compensation value 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) / 1875
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.

Examples

• 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

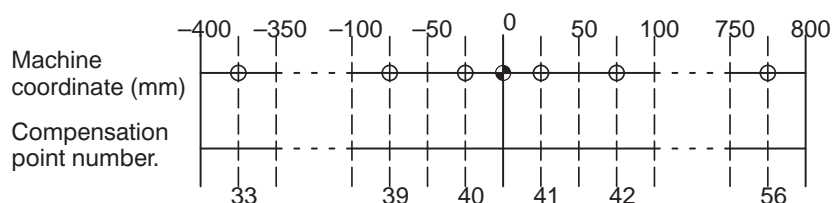
If the above is specified, the No. of the farthest compensation point in the negative direction is as follows:

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:



Compensation values are output at the positions indicated by ○ .

Therefore, set the parameters as follows:

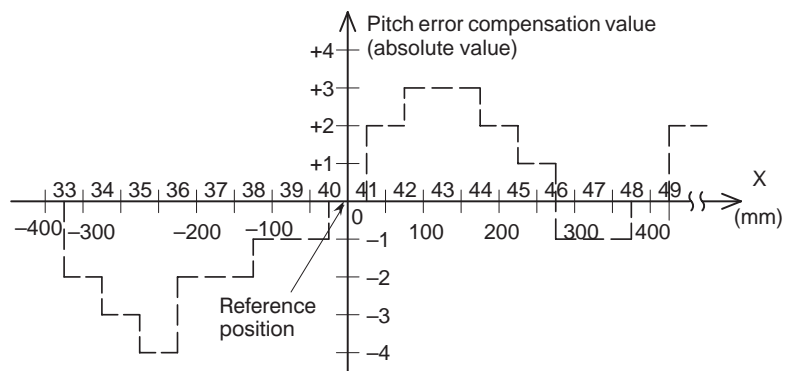
Parameter	Setting value
1000 : Compensation number for the reference position	40
011#0, #1 : Compensation magnification	#0=0, #1=0
0712 : Interval between pitch error compensation points	50000

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

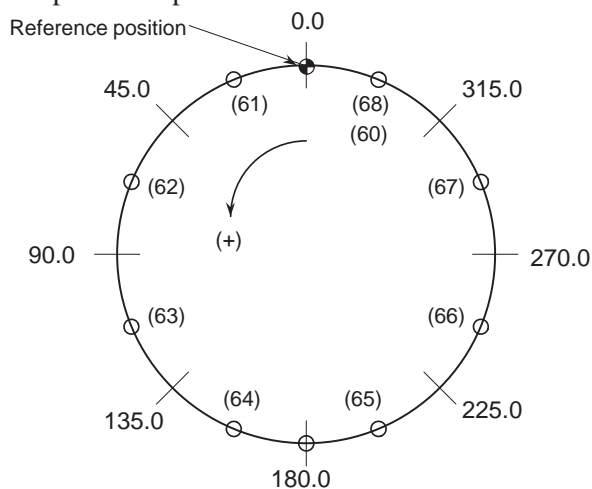
If the above is specified, the No. of the farthest compensation point in the negative direction for the rotating axis is always equal to the compensation point No. of the reference position.

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:



Compensation values are output at the positions indicated by ○.

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.

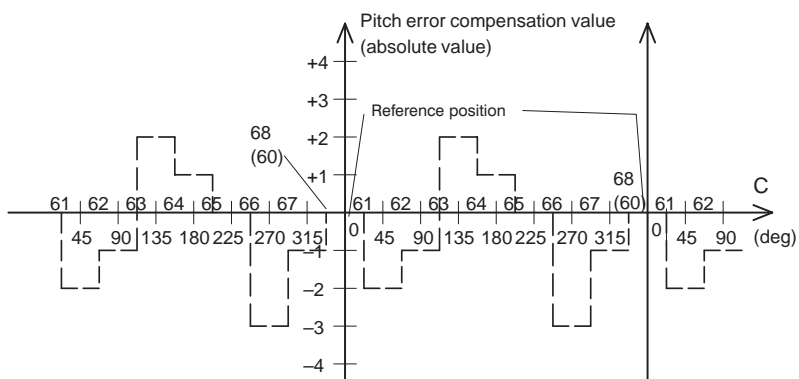
The same value must be set for compensation points 60 and 68.

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



11.6 SCREENS DISPLAYED BY FUNCTION KEY




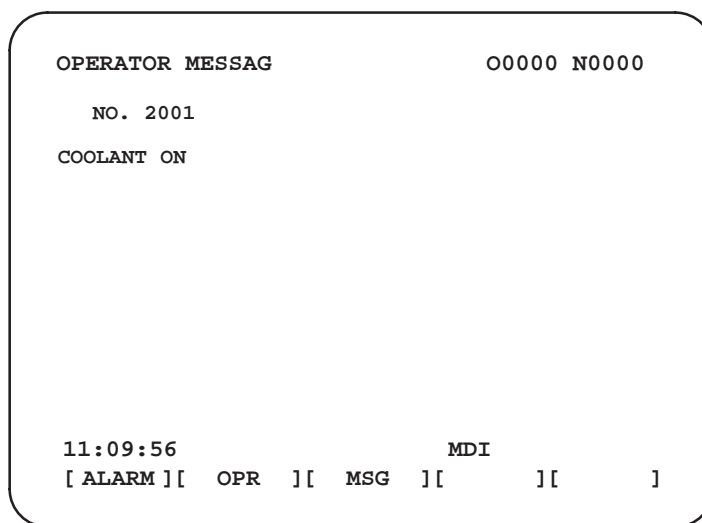
The alarm message and operator message can be displayed by pressing the  key. The software operator's panel can also be displayed and specified. For details of how to display the alarm message, see Chapter 7.

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 
- 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 chapter selection soft key **[OPR]**.
- 3 The screen consists of several pages.
Press page key  or  until the desired screen is displayed.

```
OPERATOR'S PANEL                                O0001 N0001
```

_MODE : MDI ■ AUTO EDIT HNDL JOG ZRN
HANDLE AXIS : ■ HX HZ
HANDLE MULT. : ■ *1 *10 *100
RAPID OVRD. : ■ 100% 50% 25% F0
JOG FEED : 500 MM/MIN

 * *
FEED OVERD : 90%
 * * * * *

ACTUAL POSITION (ABSOLUTE)
 X 164.000 Z 182.000
 C 0.000 Y -30.500

21:37:41 MDI
[ALARM] [OPR] [MSG] [MACRO] []

```
OPERATOR'S PANEL                                O0001 N0001
```



```
_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	164.000	Z	182.000
C	0.000	Y	-30.500

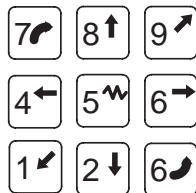

```
21:38:35                                     MDI
```

```
[ ALARM ][ OPR ] [ MSG ] [           ]
```

- 4 Move the cursor to the desired switch by pressing cursor key  or  .

5 Push the numerical key  or  to match the mark  to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the  key together with an arrow key to perform jog rapid traverse.



Explanations

• Valid operations

The valid operations on the software operator's panel are shown below. Whether to use the CRT or machine operator's panel for each group of operations can be selected by parameter 0017.

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 0017 are not displayed on the software operator's panel.

• Screens on which jog feed is valid

When the CRT 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. 0130 to 0137).

• 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.

```

PROGRAM                                O0001 N0000
Q0001 T0101 ;
S550 M08 ;
M45 ;
N010 G50 X200. Z200. ;
N011 G00 X160. Z180. ;
N012 G71 U7. R1. ;
N013 G71 P014 Q020 U4. W2. F0.3 S550 M03 ;
N014 G00 X40. F0.15 S700 ;

<
16:09:08                                EDIT
[ PRGRM ][ LIB ][FLOPPY ][           ][ C.A.P. ]

```

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 display

```

NOT READY          S    500 T0101
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



The graphic function indicates how the tool moves during automatic operation or manual operation.

12.1 GRAPHICS DISPLAY

It is possible to draw the programmed tool path on the 9-inch or 14-inch CRT screen, 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.

The drawing coordinates (parameter) and graphic parameters must be set before a tool path can be displayed.

With 0-TTC, the tool paths of two tool posts are displayed on the same screen, one on the right and the other on the left.

Graphics display procedure


Procedure

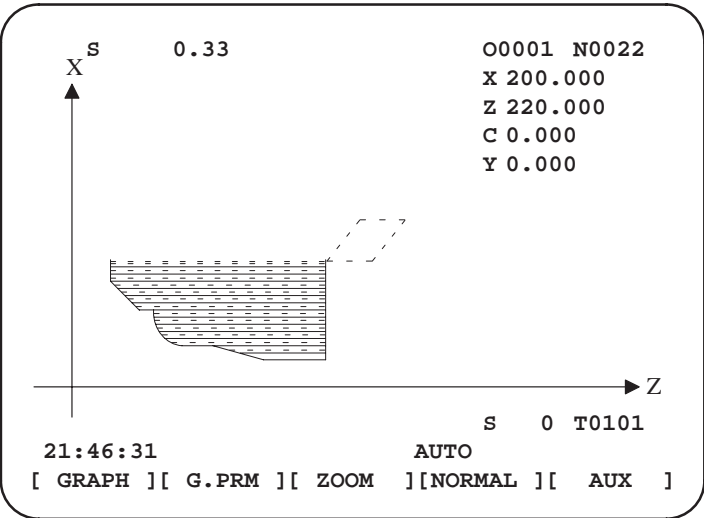
Set the drawing coordinates with parameter No.0123 before starting drawing. See "Drawing Coordinate System" for the settings and corresponding coordinates.

- 1 Press function key  .

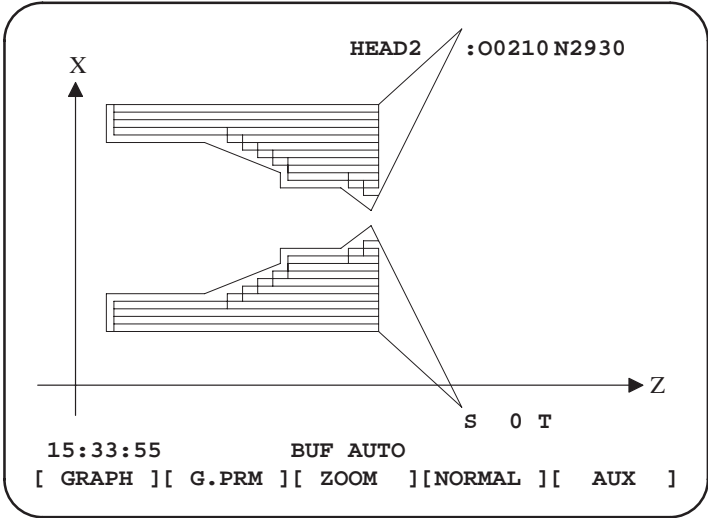
The graphic parameter screen shown below appears.

GRAPHIC PARAMETER		O0001 N0022
WORK LENGTH	W =	250000
WORK DIAMETER	D =	250000
PROGRAM STOP	N =	0
AUTO ERASE	A =	0
LIMIT	L =	0
GRAPHIC CENTER	X =	59479
	Z =	175019
SCALE	S =	33
GRAPHIC MODE4	M =	0
NO. N =	S	0 T0101
21:43:29	AUTO	
[GRAPH]	[ZOOM]	[AUX]

- 2 For the 0-TTC, determine for which tool post the data is specified, using a tool post select signal.
Specify the PROGRAM STOP (N), AUTO ERASE (A), and GRAPHIC CENTER (X,Y) parameters separately for each tool post. The other parameters are common to both tool posts. It does not matter for which tool post they are specified first.
- 3 Move the cursor with the cursor keys to a parameter to set.
- 4 Enter data, then press the  key.
- 5 Repeat steps 3 and 4 until all required parameters are specified.
- 6 Press soft key **[GRAPH]**.
- 7 Automatic or manual operation is started and machine movement is drawn on the screen.





0-TC

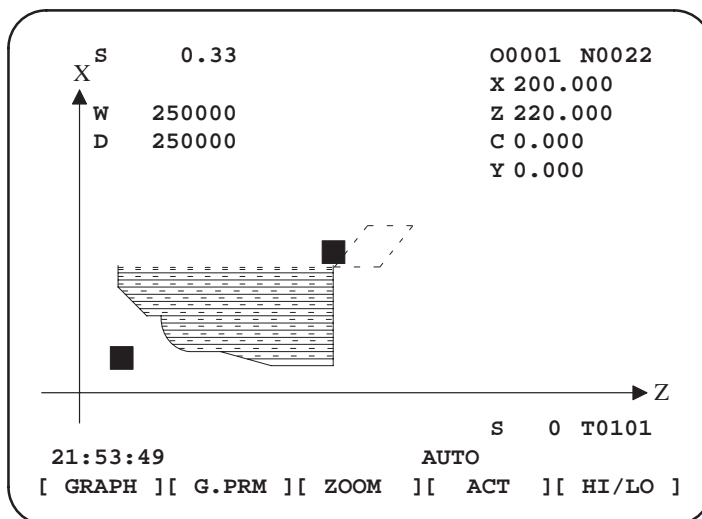


0-TTC





• Magnifying drawings

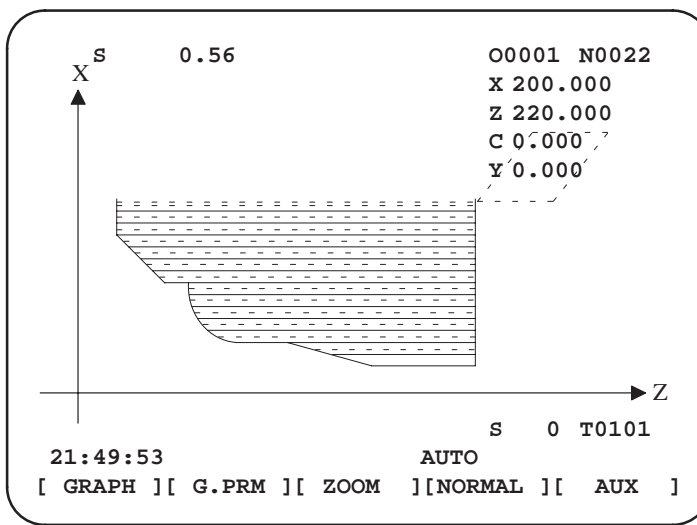
Part of a drawing on the screen can be magnified.

- 8 Press the  function key, then the **[ZOOM]** soft key to display a magnified drawing. The magnified-drawing screen contains two zoom cursors ()



A rectangle that has one of its diagonals defined by the two zoom cursors is magnified to the full size of the screen.

- 9 Using the numerical keys    , move the zoom cursors to specify a diagonal for the new screen. Pressing the **[HI/LO]** soft key toggles the zoom cursor to be moved.
- 10 To make the original drawing disappear, press **[EXEC]**.
- 11 Resume the previous operation. The part of the drawing specified with the zoom cursors will be magnified.

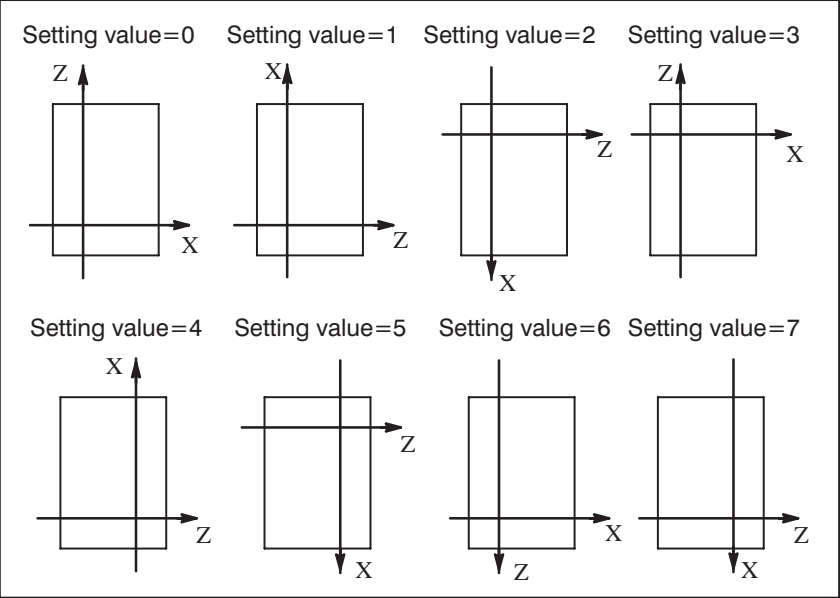


- 12 To display the original drawing, press the **[NORMAL]** soft key, then start automatic operation.

Explanation

● Setting drawing coordinate systems

Parameter No. 0123 is used to set a drawing coordinate system for using the graphic function. The relationships between setting values and drawing coordinate systems are indicated below. With 0–TTC, a different drawing coordinate system can be selected for each tool post.



● Graphics parameter

·WORK LENGTH (W), WORK DIAMETER (D)

Specify work length and work diameter. The table below lists the input unit and valid data range.

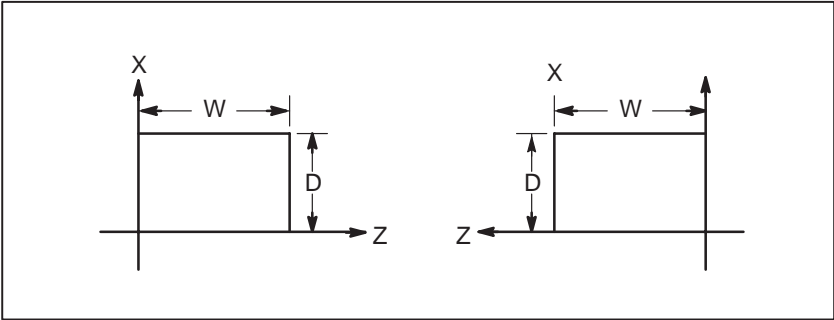


Table 12.1 Unit and Range of Drawing Data

Increment system	Unit		Valid range
	mm input	Inch input	
IS-B	0.001 mm	0.0001 inch	0 to 99999999
IS-C	0.0001 mm	0,00001 inch	

·GRAPHIC CENTER (X, Z), SCALE (S)

A screen center coordinate and drawing scale are displayed. A scale screen center coordinate are automatically calculated so that a figure set in WORK LENGTH (a) and WORK DIAMETER (b) can be fully displayed on the screen. So, the user need not set these parameters usually.

A screen center coordinate is defined in the workpiece coordinate system. Table 12. 3. 2 indicates the unit and range. The unit of SCALE is 0.001%.

·PROGRAM STOP (N)

Set the sequence number of an end block when part of the program is to be drawn. A value set in this parameter is automatically cancelled (cleared to 0) once a drawing is provided.

·AUTO ERASE (A)

If 1 is set, the previous drawing is automatically erased when automatic operation is stated from the reset state. Then, drawing is started.

·LIMIT (L)

If 1 is set, the area of stored stroke limit l is drawn with double-dot-and-dash lines.

NOTE

The parameter values for drawing are preserved even if power is turned off.

- **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**

Pressing the **[GRAPH]** soft key after graphic parameter is modified then tool paths are deleted. Setting the graphic parameter as AUTO ERASE (A) = 1 specifies that when MEMORY operation is started at reset, program execution begins after the previous drawing is erased automatically (AUTO ERASE = 1).

- **Drawing a part of a program**

When necessary to display 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.

- **Displaying coordinates**

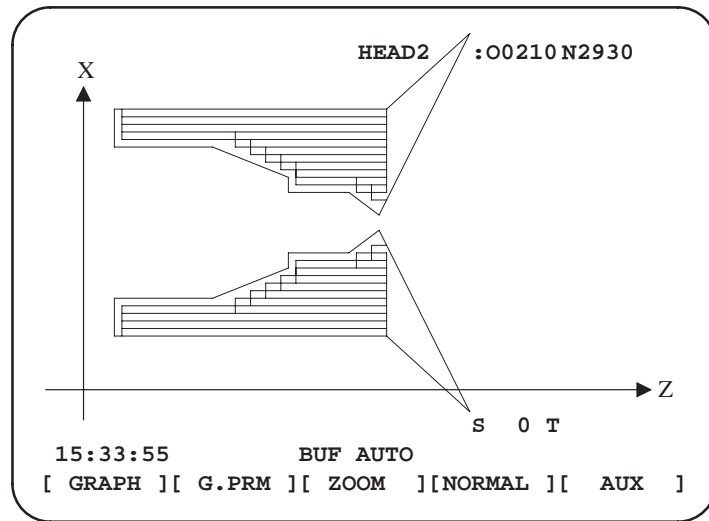
The displayed drawing is indicated with coordinates in a work coordinate system.

- **Switching from a drawing screen to another screen**

Even if the screen is switched to a non-drawing screen, drawing continues. When the drawing screen is displayed again, the entire drawing appears (no parts are missing).

- **Drawing for tool posts 1 and 2 (0-TTC)**

For the 0-TTC, bit 1 of parameter No.0047 can be used to determine whether the tool path for each tool post is to be drawn on a separate screen or whether the tool paths for both tool posts are to be drawn on the same screen.



When the tool paths of the both tool posts are to be displayed on one screen, parameter No. 123 for tool post 1 is used to set a coordinate system for drawing.

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.

- **Zooming drawings**

If the WORK and DIAMETER graphic parameters are not set correctly, the drawing cannot be magnified. To reduce a drawing, specify a negative value for the SCALE graphic parameter. The machine zero point is indicated with a question mark.

13

DISPLAY AND OPERATION OF 00-TC/00-GCC

The CRT/MDI panel of 00-TC/00-GCC 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.

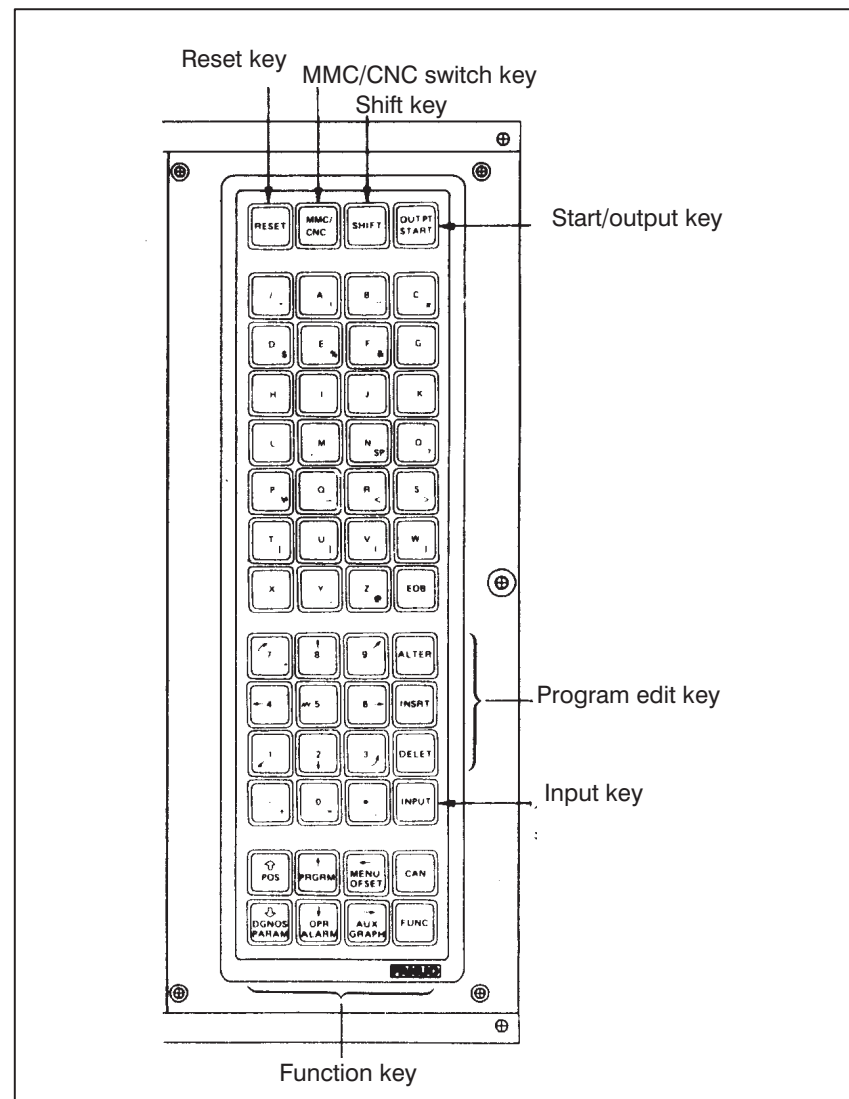


Fig. 13 CRT/MDI for 00-TC

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–TC. 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 in the Fig. 13.1. Display items of this section cannot be changed by the screen selection operation. However, its display contents are always renewed.

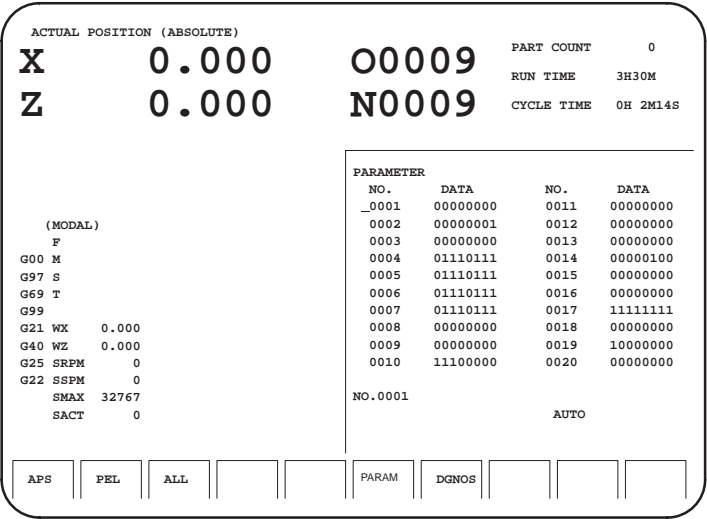










Fig. 13.1

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-TC. However, inputting data is exactly the same as that of 00-TC/00-GCC. 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  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 a number is specified with a parameter display and so forth, please use the cursor key instead of  key.

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

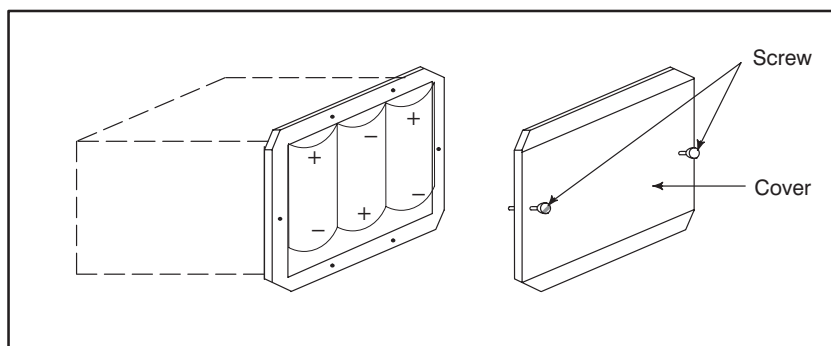
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

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.



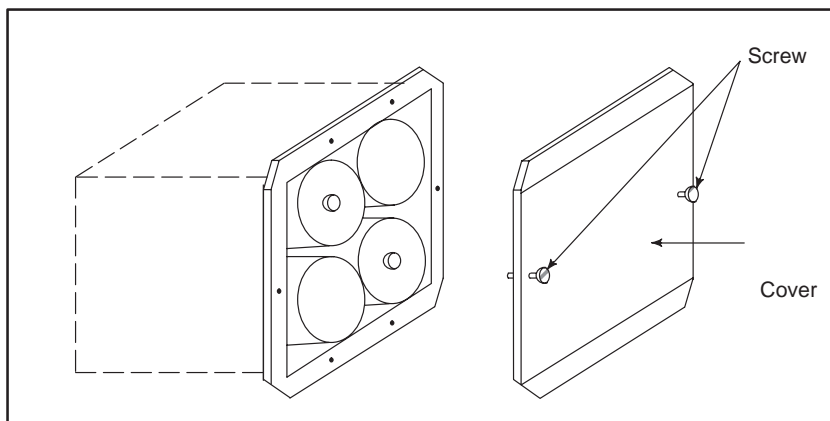
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

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)											
\$			○			○	○		—										△	○
&	○		○			○	○	○	&					○	○	○	○		△	○
▼			○			○	○	○	—										△	○
*	○		○		○	○		○	Parameter (No.0042)										△	
,	○		○		○	○	○		,			○	○	○	○		○			
;	○		○	○	○	○		○	—										△	△
<			○	○	○	○	○		—										△	△
=	○		○	○	○	○		○	Parameter (No.0043)										△	
>	○		○	○	○	○	○	○	—										△	△
?			○	○	○	○	○	○	—										△	○
@	○	○				○			—										△	○
"			○					○	—										△	△
[○	○			○	○	○	○	Parameter (No.0053)										△	
]	○	○			○	○	○	○	Parameter (No.0054)										△	

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

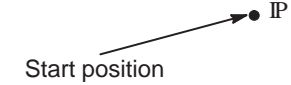
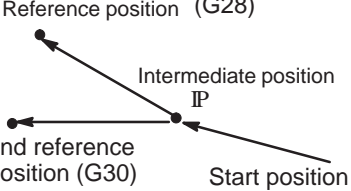
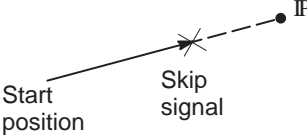
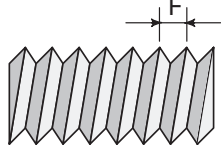
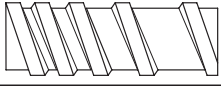
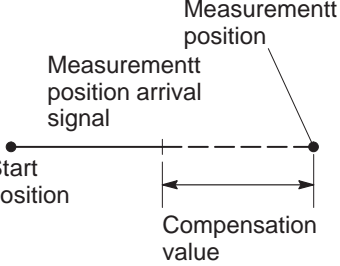
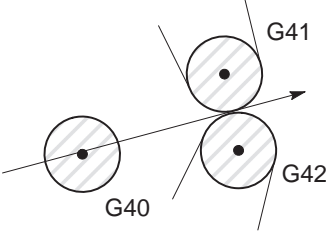
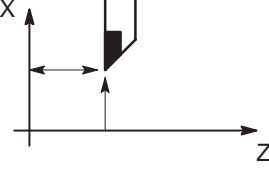
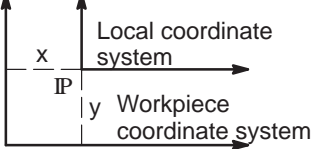
Some functions cannot be added as options depending on the model.

In the tables below, IP _; presents a combination of arbitrary axis addresses using X and Z.

x = 1st basic axis (X usually)

z = 2nd basic axis (Z usually)

Functions	Illustration	Tape format
Positioning (G00)		G00 IP _ ;
Linear interpolation (G01)		G01 IP _ F _ ;
Circular interpolation (G02, G03)		$\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_ Z_ \left\{ \begin{matrix} R_ \\ I_ K_ \end{matrix} \right\} F_ ;$
Dwell (G04)		G04 $\left\{ \begin{matrix} X_ \\ P_ \end{matrix} \right\} ;$
Change of offsetvalue by program (G10)		Tool geometry offset value G10 P_ X_ Z_ R_ Q_ ; P=1000+Geometry offset number Tool wear offset value G10 P_ X_ Z_ R_ Q_ ; P=Wear offset number
Inch/metric conversion (G20, G21)		Inch input : G20 Metric input : G21
Expanded stored stroke check (G22, G23)		G22 X_ Z_ I_ K_ ; G23 ; Cancel
Spindle speed fluctuation detection (G25, G26)		G25 ; G26 P_ Q_ R_ ;

Functions	Illustration	Tape format
Reference position return check (G27)		G27 IP_ ;
Reference position return (G28) 2nd, reference position return (G30)		G28 IP_ ; G30 IP_ ;
Skip function (G31)		G31 IP_ F_ ;
Thread cutting (G32)		Equal lead thread cutting G32 IP_ F_ ;
Variable lead thread cutting (G34)		G34 IP_ F_ K_ ;
Automatic tool compensation (G36, G37)		G36 X ; G37 Z ;
Cutter compensation (G40, G41, G42)		{ G41 } { G42 } IP_ ; G40 : Cancel
Coordinate system setting Spindle speed setting (G50)		G50 IP_ ; Coordinate system setting G50 S_ ; Spindle speed setting
Local coordinate system setting (G52)		G52 IP_ ;
Machine coordinate system selection (G53)		G53 IP_ ;

Functions	Illustration	Tape format
Workpiece coordinate system selection (G54 to G59)		$\left\{ \begin{array}{c} G54 \\ \vdots \\ G59 \end{array} \right\} IP_;$
Custom macro (G65, G66, G67)		One –shot call G65 P_ <argument> ; P : Program number Modal call G66 P_ <argument> G67 ; cancel
Balance cut (G68, G69)		G68 ; Balance cut mode G69 ; Mode is canceled
Canned cycles for drilling (G80 to G89)	Refer to the section of canned cycles for drilling of OPERATOR'S MANUAL	G80 ; Cancel $\left\{ \begin{array}{c} G83 \\ G84 \\ G85 \\ G87 \\ G88 \\ G89 \end{array} \right\} x_z_C_P_Q_R_F_K_;$
Mirror image for double turret (G68, G69)		G68 ; Mirror image for double turret on G69 ; Mirror image cancel
Feed per minute (G98) Feed per revolution (G99)	mm/min inch/min mm/rev inch/rev	G98 ... F_ ; G99 ... F_ ;
Constant surface speed control (G96/G97)	m/min or feet/min 	G96 S_ ; G97 ; Cancel
Chamfering, Corner R		$X_; \left\{ \begin{array}{c} C(K) \pm k \\ R_ \end{array} \right\} P_;$ $Z_; \left\{ \begin{array}{c} C(I) \pm i \\ R_ \end{array} \right\} P_;$

Functions	Illustration	Tape format
Canned cycle (G71 to G76) (G90, G92, G94)	Refer to II.14. FUNCTIONS TO SIM- PLIFY PROGRAMMING	N_ G70 P_ Q_ ; G71 U_ R_ ; G71 P_ Q_ U_ W_ F_ S_ T_ ; G72 W_ R_ ; G72 P_ Q_ U_ W_ F_ S_ T_ ; G73 U_ W_ R_ ; G73 P_ Q_ U_ W_ F_ S_ T_ ; G74 R_ ; G74 X(u)_ Z(w)_ P_ Q_ R_ F_ ; G75 R_ ; G75 X(u)_ Z(w)_ P_ Q_ R_ F_ ; G76 P_ Q_ R_ ; G76 X(u)_ Z(w)_ P_ Q_ R_ F_ ; $\left\{ \begin{array}{l} \text{G90} \\ \text{G92} \end{array} \right\} \text{X}_\text{Z}_\text{I}_\text{F}_\text{;} ;$ G94 X_ Z_ K_ F_ ;
tool offset		IP_ T_

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. programmable dimension		± 99999.999 mm	± 9999.9999 mm
Max. rapid traverse		100000 mm/min	24000 mm/min
Feedrate range Notes	Per min	1 to 100000 mm/min	1 to 12000 mm/min
	Per rev	0.0001 to 500.0000 mm/rev	0.0001 to 500.0000 mm/rev
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. programmable dimension		± 9999.9999 inch	± 393.70078 inch
Max. rapid traverse		100000 mm/min	24000 mm/min
Feedrate range Notes	Per min	0.01 to 4000 inch/min	0.01 to 12000 inch/min
	Per rev	0.000001 to 9.999999 inch/rev	0.000001 to 9.999999 inch/rev
Incremental feed		0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation		0 to ± 99.9999 inch	0 to ± 99.99999 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. programmable dimension		± 9999.9999 inch	± 999.99999 inch
Max. rapid traverse Notes		4000 inch/min	960 inch/min
Feedrate range Notes	Per min	0.01 to 4000 inch/min	0.01 to 480 inch/min
	Per rev	0.000001 to 9.999999 inch/rev	0.000001 to 9.999999 inch/rev
Incremental feed		0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation		0 to ± 99.9999 inch	0 to ± 99.99999 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. programmable dimension		± 99999.999 mm	± 9999.9999 mm
Max. rapid traverse Notes		4000 inch/min	960 inch/min
Feedrate range Notes	Per min	1 to 100000 mm/min	1 to 12000 mm/min
	Per rev	0.0001 to 500.0000 mm/rev	0.0001 to 500.0000 mm/rev
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. programmable dimension	± 99999.999 deg	± 9999.9999 deg
Max. rapid traverse Notes	240000 deg/min	100000 deg/min
Feedrate range (metric input) Notes	1 to 100000 deg/min	1 to 24000 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.

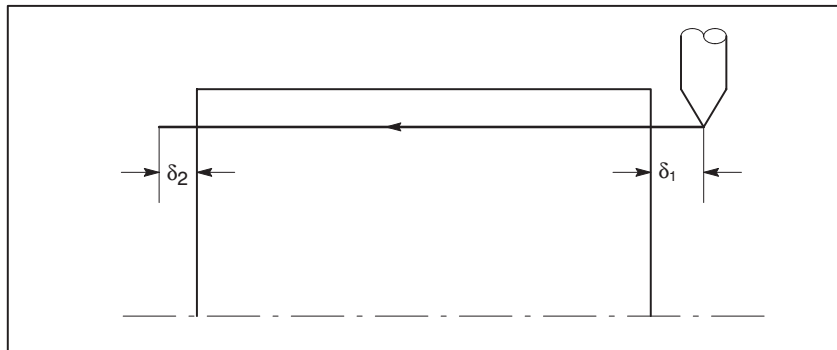


Fig. D.1 (a) Incorrect thread position

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

$$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V \dots\dots\dots (2)$$

$$a = \exp(-\frac{t}{T_1}) \dots\dots\dots (3)$$

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 HtI 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.

D.2
SIMPLE
CALCULATION OF
INCORRECT THREAD
LENGTH

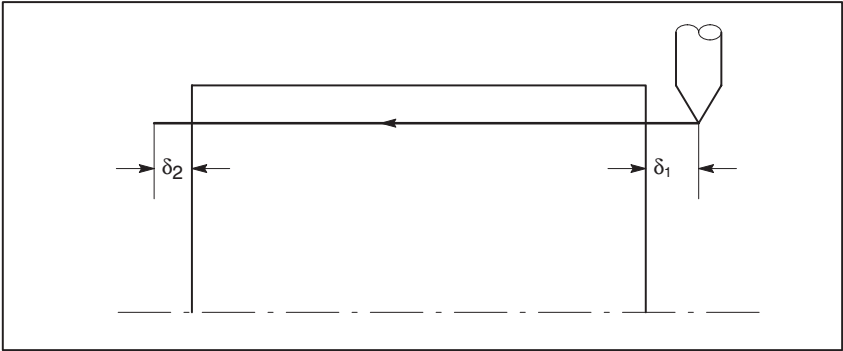


Fig. D.2 Incorrect threaded portion

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

$$\delta_1 = \frac{LR}{1800 * } (-1-\ln a) \text{ (mm)}$$
$$= \delta_2 (-1-\ln a) \text{ (mm)}$$

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.

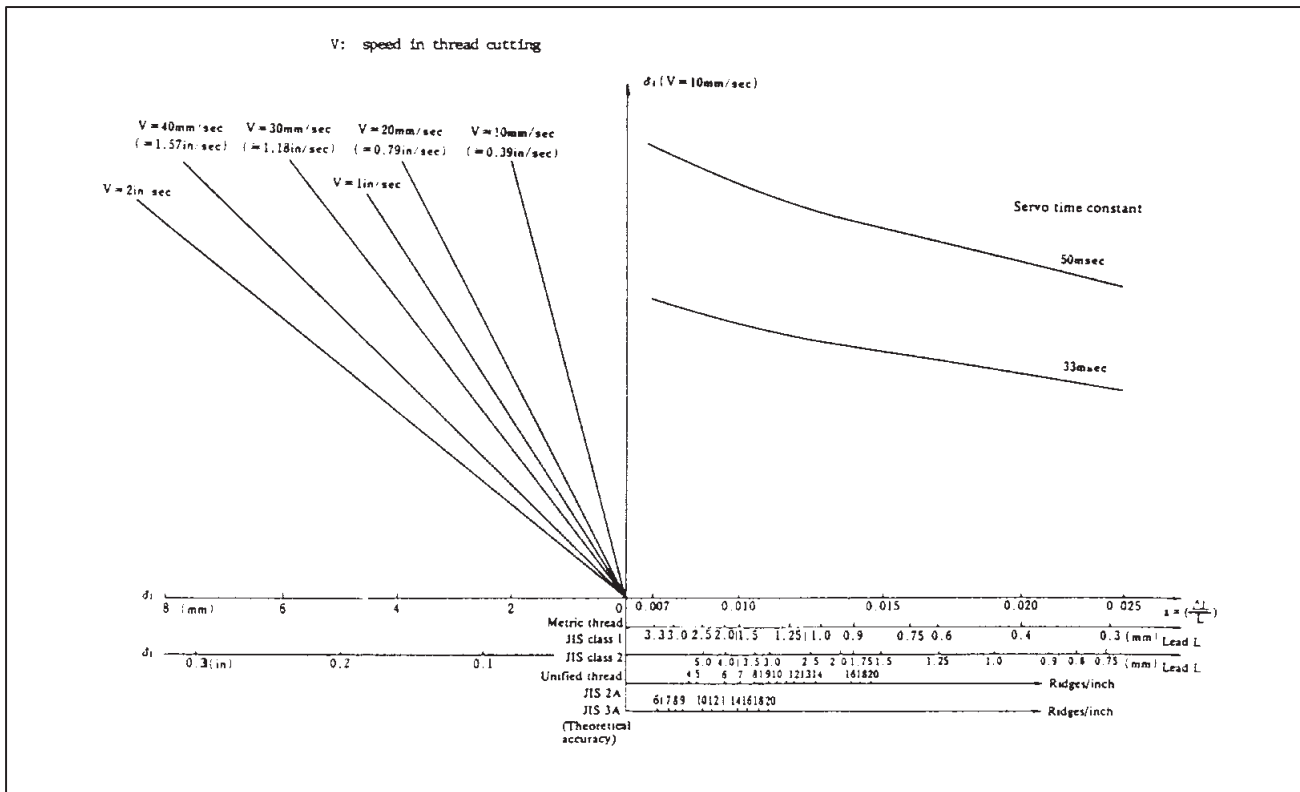
a	-1- $\ln a$
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

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.

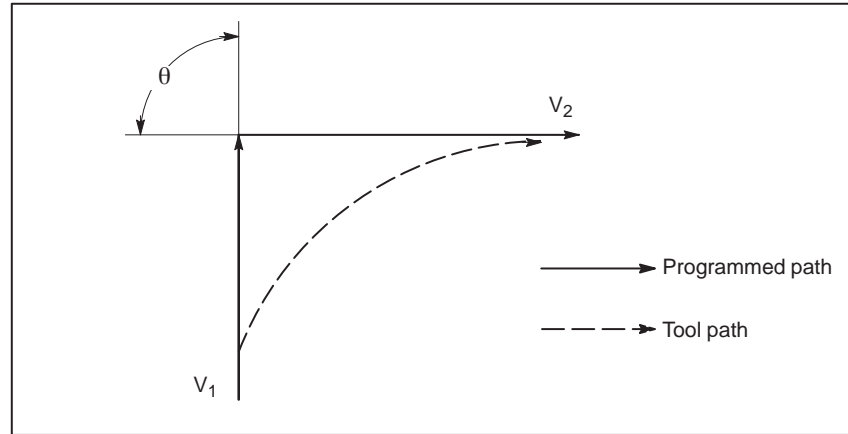


Fig. D.3 (a) Slight deviation between the tool path and the programmed path

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 with the parameter which is set as an example.

When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.

In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

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.)

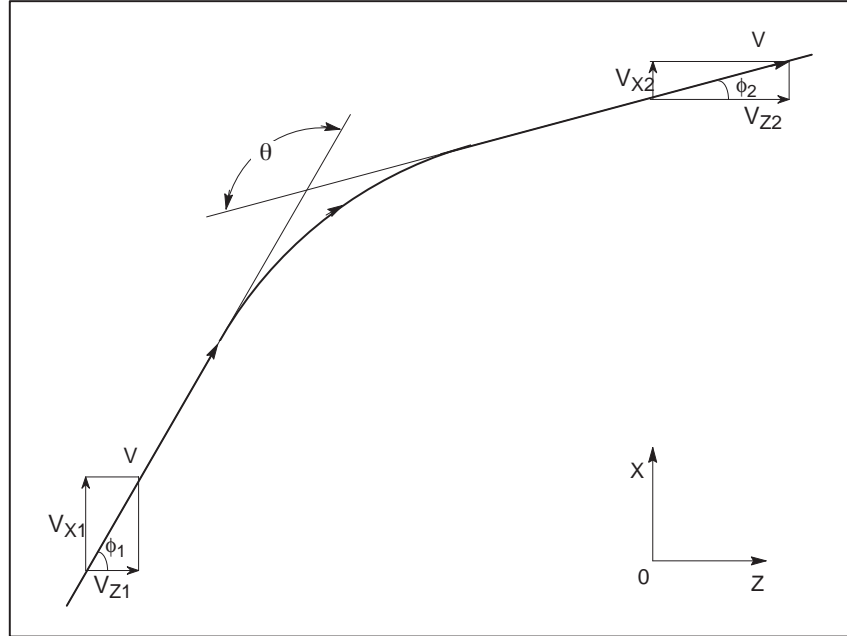


Fig. D.3 (b) Example of tool path

• Description of conditions and symbols

$$V_{x1} = V \sin \phi_1$$

$$V_{z1} = V \cos \phi_1$$

$$V_{x2} = V \sin \phi_2$$

$$V_{z2} = V \cos \phi_2$$

V : Feedrate at both blocks before and after cornering

V_{x1} : X-axis component of feedrate of preceding block

V_{z1} : Z-axis component of feedrate of preceding block

V_{x2} : X-axis component of feedrate of following block

V_{z2} : Z-axis component of feedrate of following block

θ : Corner angle

ϕ_1 : Angle formed by specified path direction of preceding block and Z-axis

ϕ_2 : Angle formed by specified path direction of following block and Z-axis

- Initial value calculation

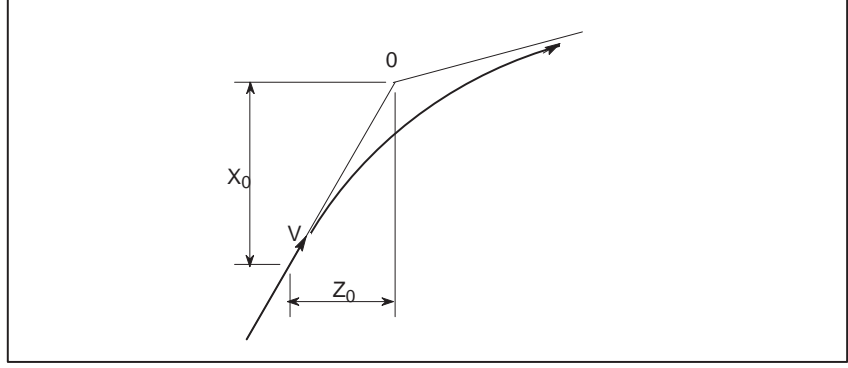


Fig. D.3 (c) Initial value

The initial value when cornering begins, that is, the Z 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.

$$X_0 = V_{x1}(T_1 + T_2)$$

$$Z_0 = V_{z1}(T_1 + T_2)$$

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 Z-axis direction.

$$V_x(t) = \frac{V_{x1}-V_{x2}}{T_1-T_2} \left(T_1 \exp\left(-\frac{t}{T_1}\right) - T_2 \exp\left(-\frac{t}{T_2}\right) \right) + V_{x2}$$

$$V_z(t) = \frac{V_{z1}-V_{z2}}{T_1-T_2} \left(T_1 \exp\left(-\frac{t}{T_1}\right) - T_2 \exp\left(-\frac{t}{T_2}\right) \right) + V_{z2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

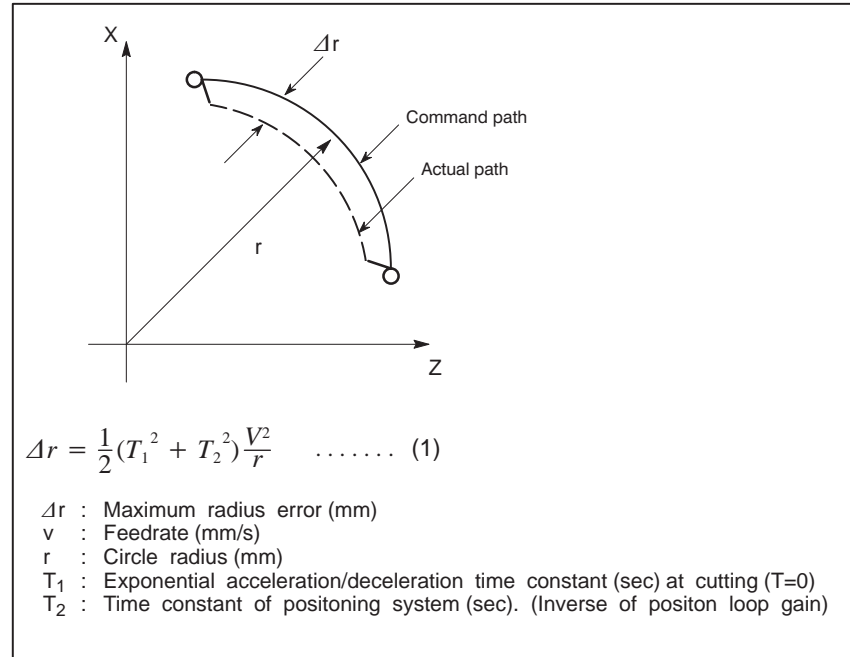
$$\begin{aligned} X(t) &= \int_0^t V_x(t) dt - X_0 \\ &= \frac{V_{x2}-V_{x1}}{T_1-T_2} \left(T_1^2 \exp\left(-\frac{t}{T_1}\right) - T_2^2 \exp\left(-\frac{t}{T_2}\right) \right) - V_{x2}(T_1 + T_2 - t) \\ Z(t) &= \int_0^t V_z(t) dt - Z_0 \\ &= \frac{V_{z2}-V_{z1}}{T_1-T_2} \left(T_1^2 \exp\left(-\frac{t}{T_1}\right) - T_2^2 \exp\left(-\frac{t}{T_2}\right) \right) - V_{z2}(T_1 + T_2 - t) \end{aligned}$$

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 045#6 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 compensation	×	Depending on parameter (No.0001#3)	○ : MDI mode Other modes depend on parameter (No.0001#3).
	Tool nose radius 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 sequence number.
- 2 When a reset is performed during execution of a subprogram, control returns main program.
Execution cannot be started from the middle of the subprogram.

F

CHARACTER-TO-CODES CORRESPONDENCE TABLE

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 parenthe- sis
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 brack- et
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	HAAtI 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
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	PROGRAM HAS AN ERROR AT THE END	The program does not end with M02/M30/M99 and the execution of EOR (%) was attempted instead. 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	ILLEGAL LEAD COMMAND	In variable lead threading, the lead incremental and decremental outputted by address K exceed the maximum command value or a command such that the lead becomes a negative value is given. Modify the program.
015	TOO MANY AXES COMMANDED	An attempt was made to move the machine along the axes, but the number of the axes exceeded the specified number of axes controlled simultaneously. Alternatively, in a block where the skip function activated by the torque-limit reached signal (G31 P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis.
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. 0876.
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.
023	ILLEGAL RADIUS COMMAND	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.

Number	Meaning	Contents
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 T code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER	The offset number in T function specified for tool offset 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 tool nose radius compensation. 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 tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN NRC	The offset plane is switched in tool nose radius compensation. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN NRC	Overcutting will occur in tool nose radius compensation. 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.
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 vertical line. Modify the program.
053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(",") is not C or R in direct drawing dimensions programming. Modify the program.
054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount.
056	NO END POINT & ANGLE IN CHF/CNR	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering command, I(K) is commanded for the X(Z) axis.
057	NO SOLUTION OF BLOCK END	Block end point is not calculated correctly in direct dimension drawing programming.

Number	Meaning	Contents
058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming.
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.
061	ADDRESS P/Q NOT FOUND IN G70–G73	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71–G76	<ol style="list-style-type: none"> 1 The depth of cut in G71 or G72 is zero or negative value. 2 The repetitive count in G73 is zero or negative value. 3 the negative value is specified to Δi or Δk is zero in G74 or G75. 4 A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. 5 A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. 6 Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7 The specified minimum depth of cut in G76 is greater than the height of thread. 8 An unusable angle of tool tip is specified in G76. Modify the program.
063	SEQUENCE NUMBER NOT FOUND	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTONOUSLY	A target shape which cannot be made by monotonic machining was specified in a repetitive canned cycle (G71 or G72).
065	ILLEGAL COMMAND IN G71–G73	<ol style="list-style-type: none"> 1 G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. 2 Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. Modify the program.
066	IMPROPER G-CODE IN G71–G73	An unallowable G code was commanded between two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT ERROR IN MDI MODE	G70, G71, G72, or G73 command with address P and Q. Modify the program.
068	TEN OR MORE POCKETS	The number of pockets is greater than or equal to ten for G71 or G72 of type II.
069	FORMAT ERROR IN G70–G73	the final move command in the blocks specified by P and Q of G70, G71, G72, and G73 ended with chamfering or corner R. Modify the program.
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), 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.

Number	Meaning	Contents
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	The number of subprograms called exceeded the limit.
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. Otherwise, a called program is being edited in back-ground processing. Correct the program, or discontinue the back-ground editing.
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.
080	G37 ARRIVAL SIGNAL NOT ASSERTED	In the automatic tool compensation function (G36, G37), the measurement position reach signal (XAE 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	Automatic tool compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.
082	T-CODE NOT ALLOWED IN G37	T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37	In automatic tool compensation (G36, G37), 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. Check the program contents.
091	MANUAL RETURN TO THE REFERENCE POSITION IS IMPOSSIBLE BECAUSE OF A TEMPORARY STOP.	A manual return to the reference position cannot be made because the system is in the temporary stop state. After pressing the RESET key, execute manual return to the reference position.
092	AXES NOT ON THE REFERENCE POINT	Automatic reference position return (G28) or the commanded axis by 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.

Number	Meaning	Contents
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.) Perform the correct operation according to the operator's manual.
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.) Perform the correct operation according to the operator's manual.
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.
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. Perform the reference position return.
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 was turned off while memory was being rewritten by program edit operation. When this alarm is issued, clear the program by setting the setting parameter (PWE) to 1, then turning on the power again while holding down the <DELETE> key.
109	P/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 Modify the program.
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°) Modify the program.
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 contains an undefined H code in a G65 block. Custom macro B contains an error in a format other than <expression>. Correct the program.

Number	Meaning	Contents
115	ILLEGAL VARIABLE NUMBER	<p>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:</p> <p>High speed cycle machining</p> <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 to 999). 3 The number of data in the header is out of the allowable range (0 to 32767). 4 The start data variable number of executable format data is out of the allowable range (#20000 to #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 is overlapped with the variable number used in the header. <p>Modify the program.</p>
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.
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	<p>Custom macro A contains an address that cannot be specified in a G65 block.</p> <p>Custom macro B contains a format error in <expression>. Correct the program.</p>
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	SPINDLE ORIENTATION PLEASE	Without any spindle orientation, an attempt was made for spindle indexing. Perform spindle orientation.
136	C/H-CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.

Number	Meaning	Contents
137	M-CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M-code related to spindle indexing. Modify the program.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Modify the value of program or parameter.
146	IMPROPER G CODE	G codes which cannot be specified in the polar coordinate interpolation mode was specified. See section II-4.4 and modify the program.
147	MOVE COMMAND TOO LARGE	The tool passes the coordinate origin, causing the amount of travel about the rotation axis to be too great. Modify the program or set bit 2 of parameter No. 399 accordingly.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. of tool life management exceeds the maximum allowable value. Modify the program. Alternatively, modify the tool life data.
151	TOOL GROUP NUMBER NOT FOUND	The tool group of 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 of 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.
155	ILLEGAL T-CODE IN M06	In the machining program, M06 and T code in the same block do not correspond to the group of tool life management 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 tool life management is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups of 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 of tool life management, power was turned off. Set again.
160	MISMATCH WAITING M-CODE (TT only)	Diffrent M code is commanded in heads 1 and 2 as waiting M code. Modify the program.
161	COMMAND G68/G69 INDEPENDENTLY (TT only)	G68 and G69 are not independently commanded in balance cut. Modify the program.
169	ILLEGAL TOOL GEOMETRY DATA (TT only)	Incorrect tool figure data in interference check.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. Modify the program.
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, G76, G81 to G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54 to G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error Modify the program.

Number	Meaning	Contents
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. SETTING ERROR	The number of controlled axes set by the parameter 597 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.
194	SPINDLE COMMAND IN SYN-CHRO-MODE	A contour control mode, spindle positioning (Cs-axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
195	MODE CHANGE ERROR	The control mode of the serial spindle cannot be changed. Check the Ladder diagram of the PMC.
197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cf-axis when the signal CON was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
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. The range for S values which can be specified in rigid tapping is set in parameter 5243. Change the setting in the parameter or 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 is not turned on.
210	CAN NOT COMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.
211	G31 (HIGH) NOT ALLOWED IN G99	G31 is commanded in the per revolution command when the high-speed skip option is provided. Modify the program.
212	ILLEGAL PLANE SELECT	The direct drawing dimensions programming is commanded for the plane other than the Z-X plane. Correct the program.
213	ILLEGAL COMMAND IN SYN-CHRO-MODE	Movement is commanded for the axis to be synchronously controlled.
214	ILLEGAL COMMAND IN SYN-CHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
217	DUPLICATE G251 (COMMANDS)	G251 is further commanded in the G251 mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251	P or Q is not commanded in the G251 block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY	G251 and G250 are not independent blocks.
220	ILLEGAL COMMAND IN SYNCHR-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis.
221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon machining operation and axis control or balance cutting are executed at a time. Modify the program.
224	RETURN TO REFERENCE POINT	Not returned to reference point before cycle start.

Number	Meaning	Contents
225	SYNCHRONOUS/MIXED CONTROL ERROR (TT only)	This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command. 1 When there is a mistake in axis number parameter setting. 2 When there is a mistake in control commanded. Modify the program or the parameter.
226	ILLEGAL COMMAND IN SYNCHRO-MODE (TT only)	A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the parameter.
229	CAN NOT KEEP SYNCHRO-STATE (TT only)	This alarm is generated in the following circumstances. 1 When the synchro/mixed state could not be kept due to system overload. 2 The above condition occurred in CMC devices (hardware) and synchro-state could not be kept. (This alarm is not generated in normal use conditions.)
233	P/S ALARM	In the skip function activated by the torque limit signal, the number of accumulated erroneous pulses exceed 32767 before the signal was input. Therefore, the pulses cannot be corrected with one distribution. Change the conditions, such as federates along axes and torque limit, and try again.
245	T-CODE NOT ALLOWED IN THIS BLOCK	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.

2) Background edit alarm

Number	Meaning	Contents
???	BP/S 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	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. (Note) Use background editing correctly.

NOTE

Because it uses the background editing function, a background editing alarm may be issued during MDI operation B.

3) Absolute pulse coder (APC) alarm

Number	Meaning	Contents
3n0	nth-axis origin return	Manual reference position return is required for the nth-axis (n=1 to 8).
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 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 – 767		CSA	BLA	PHA	RCA	BZA	CKA	SPH

CSA : The serial pulse coder is defective. Replace it.

BLA : The battery voltage is low. Replace the batteries. This alarm has nothing to do with alarm (serial pulse coder alarm).

PHA : The serial pulse coder or feedback cable is defective. Replace the serial pulse coder or cable.

RCA : The serial pulse coder is defective. Replace it.

BZA : 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).

CKA : The serial pulse coder is defective. Replace it.

SPH : 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 – 777	DTE	CRC	STB					

DTE : 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

CRC : 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.

STB : the serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective.

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 to 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.
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. 7-axis, 8-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 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.

Number	Meaning	Contents and actions
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 4n4 are displayed with diagnosis numbers 720 to 727 in the sequence of axis numbers.

	#7	#6	#5	#4	#3	#2	#1	#0
720 – 727	OVL	LV	OVC	HCAL	HVA	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 check 1/2.
5n1	OVER TRAVEL : -n	Exceeded the n-th axis - side stored stroke check 1/2.
5n2	OVER TRAVEL : +n	Exceeded the n-th axis + side stored stroke check 3.
5n3	OVER TRAVEL : -n	Exceeded the n-th axis - side stored stroke check 3.
5n4	OVER TRAVEL : +n	Exceeded the n-th axis + side stored stroke check 4.
5n5	OVER TRAVEL : -n	Exceeded the n-th axis - side stored stroke check 4.
520	OVER TRAVEL : +Z	A hardware overtravel occurred in the positive direction of the Z-axis.
590	Tool post interference alarm:+X-axis	A tool post interference alarm occurred during traveling in the positive direction on the X-axis.
591	Tool post interference alarm:-X-axis	A tool post interference alarm occurred during traveling in the negative direction on the X-axis.
592	Tool post interference alarm:+Z-axis	A tool post interference alarm occurred during traveling in the positive direction on the Z-axis.
593	Tool post interference alarm:-Z-axis	A tool post interference alarm occurred during traveling in the negative direction on the Z-axis.

8) Macro alarms

Number	Meaning	Contents and remedy
500 to 599	MACRO ALARM	<p>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.</p>

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)
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 * 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.
704	Overheat: Spindle	The spindle overheated during spindle variation detection. Check the cutting conditions.

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	Overrun 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.

Number	Meaning	Contents and remedy
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.

Alarm No.	Meaning	Description	Remedy
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.
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.

Alarm No.	Meaning	Description	Remedy
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.
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

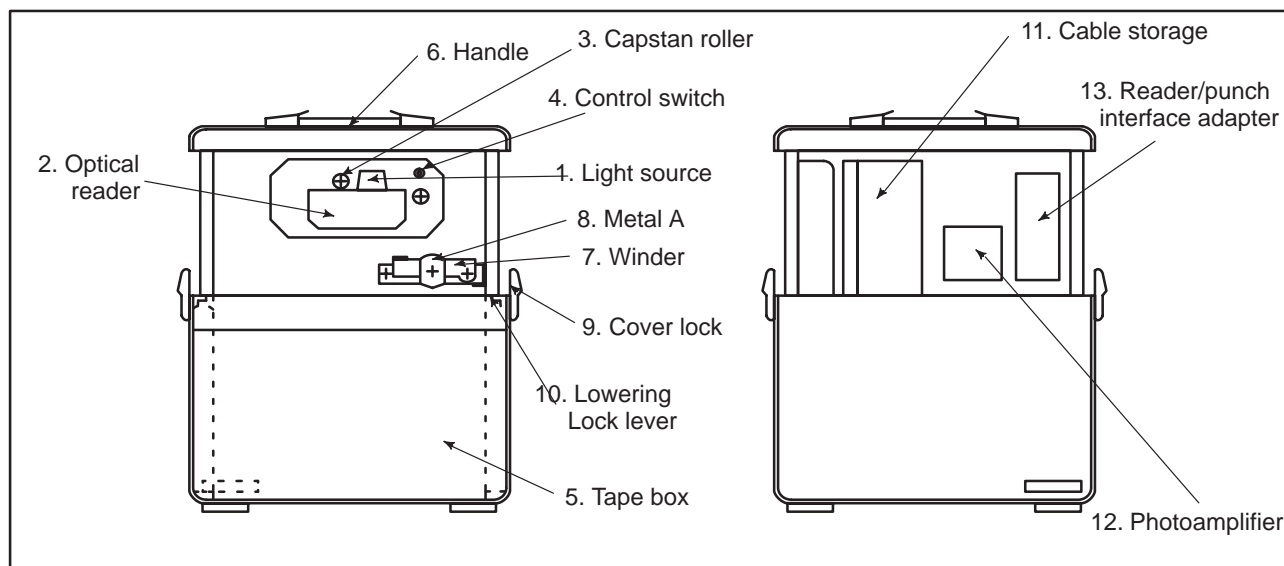


Table H Description of Each Section

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 light-source. 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>Paper tape</p> <p>Insert</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	<p>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.</p> <p>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.</p> <p>When the latch is unlocked, the tape reader can be stored in the box.</p> <p>When storing the tape reader, secure it with the cover lock.</p>
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

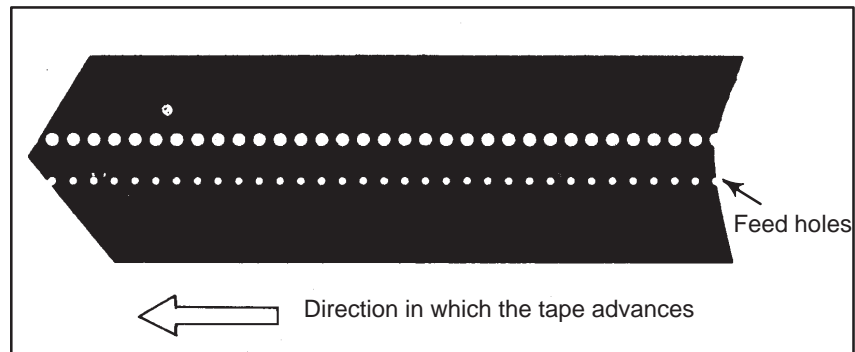
Procedure for Operating the Portable Tape Reader

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.

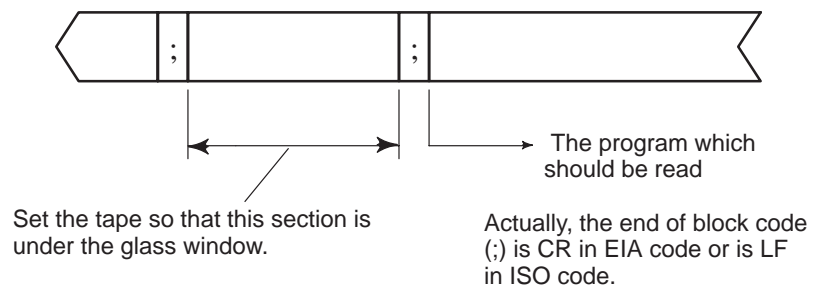
- 16 Lock the cover lock **9** and carry the tape reader with the handle **6**.

CAUTION**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.

NOTE**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.



Series 0-D SPECIFICATIONS

Controlled axis

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD II
Number of controlled axes	2 axes	○	—	—	○	—	○
	3 axes	—	○	○	☆	○	☆
	4 axes	—	—	—	☆	—	☆
Simultaneous controllable axes	2 axes	○	○	○	○	○	○
	3 axes	—	—	—	☆	—	☆
	4 axes	—	—	—	☆	—	☆
Axis control by PMC	MAX. 2 axes	—	—	—	○	○	○
Cs-axis contouring control		—	—	—	○	—	○
Cf-axis control		—	○	○	○	—	○
Y-axis 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	All axes	○	○	○	○	○	○
Machine lock	All axes	○	○	○	○	○	○
Emergency stop		○	○	○	○	○	○
Overtravel signal	Z-axis	○	○	○	○	○	○
Stored stroke check 1		○	○	○	○	○	○
Stored stroke check 2		○	○	○	○	○	○
Stored stroke check 3/4	G22/G23	—	—	—	○	—	○
Mirror image	each axis	○	○	○	○	—	—
Follow up		○	○	○	○	○	○
Servo off		○	○	○	○	○	○
Mechanical handle feed		○	○	○	○	○	○
Chamfering on/off		○	○	○	○	○	○
Backlash compensation		○	○	○	○	○	○
Stored pitch error compensa- tion		○	○	○	○	○	○
Position switch		—	—	—	○	—	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Operation

Item	Specification	0–TD Package 1	0–TD Package 2	0–TD Package 3	0–TD II	0–GCD Package 1	0–GCD 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 units	–	–	–	☆	–	☆
Manual handle feed rate	x1, x10, xm, xn m:Up to 127, n:Up to 1000	○	○	○	○	○	○
Handle interruption		–	–	–	○	–	○
Incremental feed	x1, x10, x100, x1000	○	○	○	○	○	○
Jog and handle simultaneous mode		○	○	○	○	○	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Interpolation function

Item	Specification	0–TD Package 1	0–TD Package 2	0–TD Package 3	0–TD II	0–GCD Package 1	0–GCD II
Positioning	G00	○	○	○	○	○	○
Linear interpolation		○	○	○	○	○	○
Circular interpolation	Multi quadrant is possible.	○	○	○	○	○	○
Dwell (per sec.)		○	○	○	○	○	○
Polar coordinate interpolation		–	–	–	○	–	○
Cylindrical interpolation		–	–	–	○	–	○
Thread cutting / synchronous feed		○	○	○	○	○	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD II
Thread cutting retract		—	○	○	○	—	—
Continuous threading cutting		—	—	—	○	○	○
Variable lead thread cutting		—	—	—	○	—	○
Polygon turning		—	—	—	○	—	○
Skip function	G31	—	—	—	○	—	○
High-speed skip function		—	—	—	○	—	○
Multi-step skip function		—	—	—	—	○	○
Torque limit skip		—	—	—	○	—	○
Reference position return	G28	○	○	○	○	○	○
Reference position return check	G27	○	○	○	○	○	○
2nd reference position return		○	○	○	○	○	○
3rd/4th reference position return		—	—	—	○	—	○
Rapid traverse rate	100m/min	○	○	○	○	○	○
Rapid traverse override	F0,25,50,100%	○	○	○	○	○	○
Feed per minute	mm/min.	○	○	○	○	○	○
Feed per revolution	mm/rev	○	○	○	○	○	○
Tangential speed constant control		○	○	○	○	○	○
Cutting feedrate clamp		○	○	○	○	○	○
Automatic acceleration / deceleration	Rapid traverse : linear Cutting feed : exponential	○	○	○	○	○	○
Linear acceleration / deceleration after cutting feed interpolation		—	—	—	○	—	○
Feedrate override	0 to 150%	○	○	○	○	○	○
Override cancel		○	○	○	○	○	○
Manual synchronous feed		○	○	○	○	○	○
External deceleration		—	—	—	○	—	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Programming

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD 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	○	○	○	○	○	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Item	Specification	0–TD Package 1	0–TD Package 2	0–TD Package 3	0–TD II	0–GCD Package 1	0–GCD II
Absolute / Incremental programming	It is possible to use in the same block.	○	○	○	○	○	○
Decimal point input / Pocket calculator type decimal point input programming		○	○	○	○	○	○
Diameter / radius programming (X axis)		○	○	○	○	○	○
Plane selection	G17,G18,G19	○	○	○	○	○	○
Rotary axis designation	only added axes	○	○	○	○	○	○
Rotary axis roll–over function	only added axes	○	○	○	○	○	○
Coordinate system setting		○	○	○	○	○	○
Automatic coordinate system setting		○	○	○	○	○	○
Coordinate system shift		○	○	○	○	○	○
Direct input of coordinate system shift		○	○	○	○	○	○
Workpiece coordinate system	G52,G53,G54 to G59 (Note)	○	○	○	○	–	○
Manual absolute on and off		○	○	○	○	○	○
Direct drawing dimension programming		○	○	○	○	○	○
G code system A		○	○	○	○	○	○
G code system B/C		–	○	○	○	○	○
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		○	○	○	○	○	○
Multiple repetitive cycle		○	○	○	○	–	–
Multiple repetitive cycle 2	pocket figure (Note)	–	–	–	○	–	–
Canned cycles for drilling		–	–	–	○	–	–
Canned cycles for grinding		–	–	–	–	○	○
Circular interpolation by R programming		○	○	○	○	○	○
Mirror image for double turret		–	–	–	○	–	–
Tape format for FS10/11		–	–	–	○	–	○
Menu programming		–	–	–	○	–	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Miscellaneous function / spindle function

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD II
Auxiliary function	M3 digit	○	○	○	○	○	○
2nd auxiliary function	B8 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%	○	○	○	○	○	○
Actual spindle speed output		—	—	—	○	—	○
Spindle speed fluctuation detection		—	—	—	○	—	○
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		—	—	—	○	—	○
Spindle positioning		—	—	—	○	—	○
Multi spindle control		—	—	—	○	—	○
Rigid tapping	Note)	—	—	—	○	—	—

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Tool function / tool offset function

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD II
Tool function	T2/T4	○	○	○	○	○	○
Tool offset memory	± 6 digits 9/16 pairs	—	—	—	—	○	—
	± 6 digits 32 pairs	○	○	○	○	—	○
Tool offset		○	○	○	○	○	○
Y axis offset		—	—	—	○	—	○
Tool nose radius compensation		○	○	○	○	○	○
Tool geometry / wear offset		○	○	○	○	—	○
Tool life management	Note)	—	—	—	○	—	○
Tool offset value counter input		○	○	○	○	○	○
Automatic tool offset		—	—	—	○	—	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Item	Specification	0–TD Package 1	0–TD Package 2	0–TD Package 3	0–TD II	0–GCD Package 1	0–GCD II
Direct input of offset value measured A		○	○	○	○	○	○
Direct input of offset value measured B		–	–	–	○	–	–

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Editing operation

Item	Specification	0–TD Package 1	0–TD Package 2	0–TD Package 3	0–TD II	0–GCD Package 1	0–GCD II
Part program storage length	80m	○	○	○	–	○	–
	320m	–	–	–	○	–	○
Registered programs	63 pieces	○	○	○	–	○	–
	200 pieces	–	–	–	○	–	○
Tape editing		○	○	○	○	○	○
Program protect		○	○	○	○	○	○
Background editing		–	○	○	○	–	○
Extended part program editing		–	–	–	○	–	○
Play back		–	–	–	○	–	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Setting / display

Item	Specification	0–TD Package 1	0–TD Package 2	0–TD Package 3	0–TD II	0–GCD Package 1	0–GCD 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		○	○	○	○	○	○
Directory display of floppy cassette		–	–	–	○	–	○
Axis name alternation		–	–	–	–	–	○
Graphic function		–	–	–	☆	–	☆
Servo setting screen		○	○	○	○	○	○
Spindle setting screen		○	○	○	○	○	○
Servo waveform display		–	–	–	※	–	※
Software operator's panel		–	○	○	○	–	○
Software operator's panel general purpose switch		–	○	○	○	–	○
English display		○	○	○	○	○	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD II
Japanese (Chinese characters) display		—	—	○	○	—	○
German / French display		—	—	○	○	—	○
Italian display		—	—	○	○	—	○
Chinese display		○	○	○	○	○	○
Spanish display		—	—	○	○	—	○
Korean display		—	☆	—	☆	—	☆
Data protection key		○	○	○	○	○	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Data input / output

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD 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 work number search	15 pieces	○	○	○	○	○	○
External program number search	1 to 9999	○	○	○	○	○	○

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option

Others

Item	Specification	0-TD Package 1	0-TD Package 2	0-TD Package 3	0-TD II	0-GCD Package 1	0-GCD 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: 80/56, 104/72 points, source/sink	○	○	○	●	○	●
I/O Unit-MODEL A	DI/DO Max. 1024/1024 points	—	—	—	☆	—	☆

○ : Basic ● : Basic option ☆ : option ※ : Function included in other option







































NOTE

Part program storage length is decreased by 9.75m.

J

CORRESPONDENCE BETWEEN ENGLISH KEY AND SYMBOLIC KEY

Table : 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/PA- RAM key					

[Numbers]

2 Systems Control Function (Functions Specific to 0-TTC), 340

[A]

Absolute and Incremental Programming (G90, G91), 90
 Actual Feedrate Display, 532
 Addresses and Specifiable Value Range for Series 10/11 Tape Format, 319
 Alarm and Self-Diagnosis Functions, 448
 Alarm Display, 449
 Alarm Display (See Section III-7.1), 373
 Alarm List, 625
 Altering a Word, 484
 Angular Axis Control (0-GCC/00-GCC), 338
 Arithmetic and Logic Operation, 268
 Automatic Insertion of Sequence Numbers, 506
 Automatic Operation, 364, 405
 Automatic Tool Offset (G36, G37), 241
 Auxiliary Function, 115
 Auxiliary Function (M Function), 116

[B]

Background Editing, 501
 Balance Cut (G68, G69), 355
 Branch and Repetition, 273

[C]

Canceling Spindle Positioning, 105
 Canned Cycle, 322
 Canned Cycle (G90, G92, G94), 137
 Canned Cycle for Drilling (G80 to G89), 164
 Canned Cycle for Drilling Cancel (G80), 173
 Canned Grinding Cycle (0-GCC, 00-GCC, 0-GCD/II), 178
 Chamfering and Corner R, 182
 Changing of Tool Offset Value, 240
 Changing Workpiece Coordinate System, 83
 Character-to-Codes Correspondence Table, 624

Characters and codes to be used for the pattern data input function, 315
 Check by Running the Machine, 366
 Checking by Self-Diagnostic Screen, 451
 Circular Interpolation (G02,G03), 44
 Command for Machine Operations – Miscellaneous Function, 25
 Compensation Applied to A Programmed T code, 109
 Compensation Function, 191
 Conditional Branch (IF Statement), 273
 Conditions for Making a Tool Post Interference Check, 350
 Configuration of High Speed Cycle Cutting Data, 327
 Constant Lead Threading (G32), 55
 Constant Surface Speed Control (G96, G97), 96
 Continuous Thread Cutting, 60
 Controlled Axes, 31, 32
 Conversational Programming with Graphic Function, 513
 Coordinate System, 78
 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System, 17
 Coordinate Value and Dimension, 89
 Copying an Entire Program, 492
 Copying Part of a Program, 493
 Correction in Chamfering and Corner Arcs, 235
 Correspondence Between English Key and Symbolic Key, 656
 Counter Input of Offset value, 552
 Counter Rigid Tapping, 177
 Counting a Tool Life, 113
 Creating Programs, 504
 Creating Programs in Teach in Mode, 508
 Creating Programs Using the MDI Panel, 505
 CRT/MDI Panels, 378
 Current Block Display Screen, 537
 Current Position Display (See Section III-11.1.1 to 11.1.3), 373
 Custom Macro A, 244
 Custom Macro B, 259
 Custom Macro Body, 247
 Custom Macro Command, 245
 Custom Macro Variables Common to Tool Posts, 356
 Cutting Feed, 70
 Cutting Speed – Spindle Speed Function, 23
 Cylindrical Interpolation (G107), 52

[D]

Data Input/Output, 453
 Data Output, 376
 Data Setting for the Tool Post Interference Check Function, 345
 Decimal Point Programming, 92
 Deleting a Block, 486
 Deleting a Word, 485
 Deleting All Programs, 489
 Deleting Blocks, 486
 Deleting Files, 474
 Deleting More than One Program by Specifying a Range, 490
 Deleting Multiple Blocks, 487
 Deleting One Program, 489
 Deleting Programs, 489
 Details of Functions, 298
 Details of Tool Nose Radius Compensation, 210
 Diameter and Radius Programming, 94
 Direct Drawing Dimensions Programming, 186
 Direct Input of Tool Offset measured B, 550
 Direct Input of Tool Offset Value, 548
 Direct Measured Value Input for Work Coordinate System Shift, 555
 Direction of Imaginary Tool Nose, 199
 Display, 372, 592
 Display and Operation of 00–TC, 591
 Display of Run Time and Parts Count, 533
 Displaying Directory of Floppy Disk, 469
 Displaying and Entering Setting Data, 565
 Displaying and Setting Custom Macro Common Variables, 560
 Displaying and Setting Data, 369
 Displaying and Setting Parameters, 572
 Displaying and Setting Pitch Error Compensation Data, 574
 Displaying and Setting Run Time, Parts Count, and Time, 570
 Displaying and Setting the Software Operator's Panel, 580
 Displaying and Setting the Workpiece Origin Offset Value, 559
 Displaying and Setting Tool Life Management Data, 561
 Displaying Memory Used and a List of Programs, 541

Displaying Operator Message, 579
 Displaying the Directory, 470
 Displaying the pattern menu, 307
 Displaying the Program Number and Sequence Number, 582
 Displaying the Program Number, Sequence Number, and Status, and Warning Messages for Data Setting, 582
 Displaying the Status and Warning for Data Setting, 583
 DNC Operation, 414
 Dry Run, 437
 Dwell (G04), 73
 Dwell by Turning Times of Spindle, 74

[E]

Editing a Part Program, 368
 Editing of Custom Macros B, 500
 Editing Programs, 477
 Emergency Stop, 442
 End Face Peck Drilling Cycle (G74), 157
 End Face Turning Cycle (G94), 142
 Equal-Lead Threading, 320
 Example of Making a Tool Post Interference Check, 353
 Execution of Tool Post Interference Checking, 351
 Extended Part Program Editing Function, 491
 External I/O Devices, 385
 External Output Commands, 292

[F]

FANUC FA Card, 388
 FANUC Floppy Cassette, 387
 FANUC Handy File, 387
 FANUC PPR, 388
 Feed Functions, 66
 Feed-Feed Function, 15
 Feedrate Override, 435
 File Deletion, 457
 File Search, 456
 Files, 454
 Finishing Cycle (G70), 153
 Front Boring Cycle (G85) / Side Boring Cycle (G89), 172

Front Drilling Cycle (G83) / Side Drilling Cycle (G87), 167

Front Tapping Cycle (G84) / Side Tapping Cycle (G88), 170

Function Keys, 381, 382

Functions to Simplify Programming, 136

[G]

General Flow of Operation of CNC Machine Tool, 5

General Precautions for Offset Operations, 238

General Screen Operations, 381

Graphic Display (See Section III-12), 374

Graphics Display, 585

Graphics Function, 584

[H]

Header, 327

Heading a Program, 482

High Speed Cycle Cutting, 325

How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands, 20

How to Use Canned Cycles (G90, G92, G94), 145

How to View the Position Display Change without Running the Machine, 367

[I]

Imaginary Tool Nose, 197

Inch/Metric Conversion (G20, G21), 91

Incorrect Threaded Length, 614

Increment System, 34

Incremental Feed, 397

Input Command from MDI, 237

Inputting a Program, 458

Inputting and Outputting Parameters and Pitch Error Compensation Data, 465

Inputting Custom Macro B Common Variables, 467

Inputting Offset Data, 463

Inputting Parameters, 465

Inputting/Outputting Custom Macro B Common Variables, 467

Inserting a Word, 483

Inserting, Altering and Deleting a Word, 478

Interference Check, 229

Interpolation Functions, 40

Interruption Type Custom Macro, 296

[J]

Jog Feed, 395

[K]

Key Input and Input Buffer, 383

Kind of Variables, 248

[L]

Limitations, 291

Linear Interpolation (G01), 43

List of Functions and Tape Format, 606

Local Coordinate System, 86

[M]

M98 (Single call), 245

Machine Coordinate System, 79

Machine Lock and Auxiliary Function Lock, 434

Macro Call, 277

Macro Call Using an M Code, 284

Macro Call Using G Code, 283

Macro Statements and NC Statements, 272

Manual Absolute ON and OFF, 400

Manual Handle Feed, 398

Manual Handle Interruption, 429

Manual Operation, 360, 392

Manual Reference Position Return, 393

Maximum Strokes, 35

MDI Operation, 409

Memory Operation, 406

Memory Operation by Series 10/11 Tape Format, 318

Menu Programming, 511

Merging a Program, 495

Method of Replacing Battery, 597

Mirror Image, 432

Mirror Image for Double Turret (G68, G69), 185

Modal Call (G66), 281

Moving Part of a Program, 494

Multi-step (0-GCC, 00-GCC, 0-GCD/II), 63
 Multiple M Commands in a Single Block, 117
 Multiple Repetitive Canned Turning Cycle, 323
 Multiple Repetitive Cycle (G70 TO G76), 147
 Multiple Thread Cutting Cycle (G76), 159

[N]

Names of Axes, 33
 Next Block Display Screen, 538
 Nomographs, 613
 Notes on Custom Macro, 258
 Notes on Multiple Repetitive Cycle (G70 to G76), 163
 Notes on Reading This Manual, 7
 Notes on Tool Nose Radius Compensation, 207
 Number of Control Axes, 326
 Number of Registered Cycles, 327

[O]

Offset, 194
 Offset Data Input and Output, 463
 Offset Number, 193
 Offset Number and Offset Value, 200
 Operating Monitor Display, 535
 Operation, 593
 Operation Instruction and Branch Instruction (G65), 253
 Operation of Portable Tape Reader, 644
 Operational Devices, 377
 Oscillation Direct Fixed-Dimension Grinding Cycle (G74), 181
 Oscillation Grinding Cycle (G73), 180
 Outer Diameter / Internal Diameter Drilling Cycle (G75), 158
 Outer Diameter/Internal Diameter Cutting Cycle (G90), 137
 Outputting Parameters, 466
 Outputting a Program, 460
 Outputting Custom Macro B Common Variable, 468
 Outputting Offset Data, 464
 Outputting Programs, 473
 Outputting Signal Near End Point, 119
 Overall Position Display, 530
 Overcutting by Tool Nose Radius Compensation, 234

Overtravel, 443
 Overview of Tool Nose Radius Compensation, 197

[P]







Part Drawing and Tool Movement, 16
 Parts Count Display, Run Time Display (See Section III-11.5.3), 374
 Pattern data display, 311
 Pattern data input function, 306
 Pattern Repeating (G73), 152
 Plane Selection, 88
 Polar Coordinate Interpolation (G112,G113), 48
 Polygonal Turning, 330
 Portable Tape Reader, 389
 Position Display in the Relative Coordinate System, 528
 Position Display in the Workpiece Coordinate System, 527
 Positioning (G00), 41
 Power Disconnection, 391
 Power ON/OFF, 390
 Precautions to be Taken by Operator, 173
 Preparatory Function (G Function), 36
 Processing Macro Statements, 288
 Program Check Screen, 539
 Program Components Other than Program Sections, 123
 Program Configuration, 26, 121
 Program Contents Display, 536
 Program Display (See Section III-11.2.1), 372
 Program Input/Output, 458
 Program Number Search, 488
 Program of Tool Life Data, 111
 Program Restart, 417
 Program Screen for MDI Operation, 540
 Program Section Configuration, 126
 Programmable Parameter Entry (G10), 317
 Pulse Distribution, 326

[R]

Radius Direction Error at Circle Cutting, 621
 Range of Command Value, 610
 Rapid Traverse, 69
 Rapid Traverse Override, 436

Reading Files, 472
 Reference Position, 75
 Reference Position (Machine-Specific Position), 16
 Registering Custom Macro Programs, 290
 Reorganizing Memory, 503
 Repetition (While Statement), 274
 Replacement of Words and Addresses, 498
 Replacing Batteries for Absolute Pulse Coder, 599
 Replacing CNC Battery for Memory Back-Up, 598
 Rigid Tapping, 174
 Rotary Axis Roll-Over, 337

[S]

Safety Functions, 441
 Sample Program, 286
 Scheduling Function, 422
 Screens Displayed by Function Key , 564
 Screens Displayed by Function Key , 544
 Screens Displayed by Function Key , 579
 Screens Displayed by Function Key , 526
 Screens Displayed by Function Key  (in Auto Mode or MDI Mode), 536
 Screens Displayed by Function Key  (in the Edit Mode), 541
 Selecting a Workpiece Coordinate System, 82
 Selection of Tool Used for Various Machining – Tool Function, 24
 Sequence Number Comparison and Stop, 568
 Sequence Number Search, 415
 Series 0-D Specifications, 649
 Setting a Workpiece Coordinate System, 80
 Setting and Display of Interference Forbidden Areas for Tool Post Interference Checking, 348
 Setting and Displaying Data, 517
 Setting and Displaying the Tool Offset Value, 545
 Setting the Workpiece Coordinate System Shifting Amount, 553
 Simple Calculation of Incorrect Thread Length, 616
 Simple Call (G65), 277

Simplified Tool Life Management, 108
 Single Block, 438
 Skip Function (G31), 61
 Skip Function by Torque Limit Arrival Signal, 64
 Soft Keys, 381
 Specification Method, 297
 Specifying a Tool Group in a Machining Program, 114
 Specifying the Spindle Speed Value Directly (S5-Digit Command), 96
 Specifying the Spindle Speed with a Binary Code, 96
 Spindle Orientation, 103
 Spindle Positioning, 103
 Spindle Positioning Function, 103
 Spindle Speed Fluctuation Detection Function (G25, G26), 100
 Spindle Speed Function, 95
 Status when Turning Power On, when Clear and when Reset, 622
 Stock Removal in Facing (G72), 151
 Stock Removal in Turning (G71), 147
 Stroke Check, 444
 Subprogram, 132
 Subprogram Call Function, 427
 Subprogram Call Using an M Code, 285
 Subprogram Call Using M Code, 245
 Subprogram Call Using T code, 246
 Subprogram Calling, 321
 Subprogram Calls Using a T Code, 286
 Supplementary Explanation for Copying, Moving and Merging, 496
 System Variables, 263

[T]

T code for Tool Offset, 193
 Tape Code List, 603
 Test Operation, 433
 Testing a Program, 366
 The Second Auxiliary Functions (B Codes), 118
 Thread Cutting Cycle (G92), 139
 Tool Compensation and Number of Tool Compensation, 239
 Tool Compensation Values, Number of Compensation Values, and Entering Values from the Program (G10), 239
 Tool Figure and Tool Motion by Program, 29
 Tool Function (T Function), 106

Tool Geometry Offset, 192
Tool Life Management, 111
Tool Movement Along Workpiece Parts Figure–interpolation, 12
Tool Movement by Programing – Automatic Operation, 362
Tool Movement in Offset Mode, 214
Tool Movement in Offset Mode Cancel, 226
Tool Movement in Start–Up, 212
Tool Movement Range – Stroke , 30
Tool Offset, 192
Tool Path at Corner, 618
Tool Post Interface Check, 345
Tool Selection, 107, 193
Tool Wear Offset, 192
Traverse Direct Fixed–Dimension Grinding Cycle (G72), 179
Traverse Grinding Cycle (G71), 178
Turning on the Power, 390

[U]

Unconditional Branch (GOTO Statement), 273

[V]

Variable–lead Thread Cutting (G34), 59
Variables, 247, 260

[W]

Waiting for Tool Posts, 343
Word Search, 480
Work Position and Move Command, 202
Workpiece Coordinate System, 80
Workpiece Coordinate System Shift, 85

[Y]

Y Axis Offset, 557

Revision Record
FANUC Series 0/00/0-Mate FOR LATHE OPERATOR'S MANUAL (B-61394E)

05	Dec., '94	All pages are revised.				
04	Sep., '92	Addition of Tool life Management Function Addition of Common Variables Addition of Parameters Addition of Error code list Alteration of RS-232-C/RS-422 interface				
03	Oct., '90	Addition of RS-232-C/RS-422 interface				
02	Mar., '89	Addition of 0-Mate TC	07	Aug., '97	Addition of Safety Precautions Addition of Pattern data Input Function Addition of Series 0-D/0-D II Addition of Series 0-D Specifications Correction of Errors	
01	Oct., '88	_____	06	Sep., '95	Addition of Skip Function by Torque Limit Arrival Signal (Sec.4.11) Addition of Dwell by Turning Times of Spindle (Sec.5.5) Addition of Outputting Signal near End point (Sec.11.4) Addition of Counter Rigid Tapping (Subse.13.3.7) Addition of Operating Monitor Display (Subse.11.1.6) Correction of Errors	
Revision	Date	Contents	Revision	Date	Contents	

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