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

For Lathe System
USER'S MANUAL

B-64304EN-1/01

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

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

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

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

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.
It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

CONTENTS

DEFINITION OF WARNING, CAUTION, AND NOTE s-2
GENERAL WARNINGS AND CAUTIONS s-3
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING s-6
WARNINGS AND CAUTIONS RELATED TO HANDLING s-9
WARNINGS RELATED TO DAILY MAINTENANCE s-12

DEFINITION OF WARNING, CAUTION, AND NOTE

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

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

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

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

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

GENERAL WARNINGS AND CAUTIONS

⚠ WARNING

1 Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2 Before operating the machine, thoroughly check the entered data.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3 Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate.
If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4 When using a tool compensation function, thoroughly check the direction and amount of compensation.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

⚠ WARNING

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

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

⚠ CAUTION

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

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off.

Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.

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

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

⚠ WARNING

1 **Coordinate system setting**
If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command. Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2 **Positioning by nonlinear interpolation**
When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Function involving a rotation axis**
When programming polar coordinate interpolation, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

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

⚠ WARNING

5 **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.

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

7 **Interference check for each path**
An interference check for each path 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.

⚠ WARNING

11 **Compensation function**

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.
Before issuing any of the above commands, therefore, always cancel compensation function mode.

WARNINGS AND CAUTIONS RELATED TO HANDLING

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

⚠ WARNING

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

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

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

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

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

⚠ **WARNING**

6 **Workpiece coordinate system shift**
Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

7 **Software operator's panel and menu switches**
Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands. Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

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

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.

⚠ WARNING

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 **Program editing**

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

WARNINGS RELATED TO DAILY MAINTENANCE

⚠ WARNING

1 **Memory backup battery replacement**

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

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

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

NOTE

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

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

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

Refer to the Section “Method of replacing battery” in the User’s Manual (Common to T/M series) for details of the battery replacement procedure.

⚠ WARNING

2 **Absolute pulse coder battery replacement**

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

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

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

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.

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

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

Refer to the FANUC SERVO MOTOR αi series Maintenance Manual for details of the battery replacement procedure.

⚠ WARNING

3 Fuse replacement

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

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

When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover). Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

TABLE OF CONTENTS

SAFETY PRECAUTIONS s-1
DEFINITION OF WARNING, CAUTION, AND NOTE s-2
GENERAL WARNINGS AND CAUTIONS s-3
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING s-6
WARNINGS AND CAUTIONS RELATED TO HANDLING s-9
WARNINGS RELATED TO DAILY MAINTENANCE s-12

I. GENERAL

1 GENERAL 3
1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL 7
1.2 NOTES ON READING THIS MANUAL 9
1.3 NOTES ON VARIOUS KINDS OF DATA 9

II. PROGRAMMING

1 GENERAL 13
1.1 OFFSET 14

2 PREPARATORY FUNCTION (G FUNCTION) 15

3 INTERPOLATION FUNCTION 20
3.1 POLAR COORDINATE INTERPOLATION (G12.1, G13.1) 21
3.2 CONSTANT LEAD THREADING (G32) 29
3.3 VARIABLE LEAD THREADING (G34) 33
3.4 CONTINUOUS THREADING 34
3.5 MULTIPLE THREADING 35

4 FUNCTIONS TO SIMPLIFY PROGRAMMING 37
4.1 CANNED CYCLE (G90, G92, G94) 38
4.1.1 Outer Diameter/Internal Diameter Cutting Cycle (G90) 39
4.1.1.1 Straight cutting cycle 39
4.1.1.2 Taper cutting cycle 41
4.1.2 Threading Cycle (G92) 43
4.1.2.1 Straight threading cycle 43
4.1.2.2 Taper threading cycle 47
4.1.3 End Face Turning Cycle (G94) 50
4.1.3.1 Face cutting cycle 50
4.1.3.2 Taper cutting cycle 51
4.1.4 How to Use Canned Cycles (G90, G92, G94) 53
4.1.5 Canned Cycle and Tool Nose Radius Compensation 55
4.1.6 Restrictions on Canned Cycles 57
4.2 MULTIPLE REPETITIVE CANNED CYCLE (G70-G76) 59
4.2.1 Stock Removal in Turning (G71) 60
4.2.2 Stock Removal in Facing (G72) 74
4.2.3 Pattern Repeating (G73) 79
4.2.4 Finishing Cycle (G70) 82

4.2.5 End Face Peck Drilling Cycle (G74) 86
4.2.6 Outer Diameter / Internal Diameter Drilling Cycle (G75) 88
4.2.7 Multiple Threading Cycle (G76) 90
4.2.8 Restrictions on Multiple Repetitive Canned Cycle (G70-G76) 97
4.3 CANNED CYCLE FOR DRILLING 99
4.3.1 Front Drilling Cycle (G83)/Side Drilling Cycle (G87) 103
4.3.2 Front Tapping Cycle (G84) / Side Tapping Cycle (G88) 106
4.3.3 Front Boring Cycle (G85) / Side Boring Cycle (G89) 112
4.3.4 Canned Cycle for Drilling Cancel (G80) 113
4.3.5 Precautions to be Taken by Operator 114
4.4 RIGID TAPPING 115
4.4.1 FRONT FACE RIGID TAPPING CYCLE (G84) / SIDE FACE RIGID TAPPING CYCLE (G88) 116
4.4.2 Peck Rigid Tapping Cycle (G84 or G88) 122
4.4.3 Canned Cycle Cancel (G80) 127
4.4.4 Override during Rigid Tapping 128
4.4.4.1 Extraction override 128
4.4.4.2 Override signal 130
4.5 CANNED GRINDING CYCLE (FOR GRINDING MACHINE) 131
4.5.1 Traverse Grinding Cycle (G71) 133
4.5.2 Traverse Direct Constant-Size Grinding Cycle (G72) 136
4.5.3 Oscillation Grinding Cycle (G73) 139
4.5.4 Oscillation Direct Constant-Size Grinding Cycle (G74) 142
4.6 CHAMFERING AND CORNER R 145
4.7 MIRROR IMAGE FOR DOUBLE TURRET (G68, G69) 153
4.8 DIRECT DRAWING DIMENSION PROGRAMMING 155

5 COMPENSATION FUNCTION 161
5.1 TOOL OFFSET 162
5.1.1 Tool Geometry Offset and Tool Wear Offset 162
5.1.2 T Code for Tool Offset 163
5.1.3 Tool Selection 163
5.1.4 Offset Number 163
5.1.5 Offset 164
5.1.6 Y Axis Offset 167
5.1.6.1 Y axis offset (arbitrary axes) 167
5.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42) 168
5.2.1 Imaginary Tool Nose 169
5.2.2 Direction of Imaginary Tool Nose 171
5.2.3 Offset Number and Offset Value 173
5.2.4 Workpiece Position and Move Command 175
5.2.5 Notes on Tool Nose Radius Compensation 182
5.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION 185
5.3.1 Overview 185
5.3.2 Tool Movement in Start-up 190
5.3.3 Tool Movement in Offset Mode 196
5.3.4 Tool Movement in Offset Mode Cancel 217
5.3.5 Prevention of Overcutting Due to Tool Nose Radius Compensation 224
5.3.6 Interference Check 228
5.3.6.1 Operation to be performed if an interference is judged to occur 232
5.3.6.2 Interference check alarm function 232

5.3.6.3 Interference check avoidance function....234
5.3.7 Tool Nose Radius Compensation for Input from MDI....240
5.4 CORNER CIRCULAR INTERPOLATION (G39)....242
5.5 AUTOMATIC TOOL OFFSET (G36, G37)....244

6 MEMORY OPERATION USING SERIES 10/11 FORMAT....248
6.1 ADDRESSES AND SPECIFIABLE VALUE RANGE FOR Series 10/11 PROGRAM FORMAT....249
6.2 SUBPROGRAM CALLING....249
6.3 CANNED CYCLE....250
6.3.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)....251
6.3.1.1 Straight cutting cycle....251
6.3.1.2 Taper cutting cycle....253
6.3.2 Threading Cycle (G92)....255
6.3.2.1 Straight threading cycle....255
6.3.2.2 Taper threading cycle....259
6.3.3 End Face Turning Cycle (G94)....262
6.3.3.1 Face cutting cycle....262
6.3.3.2 Taper cutting cycle....264
6.3.4 How to Use Canned Cycles....266
6.3.5 Canned Cycle and Tool Nose Radius Compensation....268
6.3.6 Restrictions on Canned Cycles....270
6.4 MULTIPLE REPETITIVE CANNED CYCLE....272
6.4.1 Stock Removal in Turning (G71)....273
6.4.2 Stock Removal in Facing (G72)....289
6.4.3 Pattern Repeating (G73)....294
6.4.4 Finishing Cycle (G70)....297
6.4.5 End Face Peck Drilling Cycle (G74)....301
6.4.6 Outer Diameter / Internal Diameter Drilling Cycle (G75)....303
6.4.7 Multiple Threading Cycle (G76)....305
6.4.8 Restrictions on Multiple Repetitive Canned Cycle....313
6.5 CANNED CYCLE FOR DRILLING....315
6.5.1 Drilling Cycle, Spot Drilling Cycle (G81)....321
6.5.2 Drilling Cycle, Counter Boring (G82)....323
6.5.3 Peck Drilling Cycle (G83)....325
6.5.4 High-speed Peck Drilling Cycle (G83.1)....327
6.5.5 Tapping Cycle (G84)....329
6.5.6 Tapping Cycle (G84.2)....331
6.5.7 Boring Cycle (G85)....333
6.5.8 Boring Cycle (G89)....335
6.5.9 Canned Cycle for Drilling Cancel (G80)....337
6.5.10 Precautions to be Taken by Operator....337

7 AXIS CONTROL FUNCTIONS....338
7.1 POLYGON TURNING (G50.2, G51.2)....339
7.2 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6)....345

8 2-PATH CONTROL FUNCTION....349
8.1 OVERVIEW....350

8.2 WAITING FUNCTION FOR PATHS 351
8.3 COMMON MEMORY BETWEEN EACH PATH 352
8.4 SPINDLE CONTROL BETWEEN EACH PATH 354
8.5 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL 355
8.6 BALANCE CUT (G68, G69) 358

III. OPERATION

1 DATA INPUT/OUTPUT 363
1.1 INPUT/OUTPUT ON EACH SCREEN 364
1.1.1 Inputting and Outputting Y-axis Offset Data 364
1.1.1.1 Inputting Y-axis offset data 364
1.1.1.2 Outputting Y-axis Offset Data 365
1.2 INPUT/OUTPUT ON THE ALL IO SCREEN 366
1.2.1 Inputting and Outputting Y-axis Offset Data 367

2 SETTING AND DISPLAYING DATA 368
2.1 SCREENS DISPLAYED BY FUNCTION KEY 369
2.1.1 Setting and Displaying the Tool Offset Value 370
2.1.2 Direct Input of Tool Offset Value 374
2.1.3 Direct Input of Tool Offset Value Measured B 376
2.1.4 Counter Input of Offset value 379
2.1.5 Setting the Workpiece Coordinate System Shift Value 380
2.1.6 Setting the Y-Axis Offset 382
2.1.7 Chuck and Tail Stock Barriers 385

APPENDIX

A PARAMETERS 395
A.1 DESCRIPTION OF PARAMETERS 396
A.2 DATA TYPE 447
A.3 STANDARD PARAMETER SETTING TABLES 448

B DIFFERENCES FROM SERIES 0*i*-C 450
B.1 SETTING UNIT 452
B.1.1 Differences in Specifications 452
B.1.2 Differences in Diagnosis Display 452
B.2 AUTOMATIC TOOL OFFSET 453
B.2.1 Differences in Specifications 453
B.2.2 Differences in Diagnosis Display 454
B.3 CIRCULAR INTERPOLATION 455
B.3.1 Differences in Specifications 455
B.3.2 Differences in Diagnosis Display 455
B.4 HELICAL INTERPOLATION 456
B.4.1 Differences in Specifications 456
B.4.2 Differences in Diagnosis Display 456
B.5 SKIP FUNCTION 457
B.5.1 Differences in Specifications 457
B.5.2 Differences in Diagnosis Display 458

B.6 MANUAL REFERENCE POSITION RETURN 459
B.6.1 Differences in Specifications 459
B.6.2 Differences in Diagnosis Display 460
B.7 WORKPIECE COORDINATE SYSTEM 461
B.7.1 Differences in Specifications 461
B.7.2 Differences in Diagnosis Display 461
B.8 LOCAL COORDINATE SYSTEM 462
B.8.1 Differences in Specifications 462
B.8.2 Differences in Diagnosis Display 463
B.9 Cs CONTOUR CONTROL 464
B.9.1 Differences in Specifications 464
B.9.2 Differences in Diagnosis Display 464
B.10 MULTI-SPINDLE CONTROL 465
B.10.1 Differences in Specifications 465
B.10.2 Differences in Diagnosis Display 465
B.11 SERIAL/ANALOG SPINDLE CONTROL 466
B.11.1 Differences in Specifications 466
B.11.2 Differences in Diagnosis Display 466
B.12 CONSTANT SURFACE SPEED CONTROL 467
B.12.1 Differences in Specifications 467
B.12.2 Differences in Diagnosis Display 467
B.13 SPINDLE POSITIONING 468
B.13.1 Differences in Specifications 468
B.13.2 Differences in Diagnosis Display 469
B.14 TOOL FUNCTIONS 470
B.14.1 Differences in Specifications 470
B.14.2 Differences in Diagnosis Display 470
B.15 TOOL COMPENSATION MEMORY 471
B.15.1 Differences in Specifications 471
B.15.2 Differences in Diagnosis Display 472
B.16 INPUT OF TOOL OFFSET VALUE MEASURED B 473
B.16.1 Differences in Specifications 473
B.16.2 Differences in Diagnosis Display 473
B.17 CUSTOM MACRO 474
B.17.1 Differences in Specifications 474
B.17.2 Differences in Diagnosis Display 476
B.17.3 Miscellaneous 476
B.18 INTERRUPTION TYPE CUSTOM MACRO 477
B.18.1 Differences in Specifications 477
B.18.2 Differences in Diagnosis Display 477
B.19 PROGRAMMABLE PARAMETER INPUT (G10) 478
B.19.1 Differences in Specifications 478
B.19.2 Differences in Diagnosis Display 478
B.20 ADVANCED PREVIEW CONTROL 479
B.20.1 Differences in Specifications 479
B.20.2 Differences in Diagnosis Display 480
B.21 MACHINING CONDITION SELECTION FUNCTION 481
B.21.1 Differences in Specifications 481
B.21.2 Differences in Diagnosis Display 481

B.22 AXIS SYNCHRONOUS CONTROL 482
B.22.1 Differences in Specifications 482
B.22.2 Differences in Diagnosis Display 486
B.23 ARBITRARY ANGULAR AXIS CONTROL 487
B.23.1 Differences in Specifications 487
B.23.2 Differences in Diagnosis Display 487
B.24 RUN HOUR AND PARTS COUNT DISPLAY 488
B.24.1 Differences in Specifications 488
B.24.2 Differences in Diagnosis Display 488
B.25 MANUAL HANDLE FEED 489
B.25.1 Differences in Specifications 489
B.25.2 Differences in Diagnosis Display 490
B.26 PMC AXIS CONTROL 491
B.26.1 Differences in Specifications 491
B.26.2 Differences in Diagnosis Display 495
B.27 EXTERNAL SUBPROGRAM CALL (M198) 496
B.27.1 Differences in Specifications 496
B.27.2 Differences in Diagnosis Display 496
B.28 SEQUENCE NUMBER SEARCH 497
B.28.1 Differences in Specifications 497
B.28.2 Differences in Diagnosis Display 497
B.29 STORED STROKE CHECK 498
B.29.1 Differences in Specifications 498
B.29.2 Differences in Diagnosis Display 499
B.30 STORED PITCH ERROR COMPENSATION 500
B.30.1 Differences in Specifications 500
B.30.2 Differences in Diagnosis Display 500
B.31 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION 501
B.31.1 Differences in Specifications 501
B.31.2 Differences in Diagnosis Display 501
B.32 RESET AND REWIND 502
B.32.1 Differences in Specifications 502
B.32.2 Differences in Diagnosis Display 502
B.33 MANUAL ABSOLUTE ON AND OFF 503
B.33.1 Differences in Specifications 503
B.33.2 Differences in Diagnosis Display 503
B.34 MEMORY PROTECTION SIGNAL FOR CNC PARAMETER 504
B.34.1 Differences in Specifications 504
B.34.2 Differences in Diagnosis Display 504
B.35 EXTERNAL DATA INPUT 505
B.35.1 Differences in Specifications 505
B.35.2 Differences in Diagnosis Display 506
B.36 DATA SERVER FUNCTION 507
B.36.1 Differences in Specifications 507
B.36.2 Differences in Diagnosis Display 507
B.37 POWER MATE CNC MANAGER 508
B.37.1 Differences in Specifications 508
B.37.2 Differences in Diagnosis Display 508

B.38 CHUCK/TAIL STOCK BARRIER 509
B.38.1 Differences in Specifications 509
B.38.2 Differences in Diagnosis Display 509
B.39 THREADING CYCLE RETRACT (CANNED CUTTING CYCLE/ MULTIPLE REPETITIVE CANNED CUTTING CYCLE) 510
B.39.1 Differences in Specifications 510
B.39.2 Differences in Diagnosis Display 510
B.40 POLAR COORDINATE INTERPOLATION 511
B.40.1 Differences in Specifications 511
B.40.2 Differences in Diagnosis Display 512
B.41 PATH INTERFERENCE CHECK (2-PATH CONTROL) 513
B.41.1 Differences in Specifications 513
B.41.2 Differences in Diagnosis Display 513
B.42 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (2-PATH CONTROL) 514
B.42.1 Differences in Specifications 514
B.42.2 Differences in Diagnosis Display 518
B.43 SUPERIMPOSED CONTROL (2-PATH CONTROL) 519
B.43.1 Differences in Specifications 519
B.43.2 Differences in Diagnosis Display 520
B.44 Y AXIS OFFSET 521
B.44.1 Differences in Specifications 521
B.44.2 Differences in Diagnosis Display 521
B.45 CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION 522
B.45.1 Differences in Specifications 522
B.45.2 Differences in Diagnosis Display 527
B.46 CANNED CYCLE FOR DRILLING 528
B.46.1 Differences in Specifications 528
B.46.2 Differences in Diagnosis Display 529
B.47 CANNED CYCLE /MULTIPLE REPETITIVE CANNED CYCLE 530
B.47.1 Differences in Specifications 530
B.47.2 Differences in Diagnosis Display 530
B.48 CANNED GRINDING CYCLE 531
B.48.1 Differences in Specifications 531
B.48.2 Differences in Diagnosis Display 531
B.49 MULTIPLE RESPECTIVE CANNED CYCLE FOR TURNING 532
B.49.1 Differences in Specifications 532
B.49.2 Differences in Diagnosis Display 536
B.50 CHAMFERING AND CORNER ROUNDING 537
B.50.1 Differences in Specifications 537
B.50.2 Differences in Diagnosis Display 537
B.51 DIRECT DRAWING DIMENSIONS PROGRAMMING 538
B.51.1 Differences in Specifications 538
B.51.2 Differences in Diagnosis Display 538

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.

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

APPENDIX
Lists parameters, valid data ranges, and alarms.

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

Applicable models

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

Special symbols

This manual uses the following symbols:

- IP

Indicates a combination of axes such as X_ Y_ Z_
In the underlined position following each address, a numeric value such as a coordinate value is placed (used in PROGRAMMING.).

- ;

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

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

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

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64302EN	
CONNECTION MANUAL (HARDWARE)	B-64303EN	
CONNECTION MANUAL (FUNCTION)	B-64303EN-1	
USER'S MANUAL (Common to Lathe System/Machining Center System)	B-64304EN	
USER'S MANUAL (For Lathe System)	B-64304EN-1	*
USER'S MANUAL (For Machining Center System)	B-64304EN-2	
MAINTENANCE MANUAL	B-64305EN	
PARAMETER MANUAL	B-64310EN	
START-UP MANUAL	B-64304EN-3	
Programming		
Macro Compiler / Macro Executor PROGRAMMING MANUAL	B-64303EN-2	
Macro Compiler OPERATOR'S MANUAL	B-64304EN-5	
C Language PROGRAMMING MANUAL	B-64303EN-3	
PMC		
PMCPROGRAMMING MANUAL	B-64393EN	
Network		
PROFIBUS-DP Board OPERATOR'S MANUAL	B-64404EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64414EN	
Operation guidance function		
MANUAL GUIDE *i* (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE *i* (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN−1	
MANUAL GUIDE 0*i* OPERATOR'S MANUAL	B-64434EN	
TURN MATE *i* OPERATOR'S MANUAL	B-64254EN	

Related manuals of SERVO MOTOR αi/βi series

The following table lists the manuals related to SERVO MOTOR αi/βi series

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR αi series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR αi series DESCRIPTIONS	B-65272EN
FANUC AC SERVO MOTOR βi series DESCRIPTIONS	B-65302EN
FANUC AC SPINDLE MOTOR βi series DESCRIPTIONS	B-65312EN
FANUC SERVO AMPLIFIER αi series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER βi series DESCRIPTIONS	B-65322EN
FANUC SERVO MOTOR αis series FANUC SERVO MOTOR αi series FANUC AC SPINDLE MOTOR αi series FANUC SERVO AMPLIFIER αi series MAINTENANCE MANUAL	B-65285EN
FANUC SERVO MOTOR βis series FANUC AC SPINDLE MOTOR βi series FANUC SERVO AMPLIFIER βi series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR αi/βi series, FANUC LINEAR MOTOR LiS series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR DiS series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR αi/βi series, BUILT-IN SPINDLE MOTOR Bi series PARAMETER MANUAL	B-65280EN

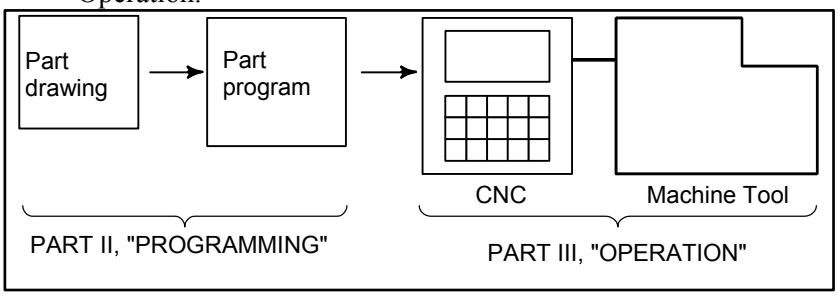
This manual mainly assumes that the FANUC SERVO MOTOR αi series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

1.1 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 Part 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 Part III, "Operation."



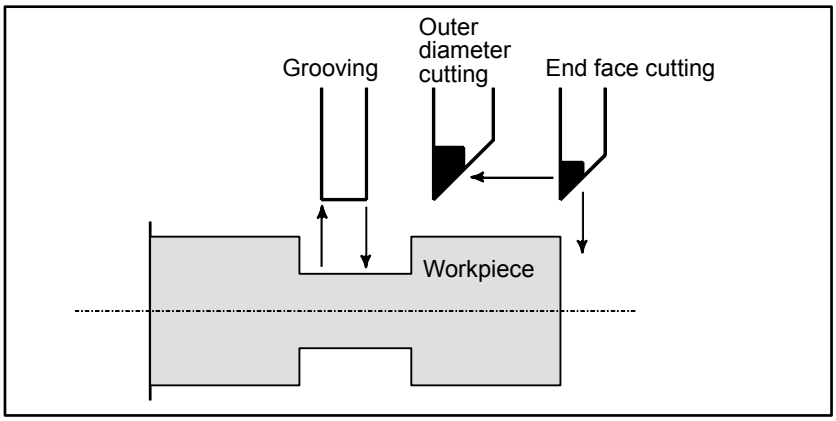
Before the actual programming, make the machining plan for how to machine the part.

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 End face cutting	2 Outer diameter cutting	3 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

⚠ CAUTION

1 The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.

2 In the header field of each page of this manual, a chapter title is indicated so that the reader can reference necessary information easily.
By finding a desired title first, the reader can reference necessary parts only.

3 This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
If a particular combination of operations is not described, it should not be attempted.

1.3 NOTES ON VARIOUS KINDS OF DATA

⚠ CAUTION

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

II. PROGRAMMING

1 GENERAL

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

1.1 OFFSET 14

1.1 OFFSET

Explanation

- 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 (see “Setting and Displaying Data” in the User’s Manual (Common to Lathe System/Machining Center System)), machining can be performed without altering the program even when the tool is changed. This function is called tool offset.

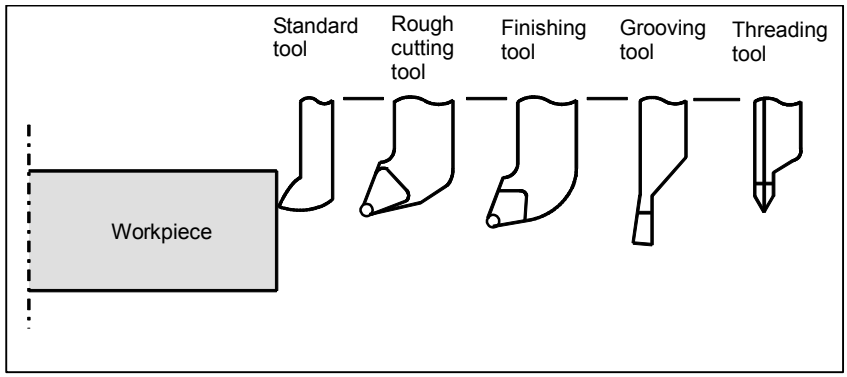


Fig. 1.1 (a) Tool offset

2 PREPARATORY FUNCTION (G FUNCTION)

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

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

(Example)

G01 and G00 are modal G codes in group 01.

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

There are three G code systems in the lathe system : A,B, and C (Table 2(a)). Select a G code system using bits 6 (GSB) and 7 (GSC) parameter No. 3401. Generally, User's Manual describes the use of G code system A, except when the described item can use only G code system B or C. In such cases, the use of G code system B or C is described.

Explanation

1. When the clear state (parameter CLR (No. 3402#6)) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◤ as indicated in Table 2.
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by parameter G23 (No. 3402#7). However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting parameter G01 (No. 3402#0).
   (5) The user can select G90 or G91 by setting parameter G91 (No. 3402#3).
   When G code system B or C is used in the lathe system, setting parameter G91 (No. 3402#3) determines which code, either G90 or G91, is effective.
2. G codes in group 00 other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010 occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle.
6. When G code system A is used, absolute or incremental programming is specified not by a G code (G90/G91) but by an address word (X/U, Z/W, C/H, Y/V). Only the initial level is provided at the return point of the canned cycle for drilling..
7. G codes are indicated by group.

Table 2 G code list

G code system			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW
G04	G04	G04	00	Dwell
G05.4	G05.4	G05.4		HRV3 on/off
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G08	G08	G08		Advanced preview control
G09	G09	G09		Exact stop
G10	G10	G10		Programmable data input
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Threading
G34	G34	G34		Variable lead threading
G36	G36	G36		Automatic tool offset (X axis)
G37	G37	G37		Automatic tool offset (Z axis)
G39	G39	G39		Tool nose radius compensation: corner rounding interpolation
G40	G40	G40	07	Tool nose radius compensation : cancel
G41	G41	G41		Tool nose radius compensation : left
G42	G42	G42		Tool nose radius compensation : right
G50	G92	G92	00	Coordinate system setting or max spindle speed clamp
G50.3	G92.1	G92.1		Workpiece coordinate system preset
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygon turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygon turning

Table 2 G code list

G code system A	G code system B	G code system C	Group	Function
G50.4	G50.4	G50.4	00	Cancel synchronous control
G50.5	G50.5	G50.5		Cancel composite control
G50.6	G50.6	G50.6		Cancel superimposed control
G51.4	G51.4	G51.4		Start synchronous control
G51.5	G51.5	G51.5		Start composite control
G51.6	G51.6	G51.6		Start superimposed control
G52	G52	G52		Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G61	G61	G61	15	Exact stop mode
G63	G63	G63		Tapping mode
G64	G64	G64		Cutting mode
G65	G65	G65	00	Macro call
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode
G69	G69	G69		Mirror image off for double turret or balance cutting mode cancel
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating cycle
G74	G74	G76		End face peck drilling cycle
G75	G75	G77		Outer diameter/internal diameter drilling cycle
G76	G76	G78		Multiple-thread cutting cycle
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct sizing/grinding cycle (for grinding machine)
G73	G73	G74		Oscillation grinding cycle (for grinding machine)
G74	G74	G75		Oscillation direct sizing/grinding cycle (for grinding machine)
G80	G80	G80	10	Canned cycle cancel for drilling Electronic gear box : synchronization cancellation
G81	G81	G81		Spot drilling (FS10/11-T format) Electronic gear box : synchronization start
G82	G82	G82		Counter boring (FS10/11-T format)
G83	G83	G83		Cycle for face drilling
G83.1	G83.1	G83.1		High-speed peck drilling cycle (FS10/11-T format)
G84	G84	G84		Cycle for face tapping
G84.2	G84.2	G84.2		Rigid tapping cycle (FS10/11-T format)
G85	G85	G85	10	Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Threading cycle
G94	G79	G24		End face turning cycle

Table 2 G code list

G code system			Group	Function
A	B	C		
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G96.1	G96.1	G96.1	00	Spindle indexing execution (waiting for completion)
G96.2	G96.2	G96.2		Spindle indexing execution (not waiting for completion)
G96.3	G96.3	G96.3		Spindle indexing completion check
G96.4	G96.4	G96.4		SV speed control mode ON
G98	G94	G94	05	Feed per minute
G99	G95	G95		Feed per revolution
-	G90	G90	03	Absolute programming
-	G91	G91		Incremental programming
-	G98	G98	11	Canned cycle : return to initial level
-	G99	G99		Canned cycle : return to R point level

3 INTERPOLATION FUNCTION

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

3.1 POLAR COORDINATE INTERPOLATION (G12.1, G13.1)...21
3.2 CONSTANT LEAD THREADING (G32).....29
3.3 VARIABLE LEAD THREADING (G34).....33
3.4 CONTINUOUS THREADING.....34
3.5 MULTIPLE THREADING.....35

3.1 POLAR COORDINATE INTERPOLATION (G12.1, G13.1)

Overview

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

Format

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

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

G112 and G113 can be used in place of G12.1 and G13.1, respectively.

Explanation

- Polar coordinate interpolation mode (G12.1)

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

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

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

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

- Polar coordinate interpolation cancel mode (G13.1)

Specifying G13.1 cancels the polar coordinate interpolation mode.

- Polar coordinate interpolation plane

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 3.1 (a)). Polar coordinate interpolation is performed on this plane.

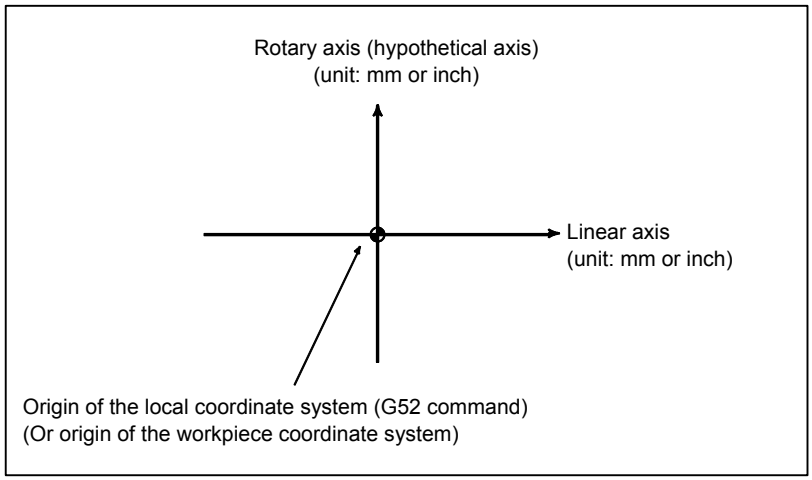


Fig. 3.1 (a) Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters Nos. 5460 and 5461 beforehand.

> ⚠ **CAUTION**
> The plane used before G12.1 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G13.1 (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).
  In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotary axis is used as the axis address for the second axis (hypothetical axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotary axis regardless of the specification for the first axis in the plane.
  The hypothetical axis is at coordinate 0 immediately after G12.1 is specified. Polar interpolation is started assuming the rotation angle of 0 for the position of the tool when G12.1 is specified.

Example)

When a value on the X-axis (linear axis) is input in millimeters

```
G12.1;
G01 X10. F1000. ; .. A 10-mm movement is made on the
                     Cartesian coordinate system.
C20. ; ..... A 20-mm movement is made on the
                     Cartesian coordinate system.
G13.1;
```

When a value on the X-axis (linear axis) is input in inches

```
G12.1;
G01 X10. F1000. ; .... A 10-inch movement is made on the
                     Cartesian coordinate system.
C20. ; ..... A 20-inch movement is made on the
                     Cartesian coordinate system.
G13.1;
```

- The unit for the feedrate is mm/min or inch/min.
  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 tool nose radius compensation.)
G65, G66, G67.....Custom macro command
G90, G91.....Absolute programming, incremental programming
(For G code system B or C)
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.

NOTE
The parallel axes U, V, and W can be used in the G code system B or C.

- 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

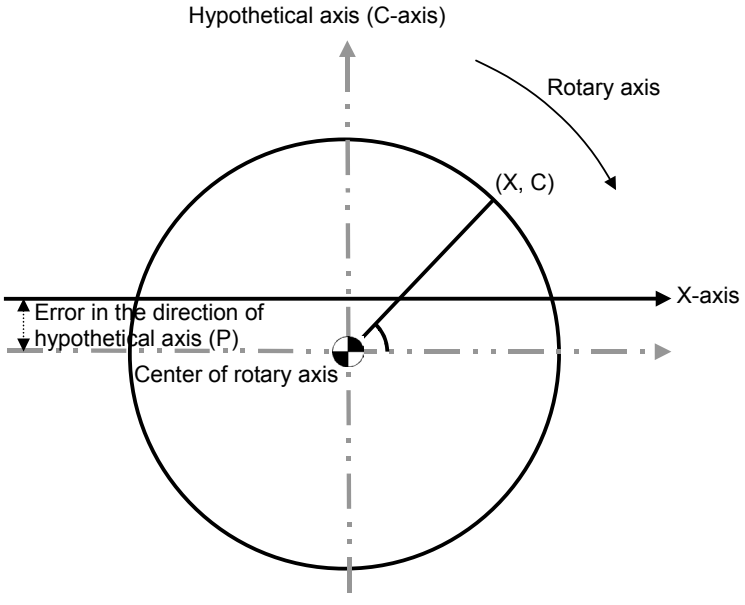
Actual coordinates are displayed. However, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane (Cartesian coordinates).

- Coordinate system for the polar coordinate interpolation

Basically, before G12.1 is specified, a local coordinate system (or workpiece coordinate system) where the center of the rotary axis is the origin of the coordinate system must be set.
In the G12.1 mode, the coordinate system must not be changed (G50, G52, G53, relative coordinate reset, G54 through G59, etc.).

- Compensation in the direction of the hypothetical axis in polar coordinate interpolation

If the first axis of the plane has an error from the center of the rotary axis in the hypothetical axis direction, in other words, if the rotary axis center is not on the X-axis, the hypothetical axis direction compensation function in the polar coordinate interpolation mode is used. With the function, the error is considered in polar coordinate interpolation. The amount of error is specified in parameter No. 5464.

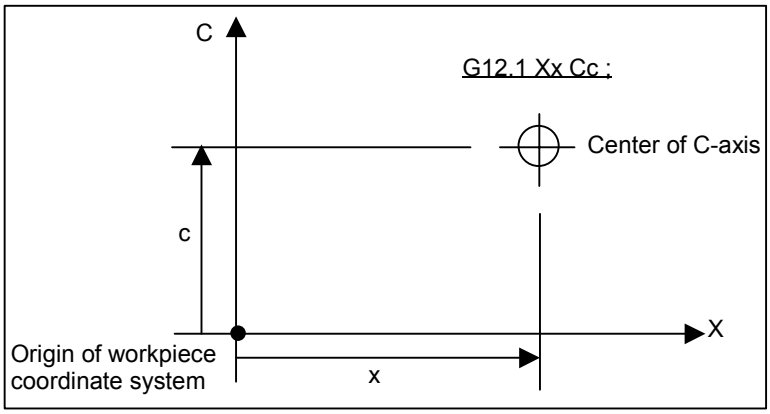


(X, C)	Point in the X-C plane (The center of the rotary axis is considered to be the origin of the X-C plane.)
X	X coordinate in the X-C plane
C	Hypothetical axis coordinate in the X-C plane
P	Error in the direction of the hypothetical axis (specified in parameter No. 5464)

- Shifting the coordinate system in polar coordinate interpolation

In the polar coordinate interpolation mode, the workpiece coordinate system can be shifted. The current position display function shows the position viewed from the workpiece coordinate system before the shift. The function to shift the coordinate system is enabled when bit 2 (PLS) of parameter No. 5450 is specified accordingly.
The shift can be specified in the polar coordinate interpolation mode, by specifying the position of the center of the rotary axis C (A, B) in the X-C (Y-A, Z-B) interpolation plane with reference to the origin of the workpiece coordinate system, in the following format.

```
G12.1 X_ C_ ;   (Polar coordinate interpolation for the X-axis and C-axis)
G12.1 Y_ A_ ;   (Polar coordinate interpolation for the Y-axis and A-axis)
G12.1 Z_ B_ ;   (Polar coordinate interpolation for the Z-axis and B-axis)
```



Limitation

- Changing the coordinate system during polar coordinate interpolation

In the G12.1 mode, the coordinate system must not be changed (G92, G52, G53, relative coordinate reset, G54 through G59, etc.).

- Tool nose radius compensation

The polar coordinate interpolation mode (G12.1 or G13.1) cannot be started or terminated in the tool nose radius compensation mode (G41 or G42). G12.1 or G13.1 must be specified in the tool nose radius compensation canceled mode (G40).

- Tool offset command

A tool offset must be specified before the G12.1 mode is set. No offset can be changed in the G12.1 mode.

- Program restart

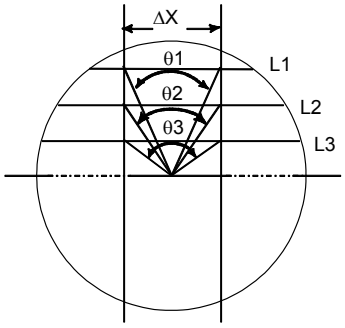
For a block in the G12.1 mode, the program cannot be restarted.

- Cutting feedrate for the rotary axis

Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotary axis (C-axis) and the linear axis (X-axis). When the tool comes close to the center of the workpiece, the C-axis velocity component increases. If the maximum cutting feedrate for the C-axis (parameter No. 1430) is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.
If the maximum cutting feedrate for the X-axis is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.

⚠ 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 θ1 to θ2 to θ3. 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. If the maximum cutting feedrate for the C-axis is exceeded, the automatic speed control function for polar coordinate interpolation automatically controls the feedrate.

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

- Automatic speed control for polar coordinate interpolation

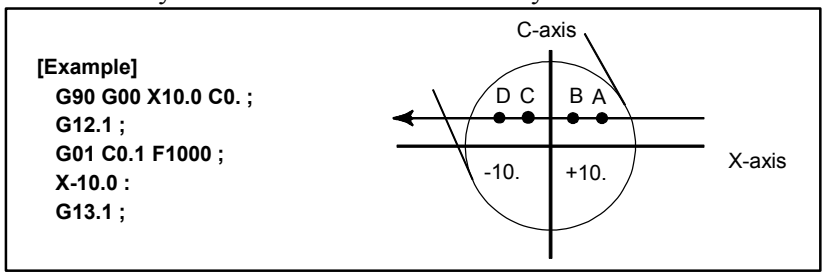
If the velocity component of the rotary axis exceeds the maximum cutting feedrate in the polar coordinate interpolation mode, the speed is automatically controlled.

- Automatic override

If the velocity component of the rotary axis exceeds the permissible velocity (maximum cutting feedrate multiplied by the permission factor specified in parameter No. 5463), the feedrate is automatically overridden as indicated below.
Override = (Permissible velocity) ÷ (Velocity component of rotary axis) × 100(%)

- Automatic speed clamp

If the velocity component of the rotary axis after automatic override still exceeds the maximum cutting feedrate, the speed of the rotary axis is automatically clamped. As a result, the velocity component of the rotary axis will not exceed the maximum cutting feedrate.
The automatic speed clamp function works only when the center of the tool is very close to the center of the rotary axis.



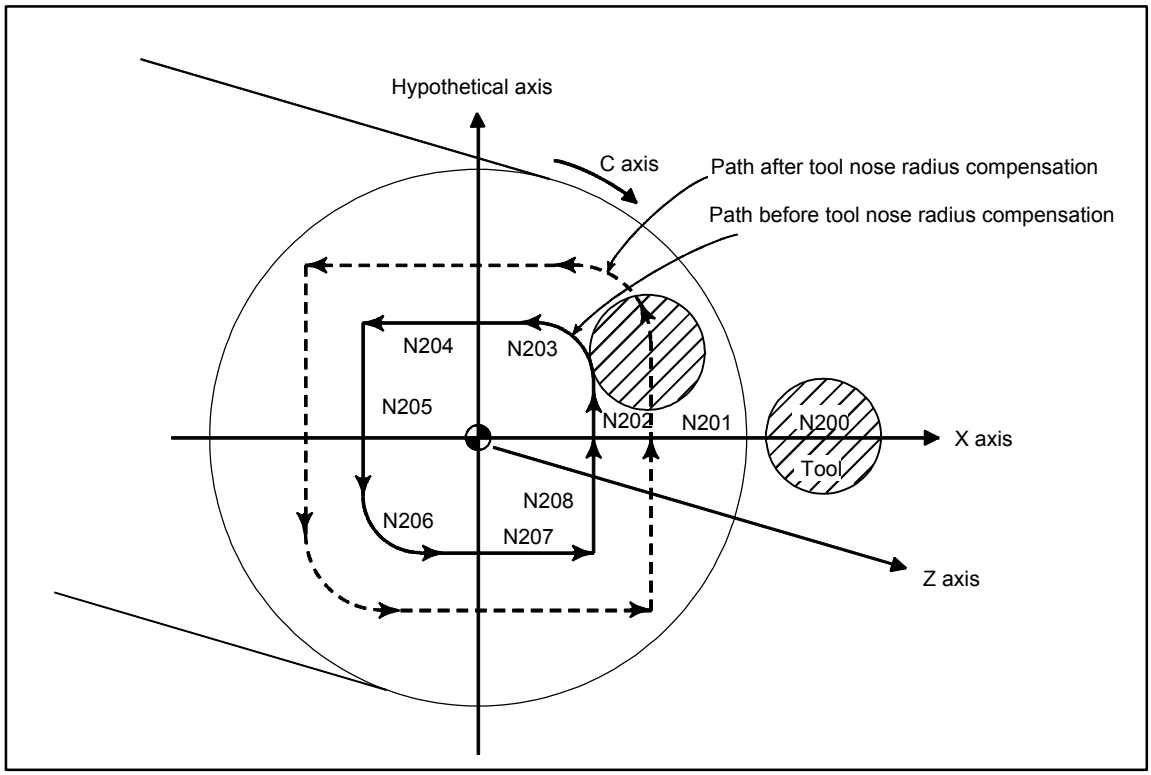
Automatic speed control for polar coordinate interpolation

Suppose that the maximum cutting feedrate of the rotary axis is 360 (3600 deg/min) and that the permission factor of automatic override for polar coordinate interpolation (parameter No. 5463) is 0 (90%). If the program indicated above is executed, the automatic override function starts working when the X coordinate becomes 2.273 (point A). The automatic speed clamp function starts working when the X coordinate becomes 0.524 (point B).
The minimum value of automatic override for this example is 3%. The automatic speed clamp function continues working until the X coordinate becomes -0.524 (point C). Then, the automatic override function works until the X coordinate becomes -2.273 (point D).
(The coordinates indicated above are the values in the Cartesian coordinate system.)

NOTE
1 While the automatic speed clamp function is working, the machine lock or interlock function may not be enabled immediately.
2 If a feed hold stop is made while the automatic speed clamp function is working, the automatic operation halt signal is output. However, the operation may not stop immediately.
3 The clamped speed may exceed the clamp value by a few percent.

Example

Sample program for polar coordinate interpolation in a Cartesian coordinate system consisting of the X-axis (a linear axis) and a hypothetical axis



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

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

3.2 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.

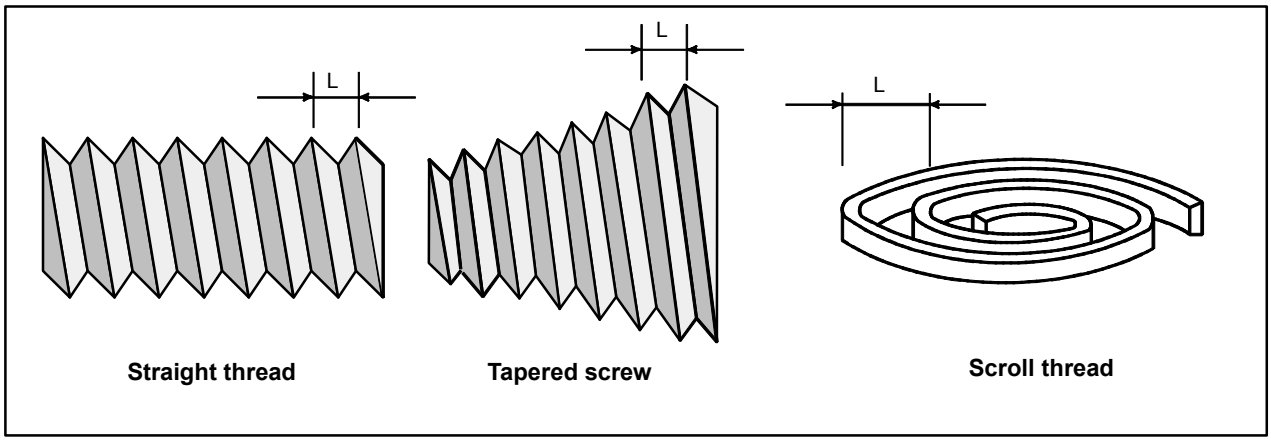


Fig. 3.2 (a) Thread types

Format

G32IP_F_;

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

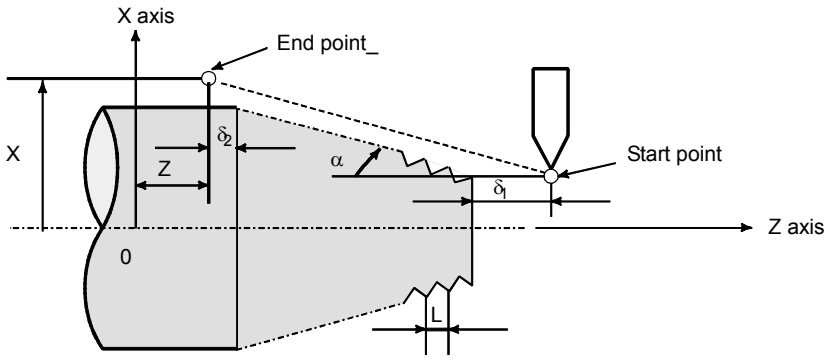


Fig. 3.2 (b) Example of threading

Explanation

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

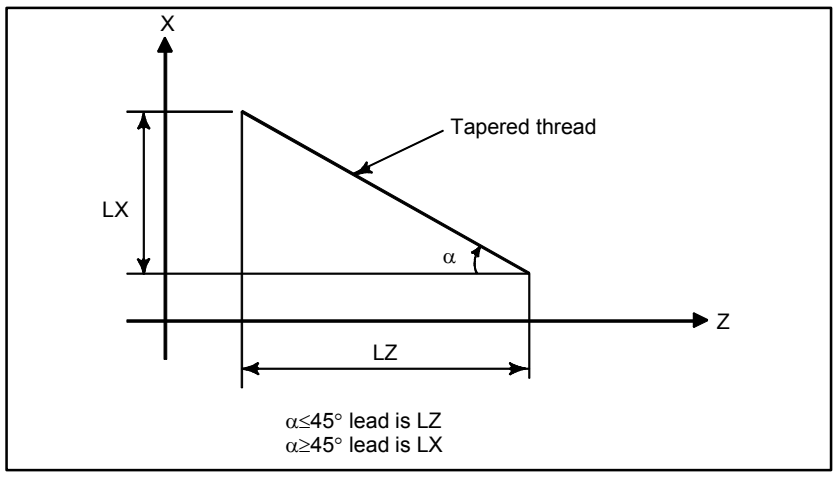


Fig. 3.2 (c) LZ and LX of a tapered thread

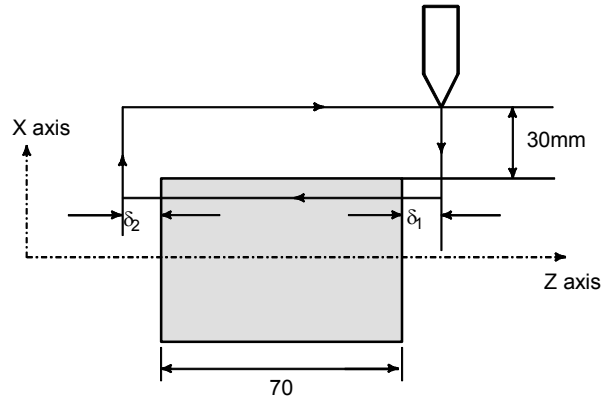
In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.
Table 3.2 (a) lists the ranges for specifying the thread lead.

Table 3.2 (a) Ranges of lead sizes that can be specified

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

Example

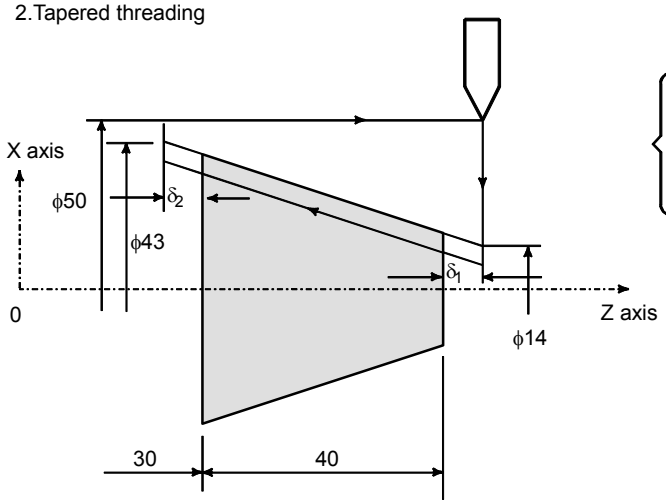
1. Straight threading



The following values are used in programming :
Thread lead :4mm
δ_1=3mm
δ_2=1.5mm
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 threading



The following values are used in programming :
Thread lead : 3.5mm in the direction of the Z axis
δ_1=2mm
δ_2=1mm
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 threading.
2 It is very dangerous to stop feeding the thread cutter without stopping the spindle. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while threading. If the feed hold button is pressed during threading, the tool will stop after a block not specifying threading 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 pressed again in the first block after threading mode that does not specify threading (or the button has been held down), the tool stops immediately at the block that does not specify threading.
4 When threading is executed in the single block status, the tool stops after execution of the first block not specifying threading.
5 When the mode was changed from automatic operation to manual operation during threading, the tool stops at the first block not specifying threading as when the feed hold button is pushed as mentioned in Warning 3.
However, when the mode is changed from one automatic operation mode to another, the tool stops after execution of the block not specifying threading as for the single block mode in Note 4.
6 When the previous block was a threading block, cutting will start immediately without waiting for detection of the one-spindle-rotation signal even if the present block is a threading block.
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 threading. Instead, use G97.
8 A movement block preceding the threading block must not specify chamfering or corner R.
9 A threading block must not specifying chamfering or corner R.
10 The spindle speed override function is disabled during threading. The spindle speed is fixed at 100%.
11 Thread cycle retract function is ineffective to G32.

3.3 VARIABLE LEAD THREADING (G34)

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

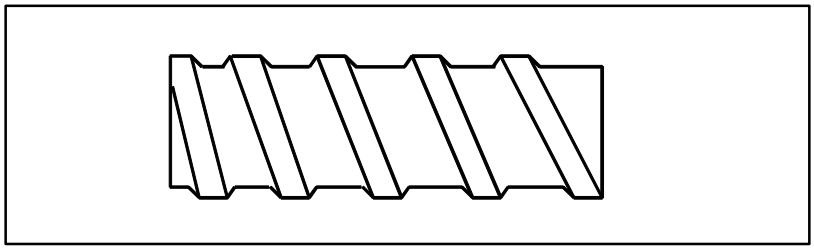


Fig. 3.3 (a) Variable lead screw

Format

G34 IP_ F_ K_ ;

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

Explanation

Address other than K are the same as in straight/taper thread cutting with G32.
The K value depends on the increment system of the reference axis, as indicated in Table 3.3 (a).
If the specified K value exceeds the range indicated in Table 3.3 (a), if the maximum lead is exceeded after a change due to the K value, or if the lead value is negative, an alarm PS0313 will be issued.

Table 3.3 (a) Range of valid K values

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

⚠ CAUTION
The "thread cutting cycle retract" is not effective for G34.

Example

Lead at the start point: 8.0 mm
Lead increment: 0.3 mm/rev

```
G34 Z-72.0 F8.0 K0.3 ;
```

3.4 CONTINUOUS THREADING

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

Explanation

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 threading operation in which the lead and shape change midway.

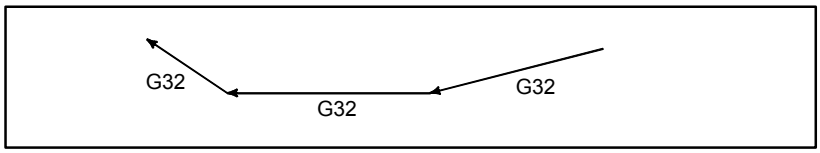


Fig. 3.4 (a) Continuous threading (Example of G32 in G code system A)

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

3.5 MULTIPLE THREADING

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

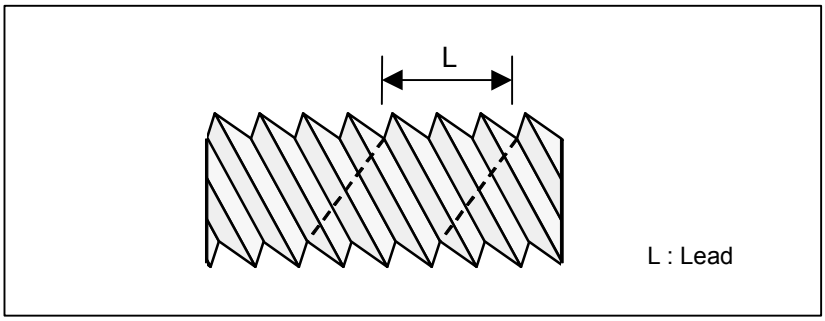


Fig. 3.5 (a) Multiple thread screws.

Format

(Constant lead threading)
G32 IP _ F_ Q_ ;
IP : End point
F_ : Lead in longitudinal direction
G32 IP _ Q_ ;
Q_ : Threading start angle

Explanation

- Available threading commands

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

Limitation

- Start angle

The start angle is not a continuous state (modal) value. It must be specified each time it is used. If a value is not specified, 0 is assumed.

- Start angle increment

The start angle (Q) increment is 0.001 degrees. Note that no decimal point can be specified.
Example:
For a shift angle of 180 degrees, specify Q180000.
Q180.000 cannot be specified, because it contains a decimal point.

- Specifiable start angle range

A start angle (Q) of between 0 and 360000 (in 0.001-degree units) can be specified. If a value greater than 360000 (360 degrees) is specified, it is rounded down to 360000 (360 degrees).

- Multiple threading cycle (G76)

For the G76 multiple threading cycle command, always use the FS10/11 command format.

Example

Program for producing double-threaded screws (with start angles of 0 and 180 degrees)

```
G00   X40.0 ;
G32   W-38.0 F4.0 Q0 ;
G00   X72.0 ;
      W38.0 ;
      X40.0 ;
G32   W-38.0 F4.0Q180000 ;
G00   X72.0 ;
      W38.0 ;
```

4 FUNCTIONS TO SIMPLIFY PROGRAMMING

Chapter 4, "FUNCTIONS TO SIMPLIFY PROGRAMMING", consists of the following sections:

4.1 CANNED CYCLE (G90, G92, G94).....38
4.2 MULTIPLE REPETITIVE CANNED CYCLE (G70-G76).....59
4.3 CANNED CYCLE FOR DRILLING 99
4.4 RIGID TAPPING.....115
4.5 CANNED GRINDING CYCLE (FOR GRINDING MACHINE).....131
4.6 CHAMFERING AND CORNER R.....145
4.7 MIRROR IMAGE FOR DOUBLE TURRET (G68, G69).....153
4.8 DIRECT DRAWING DIMENSION PROGRAMMING 155

4.1 CANNED CYCLE (G90, G92, G94)

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

NOTE
1 Explanatory figures in this section use the ZX plane as the selected plane, diameter programming for the X-axis, and radius programming for the Z-axis. When radius programming is used for the X-axis, change U/2 to U and X/2 to X.
2 A canned cycle can be performed on any plane (including parallel axes for plane definition). When G-code system A is used, however, U, V, and W cannot be set as a parallel axis.
3 The direction of the length means the direction of the first axis on the plane as follows:
ZX plane: Z-axis direction
YZ plane: Y-axis direction
XY plane: X-axis direction
4 The direction of the end face means the direction of the second axis on the plane as follows:
ZX plane: X-axis direction
YZ plane: Z-axis direction
XY plane: Y-axis direction

4.1.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

This cycle performs straight or taper cutting in the direction of the length.

4.1.1.1 Straight cutting cycle

Format

G90X(U)_Z(W)_F_;

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

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

F_ : Cutting feedrate

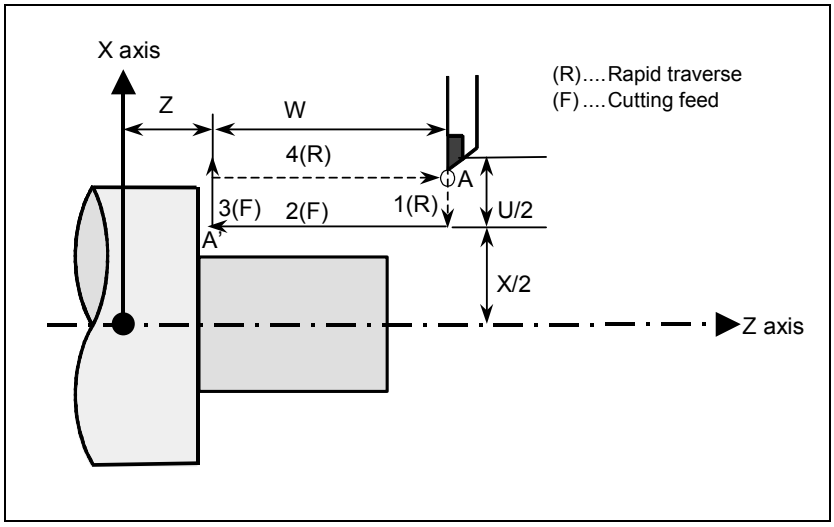


Fig. 4.1.1 (a) Straight cutting cycle

Explanation

- Operations

A straight cutting cycle performs four operations:

(1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

(2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the length.)

(3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in cutting feed.

(4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

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

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

4.1.1.2 Taper cutting cycle

Format

G90 X(U)_Z(W)_R_F_;

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

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

R_ : Taper amount (R in the figure below)

F_ : Cutting feedrate

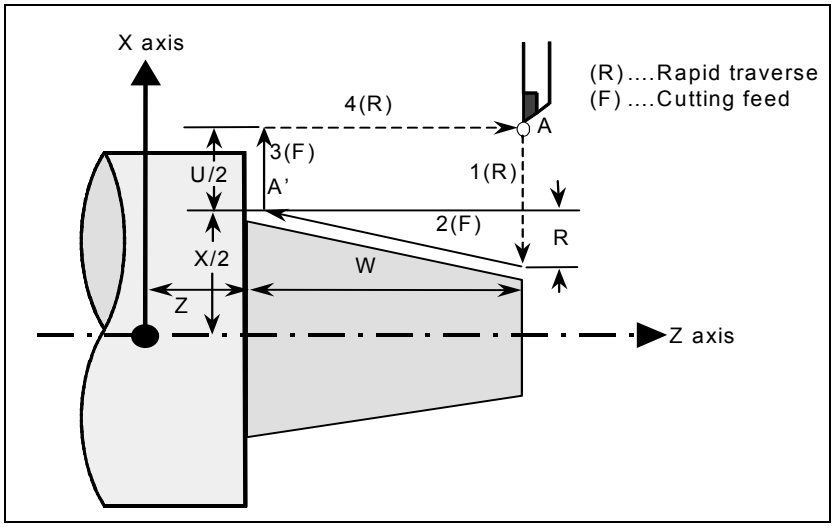


Fig. 4.1.1 (b) Taper cutting cycle

Explanation

The figure of a taper is determined by the coordinates of the cutting end point (A') in the direction of the length and the sign of the taper amount (address R). For the cycle in the figure above, a minus sign is added to the taper amount.

NOTE
The increment system of address R for specifying a taper depends on the increment system for the reference axis. Specify a radius value at R.

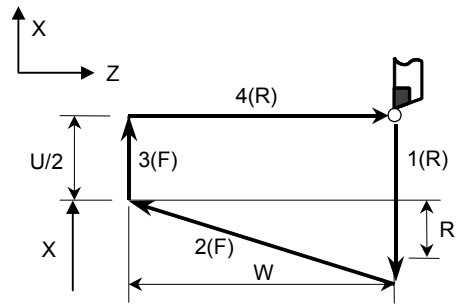
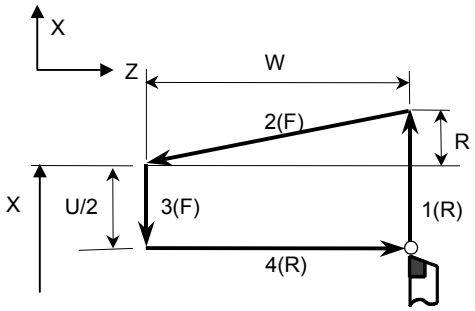
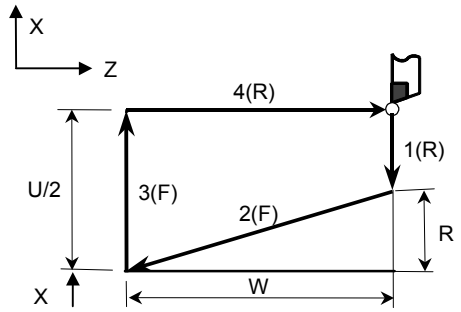
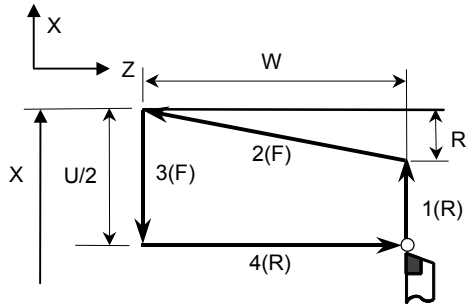
- Operations

A taper cutting cycle performs the same four operations as a straight cutting cycle.
However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
Operations 2, 3, and 4 after operation 1 are the same as for a straight cutting cycle.

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

- Relationship between the sign of the taper amount and tool path

The tool path is determined according to the relationship between the sign of the taper amount (address R) and the cutting end point in the direction of the length in the absolute or incremental programming as follows.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, R < 0	**2. U > 0, W < 0, R > 0**
	
3. U < 0, W < 0, R > 0 at |R|≤|U/2|	**4. U > 0, W < 0, R < 0 at |R|≤|U/2|**
	

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

4.1.2 Threading Cycle (G92)

4.1.2.1 Straight threading cycle

Format

G92 X(U)_Z(W)_F_Q_;

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

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

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

F_ : Thread lead (L in the figure below)

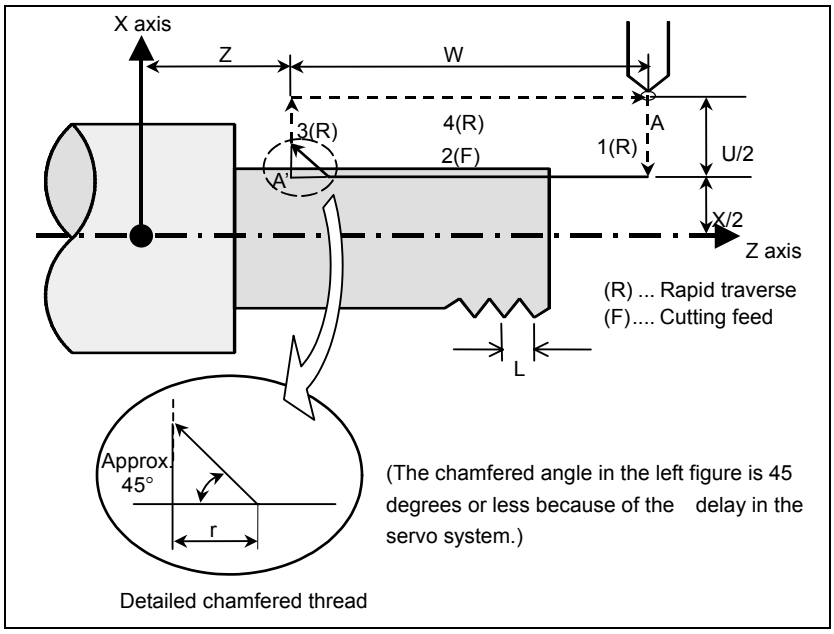


Fig. 4.1.2 (c) Straight threading

Explanation

The ranges of thread leads and restrictions related to the spindle speed are the same as for threading with G32.

- Operations

A straight threading cycle performs four operations:

(1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

(2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. At this time, thread chamfering is performed.

(3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (Retraction after chamfering)

(4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

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

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

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

- Acceleration/deceleration after interpolation for threading

Acceleration/deceleration after interpolation for threading is acceleration/deceleration of exponential interpolation type. By setting bit 5 (THLx) of parameter No. 1610, the same acceleration/deceleration as for cutting feed can be selected. (The settings of bit 0 (CTLx) of parameter No. 1610 are followed.) However, as a time constant and FL feedrate, the settings of parameter No. 1626 and No. 1627 for the threading cycle are used.

- Time constant and FL feedrate for threading

The time constant for acceleration/deceleration after interpolation for threading specified in parameter No. 1626 and the FL feedrate specified in parameter No. 1627 are used.

- Thread chamfering

Thread chamfering can be performed. A signal from the machine tool, initiates thread chamfering. The chamfering distance r 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.)
A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.
For thread chamfering, the same type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate as for threading are used.

NOTE
Common parameters for specifying the amount and angle of thread chamfering are used for this cycle and threading cycle with G76.

- Retraction after chamfering

The following table lists the feedrate, type of acceleration/deceleration after interpolation, and time constant of retraction after chamfering.

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

By setting bit 4 (ROC) of parameter No. 1403 to 1, rapid traverse override can be disabled for the feedrate of retraction after chamfering.

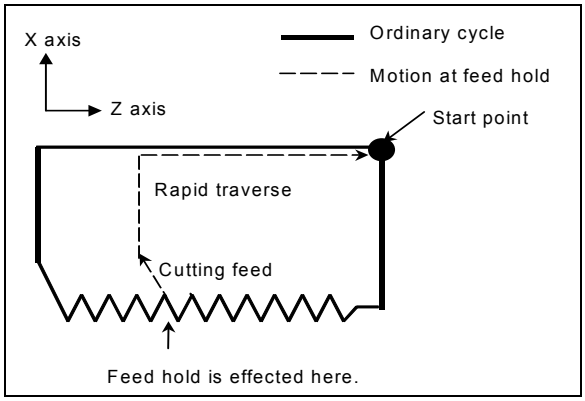
NOTE
During retraction, the machine does not stop with an override of 0% for the cutting feedrate regardless of the setting of bit 4 (RF0) of parameter No. 1401.

- Shifting the start angle

Address Q can be used to shift the threading start angle.
The start angle (Q) increment is 0.001 degrees and the valid setting range is between 0 and 360 degrees. No decimal point can be specified.

- Feed hold in a threading cycle (threading cycle retract)

Feed hold may be applied during threading (operation 2). In this case, the tool immediately retracts with chamfering and returns to the start point on the second axis (X-axis), then the first axis (Z-axis) on the plane.



The chamfered angle is the same as that at the end point.

> ⚠ **CAUTION**
> Another feed hold cannot be made during retreat.

- Inch threading

Inch threading specified with address E is not allowed.

4.1.2.2 Taper threading cycle

Format

G92 X(U)_Z(W)_R_F_Q_;

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

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

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

R_ : Taper amount (R in the figure below)

F_ : Thread lead (L in the figure below)

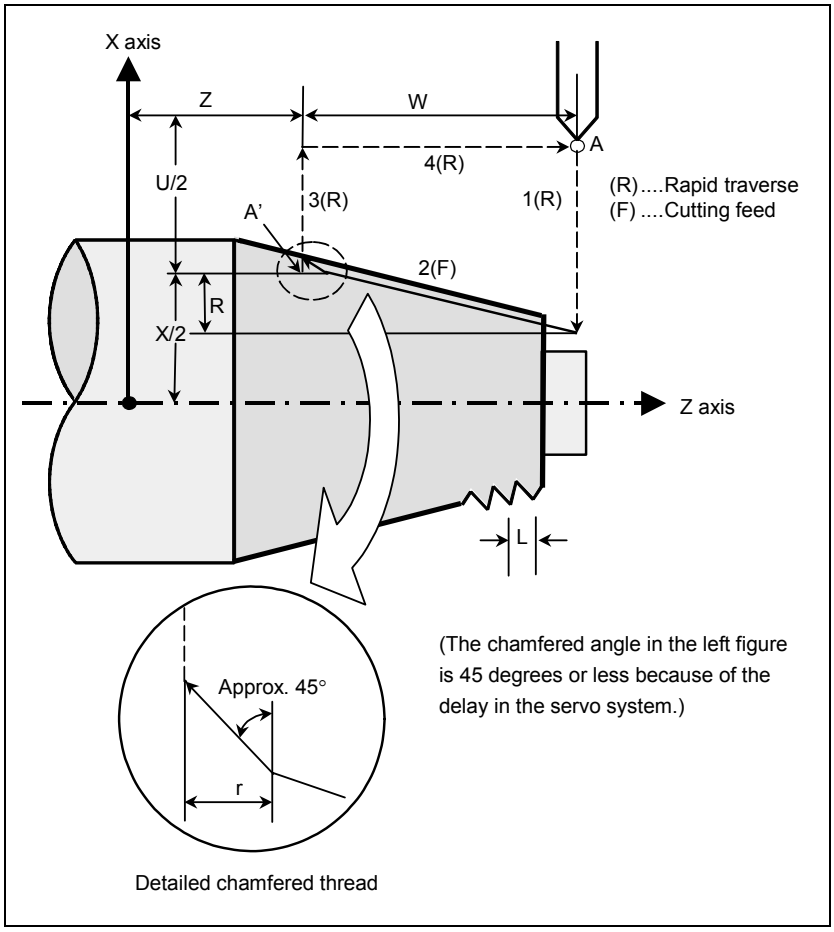


Fig. 4.1.2 (d) Taper threading cycle

Explanation

The ranges of thread leads and restrictions related to the spindle speed are the same as for threading with G32.
The figure of a taper is determined by the coordinates of the cutting end point (A') in the direction of the length and the sign of the taper amount (address R). For the cycle in the figure above, a minus sign is added to the taper amount.

NOTE
The increment system of address R for specifying a taper depends on the increment system for the reference axis. Specify a radius value at R.

- Operations

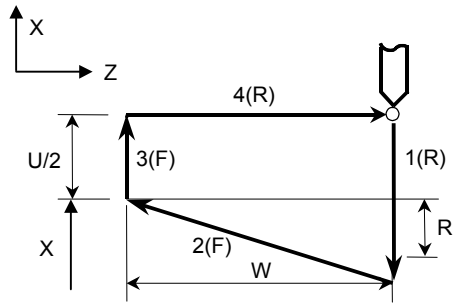
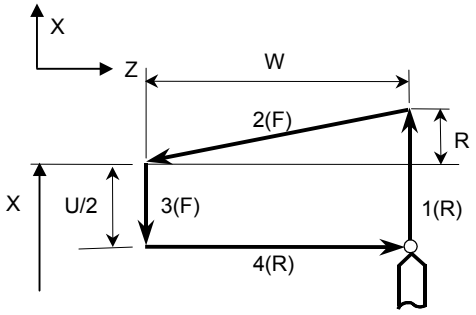
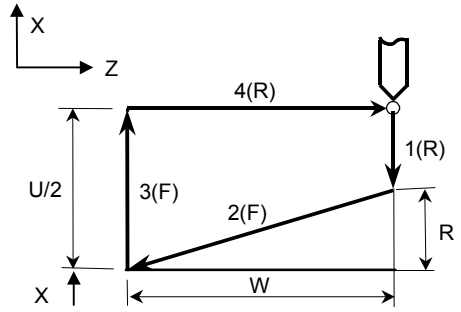
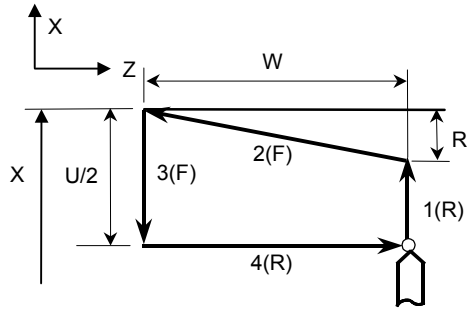
A taper threading cycle performs the same four operations as a straight threading cycle.
However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
Operations 2, 3, and 4 after operation 1 are the same as for a straight threading cycle.

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

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

- Relationship between the sign of the taper amount and tool path

The tool path is determined according to the relationship between the sign of the taper amount (address R) and the cutting end point in the direction of the length in the absolute or incremental programming as follows.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, R < 0	**2. U > 0, W < 0, R > 0**
	
3. U < 0, W < 0, R > 0 at |R|≤|U/2|	**4. U > 0, W < 0, R < 0 at |R|≤|U/2|**
	

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

- Acceleration/deceleration after interpolation for threading
- Time constant and FL feedrate for threading
- Thread chamfering
- Retraction after chamfering
- Shifting the start angle
- Threading cycle retract
- Inch threading

See the pages on which a straight threading cycle is explained.

4.1.3 End Face Turning Cycle (G94)

4.1.3.1 Face cutting cycle

Format

G94 X(U)_Z(W)_F_;

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

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

F_ : Cutting feedrate

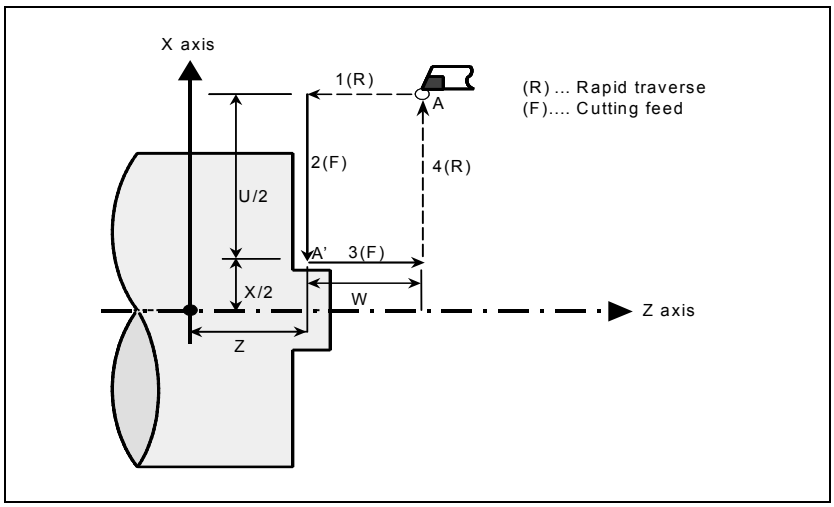


Fig. 4.1.3 (e) Face cutting cycle

Explanation

- Operations

A face cutting cycle performs four operations:

(1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.

(2) Operation 2 moves the tool to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the end face.)

(3) Operation 3 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in cutting feed.

(4) Operation 4 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

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

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

4.1.3.2 Taper cutting cycle

Format

G94 X(U)_Z(W)_R_F_;

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

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

R_ : Taper amount (R in the figure below)

F_ : Cutting feedrate

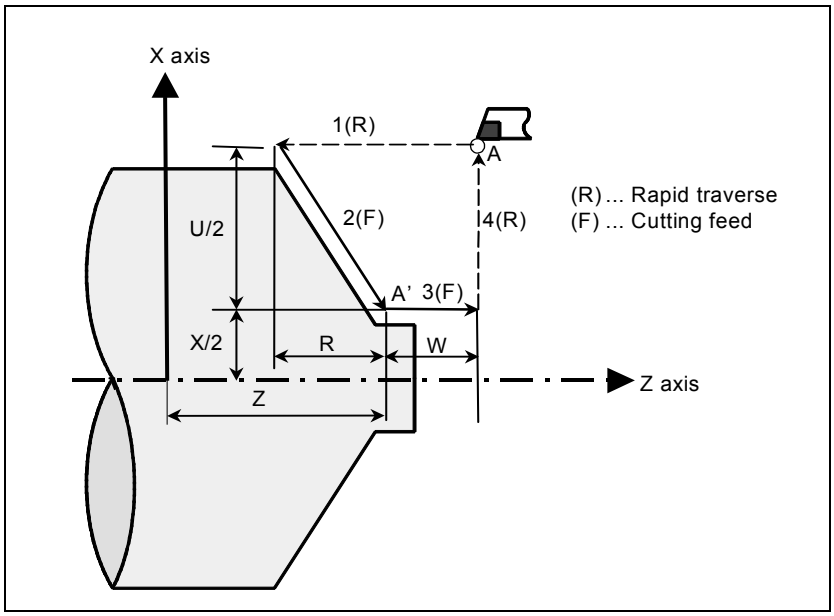


Fig. 4.1.3 (f) Taper cutting cycle

Explanation

The figure of a taper is determined by the coordinates of the cutting end point (A') in the direction of the end face and the sign of the taper amount (address R). For the cycle in the figure above, a minus sign is added to the taper amount.

NOTE
The increment system of address R for specifying a taper depends on the increment system for the reference axis. Specify a radius value at R.

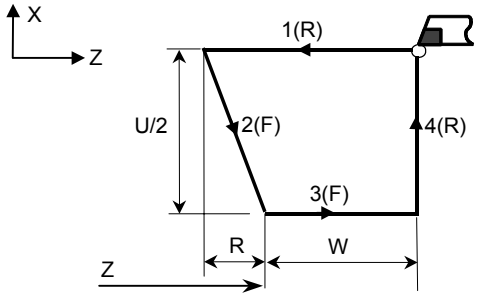
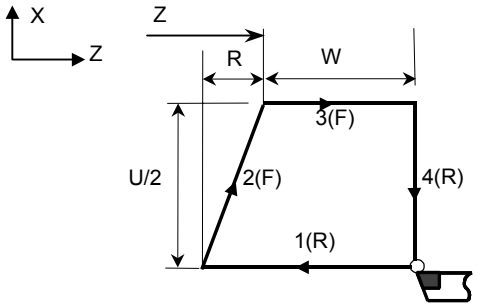
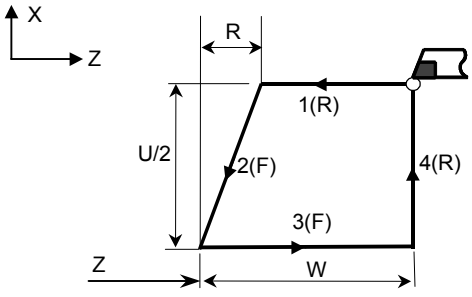
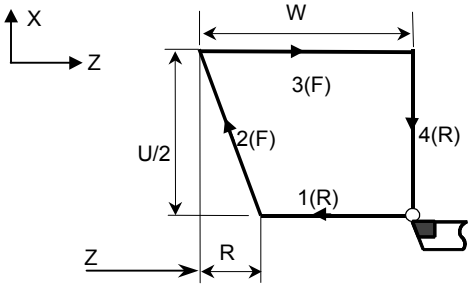
- Operations

A taper cutting cycle performs the same four operations as a face cutting cycle.
However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.
Operations 2, 3, and 4 after operation 1 are the same as for a face cutting cycle.

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

- Relationship between the sign of the taper amount and tool path

The tool path is determined according to the relationship between the sign of the taper amount (address R) and the cutting end point in the direction of the end face in the absolute or incremental programming as follows.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, R < 0	**2. U > 0, W < 0, R < 0**
	
3. U < 0, W < 0, R > 0 at |R|≤|W|	**4. U > 0, W < 0, R > 0 at |R|≤|W|**
	

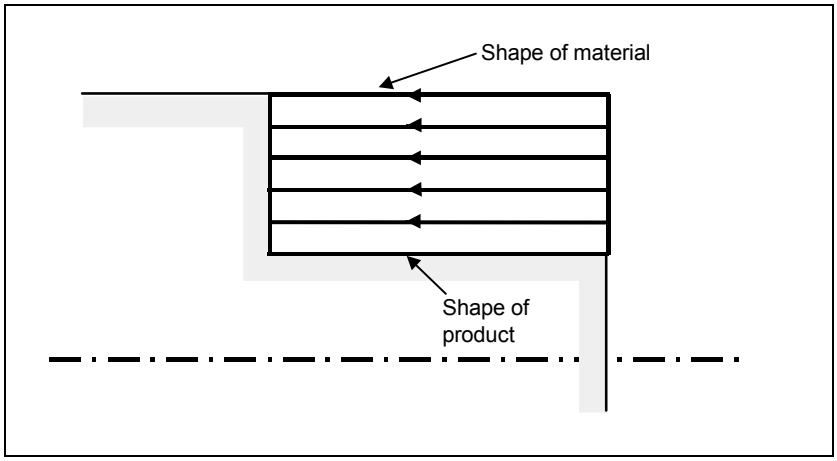
- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

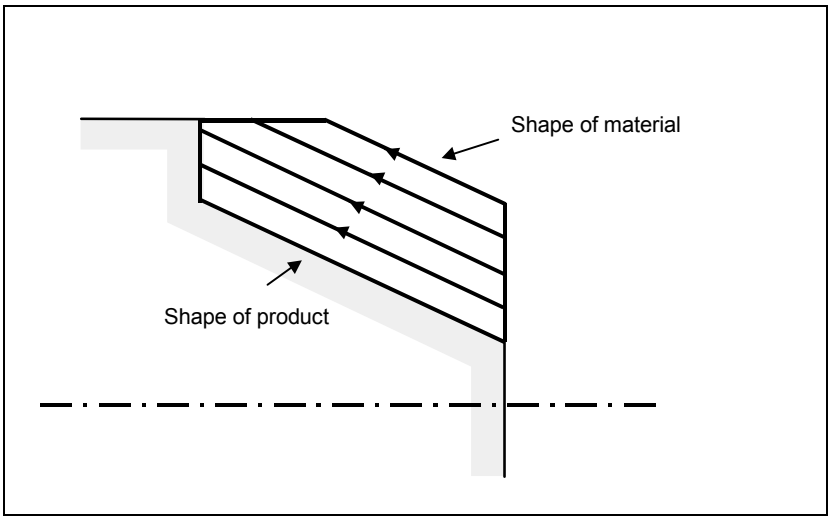
4.1.4 How to Use Canned Cycles (G90, G92, G94)

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

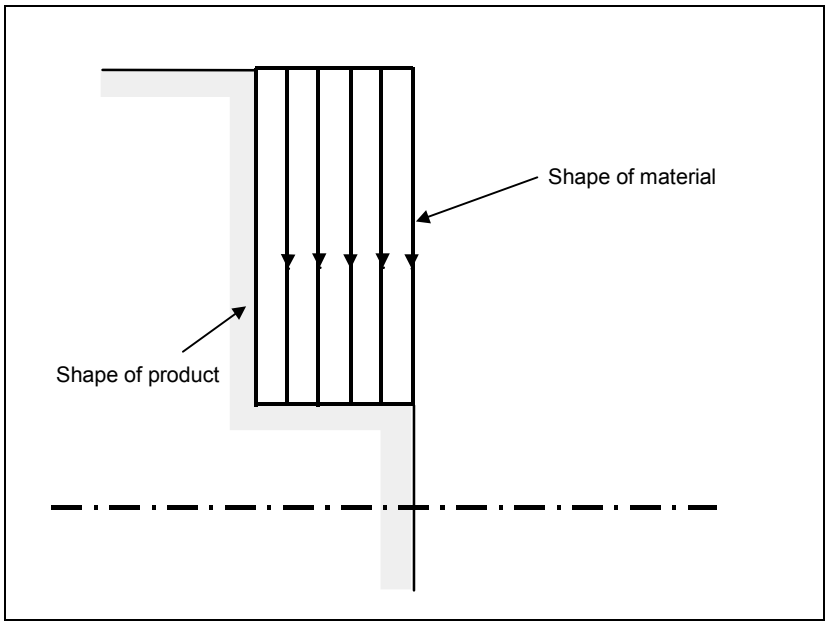
- Straight cutting cycle (G90)



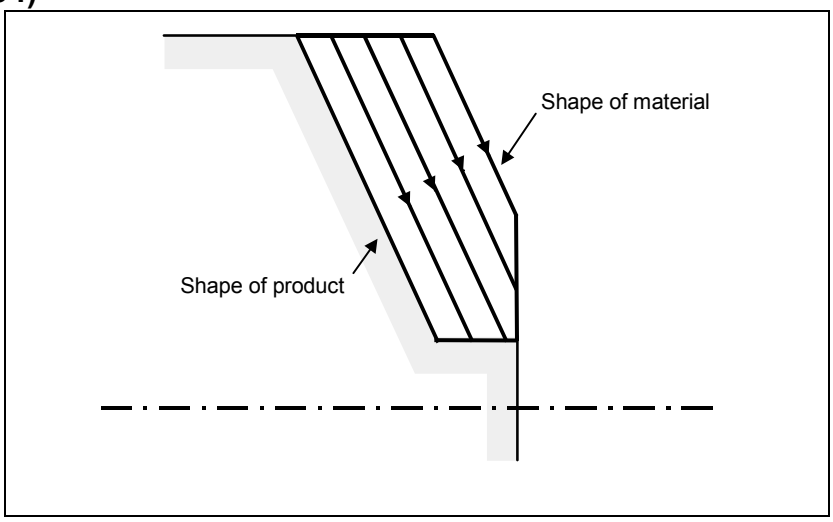
- Taper cutting cycle (G90)



- Face cutting cycle (G94)



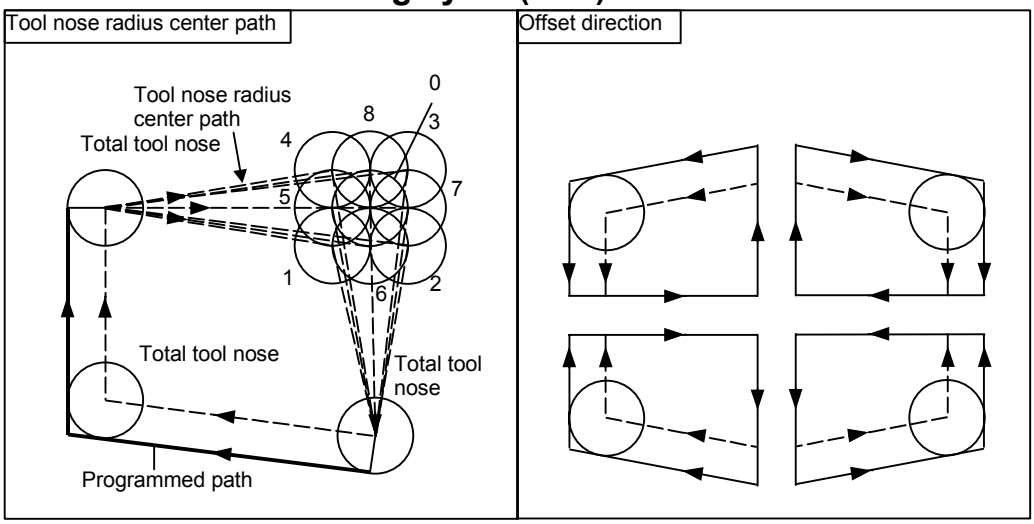
- Face taper cutting cycle (G94)



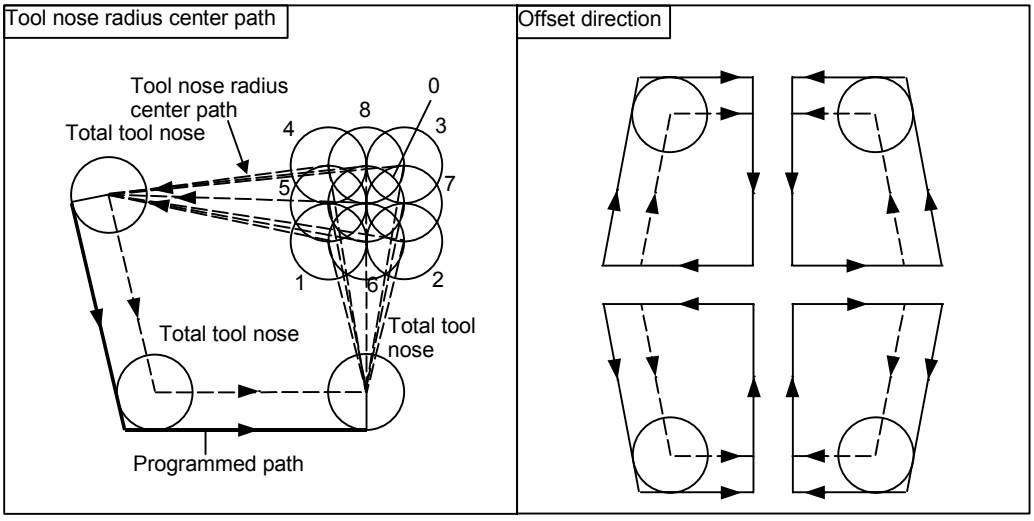
4.1.5 Canned Cycle and Tool Nose Radius Compensation

When tool nose radius compensation is applied, the tool nose center path and offset direction are as shown below. At the start point of a cycle, the offset vector is canceled. Offset start-up is performed for the movement from the start point of the cycle. The offset vector is temporarily canceled again at the return to the cycle start point and offset is applied again according to the next move command. The offset direction is determined depending of the cutting pattern regardless of the G41 or G42 mode.

Outer diameter/internal diameter cutting cycle (G90)



End face cutting cycle (G94)



Threading cycle (G92)

Tool nose radius compensation cannot be applied.

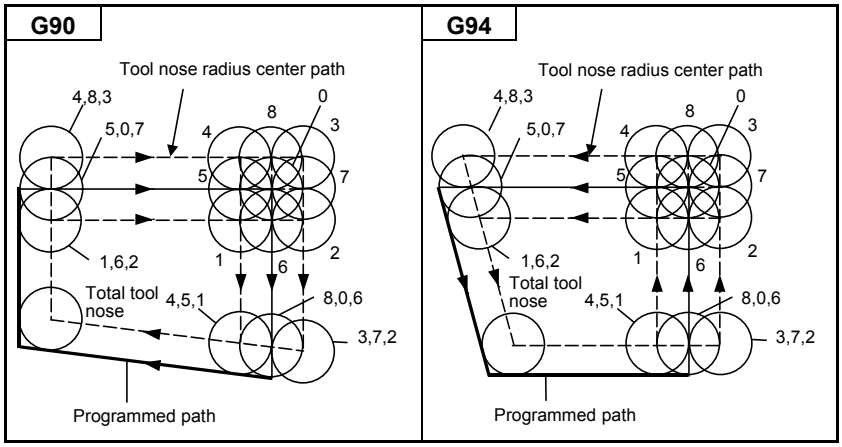
Differences between this CNC and the Series 0*i*-C

NOTE

This CNC is the same as the Series 0*i*-C in the offset direction, but differs from the series in the tool nose radius center path.

- For this CNC
  Cycle operations of a canned cycle are replaced with G00 or G01. In the first block to move the tool from the start point, start-up is performed. In the last block to return the tool to the start point, offset is canceled.
- For the Series 0*i*-C
  This series differs from this CNC in operations in the block to move the tool from the start point and the last block to return it to the start point. For details, refer to "Series 0*i*-C Operator's Manual."

How compensation is applied for the Series 0*i*-C



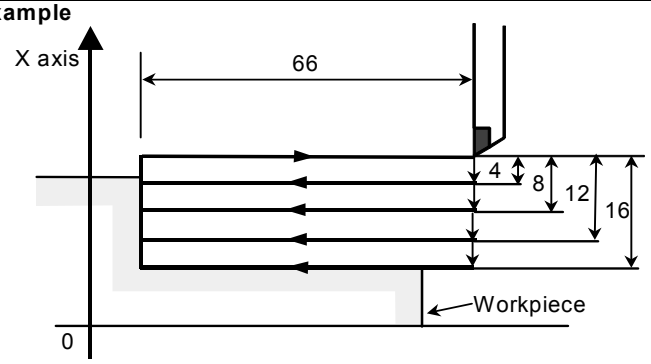
4.1.6 Restrictions on Canned Cycles

Limitation

- Modal

Since data items X (U), Z (W), and R in a canned cycle are modal values common to G90, G92, and G94. For this reason, if a new X (U), Z (W), or R value is not specified, the previously specified value is effective.
Thus, when the travel distance along the Z-axis does not vary as shown in the program example below, a canned cycle can be repeated only by specifying the travel distance along the X-axis.

Example



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;
```

The modal values common to canned cycles are cleared when a one-shot G code other than G04 is specified.
Since the canned cycle mode is not canceled by specifying a one-shot G code, a canned cycle can be performed again by specifying modal values. If no modal values are specified, no cycle operations are performed.
When G04 is specified, G04 is executed and no canned cycle is performed.

- Block in which no move command is specified

In a block in which no move command is specified in the canned cycle mode, a canned cycle is also performed. For example, a block containing only EOB or a block in which none of the M, S, and T codes, and move commands are specified is of this type of block. When an M, S, or T code is specified in the canned cycle mode, the corresponding M, S, or T function is executed together with the canned cycle. If this is inconvenient, specify a group 01 G code (G00 or G01) other than G90, G92, or G94 to cancel the canned cycle mode, and specify an M, S, or T code, as in the program example below. After the corresponding M, S, or T function has been executed, specify the canned cycle again.

Example

```
N003 T0101;
   :
   :
N010 G90 X20.0 Z10.0 F0.2;
N011 G00 T0202; ← Cancels the canned cycle mode.
N012 G90 X20.5 Z10.0;
```

- Plane selection command

Specify a plane selection command (G17, G18, or G19) before setting a canned cycle or specify it in the block in which the first canned cycle is specified.
If a plane selection command is specified in the canned cycle mode, the command is executed, but the modal values common to canned cycles are cleared.
If an axis which is not on the selected plane is specified, alarm PS0330 is issued.

- Parallel axis

When G code system A is used, U, V, and W cannot be specified as a parallel axis.

- Reset

If a reset operation is performed during execution of a canned cycle when any of the following states for holding a modal G code of group 01 is set, the modal G code of group 01 is replaced with the G01 mode:

- Reset state (bit 6 (CLR) of parameter No. 3402 = 0)
- Cleared state (bit 6 (CLR) of parameter No. 3402 = 1) and state where the modal G code of group 01 is held at reset time (bit 1 (C01) of parameter No. 3406 = 1)

Example of operation)

If a reset is made during execution of a canned cycle (X0 block) and the X20.Z1. command is executed, linear interpolation (G01) is performed instead of the canned cycle.

4.2 MULTIPLE REPETITIVE CANNED CYCLE (G70-G76)

The multiple repetitive canned cycle is 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 threading is available.

NOTE
1 Explanatory figures in this section use the ZX plane as the selected plane, diameter programming for the X-axis, and radius programming for the Z-axis. When radius programming is used for the X-axis, change U/2 to U and X/2 to X.
2 A multiple repetitive canned cycle can be performed on any plane (including parallel axes for plane definition). When G-code system A is used, however, U, V, and W cannot be set as a parallel axis.

4.2.1 Stock Removal in Turning (G71)

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

Format

ZpXp plane

G71 U(Δd) R(e) ;
G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

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

YpZp plane

G71 W(Δd) R(e) ;
G71 P(ns) Q(nf) V(Δw) W(Δu) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

XpYp plane

G71 V(Δd) R(e) ;
G71 P(ns) Q(nf) U(Δw) V(Δu) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

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

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

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

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

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

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

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

	Unit	Diameter/radius programming	Sign	Decimal point input
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
e	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed

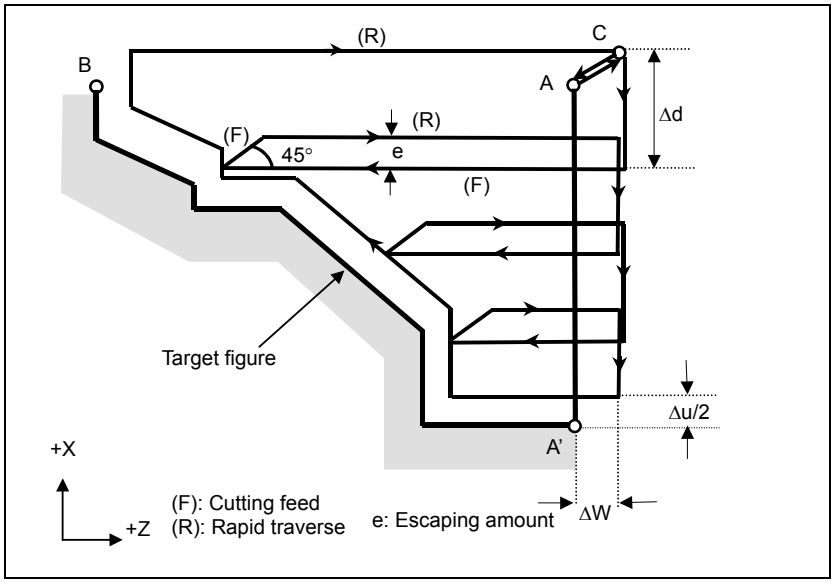


Fig. 4.2.1 (a) Cutting path in stock removal in turning (type I)

Explanation

- Operations

When a target figure passing through A, A', and B in this order is given by a program, the specified area is removed by Δd (depth of cut), with the finishing allowance specified by Δu/2 and Δw left. After the last cutting is performed in the direction of the second axis on the plane (X-axis for the ZX plane), rough cutting is performed as finishing along the target figure. After rough cutting as finishing, the block next to the sequence block specified at Q is executed.

NOTE

1 While both Δd and Δu are specified by the same address, 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.
3 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
4 When the constant surface speed control function is enabled (bit 0 (SSC) of parameter No. 8133 is set to 1), the G96 or G97 command specified in the move command between points A and B is ignored. If you want to enable the G96 or G97 command, specify the command in the G71 or previous block.

- Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the first axis on the plane (Z-axis for the ZX plane). At this time, the signs of the finishing allowances of Δu and Δw are as follows:

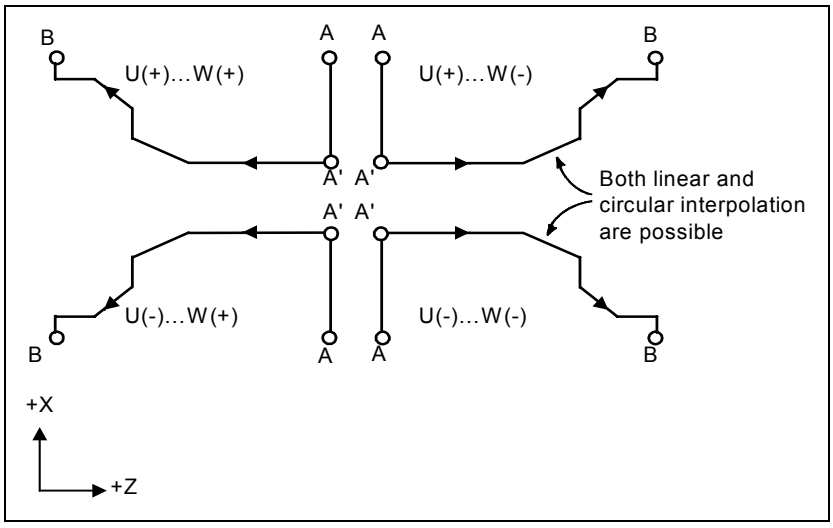


Fig. 4.2.1 (b) Four target figure patterns

Limitation

(1) For U(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For U(-), a figure for which a position lower than the cycle start point is specified cannot be machined.

(2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.

(3) For type II, the figure must show monotone increase or decrease along the first axis on the plane.

- Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.
When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.
In this start block, also select type I or II.

- Check functions

During cycle operation, whether the target figure shows monotone increase or decrease is always checked.

NOTE
When tool nose radius compensation is applied, the target figure to which compensation is applied is checked.

The following checks can also be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.
Checks the target figure before cycle operation. (Also checks that a block with the sequence number specified at address Q is contained.)	Enabled when bit 2 (FCK) of parameter No. 5104 is set to 1.

- Types I and II

Selection of type I or II

For G71, there are types I and II.
When the target figure has pockets, be sure to use type II.
Escaping operation after rough cutting in the direction of the first axis on the plane (Z-axis for the ZX plane) differs between types I and II. With type I, the tool escapes to the direction of 45 degrees. With type II, the tool cuts the workpiece along the target figure. When the target figure has no pockets, determine the desired escaping operation and select type I or II.

Selecting type I or II

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

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

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

- Type I

(1) In the block with sequence number ns, only the second axis on the plane (X-axis (U-axis) for the ZX plane) must be specified.

Example

```
ZX plane
G71 V10.0 R5.0 ;
G71 P100 Q200....;
N100 X(U)_ ;          (Specifies only the second axis on the plane.)
          :  ;
          :  ;
N200……;
```

(2) The figure along path A'-B must show monotone increase or decrease in the directions of both axes forming the plane (Z- and X-axes for the ZX plane). It must not have any pocket as shown in the figure below.

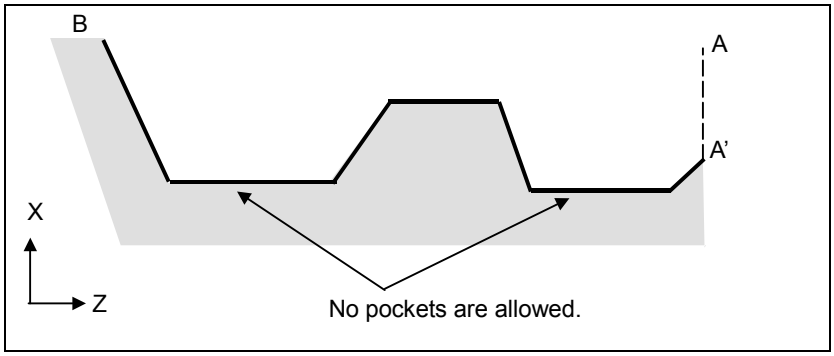


Fig. 4.2.1 (c) Figure which does not show monotone increase or decrease (type I)

> ⚠ **CAUTION**
> If a figure does not show monotone change along the first or second axis on the plane, alarm PS0064 or PS0329 is issued. If the movement does not show monotone change, but is very small, and it can be determined that the movement is not dangerous, however, the permissible amount can be specified in parameters Nos. 5145 and 5146 to specify that the alarm is not issued in this case.

(3) The tool escapes to the direction of 45 degrees in cutting feed after rough cutting.

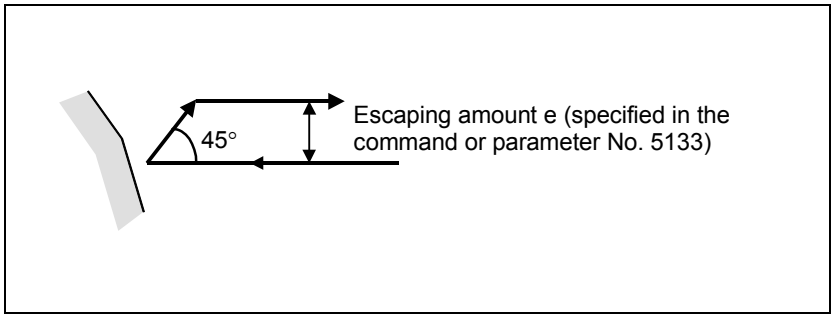


Fig. 4.2.1 (d) Cutting in the direction of 45 degrees (type I)

(4) Immediately after the last cutting, rough cutting is performed as finishing along the target figure. Bit 1 (RF1) of parameter No. 5105 can be set to 1 so that rough cutting as finishing is not performed.

- Type II

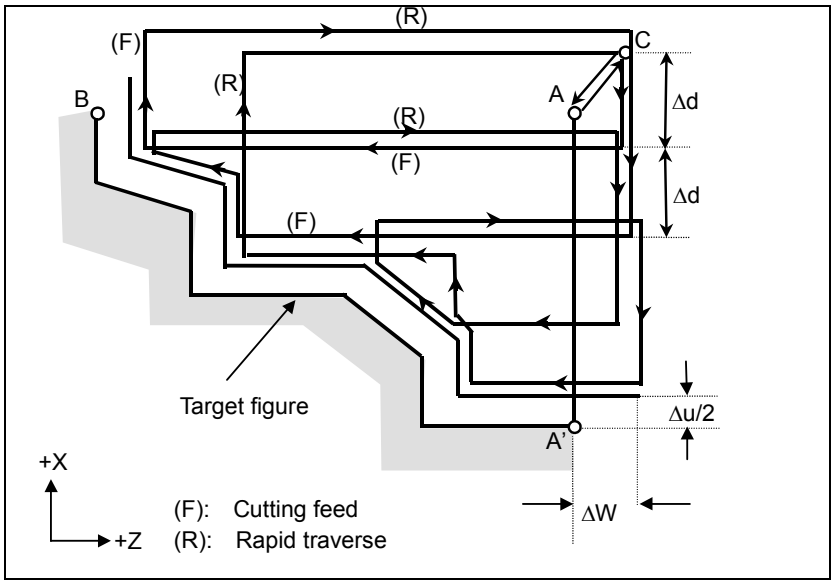


Fig. 4.2.1 (e) Cutting path in stock removal in turning (type II)

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

Type II differs from type I in the following points:

(1) In the block with sequence number ns, the two axes forming the plane (X-axis (U-axis) and Z-axis (W-axis) for the ZX plane) must be specified. When you want to use type II without moving the tool along the Z-axis on the ZX plane in the first block, specify W0.

Example

```
ZX plane
G71 V10.0 R5.0;
G71 P100 Q200.....;
N100 X(U)_ Z(W)_ ;  (Specifies the two axes
                     forming the plane.)
        :  ;
        :  ;
N200……;
```

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

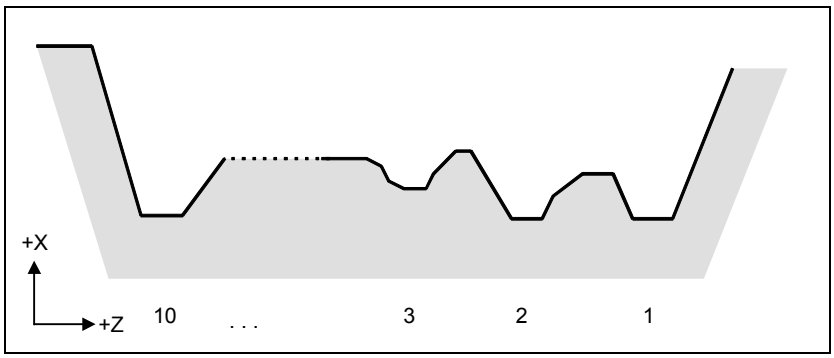


Fig. 4.2.1 (f) Figure having pockets (type II)

The figure must show monotone change in the direction of the first axis on the plane (Z-axis for the ZX plane), however. The following figure cannot be machined.

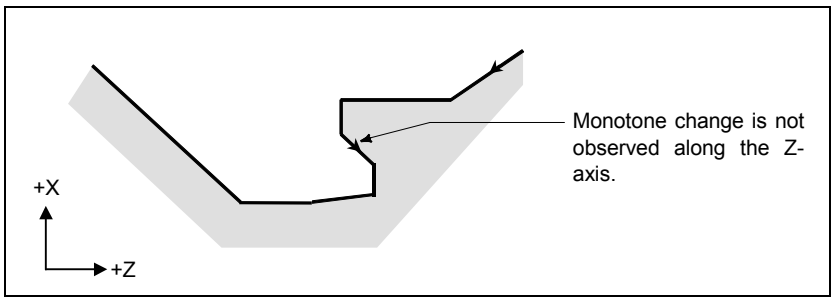


Fig. 4.2.1 (g) Figure which cannot be machined (type II)

⚠ CAUTION

For a figure along which the tool moves backward along the first axis on the plane during cutting operation (including a vertex in an arc command), the cutting tool may contact the workpiece. For this reason, for a figure which does not show monotone change, alarm PS0064 or PS0329 is issued. If the movement does not show monotone change, but is very small, and it can be determined that the movement is not dangerous, however, the permissible amount can be specified in parameter No. 5145 to specify that the alarm is not issued in this case.

The first cut portion need not be vertical. Any figure is permitted if monotone change is shown in the direction of the first axis on the plane (Z-axis for the ZX plane).

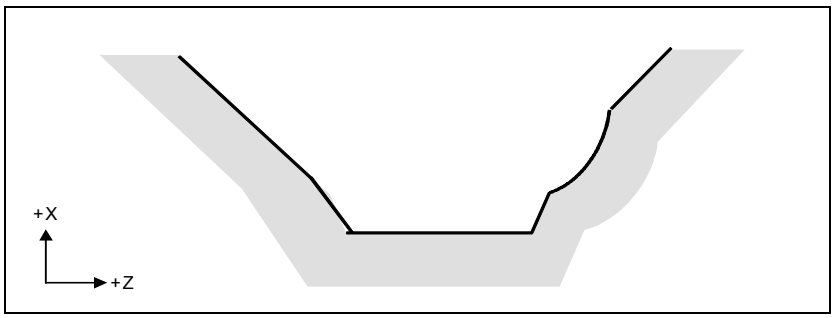


Fig. 4.2.1 (h) Figure which can be machined (type II)

(3) After turning, the tool cuts the workpiece along its figure and escapes in cutting feed.

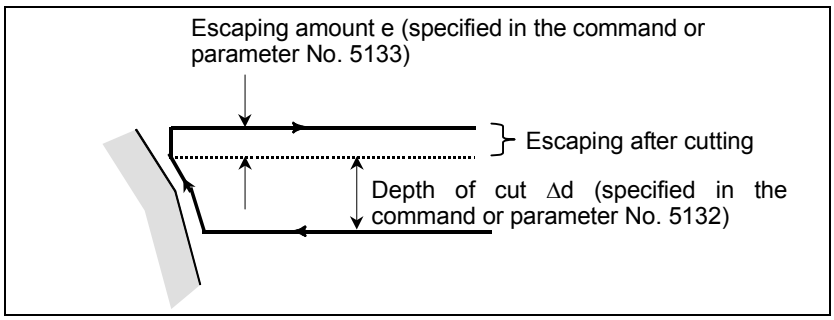


Fig. 4.2.1 (i) Cutting along the workpiece figure (type II)

The escaping amount after cutting (e) can be specified at address R or set in parameter No. 5133.
When moving from the bottom, however, the tool escapes to the direction of 45 degrees.

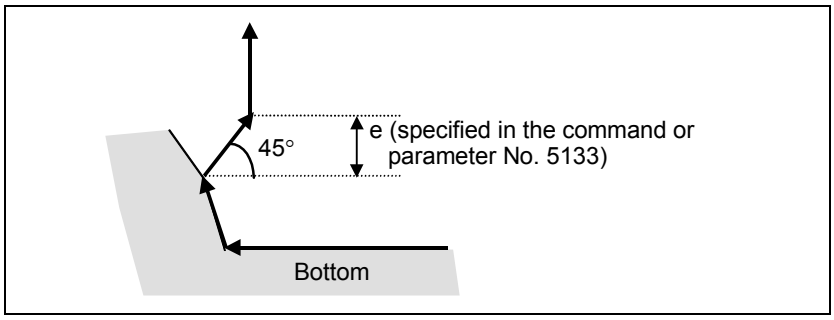


Fig. 4.2.1 (j) Escaping from the bottom to the direction of 45 degrees

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

(5) After all rough cutting terminates along the first axis on the plane (Z-axis for the ZX plane), the tool temporarily returns to the cycle start point. At this time, when there is a position whose height equals to that at the start point, the tool passes through the point in the position obtained by adding depth of cut Δd to the position of the figure and returns to the start point.
Then, rough cutting is performed as finishing along the target figure. At this time, the tool passes through the point in the obtained position (to which depth of cut Δd is added) when returning to the start point.
Bit 2 (RF2) of parameter No. 5105 can be set to 1 so that rough cutting as finishing is not performed.

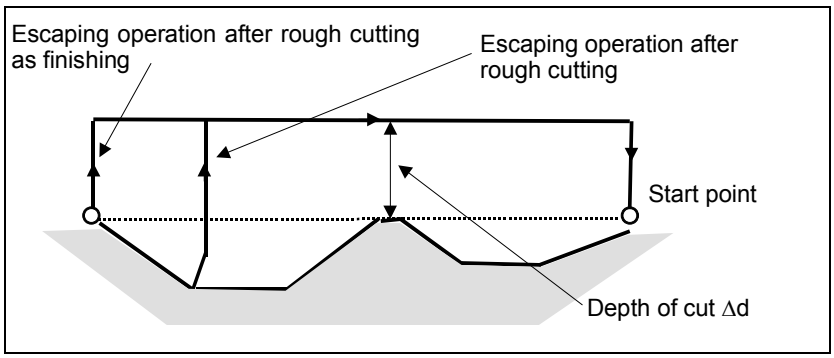


Fig. 4.2.1 (k) Escaping operation when the tool returns to the start point (type II)

(6) Order and path for rough cutting of pockets
Rough cutting is performed in the following order.
(a) When the figure shows monotone decrease along the first axis on the plane (Z-axis for the ZX plane)

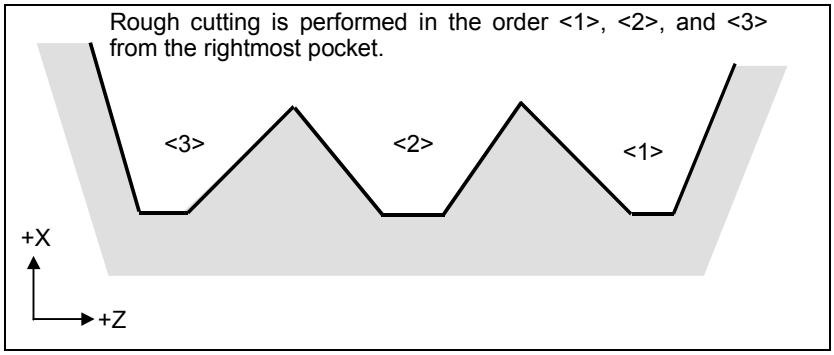


Fig. 4.2.1 (l) Rough cutting order in the case of monotone decrease (type II)

(b) When the figure shows monotone increase along the first axis on the plane (Z-axis for the ZX plane)

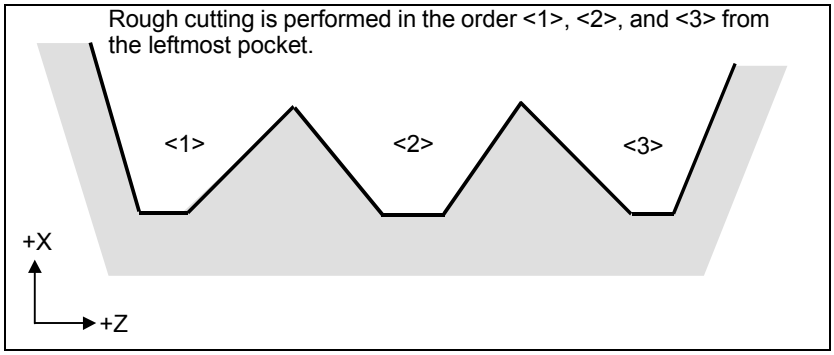


Fig. 4.2.1 (m) Rough cutting order in the case of monotone increase (type II)

The path in rough cutting is as shown below.

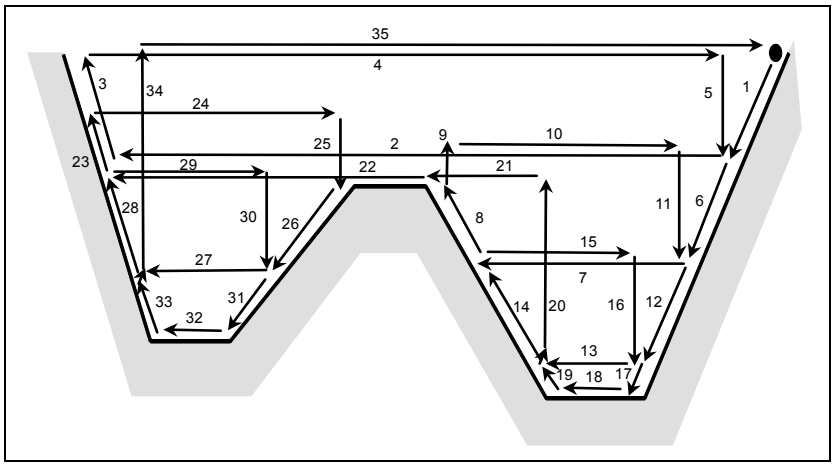


Fig. 4.2.1 (n) Cutting path for multiple pockets (type II)

The following figure shows how the tool moves after rough cutting for a pocket in detail.

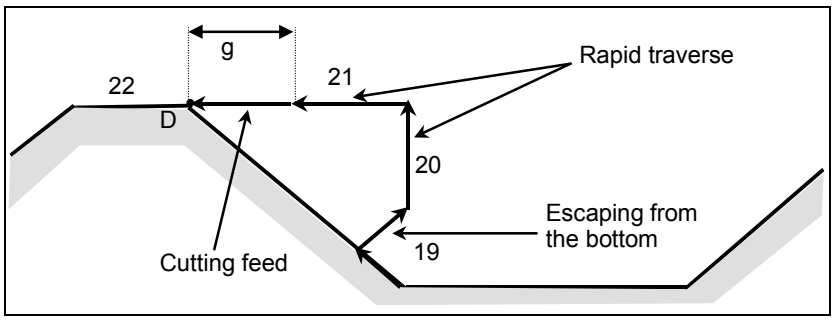


Fig. 4.2.1 (o) Details of motion after cutting for a pocket (type II)

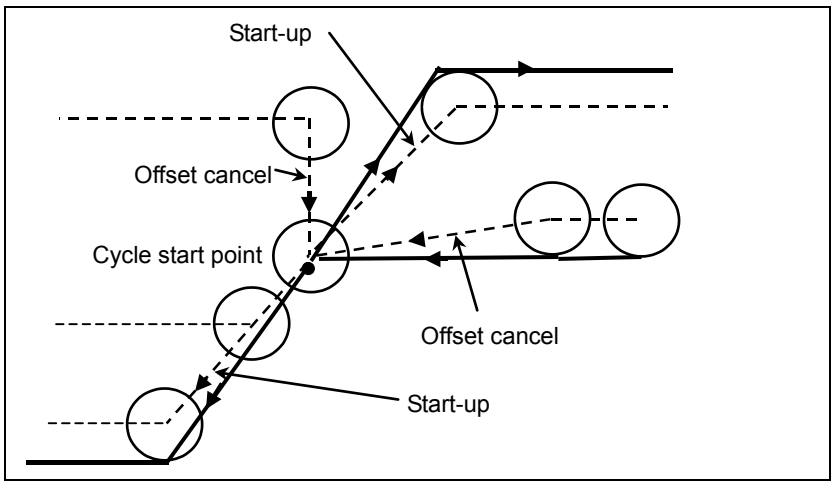
Cuts the workpiece at the cutting feedrate and escapes to the direction of 45 degrees. (Operation 19)
Then, moves to the height of point D in rapid traverse. (Operation 20)

Then, moves to the position the amount of g before point D. (Operation 21)
Finally, moves to point D in cutting feed.
The clearance g to the cutting feed start position is set in parameter No. 5134.
For the last pocket, after cutting the bottom, the tool escapes to the direction of 45 degrees and returns to the start point in rapid traverse. (Operations 34 and 35)

> ⚠ **CAUTION**
> 1 This CNC differs from the Series 0*i*-C in cutting of a pocket.
> The tool first cuts the nearest pocket to the start point. After cutting of the pocket terminates, the tool moves to the nearest but one pocket and starts cutting.
> 2 When the figure has a pocket, generally specify a value of 0 for Δw (finishing allowance). Otherwise, the tool may dig into the wall on one side.

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70, G71, G72, G73) and specify the cancel command (G40) outside the blocks (from the block specified with P to the block specified with Q) specifying a target finishing figure.
If a tool nose radius compensation command (G40, G41, or G42) is specified in the G70, G71, G72, or G73 command, alarm PS0325 is issued.
When this cycle is specified in the tool nose radius compensation mode, offset is temporarily canceled during movement to the start point. Start-up is performed in the first block. Offset is temporarily canceled again at the return to the cycle start point after termination of cycle operation. Start-up is performed again according to the next move command. This operation is shown in the figure below.



This cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

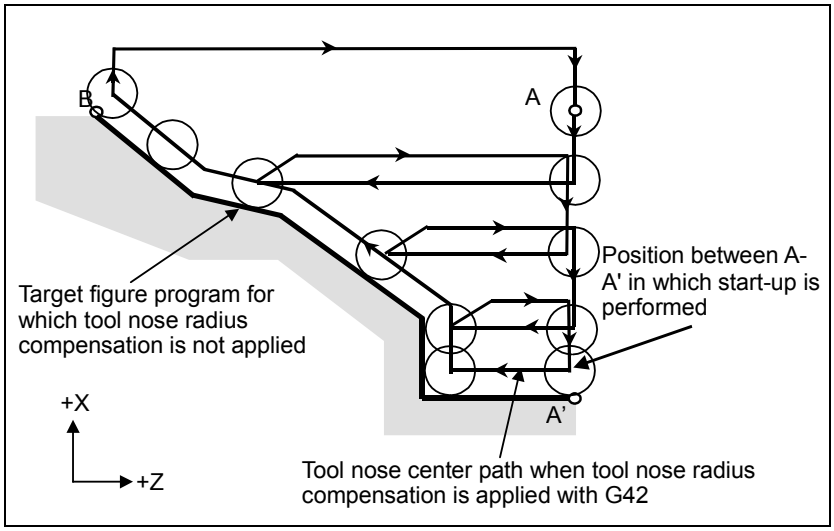
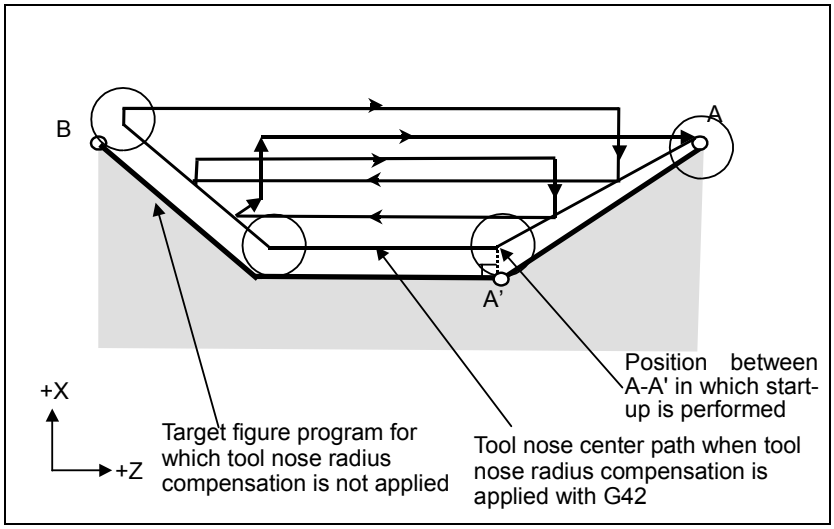


Fig. 4.2.1 (p) Path when tool nose radius compensation is applied



NOTE

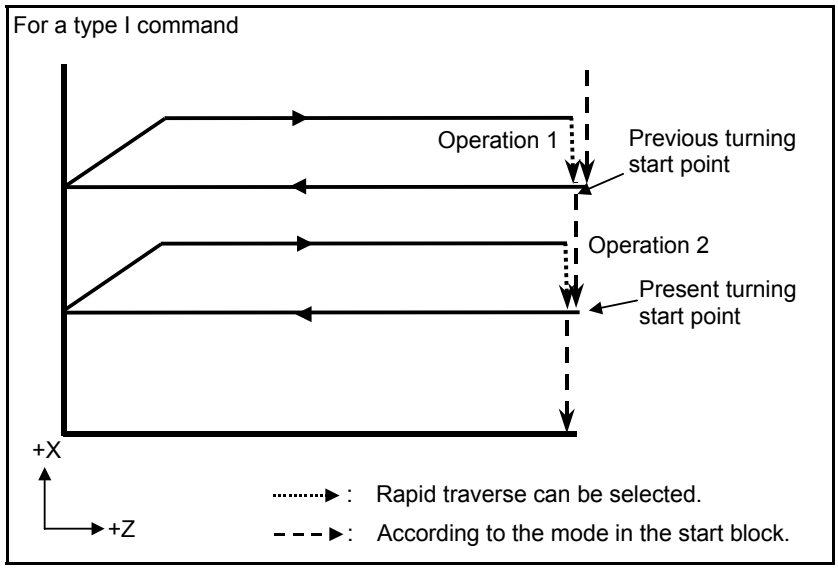
To perform pocketing in the tool nose radius compensation mode, specify the linear block A-A' outside the workpiece and specify the figure of an actual pocket. This prevents a pocket from being dug.

- Movement to the previous turning start point

Movement to the turning start point is performed with two operations. (Operations 1 and 2 in the figure below.) As movement to the present turning start point, operation 1 temporarily moves the tool to the previous turning start point, then operation 2 moves the tool to the present turning start point.

Operation 1 moves the tool in cutting feed. Operation 2 moves the tool according to the mode (G00 or G01) specified in the start block in the geometry program.

Bit 0 (ASU) of parameter No. 5107 can be set to 1 so that operation 1 moves the tool in rapid traverse.



4.2.2 Stock Removal in Facing (G72)

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

Format

ZpXp plane

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

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

YpZp plane

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

XpYp plane

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

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

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

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

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

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

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

f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G72 block is effective.

	Unit	Diameter/radius programming	Sign	Decimal point input
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
e	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed

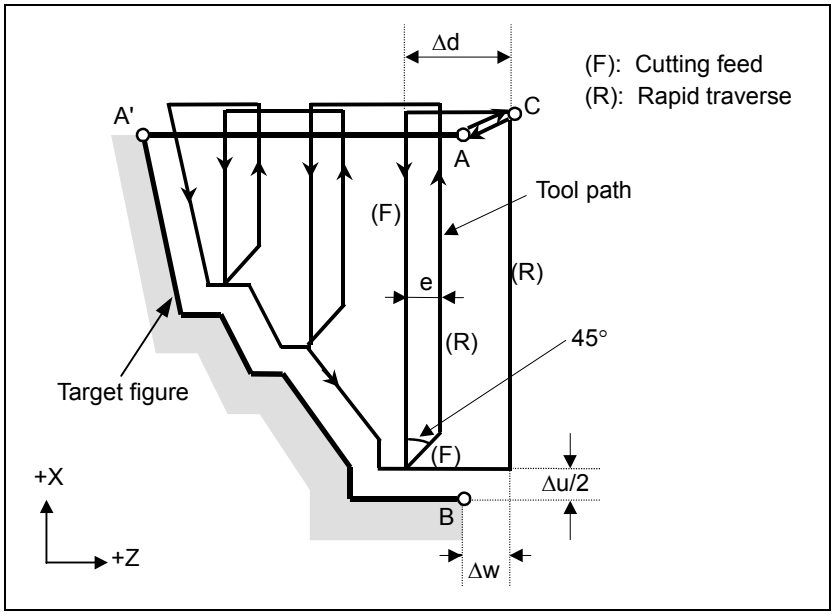


Fig. 4.2.2 (q) Cutting path in stock removal in facing (type I)

Explanation

- Operations

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

NOTE
1 While both Δd and Δu are specified by the same address, the meanings of them are determined by the presence of addresses P and Q.
2 The cycle machining is performed by G72 command with P and Q specification.
3 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G72 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
4 When the constant surface speed control function is enabled (bit 0 (SSC) of parameter No. 8133 is set to 1), the G96 or G97 command specified in the move command between points A and B is ignored. If you want to enable the G96 or G97 command, specify the command in the G71 or previous block.

- Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the second axis on the plane (X-axis for the ZX plane). At this time, the signs of the finishing allowances of Δu and Δw are as follows:

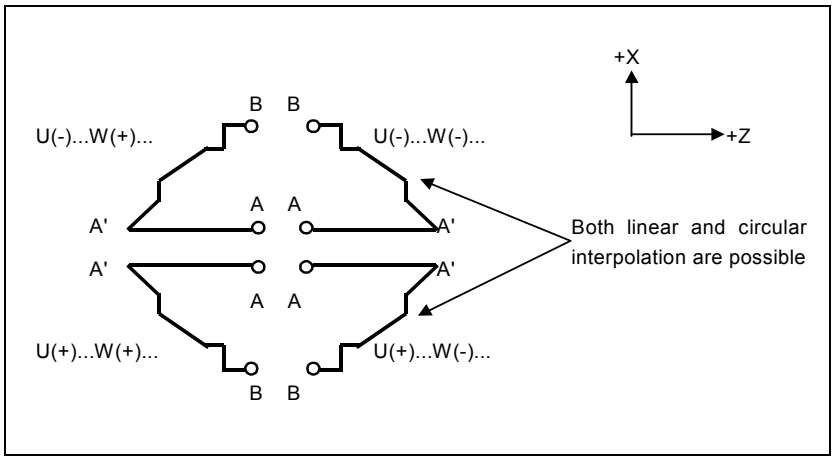


Fig. 4.2.2 (r) Signs of the values specified at U and W in stock removal in facing

Limitation

(1) For W(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For W(-), a figure for which a position lower than the cycle start point is specified cannot be machined.

(2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.

(3) For type II, the figure must show monotone increase or decrease along the second axis on the plane.

- Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.
When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.
In this start block, also select type I or II.

- Check functions

During cycle operation, whether the target figure shows monotone increase or decrease is always checked.

NOTE
When tool nose radius compensation is applied, the target figure to which compensation is applied is checked.

The following checks can also be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.
Checks the target figure before cycle operation. (Also checks that a block with the sequence number specified at address Q is contained.)	Enabled when bit 2 (FCK) of parameter No. 5104 is set to 1.

- Types I and II Selection of type I or II

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

Selecting type I or II

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

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

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

- Type I

G72 differs from G71 in the following points:

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

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

- Type II

G72 differs from G71 in the following points:

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

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

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

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

- Tool nose radius compensation

See the pages on which G71 is explained.

- Movement to the previous turning start point

See the pages on which G71 is explained.

4.2.3 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.

Format

ZpXp plane

G73 W(Δk) U(Δi) R(d) ;
G73 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

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

YpZp plane

G73 V(Δk) W(Δi) R(d) ;
G73 P(ns) Q(nf) V(Δw) W(Δu) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

XpYp plane

G73 U(Δk) V(Δi) R(d) ;
G73 P(ns) Q(nf) U(Δw) V(Δu) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

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

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

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

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

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

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

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

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

	Unit	Diameter/radius programming	Sign	Decimal point input
Δi	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed

NOTE
Decimal point input is allowed with d. However, a value rounded off to an integer is used as the number of division, regardless of the setting of bit 0 (DPI) of parameter No. 3401. When an integer is input, the input integer is used as the number of division.

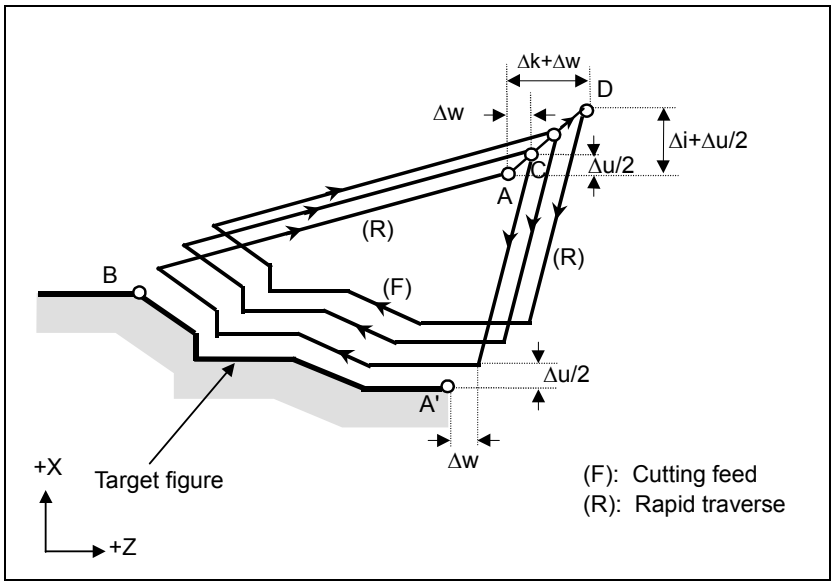


Fig. 4.2.3 (s) Cutting path in pattern repeating

Explanation

- Operations

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

NOTE
1 While the values Δi and Δk, or Δu and Δw are specified by the same address respectively, the meanings of them are determined by the presence of addresses P and Q.
2 The cycle machining is performed by G73 command with P and Q specification.
3 After cycle operation terminates, the tool returns to point A.
4 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G73 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.

- Target figure Patterns

As in the case of G71, there are four target figure patterns. Be careful about signs of Δu, Δw, Δi, and Δk when programming this cycle.

- Start block

In the start block in the program for the target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.
When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.

- Check function

The following check can be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.

- Tool nose radius compensation

Like G71, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

4.2.4 Finishing Cycle (G70)

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

Format

G70 P(ns) Q(nf) ;
ns : Sequence number of the first block for the program of finishing shape.
nf : Sequence number of the last block for the program of finishing shape.

Explanation

- Operations

The blocks with sequence numbers ns to nf in the program for a target figure are executed for finishing. The F, S, T, M, and second auxiliary functions specified in the G71, G72, or G73 block are ignored and the F, S, T, M, and second auxiliary functions specified in the blocks with sequence numbers ns to nf are effective.
When cycle operation terminates, the tool is returned to the start point in rapid traverse and the next G70 cycle block is read.

- Target figure Check function

The following check can be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.

- Storing P and Q blocks

When rough cutting is executed by G71, G72, or G73, up to three memory addresses of P and Q blocks are stored. By this, the blocks indicated by P and Q are immediately found at execution of G70 without searching memory from the beginning for them. After some G71, G72, and G73 rough cutting cycles are executed, finishing cycles can be performed by G70 at a time. At this time, for the fourth and subsequent rough cutting cycles, the cycle time is longer because memory is searched for P and Q blocks.

Example

```
G71 P100 Q200 ...;
N100 ...;
...;
...;
N200 ...;
G71 P300 Q400 ...;
N300 ...;
...;
...;
N400 ...;
...;
...;
G70 P100 Q200 ;  (Executed without a search for
                  the first to third cycles)
G70 P300 Q400 ;  (Executed after a search for the
                  fourth and subsequent cycles)
```

NOTE

The memory addresses of P and Q blocks stored during rough cutting cycles by G71, G72, and G73 are erased after execution of G70.
All stored memory addresses of P and Q blocks are also erased by a reset.

- Return to the cycle start point

In a finishing cycle, after the tool cuts the workpiece to the end point of the target figure, it returns to the cycle start point in rapid traverse.

NOTE

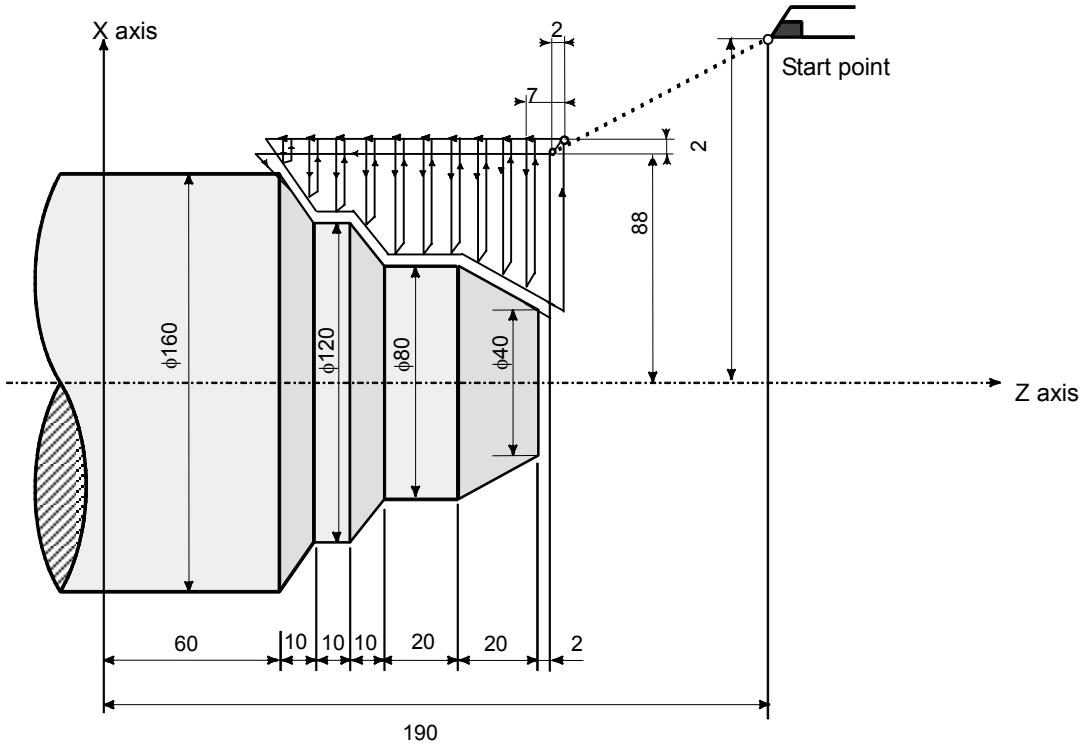
The tool returns to the cycle start point always in the nonlinear positioning mode regardless of the setting of bit 1 (LRP) of parameter No. 1401.
Before executing a finishing cycle for a target figure with a pocket cut by G71 or G72, check that the tool does not interfere with the workpiece when returning from the end point of the target figure to the cycle start point.

- Tool nose radius compensation

Like G71, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

Example

Stock removal in facing (G72)

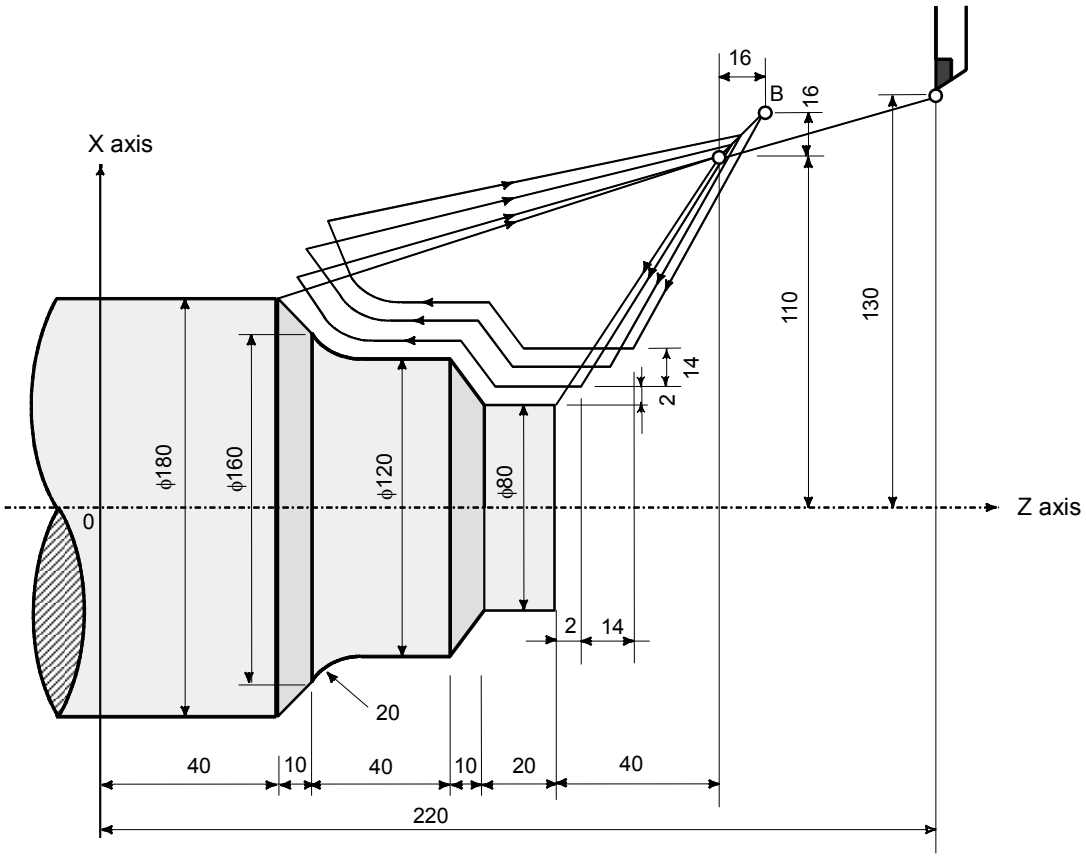


(Diameter designation for X axis, metric input)

```
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 Z56.0 S700 ;
N015  G01 X120.0 W14.0 F0.15 ;
N016  W10.0 ;
N017  X80.0 W10.0 ;
N018  W20.0 ;
N019  X36.0 W22.0 ;
N020  G70 P014 Q019 ;
```

Escaping amount: 1.0
Finishing allowance (4.0 in diameter in the X direction, 2.0 in the Z direction)

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 ;
```

4.2.5 End Face Peck Drilling Cycle (G74)

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

Format

G74R (e) ;
G74X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F (f) ;

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

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

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

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

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

Δd : Relief amount of the tool at the cutting bottom

f : Feedrate

	Unit	Diameter/radius programming	Sign	Decimal point input
e	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δi	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δd	Depends on the increment system for the reference axis.	Radius programming	NOTE	Allowed

NOTE
Normally, specify a positive value for Δd. When X (U) and Δi are omitted, specify a value with the sign indicating the direction in which the tool is to escape.

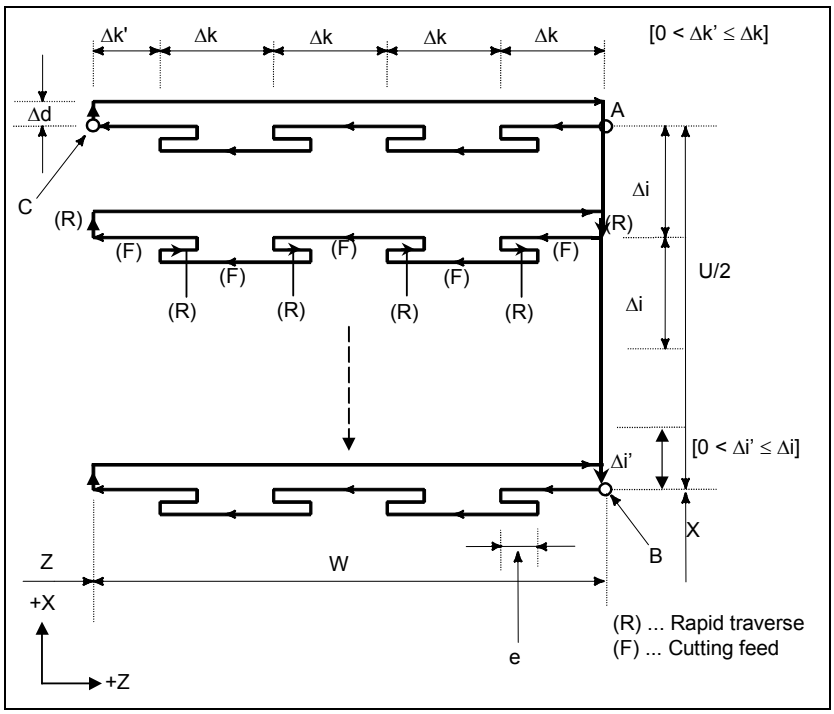


Fig. 4.2.5 (a) Cutting path in end face peek drilling cycle

Explanation

- Operations

A cycle operation of cutting by Δk and return by e is repeated.
When cutting reaches point C, the tool escapes by Δd. Then, the tool returns in rapid traverse, moves to the direction of point B by Δi, and performs cutting again.

NOTE
1 While both e and Δd are specified by the same address, the meanings of them are determined by specifying the X, Y, or Z axis. When the axis is specified, Δd is used.
2 The cycle machining is performed by G74 command with specifying the axis.

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

4.2.6 Outer Diameter / Internal Diameter Drilling Cycle (G75)

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

Format

G75R (e) ;
G75X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F (f) ;

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

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

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

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

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

Δd : Relief amount of the tool at the cutting bottom

f : Feedrate

	Unit	Diameter/radius programming	Sign	Decimal point input
e	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δi	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δd	Depends on the increment system for the reference axis.	Radius programming	NOTE	Allowed

NOTE
Normally, specify a positive value for Δd. When Z (W) and Δk are omitted, specify a value with the sign indicating the direction in which the tool is to escape.

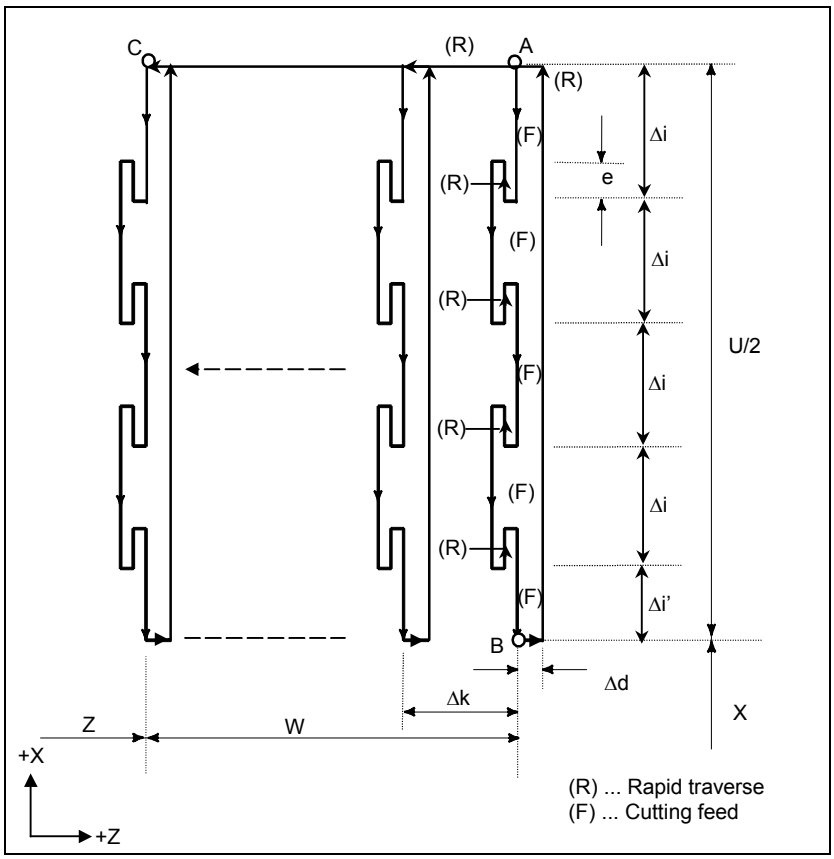


Fig. 4.2.6 (b) Outer diameter/internal diameter drilling cycle

Explanation

- Operations

A cycle operation of cutting by Δi and return by e is repeated.
When cutting reaches point B, the tool escapes by Δd. Then, the tool returns in rapid traverse, moves to the direction of point C by Δk, and performs cutting again.

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

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

4.2.7 Multiple Threading Cycle (G76)

This threading cycle performs one edge cutting by the constant amount of cut.

Format

G76 P(m) (r) (a) Q(Δdmin) R(d) ;
G76 X(U)_ Z(W)_ R(i) P(k) Q(Δd) F (L) ;

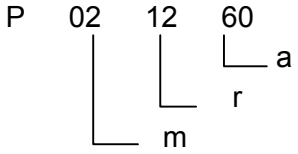
m : Repetitive count in finishing (1 to 99)
This value can be specified by the parameter No. 5142, and the parameter is changed by the program command.

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

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

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

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



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

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

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

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

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

k : Height of thread

Δd : Depth of cut in 1st cut

L : Lead of thread

	Unit	Diameter/radius programming	Sign	Decimal point input
Δdmin	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
d	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
i	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
k	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed

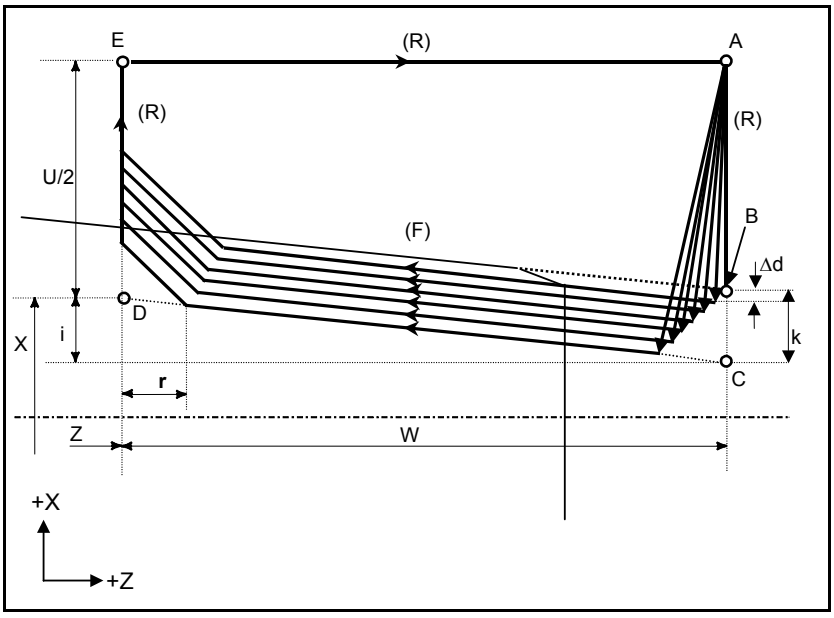


Fig. 4.2.7 (c) Cutting path in multiple threading cycle

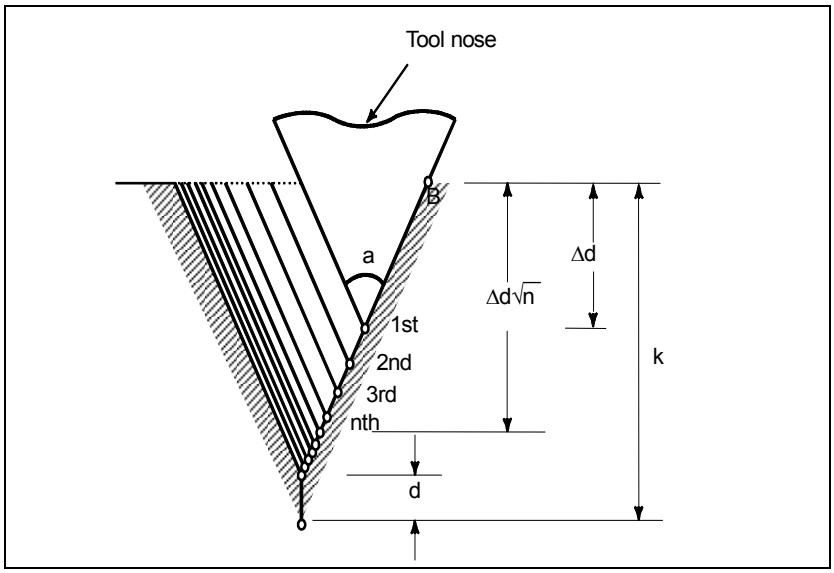
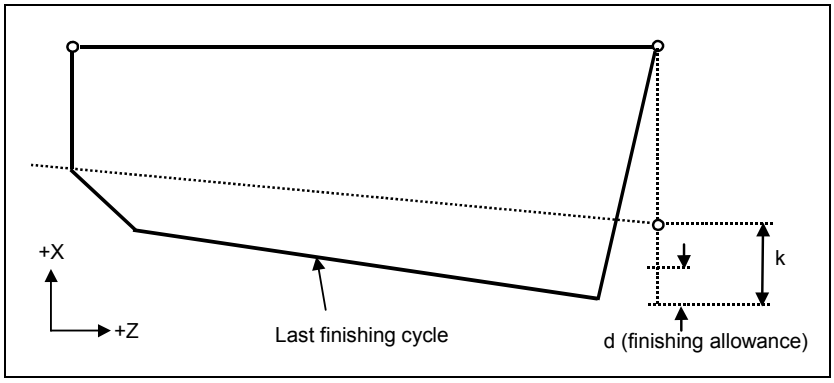


Fig. 4.2.7 (d) Detail of cutting

- Repetitive count in finishing

The last finishing cycle (cycle in which the finishing allowance is removed by cutting) is repeated.



Explanation

- Operations

This cycle performs threading so that the length of the lead only between C and D is made as specified in the F code. In other sections, the tool moves in rapid traverse.
The time constant for acceleration/deceleration after interpolation and FL feedrate for thread chamfering and the feedrate for retraction after chamfering are the same as for thread chamfering with G92 (canned cycle).

NOTE
1 The meanings of the data specified by address P, Q, and R determined by the presence of X (U) and Z (W).
2 The cycle machining is performed by G76 command with X (U) and Z (W) specification.
3 The values specified at addresses P, Q, and R are modal and are not changed until another value is specified.

⚠ **CAUTION**
Notes on threading are the same as those on G32 threading. For feed hold in a threading cycle, however, see "Feed hold in a threading cycle" described below.

- Relationship between the sign of the taper amount and tool path

The signs of incremental dimensions for the cycle shown in Fig. 4.2.7 (c) are as follows:

Cutting end point in the direction of the length for U and W:

Minus (determined according to the directions of paths A-C and C-D)

Taper amount (i):

Minus (determined according to the direction of path A-C)

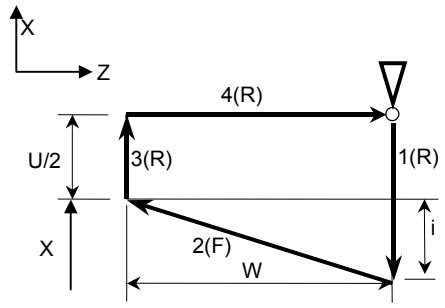
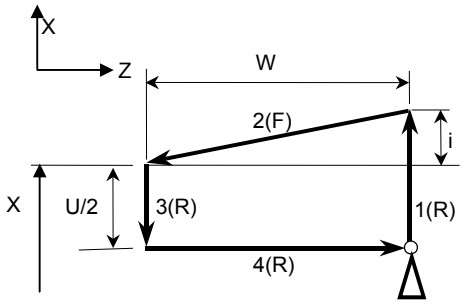
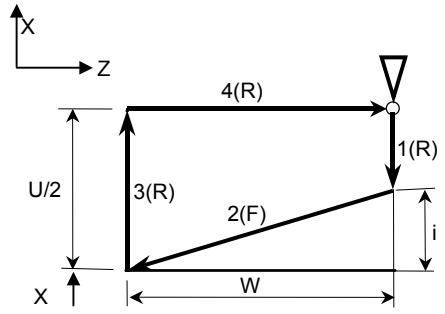
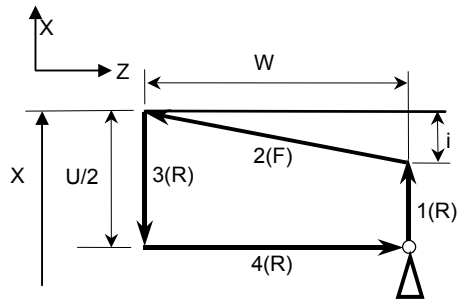
Height of thread (k):

Plus (always specified with a plus sign)

Depth of cut in the first cut (Δd):

Plus (always specified with a plus sign)

The four patterns shown in the table below are considered corresponding to the sign of each address. A female thread can also be machined.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, i < 0	**2. U > 0, W < 0, i > 0**
	
3. U < 0, W < 0, i > 0 at |i|≤|U/2|	**4. U > 0, W < 0, i < 0 at |i|≤|U/2|**
	

- Acceleration/deceleration after interpolation for threading

Acceleration/deceleration after interpolation for threading is acceleration/deceleration of exponential interpolation type. By setting bit 5 (THLx) of parameter No. 1610, the same acceleration/deceleration as for cutting feed can be selected. (The settings of bit 0 (CTLx) of parameter No. 1610 are followed.) However, as a time constant and FL feedrate, the settings of parameter No. 1626 and No. 1627 for the threading cycle are used.

- Time constant and FL feedrate for threading

The time constant for acceleration/deceleration after interpolation for threading specified in parameter No. 1626 and the FL feedrate specified in parameter No. 1627 are used.

- Thread chamfering

Thread chamfering can be performed in this threading cycle. A signal from the machine tool initiates thread chamfering.
The maximum amount of thread chamfering (r) that can be specified in the command is 99 (9.9L). The amount can be specified in a range from 0.1L to 12.7L in 0.1L increments in parameter No. 5130.
A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.
For thread chamfering, the same type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate as for threading are used.

NOTE
Common parameters for specifying the amount and angle of thread chamfering are used for this cycle and G92 threading cycle.

- Retraction after chamfering

The following table lists the feedrate, type of acceleration/deceleration after interpolation, and time constant of retraction after chamfering.

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

By setting bit 4 (ROC) of parameter No. 1403 to 1, rapid traverse override can be disabled for the feedrate of retraction after chamfering.

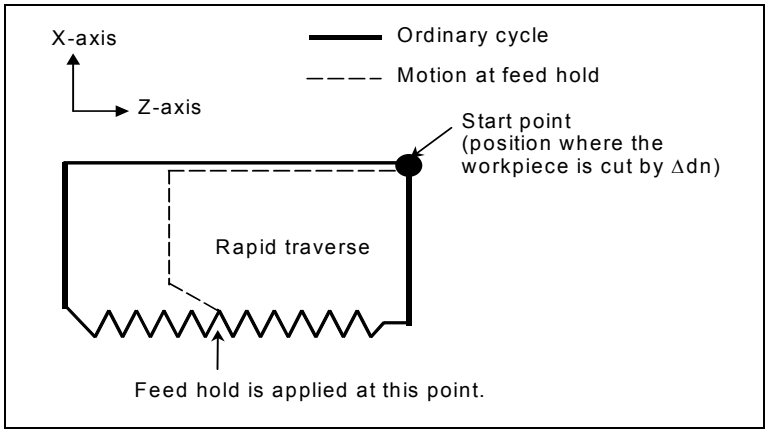
NOTE
During retraction, the machine does not stop with an override of 0% for the cutting feedrate regardless of the setting of bit 4 (RF0) of parameter No. 1401.

- Shifting the start angle

The threading start angle cannot be shifted.

- Feed hold in a threading cycle（threading cycle retract）

Feed hold may be applied during threading in a combined threading cycle (G76). In this case, the tool quickly retracts in the same way as for the last chamfering in a threading cycle and returns to the start point in the current cycle (position where the workpiece is cut by Δdn).
When cycle start is triggered, the multiple threading cycle resumes.



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

⚠ CAUTION
Another feed hold cannot be performed during retraction.

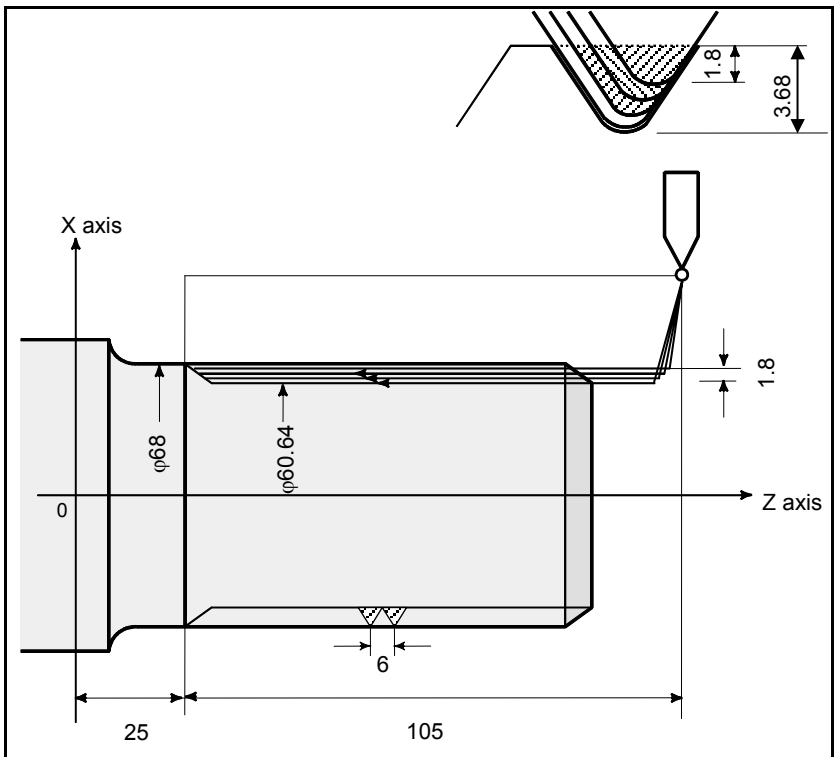
- Inch threading

Inch threading specified with address E is not allowed.

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

Example




```
G80 X80.0 Z130.0;
G76 P011060 Q100 R200 ;
G76 X60.64 Z25.0 P3680 Q1800 F6.0 ;
```

4.2.8 Restrictions on Multiple Repetitive Canned Cycle (G70-G76)

Programmed commands

- Program memory

Programs using G70, G71, G72, or G73 must be stored in the program memory. The use of the mode in which programs stored in the program memory are called for operation enables these programs to be executed in other than the MEM mode. Programs using G74, G75, or G76 need not be stored in the program memory.

- Blocks in which data related to a multiple repetitive canned cycle is specified

The addresses P, Q, X, Z, U, W, and R should be specified correctly for each block.

In a block in which G70, G71, G72, or G73 is specified, the following functions cannot be specified:

- Custom macro calls
  (simple call, modal call, and subprogram call)

- Blocks in which data related to a target figure is specified

In the block which is specified by address P of a G71, G72 or G73, G00 or G01 code in group 01 should be commanded. If it is not commanded, alarm PS0065 is generated.

In blocks with sequence numbers between those specified at P and Q in G70, G71, G72, and G73, the following commands can be specified:

- Dwell (G04)
- G00, G01, G02, and G03
  When a circular interpolation command (G02, G03) is used, there must be no radius difference between the start point and end point of the arc. If there is a radius difference, the target finishing figure may not be recognized correctly, resulting in a cutting error such as excessive cutting.
- Custom macro branch and repeat command
  The branch destination must be between the sequence numbers specified at P and Q, however. High-speed branch specified by bits 1 and 4 of parameter No. 6000 is invalid. No custom macro call (simple, modal, or subprogram call) cannot be specified.
- Direct drawing dimension programming command and chamfering and corner R command
  Direct drawing dimension programming and chamfering and corner R require multiple blocks to be specified. The block with the last sequence number specified at Q must not be an intermediate block of these specified blocks.

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.

When #1 = 2500 is executed using a custom macro, 2500.000 is assigned to #1. In such a case, P#1 is equivalent to P2500.

Relation with other functions

- Manual intervention

While a multiple repetitive canned cycle (G70 to G76) is being executed, it is possible to stop the cycle and to perform manual intervention.
The setting of manual absolute on or off is effective for manual operation.

- Interruption type macro

Any interruption type macro program cannot be executed during execution of a multiple repetitive canned cycle.

- Program restart and tool retract and recover

These functions cannot be executed in a block in a multiple repetitive canned cycle.

- Axis name and second auxiliary functions

Even if address U, V, or W is used as an axis name or second auxiliary function, data specified at address U, V, or W in a G71 to G73 block is assumed to be that for the multiple repetitive canned cycle.

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70, G71, G72, G73) and specify the cancel command (G40) outside the blocks (from the block specified with P to the block specified with Q) specifying a target finishing figure.

4.3 CANNED CYCLE FOR DRILLING

Canned cycles for drilling make it easier for the programmer to create programs. With a canned cycle, a frequently-used machining operation can be specified in a single block with a G function; without canned cycles, more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.
Table 4.3 (a) lists canned cycles for drilling.

Table 4.3 (a) Canned cycles for drilling

G code	Drilling axis	Hole machining operation	Operation in the bottom hole position	Retraction operation	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	Dwell	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

Explanation

The canned cycle for drilling consists of the following six operation sequences.

Operation 1 ..... Positioning of X (Z) and C axis
Operation 2 ..... Rapid traverse up to point R level
Operation 3 ..... Hole machining
Operation 4 ..... Operation at the bottom of a hole
Operation 5 ..... Retraction to point R level
Operation 6 ..... Rapid traverse up to the initial level

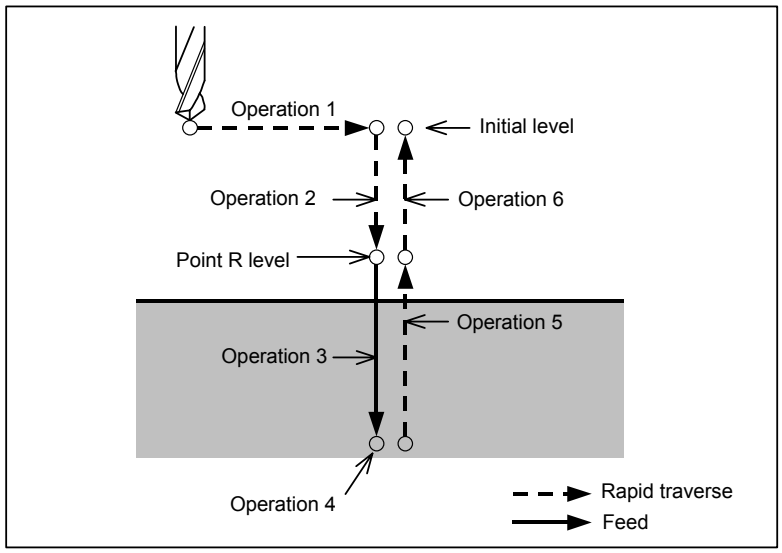


Fig. 4.3 (a) Operation sequence of canned cycle for drilling

- Positioning axis and drilling axis

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. A drilling G code specifies positioning axes and a drilling axis as shown below.
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 4.3 (b) Positioning axis and drilling axis

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

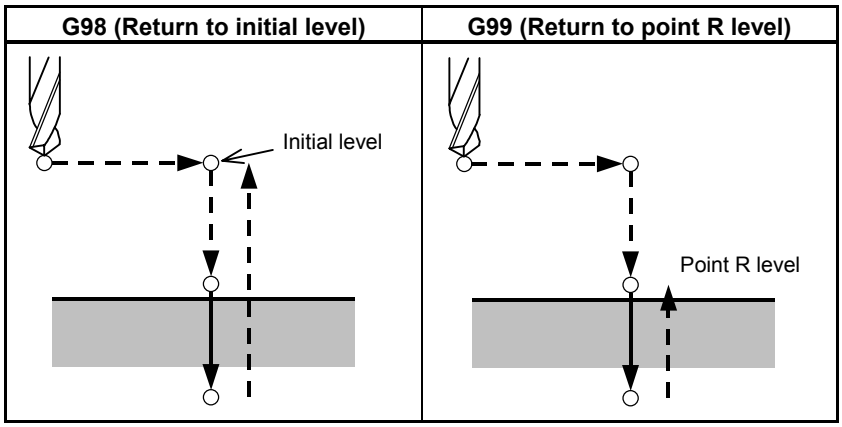
G83 and G87, G84 and G88, and G85 and G89 have the same function respectively except for axes specified as positioning axes and a drilling axis.

- Drilling mode

G83 to G85/G87 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.
Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.
Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.
The feedrate specified at F is retained also after the drilling cycle is canceled. When Q data is required, it must be specified in each block.
Once specified, the M code used for C-axis clamp/unclamp functions as a modal code. It is canceled by specifying G80.

- 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.



- Number of repeats

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

Number of repeats K	The maximum command value = 9999

When K0 is specified, drilling data is just stored without drilling being performed.

NOTE
For K, specify an integer of 0 or 1 to 9999.

- M code used for C-axis clamp/unclamp

When an M code specified in parameter No. 5110 for C-axis clamp/unclamp is coded in a program, the following operations occur.

- The CNC issues the M code for C-axis clamp after the tool is positioned and while the tool is being fed in rapid traverse to the point-R level.
- The CNC issues the M code for C-axis unclamp (the M code for C-axis clamp +1) after the tool retracts to the point-R level.
- After the CNC issues the M code for C-axis unclamp, the tool dwells for the time specified in parameter No. 5111.

- Cancel

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

Group 01 G codes (Example)

G00 : Positioning (rapid traverse)
G01 : Linear interpolation
G02 : Circular interpolation (CW)
G03 : Circular interpolation (CCW)

- Symbols in figures

Subsequent subsections 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.5111
Mα	Issuing the M code for C-axis clamp (The value of α is specified with parameter No. 5110.)
M (α + 1)	Issuing the M code for C-axis unclamp

⚠ CAUTION

1 In each canned cycle, addresses R, Z, and X are handled as follows:
R_ : Always handled as a radius.
Z_ or X_: Depends on diameter/radius programming.

2 For the B or C G-code system, G90 or G91 can be used to select an incremental or absolute programming for hole position data (X, C or Z, C), the distance from point R to the bottom of the hole (Z or X), and the distance from the initial level to the point R level (R).

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

The peck drilling cycle or high-speed peck drilling cycle is used depending on the setting in RTR, bit 2 of parameter No. 5101. If depth of cut for each drilling is not specified, the normal drilling cycle is used.

- High-speed peck drilling cycle (G83, G87) (parameter RTR (No. 5101#2) =0)

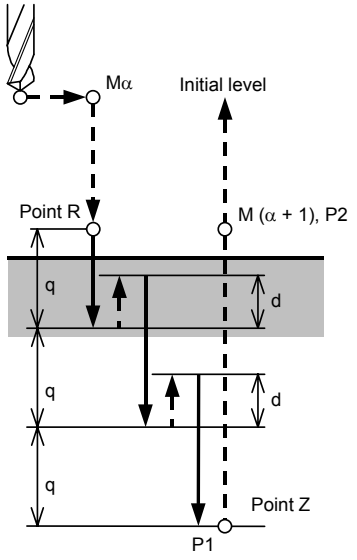
This cycle performs high-speed peck drilling. The drill repeats the cycle of drilling at the cutting feedrate and retracting the specified retraction distance intermittently to the bottom of a hole. The drill draws cutting chips out of the hole when it retracts.

Format

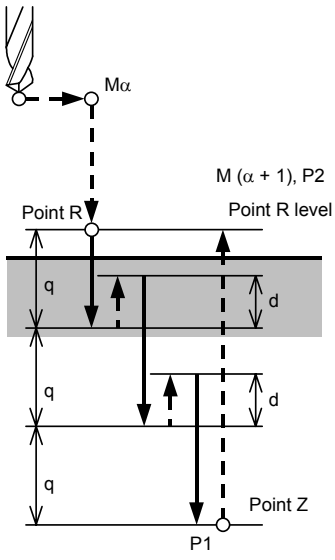
G83 X(U)_ C(H)_ Z(W)_ R_ P_ Q_ F_ K_ M_ ;
or
G87 Z(W)_ C(H)_ X(U)_ R_ P_ Q_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ 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
Q_ : Depth of cut for each cutting feed
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed)
M_ : M code for C-axis clamp (When it is needed.)

G83 or G87 (G98 mode)



G83 or G87 (G99 mode)



Mα : M code for C-axis clamp
M (α + 1) : M code for C-axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5111
d : Retraction distance specified in parameter No. 5114

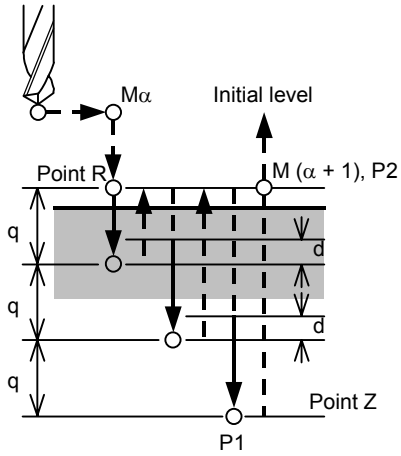
- Peck drilling cycle (G83, G87) (parameter No. 5101#2 =1)

Format

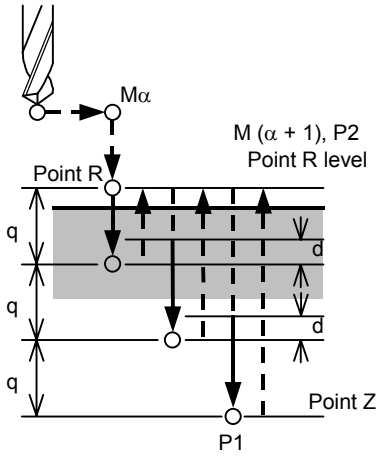
G83 X(U)_ C(H)_ Z(W)_ R_ P_ Q_ F_ K_ M_ ;
or
G87 Z(W)_ C(H)_ X(U)_ R_ P_ Q_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ 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
Q_ : Depth of cut for each cutting feed
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C-axis clamp (When it is needed.)

G83 or G87 (G98 mode)



G83 or G87 (G99 mode)



Mα : M code for C-axis clamp
M (α + 1) : M code for C-axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5111
d : Retraction distance specified in parameter No. 5115

Example

```
M51 ;                                Setting C-axis index mode ON
M3 S2000 ;                           Rotating the drill
G00 X50.0 C0.0 ;                     Positioning the drill along the X- and C-axes
G83 Z-40.0 R-5.0 Q5000 F5.0 M31 ;    Drilling hole 1
C90.0 Q5000 M31 ;                    Drilling hole 2
C180.0 Q5000 M31 ;                   Drilling hole 3
C270.0 Q5000 M31 ;                   Drilling hole 4
G80 M05 ;                            Canceling the drilling cycle and stopping
                                     drill rotation
M50 ;                                Setting C-axis index mode off
```

NOTE
If the depth of cut for each cutting feed (Q) is not commanded, normal drilling is performed. (See the description of the drilling cycle.)

- Drilling cycle (G83 or G87)

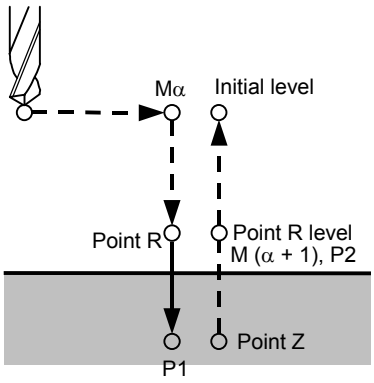
If depth of cut (Q) is not specified for each drilling, the normal drilling cycle is used. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

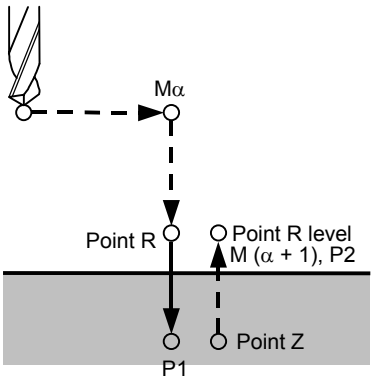
G83 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_ ;
or
G87 Z(W)_ C(H)_ X(U)_ R_ P_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ 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.)

G83 or G87 (G98 mode)



G83 or G87 (G99 mode)



Mα : M code for C-axis clamp
M (α + 1) : M code for C-axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5111

Example

```
M51 ;                          Setting C-axis index mode ON
M3 S2000 ;                     Rotating the drill
G00 X50.0 C0.0 ;               Positioning the drill along the X- and
                               C-axes
G83 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
```

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

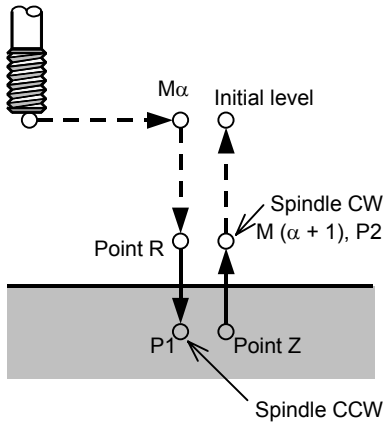
This cycle performs tapping.
In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format

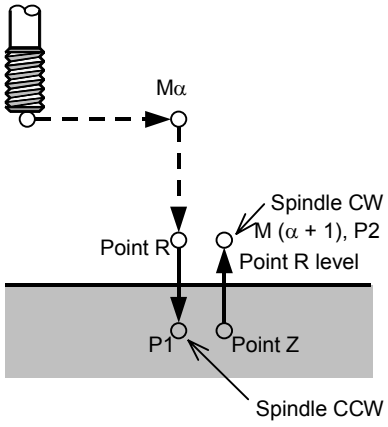
G84 X(U)_ C(H)_ Z(W)_ R_ P_ Q_ F_ K_ M_ ;
or
G88 Z(W)_ C(H)_ X(U)_ R_ P_ Q_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ 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
Q_ : Depth of cut for each cutting feed (Bit 6 (PCT) of parameter No. 5104 = "1")
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C-axis clamp (when it is needed.)

G84 or G88 (G98 mode)



G84 or G88 (G99 mode)



Mα : M code for C-axis clamp
M (α + 1) : M code for C-axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5111

Explanation

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 3 (M5T) of parameter No. 5105 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.

- Q command

After setting bit 6 (PCT) of parameter No. 5104 to 1, add address Q to the ordinary tapping cycle command format and specify the depth of cut for each tapping.
In the peck tapping cycle, the tool is retracted to point R for each tapping. In the high-speed peck tapping cycle, the tool is retracted by the retraction distance specified for parameter No. 5213 in advance. Which operation is to be performed can be selected by setting bit 5 (PCP) of parameter No. 5200.

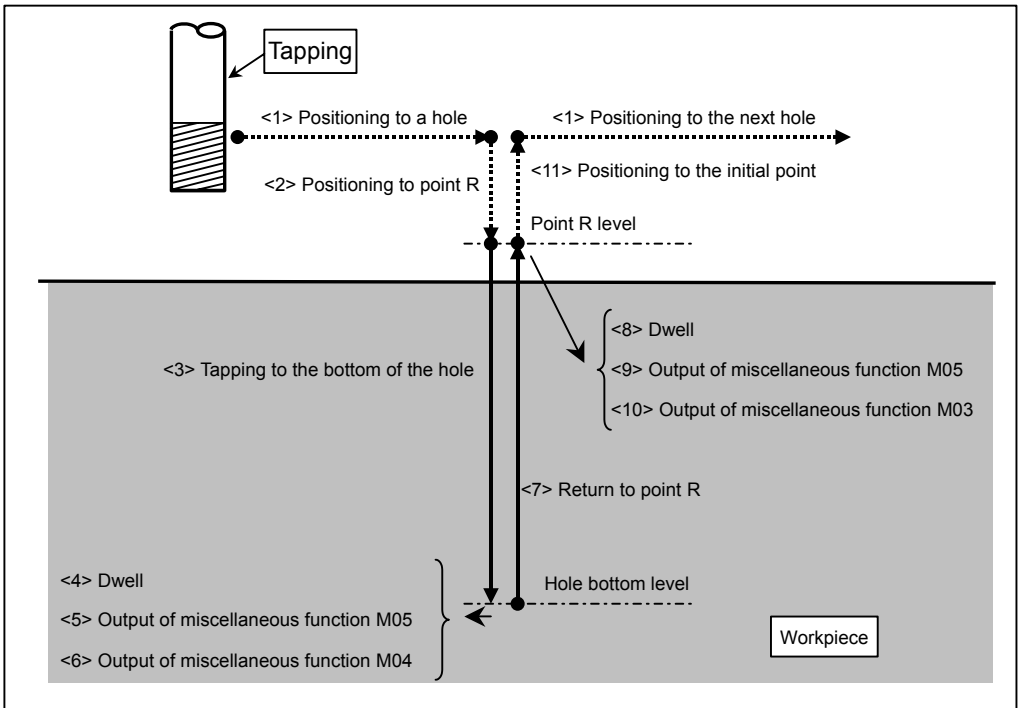
Operation

First, ordinary tapping cycle operation is explained as basic operation.

Before specifying a tapping cycle, rotate the spindle using a miscellaneous function.

1. When a command to position the tool to a hole position, positioning is performed.
2. When point R is specified, positioning to point R is performed.
3. Tapping is performed to the bottom of the hole in cutting feed.
4. When a dwell time (P) is specified, the tool dwells.
5. Miscellaneous function M05 (spindle stop) is output and the machine enters the FIN wait state.
6. When FIN is returned, miscellaneous function M04 (reverse spindle rotation) is output and the machine enters the FIN wait state.
7. When FIN is returned, the tap is removed until point R is reached in cutting feed.
8. When a dwell time (P) is specified, the tool dwells.
9. Miscellaneous function M05 (spindle stop) is output and the machine enters the FIN wait state.
10. When FIN is returned, miscellaneous function M03 (forward spindle rotation) is output, and the machine enters the FIN wait state.
11. When FIN is returned, the tool returns to the initial point in rapid traverse when return to the initial level is specified.

When the repetitive count is specified, operation is repeated from step 1.



Peck tapping cycle

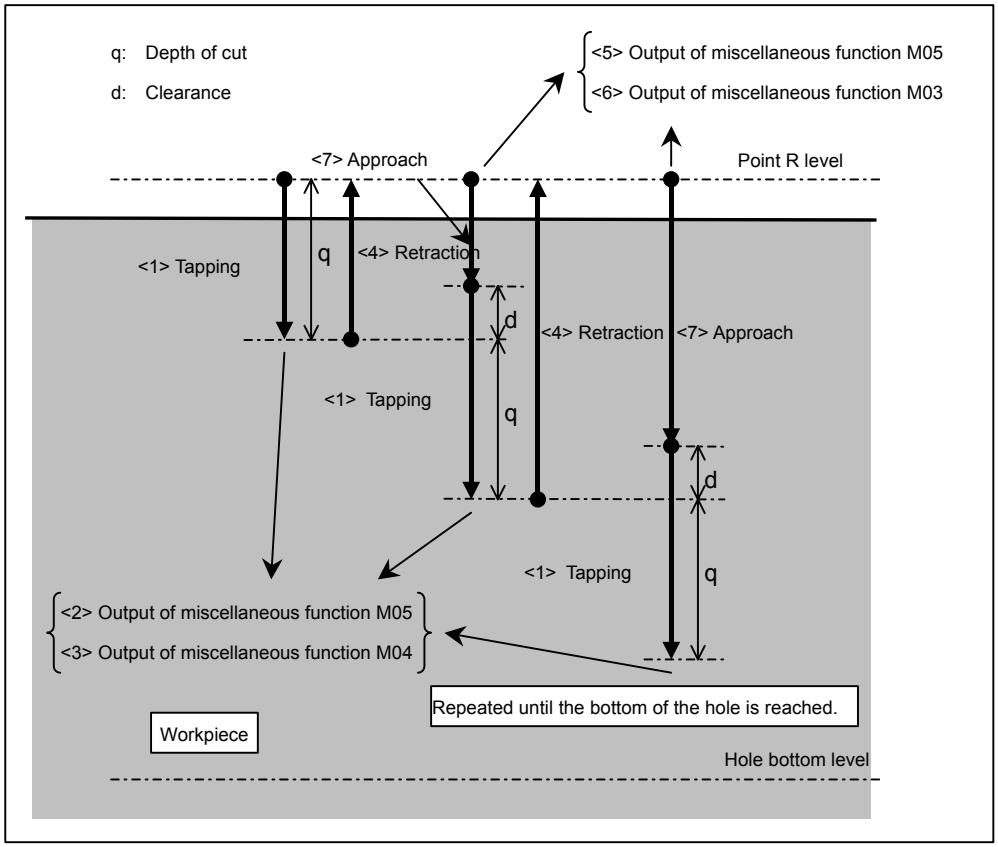
When bit 6 (PCT) of parameter No. 5104 is set 1 and bit 5 (PCP) of parameter No. 5200 is set to 1, the peck tapping cycle is used.
Step 3 of the tapping cycle operation described above changes as follows:

3-1. The tool cuts the workpiece by the depth of cut q specified by address Q.
3-2. Miscellaneous function M05 (spindle stop) is output, and the machine enters the FIN wait state.
3-3. When FIN is returned, miscellaneous function M04 (reverse spindle rotation) is output, and the machine enters the FIN wait state.
3-4. When FIN is returned, the tool is retracted to point R in cutting feed.
3-5. Miscellaneous function M05 (spindle stop) is output, and the machine enters the FIN wait state.
3-6. When FIN is returned, miscellaneous function M03 (forward spindle rotation) is output, and the machine enters the FIN wait state.
3-7. When FIN is returned, the tool moves to the position the clearance d (parameter No. 5213) apart from the previous cutting point in cutting feed (approach).

3-1. The tool cuts the workpiece by the clearance d (parameter No. 5213) + depth of cut q (specified by address Q).

Tapping is performed to the bottom of the hole by repeating the above steps.

When a dwell time (P) is specified, the tool dwells only when it reaches at the bottom of the hole and reaches point R last.



High-speed peck tapping cycle

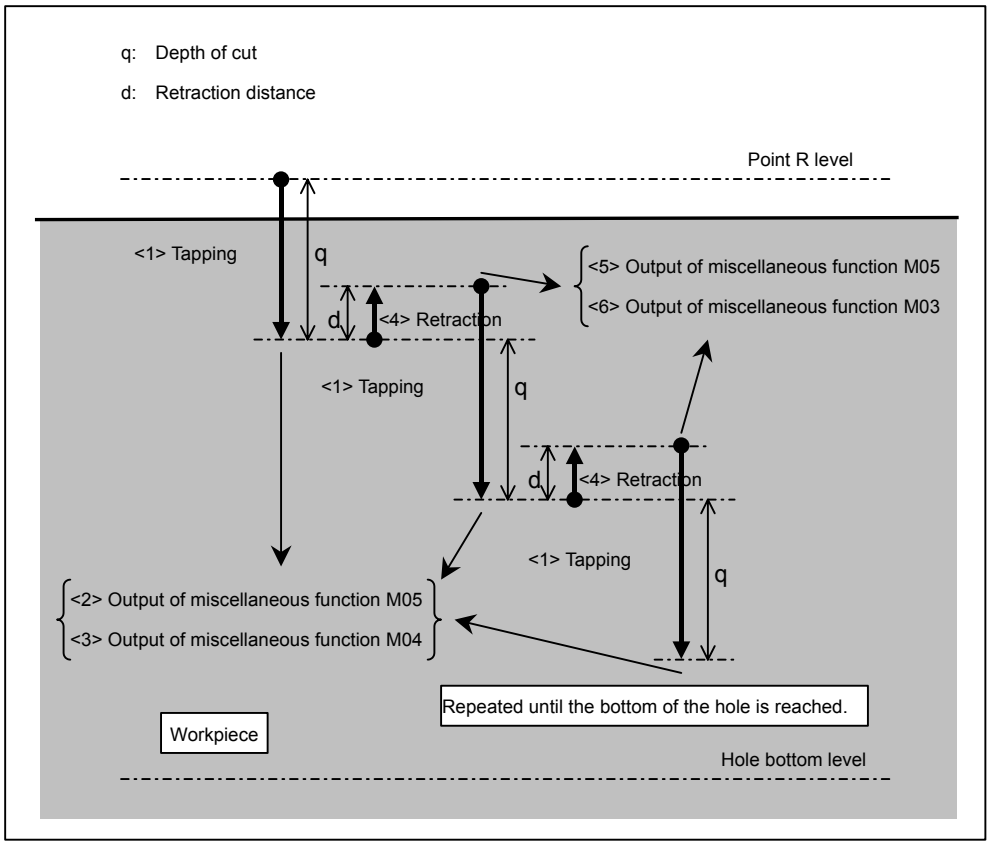
When bit 6 (PCT) of parameter No. 5104 is set 1 and bit 5 (PCP) of parameter No. 5200 is set to 0, the high-speed peck tapping cycle is used.

Step 3 of the tapping cycle operation described above changes as follows:

3-1. The tool cuts the workpiece by the depth of cut q specified by address Q.
3-2. Miscellaneous function M05 (spindle stop) is output, and the machine enters the FIN wait state.
3-3. When FIN is returned, miscellaneous function M04 (reverse spindle rotation) is output, and the machine enters the FIN wait state.
3-4. When FIN is returned, the tool is retracted by the retraction distance d specified by parameter No. 5213 in cutting feed.
3-5. Miscellaneous function M05 (spindle stop) is output, and the machine enters the FIN wait state.
3-6. When FIN is returned, miscellaneous function M03 (forward spindle rotation) is output, and the machine enters the FIN wait state.

3-1. When FIN is returned, the tool cuts the workpiece by the retraction distance d (parameter No. 5213) + depth of cut q (specified by address Q).

Tapping is performed to the bottom of the hole by repeating the above steps.
When a dwell time (P) is specified, the tool dwells only when it reaches at the bottom of the hole and reaches point R.



Notes

1. The depth of cut specified by address Q is stored as a modal value until the canned cycle mode is canceled.
In both examples 1 and 2 below, address Q is not specified in the N20 block, but the peck tapping cycle is performed because the value specified by address Q is valid as a modal value. If this operation is not suitable, specify G80 to cancel the canned cycle mode as shown in N15 in example 3 or specify Q0 in the tapping block as shown in N20 in example 4.

Example 1
N10 G84 X100. Y150. Z-100. Q20. ;
N20 X150. Y200 ; ← The peck tapping cycle is also performed in this block.
N30 G80 ;

Example 2
N10 G83 X100. Y150. Z-100. Q20. ;
N20 G84 Z-100. ; ← The peck tapping cycle is also performed in this block.
N30 G80 ;

Example 3
N10 G83 X100. Y150. Z-100. Q20. ;
N15 G80 ; ← The canned cycle mode is canceled.
N20 G84 Z-100. ;
N30 G80 ;

Example 4
N10 G83 X100. Y150. Z-100. Q20. ;
N20 G84 Z-100. Q0 ; ←Q0 is added.
N30 G80 ;

2. The unit for the reference axis that is set by parameter No. 1031, not the unit for the drilling axis is used as the unit of Q. Any sign is ignored.

3. Specify a radius value at address Q even when a diameter axis is used.

4. Perform operation in the peck tapping cycle within point R. That is, set a value which does not exceed point R for d (parameter No. 5213).

Example

M51 ;	Setting C-axis index mode ON
M3 S2000 ;	Rotating the drill
G00 X50.0 C0.0 ;	Positioning the drill along the X- and C-axes
G84 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

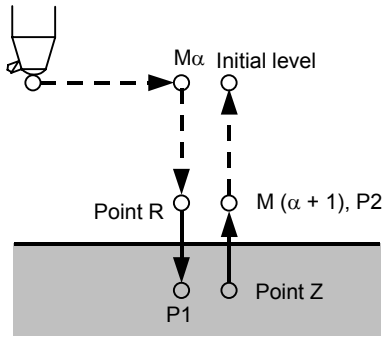
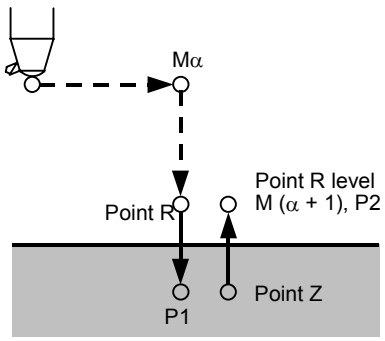
4.3.3 Front Boring Cycle (G85) / Side Boring Cycle (G89)

This cycle is used to bore a hole.

Format

G85 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_ ;
or
G89 Z(W)_ C(H)_ X(U)_ R_ P_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ 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.)

G85 or G89 (G98 mode)	G85 or G89 (G99 mode)
	

Mα : M code for C-axis clamp
M (α + 1) : M code for C-axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5111

Explanation

After positioning, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
After the tool reaches point Z, it returns to point R at a feedrate twice the cutting feedrate.

Example

M51 ;	Setting C-axis index mode ON
M3 S2000 ;	Rotating the drill
G00 X50.0 C0.0 ;	Positioning the drill along the X- and C-axes
G85 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

4.3.4 Canned Cycle for Drilling Cancel (G80)

G80 cancels canned cycle for drilling.

Format

G80 ;

Explanation

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

Example

M51 ;	Setting C-axis index mode ON
M3 S2000 ;	Rotating the drill
G00 X50.0 C0.0 ;	Positioning the drill along the X- and C-axes.
G83 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

4.3.5 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, 6 in Fig. 4.3 (a).
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. If there is a remaining repetitive count at the end of operation 6, the operation is stopped by feed hold. If there is no remaining repetitive count, the operation is stopped in the single block stop state.

- 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 6.

- Override

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

4.4 RIGID TAPPING

Front face tapping cycles (G84) and side face tapping cycles (G88) can be performed either in conventional mode or rigid mode.

In conventional mode, the spindle is rotated or stopped, in synchronization with the motion along the tapping axis according to miscellaneous functions M03 (spindle CW rotation), M04 (spindle CCW rotation), and M05 (spindle stop).

In rigid mode, the spindle motor is controlled in the same way as a control motor, by the application of compensation to both motion along the tapping axis and that of the spindle.

For rigid tapping, each turn of the spindle corresponds to a certain amount of feed (screw lead) along the spindle axis. This also applies to acceleration/deceleration. This means that rigid tapping does not demand the use of float tappers as in the case of conventional tapping, thus enabling high-speed, high-precision tapping.

When multispindle control is enabled (bit 3 (MSP) of parameter No. 8133 is set to 1), the second spindle can be used for rigid tapping.

4.4.1 FRONT FACE RIGID TAPPING CYCLE (G84) / SIDE FACE RIGID TAPPING CYCLE (G88)

Controlling the spindle motor in the same way as a servo motor in rigid mode enables high-speed tapping.

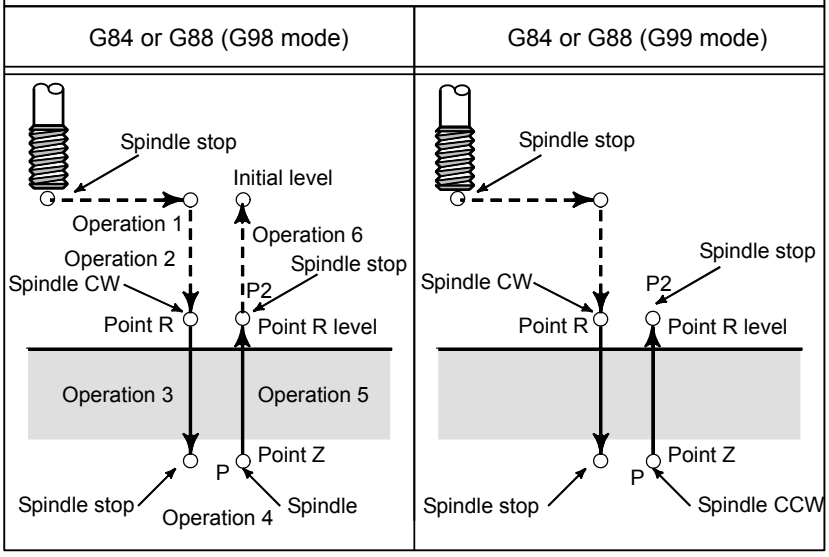
Format

G84 X (U)_ C (H)_ Z (W)_ R_ P_ F_ K_ M_ ;

or

G88 Z (W)_ C (H)_ X (U)_ R_ P_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ 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.)



P2 performs dwelling of C-axis unclamp. (The dwell time is set in parameter No. 5111.)

In front face rigid tapping (G84), the plane first axis is used as the drilling axis and the other axes are used as positioning axes.

Parameter RTX(No.5209#0)	Plane selection	Drilling axis
0	G17 Xp-Yp plane	Xp
	G18 Zp-Xp plane	Zp
	G19 Yp-Zp plane	Yp
1 (Note)		Zp

Xp: X axis or its parallel axis
Yp: Y axis or its parallel axis
Zp: Z axis or its parallel axis

NOTE
Invalid with the Series 10/11 format.

In side face rigid tapping (G88), the plane first axis is used as the drilling axis and the other axes are used as positioning axes.

Parameter RTX(No.5209#0)	Plane selection	Drilling axis
0	G17 Xp-Yp plane	Yp
	G18 Zp-Xp plane	Xp
	G19 Yp-Zp plane	Zp
1 (Note)		Xp

Xp: X axis or its parallel axis
Yp: Y axis or its parallel axis
Zp: Z axis or its parallel axis

NOTE
Invalid with the Series 10/11 format.

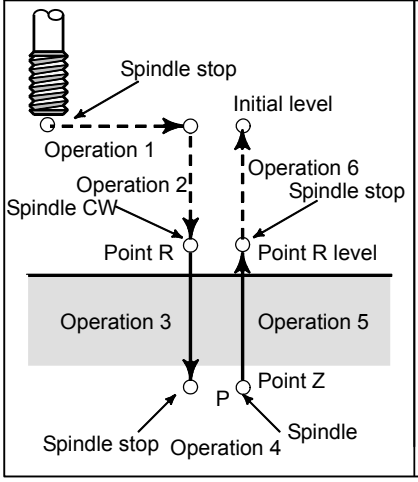
(Series 10/11 format)

G84.2 X (U)_ C (H)_ Z (W)_ R_ P_ F_ L_ S_ ;

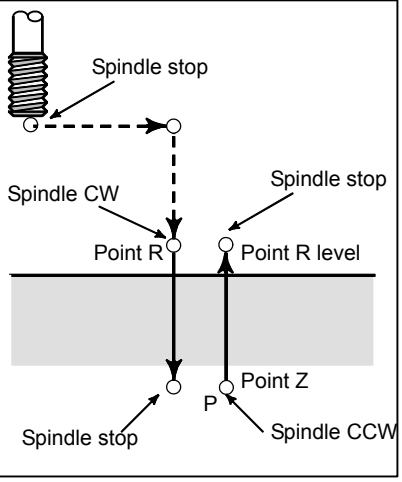
X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)
S_ : Spindle speed

C-axis clamp cannot be performed during specification of the Series 15 format.

G84.2 (G98 mode)



G84.2 (G99 mode)



A G code cannot discriminate between front face tapping cycle and side face tapping cycle using Series 10/11 format commands. The drilling axis is determined by plane selection (G17/G18/G19). Specify the plane selection that becomes equivalent to front face tapping cycle or side face tapping cycle as appropriate. (When FXY (bit 0 of parameter No. 5101) is 0, the Z-axis is used as the drilling axis. When FXY is 1, plane selection is as follows.)

Plane selection	Drilling axis
G17 Xp-Yp plane	Zp
G18 Zp-Xp plane	Yp
G19 Yp-Zp plane	Xp

Xp: X axis or its parallel axis
Yp: Y axis or its parallel axis
Zp: Z axis or its parallel axis

Explanation

Once positioning for the X-axis (G84) or Z-axis (G88) has been completed, the spindle is moved, by rapid traverse, to point R. Tapping is performed from point R to point Z, after which the spindle stops and observes a dwell time. Then, the spindle starts reverse rotation, retracts to point R, stops rotating, then moves to the initial level by rapid traverse.
During tapping, the feedrate override and spindle override are assumed to be 100%. For retraction (operation 5), however, a fixed override of up to 2000% can be applied by setting parameters DOV (No. 5200#4), OVU (No.5201#3), and No. 5211.

- Rigid mode

Rigid mode can be specified by applying any of the following methods:

- Specifying M29S***** before a tapping block
- Specifying M29S***** within a tapping block
- Handling G84 or G88 as a G code for rigid tapping (Set parameter G84 (No. 5200#0) to 1.)

- Thread lead

In feed per minute mode, the feedrate divided by the spindle speed is equal to the thread lead. In feed per rotation mode, the feedrate is equal to the thread lead.

- Series 10/11 format command

When bit 1 (FCV) of parameter No. 0001 is set to 1, rigid tapping can be executed with G84.2. The same operation as with G84 is performed. The command format for the repetitive count is L, however.

- Acceleration/deceleration after interpolation

Linear or bell-shaped acceleration/deceleration can be applied.
Details are given later.

- Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

- Override

Various types of override functions are invalid. The following override functions can be enabled by setting corresponding parameters:

- Extraction override
- Override signal

- Dry run

Dry run can be executed also in G84 (G88). When dry run is executed at the feedrate for the drilling axis in G84 (G88), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock

Machine lock can be executed also in G84 (G88).
When G84 (G88) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

- Reset

When a reset is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G88) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G88).

- Feed hold and single block

When parameter FHD (No. 5200#6) is set to 0, feed hold and single block are invalid in the G84 (G88) mode. When this bit is set to 1, they are valid.

- Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle."
With other manual operations, rigid tapping cannot be performed.

- Backlash compensation

In the rigid tapping mode, backlash compensation is applied to compensate the lost motion when the spindle rotates clockwise or counterclockwise. Set the amount of backlash in parameters Nos. 5321 to 5324.
Along the drilling axis, backlash compensation has been applied.

- C-axis clamp, C-axis unclamp

It is possible to specify an M code for mechanically fixing or releasing the C-axis during rigid tapping. Adding an M code for clamp to the G84 (G88) block outputs both M codes. Descriptions of timing are provided later.
An M code for clamp is set in parameter No. 5110. An M code for unclamp is assumed as follows depending on the setting of parameter No. 5110.

Parameter No.5110	
0	**Non-0**
No M codes are output.	The setting of parameter No.5110 + 1 is assumed.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm PS0206 is issued.

- S commands

When a value exceeding the maximum rotation speed for the gear being used is specified, alarm PS0200 is issued. If such a command that the number of pulses of 8 msec is 32768 or more on the detection unit level or the number of pulses of 8 msec is 32768 or more for a serial spindle is specified, alarm PS0202 is issued.

<Example>

For a built-in motor equipped with a detector having a resolution of 4095 pulses per rotation, the maximum spindle speed during rigid tapping is as follows (in the case of 8 msec):
$(4095\times1000\div8\times60)\div4095=7500\ (\text{min}^{-1})$
For a serial spindle
$(32767\times1000\div8\times60)\div4095=60012(\text{min}^{-1})$ [Note:Ideal value]

When rigid tapping is canceled, the S value used for rigid tapping is cleared (as if S0 is specified).

- F commands

Specifying a value larger than the upper limit for cutting feed will cause alarm PS0201 to be issued.

- Unit of F command

	Metric input	Inch input	Remarks
G98	1mm/min	0.01inch/min	Decimal point programming allowed
G99	0.01mm/rev	0.0001inch/rev	Decimal point programming allowed

- M29

If an S command and axis movement are specified between M29 and G84, alarm PS0203 is issued. If M29 is specified in a tapping cycle, alarm PS0204 is issued.

- Rigid tapping command M code

The M code used to specify the rigid tapping mode is set in parameter No. 5210.

- P

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G84 in a single block. Otherwise, G84 will be canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Program restart

A program cannot be restarted during rigid tapping.

- R

The value of R must be specified in a block which performs drilling. If the value is specified in a block which does not perform drilling, it is not stored as modal data.

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P_ in an independent block.

Example

Tapping axis feedrate: 1000 mm/min
Spindle speed: 1000 min^{-1}
Screw lead: 1.0 mm
<Programming for feed per minute>

```
G98 ; .....Command for feed per minute
G00 X100.0 ; ..... Positioning
M29 S1000; ..... Command for specifying rigid mode
G84 Z-100.0 R-20.0 F1000 ; .....Rigid tapping
```

<Programming for feed per rotation>

```
G99 ; ..... Command for feed per rotation
G00 X100.0 ; ..... Positioning
M29 S1000 ; ..... Command for specifying rigid mode
G84 Z-100.0 R-20.0 F1.0 ; .....Rigid tapping
```

4.4.2 Peck Rigid Tapping Cycle (G84 or G88)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.
In this cycle, cutting is performed several times until the bottom of the hole is reached. Two peck tapping cycles are available: High-speed peck tapping cycle and standard peck tapping cycle. These cycles are selected using the bit 5 (PCP) of parameter No. 5200.

Format

When rigid tapping is specified with G84 (G88) if bit 5 (PCP) of parameter No. 5200 = 0, high-speed peck rigid tapping is assumed.

G84 X(U)_ C(H)_Z(W)_ R_ P_ Q_ F_ K_ M_ ;
or
G88 Z(W)_ C(H)_X(U)_ R_ P_ Q_ F_ K_ M_ ;

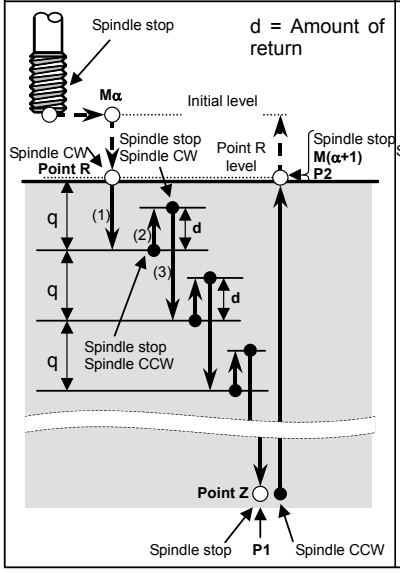
X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of the hole
Q_ : Depth of cut for each cutting feed
F_ : The cutting feedrate
K_ : Number of repeats (when it is needed.)
M_ : M code for C-axis clamp (when it is needed.)

- High-speed peck rigid tapping cycle
  In the first cutting from point R, perform cutting by depth "q" specified by address Q while rotating the spindle in the forward direction (operation <1>).
  Then, perform returning by the amount specified by parameter No. 5213 while rotating the spindle in the reverse direction (operation <2>).
  Then, perform tapping by (d+q) while rotating the spindle in the forward direction (operation <3>).

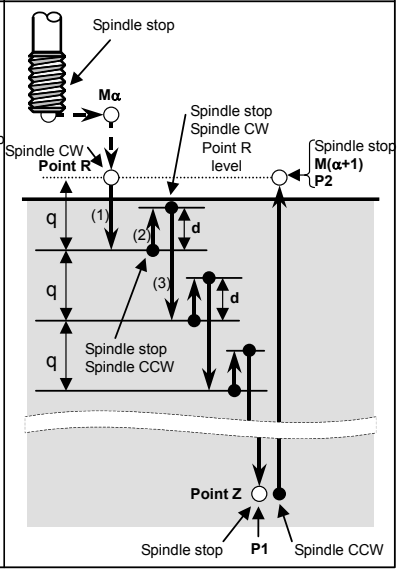
  Repeat operations <2> and <3> until the bottom of the hole is reached.

  The cutting speed and rigid tapping time constant are used for operations <1> and <3>.
  For operation <2> and travel from the bottom of the hole (point Z) to point R, rigid tapping extract override is enabled and the rigid tapping extract time constant is used.

G84 or G88(G98 mode)



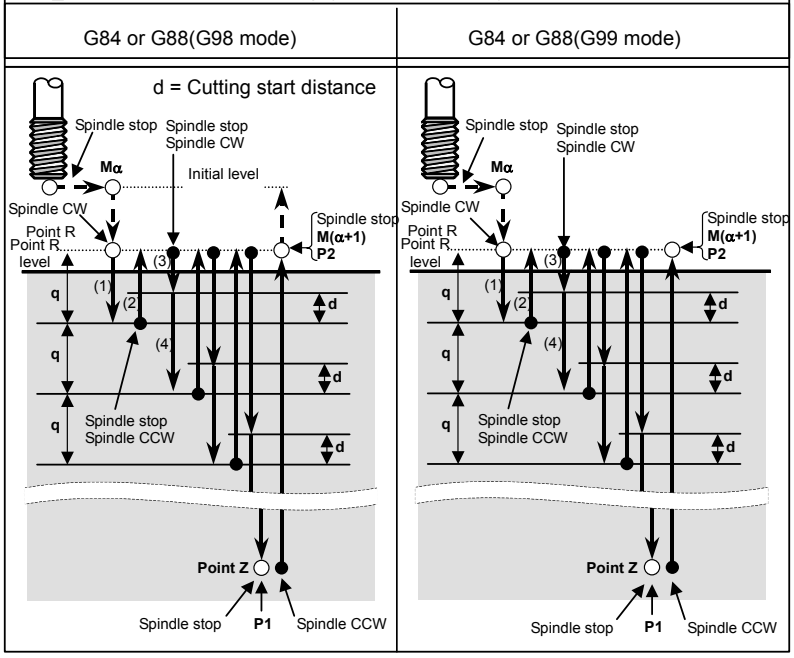
G84 or G88(G99 mode)



When rigid tapping is specified with G84 (G88) if bit 5 (PCP) of parameter No. 5200 = 1, peck rigid tapping is assumed.

G84 X(U)_ C(H)_Z(W)_ R_ P_ Q_ F_ K_ M_ ;
or
G88 Z(W)_ C(H)_X(U)_ R_ P_ Q_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ 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 the hole
Q_ : Depth of cut for each cutting feed
F_ : The cutting feedrate
K_ : Number of repeats (when it is needed.)
M_ : M code for C-axis clamp (when it is needed.)



- Peck rigid tapping cycle

In the first cutting from point R, perform cutting by depth "q" specified by address Q while rotating the spindle in the forward direction (operation <1>).
Then, return to point R by rotating the spindle in the reverse direction (operation <2>).
Then, rotate the spindle in the forward direction and perform cutting to the position indicated by [(Position to which cutting was performed previously) - (Cutting start distance set in parameter No. 5213)] as movement to the cutting start point (operation <3>).
Continue cutting by (d+q) (operation <4>).

Repeat operations <2> to <4> until the bottom of the hole is reached.

The cutting speed and rigid tapping time constant are used for operations <1> and <4>.
For operations <2>, <3>, and travel from the bottom of the hole (point Z) to point R, rigid tapping extract override is enabled and the rigid tapping extract time constant is used.

The symbols in the figure above indicate the following operations.

- - - → : Positioning (Rapid traverse G00)
——→ : Cutting feed (Linear interpolation G01)
P1 : Dwell programmed by the address P command
Mα : Output of the M code for C-axis clamp (The α value is set in parameter No. 5110.)
M(α+1) : Output of the M code for C-axis unclamp
P2 : Dwell set by parameter No.5111

Note P1, Mα, M(α+1), and P2 are not executed or output without being specified or set.

Explanation

- Cutting start distance

Cutting start distance d is set by parameter No. 5213.

- Amount of return

Amount of return for each time d is set by parameter No. 5213.

- Return speed

For the speed of return operation, a maximum of 2000% of override can be enabled by setting DOV (bit 4 of parameter No. 5200), OVU (bit 3 of parameter No. 5201), and parameter No. 5211 as with travel from the bottom of the hole (point Z) to point R.

- Speed during cutting into the cutting start point

For the speed during cutting into the cutting start point, a maximum of 2000% of override can be enabled by setting DOV (bit 4 of parameter No. 5200), OVU (bit 3 of parameter No. 5201), and parameter No. 5211 as with travel from the bottom of the hole (point Z) to point R.

- Acceleration/deceleration after interpolation

Linear or bell-shaped acceleration/deceleration can be applied.

- Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

- Override

Various types of override functions are invalid. The following override functions can be enabled by setting corresponding parameters:

- Extraction override
- Override signal

Details are given later.

- Dry run

Dry run can be executed also in G84 (G88). When dry run is executed at the feedrate for the drilling axis in G84 (G88), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock

Machine lock can be executed also in G84 (G88).
When G84 (G88) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

- Reset

When a reset is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G88) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G88).

- Feed hold and single block

When parameter FHD (No. 5200#6) is set to 0, feed hold and single block are invalid in the G84 (G88) mode. When this bit is set to 1, they are valid.

- Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle."
With other manual operations, rigid tapping cannot be performed.

- Backlash compensation

In the rigid tapping mode, backlash compensation is applied to compensate the lost motion when the spindle rotates clockwise or counterclockwise. Set the amount of backlash in parameters Nos. 5321 to 5324.
Along the drilling axis, backlash compensation has been applied.

- Series 10/11 format

When bit 1 (FCV) of parameter No. 0001 is set to 1, execution is enabled with G84.2. The same operation as with G84 is performed. However, the command format for the repetitive count is L.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm PS0206 is issued.

- S commands

If a speed higher than the maximum speed for the gear being used is specified, alarm PS0200 is issued.
When the rigid tapping canned cycle is cancelled, the S command used for rigid tapping is cleared to S0.

- Distribution amount for the spindle

The maximum distribution amount is 32,767 pulses per 8 msec for a serial spindle. (displayed on diagnostic display No. 451)
This amount is changed according to the gear ratio setting for the position coder or rigid tapping command. If a setting is made to exceed the upper limit, alarm PS0202 is issued.

- F command

Specifying a value larger than the upper limit for cutting feed will cause alarm PS0011 to be issued.

- Unit of F command

	Metric input	Inch input	Remarks
G98	1mm/min	0.01inch/min	Decimal point programming allowed
G99	0.01mm/rev	0.0001inch/rev	Decimal point programming allowed

- M29

If an S command and axis movement are specified between M29 and G84, alarm PS0203 is issued. If M29 is specified in a tapping cycle, alarm PS0204 is issued.

- Rigid tapping command M code

The M code used to specify the rigid tapping mode is set in parameter No. 5210.

- P/Q

Specify P and Q in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data.
When Q0 is specified, the peck rigid tapping cycle is not performed.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G84 in a single block. Otherwise, G84 will be canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P_ in an independent block.

- d (parameter No.5213)

Perform operation in the peck tapping cycle within point R. That is, set a value which does not exceed point R for d (parameter No. 5213).

4.4.3 Canned Cycle Cancel (G80)

The rigid tapping canned cycle is canceled. For how to cancel this cycle, see II-4.3.4.

NOTE
When the rigid tapping canned cycle is cancelled, the S value used for rigid tapping is also cleared (as if S0 is specified).
Accordingly, the S command specified for rigid tapping cannot be used in a subsequent part of the program after the cancellation of the rigid tapping canned cycle.
After canceling the rigid tapping canned cycle, specify a new S command as required.

4.4.4 Override during Rigid Tapping

Various types of override functions are invalid. The following override functions can be enabled by setting corresponding parameters:

- Extraction override
- Override signal

4.4.4.1 Extraction override

For extraction override, the fixed override set in the parameter or override specified in a program can be enabled at extraction (including retraction during peck drilling/high-speed peck drilling).

Explanation

- Specifying the override in the parameter

Set bit 4 (DOV) of parameter No. 5200 to 1 and set the override in parameter No. 5211.
An override from 0% to 200% in 1% steps can be set. Bit 3 (OVU) of parameter No. 5201 can be set to 1 to set an override from 0% to 2000% in 10% steps.

- Specifying the override in a program

Set bit 4 (DOV) of parameter No. 5200 and bit 4 (OV3) of parameter No. 5201 to 1. The spindle speed at extraction can be specified in the program.
Specify the spindle speed at extraction using address "J" in the block in which rigid tapping is specified.
Example)

To specify 1000 min^{-1} for S at cutting and 2000 min^{-1} for S at extraction

```
.
M29 S1000 ;
G84 Z-100. F1000. J2000 ;
.
```

The difference in the spindle speed is converted to the actual override by the following calculation.
Therefore, the spindle speed at extraction may not be the same as that specified at address "J". If the override does not fall in the range between 100% and 200%, it is assumed to be 100%.

$$\text{Override}(\%) = \frac{\text{Spindle speed at extraction (specified at } J)}{\text{Spindle speed (specified at } S)} \times 100$$

Bit 6 (OVE) of parameter No. 5202 can be set to 1 to extend the override value to 100% to 2000%. If the specified override value is outside the range between 100% and 2000%, it is assumed to be 100%.

The override to be applied is determined according to the setting of parameters and that in the command as shown in the table below.

When bit 6 (OVE) of parameter No. 5202 is set to 0

Command \ Parameter setting		DOV=1		DOV=0
		OV3=1	OV3=0	
Spindle speed at extraction specified at address J	Within the range between 100 to 200%	Command in the program	Parameter No. 5211	100%
	Within the range between 100 to 200%	100%		
No spindle speed at extraction specified at address J		Parameter No. 5211		

When bit 6 (OVE) of parameter No. 5202 is set to 1

Command \ Parameter setting		DOV=1		DOV=0
		OV3=1	OV3=0	
Spindle speed at extraction specified at address J	Within the range between 100 to 2000%	Command in the program	Parameter No. 5211	100%
	Within the range between 100 to 2000%	100%		
No spindle speed at extraction specified at address J		Parameter No. 5211		

NOTE

1 Do not use a decimal point in the value specified at address "J".
If a decimal point is used, the value is assumed as follows:
Example) When the increment system for the reference axis is IS-B
- When pocket calculator type decimal point programming is not used
The specified value is converted to the value for which the least input increment is considered.
"J200." is assumed to be 200000 min^{-1}.
- When pocket calculator type decimal point programming is used
The specified value is converted to the value obtained by rounding down to an integer.
"J200." is assumed to be 200 min^{-1}.

2 Do not use a minus sign in the value specified at address "J".
If a minus sign is used, a value outside the range is assumed to be specified.

3 The maximum override is obtained using the following equation so that the spindle speed to which override at extraction is applied do not exceed the maximum used gear speed (specified in parameters Nos. 5241 to 5244). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

$$\text{Maximum override (\%)} = \frac{\text{Maximum spindle speed (specified in parameters)}}{\text{Spindle speed (specified at } S\text{)}} \times 100$$

4 When a value is specified at address "J" for specifying the spindle speed at extraction in the rigid tapping mode, it is valid until the canned cycle is canceled.

4.4.4.2 Override signal

By setting bit 4 (OVS) of parameter No. 5203 to 1, override can be applied to cutting/extraction operation during rigid tapping as follows:

- Applying override using the feedrate override signal
- Canceling override using the override cancel signal

There are the following relationships between this function and override to each operation:

- At cutting
  - When the override cancel signal is set to 0
    Value specified by the override signal
  - When the override cancel signal is set to 1
    100%
- At extraction
  - When the override cancel signal is set to 0
    Value specified by the override signal
  - When the override cancel signal is set to 1 and extraction override is disabled
    100%
  - When the override cancel signal is set to 1 and extraction override is enabled
    Value specified for extraction override

NOTE

1 The maximum override is obtained using the following equation so that the spindle speed to which override is applied do not exceed the maximum used gear speed (specified in parameters Nos. 5241 to 5244). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

$$\text{Maximum override (\%)} = \frac{\text{Maximum spindle speed (specified in parameters)}}{\text{Spindle speed (specified at } S\text{)}} \times 100$$

2 Since override operation differs depending on the machine in use, refer to the manual provided by the machine tool builder.

4.5 CANNED GRINDING CYCLE (FOR GRINDING MACHINE)

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

- Traverse grinding cycle (G71)
  (G72 when G code system C is used)
- Traverse direct constant-size grinding cycle (G72)
  (G73 when G code system C is used)
- Oscillation grinding cycle (G73)
  (G74 when G code system C is used)
- Oscillation direct constant-size grinding cycle (G74)
  (G75 when G code system C is used)

In the descriptions below, an axis used for cutting with a grinding wheel and an axis used for grinding with a grinding wheel are referred to as follows:

Axis used for cutting with a grinding wheel: Cutting axis
Axis used for grinding with a grinding wheel: Grinding axis

For a depth of cut on a cutting axis and a distance of grinding on a grinding axis, the incremental system (parameter No. 1013) for the reference axis (parameter No. 1031) is used. If 0 is set in parameter No. 1031 (reference axis), the incremental system for the first axis is used.

NOTE
The canned grinding cycle is an optional function. The canned grinding cycle and multiple repetitive cycle cannot be used simultaneously for the same path.
To use the canned grinding cycle, set bit 0 (GFX) of parameter No. 5106 to 1.

⚠ WARNING

The G codes for canned grinding cycles G71, G72, G73, and G74 (G72, G73, G74, and G75 when G code system C is used) are G codes of group 01. A G code for cancellation such as G80 used for a canned cycle for drilling is unavailable. By specifying a G code of group 00 other than G04, modal information such as a depth of cut is cleared but no canned grinding cycle can be canceled. To cancel a canned grinding cycle, a G code of group 01 other than G71, G72, G73, and G74 needs to be specified. So, when switching to another axis move command from G71, G72, G73 or G74, for example, be sure to specify a G code of group 01 such as G00 or G01 to cancel the canned grinding cycle. If another axis move command is specified without canceling the canned grinding cycle, an unpredictable operation can result because of continued cycle operation.

NOTE

1 If the G code for a canned grinding cycle (G71, G72, G73, or G74) is specified, the canned grinding cycle is executed according to the values of A, B, W, U, I, and K preserved as modal data while the cycle is valid, even if a block specified later specifies none of G71, G72, G73, and G74.
Example:

```
G71 A_ B_ W_ U_ I_ K_ H_ ;
  ;   ← The canned grinding cycle is executed
        even if an empty block is specified.
%
```

2 When switching from a canned cycle for drilling to a canned grinding cycle, specify G80 to cancel the canned cycle for drilling.
3 When switching from a canned grinding cycle to another axis move command, cancel the canned cycle according to the warning above.
4 A canned grinding cycle and multiple repetitive canned cycle cannot be used simultaneously on the same path.

When the canned grinding cycle option is enabled, which function is to be used can be selected using bit 0 (GFX) of parameter No. 5106.

4.5.1 Traverse Grinding Cycle (G71)

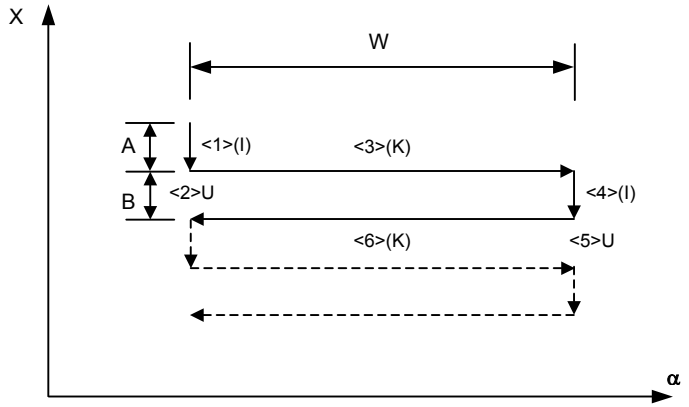
A traverse grinding cycle can be executed.

Format

G71 A_ B_ W_ U_ I_ K_ H_ ;

A_ : First depth of cut (The cutting direction depends on the sign.)
B_ : Second depth of cut (The cutting direction depends on the sign.)
W_ : Grinding range (The grinding direction depends on the sign.)
U_ : Dwell time
I_ : Feedrate for A and B
K_ : Feedrate for W
H_ : Repetitive count (from 1 to 9999)

G71



Explanation

The traverse grinding cycle consists of six operations.
The operations from <1> to <6> are repeated until the repetitive count specified in address H is reached. In the case of single block operation, the operations from <1> to <6> are executed with one cycle start operation.

- Operation sequence in a cycle

<1> Cutting with a grinding wheel

By the first depth of cut specified in A, cutting is performed by cutting feed in the X-axis direction. The feedrate specified in I is used.

<2> Dwell

A dwell operation is performed for the time specified in U.

<3> Grinding

A movement is made by the distance specified in W by cutting feed. Set a grinding axis in parameter No.5176. The feedrate specified in K is used.

<4> Cutting with a grinding wheel

By the second depth of cut specified in B, cutting is performed by cutting feed in the X-axis direction. The feedrate specified in I is used.

<5> Dwell

A dwell operation is performed for the time specified in U.

<6> Grinding (return direction)

A movement is made at the feedrate specified in K in the reverse direction by the distance specified in W.

Limitation

- Cutting axis

As a cutting axis, the first controlled axis is used. By setting bit 0 (FXY) of parameter No. 5101 to 1, the axis can be switched using a plane selection command (G17, G18, or G19).

- Grinding axis

Specify a grinding axis by setting an axis number for other than the cutting axis in parameter No. 5176. Specify a grinding command in W at all times without using an axis name. The axis name corresponding to the set axis number can also be used for specification.

- A, B, W

The commands of A, B, and W are all incremental commands.
When none of A and B are specified or A=B=0, spark-out operation (execution of only movement in the grinding direction) is performed.

- H

When H is not specified or H=0, the specification of H=1 is assumed.
The specification of H is valid only in the block where H is specified.

- Clear

The data A, B, W, U, I, and K in the canned cycle is modal information common to G71, G72, G73, and G74. So, the data remains valid until new data is specified. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G71, G72, G73, and G74 is specified. The specification of H is valid only in the block where H is specified.

- B code

During the canned cycle, no B code (second auxiliary function) can be specified.

NOTE
1 If no grinding axis is specified when G71 is specified, alarm PS0455 is issued.
2 If the specified cutting axis number and grinding axis number are the same, alarm PS0456 is issued.
3 Even if G90 (absolute command) is specified while this cycle is valid, each of the A, B, and W commands is an incremental command.

4.5.2 Traverse Direct Constant-Size Grinding Cycle (G72)

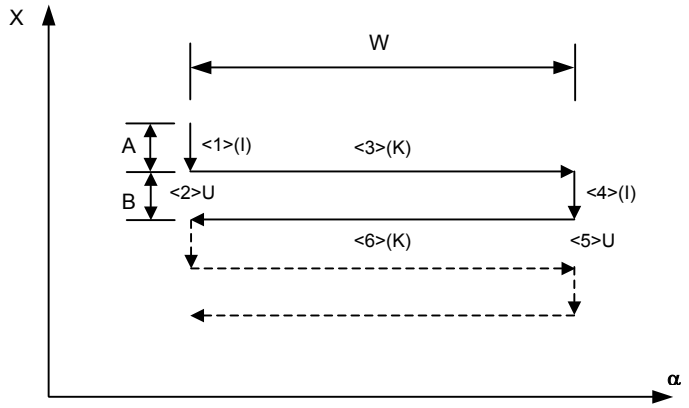
A traverse direct constant-size grinding cycle can be executed.

Format

G72 P_ A_ B_ W_ U_ I_ K_ H_ ;

P_ : Gage number (1 to 4)
A_ : First depth of cut (The cutting direction depends on the sign.)
B_ : Second depth of cut (The cutting direction depends on the sign.)
W_ : Grinding range (The grinding direction depends on the sign.)
U_ : Dwell time
I_ : Feedrate for A and B
K_ : Feedrate for W
H_ : Repetitive count (from 1 to 9999)

G72



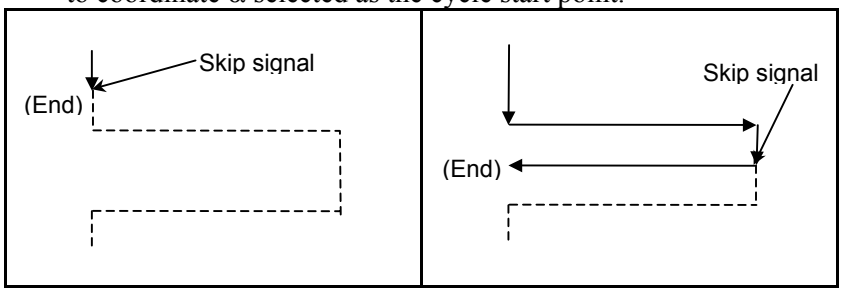
Explanation

If the multi-step skip option is specified, a gage number can be specified. The method of gage number specification is the same as for the multi-step skip option. If the multi-step skip option is not specified, the conventional skip signal is used.
The commands and operations other than gage number specification are the same as for G71.

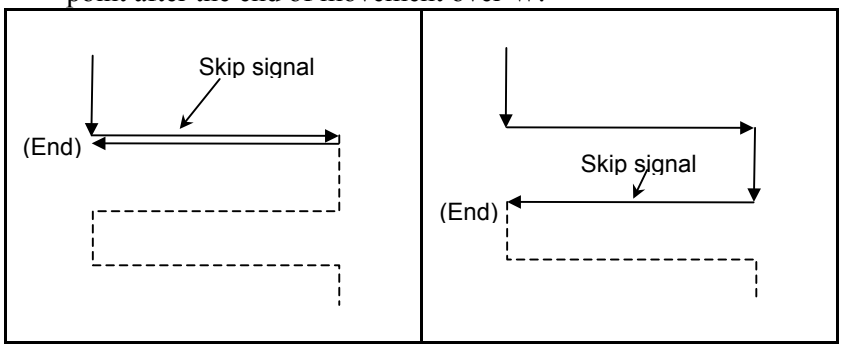
- Operation performed when the skip signal is input

A G72 cycle can be terminated after interrupting the current operation (or after ending the current operation) by inputting the skip signal during execution of the cycle.
Each operation of the sequence performed when the skip signal is input is described below.

- If the skip signal is input during operation <1> or <4> (movement by A or B), cutting is immediately stopped to return to coordinate α selected as the cycle start point.



- If the skip signal is input during operation <2> or <5> (dwell), dwell operation is immediately stopped to return to coordinate α selected as the cycle start point.
- If the skip signal is input during operation <3> or <6> (grinding feed), the tool returns to coordinate α selected as the cycle start point after the end of movement over W.



Limitation

- Cutting axis

As a cutting axis, the first controlled axis is used. By setting bit 0 (FXY) of parameter No. 5101 to 1, the axis can be switched using a plane selection command (G17, G18, or G19).

- Grinding axis

Specify a grinding axis by setting an axis number for other than the cutting axis in parameter No. 5177. Specify a grinding command in W at all times without using an axis name. The axis name corresponding to the set axis number can also be used for specification.

- P

If a value other than P1 to P4 is specified, the skip function is disabled.
The specification of P is valid only in the block where P is specified.

- A, B, W

The commands of A, B, and W are all incremental commands.
When none of A and B are specified or A=B=0, spark-out operation (execution of only movement in the grinding direction) is performed.

- H

When H is not specified or H=0, the specification of H=1 is assumed.
The specification of H is valid only in the block where H is specified.

- Clear

The data A, B, W, U, I, and K in the canned cycle is modal information common to G71, G72, G73, and G74. So, the data remains valid until new data is specified. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G71, G72, G73, and G74 is specified. The specification of P or H is valid only in the block where P or H is specified.

- B code

During the canned cycle, no B code (second auxiliary function) can be specified.

NOTE
1 If no grinding axis is specified when G72 is specified, alarm PS0455 is issued.
2 If the specified cutting axis number and grinding axis number are the same, alarm PS0456 is issued.
3 Even if G90 (absolute command) is specified while this cycle is valid, each of the A, B, and W commands is an incremental command.
4 If a value from P1 to P4 is specified without specifying the multi-step skip option, alarm PS0370 is issued.

4.5.3 Oscillation Grinding Cycle (G73)

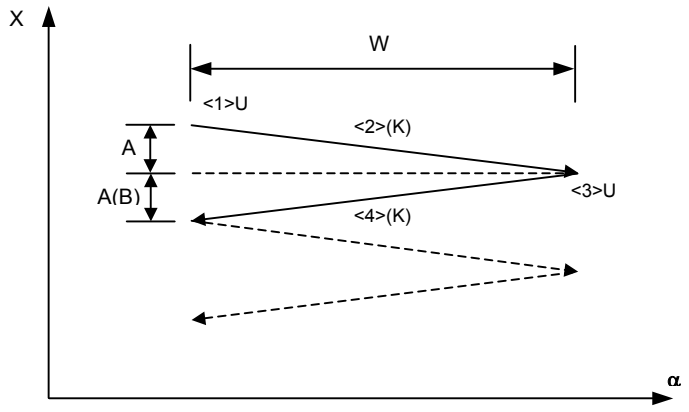
An oscillation grinding cycle can be executed.

Format

G73 A_ (B_) W_ U_ K_ H_ ;

A_ : First depth of cut (The cutting direction depends on the sign.)
B_ : Second depth of cut (The cutting direction depends on the sign.)
W_ : Grinding range (The grinding direction depends on the sign.)
U_ : Dwell time
K_ : Feedrate for W
H_ : Repetitive count (from 1 to 9999)

G73



Explanation

The oscillation grinding cycle consists of four operations.
The operations from <1> to <4> are repeated until the repetitive count specified in address H is reached. In the case of single block operation, the operations from <1> to <4> are executed with one cycle start operation.

- Operation sequence in a cycle

<1> Dwell

A dwell operation is performed for the time specified in U.

<2> Cutting + grinding with a grinding wheel

Cutting feed is performed on the cutting axis (X-axis) and a grinding axis at the same time. The amount of movement on the cutting axis (depth of cut) is the first depth of cut specified in A, and the amount of movement on a grinding axis is the distance specified in W. Set a grinding axis in parameter No.5178. The feedrate specified in K is used.

<3> Dwell

A dwell operation is performed for the time specified in U.

<4> Cutting + grinding with a grinding wheel (return direction)

Cutting feed is performed on the cutting axis (X-axis) and a grinding axis at the same time. The amount of movement on the cutting axis (depth of cut) is the second depth of cut specified in B, and the amount of movement on the grinding axis is the distance specified in W. The feedrate specified in K is used.

Limitation

- Cutting axis

As a cutting axis, the first controlled axis is used. By setting bit 0 (FXY) of parameter No. 5101 to 1, the axis can be switched using a plane selection command (G17, G18, or G19).

- Grinding axis

Specify a grinding axis by setting an axis number for other than the cutting axis in parameter No. 5178. Specify a grinding command in W at all times without using an axis name. The axis name corresponding to the set axis number can also be used for specification.

- B

If B is not specified, B=A is assumed.
The specification of B is valid only in the block where B is specified.

- A, B, W

The commands of A, B, and W are all incremental commands.
When none of A and B are specified or A=B=0, spark-out operation (execution of only movement in the grinding direction) is performed.

- H

When H is not specified or H=0, the specification of H=1 is assumed.
The specification of H is valid only in the block where H is specified.

- Clear

The data A, W, U, and K in the canned cycle is modal information common to G71, G72, G73, and G74. So, the data remains valid until new data is specified. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G71, G72, G73, and G74 is specified. The specification of B or H is valid only in the block where B or H is specified.

- B code

During the canned cycle, no B code (second auxiliary function) can be specified.

NOTE
1 If no grinding axis is specified when G73 is specified, alarm PS0455 is issued.
2 If the specified cutting axis number and grinding axis number are the same, alarm PS0456 is issued.
3 Even if G90 (absolute command) is specified while this cycle is valid, each of the A, B, and W commands is an incremental command.

4.5.4 Oscillation Direct Constant-Size Grinding Cycle (G74)

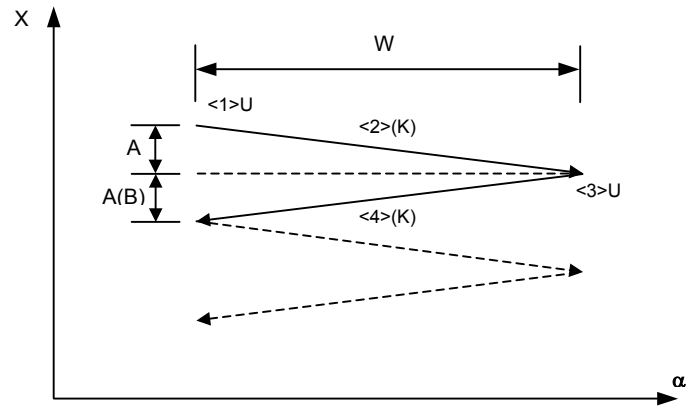
An oscillation direct constant-size grinding cycle can be executed.

Format

G74 P_ A_ (B_) W_ U_ K_ H_ ;

P_ : Gage number (1 to 4)
A_ : First depth of cut (The cutting direction depends on the sign.)
B_ : Second depth of cut (The cutting direction depends on the sign.)
W_ : Grinding range (The grinding direction depends on the sign.)
U_ : Dwell time
K_ : Feedrate for W
H_ : Repetitive count (from 1 to 9999)

G74



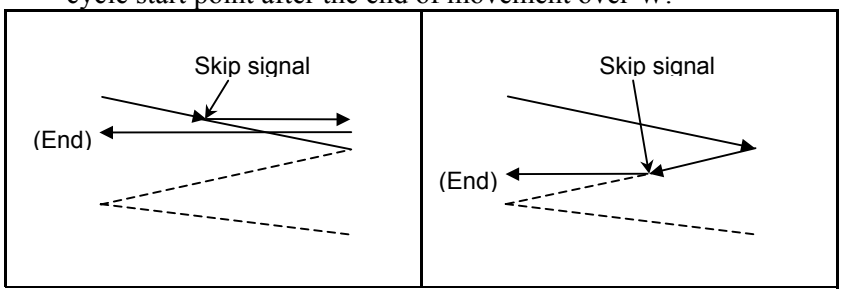
Explanation

If the multi-step skip option is specified, a gage number can be specified. The method of gage number specification is the same as for the multi-step skip option. If the multi-step skip option is not specified, the conventional skip signal is used.
The commands and operations other than gage number specification are the same as for G73.

- Operation performed when the skip signal is input

A G74 cycle can be terminated after interrupting the current operation (or after ending the current operation) by inputting the skip signal during execution of the cycle.
Each operation of the sequence performed when the skip signal is input is described below.

- If the skip signal is input during operation <1> or <3> (dwell), dwell operation is immediately stopped to return to coordinate α selected as the cycle start point.
- If the skip signal is input during operation <2> or <4> (A, B, grinding feed), the tool returns to coordinate α selected as the cycle start point after the end of movement over W.



Limitation

- Cutting axis

As a cutting axis, the first controlled axis is used. By setting bit 0 (FXY) of parameter No. 5101 to 1, the axis can be switched using a plane selection command (G17, G18, or G19).

- Grinding axis

Specify a grinding axis by setting an axis number for other than the cutting axis in parameter No. 5179. Specify a grinding command in W at all times without using an axis name. The axis name corresponding to the set axis number can also be used for specification.

- P

If a value other than P1 to P4 is specified, the skip function is disabled.
The specification of P is valid only in the block where P is specified.

- B

If B is not specified, B=A is assumed.
The specification of B is valid only in the block where B is specified.

- A, B, W

The commands of A, B, and W are all incremental commands.
When none of A and B are specified or A=B=0, spark-out operation (execution of only movement in the grinding direction) is performed.

- H

When H is not specified or H=0, the specification of H=1 is assumed.
The specification of H is valid only in the block where H is specified.

- Clear

The data A, W, U, and K in the canned cycle is modal information common to G71, G72, G73, and G74. So, the data remains valid until new data is specified. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G71, G72, G73, and G74 is specified. The specification of P, B, or H is valid only in the block where P, B, or H is specified.

- B code

During the canned cycle, no B code (second auxiliary function) can be specified.

NOTE
1 If no grinding axis is specified when G74 is specified, alarm PS0455 is issued.
2 If the specified cutting axis number and grinding axis number are the same, alarm PS0456 is issued.
3 Even if G90 (absolute command) is specified while this cycle is valid, each of the A, B, and W commands is an incremental command.
4 If a value from P1 to P4 is specified without specifying the multi-step skip option, alarm PS0370 is issued.

4.6 CHAMFERING AND CORNER R

Overview

A chamfering or corner R block can automatically be inserted between linear interpolation (G01) along a single axis and that along a single axis normal to that single axis.
Chamfering or corner R is inserted for a command to move the tool along two axes on the plane determined by the plane selection (G17, G18, or G19) command.

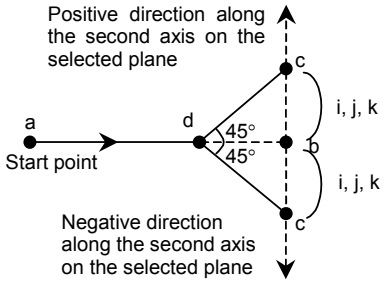
NOTE
To enable the chamfering and corner R function, bit 2 (CCR) of parameter No. 8134 to 1.

Format

- Chamfering

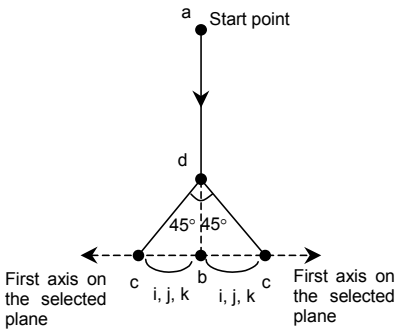
First axis on the selected plane → second axis on the selected plane (G17 plane: $X_P \rightarrow Y_P$, G18 plane: $Z_P \rightarrow X_P$, G19 plane: $Y_P \rightarrow Z_P$)

Format		
G17 plane:	**G01**	**X_P(U)_ J(C)±j ;**
G18 plane:	**G01**	**Z_P(W)_ I(C)±i ;**
G19 plane:	**G01**	**Y_P(V)_ K(C)±k ;**

Explanation		Tool movement
X_P(U)__ **Y_P(V)__** **Z_P(W)__**	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right. X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.	 Moves from a to d and c.
I(C)±i **J(C)±j** **K(C)±k**	Specify the distance between points b and c in the figure shown at right with a sign following address I, J, K, or C. (Use I, J, or K when bit 4 (CCR) of parameter No. 3405 is set to 0 or C when the bit is set to 1.)	(Positive direction along the second axis on the selected plane when a plus sign is specified at I, J, K, or C or negative direction when a minus sign is specified at I, J, K, or C)

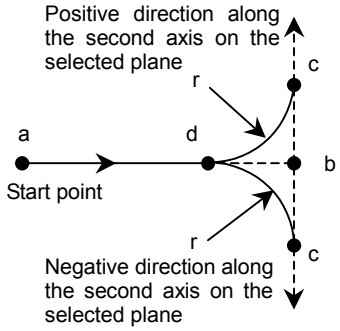
- Chamfering

Second axis on the selected plane → first axis on the selected plane (G17 plane: $Y_P \rightarrow X_P$, G18 plane: $X_P \rightarrow Z_P$, G19 plane: $Z_P \rightarrow Y_P$)

<table>
<tr><th colspan="3">Format</th></tr>
<tr><td colspan="3">G17 plane: G01 Y_P(V)_ I(C)±i ;
G18 plane: G01 X_P(U)_ K(C)±k ;
G19 plane: G01 Z_P(W)_ J(C)±j ;</td></tr>
<tr><th colspan="2">Explanation</th><th>Tool movement</th></tr>
<tr><td>X_P(U)__
Y_P(V)__
Z_P(W)__</td><td>Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.
X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.</td><td rowspan="2">Moves from a to d and c.
(Positive direction along the first axis on the selected plane when a plus sign is specified at I, J, K, or C or negative direction when a minus sign is specified at I, J, K, or C)

</td></tr>
<tr><td>I(C)±i
J(C)±j
K(C)±k</td><td>Specify the distance between points b and c in the figure shown at right with a sign following address I, J, K, or C. (Use I, J, or K when bit 4 (CCR) of parameter No. 3405 is set to 0 or C when the bit is set to 1.)</td></tr>
</table>

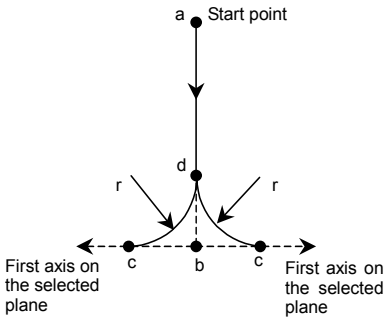
- Corner R

First axis on the selected plane → second axis on the selected plane
(G17 plane: $X_P \rightarrow Y_P$, G18 plane: $Z_P \rightarrow X_P$, G19 plane: $Y_P \rightarrow Z_P$)

Format		
G17 plane: **G01 X_P(U)_ R±r ;** **G18 plane:** **G01 Z_P(W)_ R±r ;** **G19 plane:** **G01 Y_P(V)_ R±r ;**		
Explanation		**Tool movement**
X_P(U)__ **Y_P(V)__** **Z_P(W)__**	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right. X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.	 Moves from a to d and c. (Positive direction along the second axis on the selected plane when +r is specified at R or negative direction when -r is specified at R)
R±r	Specify the radius of the arc connecting points d and c in the figure shown at right with a sign following address R.	

- Corner R

Second axis on the selected plane → first axis on the selected plane (G17 plane: $Y_P \rightarrow X_P$, G18 plane: $X_P \rightarrow Z_P$, G19 plane: $Z_P \rightarrow Y_P$)

Format		
G17 plane: **G01 Y_P(V)_ R±r ;** **G18 plane:** **G01 X_P(U)_ R±r ;** **G19 plane:** **G01 Z_P(W)_ R±r ;**		
Explanation		**Tool movement**
X_P(U)__ **Y_P(V)__** **Z_P(W)__**	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right. X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.	Moves from a to d and c. (Positive direction along the first axis on the selected plane when +r is specified at R or negative direction when -r is specified at R) 
R±r	Specify the radius of the arc connecting points d and c in the figure shown at right with a sign following address R.	

Explanation

By G01 specified for chamfering or corner R, the tool must be moved only along one of the two axes on the selected plane. The command in the next block must move the tool only along the other axis on the selected plane.

Example:

When the A-axis is set as an axis parallel to the basic X-axis (by setting parameter No. 1022 to 5), the following program performs chamfering between cutting feed along the A-axis and that along the Z-axis:

```
G18 A0 Z0
G00 A100.0 Z100.0
G01 A200.0 F100 K30.0
Z200.0
```

The following program causes an alarm. (Because chamfering is specified in the block to move the tool along the X-axis, which is not on the selected plane)

```
G18 A0 Z0
G00 A100.0 Z100.0
G01 X200.0 F100 K30.0
Z200.0
```

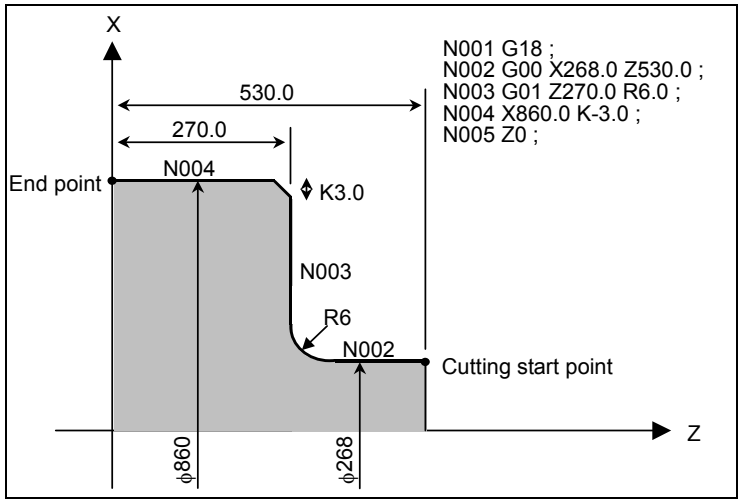
The following program also causes an alarm. (Because the block next to the chamfering command moves the tool along the X-axis, which is not on the selected plane)

```
G18 A0 Z0
G00 A100.0 Z100.0
G01 Z200.0 F100 I30.0
X200.0
```

A radius value is specified at I, J, K, R, and C.

In an incremental programming, use point b in the figure in "Format" as the start point in the block next to a chamfering or corner R block. That is, specify the distance from point b. Do not specify the distance from point c.

Example



Limitation

- Alarms

In the following cases, an alarm is issued:

1) Chamfering or corner R is specified in a block for threading (alarm PS0050).
2) G01 is not specified in the block next to the G01 block in which chamfering or corner R is specified (alarm PS0051 or PS0052).
3) An axis which is not on the selected plane is specified as a move axis in the block in which chamfering or corner R is specified or the next block (alarm PS0051 or PS0052).
4) A plane selection command (G17, G18, or G19) is specified in the block next to the block in which chamfering or corner R is specified (alarm PS0051).
5) When bit 4 (CCR) of parameter No. 3405 is set to 0 (to specify chamfering at I, J, or K), two or more of I, J, K, and R are specified in G01 (alarm PS0053).
6) Chamfering or corner R is specified in the G01 block to move the tool along more than one axis (alarm PS0054).
7) The travel distance along an axis specified in the block in which chamfering or corner R is specified is smaller than the amount of chamfering or corner R (alarm PS0055). (See the figure below.)

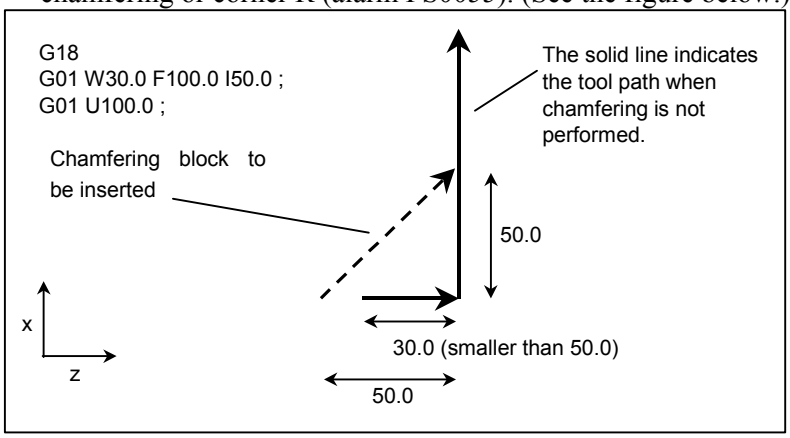


Fig. 4.6 (a) Example of machining which causes alarm PS0055

8) An invalid combination of a move axis and I, J, or K is specified for chamfering (alarm PS0306).

9) An invalid sign is specified at I, J, K, R, or C (chamfering or corner R in the direction opposite to the movement in the next block is specified) (alarm PS0051). (See the figure below.)

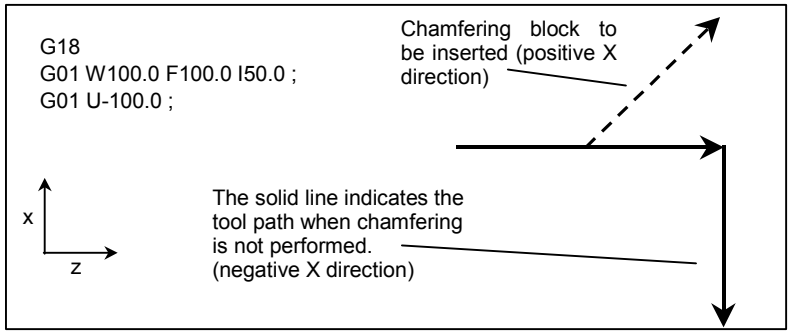


Fig. 4.6 (b) Example of machining which causes alarm PS0051

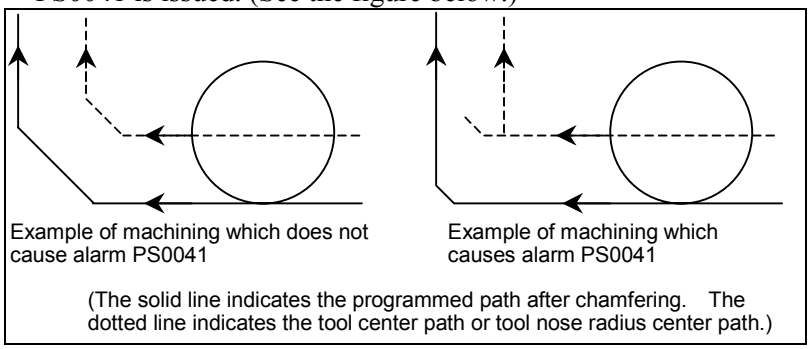
- Single block operation

When the block in which chamfering or corner R is specified is executed in the single block mode, operation continues to the end point of the inserted chamfering or corner R block and the machine stops in the feed hold mode at the end point. When bit 0 (SBC) of parameter No. 5105 is set to 1, the machine stops in the feed hold mode also at the start point of the inserted chamfering or corner R block.

- Tool nose radius compensation

When applying tool nose radius compensation, note the following points:

1. If the amount of inner chamfering or corner R is too small as compared with compensation and cutting is generated, alarm PS0041 is issued. (See the figure below.)



2. A function is available which intentionally changes the compensation direction by specifying the I, J, or K command in the G01 block in the tool nose radius compensation mode (see the explanations of tool nose radius compensation). To use this function when the chamfering and corner R function is enabled (bit 2 (CCR) of parameter No. 8134 is set to 1), set bit 4 (CCR) of parameter No. 3405 is set to 1 so that the I, J, and K commands are not used as chamfering commands. Operation to be performed under each condition is explained below.

(1) When the chamfering and corner R function is not used (bit 2 (CCR) of parameter No.8134 = 0)
In the G01 block in the tool nose radius compensation mode, the tool nose radius compensation direction can be specified at address I, J, or K.
No chamfering is performed.

(2) When the chamfering and corner R function is used (bit 2 (CCR) of parameter No.8134 = 1)

(2-1) When bit 4 (CCR) of parameter No. 3405 is set to 0
In the G01 block in the tool nose radius compensation mode, chamfering can be specified at address I, J, or K. Corner R can also be specified at address R.
The tool nose radius compensation direction cannot be specified.

(2-2) When bit 4 (CCR) of parameter No. 3405 is set to 1
In the G01 block in the tool nose radius compensation mode, the tool nose radius compensation direction can be specified at address I, J, or K.
Chamfering or corner R can also be specified at address C or R.

- Direct drawing dimension programming

The chamfering and corner R function and direct drawing dimension programming cannot be used simultaneously.
When the chamfering and corner R function is enabled (bit 2 (CCR) of parameter No. 8134 is set to 1), bit 0 (CRD) of parameter No. 3453 can be set to 1 to enable direct drawing dimension programming. (With this setting, the chamfering and corner R function is disabled.)

4.7 MIRROR IMAGE FOR DOUBLE TURRET (G68, G69)

Overview

When a unit has a double turret consisting of two tool posts which face each other on the same controlled axis, mirror image can be applied to the X-axis with a G code command. Symmetrical cutting can be performed by creating a machining program for the facing tool posts as if they were in the coordinate system on the same side.

Format

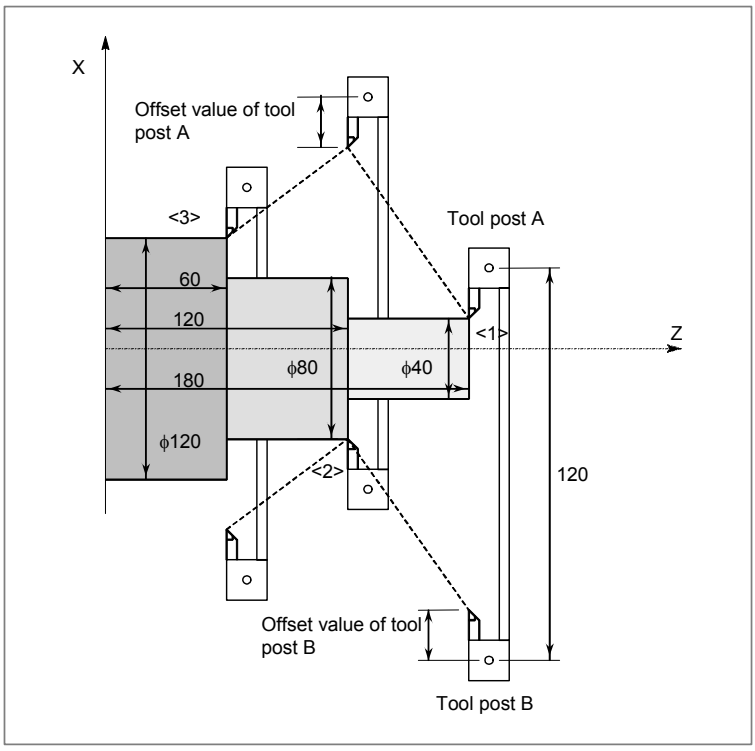
G68 : Double turret mirror image on
G69 : Mirror image cancel

Explanation

Mirror image can be applied to the X-axis of the three basic axes that is set by parameter No. 1022 with the G code command.
When G68 is designated, the coordinate system is shifted to the double turret side, and the X-axis sign is reversed from the programmed command to perform symmetrical cutting. This function is called the mirror image for double turret.
To use this function, set the distance between the two tool posts to a parameter No. 1290.

Example

- For turning



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

NOTE
A diameter value is specified for the X-axis.

Limitation

NOTE
1 When the G68 command based on this function is enabled, the X-axis coordinate value that can be read with the custom macro system variables #5041 and up or #100101 and up (current specified position (in the workpiece coordinate system)) is a position with mirror image applied.
2 This function cannot be used together with the balanced cutting function (for a 2-path system). To use this function, set bit 0 (NVC) of parameter No. 8137 to 1.

4.8 DIRECT DRAWING DIMENSION PROGRAMMING

Overview

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

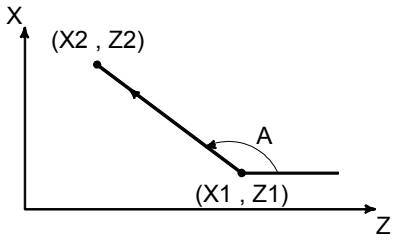
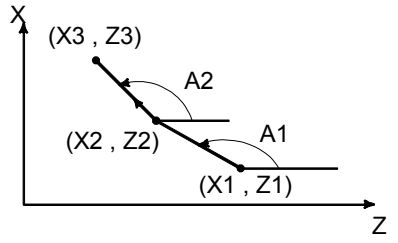
NOTE
To use direct drawing dimension programming when the chamfering and corner R function is enabled (bit 2 (CCR) of parameter No. 8134 is set to 1), set bit 0 (CRD) of parameter No. 3453 to 1. (With this setting, the chamfering and corner R function is disabled.)

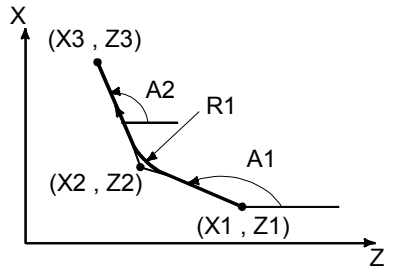
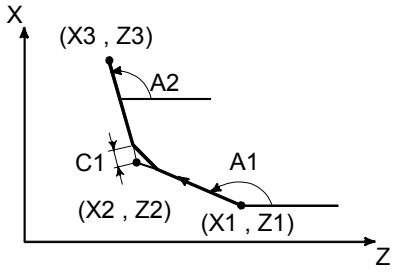
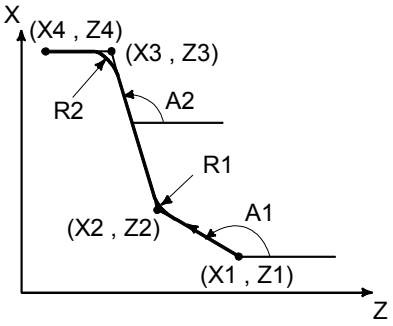
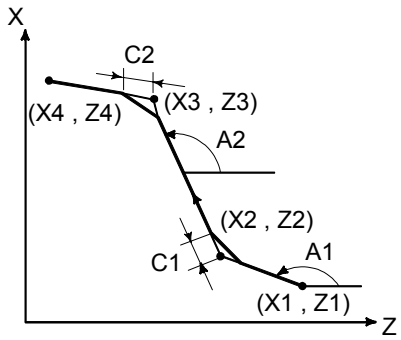
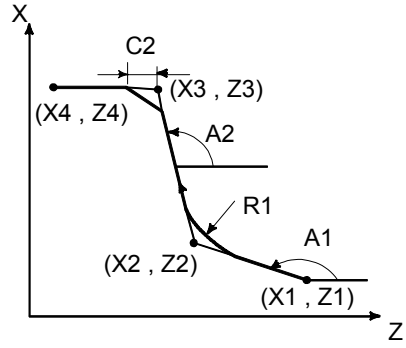
Format

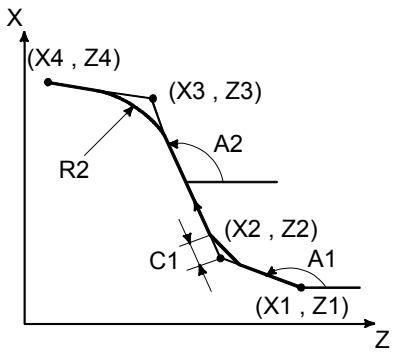
Examples of command formats for the G18 plane (ZX plane) are shown. This function can be specified in the following formats also for the G17 plane (XY plane) and G19 plane (YZ plane).
The following formats are changed as follows:

For the G17 plane: Z → X, X → Y
For the G19 plane: Z → Y, X → Z

Table 4.8 (a) Commands table

	Commands	Movement of tool
1	X2_ (Z2_), A_ ;	
2	,A1_ ; X3_ Z3_, A2_ ;	

	Commands	Movement of tool
3	X2_ Z2_, R1_ ; X3_ Z3_ ; or ,A1_, R1_ ; X3_ Z3_, A2_ ;	
4	X2_ Z2_, C1_ ; X3_ Z3_ ; or ,A1_, C1_ ; X3_ Z3_, A2_ ;	
5	X2_ Z2_ , R1_ ; X3_ Z3_ , R2_ ; X4_ Z4_ ; or ,A1_, R1_ ; X3_ Z3_, A2_, R2_ ; X4_ Z4_ ;	
6	X2_ Z2_ , C1_ ; X3_ Z3_ , C2_ ; X4_ Z4_ ; or ,A1_, R1_ ; X3_ Z3_, A2_, C2_ ; X4_ Z4_ ;	
7	X2_ Z2_ , R1_ ; X3_ Z3_ , C2_ ; X4_ Z4_ ; or ,A1_, R1_ ; X3_ Z3_, A2_, C2_ ; X4_ Z4_ ;	

	Commands	Movement of tool
8	X2_ Z2_ , C1_ ; X3_ Z3_ , R2_ ; X4_ Z4_ ; or ,A1_, C1_ ; X3_ Z3_, A2_, R2_ ; X4_ Z4_ ;	

Explanation

A program for machining along the curve shown in Fig. 4.8 (a) is as follows :

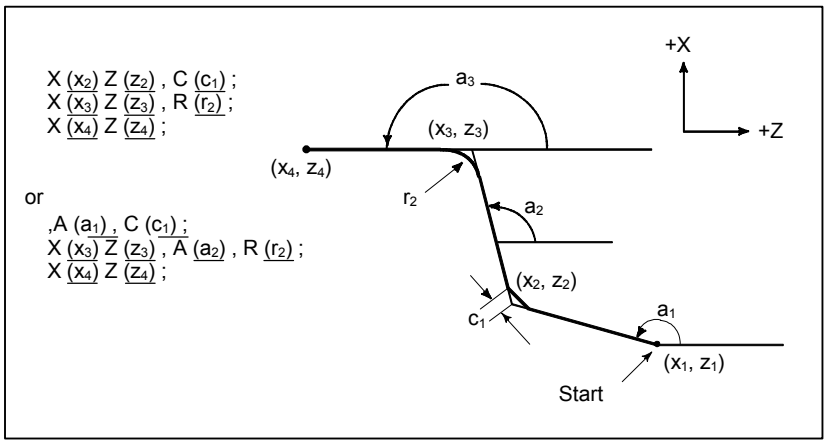


Fig. 4.8 (a) Machining Drawing (example)

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 a comma (,) as follows :

,A_
,C_
,R_

By specifying 1 to bit 4 (CCR) of parameter No.3405 on the system which does not use A or C as an axis name, the degree of a straight line or the value of chamfering or corner R can be commanded without a comma (,) as follows :

A_
C_
R_

- Command using a supplement

When bit 5 (DDP) of parameter No. 3405 is set to 1, an angle can be specified using a supplement.
There is the following relationship, assuming that the supplement is A' and the actual specified angle is A:

A = 180 – A'

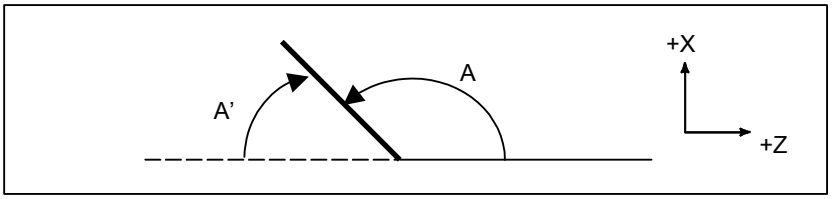


Fig. 4.8 (b) Supplement

Limitation

NOTE

1 Direct drawing dimension programming commands are valid only during memory operation.
2 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.
   (a) G codes other than G04 in group 00
   (b) G codes other than G00, G01, and G33 in group 01
   (c) G codes in group 10 (canned cycle for drilling)
   (d) G codes in group 16 (plane selection)
   (e) G22 and G23
3 Corner R cannot be inserted into a threading block.
4 When the chamfering and corner R function is enabled (bit 2 (CCR) of parameter No. 8134 is set to 1), both functions cannot be used simultaneously. When bit 0 (CRD) of parameter No. 3453 is set to 1, direct drawing dimension programming is enabled. (At this time, chamfering and corner R are disabled.)
5 When the end point of the previous block is determined in the next block according to sequential commands of direct drawing dimension programming during single block operation, the machine does not stop in the single block stop mode, but stop in the feed hold stop mode at the end point of the previous block.

NOTE

6 The angle allowance in calculating the point of intersection in the program below is ±1°.
(Because the travel distance to be obtained in this calculation is too large.)
(a) X_ ,A_ ; (If a value within 0°±1° or 180°±1° is specified for the angle instruction A, the alarm PS0057 occurs.)
(b) Z_ ,A_ ; (If a value within 90°±1° or 270°±1° is specified for the angle instruction A, the alarm PS0057 occurs.)

7 An alarm PS0058 occurs if the angle made by the 2 lines is within ±1° when calculating the point of intersection.

8 Chamfering or corner R is ignored if the angle made by the 2 lines is within ±1°.

9 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 command must be specified in block No. 3. If the angle command is not specified, alarm PS0056 is issued. If the coordinates are not specified with an absolute programming, alarm PS0312 is issued.

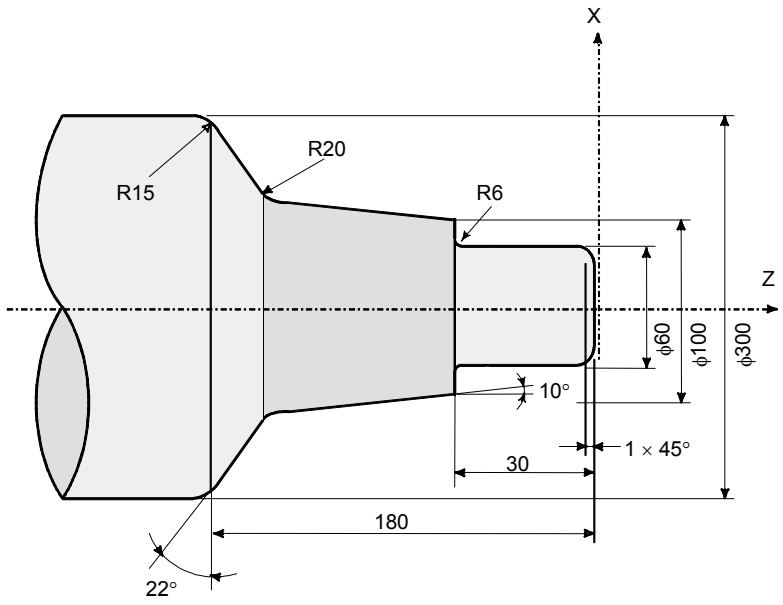
10 In the tool nose radius compensation mode, a block in which only the angle command is specified in direct drawing dimension programming is assumed to be a block with no move command. For details of compensation when sequential blocks with no move command are specified, see the explanation of tool nose radius compensation.

11 If two or more blocks with no move command are specified between sequential commands of direct drawing dimension programming, alarm PS0312 is issued.

12 When bit 4 (CCR) of parameter No. 3405 is set to 1, address A in the G76 (multiple threading cycle) block specifies the tool nose angle.
When A or C is used as an axis name, it cannot be used in the angle or chamfering command in direct drawing dimension programming. Use ,A_ or ,C_ (when bit 4 (CCR) of parameter No. 3405 is set to 0).

NOTE
13 In a multiple repetitive canned cycle, in blocks with sequence numbers between those specified at P and Q, a program using direct drawing dimension programming can be used. The block with the last sequence number specified at Q must not be an intermediate block of these specified blocks.

Example



(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 ;
                  :
                  :
```

5 COMPENSATION FUNCTION

Chapter 5, "COMPENSATION FUNCTION", consists of the following sections:

5.1 TOOL OFFSET.....162
5.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42).....168
5.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION.....185
5.4 CORNER CIRCULAR INTERPOLATION (G39).....242
5.5 AUTOMATIC TOOL OFFSET (G36, G37).....244

5.1 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).

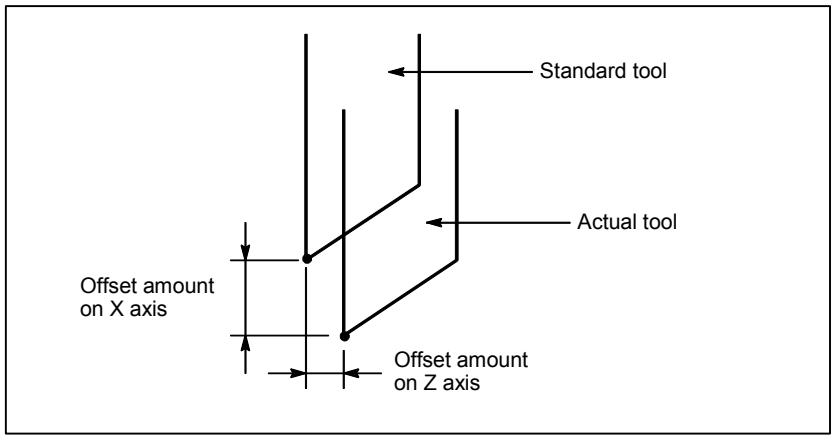


Fig. 5.1 (a) Tool offset

5.1.1 Tool Geometry Offset and Tool Wear Offset

Tool geometry offset and tool wear offset are possible to divide the tool offset to the tool geometry offset for compensating the tool shape or tool mounting position and the tool wear offset for compensating the tool nose wear. The tool geometry offset value and tool wear offset value can be set individually. When these values are not distinguished from each other, the total of the values is set as the tool offset value.

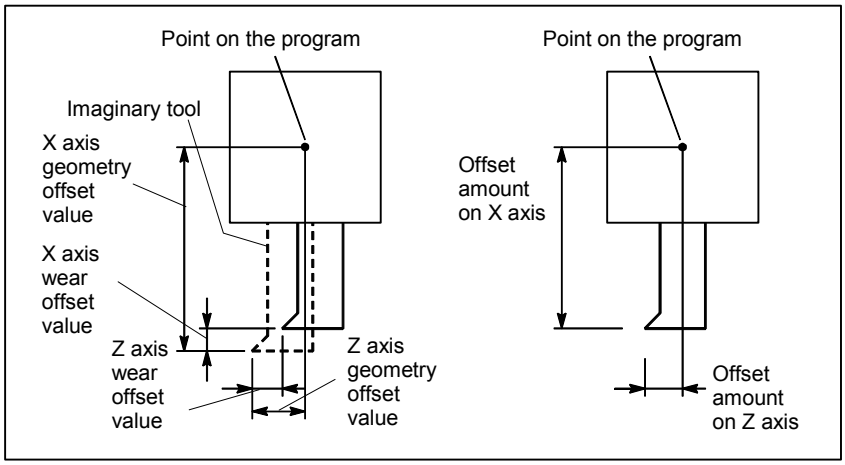


Fig. 5.1.1 (a) If tool geometry offset and tool wear offset are distinguished from each other (left) and if they are not (right)

5.1.2 T Code for Tool Offset

Format

Select a tool with a numeric value after a T code. A part of the numeric value is used as a tool offset number for specifying data such as a tool offset value. The following selections can be made according to the specification method and parameter setting:

Meaning of T code (*1)		Parameter setting for specifying of offset No. (*2)
LGN(No.5002#1)=0	LGN(No.5002#1)=1	
T x x x x x x x y xxxxxxx : Tool selection y : Tool wear and tool geometry offset	T x x x x x x x y xxxxxxx : Tool selection and tool geometry offset y : Tool wear offset	A tool wear offset number is specified using the lower one digit of a T code. When parameter No. 5028 is set to 1
T x x x x x x y y xxxxxx : Tool selection yy : Tool wear and tool geometry offset	T x x x x x x y y xxxxxx : Tool selection and tool geometry offset yy : Tool wear offset	A tool wear offset number is specified using the lower two digits of a T code. When parameter No. 5028 is set to 2
T x x x x x y y y xxxxx : Tool selection yyy : Tool wear and tool geometry offset	T x x x x x y y y xxxxx : Tool selection and tool geometry offset yyy : Tool wear offset	A tool wear offset number is specified using the lower three digits of a T code. When parameter No. 5028 is set to 3

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

*2 When parameter No. 5028 is set to 0, the number of digits of a T code used for offset number specification depends on the number of tool offsets.
Example)
When the number of tool offsets is 1 to 9: Lower one digit
When the number of tool offsets is 10 to 99: Lower two digits
When the number of tool offsets is 100 to 200: Lower three digits

5.1.3 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.

5.1.4 Offset Number

Tool offset number has two meanings. It is specifies the offset distance corresponding to the number that is selected to begin the offset function. A tool offset number of 0 indicates that the offset amount is 0 and the offset is cancelled.

5.1.5 Offset

Explanation

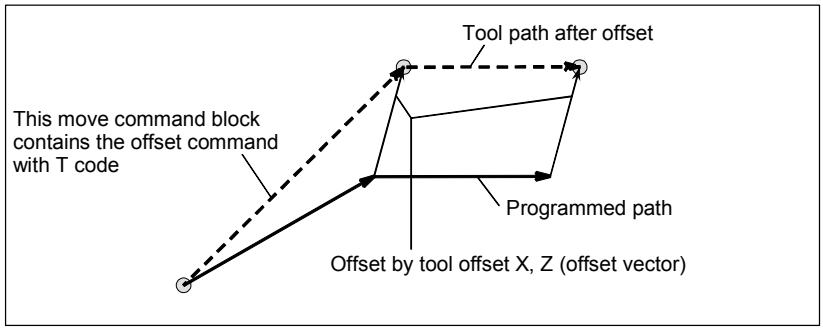
- Offset methods

There are the following two methods are available for tool geometry and wear compensation: Tool movement and coordinate shift methods. Either of these methods can be selected using bits 2 (LWT) and 4 (LGT) of parameter No. 5002. When tool geometry and wear compensation is disabled (bit 6 (NGW) of parameter No. 8136 is set to 1), however, compensation with tool movement is used unconditionally.

Bit 6 (NGW) of No.8136	Compensation element	Parameter			
		LWT=0 LGT=0	LWT=1 LGT=0	LWT=0 LGT=1	LWT=1 LGT=1
1	Wear and geometry not distinguished	Tool movement			
0	Wear compensation	Tool movement	Coordinate shift	Tool movement	Coordinate shift
	Geometry compensation	Coordinate shift	Coordinate shift	Tool movement	Tool movement

- Offset with tool movement

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

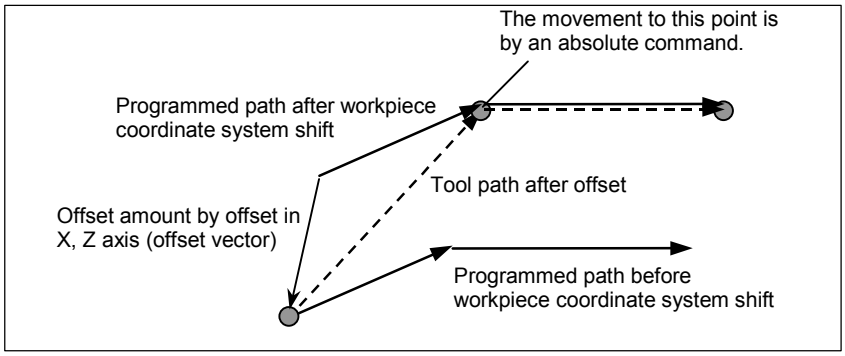


Offset operation with tool movement

NOTE
1 When G50 X_Z_T_ ; is specified, the 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 offset value corresponding to the tool offset number specified in the T code.
2 The G codes in the 00 group other than G50 must not be specified in the same block as that containing a T code. If an invalid G code is specified, alarm PS0245 is issued.

- Offset with coordinate shift

The workpiece coordinate system is shifted by the X, Y, and Z tool offset amounts. Namely, the offset amount corresponding to the number designated with the T code is added to or subtracted from the absolute coordinates.



Offset operation with coordinate shift

- Starting and canceling offset by specifying a T code

Specifying an tool offset number with a T code means to select the tool offset value corresponding to it and to start offset. Specifying 0 as a tool offset number means to cancel offset.
For offset with tool movement, whether to start or cancel the offset can be specified with parameter LWN (No. 5002#6). For compensation with coordinate shift, the offset is started and canceled when a T code is specified. For the cancellation of geometry compensation, its operation can be selected with LGC (No. 5002#5).

Offset method	LWM (No.5002#6)=0	LWM (No.5002#6)=1
Tool movement	When a T code is specified	When an axial movement is specified
Coordinate shift	When a T code is specified (Note that geometry offset can be canceled only if LGC (No. 5002#5) = 1.)	

- Canceling offset with reset

Tool offset is canceled under one of the following conditions:

<1> The power to the CNC is turned off and turned back on
<2> The reset button on the MDI unit is pressed
<3> A reset signal is input from the machine to the CNC

In cases <2> and <3> above, it is possible to select a cancel operation using parameters LVC (No. 5006#3) and TGC (No. 5003#7).

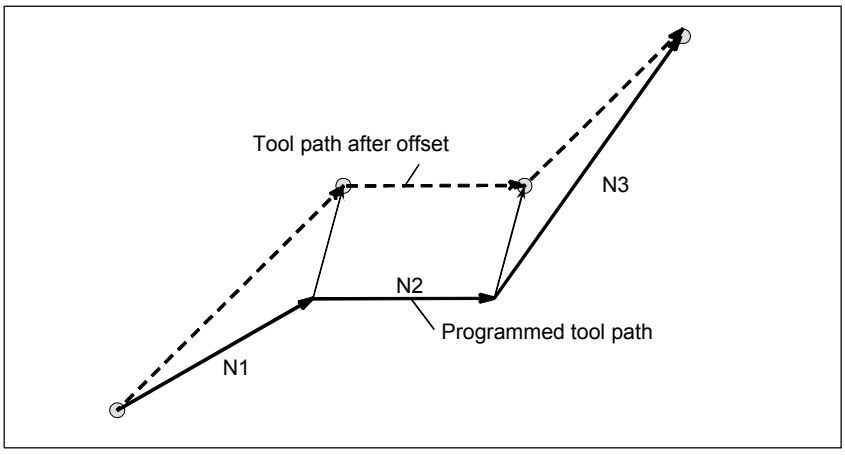
Offset method		Parameter			
		LVC=0 TGC=0	LVC=1 TGC=0	LVC=0 TGC=1	LVC=1 TGC=1
Tool movement	Wear offset	x	o (When axial movement is specified)	x	o (When axial movement is specified)
	Geometry offset				
Coordinate shift	Wear offset	x	o	x	o
	Geometry offset	x	x	o	o

o: Canceled.
x: Not canceled.

Example

```
N1 X60.0 Z50.0 T0202 ;
```
Creates the offset vector corresponding to tool offset number 02.

```
N2 Z100.0 ;
N3 X200.0 Z150.0 T0200 ;
```
Cancels the offset vector with offset number 0.



Limitation

- Helical interpolation (G02, G03)

Tool offset cannot be specified in a block in which helical interpolation is used.

- Workpiece coordinate system preset (G50.3)

Performing workpiece coordinate system preset causes tool offset with tool movement to be canceled; this does not cause tool offset with coordinate shift to be canceled.

-Machine coordinate system setting (G53), reference position return (G28), second, third, and fourth reference position return (G30), and manual reference position return

Basically, before performing these commands or operations, cancel tool offset. These operations do not cause tool offset to be canceled. The following actions take place:

	When the command or operation is specified	When the next axial movement command is specified
Tool movement	The tool offset value is temporarily canceled.	The tool offset value is reflected.
Coordinate shift	Coordinates with the tool offset value reflected are assumed.	Coordinates with the tool offset value reflected are assumed.

5.1.6 Y Axis Offset

Overview

When the Y axis, one of the basic three axes, is used with a lathe system, this function performs Y axis offset.
When tool geometry and wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is set to 0), the compensation is also enabled for the Y axis offset.

Explanation

Y axis offset results in the same operation as tool offset. For an explanation of the operation, related parameters, and the like, refer to the item “Tool Offset.”

5.1.6.1 Y axis offset (arbitrary axes)

Overview

In a lath system, Y axis offset has been usable with the basic three axes only. This function enables Y axis offset to be used with arbitrary axes other than the Y axis, which is one of the basic three axes. Specify an axis number for which to use Y axis offset for parameter No. 5043.

5.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)

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

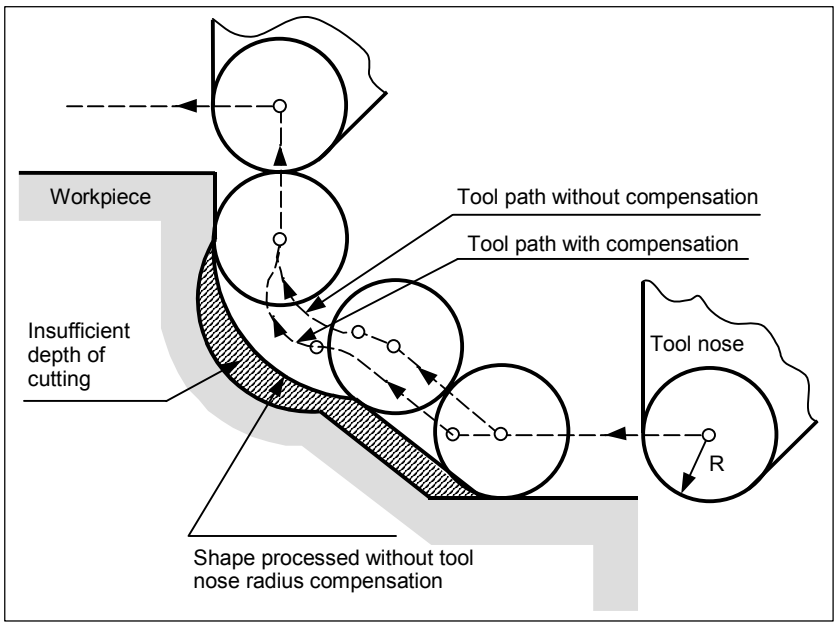


Fig 5.2 (a) Tool path of tool nose radius compensation

NOTE
To use tool nose radius compensation, set bit 7 (NCR) of parameter No. 8136 to 0.

5.2.1 Imaginary Tool Nose

The tool nose at position A in Fig. 5.2.1 (a) does not actually exist.
The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start point than the imaginary tool nose.
Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.
The position relationship when the tool is set to the start point is shown in Fig. 5.2.1 (a).

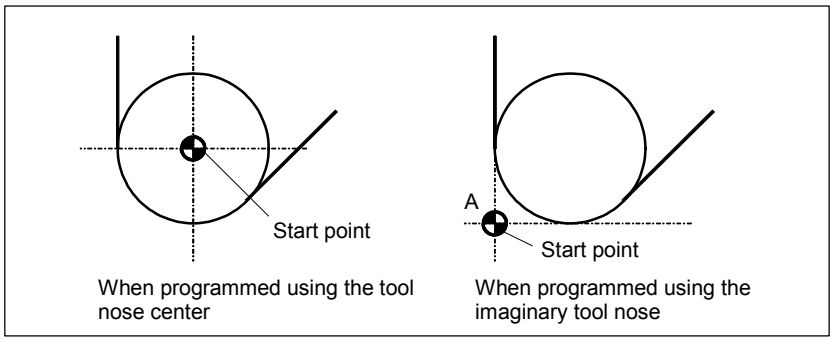


Fig. 5.2.1 (a) Tool nose radius center and imaginary tool nose

⚠ CAUTION

In a machine with reference positions, a standard position like the turret center can be placed over the start point. 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 point, 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.

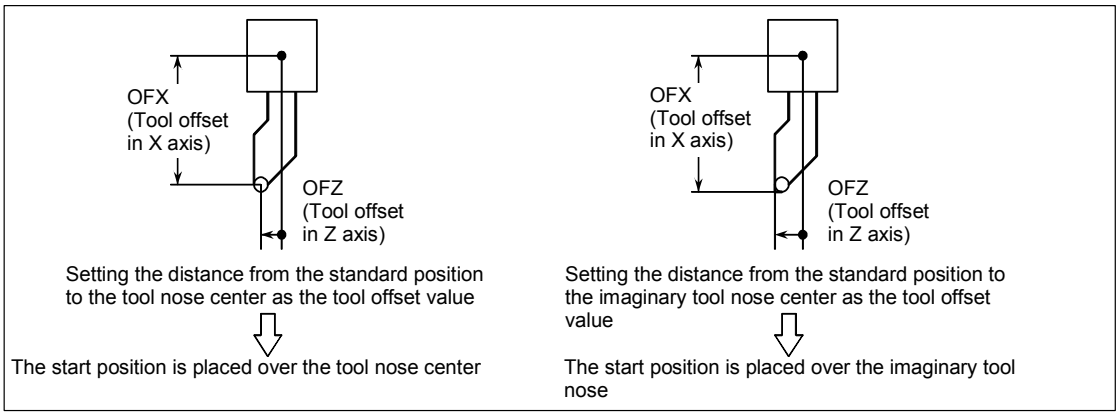


Fig. 5.2.1 (b) Tool offset value when the turret center is placed over the start point

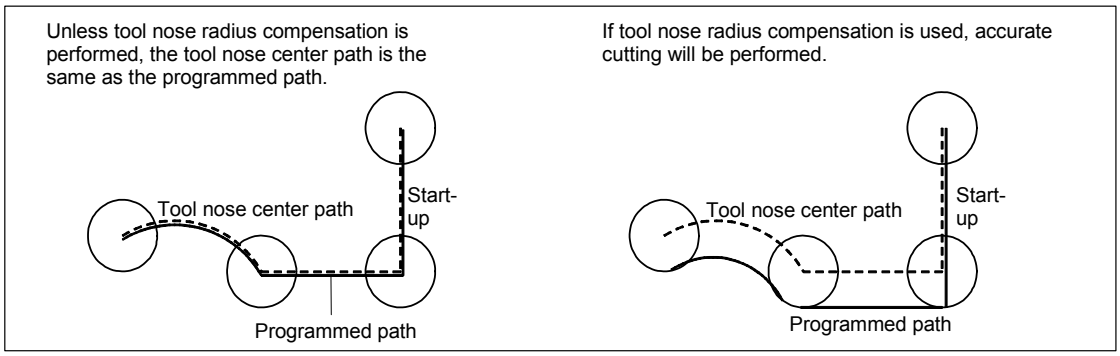


Fig. 5.2.1 (c) Tool path when programming using the tool nose center

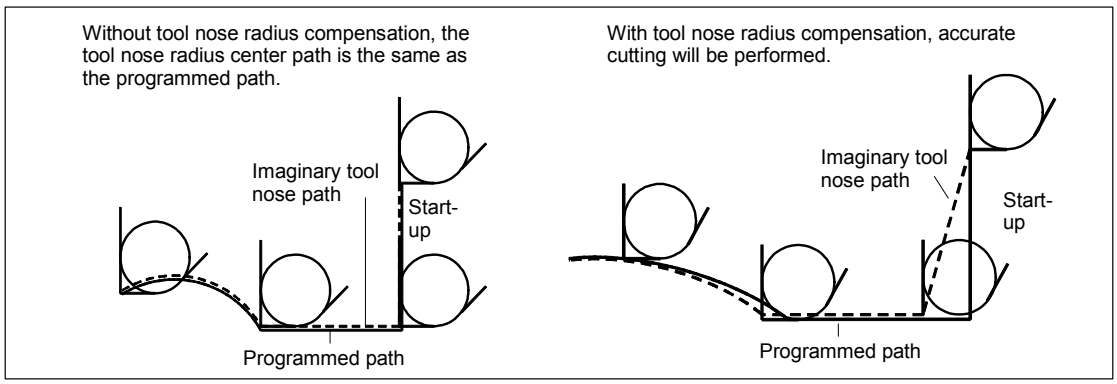


Fig. 5.2.1 (d) Tool path when programming using the imaginary tool nose

5.2.2 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. 5.2.2 (a) below together with their corresponding codes. This Fig 5.2.2 (a) illustrates the relation between the tool and the start point. The following apply when the tool geometry offset and tool wear offset option are selected.

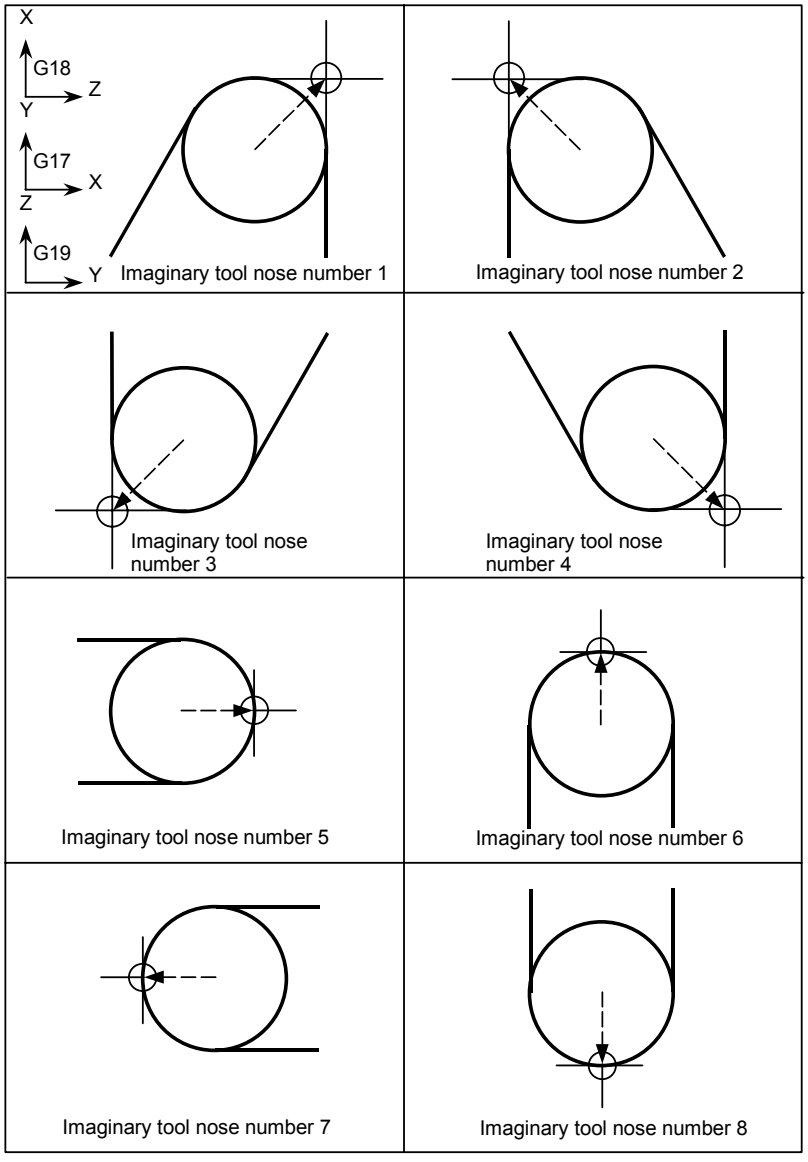
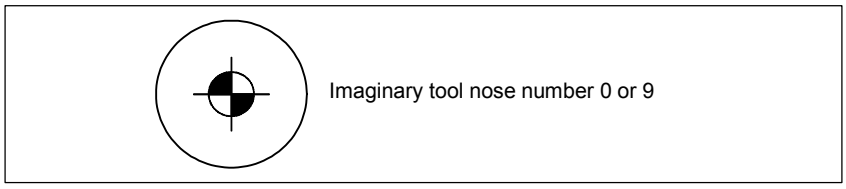


Fig. 5.2.2 (a) Direction of imaginary tool nose

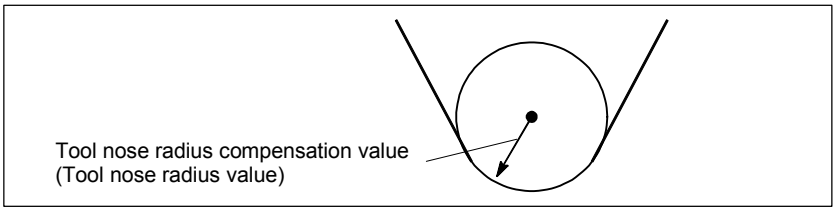
Imaginary tool nose numbers 0 and 9 are used when the tool nose center coincides with the start point. Set imaginary tool nose number to address OFT for each offset number.
Bit 7 (WNP) of parameter No. 5002 is used to determine whether the tool geometry offset number or the tool wear offset number specifies the direction of the virtual tool nose for tool nose radius compensation.



5.2.3 Offset Number and Offset Value

Explanation

- Offset number and offset value



When tool geometry and wear compensation is disabled (bit 6 (NGW) of parameter No. 8136 is set to 1), the following numbers and values are used:

Table 5.2.3 (a) Offset number and offset value (example)

Offset number Up to 999 sets	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)
001	0.040	0.020	0.200	1	0.030
002	0.060	0.030	0.250	2	0.040
003	0.050	0.015	0.120	6	0.025
004	:	:	:	:	:
005	:	:	:	:	:
:	:	:	:	:	:

When tool geometry and wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is set to 0), the following numbers and values are used:

Table 5.2.3 (b) Tool geometry offset (example)

Geometry offset number	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)
G001	10.040	50.020	0	1	70.020
G002	20.060	30.030	0	2	90.030
G003	0	0	0.200	6	0
G004	:	:	:	:	:
G005	:	:	:	:	:
:	:	:	:	:	:

Table 5.2.3 (c) Tool geometry offset (example)

Wear offset number	OFWX (X-axis wear offset amount)	OFWZ (Z-axis wear offset amount)	OFWR (Tool nose radius wear offset value)	OFT (Imaginary tool nose direction)	OFWY (Y-axis wear offset amount)
W001	0.040	0.020	0	1	0.010
W002	0.060	0.030	0	2	0.020
W003	0	0	0.200	6	0
W004	:	:	:	:	:
W005	:	:	:	:	:
:	:	:	:	:	:

- Tool nose radius compensation

When tool geometry and wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is set to 0), the total of the geometry and wear offset amounts is used as the tool nose radius compensation value during execution.
OFR=OFGR+OFWR

- Imaginary tool nose direction

The imaginary tool nose direction is common to geometry and wear offsets.

- Command of offset value

A offset number is specified with the same T code as that used for tool offset.

NOTE
When the geometry offset number is made common to the tool selection by the parameter LGN (No.5002#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.
Example) T0102
$OFR=OFGR_{01}+OFWR_{02}$
$OFT=OFT_{01}$
By setting parameter WNP (No. 5002#7) appropriately, the imaginary tool nose direction specified with the wear offset number can be made valid.

- Setting range of offset value

The range of values that can be set as a compensation value is either of the following, depending on the bits 1 (OFC) and 0 (OFA) of parameter No. 5042).

Valid compensation range (metric input)

OFC	OFA	Range
0	1	±9999.99mm
0	0	±9999.999mm
1	0	±9999.9999mm

Valid compensation range (inch input)

OFC	OFA	Range
0	1	±999.999inch
0	0	±999.9999inch
1	0	±999.99999inch

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

5.2.4 Workpiece 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.

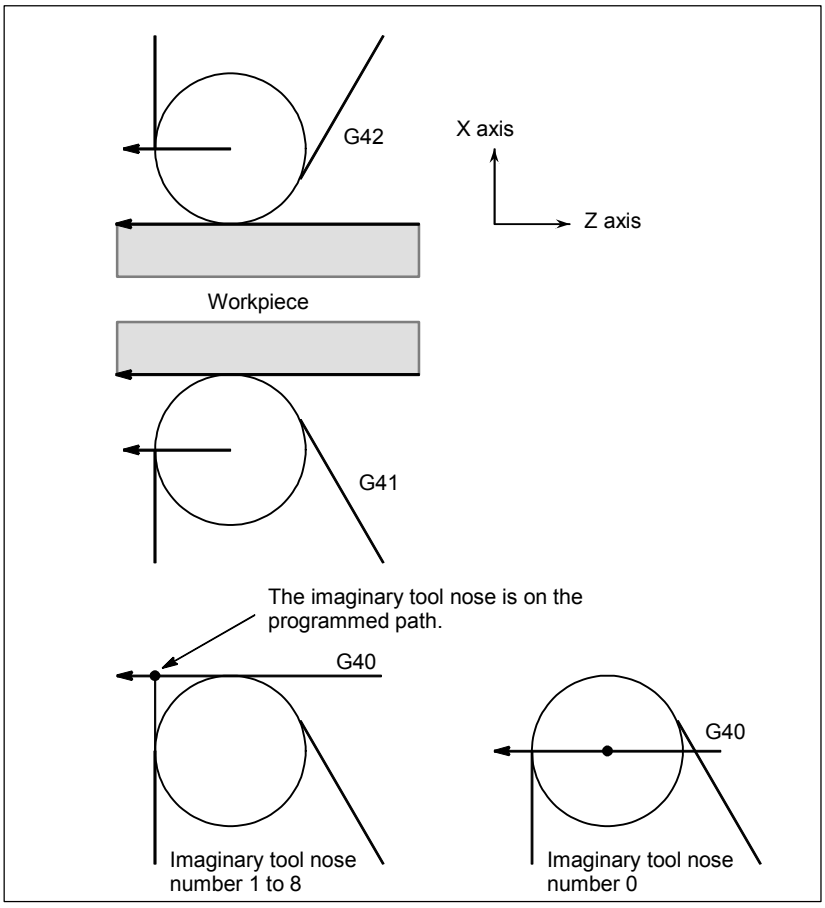


Fig. 5.2.4 (a) Workpiece position

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

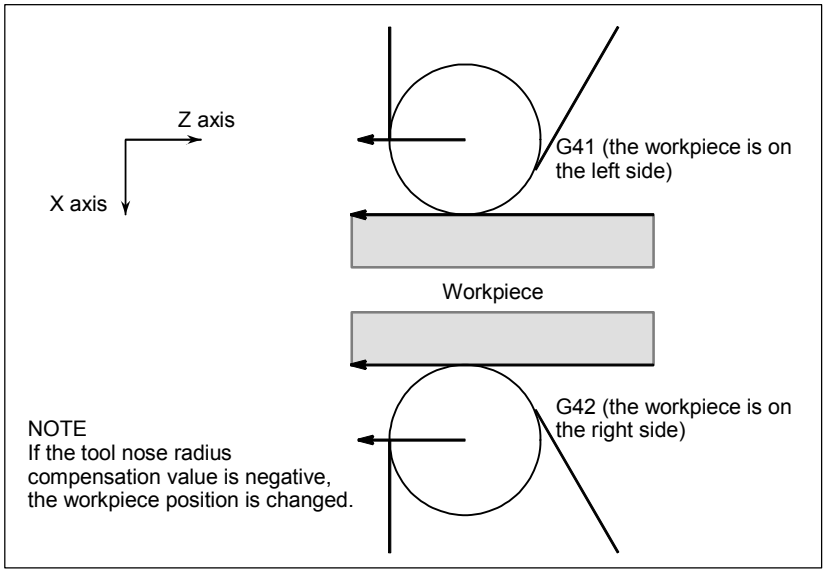


Fig. 5.2.4 (b) When the workpiece position is changed

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.

> ⚠ **CAUTION**
> If the sign of the compensation value is changed from plus to minus and vice versa, the offset vector of tool nose radius compensation is reversed, but the direction of the imaginary tool tip does not change. For a use in which the imaginary tool tip is adjusted to the starting point, therefore, do not change the sign of the compensation value for the assumed program.

Explanation

- Tool movement when the workpiece position does not change

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

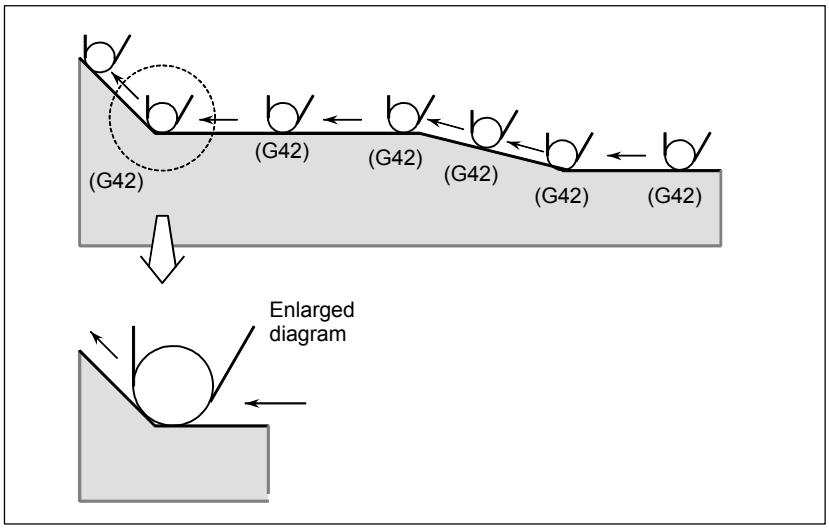


Fig. 5.2.4 (c) Tool movement when the workpiece position does not change

- 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.

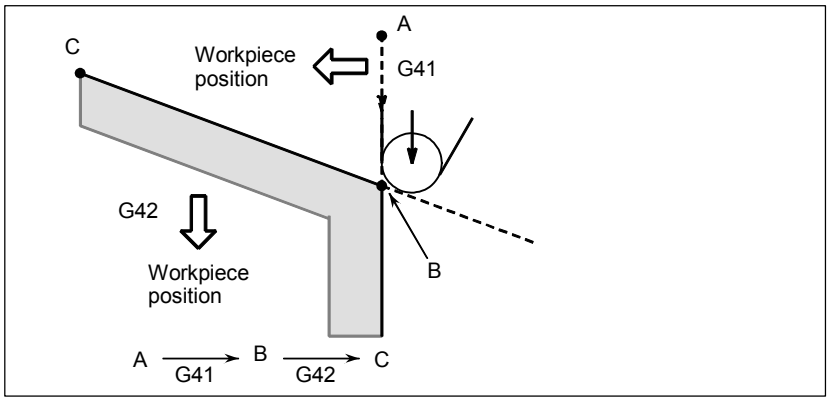


Fig. 5.2.4 (d) Tool movement when the workpiece position changes

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 position must not be changed in the block next to the start-up block. In the above example, if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

- 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 point.

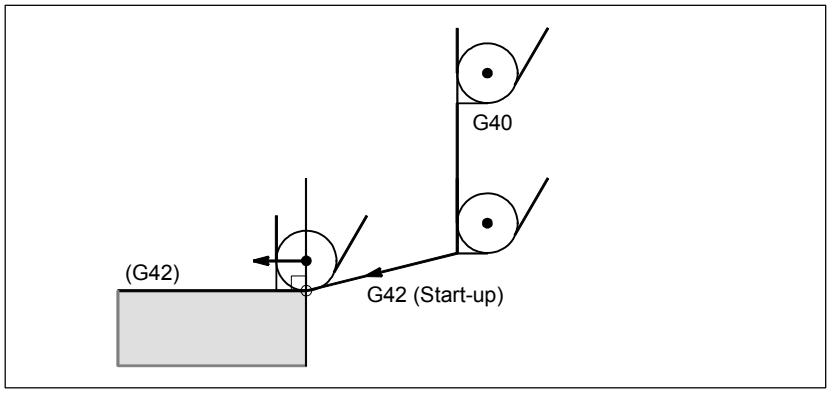


Fig. 5.2.4 (e) Start-up

- 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.

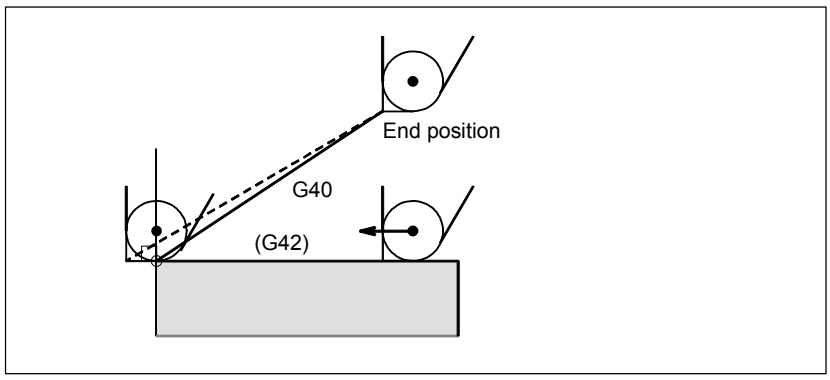


Fig. 5.2.4 (f) Offset cancel

- Changing the compensation value

In general, the compensation value is to be changed when the tool is changed in offset cancel mode. If the compensation value is changed in offset mode, however, the vector at the end point of the block is calculated using the compensation value specified in that same block. The same applies if the imaginary tool nose direction and the tool offset value are changed.

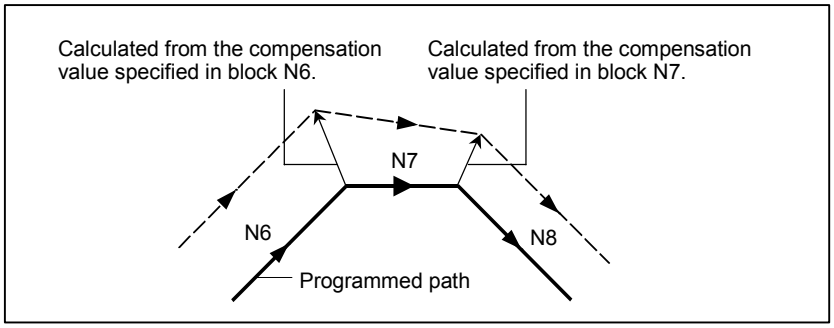


Fig. 5.2.4 (g) Changing the compensation value

- Specification of G41/G42 in G41/G42 mode

When a G41 or G42 code 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.

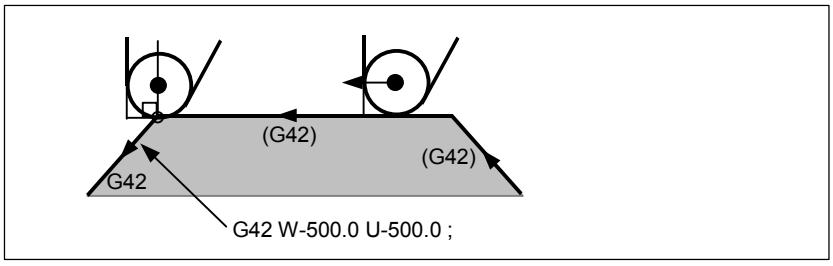


Fig. 5.2.4 (h) Specification of G41/G42 in G41/G42 mode

In the block that first changes from G40 to 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 (offset cancel) 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) canceling 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 _ ;

where I and K are the direction of the workpiece in the next block, which is specified in incremental mode.

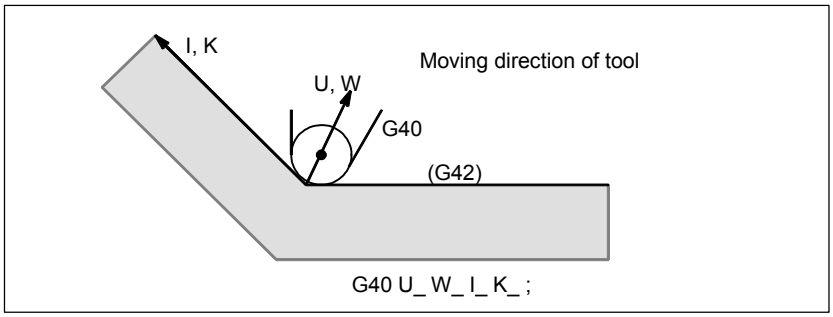


Fig. 5.2.4 (i) If I and K are specified in the same block as G40

Thus, this prevents the tool from overcutting, as shown in Fig. 5.2.4 (j).

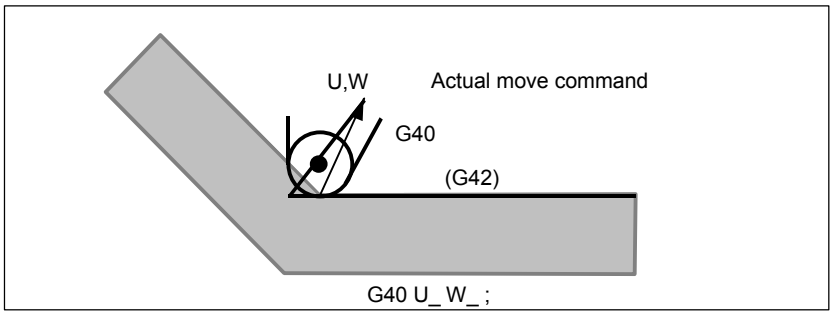


Fig. 5.2.4 (j) Case in which overcutting occurs in the same block as G40

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

Specify I_K_; in the same block as G40. If it is specified in the same block as G02 or G03, it is assumed to be the center of the arc.

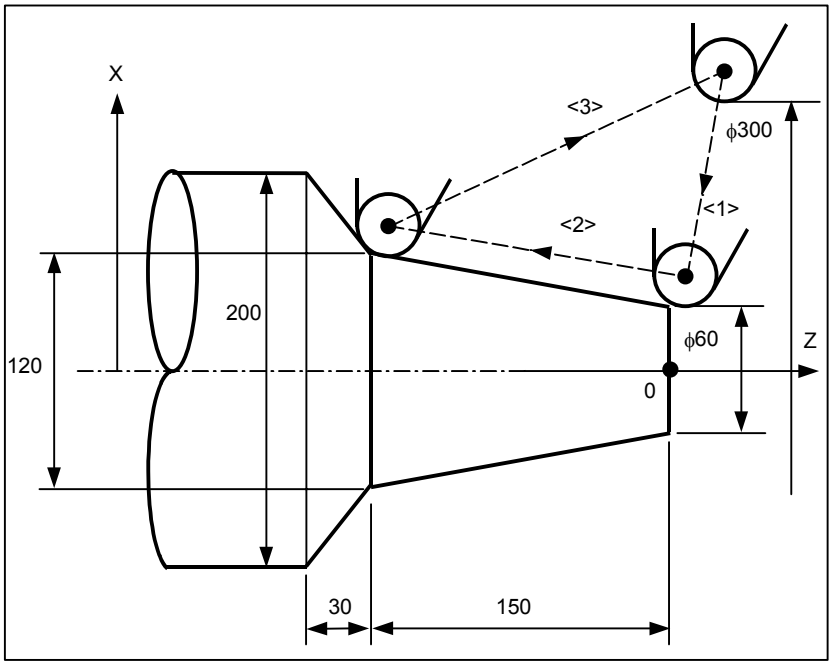
G40 X_ Z_ I_ K_ ;	Tool nose radius compensation
G02 X_ Z_ I_ K_ ;	Circular interpolation

If I and/or K is specified with G40 in the offset cancel mode, the I and/or K is ignored. The numeral is followed I and K should always be specified as radius values.

G40 G01 X_ Z_ ;

G40 G01 X_ Z_ I_ K_ ; Offset cancel mode (I and K are ineffective.)

Example



(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 ;

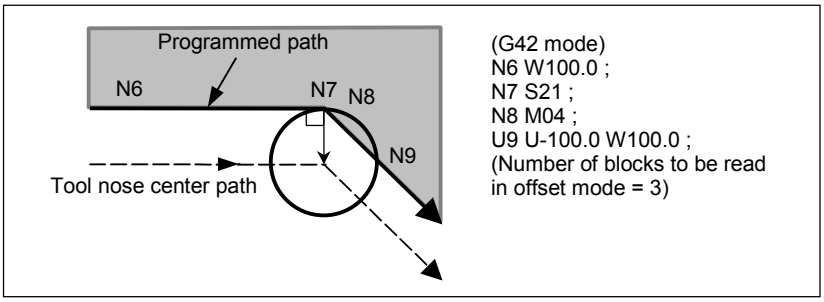
5.2.5 Notes on Tool Nose Radius Compensation

Explanation

- Blocks without a move command that are specified in offset mode

```
<1> M05 ;                              M code output
<2> S210 ;                             S code output
<3> G04 X10.0 ;                        Dwell
<4> G22 X100000 ;                      Machining area setting
<5> G01 U0 ;                           Feed distance of zero
<6> G98 ;                              G code only
<7> G10 P01 X10.0 Z20.0 R0.5 Q2 ;      Offset change
```

If the number of such blocks consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)), the tool arrives at the position vertical to this block at the end point of the previous block.
If the feed distance is 0 (<5>), this applies even if only one block is specified.

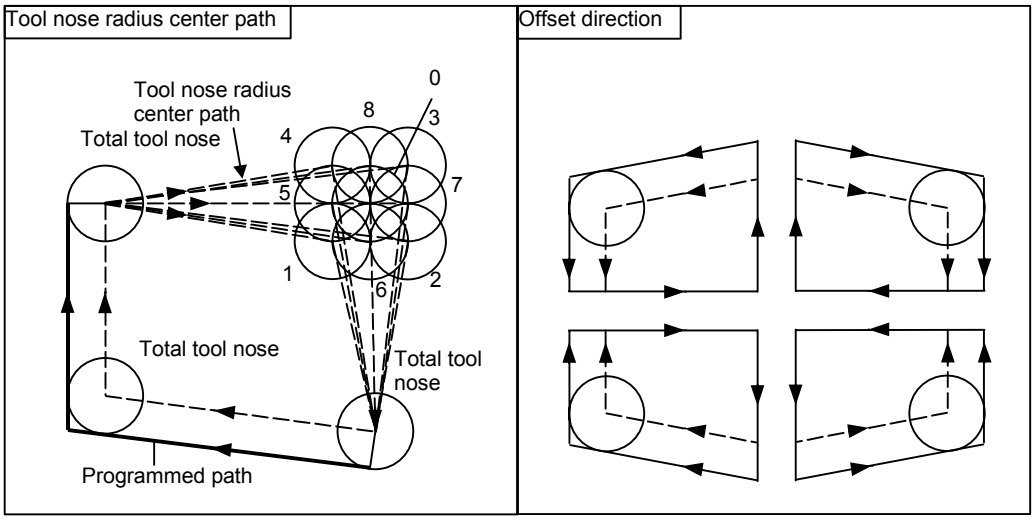


Overcutting may, therefore, occur in the above figure.

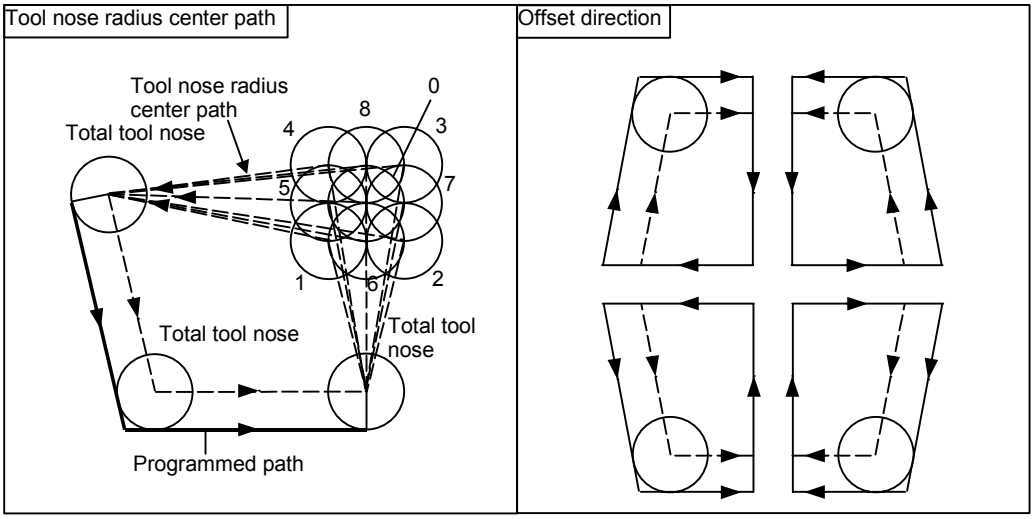
- Tool nose radius compensation with G90 or G94

The tool nose center path and the offset direction are as shown below if tool nose radius compensation is applied. At the cycle start point, the offset vector disappears, and offset starts up with tool movement from the cycle start point. In addition, during a return to the cycle start point, the offset vector disappears temporarily, and offset is applied again with the next move command. The offset direction is determined by the cutting pattern, regardless of G41 or G42.

- Outer/inner turning cycle (G90)



- End cutting cycle (G94)



- Difference from Series 0*i*-C

NOTE

The offset direction is the same as that of Series 0*i*-C, but the tool nose radius center path is different.

- For this CNC
  The operation is the same as that performed if the canned cycle operation is replaced with G00 or G01, start-up is performed in the first block for movement from the start point, and offset cancel is performed in the last block for returning to the start point.
- For Series 0*i*-C
  The operation with the block for movement from the start point and the last block for returning to the start point differs from that of this CNC. For details, refer to the Series 0*i*-C Operator's Manual.

- Tool nose radius compensation with G71 to G73

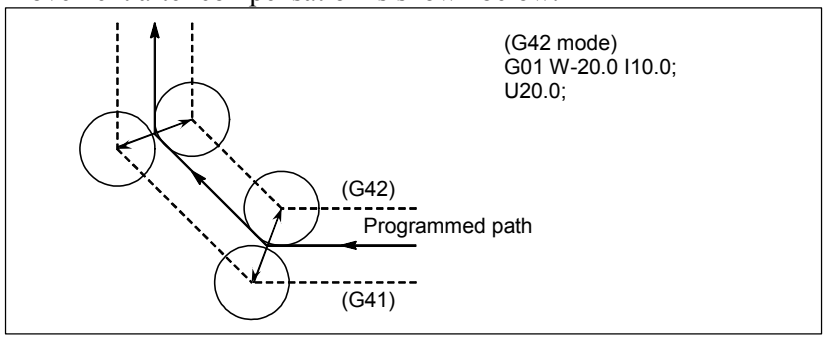
Tool nose radius compensation performed with G71 (outer surface rough cutting cycle or traverse grinding cycle), G72 (end rough cutting cycle or traverse direct constant-size grinding cycle), and G73 (closed loop cutting cycle or oscillation direct constant-size grinding cycle), see the explanations of the respective cycles.

- Tool nose radius compensation with G74 to G76 and G92

With G74 (end cutting off cycle), G75 (outer/inner surface cutting off cycle), G76 (multiple threading cycle), and G92 (threading cycle), tool nose radius compensation cannot be applied.

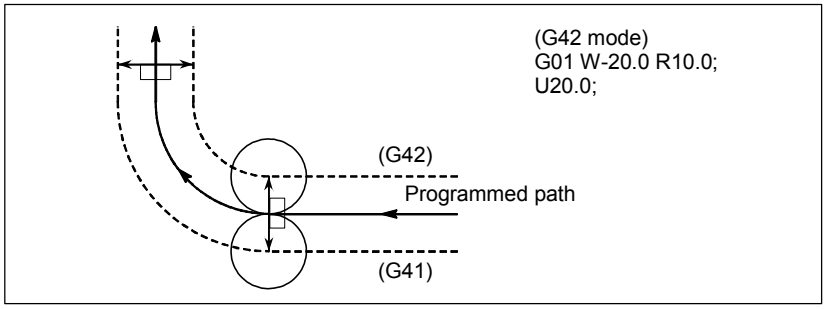
- Tool nose radius compensation when chamfering is performed

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 for MDI operation.

Tool nose radius compensation is valid for MDI operation.

NOTE
For Series 0*i*-C, tool nose radius compensation is invalid for MDI operation.

5.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION

5.3.1 Overview

This subsection details tool movement in tool nose radius compensation.

- Tool nose radius center offset vector

The tool nose radius center offset vector is a two dimensional vector equal to the offset value specified in a T code, and the vector 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 radius) 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

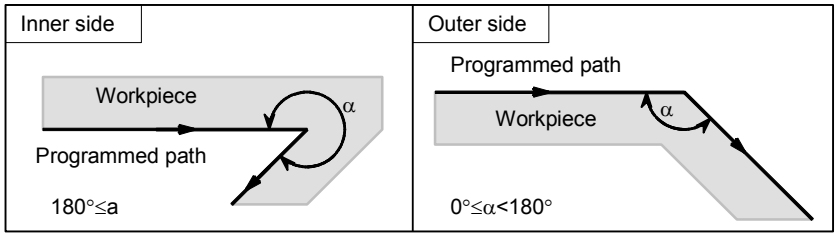
G40, G41 or G42 is used to delete or generate vectors.
These codes are used together with G00, G01, G02, or G32 to specify a mode for tool motion (Offsetting).

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

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

- Inner side and outer side

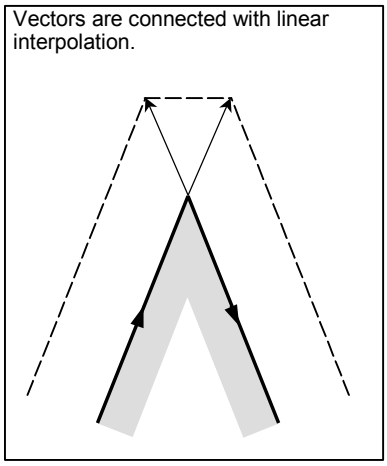
When an angle of intersection of the tool paths specified with move commands for two blocks on the workpiece side is over 180°, it is referred to as "inner side." When the angle is between 0° and 180°, it is referred to as "outer side."



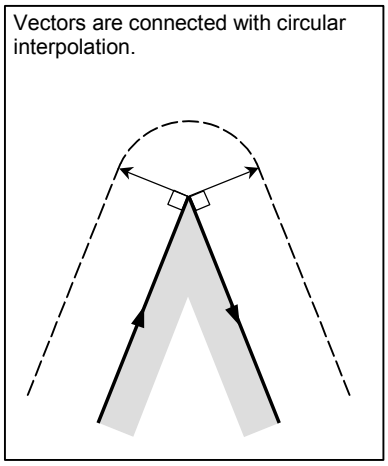
- Outer corner connection method

If the tool moves around an outer corner in tool nose radius compensation mode, it is possible to specify whether to connect compensation vectors with linear interpolation or with circular interpolation, using parameter CCC (No. 19607#2).

<1> Linear connection type [Parameter CCC (No.19607#2) = 0]



<2> Circular connection type [Parameter CCC (No.19607#2) = 1]



- Cancel mode

The tool nose radius compensation enters the cancel mode under the following conditions. (The system may not enter the cancel mode depending on the machine tool.)

<1> Immediately after the power is turned on

<2> After the RESET key on the MDI panel is pushed

<3> After a program is forced to end by executing M02 or M30

<4> After the tool nose radius compensation cancel command (G40) is exercised

In the cancel mode, the magnitude of a compensation vector is 0 at all times and the path of the virtual tool nose matches the programmed path. A program must end in cancel mode. If it ends in the tool nose radius compensation mode, the tool cannot be positioned at the end point, and the tool stops at a location the compensation vector length away from the end point.

NOTE

The operation performed when a reset operation is performed during tool nose radius compensation varies according to the setting of bit 6 (CLR) of parameter No. 3402.

- When CLR=0
  The reset state is set. The modal information of G41/G42 in group 07 is preserved. To perform tool nose radius compensation, however, an offset number (T code) needs to be specified again.
- When CLR=1
  The cleared state is set. The modal information of G40 in group 07 is preserved. To perform tool nose radius compensation, G41/G42 and an offset number (T code) need to be specified.

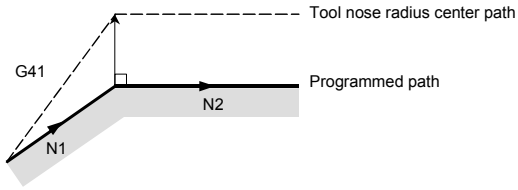
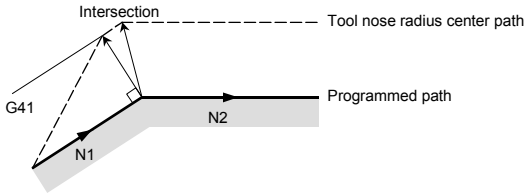
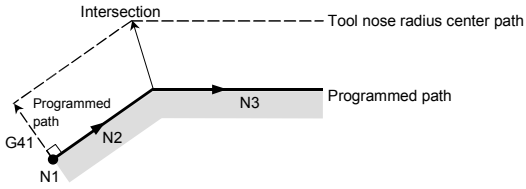
- Start-up

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

<1> G41 or G42 is contained in the block, or has been specified to place the CNC in the offset mode.
<2> 0 < compensation number of tool nose radius compensation ≤ maximum compensation number
<3> Positioning (G00) or linear interpolation (G01) mode
<4> A compensation plane axis command with a travel distance of 0 (except start-up type C) is specified.

If start-up is specified in circular interpolation (G02, G03) mode, alarm PS0034 will occur.
As a start-up operation, one of the three types A, B, and C can be selected by setting bits 0 (SUP) and 1 (SUV) of parameter No. 5003 appropriately. The operation to be performed if the tool moves around an inner side is of single type only.

Table 5.3.1 (a) Start-up/cancel operation

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector is output, which is vertical to the block subsequent to the start-up block and the block preceding the cancel block. 
0	1	Type B	A compensation vector is output, which is vertical to the start-up block and the cancel block. An intersection vector is also output. 
1	0 1	Type C	When the start-up block and the cancel block are blocks without tool movement, the tool moves by the tool nose radius compensation value in the direction vertical to the block subsequent to the start-up block and the block preceding the cancel block.  For a block with tool movement, the tool follows the SUP setting: If it is 0, type A is assumed and if 1, type B is assumed.

- Reading input commands in tool nose radius compensation mode

In tool nose radius compensation mode, input commands are read from usually three blocks and up to eight blocks depending on the setting of parameter (No. 19625) to perform intersection calculation or an interference check, described later, regardless of whether the blocks are with or without tool movement, until a cancel command is received.
To perform intersection calculation, it is necessary to read at least two blocks with tool movement. To perform an interference check, it is necessary to read at least three blocks with tool movement.

As the setting of parameter (No. 19625), that is, the number of blocks to read, increases, it is possible to predict overcutting (interference) for up to more subsequent commands. Increases in blocks to read and analyze, however, cause reading and analysis to take more time.

- Meaning of symbols

The following symbols are used in subsequent figures:

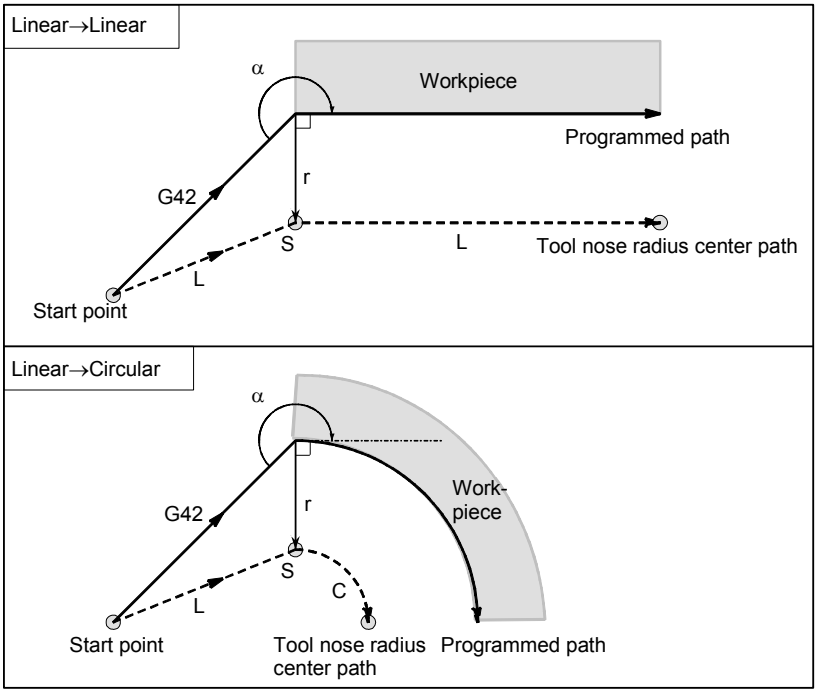
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves 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 the center of the tool nose radius.

5.3.2 Tool Movement in Start-up

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

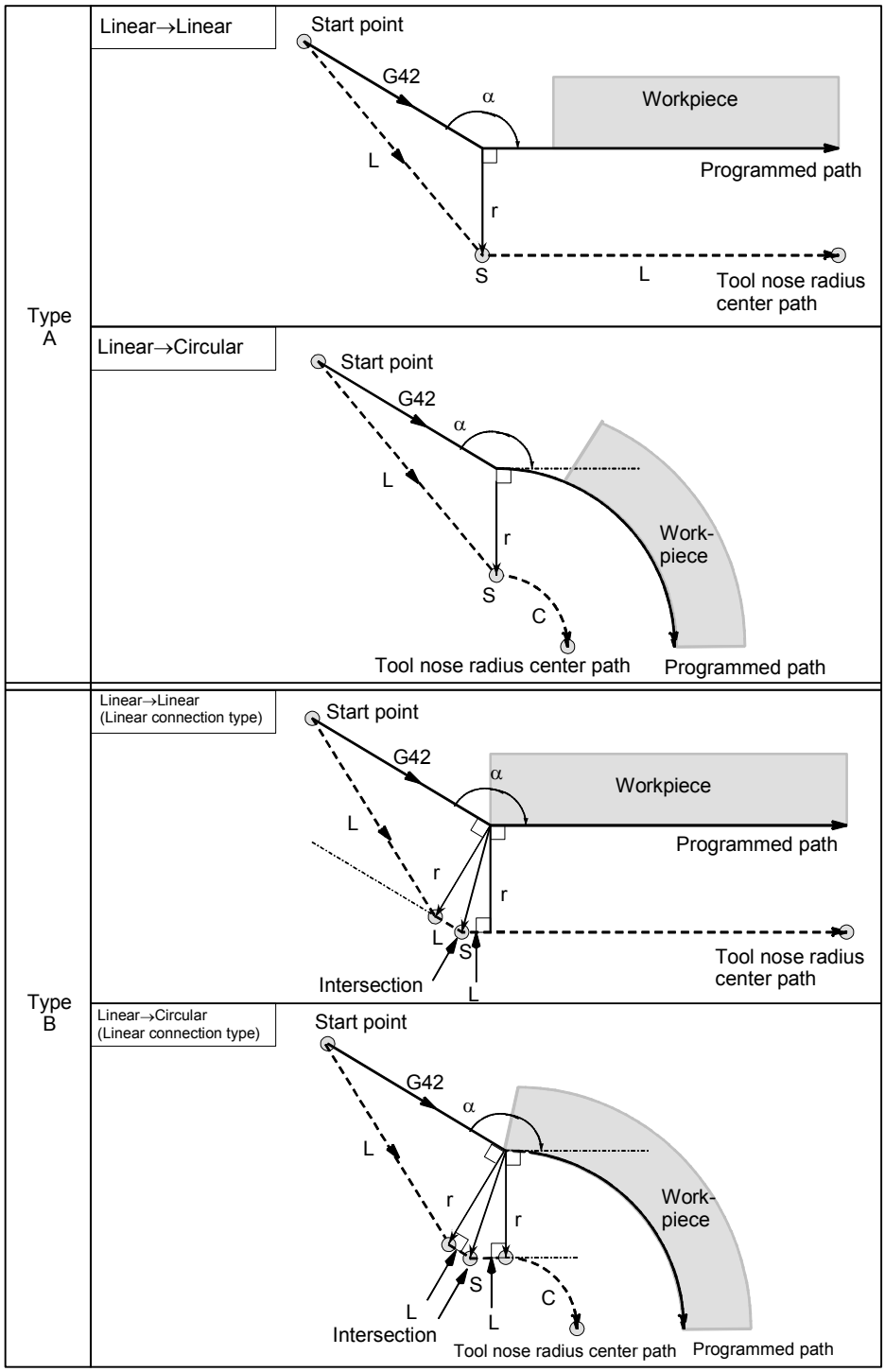
Explanation

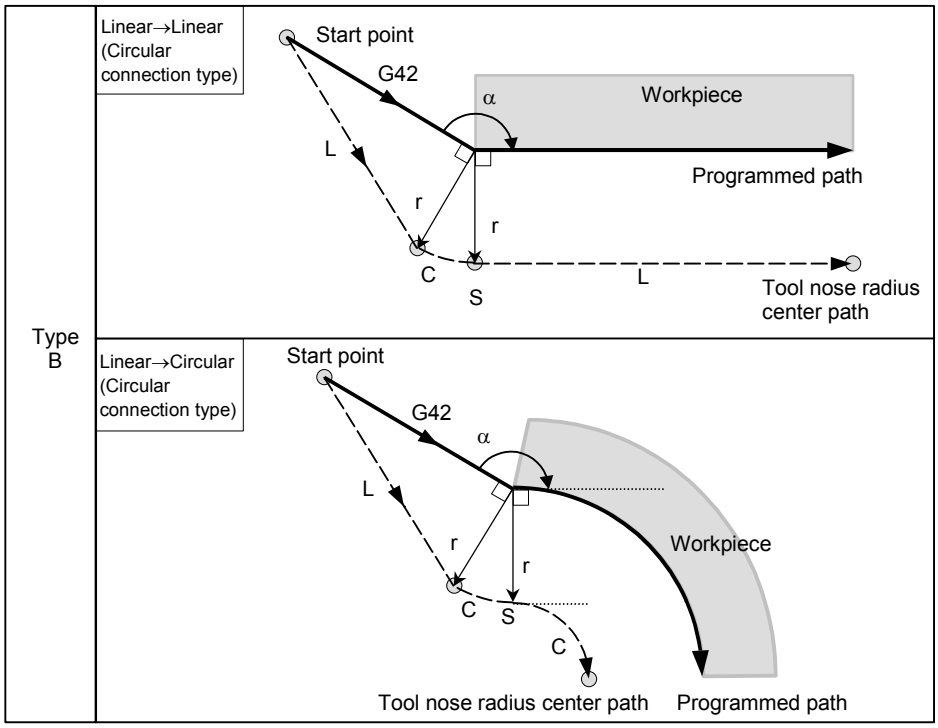
- Tool movement around an inner side of a corner (180°≤ α)



- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an obtuse angle (90°≤ α<180°)

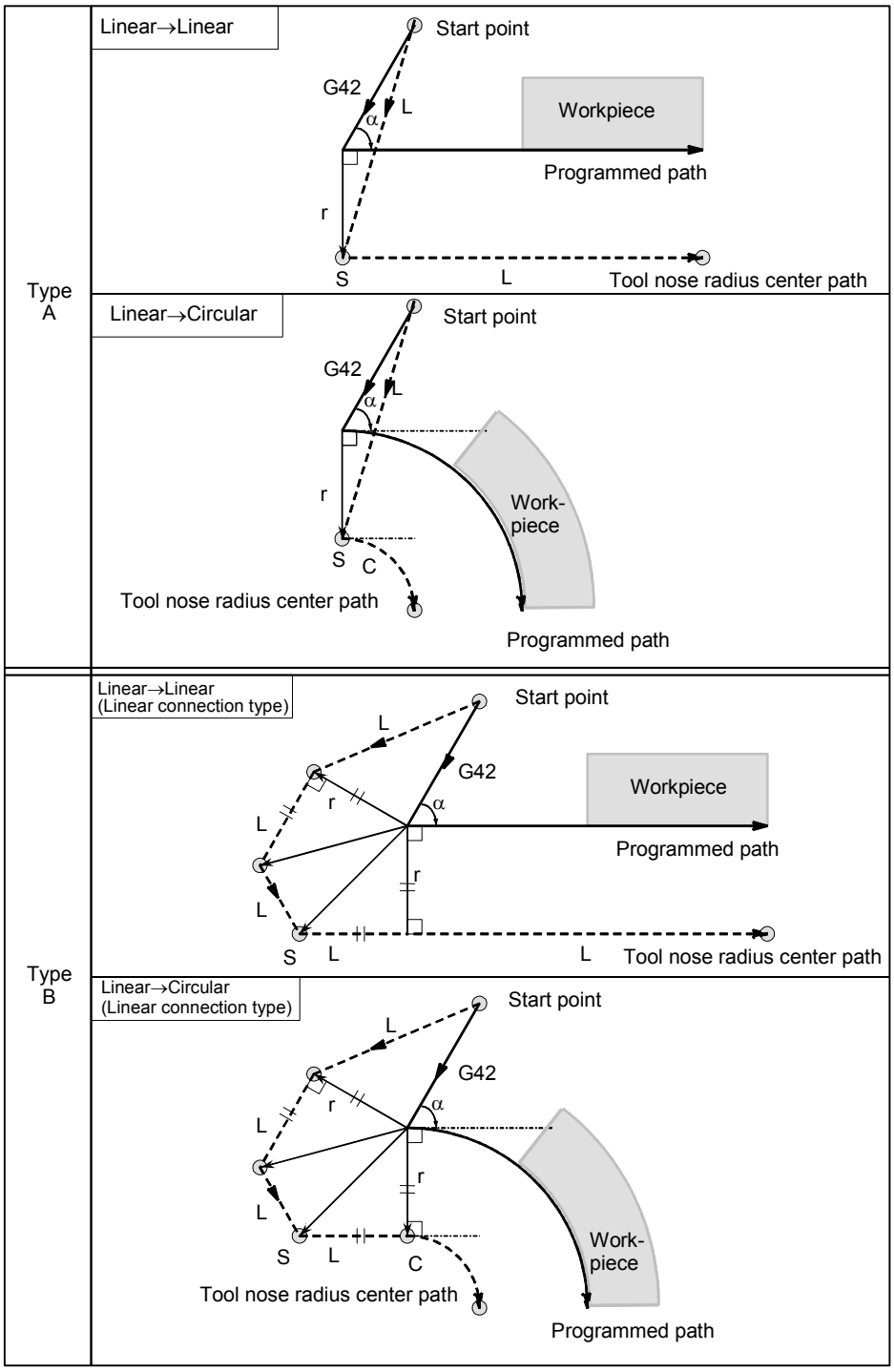
Tool path in start-up has two types A and B, and they are selected by parameter SUP (No.5003#0).

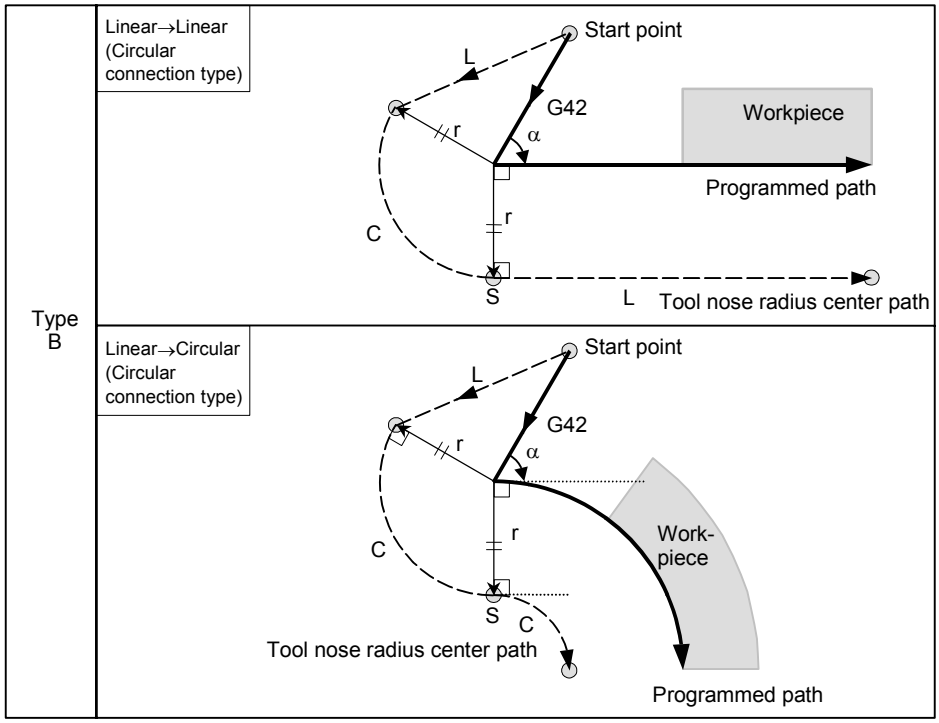



Type B
Linear→Linear (Circular connection type)
Start point
G42
α
Workpiece
L
Programmed path
r
r
C
S
L
Tool nose radius center path
Linear→Circular (Circular connection type)
Start point
G42
α
L
r
r
Workpiece
C
S
C
Tool nose radius center path
Programmed path

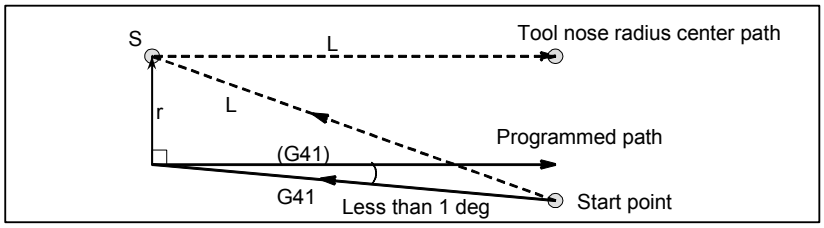
- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle (α<90°)

Tool path in start-up has two types A and B, and they are selected by parameter SUP (No.5003#0).



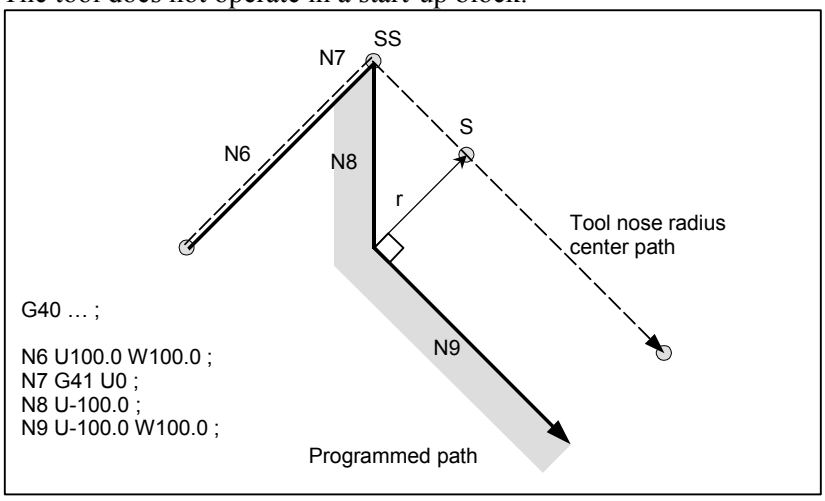


- Tool movement around the outside linear → linear at an acute angle less than 1 degree (α<1°)

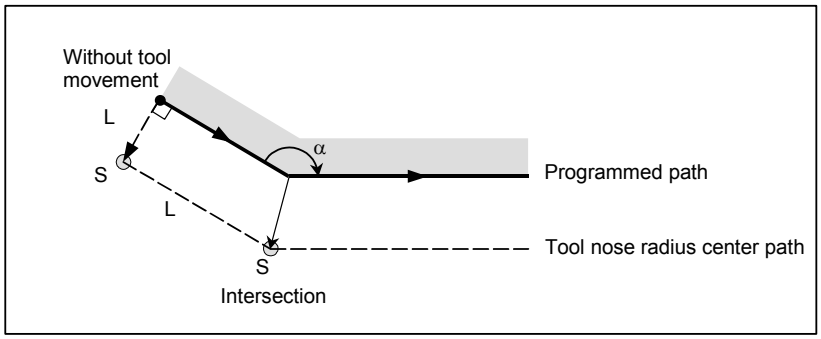


- A block without tool movement specified at start-up

For type A and B
If the command is specified at start-up, the offset vector is not created.
The tool does not operate in a start-up block.



For type C
The tool shifts by the compensation value in the direction vertical to the block with tool movement subsequent to the start-up block.



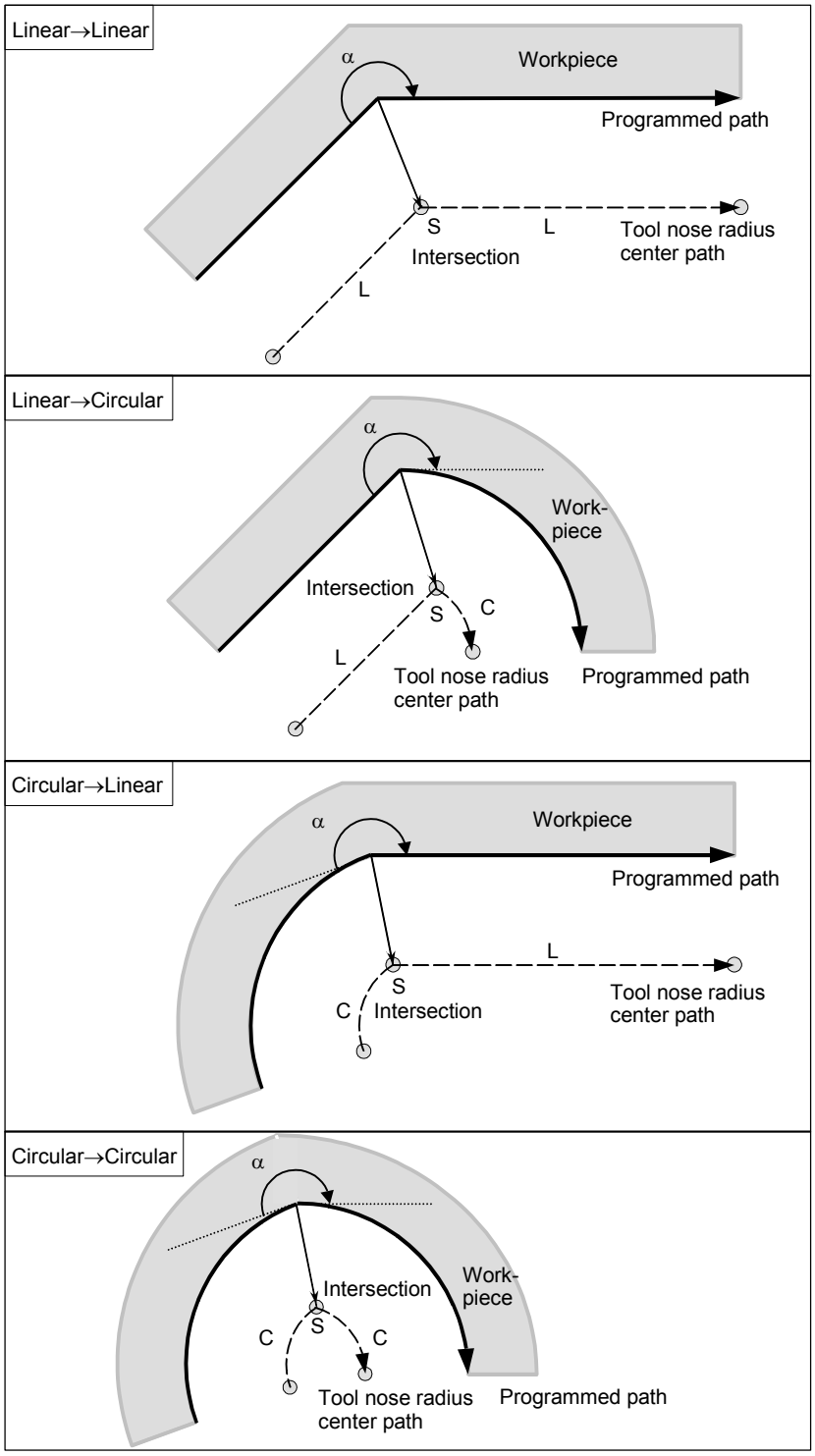
5.3.3 Tool Movement in Offset Mode

In offset mode, compensation is performed even for positioning commands, not to speak of linear and circular interpolations. To perform intersection calculation, it is necessary to read at least two blocks with tool movement. If, therefore, two or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as auxiliary function independent commands and dwell, are specified in succession, excessive or insufficient cutting may occur because intersection calculation fails. Assuming the number of blocks to read in offset mode, which is determined by parameter (No. 19625), to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which intersection calculation is possible is $(N - 2) \geq M$. For example, if the maximum number of blocks to read in offset mode is 5, intersection calculation is possible even if up to three blocks without tool movement are specified.

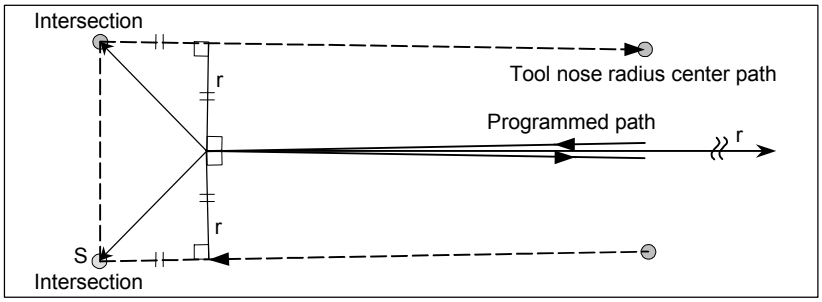
NOTE
The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the interference check.

If a G or M code in which buffering is suppressed is specified, no subsequent commands can be read before that block is executed, regardless of the setting of parameter (No. 19625). Excessive or insufficient cutting may, therefore, occur because of an intersection calculation failure.

- Tool movement around the inside of a corner (180°≤ α)

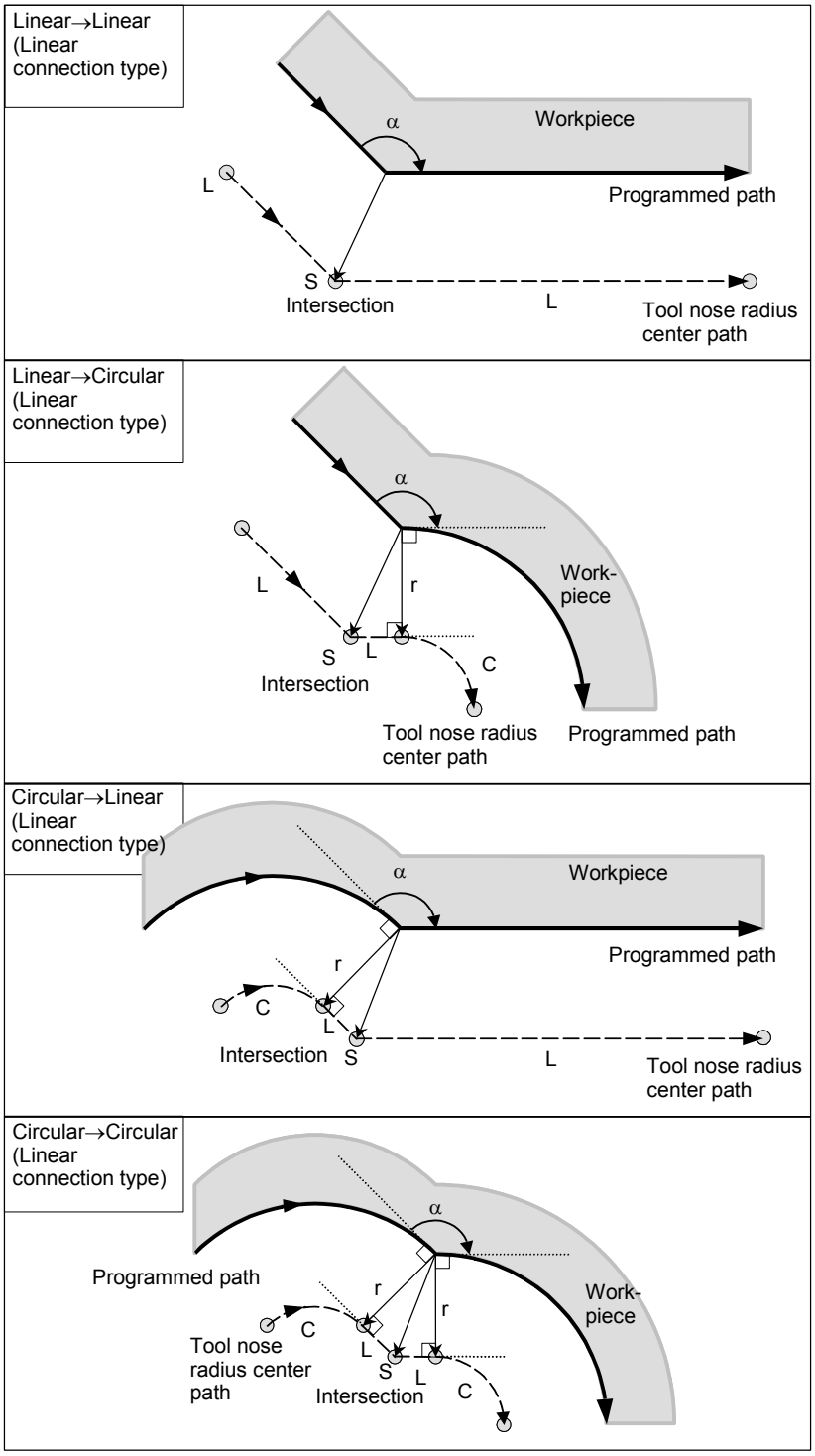


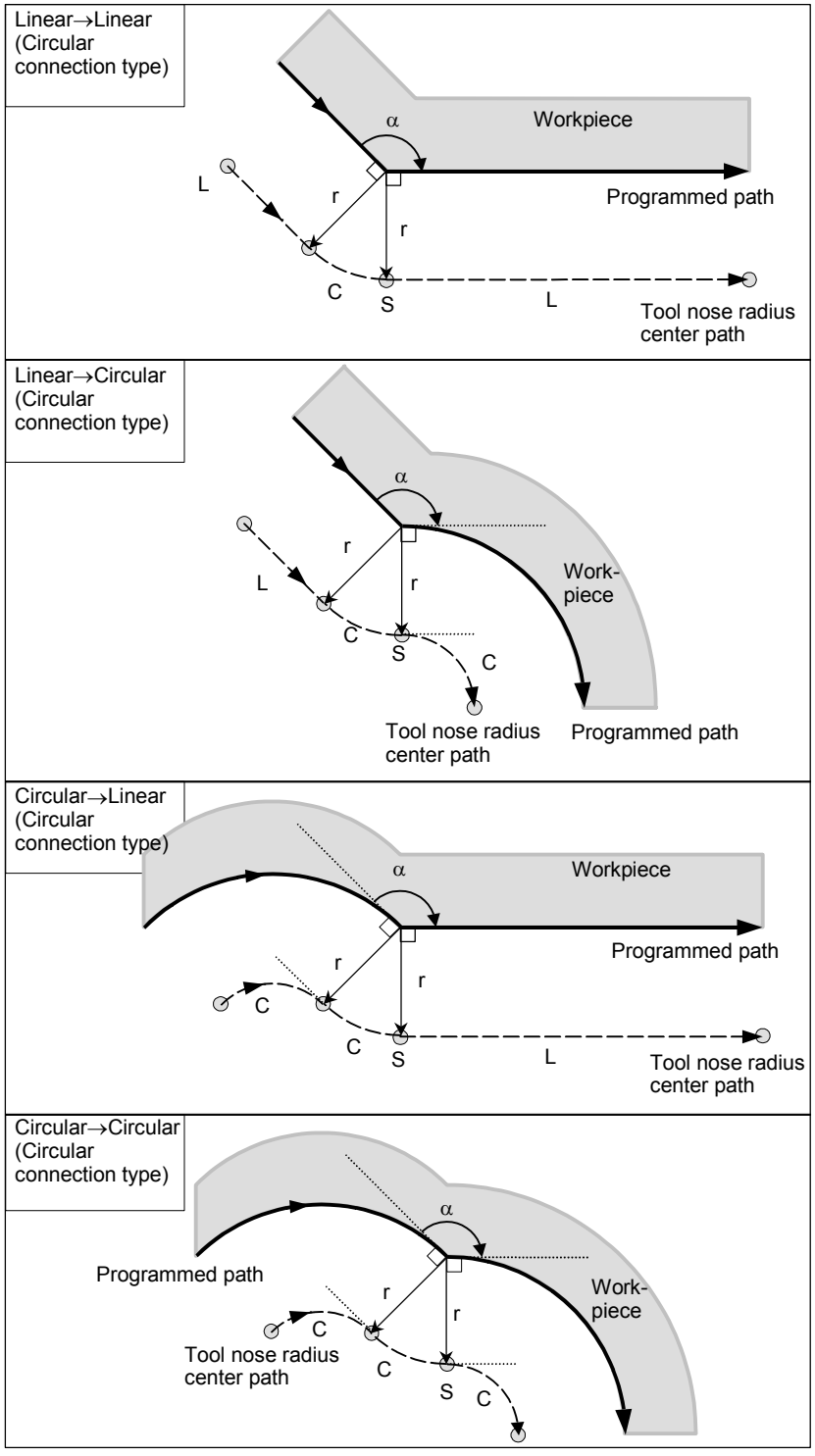
- Tool movement around the inside (α<1°) with an abnormally long vector, linear → linear



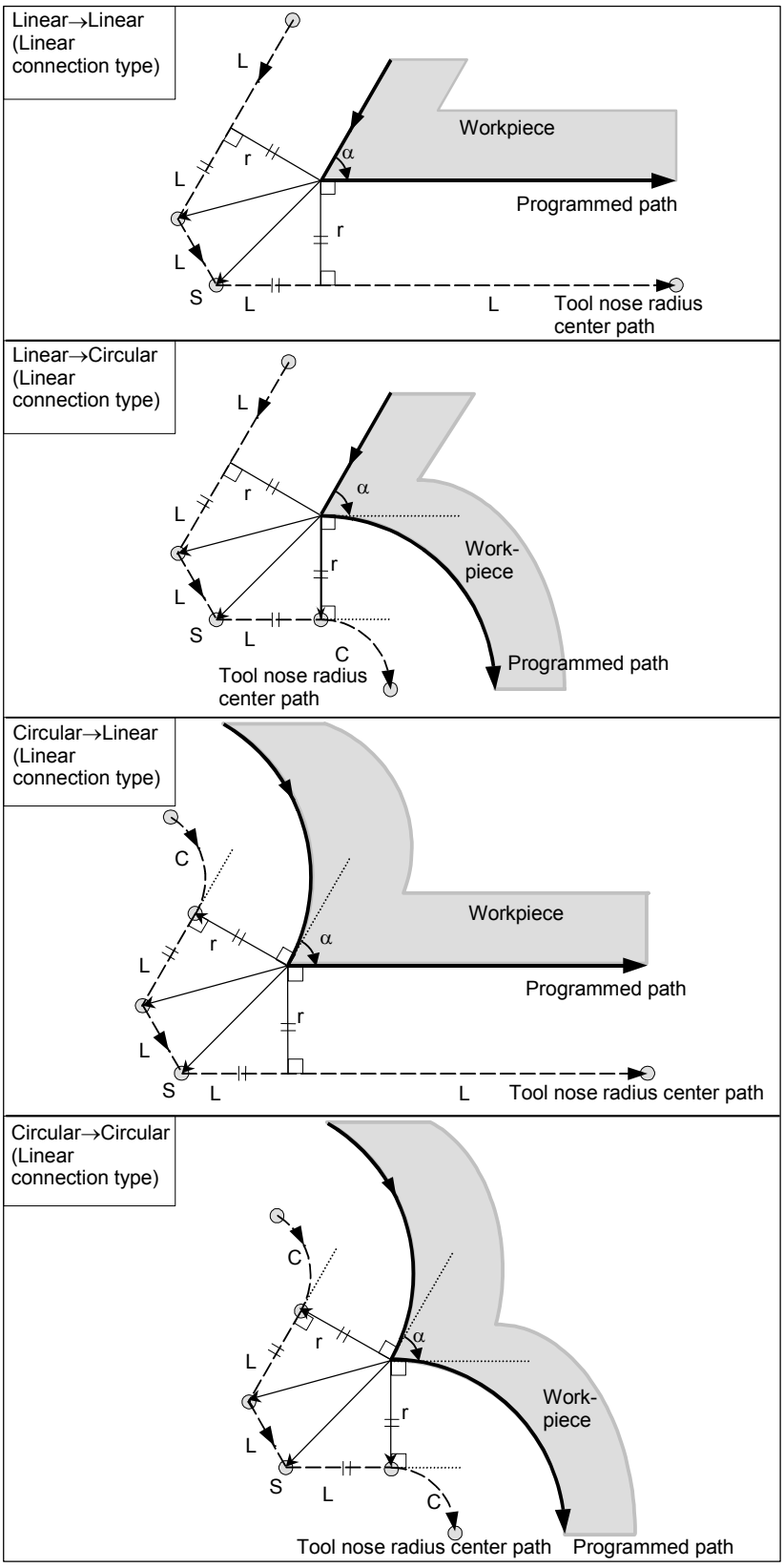
Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

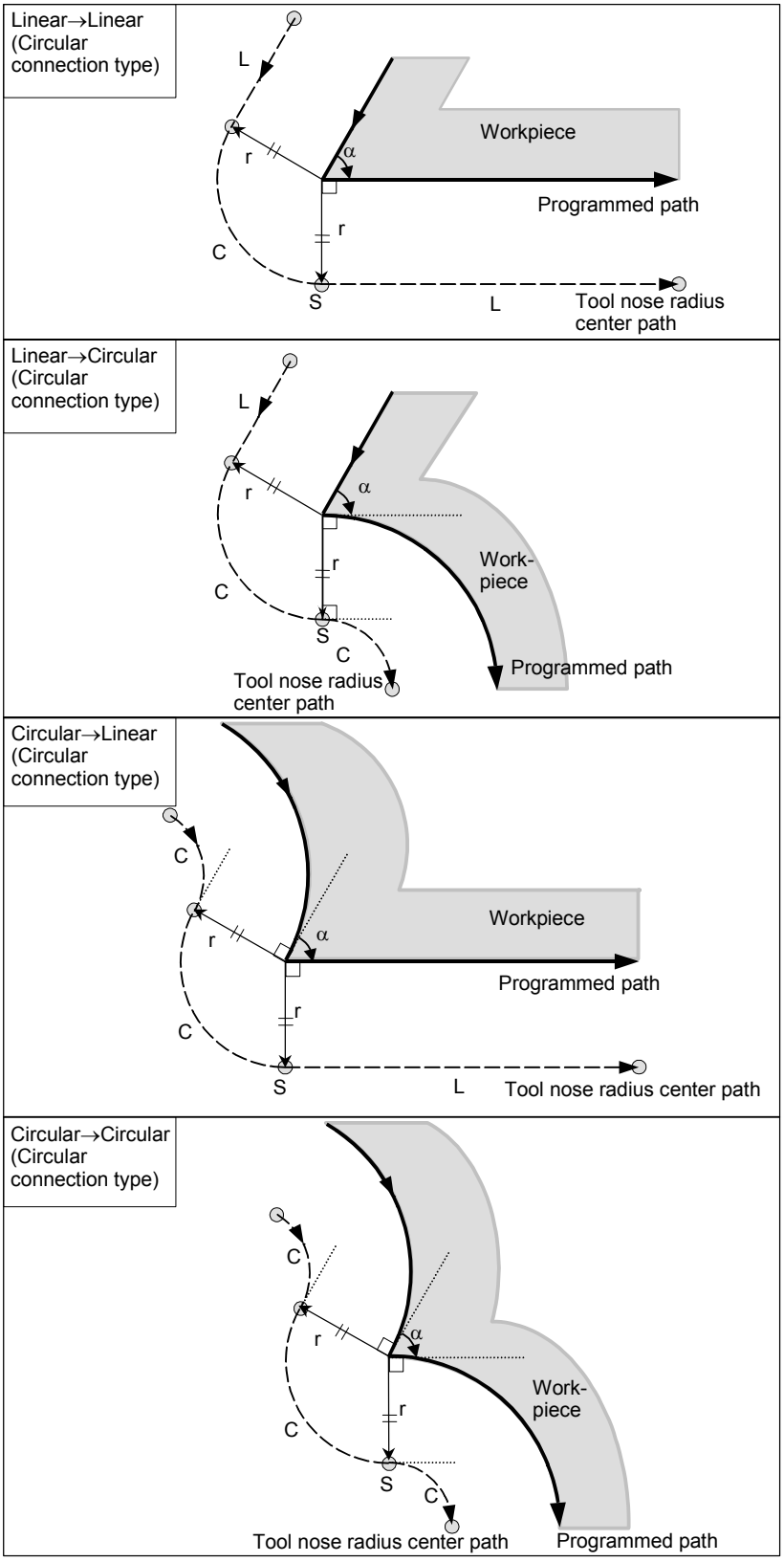
- Tool movement around the outside corner at an obtuse angle (90°≤α<180°)




Linear→Linear
(Circular
connection type)
α
Workpiece
L
r
r
Programmed path
C
S
L
Tool nose radius
center path
Linear→Circular
(Circular
connection type)
α
r
L
r
Work-
piece
C
S
C
Tool nose radius
center path
Programmed path
Circular→Linear
(Circular
connection type)
α
Workpiece
Programmed path
r
r
C
C
S
L
Tool nose radius
center path
Circular→Circular
(Circular
connection type)
α
Programmed path
r
Work-
piece
C
r
Tool nose radius
center path
C
S
C

- Tool movement around the outside corner at an acute angle (α<90°)

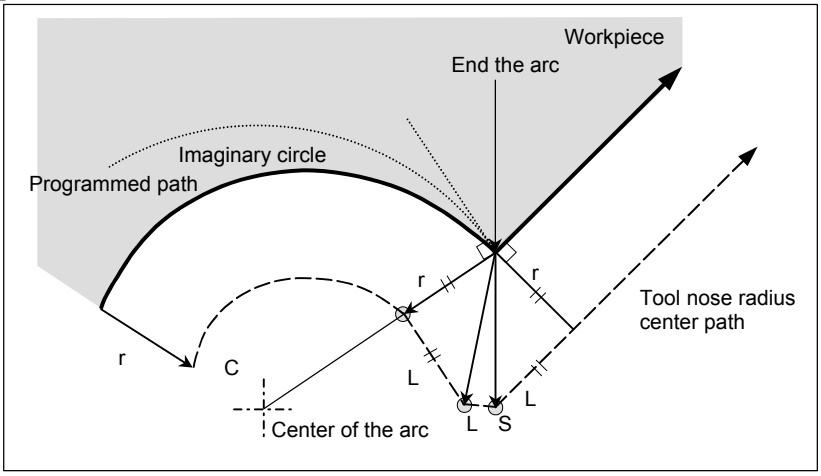



Linear→Linear
(Circular
connection type)
L
r
α
Workpiece
Programmed path
r
C
S
L
Tool nose radius
center path
Linear→Circular
(Circular
connection type)
L
r
α
r
Work-
piece
C
S
C
Tool nose radius
center path
Programmed path
Circular→Linear
(Circular
connection type)
C
r
α
Workpiece
Programmed path
r
C
S
L
Tool nose radius center path
Circular→Circular
(Circular
connection type)
C
r
α
Work-
piece
r
C
S
C
Tool nose radius center path
Programmed path

- When it is exceptional

End position for the arc is not on the arc

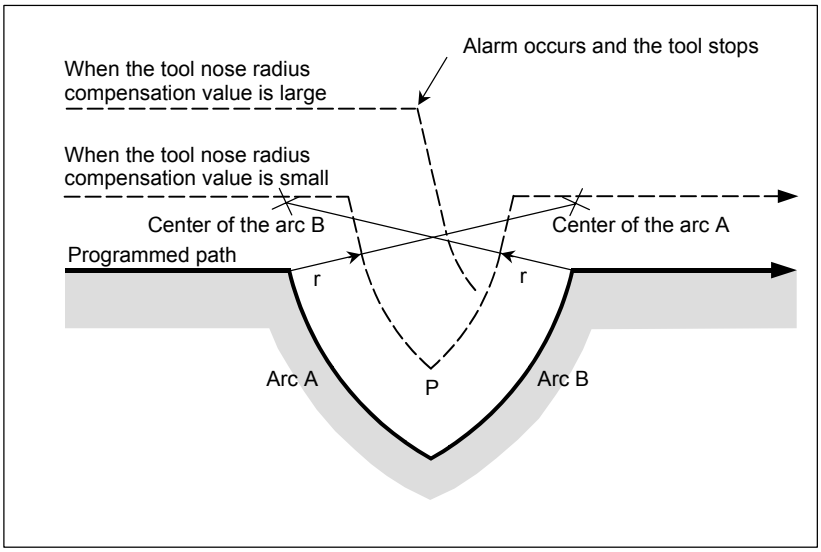
If the end of a line leading to an arc is not on the arc as illustrated below, the system assumes that the tool nose radius compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system creates a vector and carries out compensation. The same description applies to tool movement between two circular paths.



There is no inner intersection

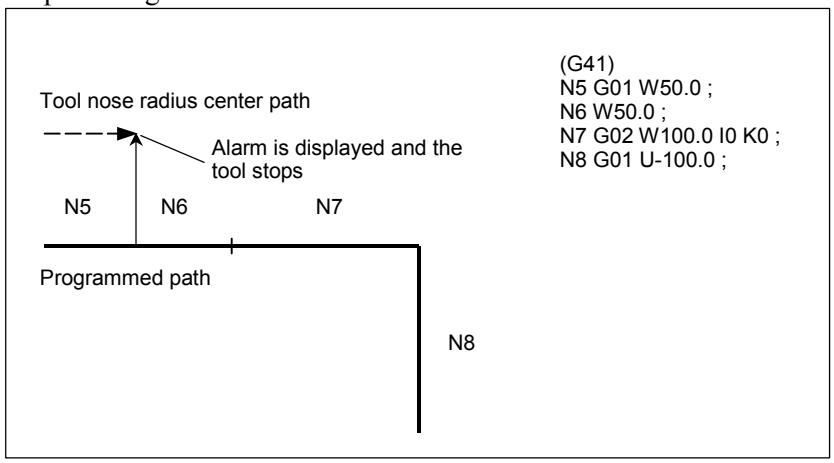
If the tool nose radius compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool nose radius compensation. When this is predicted, alarm PS0033 occurs at the end of the previous block and the tool is stopped.

In the example shown below, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool nose radius compensation. If an excessively large value is specified, this intersection does not occur.



- When the center of the arc is identical with the start point or the end position

If the center of the arc is identical with the start point or end point, alarm PS0041 is displayed, and the tool will stop at the start point of the preceding block of the arc.



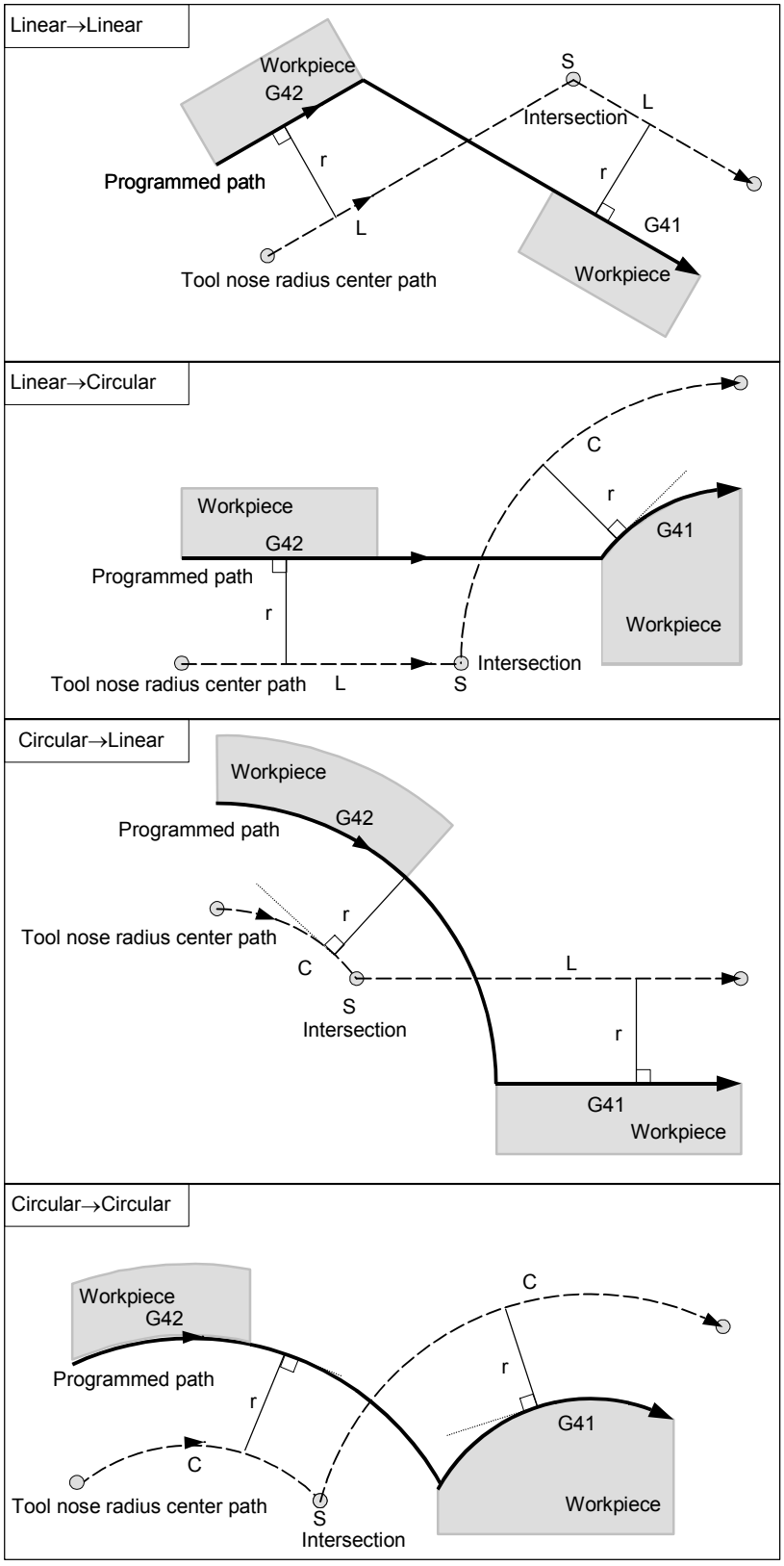
- Change in the offset direction in the offset mode

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

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

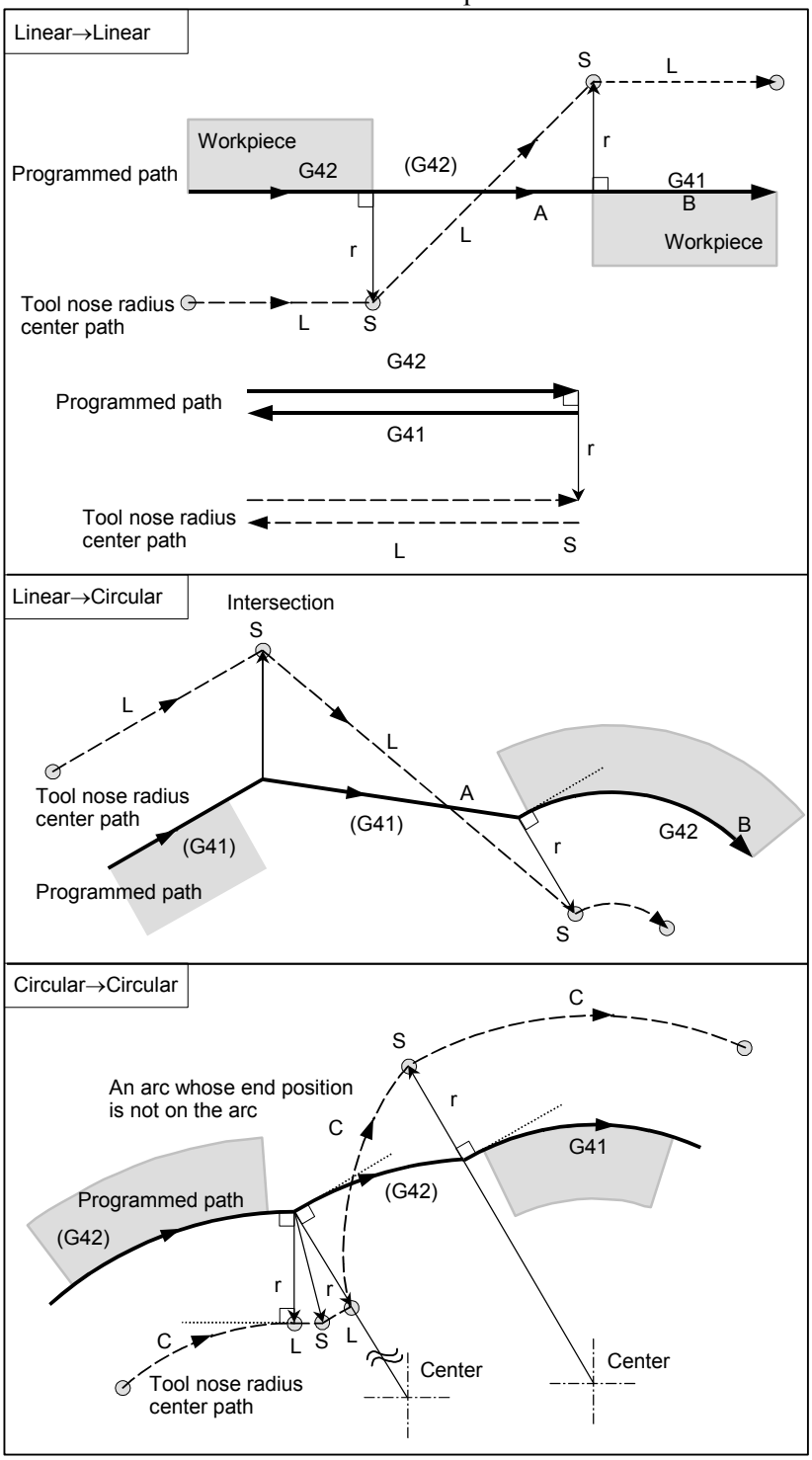
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool nose radius center path of that block and the tool nose radius center path of a preceding block.
However, the change is not available in the start-up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

- Tool nose radius center path with an intersection



- Tool nose radius center path without an intersection

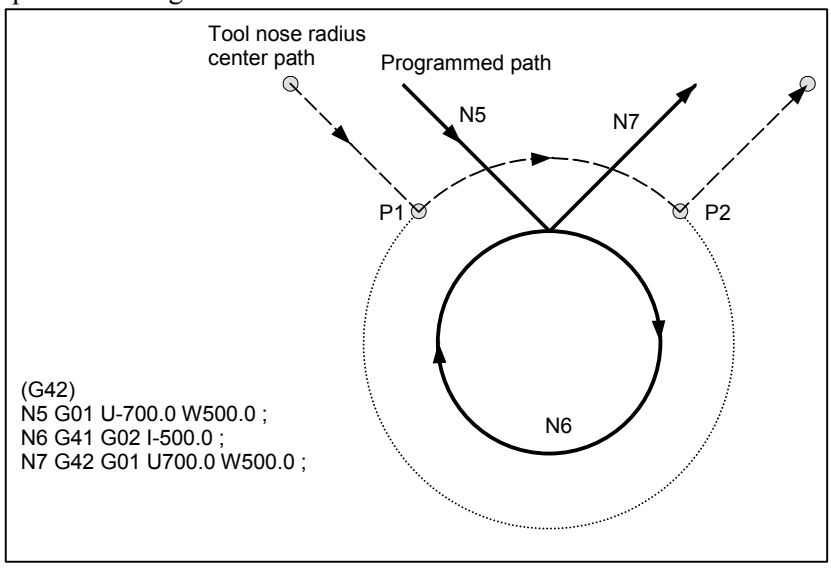
When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.



The length of tool center path larger than the circumference of a circle

Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I, J, and K this situation can occur.

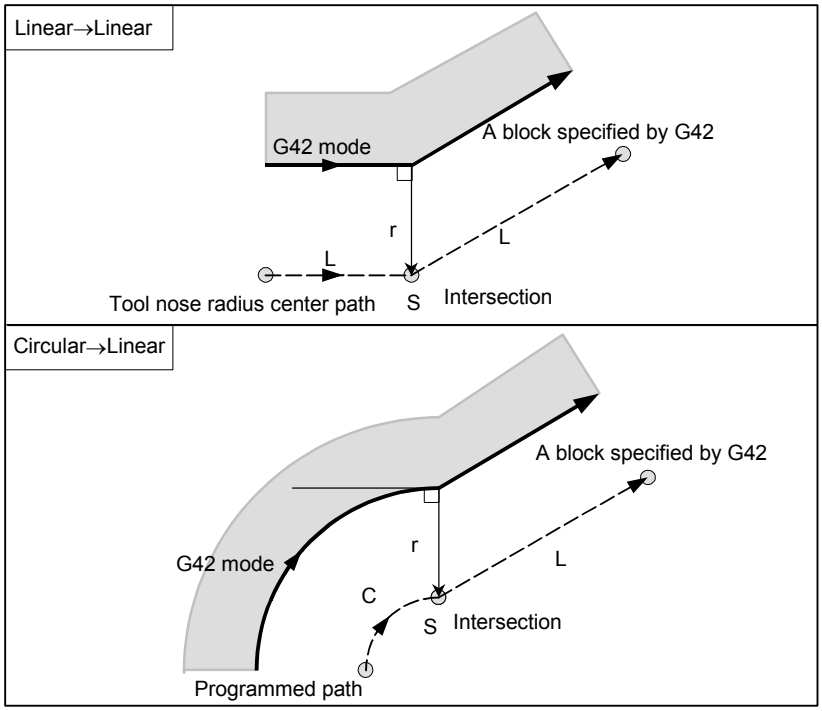
In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from P_1 to P_2 as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.



- Tool nose radius compensation G code in the offset mode

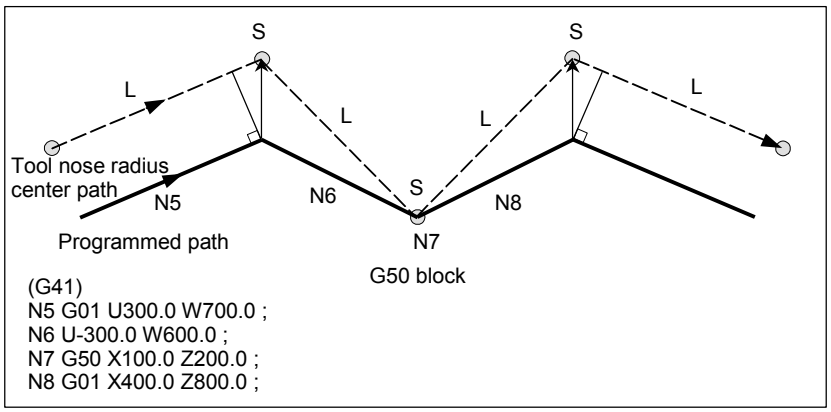
The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining inner or outer side, by commanding the tool nose radius compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct 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), see "Change in the offset direction in the offset mode".



- Command canceling the offset vector temporarily

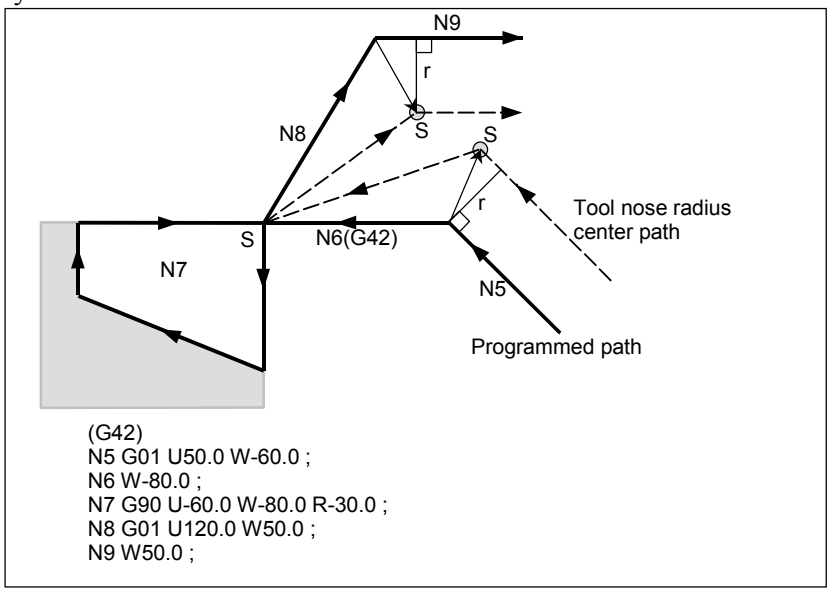
During offset mode, if G50 (workpiece coordinate system setting) or G52 (local coordinate system setting) is commanded, the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.
In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled.
Also when restored to offset mode, the tool moves directly to the intersecting point.



Before specifying G28 (reference position return), G30 (second, third, and fourth reference position returns), and G53 (machine coordinate system selection) commands, cancel offset mode, using G40. If an attempt is made to specify any of the commands in offset mode, the offset vector temporarily disappears.

- Canned cycles (G90, G92, G94) and multiple repetitive canned cycles (G71 to G76)

See the cautions for the tool nose radius compensation related canned cycles.



- If I, J, and K are specified in a G00/G01 mode block

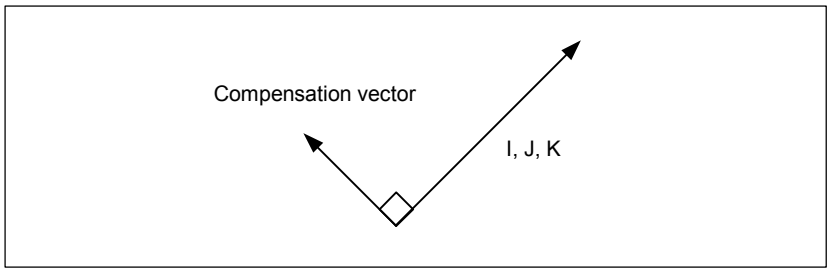
At the start of tool nose radius compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This makes it possible to change the compensation direction intentionally.

IJ type vector (XY plane)

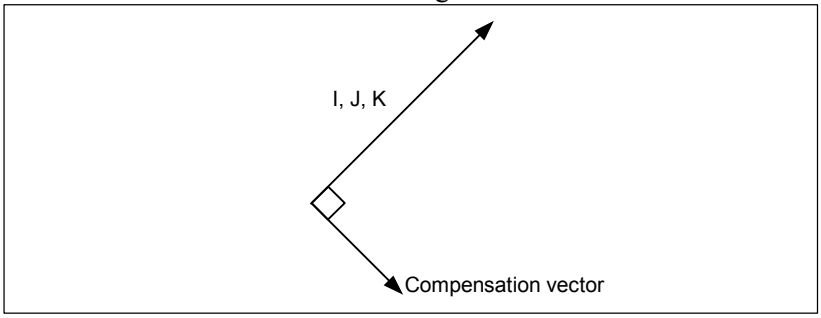
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) As shown in the figure below, it is assumed that the compensation vector (IJ type vector) is the vector with a size equal to the compensation value and vertical to the direction specified by I and J, without performing intersection calculation on the programmed path. I and J can be specified both at the start of tool nose radius compensation and in that mode. If they are specified at the start of compensation, any start-up type set in the appropriate parameter will be invalid, and an IJ type vector is assumed.

Offset vector direction

In G41 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the left side.



In G42 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the right side.

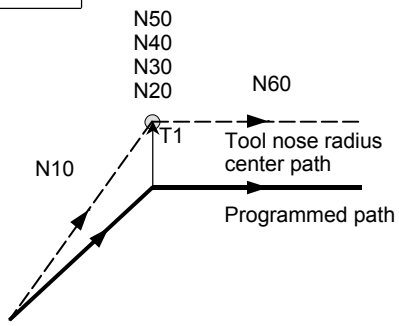


Example

If I and J are specified at the start of compensation (with tool movement)

```
      (G40)
N10 G41 U100.0 W100.0
      K1 T0101 ;
N20 G04 X1000 ;
N30 G01 F1000 ;
N40 S300 ;
N50 M50 ;
N60 W150. ;
```

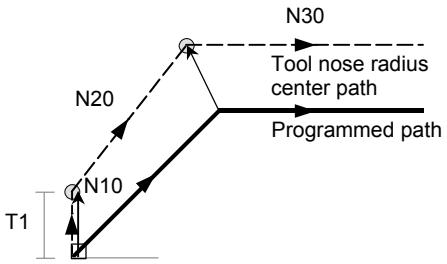
Note) In N10, a vector is specified with a size of T1 in the direction vertical to the Z axis, using K1.



If I and J are specified at the start of compensation (without tool movement)

```
      (G40)
N10 G41 K1 T0101 ;
N20 U100.0 W100.0 ;
N30 W150.0 ;
```

Note) In N10, a vector is specified with a size of T1 in the direction vertical to the Z axis, using K1.

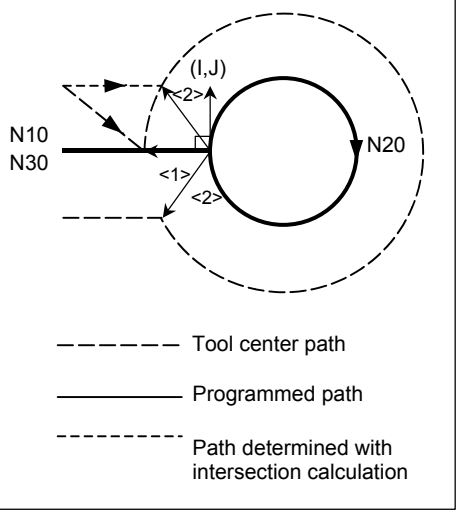


If I and J are specified at the start of compensation (with tool movement)

```
      (G17 G41 T0101)
N10 G00 U150.0 J50.0 ;
N20 G02 I50.0 ;
N30 G00 U-150.0 ;
```

Note) In N10, a vector is specified with a size of T1 in the direction vertical to the Y axis, using J50.

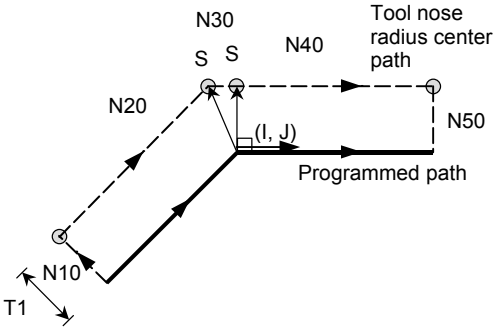
<1> IJ type vector
<2> Vector determined with intersection calculation



If I and J are specified in a block without tool movement in compensation mode

Start-up/cancel type C

```
N10 G41 T0101 G01 F1000 ;
N20 U100.0 W100.0 ;
N30 K10.0 ;
N40 W150.0 ;
N50 G40 ;
```

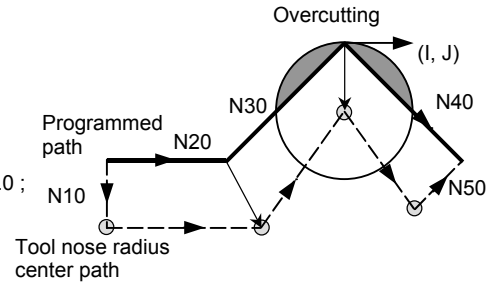


Limitation

If an IJ type vector is specified, tool interference may occur due to that vector alone, depending on the direction. If this occurs, no interference alarm will occur, or no interference avoidance will be performed. Overcutting may, therefore, result.

Start-up/cancel
Type C

```
N10 G42 T0101 F1000 ;
N20 W100.0;
N30 U100.0 W100.0 K10.0 ;
N40 U-100.0 W100.0 ;
N50 G40 ;
```



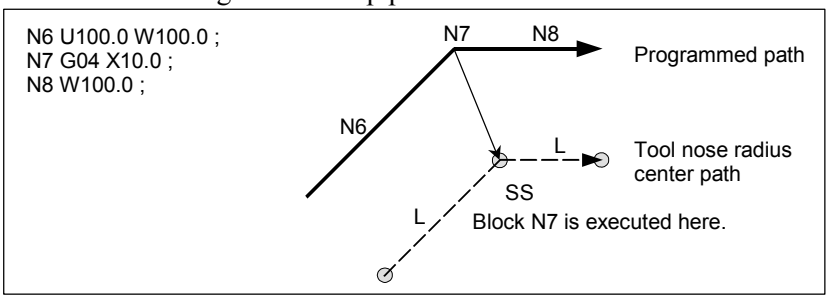
- A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

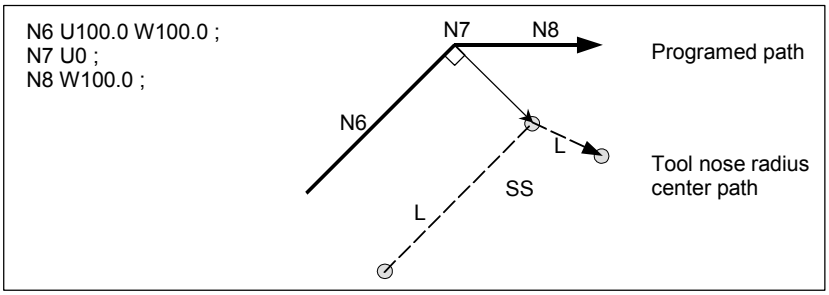
M05 ;	: M code output
S21 ;	: S code output
G04 X10.0 ;	: Dwell
G22 X100000 ;	: Machining area setting
G10 P01 X10 Z20 R10.0 ;	: Tool nose radius compensation value setting/changing
(G18) Y200.0 ;	: Move command not included in the offset plane.
G98 ;, O10 ;, N20 ;	: G, O, and N codes only
U0 ;	: Move distance is zero.

- A block without tool movement specified in offset mode

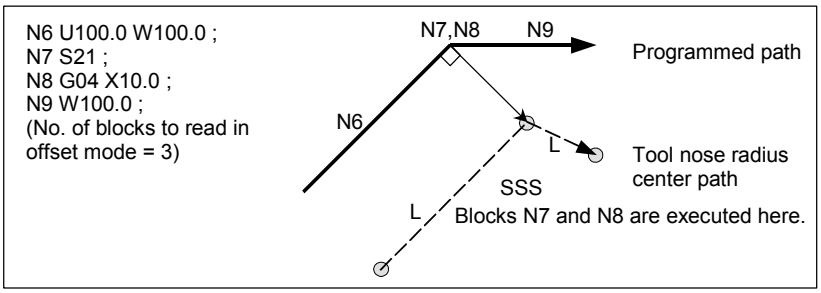
Unless the number of blocks without movement consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)) in offset mode, the vector and the tool nose radius center path will be as usual. This block is executed at the single block stop point.



For an axis command for which the move distance is zero, however, a vector with a size equal to the compensation value will be created vertical to the movement direction in the previous block, even if the number of block is 1. Note that specifying such a command may result in overcutting.

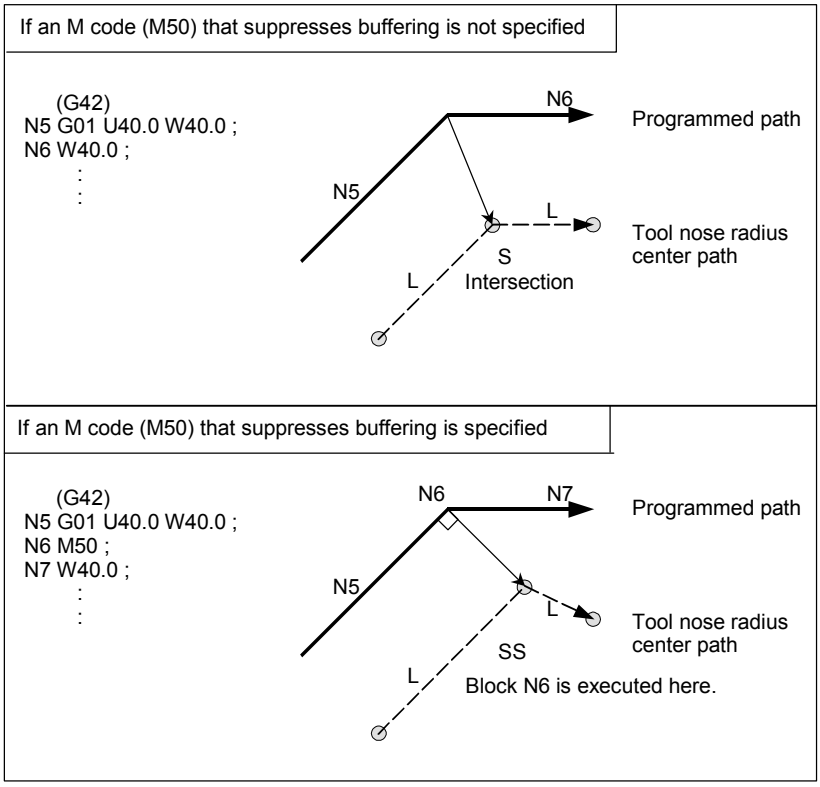


In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter (No. 19625)). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.



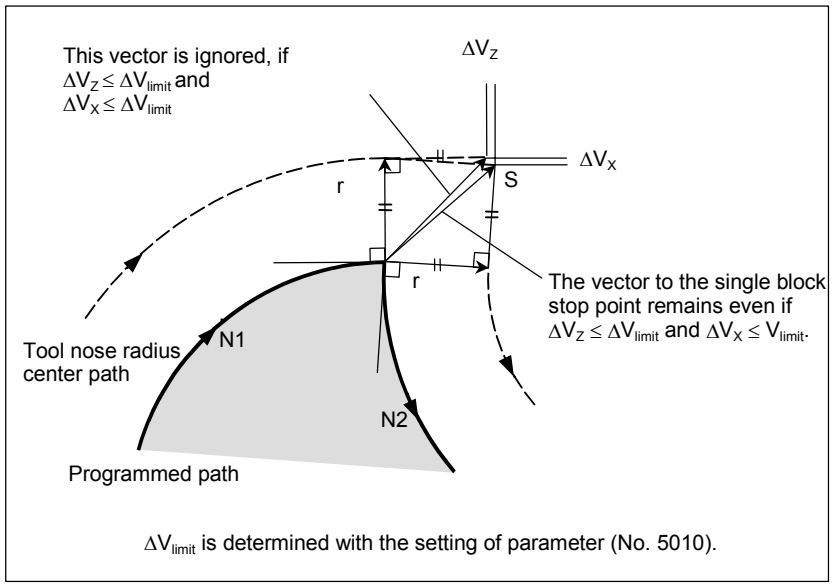
- If an M/G code that suppresses buffering is specified

If an M/G code that suppresses buffering is specified in offset mode, it is no longer possible to read and analyze subsequent blocks regardless of the number of blocks to read in offset mode, which is determined by parameter (No. 19625). Then, intersection calculation and a interference check, described later, are no longer possible. If this occurs, overcutting may occur because a vertical vector is output in the immediately preceding block.

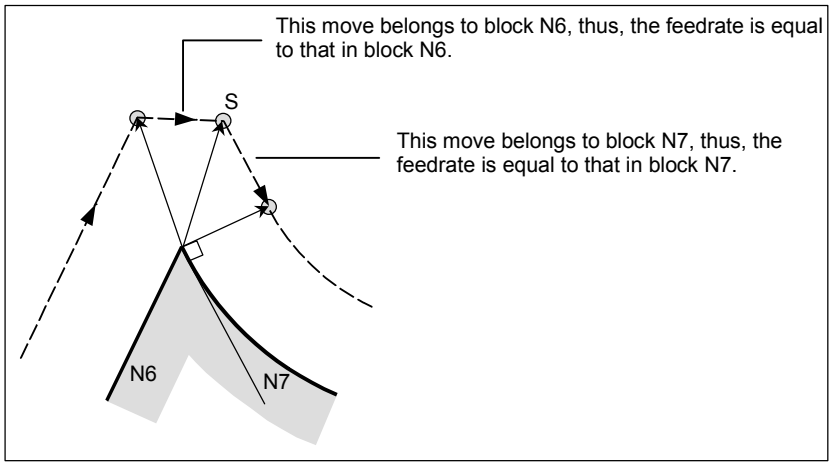


- Corner movement

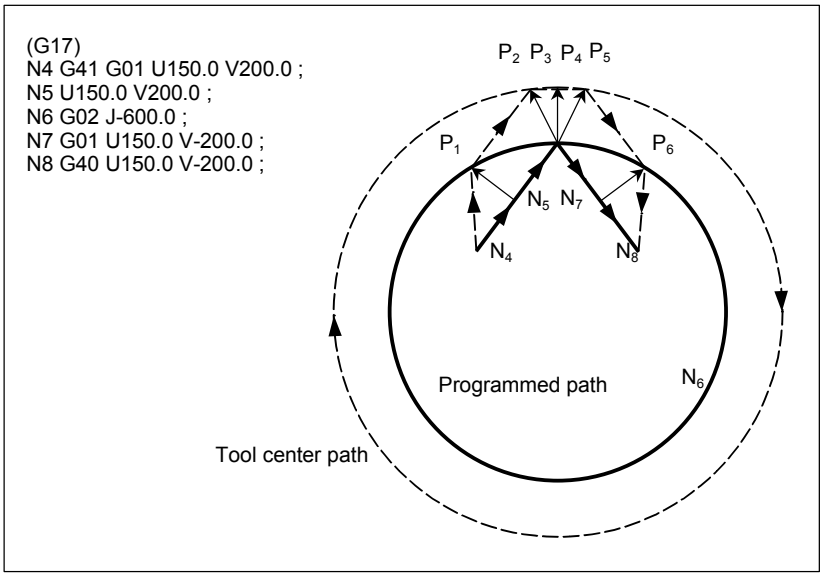
When two or more offset vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.
If these vectors almost coincide with each other (the distance of corner movement between the vectors is judged short due to the setting of parameter (No. 5010)), corner movement is not performed. In this case, the vector to the single block stop point takes precedence and remains, while other vectors are ignored. This makes it possible to ignore the very small movements arising from performing tool nose radius compensation, thereby preventing velocity changes due to interruption of buffering.



If the vectors are not judged to almost coincide (therefore, are not erased), movement to turn around the corner is performed. The corner movement that precedes the single block stop point belongs to the previous block, while the corner movement that succeeds the single block stop point belongs to the latter block.



However, if the path of the next block is semicircular or more, the above function is not performed.
The reason for this is as follows:



If the vector is not ignored, the tool path is as follows:
$P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow$ (Circle) $\rightarrow P_4 \rightarrow P_5 \rightarrow P_6$
But if the distance between P_2 and P_3 is negligible, the point P_3 is ignored. Therefore, the tool path is as follows:
$P_2 \rightarrow P_4$
Namely, circle cutting by the block N6 is ignored.

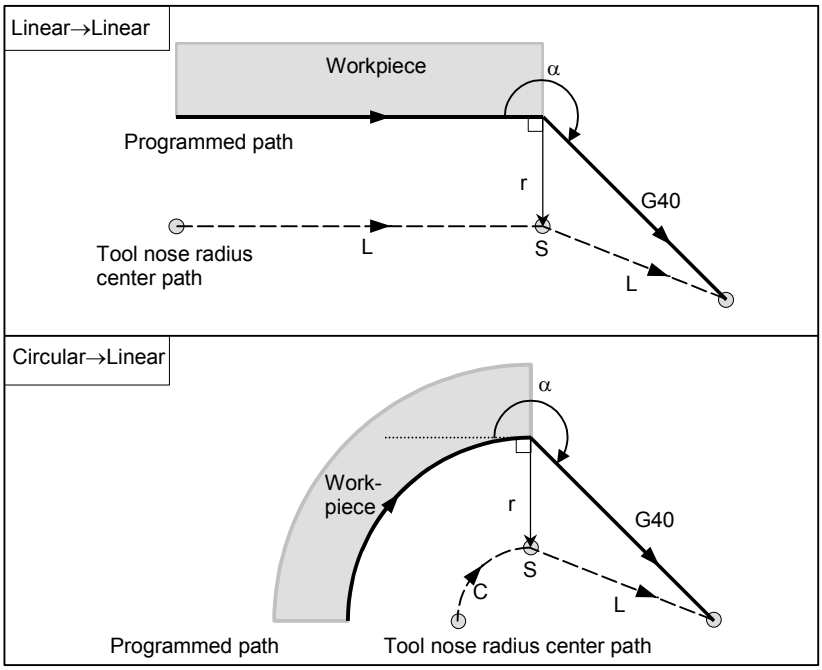
- Interruption of manual operation

For manual operation during the offset mode, see "Manual Absolute ON and OFF."

5.3.4 Tool Movement in Offset Mode Cancel

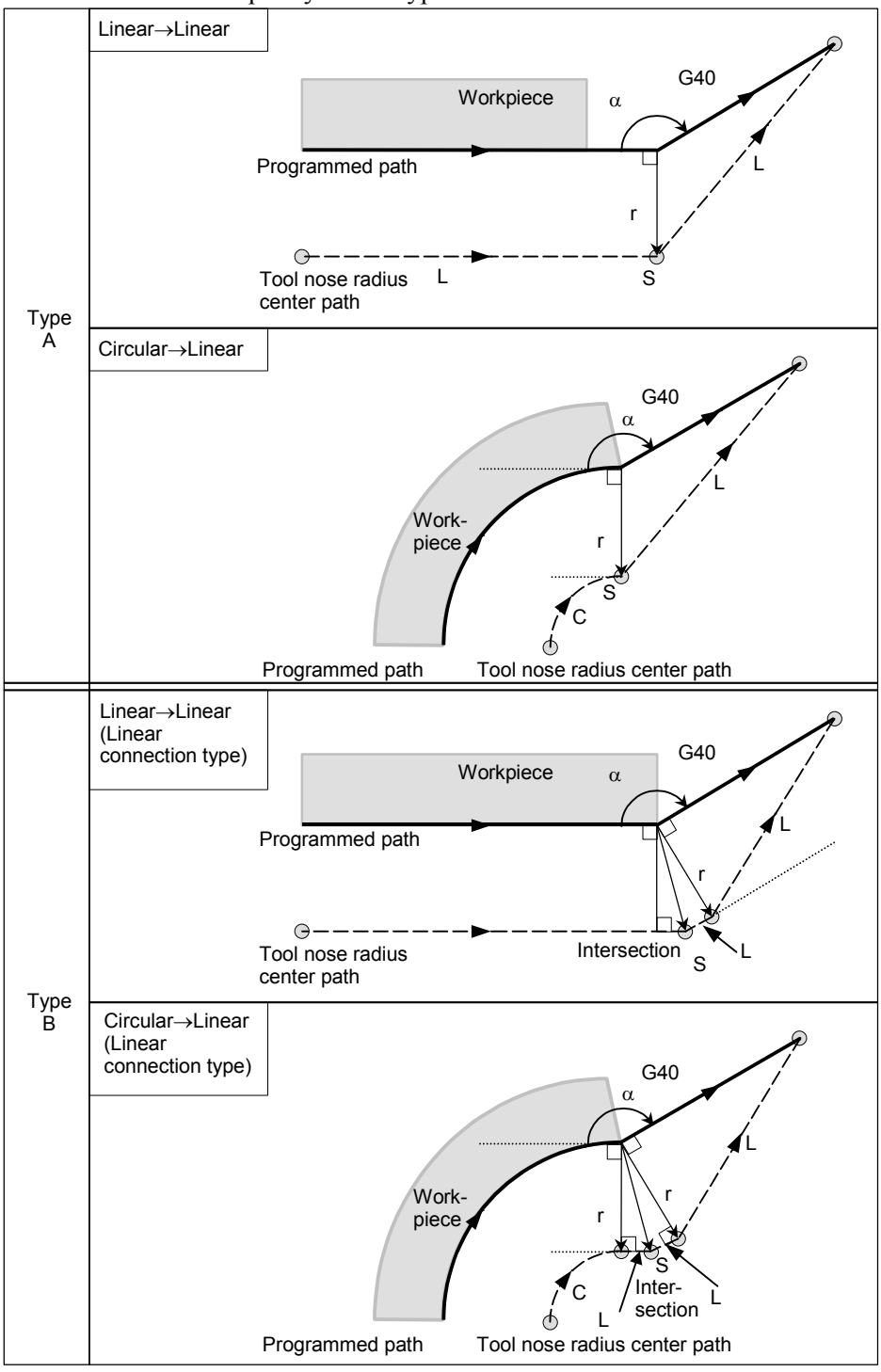
Explanation

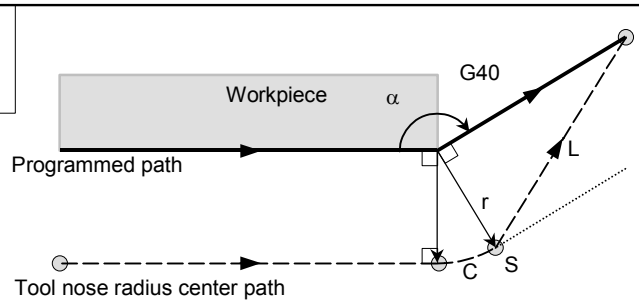
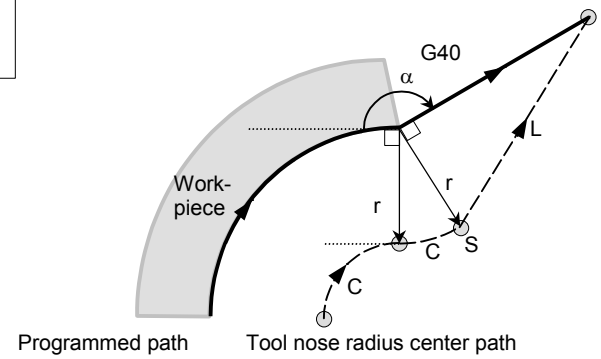
- If the cancel block is a block with tool movement, and the tool moves around the inside (180° ≤ α)



- If the cancel block is a block with tool movement, and the tool moves around the outside at an obtuse angle (90° ≤ α < 180°)

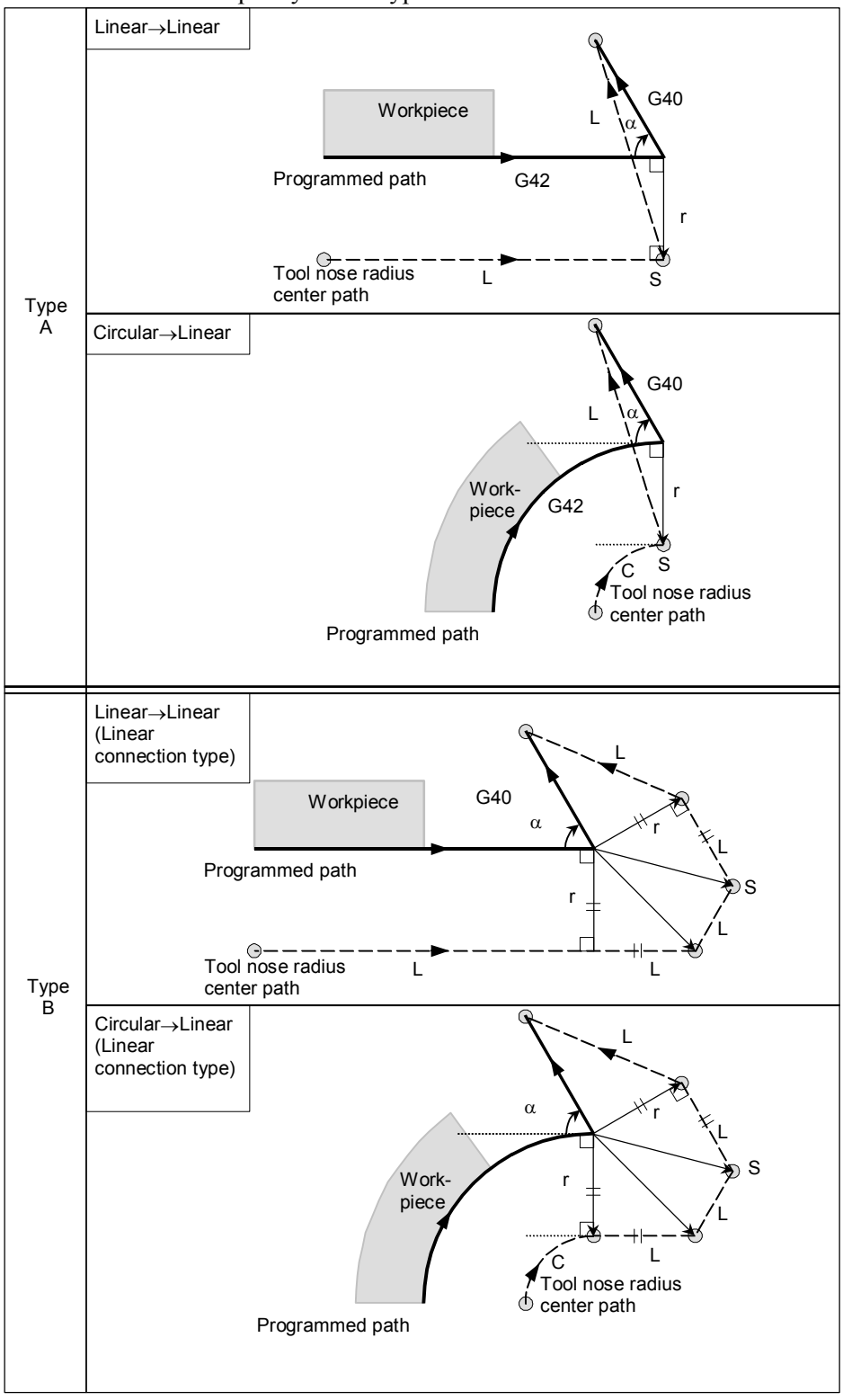
The two types, A and B, are available. Set bit 0 (SUP) of parameter No. 5003 to specify which type is to be used.

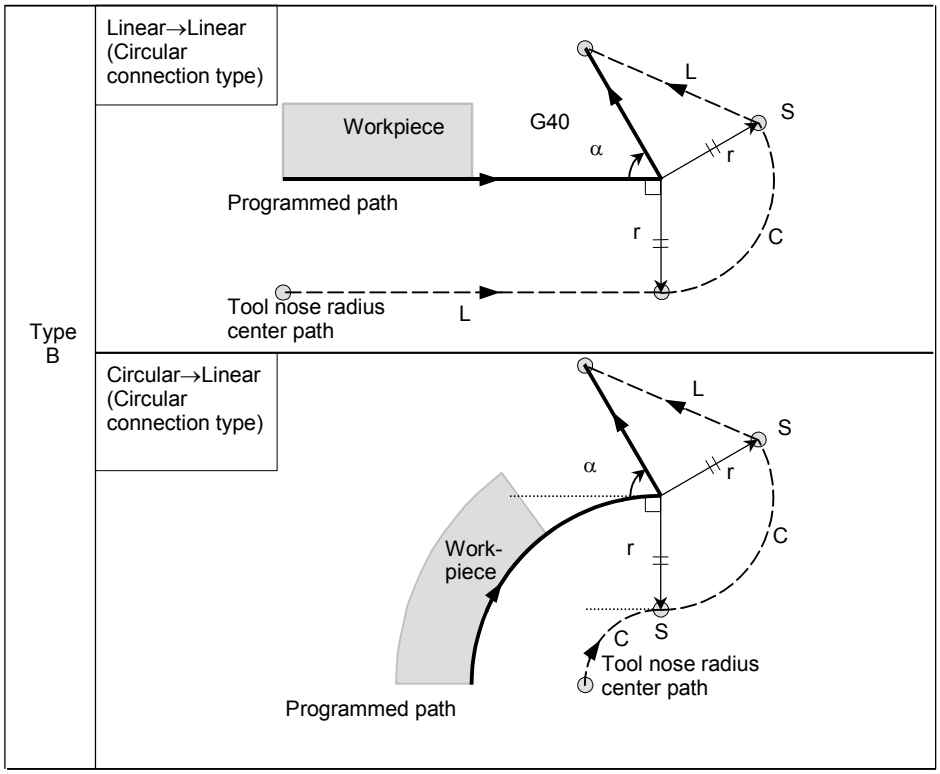


<table>
<tr><td rowspan="2">Type B</td><td>Linear→Linear (Circular connection type)</td><td>
</td></tr>
<tr><td>Circular→Linear (Circular connection type)</td><td>
</td></tr>
</table>

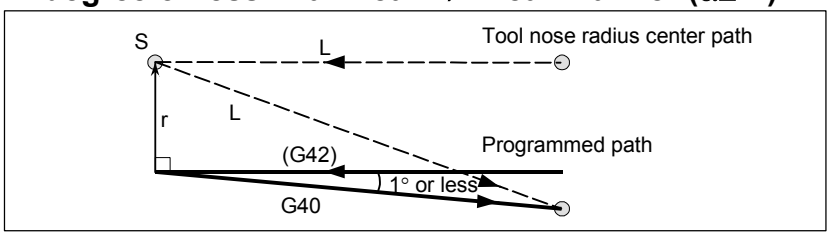
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle (α<90°)

The two types, A and B, are available. Set bit 0 (SUP) of parameter No. 5003 to specify which type is to be used.





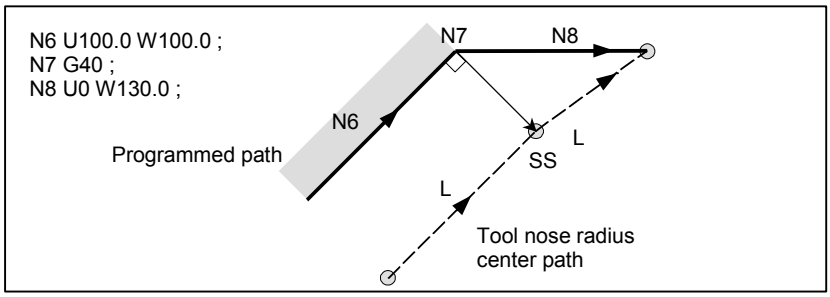
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle of 1 degree or less in a linear → linear manner (α≤1°)



- A block without tool movement specified together with offset cancel

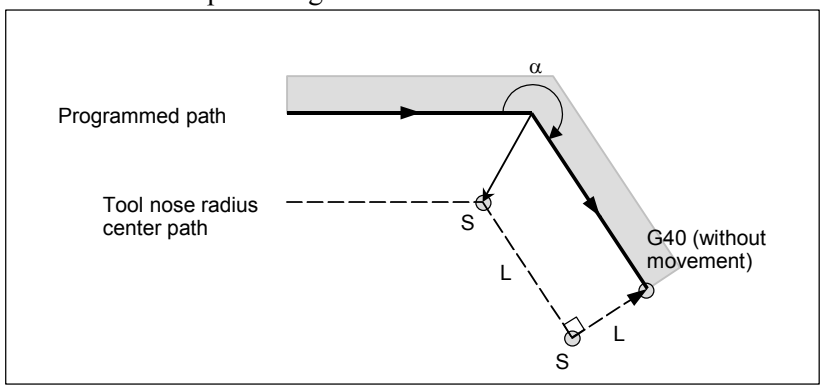
For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the tool nose radius compensation value in the vertical direction. The tool does not operate in the cancel block. The remaining vectors are canceled with the next move command.



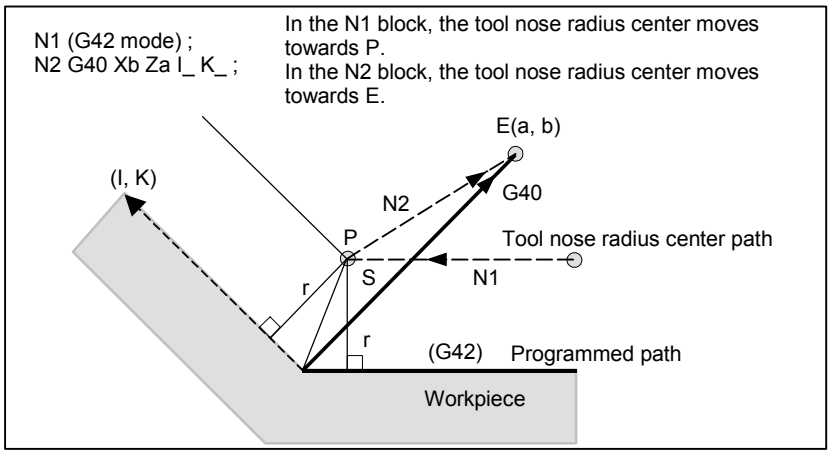
For type C

The tool shifts by the compensation value in the direction vertical to the block preceding the cancel block.

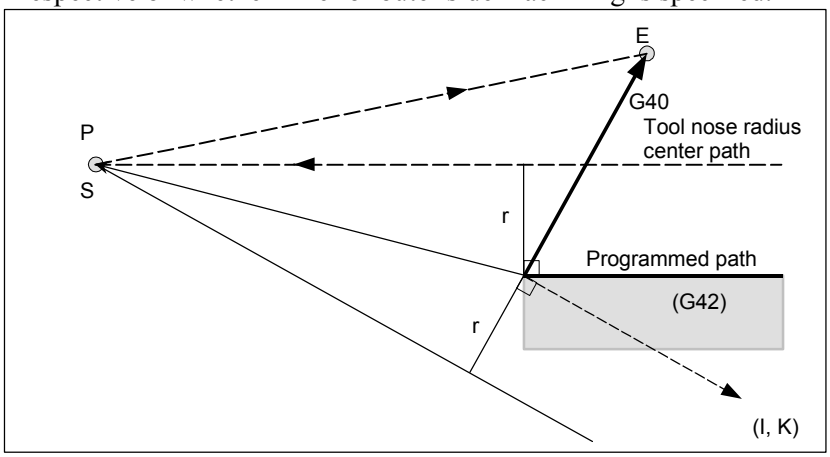


- Block containing G40 and I_J_K_ The previous block contains G41 or G42

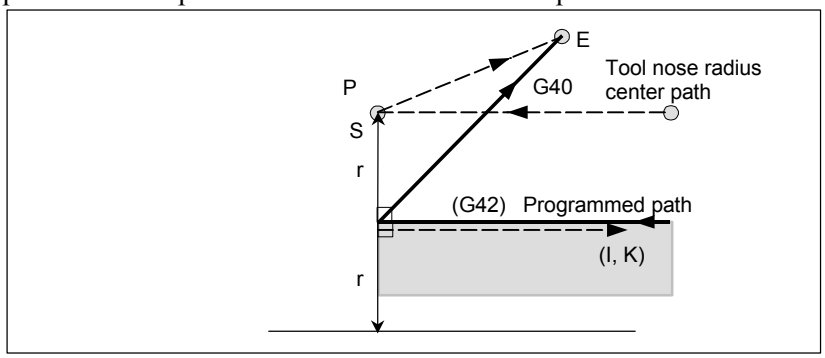
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified.

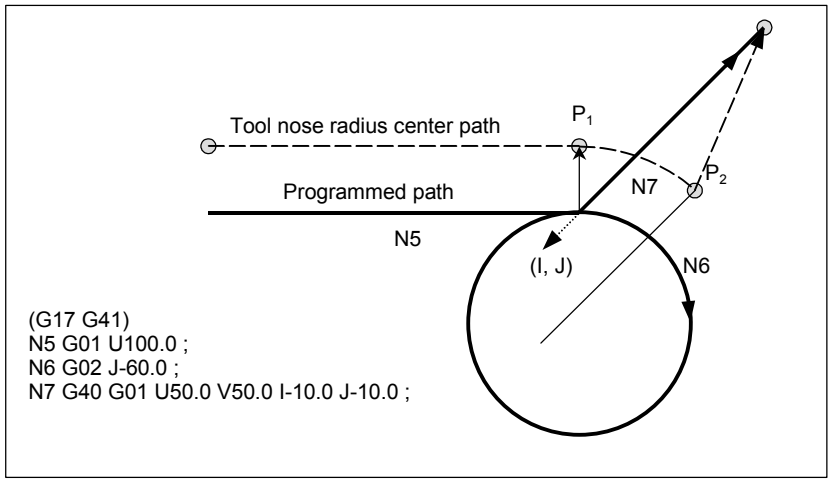


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



- Length of the tool center path larger than the circumference of a circle

In the example shown below, the tool does not trace the circle more than once. It moves along the arc from P_1 to P_2. The interference check function described below may raise an alarm.
To make the tool trace a circle more than once, program two or more arcs.



5.3.5 Prevention of Overcutting Due to Tool Nose Radius Compensation

Explanation

- Machining a groove smaller than the diameter of the tool nose

Since the tool nose radius compensation forces the path of the center of the tool nose radius to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.

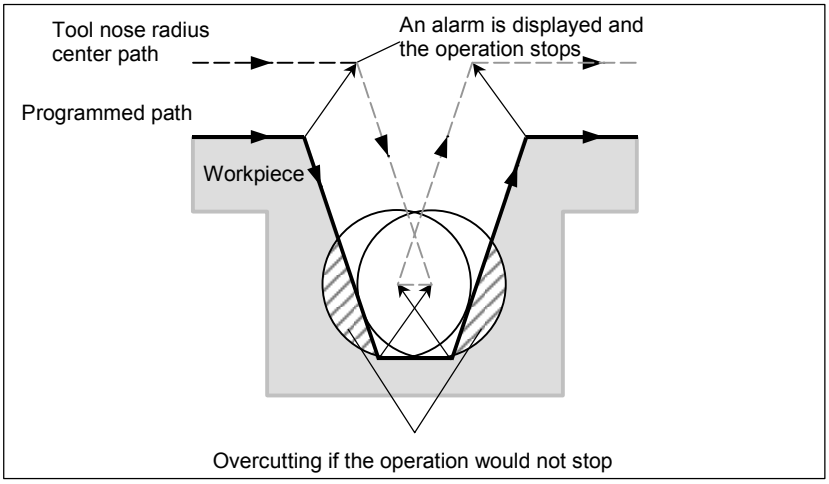


Fig. 5.3.5 (a) Machining a groove smaller than the diameter of the tool nose

- Machining a step smaller than the tool nose radius

For a figure in which a workpiece step is specified with an arc, the tool nose radius center path will be as shown in Fig. 5.3.5 (b). If the step is smaller than the tool nose radius, the tool nose radius center path usually compensated as shown in Fig. 5.3.5 (c) may be in the direction opposite to the programmed path. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued.
If the step is of linear, no alarm will be generated and cut correctly. However uncut part will remain.

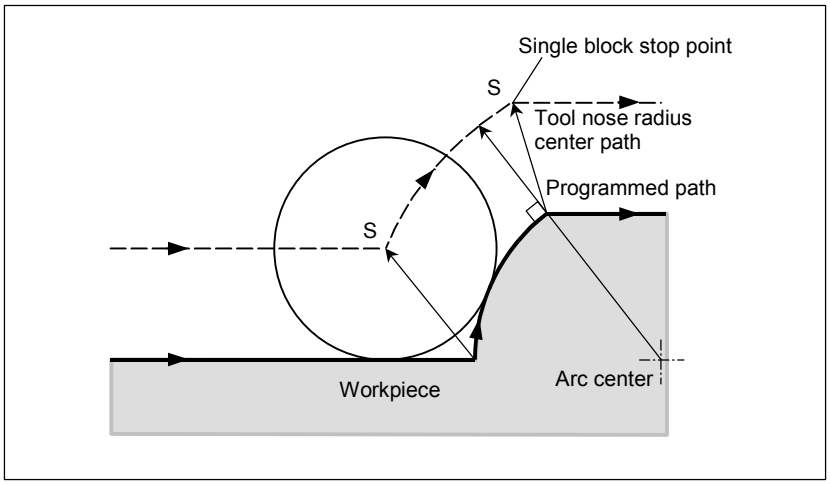


Fig. 5.3.5 (b) Machining a step larger than the tool nose radius

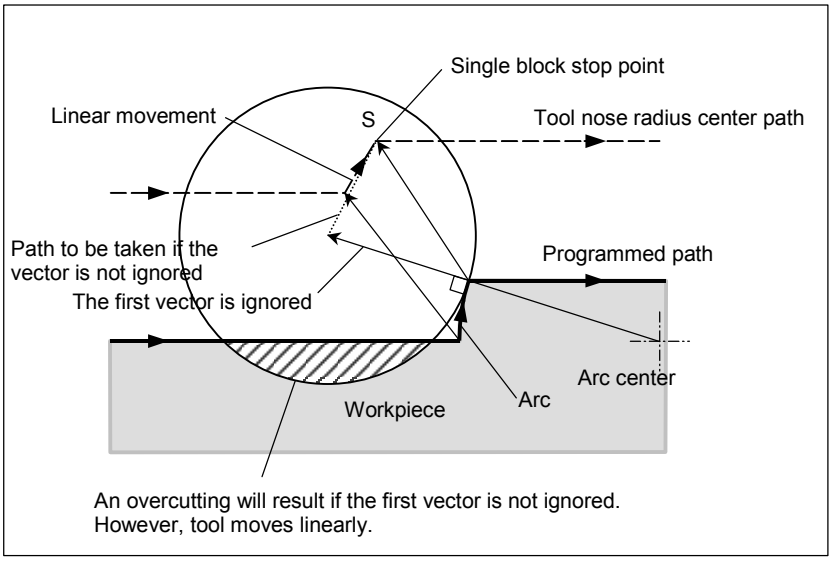
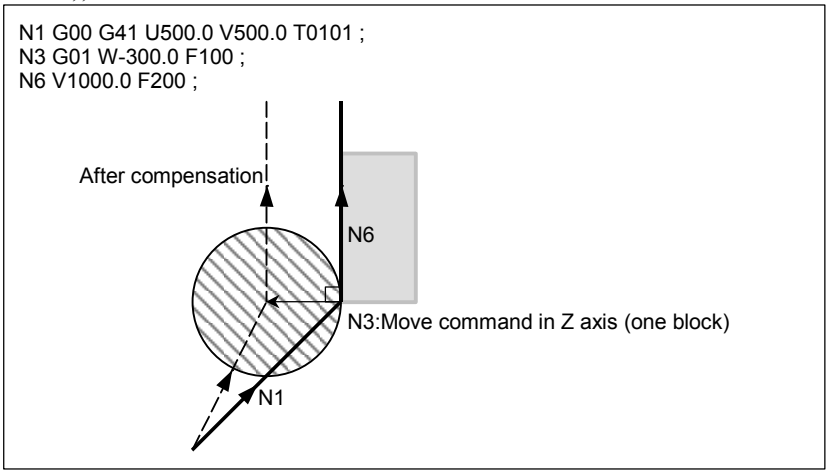


Fig. 5.3.5 (c) Machining a step smaller than the tool nose radius

- Starting compensation and cutting along the Z-axis

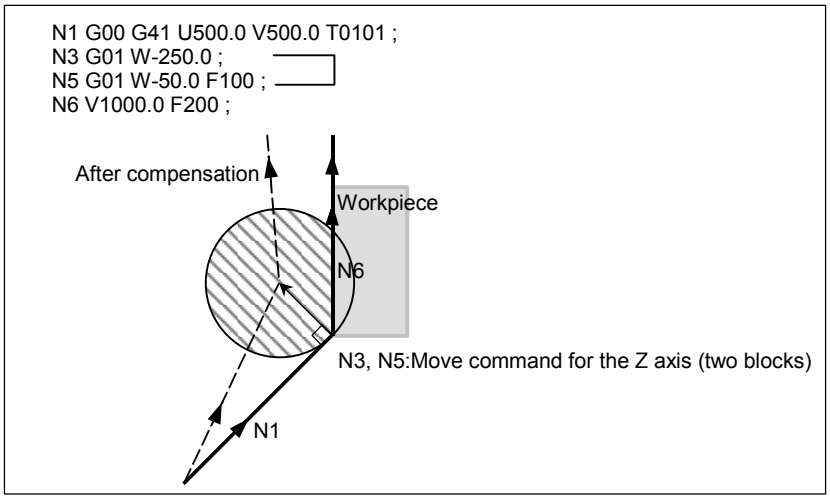
It is usually used such a method that the tool is moved along the Z axis after the tool nose radius compensation (normally XY plane) is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.

Let us consider the following program, assuming the number of blocks to read in tool nose radius compensation mode (parameter (No. 19625)) to be 3.



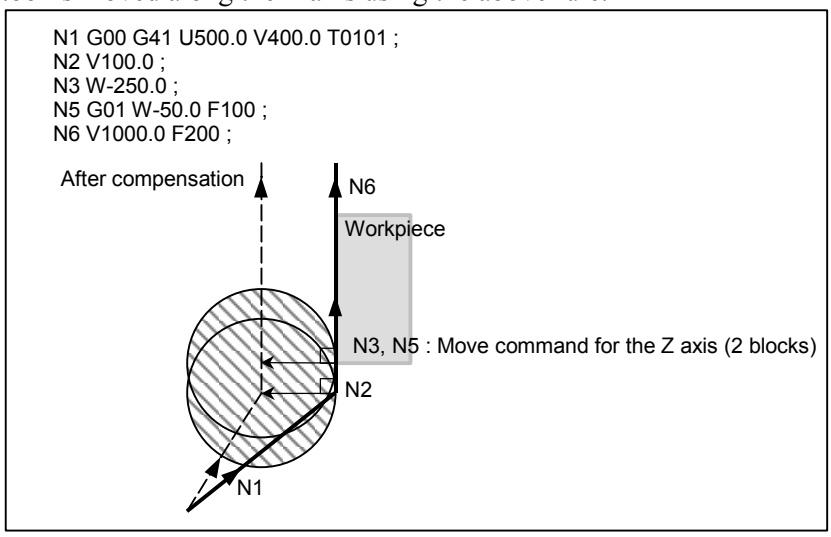
In the program example above, when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the figure above.

Then, suppose that the block N3 (move command in Z axis) is divided into N3 and N5.



At this time, because the number of blocks to read is 3, blocks up to N5 can be read at the start of N1 compensation, but block N6 cannot be read. As a result, compensation is performed only on the basis of the information in block N1, and a vertical vector is created at the end of the compensation start block. Usually, therefore, overcutting will result as shown in the figure above.

In such a case, it is possible to prevent overcutting by specifying a command with the exactly the same direction as the advance direction immediately before movement along the Z axis beforehand, after the tool is moved along the Z axis using the above rule.



As the block N2 has the move command in the same direction as that of the block N6, the correct compensation is performed.

Alternatively, it is possible to prevent overcutting in the same way by specifying an IJ type vector with the same direction as the advance direction in the start-up block, as in N1 G00 G41 U500.0 V500.0 I0 J1 T0101;, after the tool has moved along the Z axis.

5.3.6 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

Explanation

- Condition under which an interference check is possible

To perform an interference check, it is necessary to read at least three blocks with tool movement. If, therefore, three or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as independent auxiliary function and dwell, are specified in succession, excessive or insufficient cutting may occur because an interference check fails. Assuming the number of blocks to read in offset mode, which is determined by parameter (No. 19625), to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which an interference check is possible is

$$(N - 3) \geq M.$$

For example, if the maximum number of blocks to read in offset mode is 8, an interference check is possible even if up to five blocks without tool movement are specified. In this case, three adjacent blocks can be checked for interference, but any subsequent interference that may occur cannot be detected.

- Interference check method

Two interference check methods are available, direction check and circular angle check. Parameter CNC (No. 5008#1) and parameter CNV (No. 5008#3) are used to specify whether to enable these methods.

CNV	CNC	Operation
0	0	An interference check is enabled, and a direction check and a circular angle check can be performed.
0	1	An interference check is enabled, and only a circular angle check is performed.
1	–	An interference check is disabled.

NOTE
There are no settings for performing a direction check only.

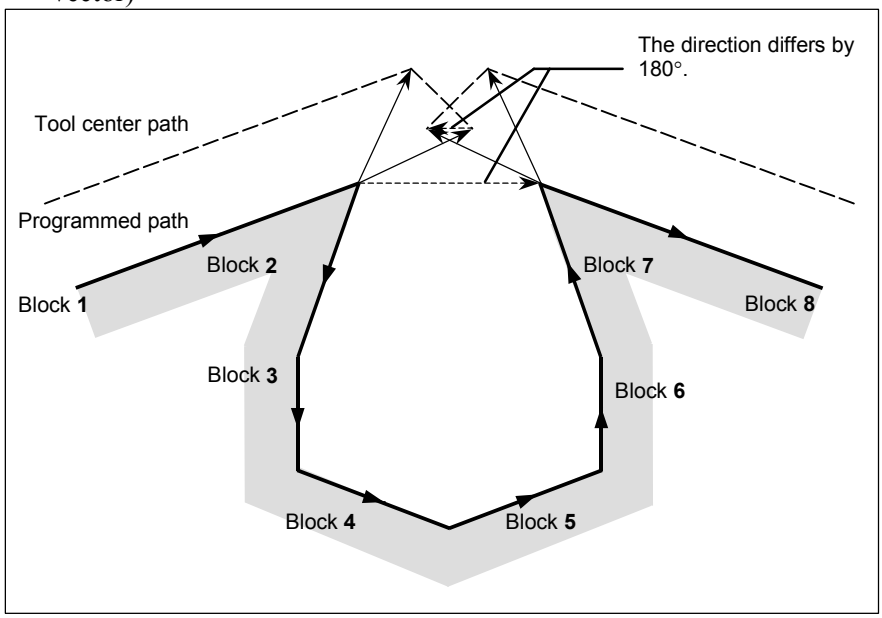
- Interference reference <1> (direction check)

Assuming the number of blocks to read during tool nose radius compensation to be N, a check is first performed on the compensation vector group calculated in (block 1 - block 2) to be output this time and the compensation vector group calculated in (block N-1 - block N); if they intersect, they are judged to interfere. If no interference is found, a check is performed sequentially in the direction toward the compensation vector group to be output this time, as follows:

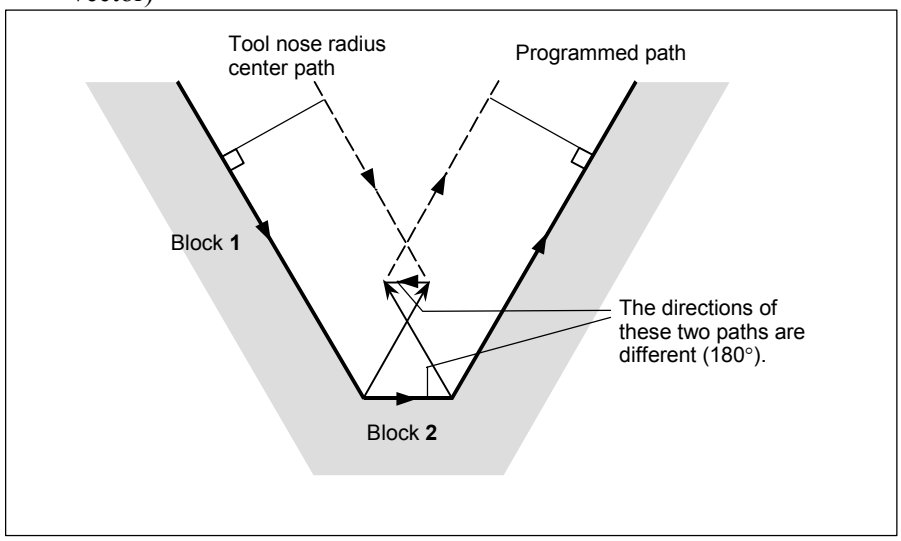
(Block 1 - block 2) and (block N-2 - block N-1)
(Block 1 - block 2) and (block N-3 - block N-2)
:
:
(Block 1 - block 2) and (block 2 - block 3)

Even if multiple number of compensation vector groups are generated, a check is performed on all pairs.
The judgment method is as follows: For a check on the compensation vector group in (block 1 - block 2) and those in (block N-1 - block N), the direction vector from the specified (end point of block 1) to the (end point of block N-1) is compared with the direction vector from the (point resulting from adding the compensation vector to be checked to the end of block 1) to the (point resulting from adding the compensation vector to be checked to the end of block N-1), and if the direction is 90^o or greater or 270^o or less, they are judged to intersect and interfere. This is called a direction check.

Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 7 end-point vector)



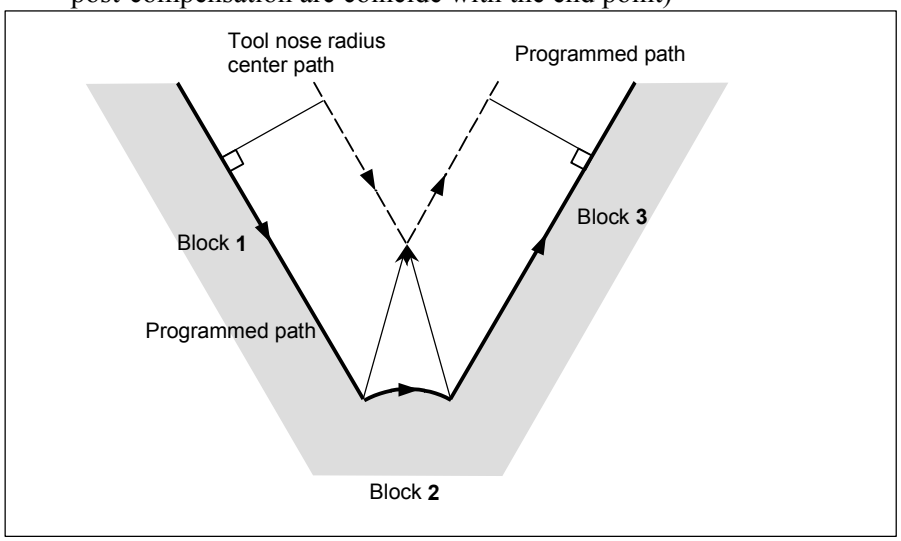
Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 2 end-point vector)



- Interference reference <2> (circular angle check)

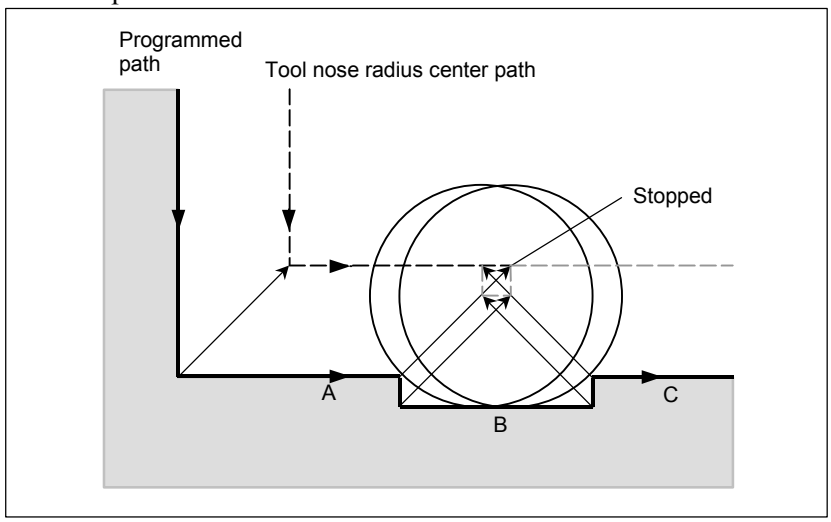
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed path and the circular angle of the start and end point of the post-compensation path, in addition to direction check <1>. If the difference is 180^{o} or greater, the blocks are judged to interfere. This is called a circular angle check.

Example of <2> (if block 2 is circular and the start point of the post-compensation arc coincide with the end point)



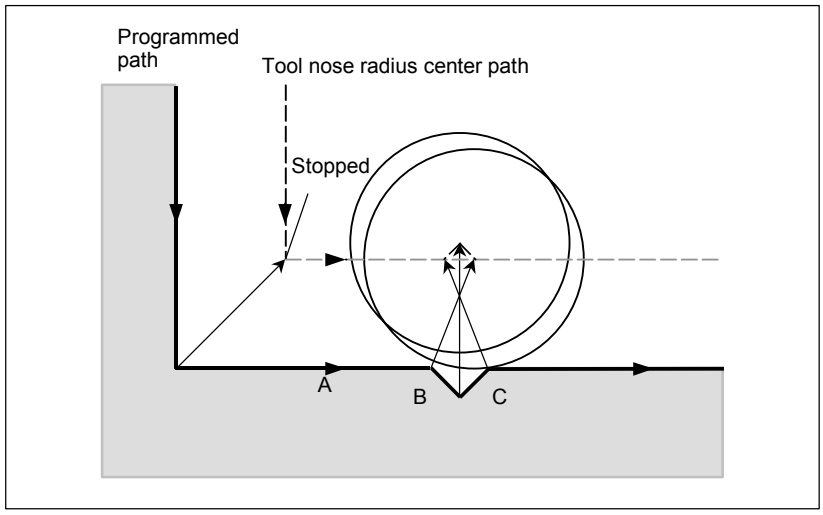
- When interference is assumed although actual interference does not occur

<1> Depression which is smaller than the tool 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 the tool nose radius compensation, the tool stops and an alarm is displayed.

<2> Groove which is smaller than the tool nose radius compensation value



Like <1>, an alarm is displayed because of the interference as the direction is reverse in block B.

5.3.6.1 Operation to be performed if an interference is judged to occur

Explanation

The operation to be performed if an interference check judges that an interference (due to overcutting) occurs can be either of the following two, depending on the setting of parameter CAV (No. 19607#5).

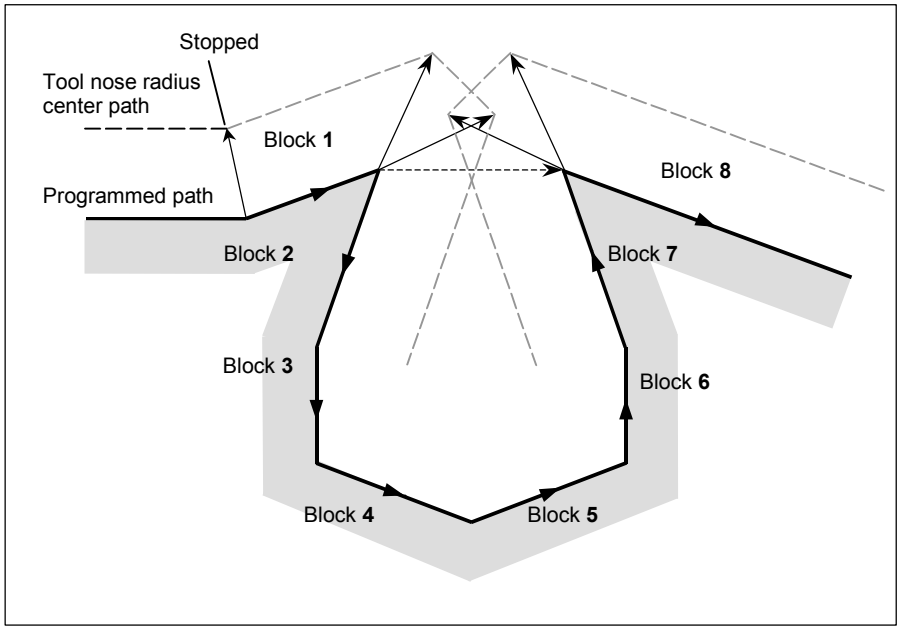
CAV	Function	Operation
0	Interference check alarm function	An alarm stop occurs before the execution of the block in which overcutting (interference) occurs.
1	Interference check avoidance function	The tool path is changed so that overcutting (interference) does not occur, and processing continues.

5.3.6.2 Interference check alarm function

Explanation

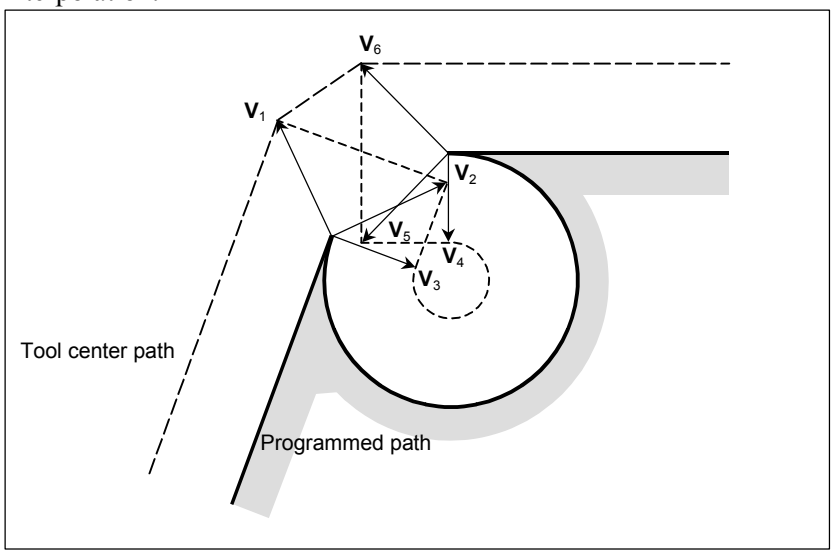
- Interference other than those between adjacent three blocks

If the end-point vector of block 1 and the end-point vector of block 7 are judged to interfere as shown in the figure, an alarm will occur before the execution of block 1 so that the tool stops. In this case, the vectors will not be erased.

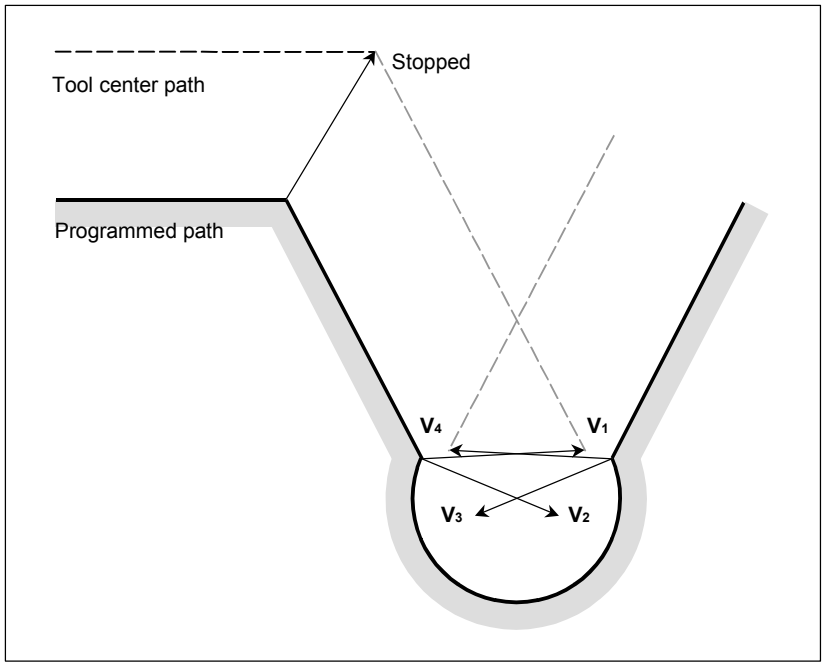


- Interference between adjacent three blocks

If an interference is judged to occur between adjacent three blocks, the interfering vector, as well as any vectors existing inside of it, is erased, and a path is created to connect the remaining vectors. In the example shown in the figure below, V_2 and V_5 interfere, so that V_2 and V_5 are erased, so are V_3 and V_4, which are inside of them, and V_1 is connected to V_6. The operation during this time is linear interpolation.



If, after vector erasure, the last single vector still interferes, or if there is only one vector at the beginning and it interferes, an alarm will occur immediately after the start of the previous block (end point for a single block) and the tool stops. In the example shown in the figure below, V_2 and V_3 interfere, but, even after erasure, an alarm will occur because the final vectors V_1 and V_4 interfere.



5.3.6.3 Interference check avoidance function

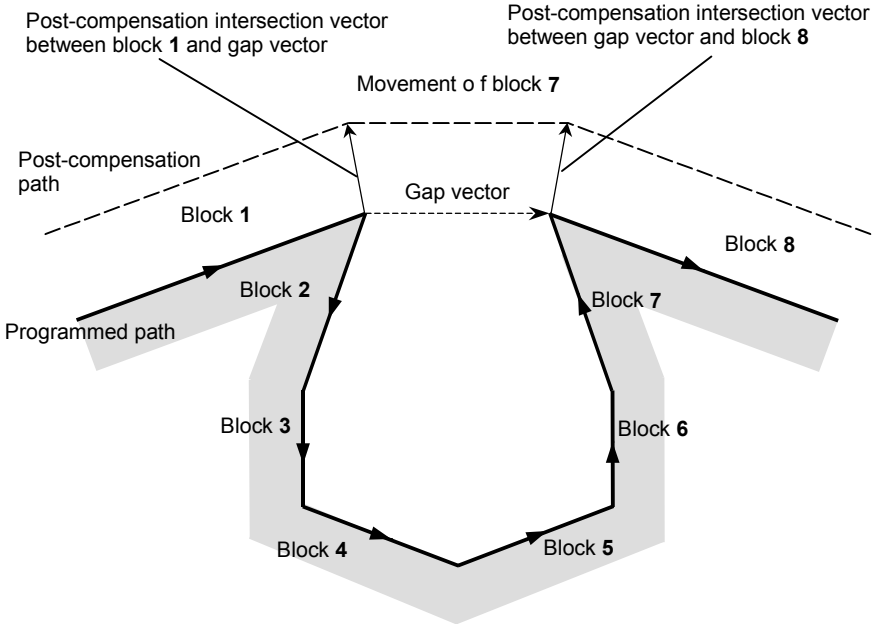
Overview

If a command is specified which satisfies the condition under which the interference check alarm function generates an interference alarm, this function suppresses the generation of the interference alarm, but causes a new compensation vector to be calculated as a path for avoiding interference, thereby continuing machining. For the path for avoiding interference, insufficient cutting occurs in comparison with the programmed path. In addition, depending on the specified figure, no path for avoiding interference can be determined or the path for avoiding interference may be judged dangerous. In such a case, an alarm stop will occur. For this reason, it is not always possible to avoid interference for all commands.

Explanation

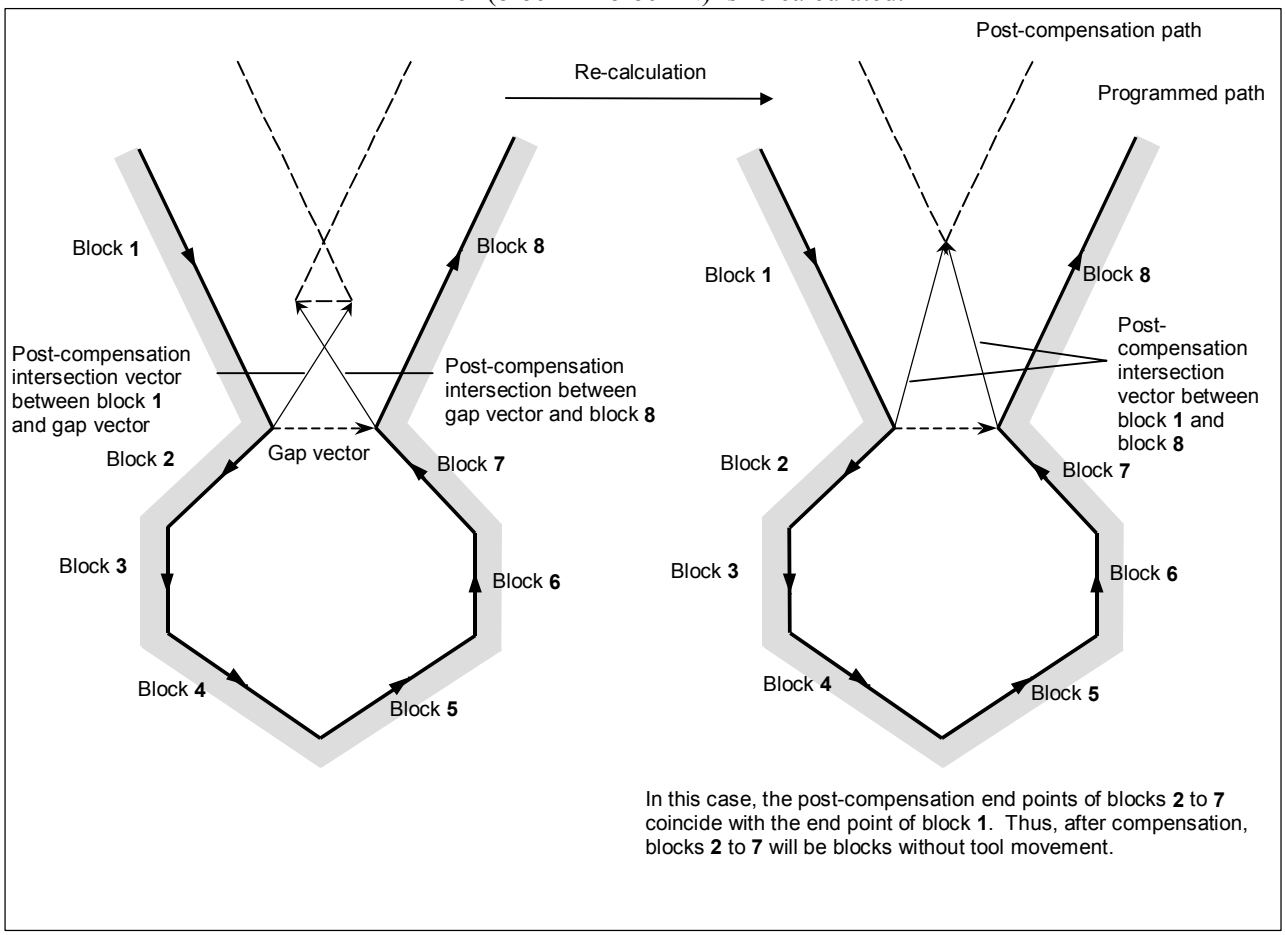
- Interference avoidance method

Let us consider a case in which an interference occurs between the compensation vector between (block 1 - block 2) and the compensation vector between (block N-1 - block N). The direction vector from the end point of block 1 to the end point of block N-1 is called a gap vector. At this time, a post-compensation intersection vector between (block 1 - gap vector) and a post-compensation intersection vector between (gap vector - block N) is determined, and a path connecting them is created.

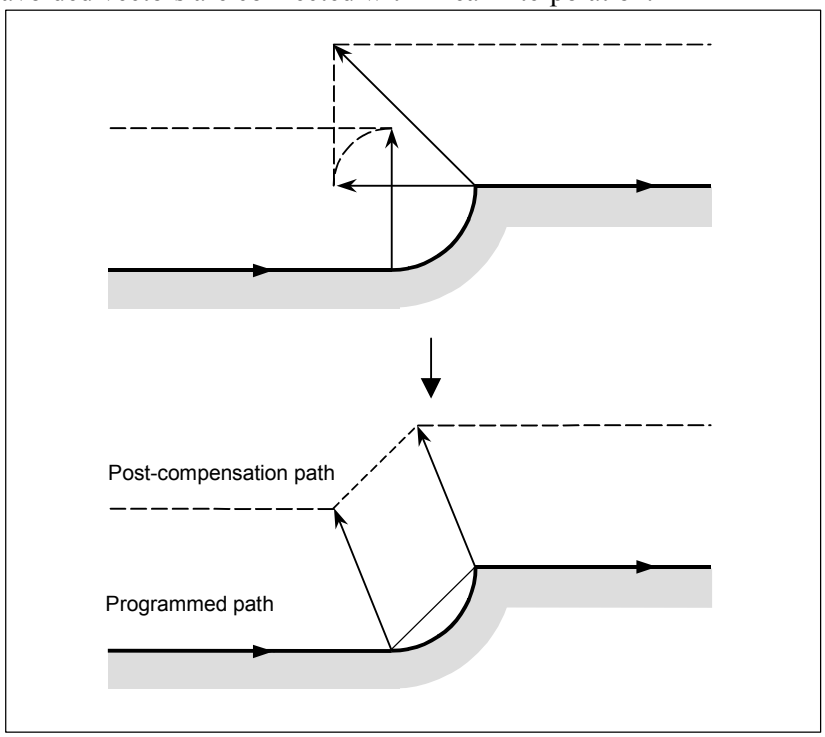


In this case, the post-compensation end points of blocks **2** to **6** coincide with the end point of block **1**. Thus, after compensation, blocks **2** to **6** will be blocks without tool movement.

If the post-compensation intersection vector of (block 1 - gap vector) and the post-compensation intersection vector of (gap vector - block N) further intersect, vector erasure is first performed in the same way as in "Interference between adjacent three blocks". If the last vectors that remains still intersects, the post-compensation intersection vector of (block 1 - block N) is re-calculated.

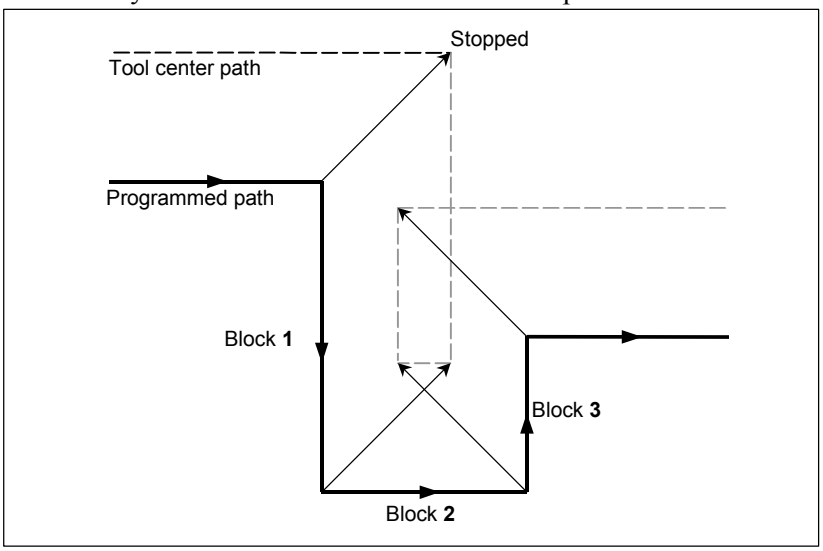


If the tool nose radius compensation value is greater than the radius of the specified arc as shown in the figure below, and a command is specified which results in compensation with respect to the inside of the arc, interference is avoided by performing intersection calculation with an arc command being assumed a linear one. In this case, avoided vectors are connected with linear interpolation.

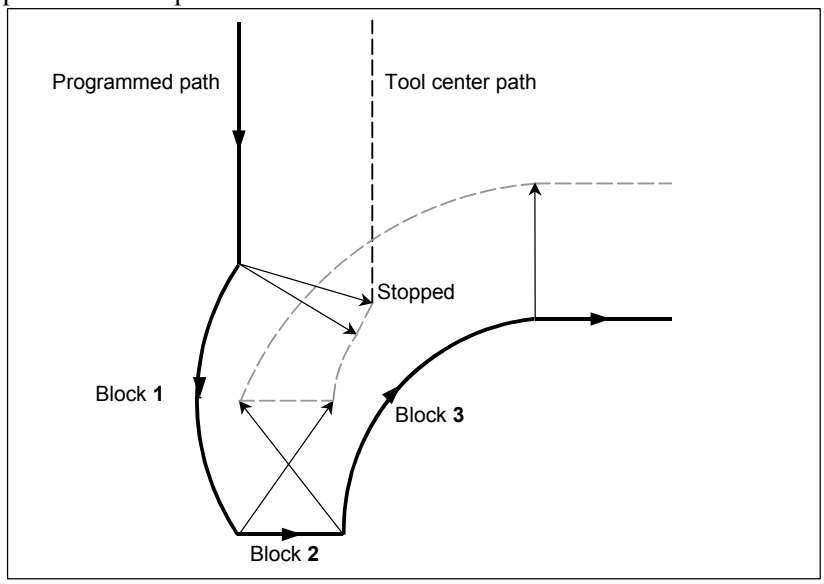


- If no interference avoidance vector exists

If the parallel pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are parallel to each other, no intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop.

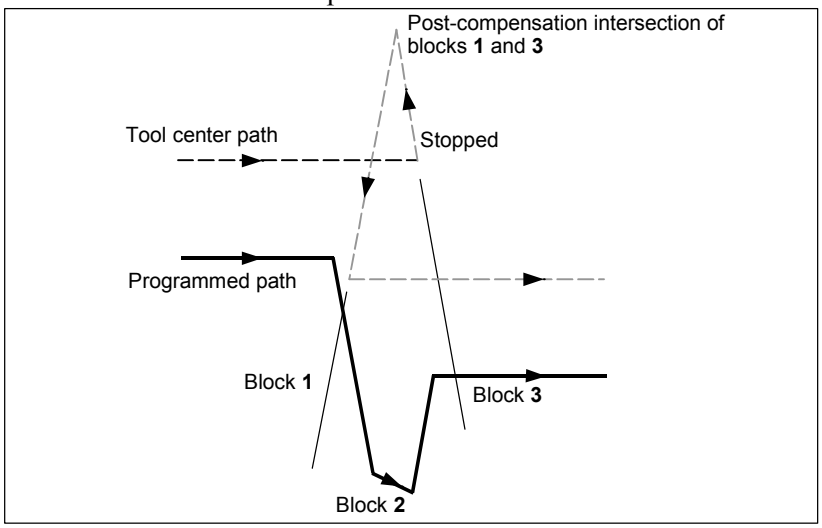


If the circular pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are circular, no post-compensation intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop, as in the previous example.

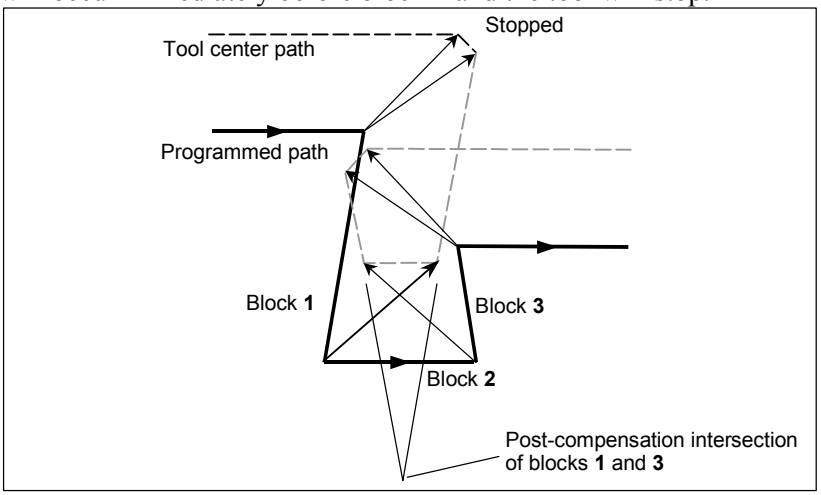


- If it is judged dangerous to avoid interference

If the acute-angle pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the movement direction of the post-avoidance path extremely differs from the previously specified direction. If the post-avoidance path extremely differs from that of the original command (90° or greater or 270° or less), interference avoidance operation is judged dangerous; an alarm will occur immediately before block 1 and the tool will stop.

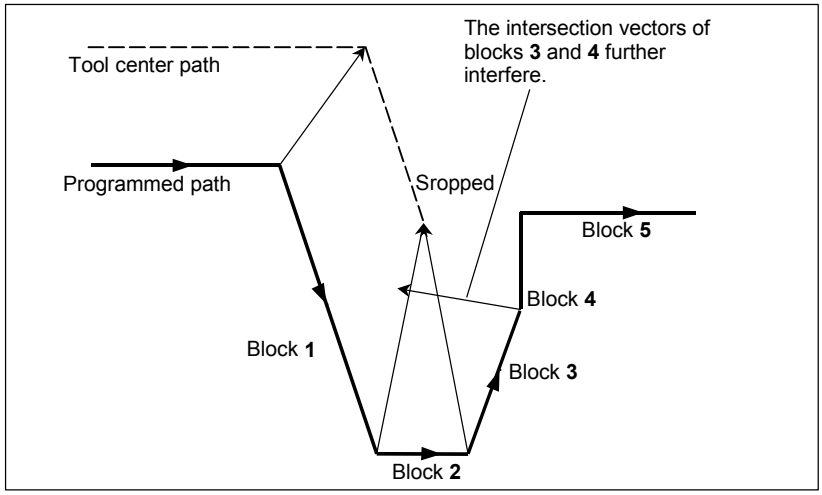


If a pocket in which the bottom is wider than the top, such as that shown in the figure, is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the relation between blocks 1 and 3 is judged an outer one, the post-avoidance path results in overcutting as compared with the original command. In such a case, interference avoidance operation is judge dangerous; an alarm will occur immediately before block 1 and the tool will stop.



- If further interference with an interference avoidance vector occurs

If the pocket shown in the figure is to be machined, if the number of blocks to read is 3, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, however, the end-point vector of block 3 that is to be calculated next further interferes with the previous interference avoidance vector. If a further interference occurs to the interference avoidance vector once created and output, the movement in the block will not be performed; an alarm will occur immediately before the block and the tool will stop.



NOTE

1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting parameter NAA (No. 19607#6) appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", however, it is not possible to avoid an alarm regardless of the setting of this parameter.

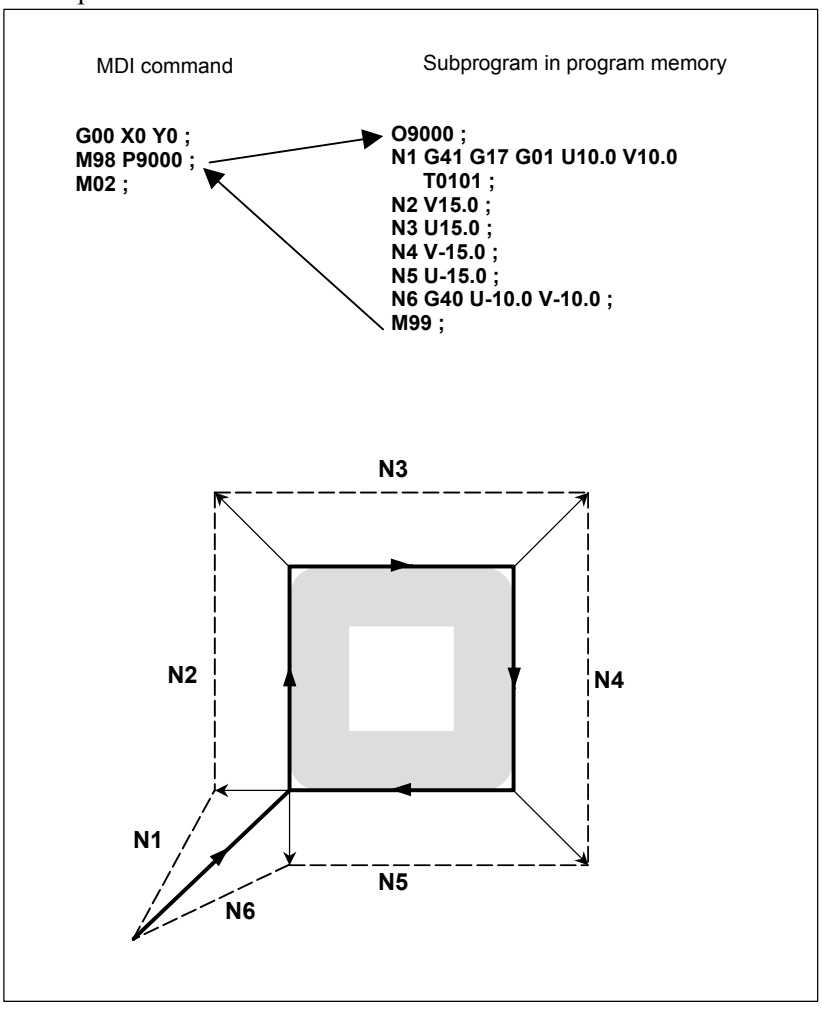
2 If a single block stop occurs during interference avoidance operation, and an operation is performed which differs from the original movement, such as manual intervention, MDI intervention, tool nose radius compensation value change, intersection calculation is performed with a new path. If such an operation is performed, therefore, an interference may occur again although interference avoidance has been performed once.

5.3.7 Tool Nose Radius Compensation for Input from MDI

Explanation

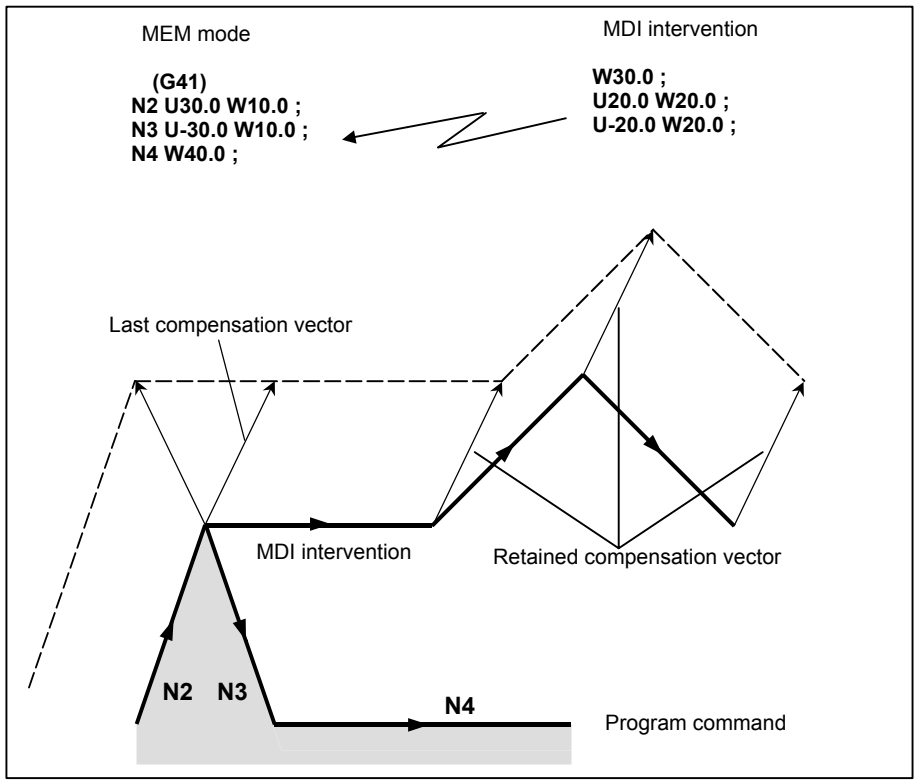
- MDI operation

During MDI operation, that is, if a program command is specified in MDI mode in the reset state to make a cycle start, intersection calculation is performed for compensation in the same way as in memory operation/DNC operation. Compensation is performed in the same way if a subprogram is called from program memory due to MDI operation.



- MDI intervention

If MDI intervention is performed, that is, if a single block stop is performed to enter the automatic operation stop state in the middle of memory operation, DNC operation, and the like, and a program command is specified in MDI mode to make a cycle start, tool nose radius compensation does not perform intersection calculation, retaining the last compensation vector before the intervention.



5.4 CORNER CIRCULAR INTERPOLATION (G39)

By specifying G39 in offset mode during tool nose radius compensation, corner circular interpolation can be performed. The radius of the corner circular interpolation equals the compensation value.

Format

In offset mode

G39 ;

or

G39 { **I_J_** / **I_K_** / **J_K_** } **;**

Explanation

- Corner circular interpolation

When the command indicated above is specified, corner circular interpolation in which the radius equals compensation value can be performed. G41 or G42 preceding the command determines whether the arc is clockwise or counterclockwise. G39 is a one-shot G code.

- G39 without I, J, or K

When G39; is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block.

- G39 with I, J, and K

When G39 is specified with I, J, and K, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the vector defined by the I, J, and K values.

Limitation

- Move command

In a block containing G39, no move command can be specified. Otherwise, an alarm will occur.

- Inner corner

In an inner corner block, G39 cannot be specified. Otherwise, overcutting will occur.

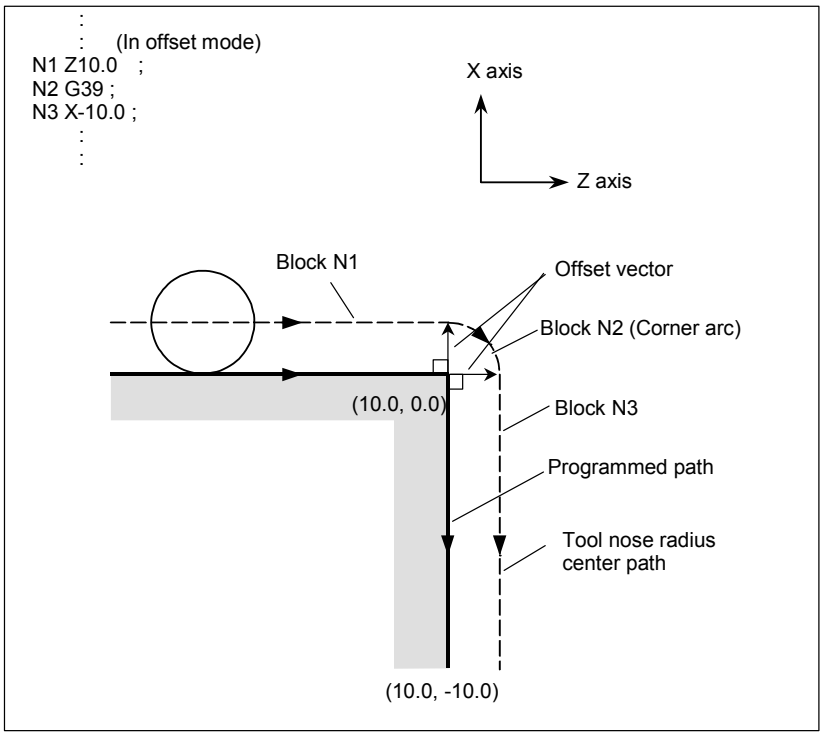
- Corner arc velocity

If a corner arc is specified with G39 in G00 mode, the corner arc block velocity will be that of the F command previously specified.

Example

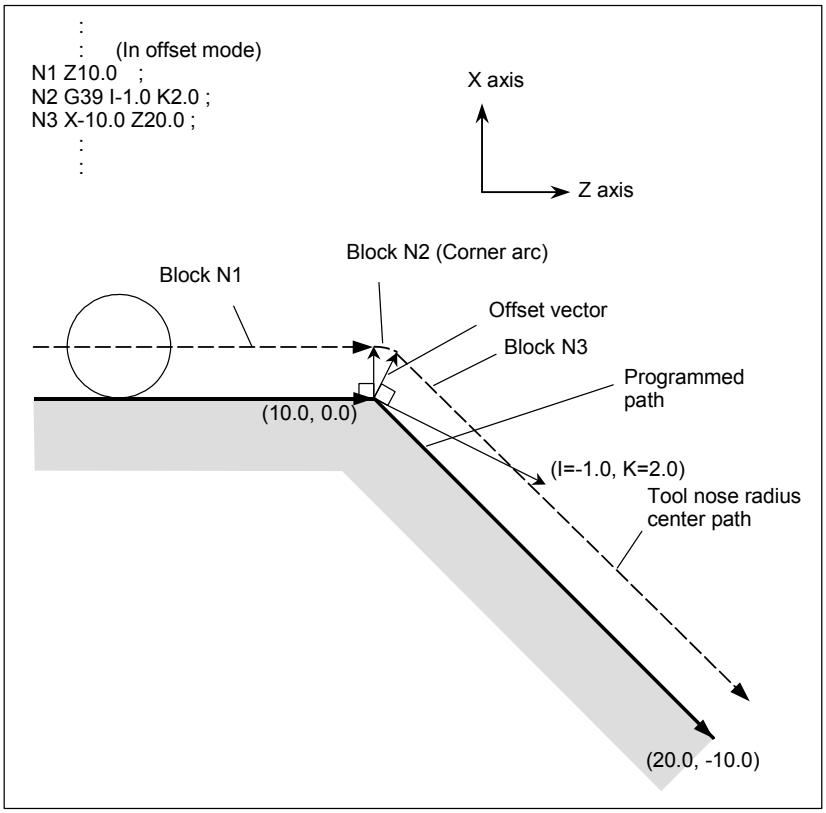
- G39 without I, J, or K

```
      :
      :   (In offset mode)
N1 Z10.0   ;
N2 G39 ;
N3 X-10.0 ;
      :
      :
```



- G39 with I, J, and K

```
      :
      :   (In offset mode)
N1 Z10.0   ;
N2 G39 I-1.0 K2.0 ;
N3 X-10.0 Z20.0 ;
      :
      :
```



5.5 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.

NOTE
To use automatic tool offset, set bit 7 (IGA) of parameter No. 6240 to 0.

Explanation

- Coordinate system

When moving the tool to a position for measurement, the coordinate system must be set in advance. (The workpiece 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 MEM mode :

G36 Xxa ; or G37 Zza ;

In this case, the measurement position should be xa or za (absolute programming).
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 xa or za specified in G36Xxa or G37Zza.

Offset value x = Current offset value x+(α-xa)
Offset value z = Current offset value z+(β-za)
xa : Programmed X-axis measurement point
za : 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 stating position toward the measurement position predicted by xa or za in G36 or G37, is feed at the rapid traverse rate across area A. Then the tool stops at point T (xa-γ or za-γ) and moves at the measurement feedrate set by parameter (No. 6241) 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 PS0080 is generated.

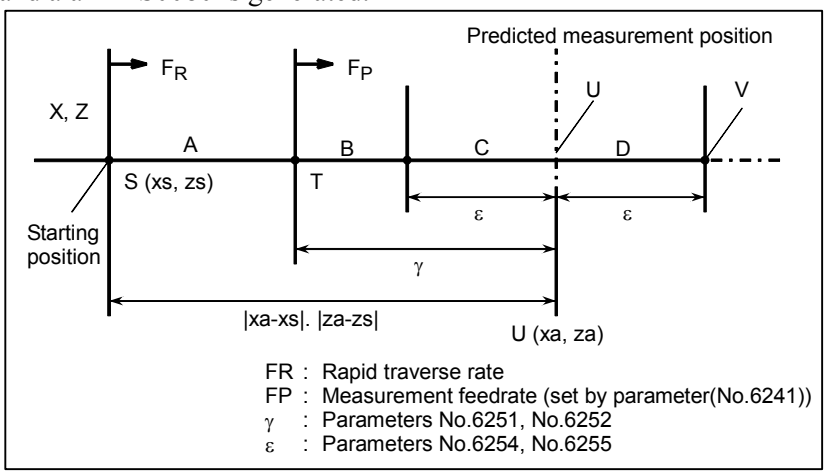
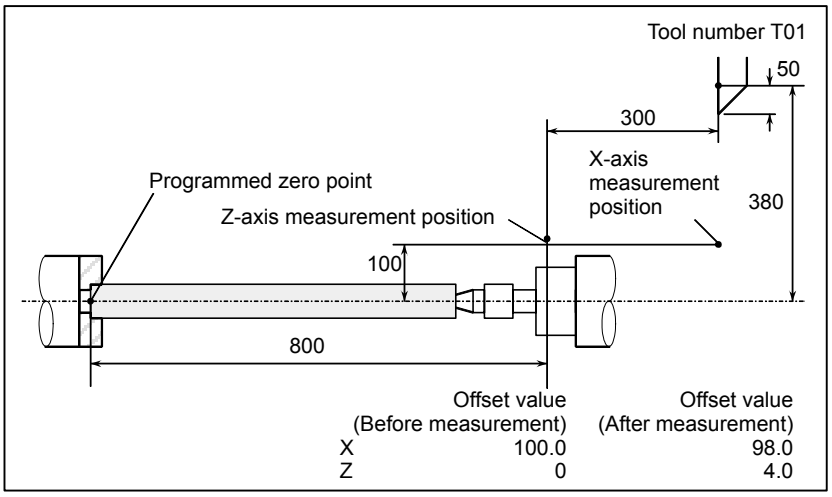


Fig. 5.5 (a) Feedrate and alarm

Example



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=-2.0mm.

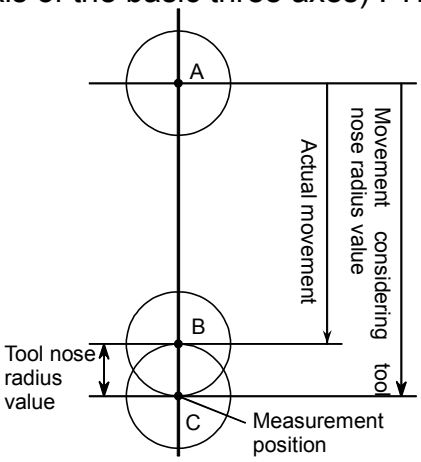
G00 X204.0 ; Refracts a little along the X axis.

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.6241, γ : No.6251, ε : No.6254) by machine tool builder. ε must be positive numbers so that $\gamma>\varepsilon$.
2 Cancel the tool nose radius compensation before G36, G37.
3 A delay or variation in detection of the measurement position arrival signal is 0 to 2 msec on the CNC side excluding the PMC side. Therefore, the measurement error is the sum of 2 msec and a delay or variation (including a delay or variation on the receiver side) in propagation of the measurement position arrival signal on the PMC side, multiplied by the feedrate set in parameter No. 6241.
4 A delay or variation in time after detection of the measurement position arrival signal until a feed stops is 0 to 8 msec. To calculate the amount of overrun, further consider a delay in acceleration/deceleration, servo delay, and delay on the PMC side.
5 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.
6 When tool nose radius compensation is enabled (bit 7 (NCR) of parameter No. 8136 is set to 0), the tool offset amount is calculated with considering the tool nose radius value. Make sure that tool nose radius value is set correctly.
(Condition under which the tool-nose radius compensation is considered)
For the X-axis (first axis of the basic three axes) : TIP=0/5/7
For the Z-axis (third axis of the basic three axes) : TIP=0/6/8
For the Y-axis (second axis of the basic three axes) : TIP=0



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 PS0081 is generated.
2 When a T code is specified in the same block as G36 or G37, alarm PS0082 is generated.

6 MEMORY OPERATION USING Series 10/11 FORMAT

By setting the setting-related parameter (bit 1 of parameter No. 0001), a program created in the Series 10/11 program format can be registered in memory for memory operation. Memory operation are possible for the functions which use the same program format as that for the Series 10/11 as well as for the following functions which use a different program format:

- Subprogram calling
- Canned cycle
- Multiple repetitive canned cycle
- Canned cycle for drilling

NOTE
Memory operation are possible only for the functions available in this CNC.

Chapter 6, "MEMORY OPERATION BY Series 10/11 FORMAT", consists of the following sections:

6.1 ADDRESSES AND SPECIFIABLE VALUE RANGE FOR Series 10/11 PROGRAM FORMAT.....249
6.2 SUBPROGRAM CALLING.....249
6.3 CANNED CYCLE.....250
6.4 MULTIPLE REPETITIVE CANNED CYCLE.....272
6.5 CANNED CYCLE FOR DRILLING.....315

6.1 ADDRESSES AND SPECIFIABLE VALUE RANGE FOR Series 10/11 PROGRAM FORMAT

Some addresses which cannot be used for the this CNC can be used in the Series 10/11 program format. The specifiable value range for the Series 10/11 program format is basically the same as that for the this CNC. Sections II-6.2 to II-6.5 describe the addresses with a different specifiable value range. If a value out of the specifiable value range is specified, an alarm is issued.

6.2 SUBPROGRAM CALLING

Format

M98 Pxxxx Lyyyy ;
P : Subprogram number
L : Repetition count

Explanation

- Address

Address L cannot be used in this CNC tape format but can be used in the Series 10/11 format.

- Subprogram number

The specifiable value range is the same as that for this CNC (1 to 9999).
If a value of more than four digits is specified, the last four digits are assumed as the subprogram number.

- 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.

6.3 CANNED CYCLE

Explanation

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

NOTE

1 Explanatory figures in this section use the ZX plane as the selected plane, diameter programming for the X-axis, and radius programming for the Z-axis. When radius programming is used for the X-axis, change U/2 to U and X/2 to X.

2 A canned cycle can be performed on any plane (including parallel axes for plane definition). When G code system A is used, however, U, V, and W cannot be set as a parallel axis.

3 The direction of the length means the direction of the first axis on the plane as follows:
ZX plane: Z-axis direction
YZ plane: Y-axis direction
XY plane: X-axis direction

4 The direction of the end face means the direction of the second axis on the plane as follows:
ZX plane: X-axis direction
YZ plane: Z-axis direction
XY plane: Y-axis direction

6.3.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

This cycle performs straight or taper cutting in the direction of the length.

6.3.1.1 Straight cutting cycle

Format

G90X(U)_Z(W)_F_;

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

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

F_ : Cutting feedrate

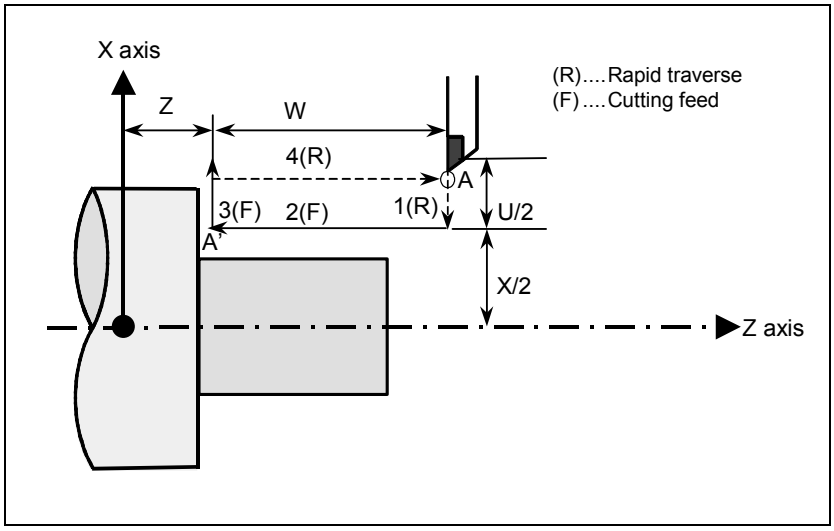


Fig. 6.3.1 (a) Straight cutting cycle

Explanation

- Operations

A straight cutting cycle performs four operations:

(1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

(2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the length.)

(3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in cutting feed.

(4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

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

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

6.3.1.2 Taper cutting cycle

Format

ZpXp-plane
G90 X(U)_ Z(W)_ I_ F_ ;
YpZp-plane
G90 Y(V)_ Z(W)_ K_ F_ ;
XpYp-plane
G90 X(U)_ Y(V)_ J_ F_ ;

X_,Y_,Z_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length
U_,V_,W_ : Travel distance to the cutting end point (point A' in the figure below) in the direction of the length
I_,J_,K_ : Taper amount (I in the figure below)
F_ : Cutting feedrate

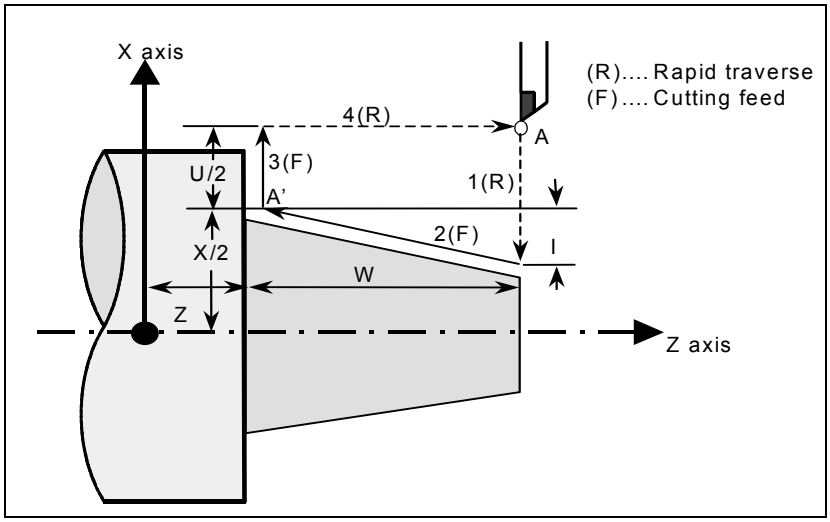


Fig. 6.3.1 (b) Taper cutting cycle

Explanation

Address I, J, or K for specifying a taper varies with the plane selected. The figure of a taper is determined by the coordinates of the cutting end point (A') in the direction of the length and the sign of the taper amount (address I, J, or K). For the cycle in the figure above, a minus sign is added to the taper amount.

NOTE
The increment system of address I, J, or K for specifying a taper depends on the increment system for the reference axis. Specify a radius value at I, J, or K.

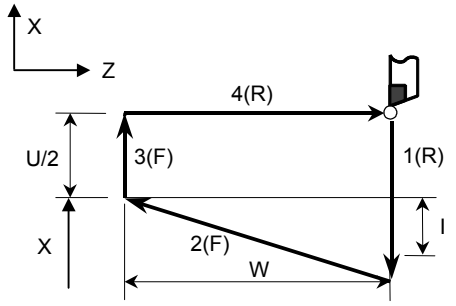
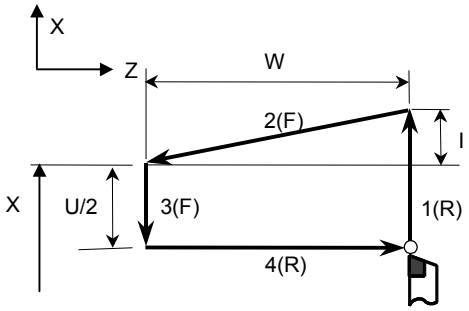
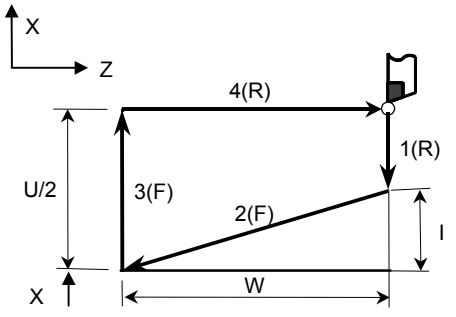
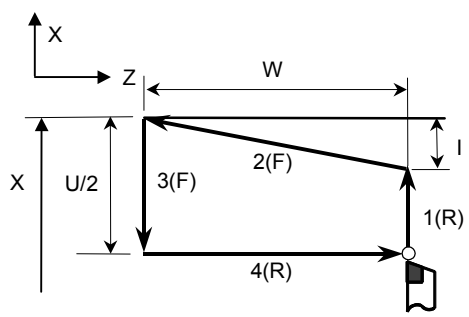
- Operations

A taper cutting cycle performs the same four operations as a straight cutting cycle.
However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
Operations 2, 3, and 4 after operation 1 are the same as for a straight cutting cycle.

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

- Relationship between the sign of the taper amount and tool path

The tool path is determined according to the relationship between the sign of the taper amount (address I, J, or K) and the cutting end point in the direction of the length in the absolute or incremental programming as follows.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, I < 0	**2. U > 0, W < 0, I > 0**
	
3. U < 0, W < 0, I > 0 at |I|≤|U/2|	**4. U > 0, W < 0, I < 0 at |I|≤|U/2|**
	

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

6.3.2 Threading Cycle (G92)

6.3.2.1 Straight threading cycle

Format

G92 X(U)_Z(W)_F_Q_;

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

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

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

F_ : Thread lead (L in the figure below)

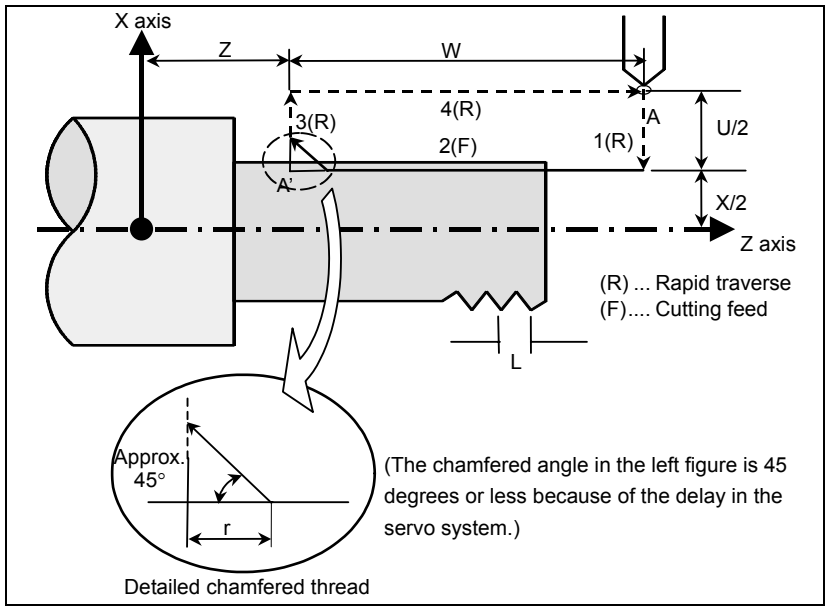


Fig. 6.3.2 (c) Straight threading

Explanation

The ranges of thread leads and restrictions related to the spindle speed are the same as for threading with G32.

- Operations

A straight threading cycle performs four operations:

(1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

(2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. At this time, thread chamfering is performed.

(3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (Retraction after chamfering)

(4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

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

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

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

- Acceleration/deceleration for threading after interpolation

Acceleration/deceleration for threading after interpolation is acceleration/deceleration of exponential interpolation type. By setting bit 5 (THLx) of parameter No. 1610, the same acceleration/deceleration as for cutting feed can be selected. (The settings of bit 0 (CTLx) of parameter No. 1610 are followed.) However, as a time constant and FL feedrate, the settings of parameter No. 1626 and No. 1627 for the threading cycle are used.

- Time constant and FL feedrate for threading

The time constant for acceleration/deceleration after interpolation for threading specified in parameter No. 1626 and the FL feedrate specified in parameter No. 1627 are used.

- Thread chamfering

Thread chamfering can be performed. A signal from the machine tool, initiates thread chamfering. The chamfering distance r 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.)
A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.
For thread chamfering, the same type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate as for threading are used.

NOTE
Common parameters for specifying the amount and angle of thread chamfering are used for this cycle and threading cycle with G76.

- Retraction after chamfering

The following table lists the feedrate, type of acceleration/deceleration after interpolation, and time constant of retraction after chamfering.

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

By setting bit 4 (ROC) of parameter No. 1403 to 1, rapid traverse override can be disabled for the feedrate of retraction after chamfering.

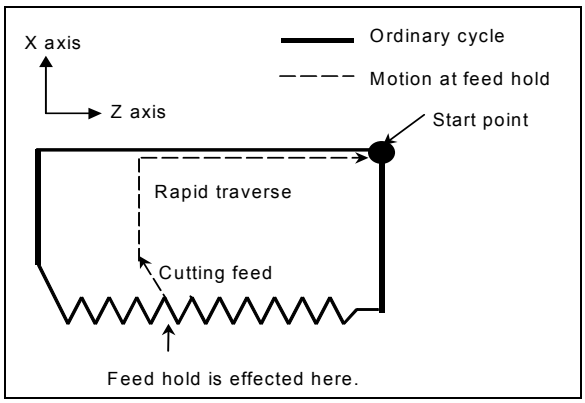
NOTE
During retraction, the machine does not stop with an override of 0% for the cutting feedrate regardless of the setting of bit 4 (RF0) of parameter No. 1401.

- Shifting the start angle

Address Q can be used to shift the threading start angle.
The start angle (Q) increment is 0.001 degrees and the valid setting range is between 0 and 360 degrees. No decimal point can be specified.

Feed hold in a threading cycle (threading cycle retract)

Feed hold may be applied during threading (operation 2). In this case, the tool immediately retracts with chamfering and returns to the start point on the second axis (X-axis), then the first axis (Z-axis) on the plane.



The chamfered angle is the same as that at the end point.

> ⚠ **CAUTION**
> Another feed hold cannot be made during retreat.

- Inch threading

Inch threading specified with address E is allowed.

6.3.2.2 Taper threading cycle

Format

ZpXp-plane
G92 X(U)_ Z(W)_ I_ F_ Q_ ;
YpZp-plane
G92 Y(V)_ Z(W)_ K_ F_ Q_ ;
XpYp-plane
G92 X(U)_ Y(V)_ J_ F_ Q_ ;

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

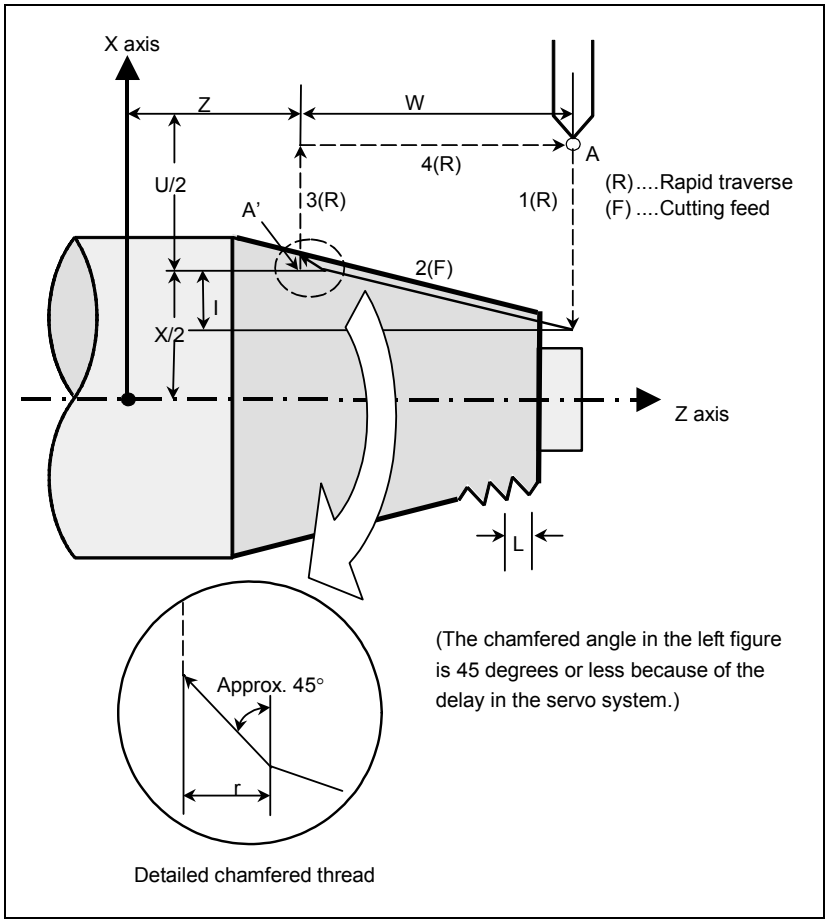


Fig. 6.3.2 (d) Taper threading cycle

Explanation

The ranges of thread leads and restrictions related to the spindle speed are the same as for threading with G32.
The figure of a taper is determined by the coordinates of the cutting end point (A') in the direction of the length and the sign of the taper amount (address I, J, or K). For the cycle in the figure above, a minus sign is added to the taper amount.

NOTE
The increment system of address I, J, or K for specifying a taper depends on the increment system for the reference axis. Specify a radius value at I, J, or K.

- Operations

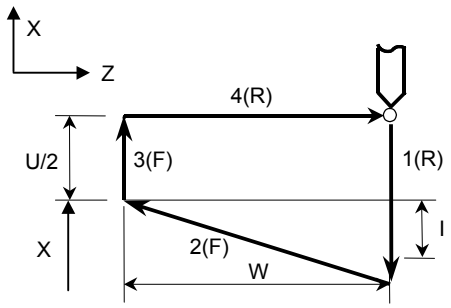
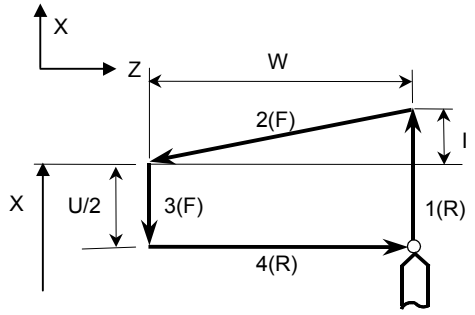
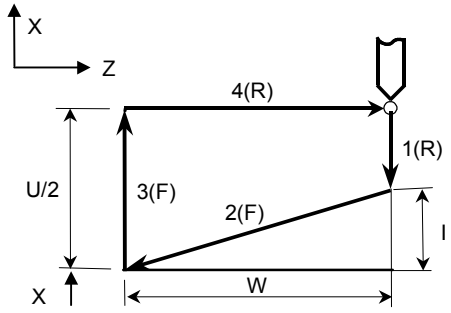
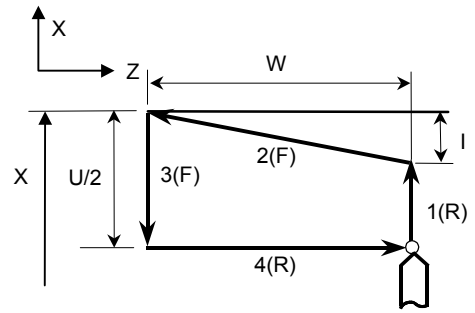
A taper threading cycle performs the same four operations as a straight threading cycle.
However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
Operations 2, 3, and 4 after operation 1 are the same as for a straight threading cycle.

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

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

- Relationship between the sign of the taper amount and tool path

The tool path is determined according to the relationship between the sign of the taper amount (address I, J, or K) and the cutting end point in the direction of the length in the absolute or incremental programming as follows.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, I < 0	**2. U > 0, W < 0, I > 0**
	
3. U < 0, W < 0, I > 0 at |I|≤|U/2|	**4. U > 0, W < 0, I < 0 at |I|≤|U/2|**
	

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

- Acceleration/deceleration for threading after interpolation
- Time constant and FL feedrate for threading
- Thread chamfering
- Retraction after chamfering
- Shifting the start angle
- Threading cycle retract
- Inch threading

See the pages on which a straight threading cycle is explained.

6.3.3 End Face Turning Cycle (G94)

6.3.3.1 Face cutting cycle

Format

G94 X(U)_Z(W)_F_;

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

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

F_ : Cutting feedrate

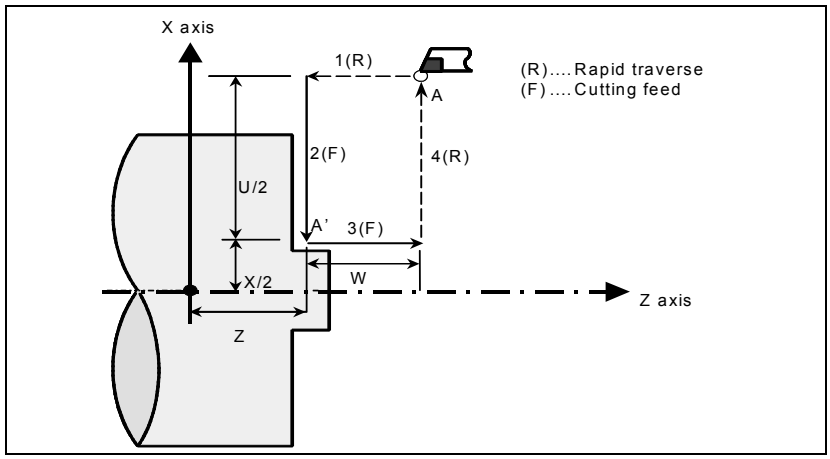


Fig. 6.3.3 (e) Face cutting cycle

Explanation

- Operations

A face cutting cycle performs four operations:

(1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.

(2) Operation 2 moves the tool to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the end face.)

(3) Operation 3 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in cutting feed.

(4) Operation 4 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

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

- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

6.3.3.2 Taper cutting cycle

Format

ZpXp-plane
G94 X(U)_ Z(W)_ K _ F_ ;
YpZp-plane
G94 Y(V)_ Z(W)_ J _ F_ ;
XpYp-plane
G94 X(U)_ Y(V)_ I _ F_ ;

X_,Y_,Z_	:	Coordinates of the cutting end point (point A' in the figure below) in the direction of the end face
U_,V_,W_	:	Travel distance to the cutting end point (point A' in the figure below) in the direction of the end face
I_,J_,K_	:	Taper amount (K in the figure below)
F_	:	Cutting feedrate

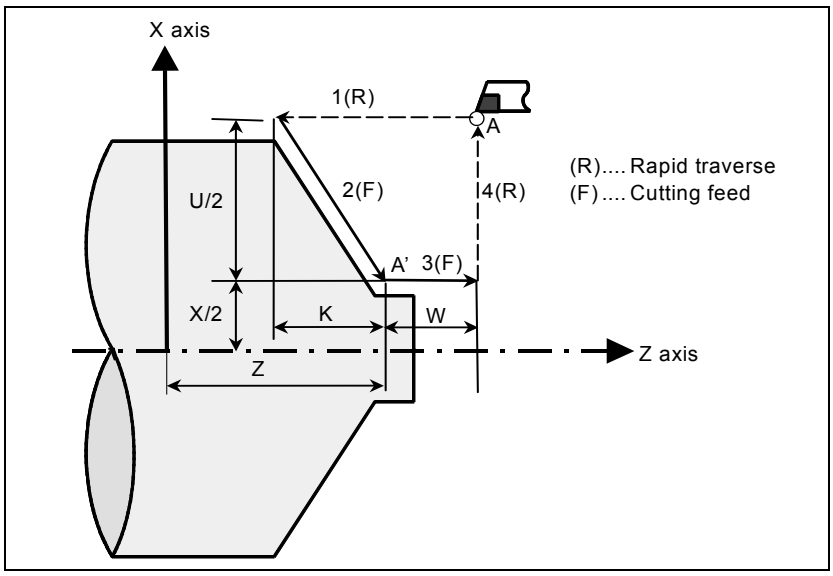


Fig. 6.3.3 (f) Taper cutting cycle

Explanation

The figure of a taper is determined by the coordinates of the cutting end point (A') in the direction of the end face and the sign of the taper amount (address I, J, or K). For the cycle in the figure above, a minus sign is added to the taper amount.

NOTE
The increment system of address I, J, or K for specifying a taper depends on the increment system for the reference axis. Specify a radius value at I, J, or K.

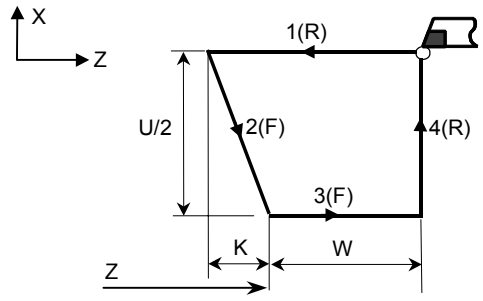
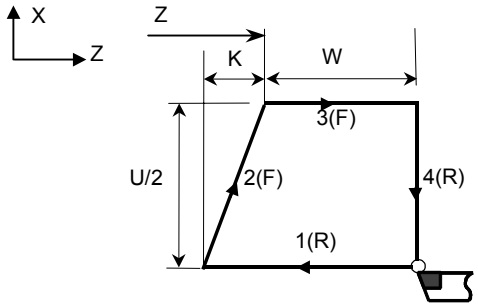
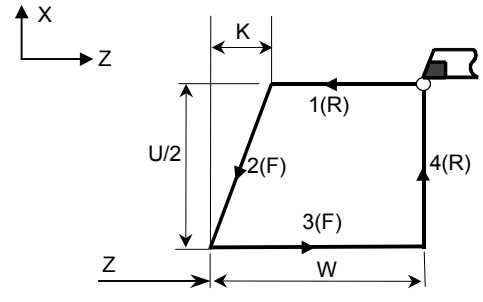
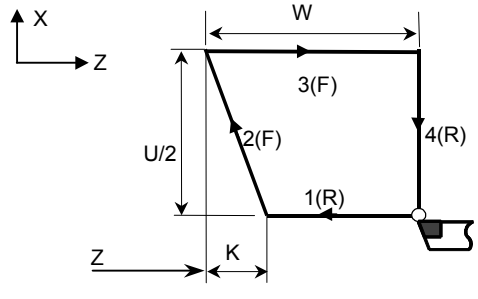
- Operations

A taper cutting cycle performs the same four operations as a face cutting cycle.
However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.
Operations 2, 3, and 4 after operation 1 are the same as for a face cutting cycle.

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

- Relationship between the sign of the taper amount and tool path

The tool path is determined according to the relationship between the sign of the taper amount (address I, J, or K) and the cutting end point in the direction of the end face in the absolute or incremental programming as follows.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, K < 0	**2. U > 0, W < 0, K > 0**
	
3. U < 0, W < 0, K > 0 at |K|≤|W|	**4. U > 0, W < 0, K < 0 at |K|≤|W|**
	

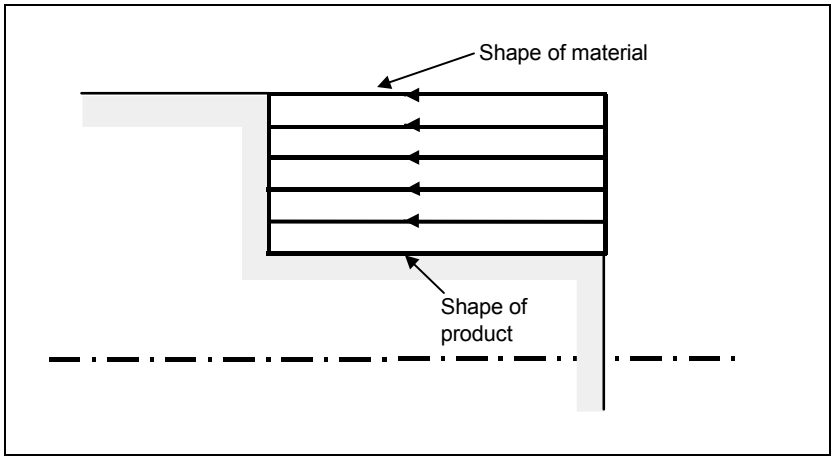
- Canceling the mode

To cancel the canned cycle mode, specify a group 01 G code other than G90, G92, or G94.

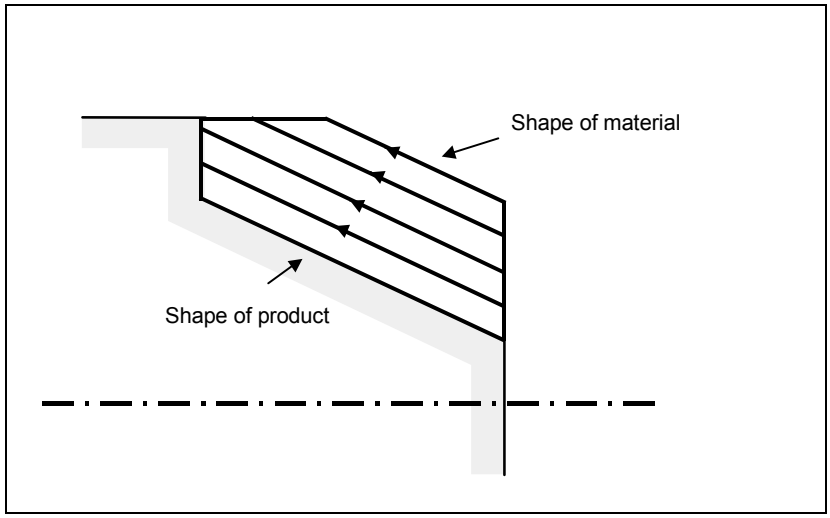
6.3.4 How to Use Canned Cycles

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

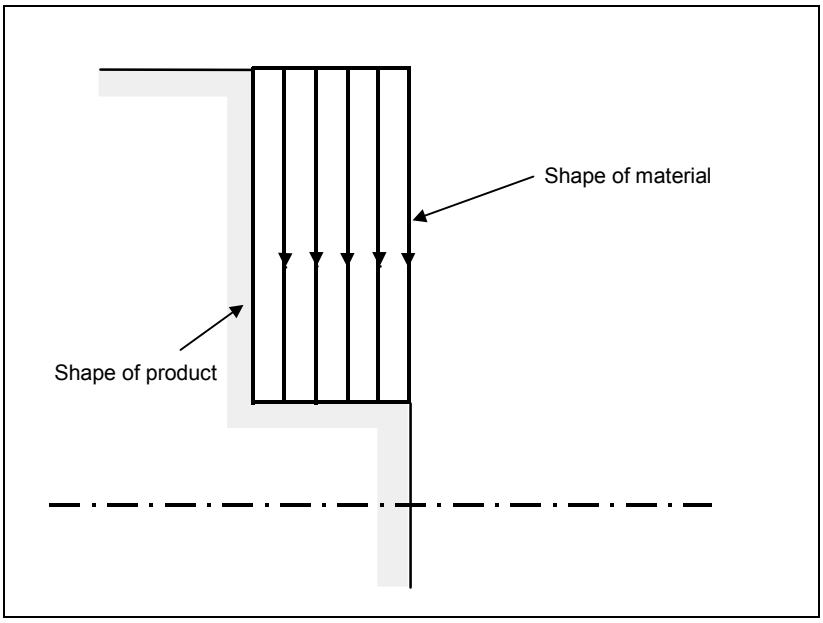
- Straight cutting cycle (G90)



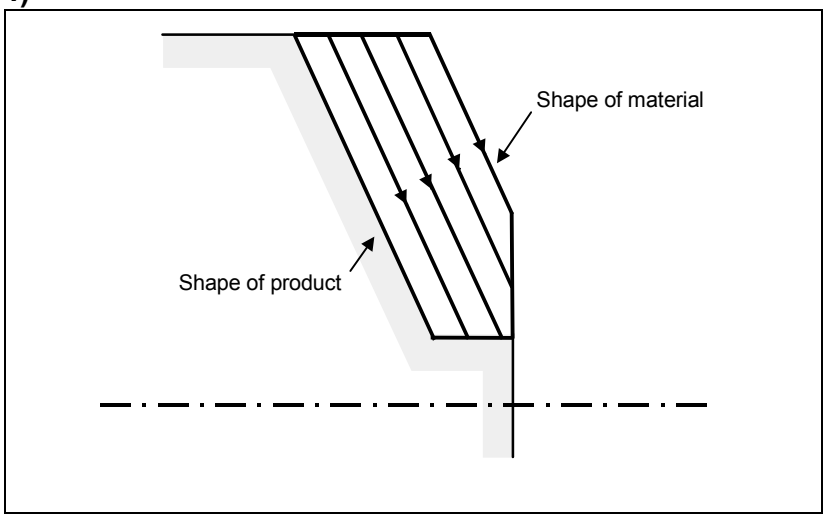
- Taper cutting cycle (G90)



- Face cutting cycle (G94)



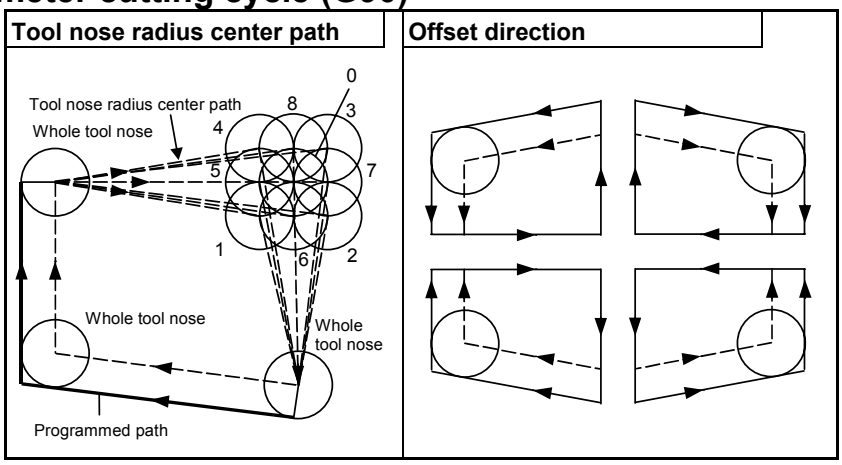
- Face taper cutting cycle (G94)



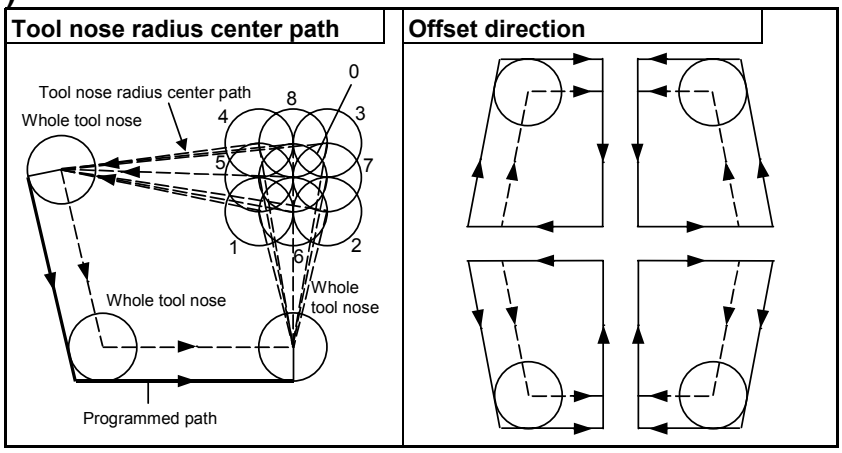
6.3.5 Canned Cycle and Tool Nose Radius Compensation

When tool nose radius compensation is applied, the tool nose center path and offset direction are as shown below. At the start point of a cycle, the offset vector is canceled. Offset start-up is performed for the movement from the start point of the cycle. The offset vector is temporarily canceled again at the return to the cycle start point and offset is applied again according to the next move command. The offset direction is determined depending of the cutting pattern regardless of the G41 or G42 mode.

Outer diameter/internal diameter cutting cycle (G90)



End face cutting cycle (G94)



Threading cycle (G92)

Tool nose radius compensation cannot be applied.

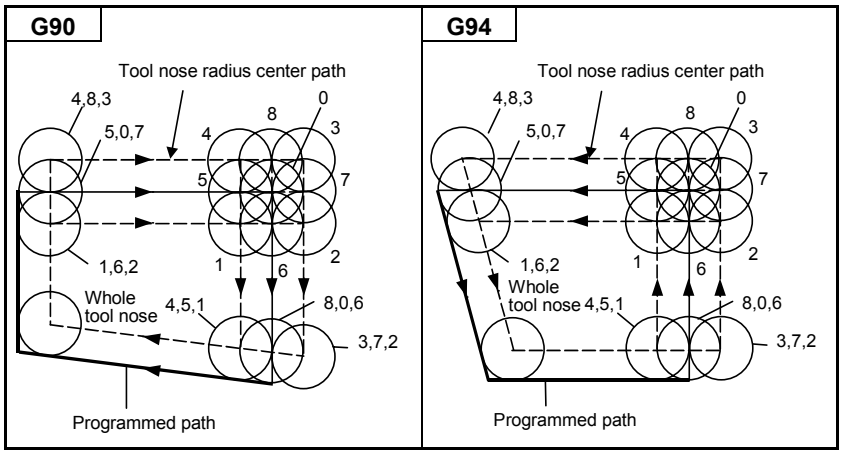
Differences between this CNC and the Series 0*i*-C

NOTE

This CNC is the same as the Series 0*i*-C in the offset direction, but differs from the series in the tool nose radius center path.

- For this CNC
  Cycle operations of a canned cycle are replaced with G00 or G01. In the first block to move the tool from the start point, start-up is performed. In the last block to return the tool to the start point, offset is canceled.
- For the Series 0*i*-C
  This series differs from this CNC in operations in the block to move the tool from the start point and the last block to return it to the start point. For details, refer to "Series 0*i*-C Operator's Manual."

How compensation is applied for the Series 0*i*-C



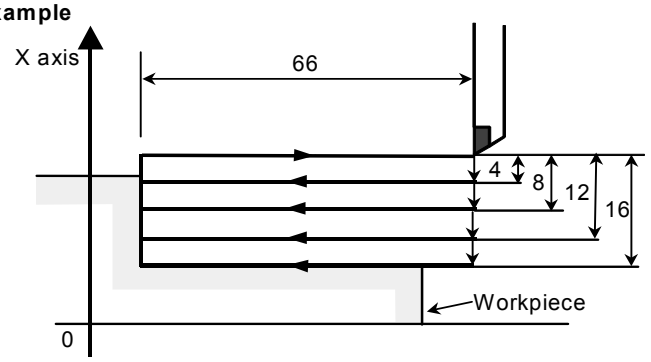
6.3.6 Restrictions on Canned Cycles

Limitation

- Modal

Since data items X (U), Z (W), and R in a canned cycle are modal values common to G90, G92, and G94. For this reason, if a new X (U), Z (W), or R value is not specified, the previously specified value is effective.
Thus, when the travel distance along the Z-axis does not vary as shown in the program example below, a canned cycle can be repeated only by specifying the travel distance along the X-axis.

Example



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;
```

The modal values common to canned cycles are cleared when a one-shot G code other than G04 is specified.
Since the canned cycle mode is not canceled by specifying a one-shot G code, a canned cycle can be performed again by specifying modal values. If no modal values are specified, no cycle operations are performed.
When G04 is specified, G04 is executed and no canned cycle is performed.

- Block in which no move command is specified

In a block in which no move command is specified in the canned cycle mode, a canned cycle is also performed. For example, a block containing only EOB or a block in which none of the M, S, and T codes, and move commands are specified is of this type of block. When an M, S, or T code is specified in the canned cycle mode, the corresponding M, S, or T function is executed together with the canned cycle. If this is inconvenient, specify a group 01 G code (G00 or G01) other than G90, G92, or G94 to cancel the canned cycle mode, and specify an M, S, or T code, as in the program example below. After the corresponding M, S, or T function has been executed, specify the canned cycle again.

Example

```
N003 T0101;
   :
   :
N010 G90 X20.0 Z10.0 F0.2;
N011 G00 T0202;              ← Cancels the canned cycle mode.
N012 G90 X20.5 Z10.0;
```

- Plane selection command

Specify a plane selection command (G17, G18, or G19) before setting a canned cycle or specify it in the block in which the first canned cycle is specified.
If a plane selection command is specified in the canned cycle mode, the command is executed, but the modal values common to canned cycles are cleared.
If an axis which is not on the selected plane is specified, alarm PS0330 is issued.

- Parallel axis

When G code system A is used, U, V, and W cannot be specified as a parallel axis.

- Reset

If a reset operation is performed during execution of a canned cycle when any of the following states for holding a modal G code of group 01 is set, the modal G code of group 01 is replaced with the G01 mode:

- Reset state (bit 6 (CLR) of parameter No. 3402 = 0)
- Cleared state (bit 6 (CLR) of parameter No. 3402 = 1) and state where the modal G code of group 01 is held at reset time (bit 1 (C01) of parameter No. 3406 = 1)

Example of operation)

If a reset is made during execution of a canned cycle (X0 block) and the X20.Z1. command is executed, linear interpolation (G01) is performed instead of the canned cycle.

6.4 MULTIPLE REPETITIVE CANNED CYCLE

The multiple repetitive canned cycle is canned cycles to make CNC programming easy. For instance, the data of the finish workpiece shape describes the tool path for rough machining. And also, a canned cycles for the threading is available.

NOTE
1 Explanatory figures in this section use the ZX plane as the selected plane, diameter programming for the X-axis, and radius programming for the Z-axis. When radius programming is used for the X-axis, change U/2 to U and X/2 to X.
2 A multiple repetitive canned cycle can be performed on any plane (including parallel axes for plane definition). When G code system A is used, however, U, V, and W cannot be set as a parallel axis.

6.4.1 Stock Removal in Turning (G71)

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

Format

ZpXp plane

G71 P(ns) Q(nf) U(Δu) W(Δw) I(Δi) K(Δk) D(Δd) F(f) S(s) T(t);

N (ns) ;
...
N (nf) ;

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

YpZp plane

G71 P(ns) Q(nf) V(Δw) W(Δu) J(Δk) K(Δi) D(Δd) F(f) S(s) T(t);
N (ns) ;
...
N (nf) ;

XpYp plane

G71 P(ns) Q(nf) U(Δw) V(Δu) I(Δk) J(Δi) D(Δd) F(f) S(s) T(t);
N (ns) ;
...
N (nf) ;

Δd : Depth of cut
The cutting direction depends on the direction AA'.
ns : Sequence number of the first block for the program of finishing shape.
nf : Sequence number of the last block for the program of finishing shape.
Δu : Distance of the finishing allowance in the direction of the second axis on the plane (X-axis for the ZX plane)
Δw : Distance of the finishing allowance in the direction of the first axis on the plane (Z-axis for the ZX plane)
Δi : Distance of the finishing allowance of the roughing in the direction of the second axis on the plane (X-axis for the ZX plane)
Δk : Distance of the finishing allowance of the roughing in the direction of the first axis on the plane (Z-axis for the ZX plane)
f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

NOTE
Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

	Unit	Diameter/radius programming	Sign	Decimal point input
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed
Δi	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed

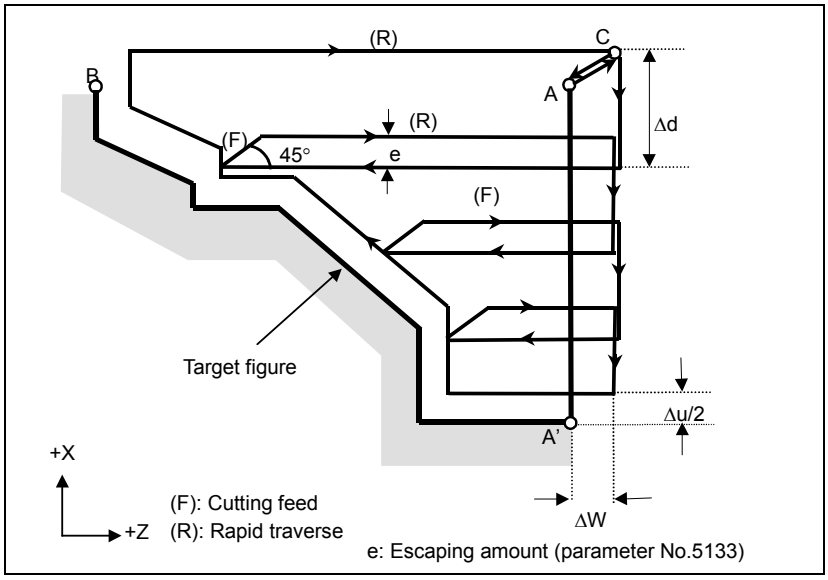


Fig. 6.4.1 (a) Cutting path of an outer surface rough cutting cycle without rough cutting finishing allowance (type I)

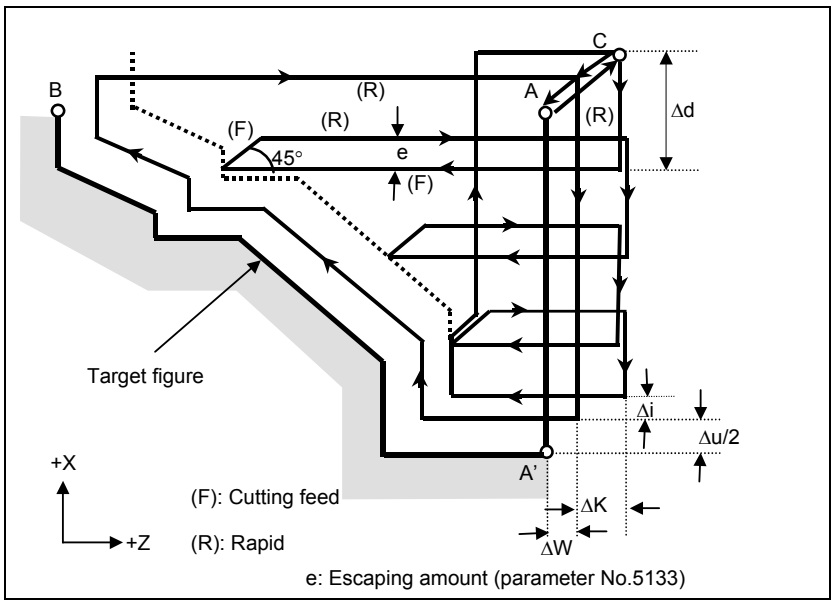


Fig. 6.4.1 (b) Cutting path of an outer surface rough cutting cycle with rough cutting finishing allowance (type I)

Explanation

- Operations

If a target figure passing through A, A', and B in this order is given by the program, a workpiece is cut away by depth of cut Δd at a time. The machining path varies as follows depending on whether the rough machining finishing allowance is specified.

(1) When the rough cutting finishing allowance is not specified
Cutting is performed by depth of cut Δd with finishing allowances Δu/2 and Δw left, and rough cutting as finishing is performed according to the target figure program after the last machining.

(2) When the rough cutting finishing allowance is specified
Cutting is performed by depth of cut Δd with cutting allowances Δu/2+Δi and Δw+Δk left, and the tool returns to the start point (A) after the last cutting is performed. Then, rough machining as finishing is performed along the target figure to remove cutting allowances Δi and Δk.

Upon completion of rough machining as finishing, the block next to the sequence block specified by Q is executed.

NOTE
1 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
2 When the constant surface speed control function is enabled (bit 0 (SSC) of parameter No. 8133 is set to 1), the G96 or G97 command specified in the move command between points A and B is ignored. If you want to enable the G96 or G97 command, specify the command in the G71 or previous block.

- Escaping amount (e)

The escaping amount (e) is set in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5133	Depends on the increment system for the reference axis.	Radius programming	Not required

- Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the first axis on the plane (Z-axis for the ZX plane). At this time, the signs of the finishing allowances of Δu and Δw are as follows:

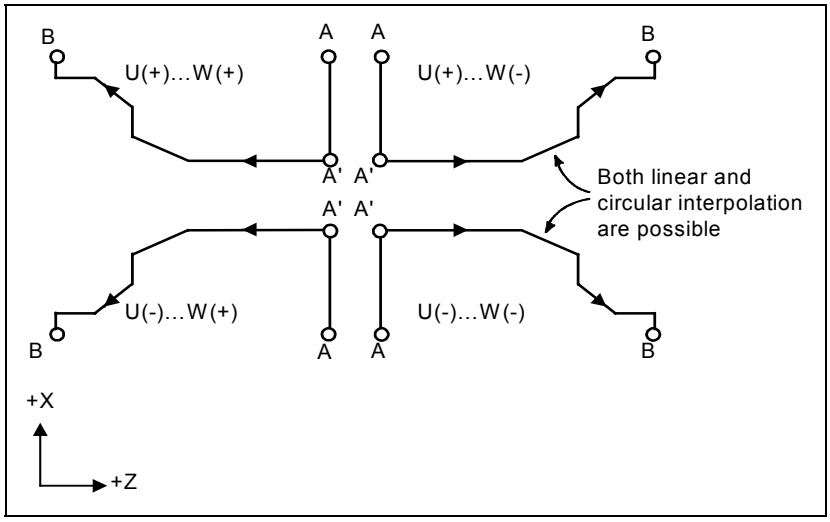


Fig. 6.4.1 (c) Four target figure patterns

Limitation

(1) For U(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For U(-), a figure for which a position lower than the cycle start point is specified cannot be machined.

(2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.

(3) For type II, the figure must show monotone increase or decrease along the first axis on the plane.

Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.
When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.
In this start block, also select type I or II.

Check functions

During cycle operation, whether the target figure shows monotone increase or decrease is always checked.

NOTE
When tool nose radius compensation is applied, the target figure to which compensation is applied is checked.

The following checks can also be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.
Checks the target figure before cycle operation. (Also checks that a block with the sequence number specified at address Q is contained.)	Enabled when bit 2 (FCK) of parameter No. 5104 is set to 1.

- Types I and II Selection of type I or II

For G71, there are types I and II.
When the target figure has pockets, be sure to use type II.
Escaping operation after rough cutting in the direction of the first axis on the plane (Z-axis for the ZX plane) differs between types I and II. With type I, the tool escapes to the direction of 45 degrees. With type II, the tool cuts the workpiece along the target figure. When the target figure has no pockets, determine the desired escaping operation and select type I or II.

Selecting type I or II

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

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

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

- Type I

(1) In the block with sequence number ns, only the second axis on the plane (X-axis (U-axis) for the ZX plane) must be specified.

```
Example
    ZX plane
    G71 V10.0 R5.0 ;
    G71 P100 Q200....;
    N100 X(U)_ ;          (Specifies only the second axis on the plane.)
                :  ;
                :  ;
    N200……;
```

(2) The figure along path A'-B must show monotone increase or decrease in the directions of both axes forming the plane (Z- and X-axes for the ZX plane). It must not have any pocket as shown in the figure below.

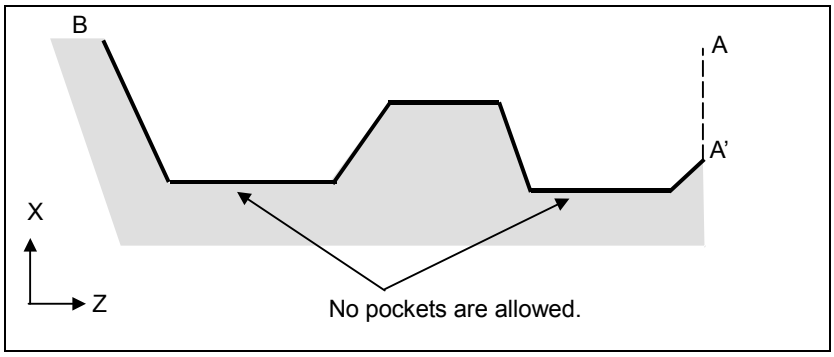


Fig. 6.4.1 (d) Figure which does not show monotone increase or decrease (type I)

> ⚠ **CAUTION**
> If a figure does not show monotone change along the first or second axis on the plane, alarm PS0064 or 0329 is issued. If the movement does not show monotone change, but is very small, and it can be determined that the movement is not dangerous, however, the permissible amount can be specified in parameters Nos. 5145 and 5146 to specify that the alarm is not issued in this case.

(3) The tool escapes to the direction of 45 degrees in cutting feed after rough cutting.

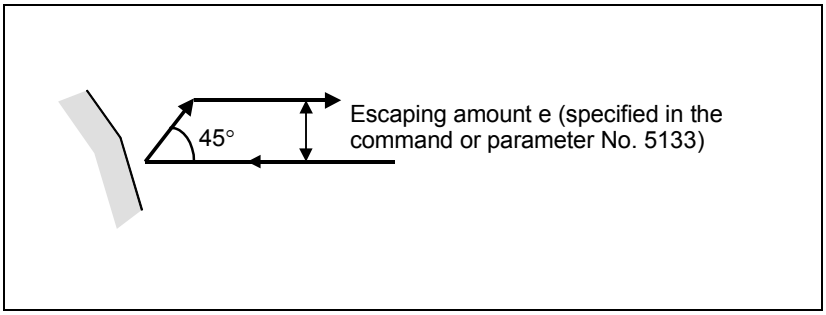


Fig. 6.4.1 (e) Cutting in the direction of 45 degrees (type I)

(4) Immediately after the last cutting, rough cutting is performed as finishing along the target figure. Bit 1 (RF1) of parameter No. 5105 can be set to 1 so that rough cutting as finishing is not performed. When the rough cutting finishing allowance is specified, however, rough cutting as finishing is performed.

- Type II

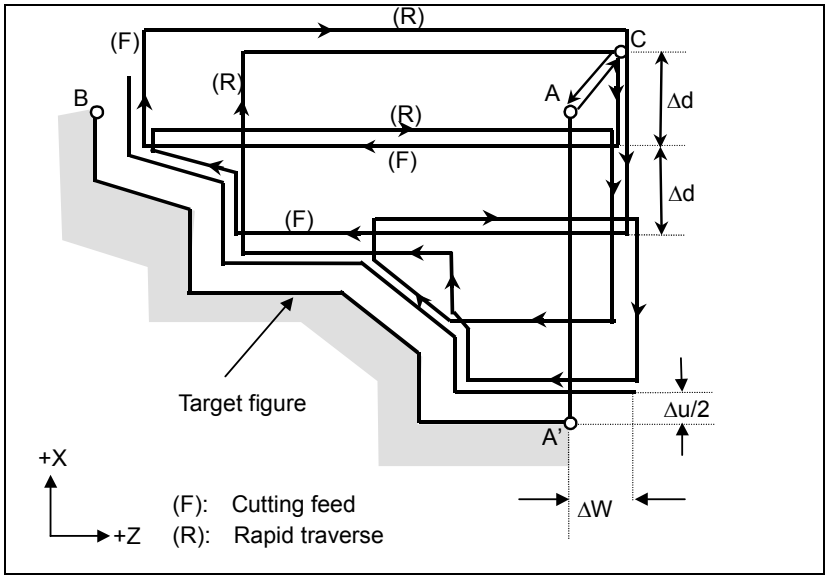


Fig. 6.4.1 (f) Cutting path in stock removal in turning (type II)

When the figure program for instructing a target figure passing through A, A', and B in this order is specified, a workpiece is cut away by depth of cut Δd at a time. In type II, cutting is performed along the figure after rough cutting in the direction of the plane first axis (z-axis for the ZX plane).
The machining path varies as follows depending on whether the rough machining finishing allowance is specified.

(1) When the rough cutting finishing allowance is not specified
Cutting is performed by depth of cut Δd with finishing allowances Δu/2 and Δw left, and the tool returns to the start point (A) after the last cutting is performed (one pocket is assumed because Pn→Pm is parallel to the z-axis in the above figure, and the zone is cut). Then, rough machining as finishing is performed according to the finishing figure program with finishing allowances Δu/2 and Δw left.

(2) When the rough cutting finishing allowance is specified
Cutting is performed by depth of cut Δd with cutting allowances Δu/2+Δi and Δw+Δk left, and the tool returns to the start point (A) after the last cutting is performed. Then, rough machining as finishing is performed along the target figure to remove cutting allowances Δi and Δk.

Upon completion of rough machining as finishing, the block next to the sequence block specified by Q is executed.

Type II differs from type I in the following points:

(1) In the block with sequence number ns, the two axes forming the plane (X-axis (U-axis) and Z-axis (W-axis) for the ZX plane) must be specified. When you want to use type II without moving the tool along the Z-axis on the ZX plane in the first block, specify W0.

```
Example
    ZX plane
    G71 V10.0 R5.0;
    G71 P100 Q200.....;
    N100 X(U)_ Z(W)_ ;      (Specifies the two axes forming the plane.)
                 :  ;
                 :  ;
    N200…….;
```

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

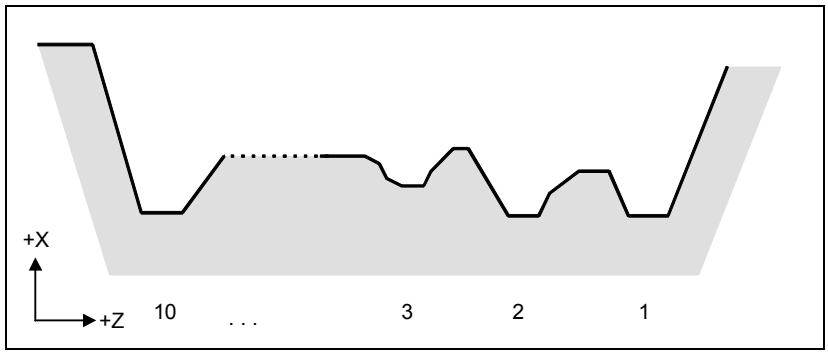


Fig. 6.4.1 (g) Figure having pockets (type II)

The figure must show monotone change in the direction of the first axis on the plane (Z-axis for the ZX plane), however. The following figure cannot be machined.

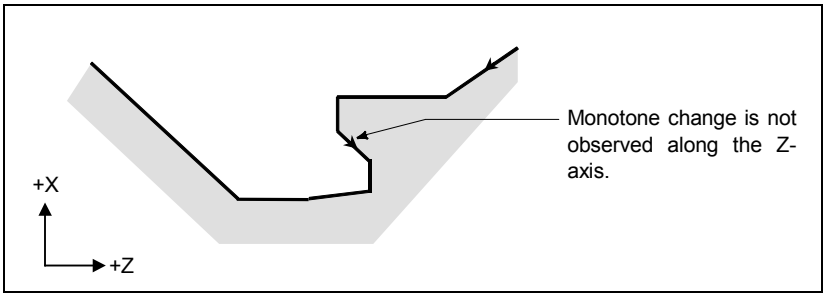


Fig. 6.4.1 (h) Figure which cannot be machined (type II)

⚠ CAUTION

For a figure along which the tool moves backward along the first axis on the plane during cutting operation (including a vertex in an arc command), the cutting tool may contact the workpiece. For this reason, for a figure which does not show monotone change, alarm PS0064 or PS0329 is issued. If the movement does not show monotone change, but is very small, and it can be determined that the movement is not dangerous, however, the permissible amount can be specified in parameter No. 5145 to specify that the alarm is not issued in this case.

The first cut portion need not be vertical. Any figure is permitted if monotone change is shown in the direction of the first axis on the plane (Z-axis for the ZX plane).

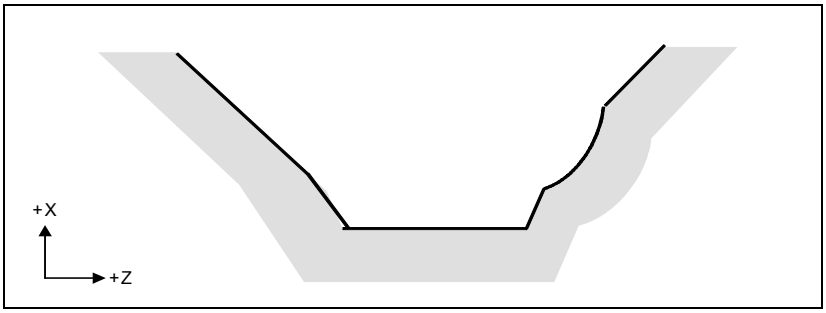


Fig. 6.4.1 (i) Figure which can be machined (type II)

(3) After turning, the tool cuts the workpiece along its figure and escapes in cutting feed.

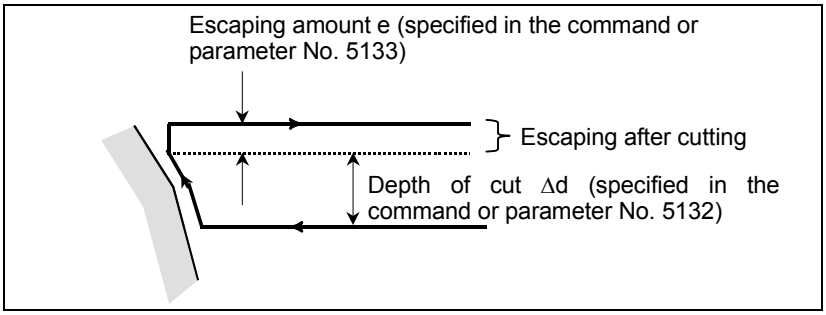


Fig. 6.4.1 (j) Cutting along the workpiece figure (type II)

The escaping amount e after cutting is set in parameter No. 5133. When moving from the bottom, however, the tool escapes to the direction of 45 degrees.

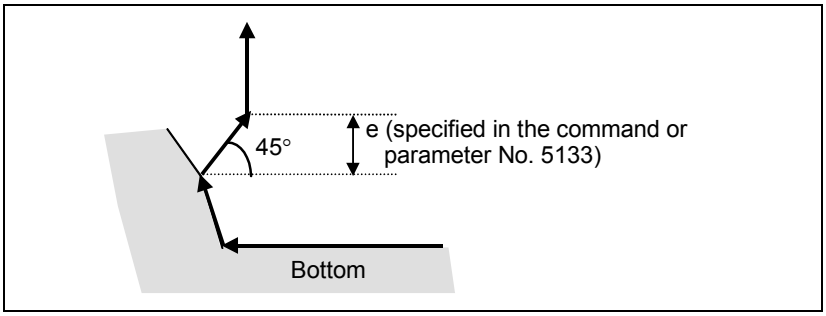


Fig. 6.4.1 (k) Escaping from the bottom to the direction of 45 degrees

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

(5) After all rough cutting terminates along the first axis on the plane (Z-axis for the ZX plane), the tool temporarily returns to the cycle start point. At this time, when there is a position whose height equals to that at the start point, the tool passes through the point in the position obtained by adding depth of cut Δd to the position of the figure and returns to the start point.
Then, rough cutting is performed as finishing along the target figure. At this time, the tool passes through the point in the obtained position (to which depth of cut Δd is added) when returning to the start point.
Bit 2 (RF2) of parameter No. 5105 can be set to 1 so that rough cutting as finishing is not performed.

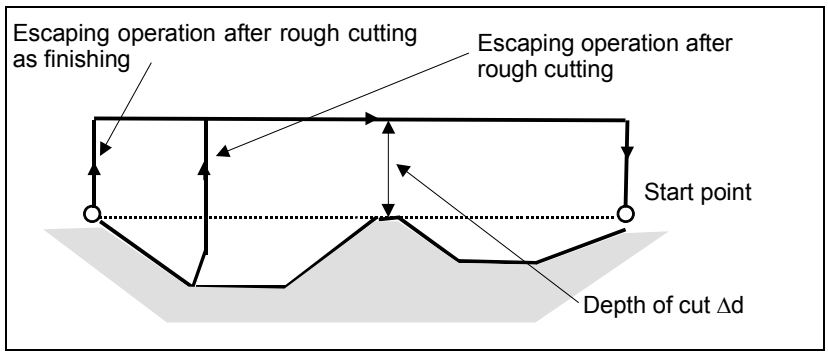


Fig. 6.4.1 (l) Escaping operation when the tool returns to the start point (type II)

(6) Order and path for rough cutting of pockets
Rough cutting is performed in the following order.

(a) When the figure shows monotone decrease along the first axis on the plane (Z-axis for the ZX plane)

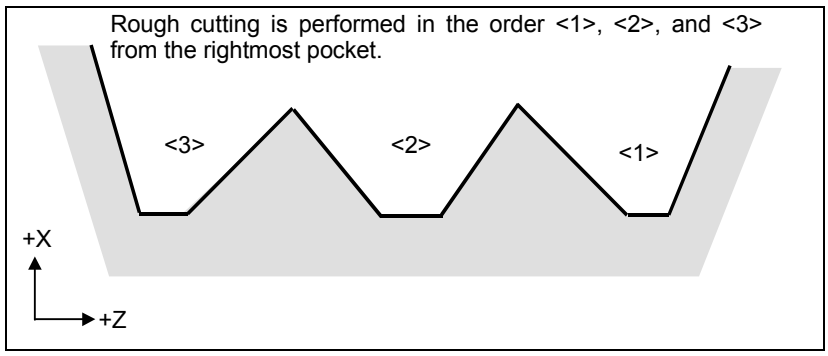


Fig. 6.4.1 (m) Rough cutting order in the case of monotone decrease (type II)

(b) When the figure shows monotone increase along the first axis on the plane (Z-axis for the ZX plane)

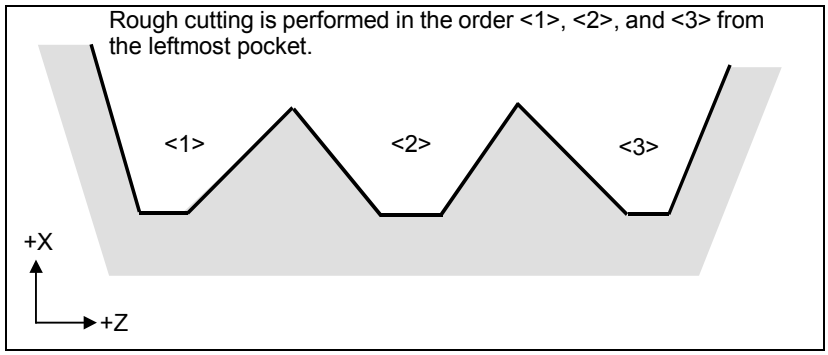


Fig. 6.4.1 (n) Rough cutting order in the case of monotone increase (type II)

The path in rough cutting is as shown below.

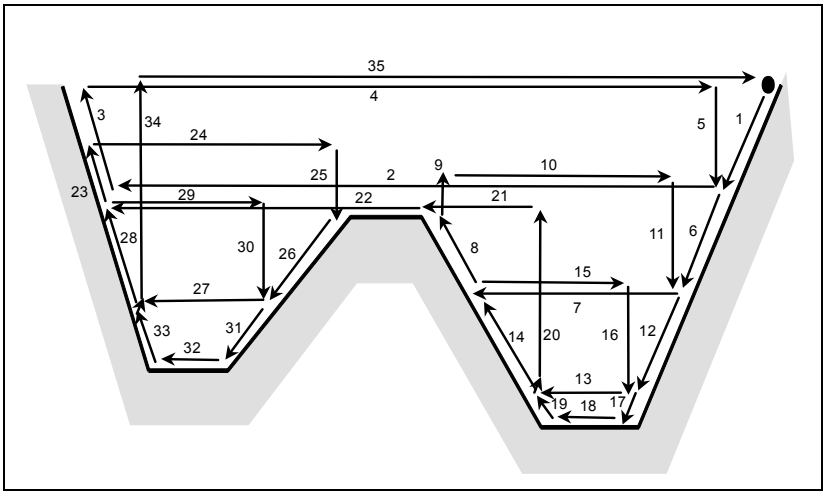


Fig. 6.4.1 (o) Cutting path for multiple pockets (type II)

The following figure shows how the tool moves after rough cutting for a pocket in detail.

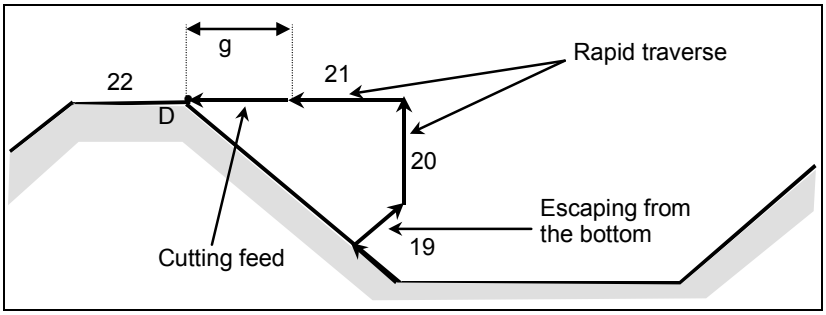


Fig. 6.4.1 (p) Details of motion after cutting for a pocket (type II)

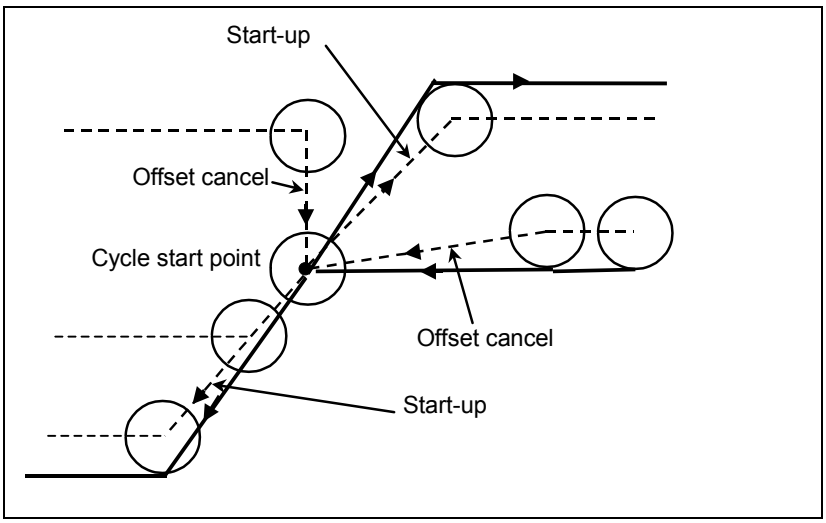
Cuts the workpiece at the cutting feedrate and escapes to the direction of 45 degrees. (Operation 19)
Then, moves to the height of point D in rapid traverse. (Operation 20)
Then, moves to the position the amount of g before point D. (Operation 21)
Finally, moves to point D in cutting feed.
The clearance g to the cutting feed start position is set in parameter No. 5134.
For the last pocket, after cutting the bottom, the tool escapes to the direction of 45 degrees and returns to the start point in rapid traverse. (Operations 34 and 35)

> ⚠ **CAUTION**
> 1 This CNC differs from the Series 0*i*-C in cutting of a pocket.
> The tool first cuts the nearest pocket to the start point. After cutting of the pocket terminates, the tool moves to the nearest but one pocket and starts cutting.
> 2 When the figure has a pocket, generally specify a value of 0 for Δw (finishing allowance). Otherwise, the tool may dig into the wall on one side.

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70, G71, G72, G73) and specify the cancel command (G40) outside the blocks (from the block specified with P to the block specified with Q) specifying a target finishing figure. If a tool nose radius compensation command (G40, G41, or G42) is specified in the G70, G71, G72, or G73 command, alarm PS0325 is issued.

When this cycle is specified in the tool nose radius compensation mode, offset is temporarily canceled during movement to the start point. Start-up is performed in the first block. Offset is temporarily canceled again at the return to the cycle start point after termination of cycle operation. Start-up is performed again according to the next move command. This operation is shown in the figure below.



This cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

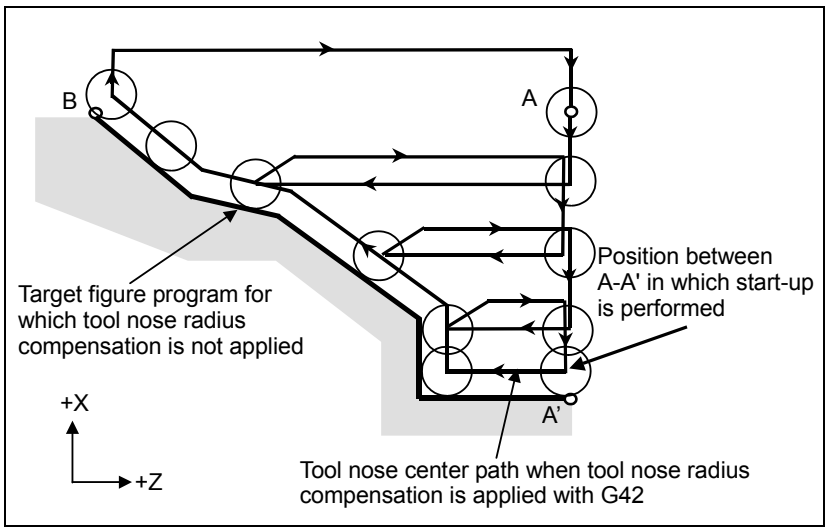
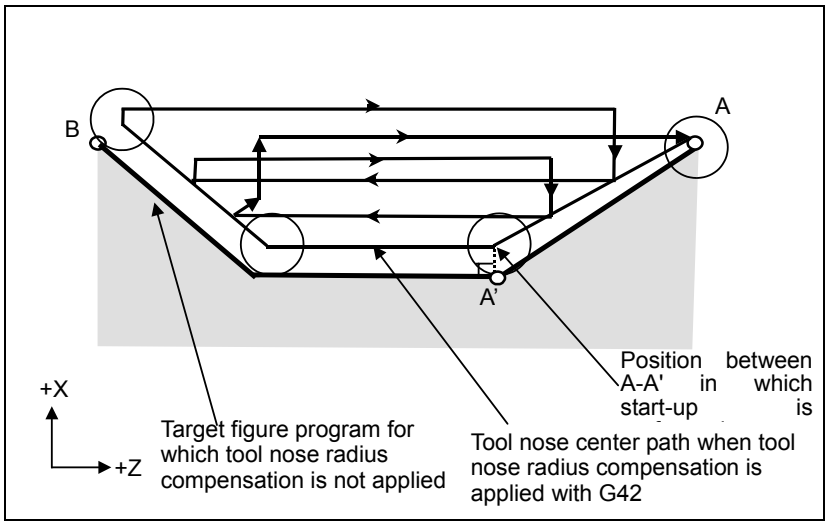


Fig. 6.4.1 (q) Path when tool nose radius compensation is applied



NOTE

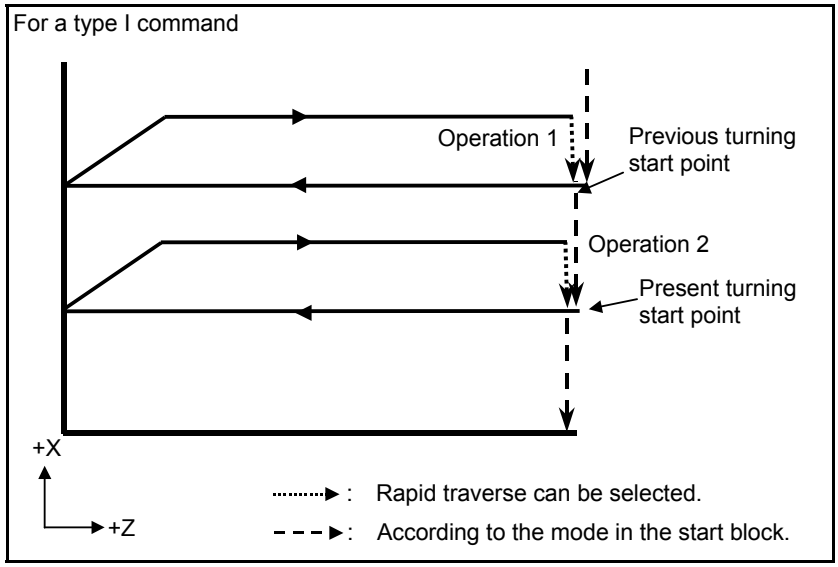
To perform pocketing in the tool nose radius compensation mode, specify the linear block A-A' outside the workpiece and specify the figure of an actual pocket. This prevents a pocket from being dug.

- Movement to the previous turning start point

Movement to the turning start point is performed with two operations. (Operations 1 and 2 in the figure below.) As movement to the present turning start point, operation 1 temporarily moves the tool to the previous turning start point, then operation 2 moves the tool to the present turning start point.

Operation 1 moves the tool in cutting feed. Operation 2 moves the tool according to the mode (G00 or G01) specified in the start block in the geometry program.

Bit 0 (ASU) of parameter No. 5107 can be set to 1 so that operation 1 moves the tool in rapid traverse.


For a type I command
Operation 1
Previous turning start point
Operation 2
Present turning start point
+X
+Z
: Rapid traverse can be selected.
: According to the mode in the start block.

6.4.2 Stock Removal in Facing (G72)

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

Format

ZpXp plane

G72 P(ns) Q(nf) U(Δu) W(Δw) I(Δi) K(Δk) D(Δd) F(f) S(s) T(t);
N (ns) ;
...
N (nf) ;

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

YpZp plane

G72 P(ns) Q(nf) V(Δw) W(Δu) J(Δk) K(Δi) D(Δd) F(f) S(s) T(t);
N (ns) ;
...
N (nf) ;

XpYp plane

G72 P(ns) Q(nf) U(Δw) V(Δu) I(Δk) J(Δi) D(Δd) F(f) S(s) T(t);
N (ns) ;
...
N (nf) ;

Δd : Depth of cut
The cutting direction depends on the direction AA'.
ns : Sequence number of the first block for the program of finishing shape.
nf : Sequence number of the last block for the program of finishing shape.
Δu : Distance of the finishing allowance in the direction of the second axis on the plane (X-axis for the ZX plane)
Δw : Distance of the finishing allowance in the direction of the first axis on the plane (Z-axis for the ZX plane)
Δi : Distance of the finishing allowance of the roughing in the direction of the second axis on the plane (X-axis for the ZX plane)
Δk : Distance of the finishing allowance of the roughing in the direction of the first axis on the plane (Z-axis for the ZX plane)
f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G72 block is effective.

NOTE
Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

	Unit	Diameter/radius programming	Sign	Decimal point input
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed
Δi	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed

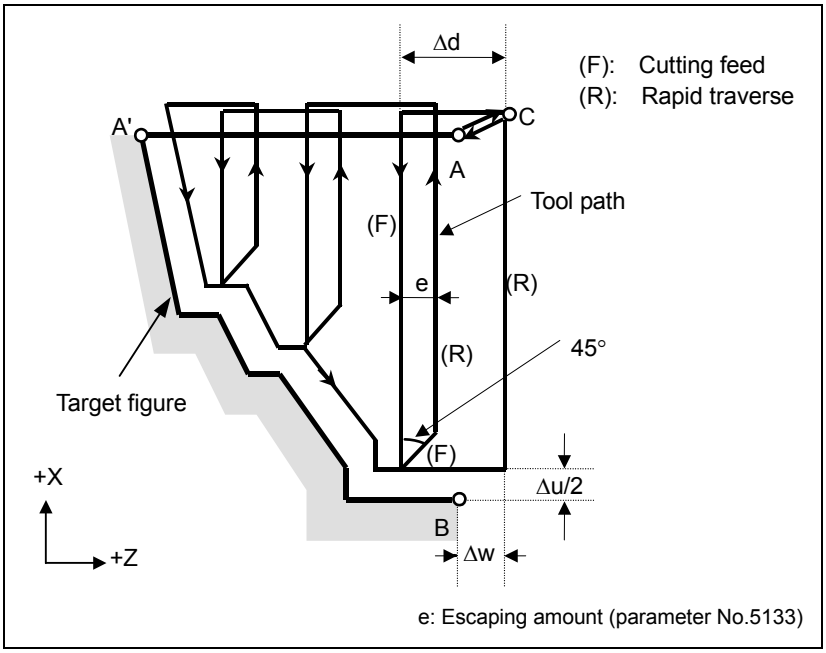


Fig. 6.4.2 (r) Cutting path in stock removal in facing (type I)

Explanation

- Operations

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

NOTE
1 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G72 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
2 When the constant surface speed control function is enabled (bit 0 (SSC) of parameter No. 8133 is set to 1), the G96 or G97 command specified in the move command between points A and B is ignored. If you want to enable the G96 or G97 command, specify the command in the G71 or previous block.

- Escaping amount (e)

The escaping amount (e) is set in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5133	Depends on the increment system for the reference axis.	Radius programming	Not required

- Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the second axis on the plane (X-axis for the ZX plane). At this time, the signs of the finishing allowances of Δu and Δw are as follows:

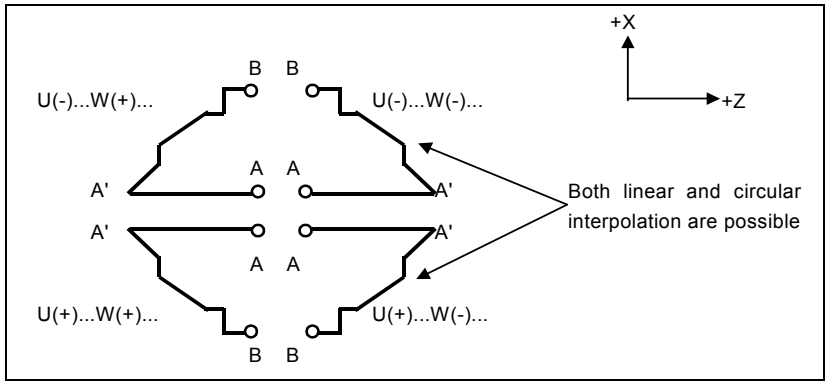


Fig. 6.4.2 (s) Signs of the values specified at U and W in stock removal in facing

Limitation

(1) For W(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For W(-), a figure for which a position lower than the cycle start point is specified cannot be machined.

(2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.

(3) For type II, the figure must show monotone increase or decrease along the second axis on the plane.

Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.
When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.
In this start block, also select type I or II.

Check functions

During cycle operation, whether the target figure shows monotone increase or decrease is always checked.

NOTE
When tool nose radius compensation is applied, the target figure to which compensation is applied is checked.

The following checks can also be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.
Checks the target figure before cycle operation. (Also checks that a block with the sequence number specified at address Q is contained.)	Enabled when bit 2 (FCK) of parameter No. 5104 is set to 1.

- Types I and II Selection of type I or II

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

Selecting type I or II

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

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

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

- Type I

G72 differs from G71 in the following points:

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

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

- Type II

G72 differs from G71 in the following points:

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

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

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

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

- Tool nose radius compensation

See the pages on which G71 is explained.

- Movement to the previous turning start point

See the pages on which G71 is explained.

6.4.3 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 workpiece whose rough shape has already been made by a rough machining, forging or casting method, etc.

Format

ZpXp plane

G73 P(ns) Q(nf) U(Δu) W(Δw) I(Δi) K(Δk) D(d) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

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

YpZp plane

G73 P(ns) Q(nf) V(Δw) W(Δu) J(Δk) K(Δi) D(d) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

XpYp plane

G73 P(ns) Q(nf) U(Δw) V(Δu) I(Δk) J(Δi) D(d) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

Δi : Distance of escape in the direction of the second axis on the plane (X-axis for the ZX plane)
Δk : Distance of escape in the direction of the first axis on the plane (Z-axis for the ZX plane)
d : The number of division
This value is the same as the repetitive count for rough cutting.
ns : Sequence number of the first block for the program of finishing shape.
nf : Sequence number of the last block for the program of finishing shape.
Δu : Distance of the finishing allowance in the direction of the second axis on the plane (X-axis for the ZX plane)
Δw : Distance of the finishing allowance in the direction of the first axis on the plane (Z-axis for the ZX plane)
f, s, t : Any F, S, and T function contained in the blocks between sequence number "ns" and "nf" are ignored, and the F, S, and T functions in this G73 block are effective.

NOTE
Even if pocket calculator type decimal point programming is specified (bit 0 (DPI) of parameter No. 3401 = 1), the unit of address D is the least input increment. In addition, when a decimal point is input in address D, alarm PS0007 is issued.

	Unit	Diameter/radius programming	Sign	Decimal point input
Δi	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed

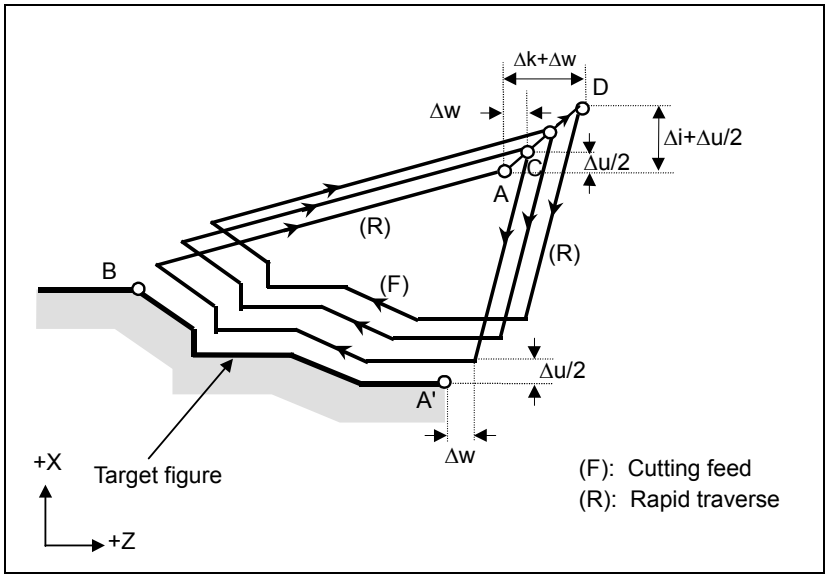


Fig. 6.4.3 (t) Cutting path in pattern repeating

Explanation

- Operations

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

NOTE
1 After cycle operation terminates, the tool returns to point A.
2 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G73 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.

- Target figure patterns

As in the case of G71, there are four target figure patterns. Be careful about signs of Δu, Δw, Δi, and Δk when programming this cycle.

- Start block

In the start block in the program for the target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.
When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.

- Check function

The following check can be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.

- Tool nose radius compensation

Like G71, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

6.4.4 Finishing Cycle (G70)

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

Format

G70 P(ns) Q(nf) ;

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

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

Explanation

- Operations

The blocks with sequence numbers ns to nf in the program for a target figure are executed for finishing. The F, S, T, M, and second auxiliary functions specified in the G71, G72, or G73 block are ignored and the F, S, T, M, and second auxiliary functions specified in the blocks with sequence numbers ns to nf are effective.
When cycle operation terminates, the tool is returned to the start point in rapid traverse and the next G70 cycle block is read.

- Target figure check function

The following check can be made.

Check	Related parameter
Checks that a block with the sequence number specified at address Q is contained in the program before cycle operation.	Enabled when bit 2 (QSR) of parameter No. 5102 is set to 1.

- Storing P and Q blocks

When rough cutting is executed by G71, G72, or G73, up to three memory addresses of P and Q blocks are stored. By this, the blocks indicated by P and Q are immediately found at execution of G70 without searching memory from the beginning for them. After some G71, G72, and G73 rough cutting cycles are executed, finishing cycles can be performed by G70 at a time. At this time, for the fourth and subsequent rough cutting cycles, the cycle time is longer because memory is searched for P and Q blocks.

Example

```
G71 P100 Q200 ...;
N100 ...;
...;
...;
N200 ...;
G71 P300 Q400 ...;
N300 ...;
...;
...;
N400 ...;
...;
...;
G70 P100 Q200 ;  (Executed without a search for
                  the first to third cycles)
G70 P300 Q400 ;  (Executed after a search for the
                  fourth and subsequent cycles)
```

NOTE

The memory addresses of P and Q blocks stored during rough cutting cycles by G71, G72, and G73 are erased after execution of G70.
All stored memory addresses of P and Q blocks are also erased by a reset.

- Return to the cycle start point

In a finishing cycle, after the tool cuts the workpiece to the end point of the target figure, it returns to the cycle start point in rapid traverse.

NOTE

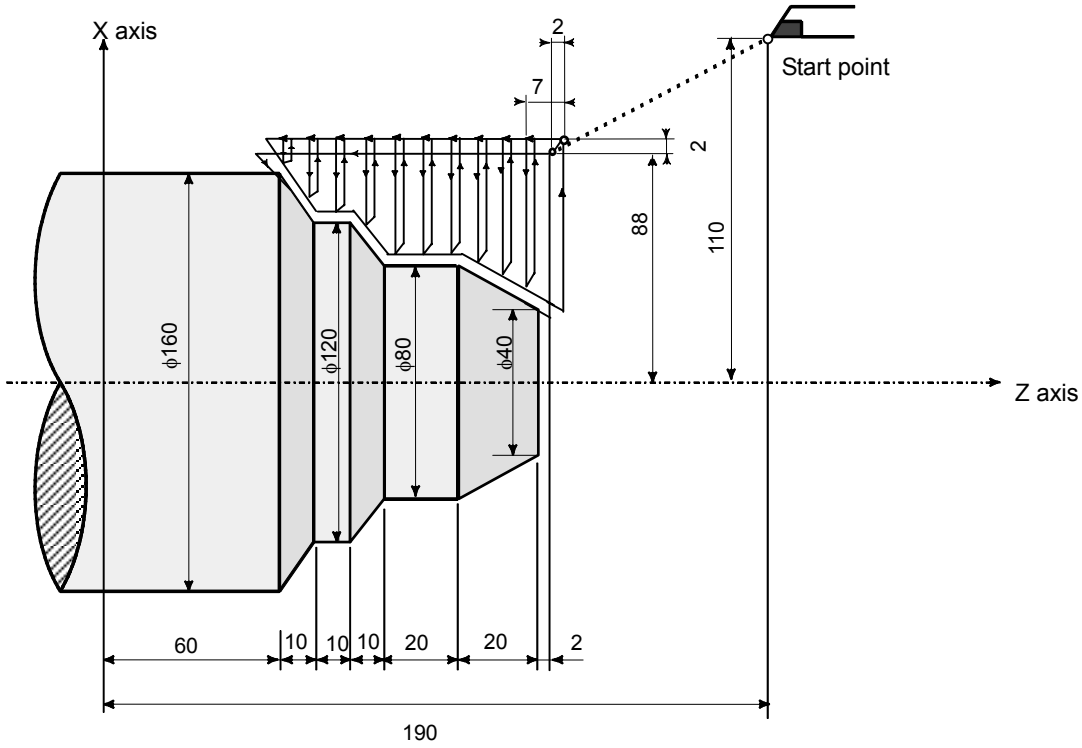
The tool returns to the cycle start point always in the nonlinear positioning mode regardless of the setting of bit 1 (LRP) of parameter No. 1401.
Before executing a finishing cycle for a target figure with a pocket cut by G71 or G72, check that the tool does not interfere with the workpiece when returning from the end point of the target figure to the cycle start point.

- Tool nose radius compensation

Like G71, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

Example

Stock removal in facing (G72)

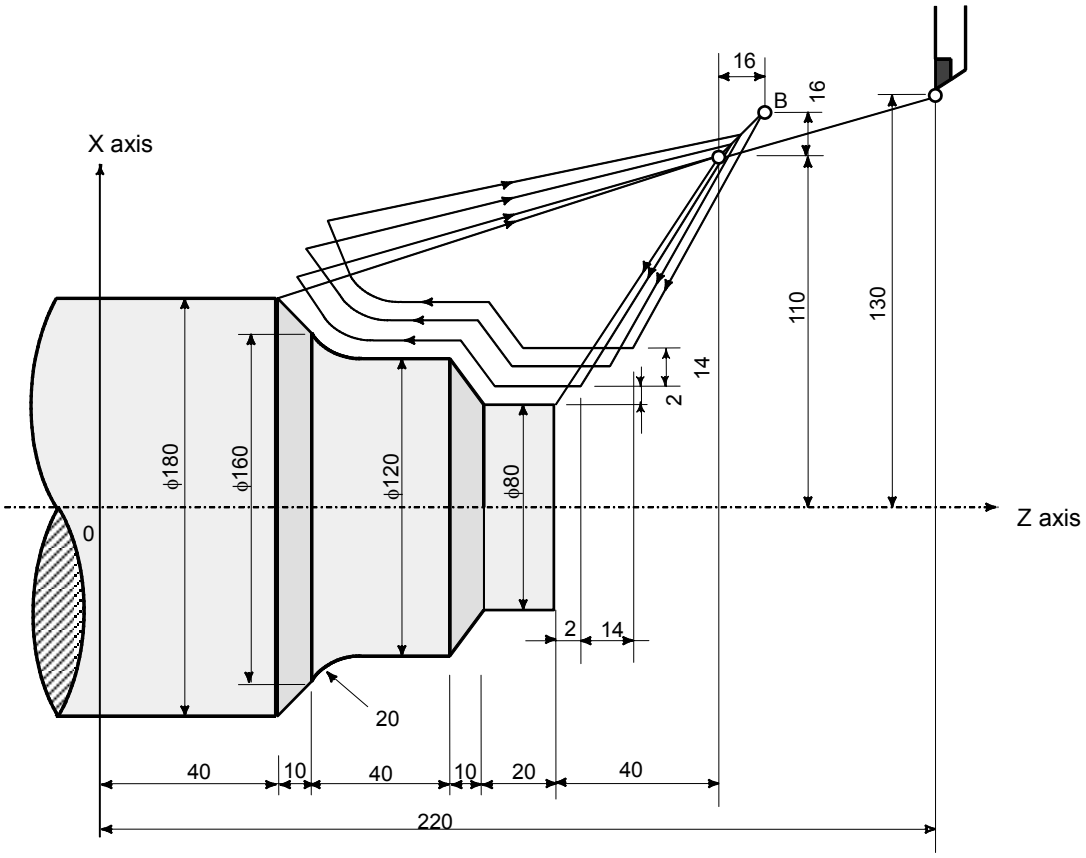


(Diameter designation for X axis, metric input)

```
N011    G50 X220.0 Z190.0 ;
N012    G00 X176.0 Z132.0 ;
N013    G72 P014 Q019 U4.0 W2.0 D7000 F0.3 S550 ;
N014    G00 Z56.0 S700 ;
N015     G01 X120.0 W14.0 F0.15 ;
N016    W10.0 ;
N017    X80.0 W10.0 ;
N018    W20.0 ;
N019    X36.0 W22.0 ;
N020    G70 P014 Q019 ;
```

Parameter No. 5133 = 1.0 (escaping amount)
Finishing allowance (4.0 in diameter in the X direction, 2.0 in the Z direction)

Pattern repeating (G73)



(Diameter designation, metric input)

```
N011    G50 X260.0 Z220.0 ;
N012    G00 X220.0 Z160.0 ;
N013    G73 P014 Q019 U4.0 W2.0 I14.0 K14.0 D3 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 ;
```

6.4.5 End Face Peck Drilling Cycle (G74)

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

Format

ZpXp-plane
G74X(U)_ Z(W)_ I(Δi) K(Δk) D(Δd) F(f) ;
YpZp-plane
G74Y(V)_ Z(W)_ J(Δk) K(Δi) D(Δd) F(f) ;
XpYp-plane
G74X(U)_ Y(V)_ I(Δk) J(Δi) D(Δd) F(f) ;

X_,Z_ : Coordinate of the second axis on the plane (X-axis for the ZX plane) at point B and
Coordinate of the first axis on the plane (Z-axis for the ZX plane) at point C
U_,W_ : Travel distance along the second axis on the plane (U for the ZX plane) from point A to B
Travel distance along the first axis on the plane (W for the ZX plane) from point A to C
(When G code system A is used. In other cases, X_,Z_ is used for specification.)
Δi : Travel distance in the direction of the second axis on the plane (X-axis for the ZX plane)
Δk : Depth of cut in the direction of the first axis on the plane (Z-axis for the ZX plane)
Δd : Relief amount of the tool at the cutting bottom
f : Feedrate

	Unit	Diameter/radius programming	Sign	Decimal point input
Δi	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δd	Depends on the increment system for the reference axis.	Radius programming	NOTE 1	Not allowed

NOTE
1 Normally, specify a positive value for Δd. When X (U) and Δi are omitted, specify a value with the sign indicating the direction in which the tool is to escape.
2 Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

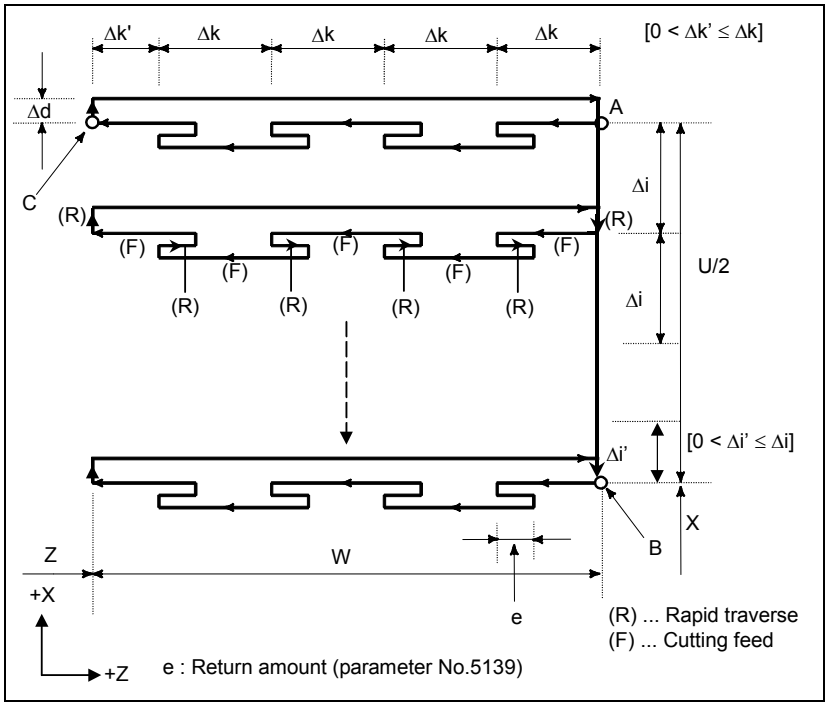


Fig. 6.4.5 (a) Cutting path in end face peek drilling cycle

Explanation

- Operations

A cycle operation of cutting by Δk and return by e is repeated.
When cutting reaches point C, the tool escapes by Δd. Then, the tool returns in rapid traverse, moves to the direction of point B by Δi, and performs cutting again.

- Return amount (e)

The escaping amount (e) is set in parameter No. 5139.

No.	Unit	Diameter/radius programming	Sign
5139	Depends on the increment system for the reference axis.	Radius programming	Not required

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

6.4.6 Outer Diameter / Internal Diameter Drilling Cycle (G75)

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

Format

ZpXp-plane
G75 X(U)_ Z(W)_ I(Δi) K(Δk) D(Δd) F (f) ;
YpZp-plane
G75 Y(V)_ Z(W)_ J(Δk) K(Δi) D(Δd) F(f) ;
XpYp-plane
G75 X(U)_ Y(V)_ I(Δk) J(Δi) D(Δd) F(f) ;

X_, Z_ : Coordinate of the second axis on the plane (X-axis for the ZX plane) at point B and
Coordinate of the first axis on the plane (Z-axis for the ZX plane) at point C
U_, W_ : Travel distance along the second axis on the plane (U for the ZX plane) from point A to B
Travel distance along the first axis on the plane (W for the ZX plane) from point A to C
Δi : Depth of cut in the direction of the second axis on the plane (X-axis for the ZX plane)
Δk : Travel distance in the direction of the first axis on the plane (Z-axis for the ZX plane)
Δd : Relief amount of the tool at the cutting bottom
f : Feedrate

	Unit	Diameter/radius programming	Sign	Decimal point input
Δi	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δd	Depends on the increment system for the reference axis.	Radius programming	NOTE 1	Not allowed

NOTE
1 Normally, specify a positive value for Δd. When Z (W) and Δk are omitted, specify a value with the sign indicating the direction in which the tool is to escape.
2 Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

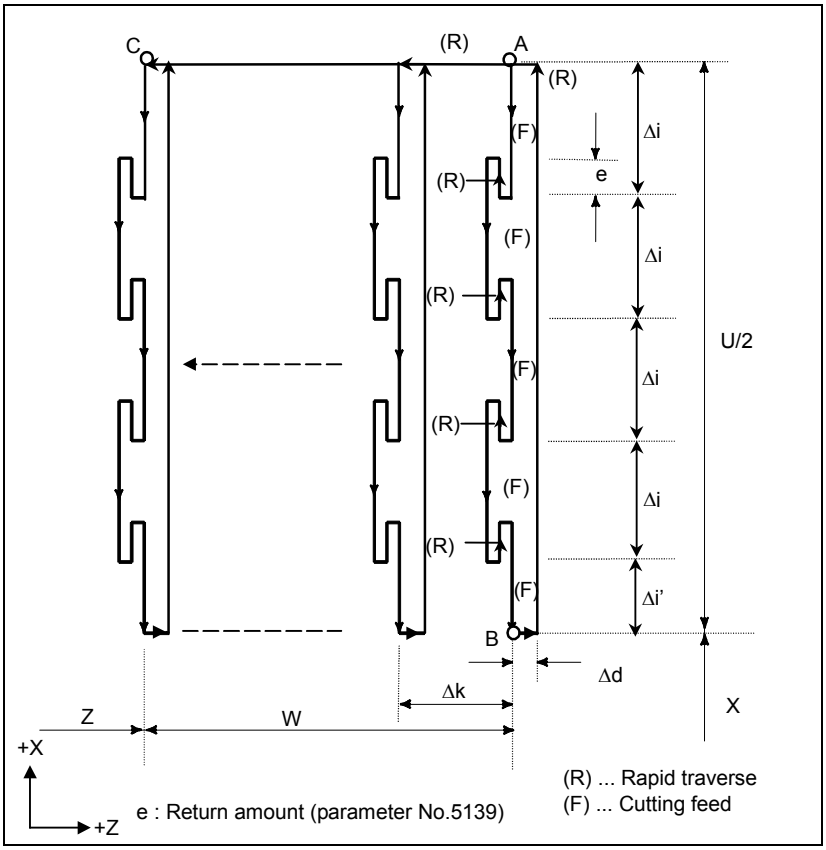


Fig. 6.4.6 (a) Outer diameter/internal diameter drilling cycle

Explanation

- Operations

A cycle operation of cutting by Δi and return by e is repeated.
When cutting reaches point B, the tool escapes by Δd. Then, the tool returns in rapid traverse, moves to the direction of point C by Δk, and performs cutting again.

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

- Return amount (e)

The escaping amount (e) is set in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5139	Depends on the increment system for the reference axis.	Radius programming	Not required

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

6.4.7 Multiple Threading Cycle (G76)

The multiple threading cycle can select four cutting methods.

Format

ZpXp-plane
G76 X(U)_ Z(W)_ I(i) K(k) D(Δd) A(a) F(L) P(p) Q(q) ;
YpZp-plane
G76 Y(V)_ Z(W)_ J(k) K(i) D(Δd) A(a) F(L) P(p) Q(q) ;
XpYp-plane
G76 X(U)_ Y(V)_ I(k) J(i) D(Δd) A(a) F(L) P(p) Q(q) ;

X_, Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length
U_, W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length
a : Angle of tool nose
From 0 to 120 in steps of 1 degree
(The default is 0.)
i : Taper amount
If i = 0, ordinary straight threading can be made.
k : Height of thread
Δd : Depth of cut in 1st cut
L : Lead of thread
p : Cutting method (one-edge threading with constant cutting amount by default or for P0)
P1: One-edge threading with constant cutting amount
P2: Both-edge zigzag threading with constant cutting amount
P3: One-edge threading with constant depth of cut
P4: Both-edge zigzag threading with constant depth of cut
q : Threading start angle shift
(From 0 to 360 degrees in steps of 0.001 degrees)

NOTE
1 Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.
2 A decimal point included in address A has no meaning. That is, A120. is equivalent to A120 in specifying 120 degrees.
3 To use P2, P3, or P4 as a cutting method, the option for multiple repetitive canned cycle II is required.
4 Address Q does not allow decimal point input.

	Unit	Diameter/radius programming	Sign	Decimal point input
i	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
k	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed

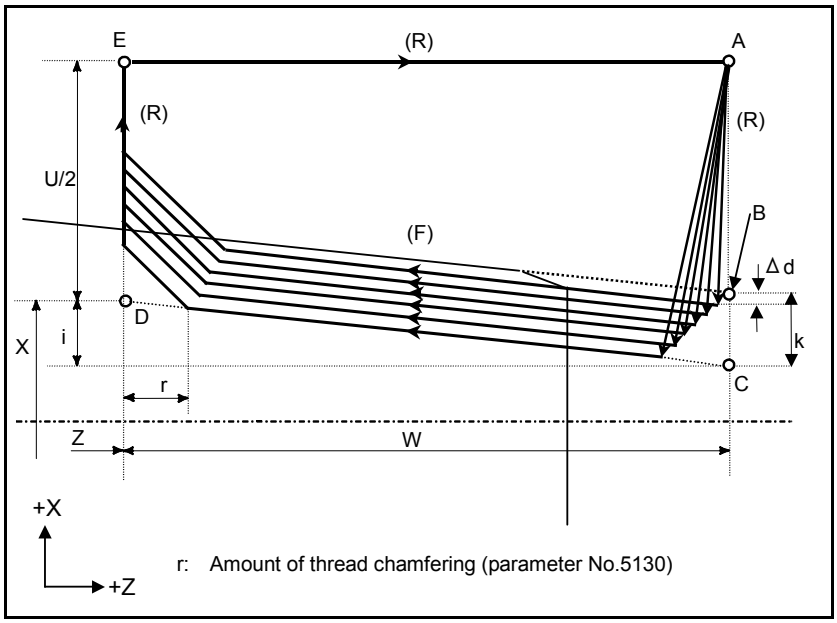


Fig. 6.4.7 (a) Cutting path in multiple threading cycle

Explanation

- Operations

This cycle performs threading so that the length of the lead only between C and D is made as specified in the F code. In other sections, the tool moves in rapid traverse.
The time constant for acceleration/deceleration after interpolation and FL feedrate for thread chamfering and the feedrate for retraction after chamfering are the same as for thread chamfering with canned cycle.

> ⚠ **CAUTION**
> Notes on threading are the same as those on G32 threading. For feed hold in a threading cycle, however, see "Feed hold in a threading cycle" described below.

- Cutting method

There are four cutting methods.

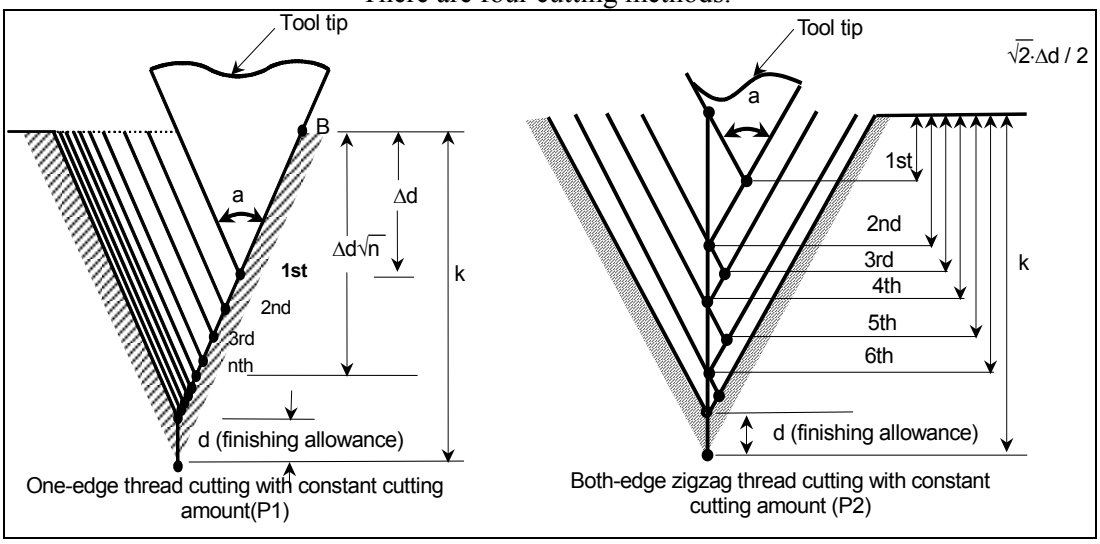


Fig. 6.4.7 (b) One-edge threading with constant cutting amount, both-edge zigzag threading with constant cutting amount (P1/2)

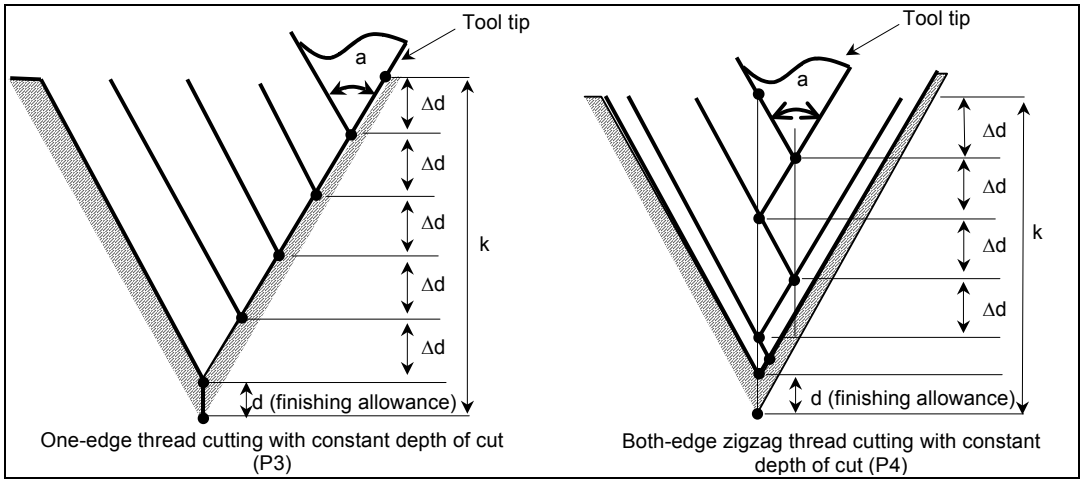
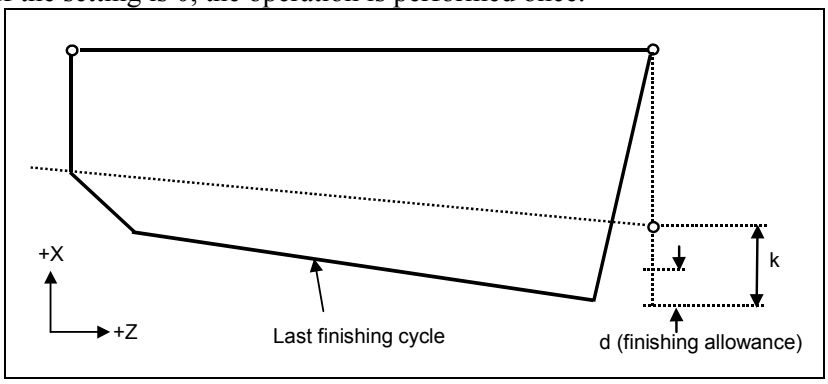


Fig. 6.4.7 (c) One-edge threading with constant depth of cut, both-edge zigzag threading with constant depth of cut (P3/4)

- Repetitive count in finishing

The last finishing cycle (cycle in which the finishing allowance is removed by cutting) is repeated.
The repetitive count is set in parameter No. 5142.
If the setting is 0, the operation is performed once.



- Minimum depth of cut

When a cutting method with constant cutting amount is selected (P1 or P2), clamping can be performed with the minimum depth of cut to prevent the depth of cut from becoming too small.
The minimum depth of cut is set in parameter No. 5140.

No.	Unit	Diameter/radius programming	Sign
5140	Depends on the increment system for the reference axis.	Radius programming	Not required

- Finishing allowance

The finishing allowance is set in parameter No. 5141.

No.	Unit	Diameter/radius programming	Sign
5141	Depends on the increment system for the reference axis.	Radius programming	Not required

- Relationship between the sign of the taper amount and tool path

The signs of incremental dimensions for the cycle shown in Fig. 6.4.7 (a) are as follows:

Cutting end point in the direction of the length for U and W:

Minus (determined according to the directions of paths A-C and C-D)

Taper amount (i):

Minus (determined according to the direction of path A-C)

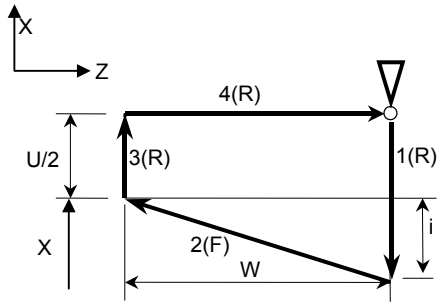
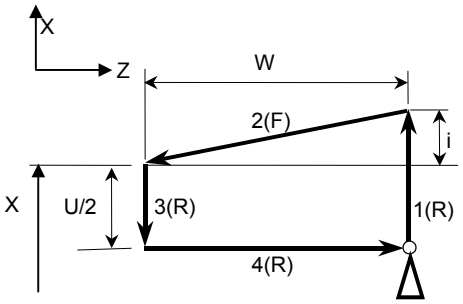
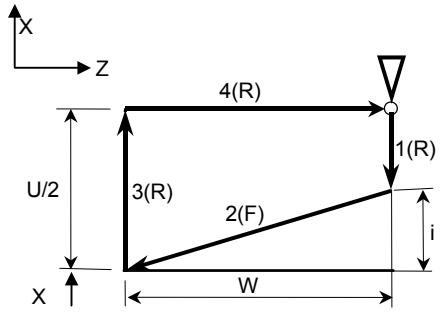
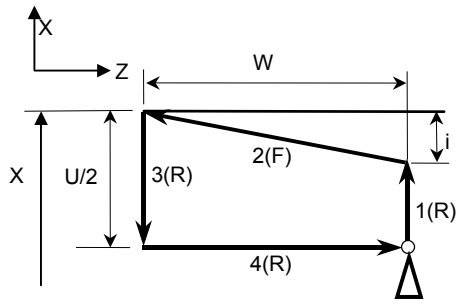
Height of thread (k):

Plus (always specified with a plus sign)

Depth of cut in the first cut (Δd):

Plus (always specified with a plus sign)

The four patterns shown in the table below are considered corresponding to the sign of each address. A female thread can also be machined.

Outer diameter machining	Internal diameter machining
1. U < 0, W < 0, i < 0	**2. U > 0, W < 0, i > 0**
	
3. U < 0, W < 0, i > 0 at |i|≤|U/2|	**4. U > 0, W < 0, i < 0 at |i|≤|U/2|**
	

- Acceleration/deceleration after interpolation for threading

Acceleration/deceleration after interpolation for threading is acceleration/deceleration of exponential interpolation type. By setting bit 5 (THLx) of parameter No. 1610, the same acceleration/deceleration as for cutting feed can be selected. (The settings of bit 0 (CTLx) of parameter No. 1610 are followed.) However, as a time constant and FL feedrate, the settings of parameter No. 1626 and No. 1627 for the threading cycle are used.

- Time constant and FL feedrate for threading

The time constant for acceleration/deceleration after interpolation for threading specified in parameter No. 1626 and the FL feedrate specified in parameter No. 1627 are used.

- Thread chamfering

Thread chamfering can be performed in this threading cycle. A signal from the machine tool initiates thread chamfering.
The maximum amount of thread chamfering (r) can be specified in a range from 0.1L to 12.7L in 0.1L increments in parameter No. 5130.
A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.
For thread chamfering, the same type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate as for threading are used.

NOTE
Common parameters for specifying the amount and angle of thread chamfering are used for this cycle and G92 threading cycle.

- Retraction after chamfering

The following table lists the feedrate, type of acceleration/deceleration after interpolation, and time constant of retraction after chamfering.

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

By setting bit 4 (ROC) of parameter No. 1403 to 1, rapid traverse override can be disabled for the feedrate of retraction after chamfering.

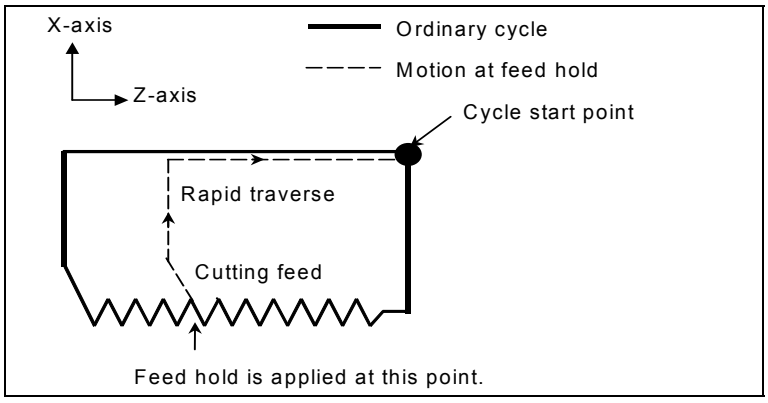
NOTE
During retraction, the machine does not stop with an override of 0% for the cutting feedrate regardless of the setting of bit 4 (RF0) of parameter No. 1401.

- Shifting the start angle

Address Q can be used to shift the start angle of threading.
The start angle (Q) increment is 0.001 degrees and the valid setting range is between 0 and 360 degrees. No decimal point can be specified.

- Feed hold when the threading cycle retract function is used

Feed hold may be applied during threading in a multiple threading cycle (G76). In this case, the tool quickly retracts in the same way as for the last chamfering in a threading cycle and returns to the start point in the current cycle (position where the workpiece is cut by Δdn).
When cycle start is triggered, the multiple threading cycle resumes.



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

⚠ CAUTION
Feed hold operation during retraction is disabled.

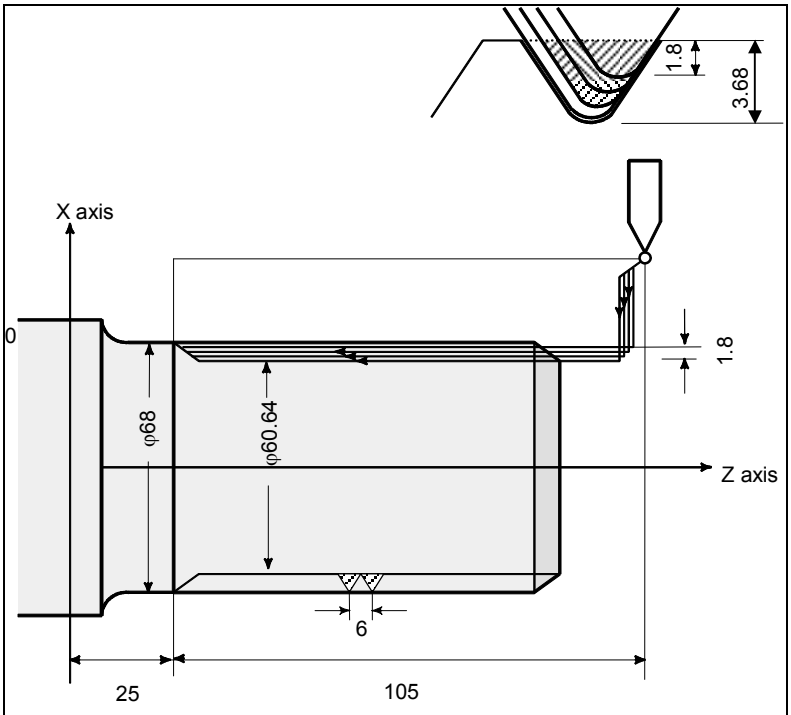
- Inch threading

Inch threading specified with address E is allowed.

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

Example



G00 X80.0 Z130.0;
G76 X60.64 Z25.0 K3680 D1800 A60 P1 F6.0 ;
Parameter No.5130 = 10(1.0L)

6.4.8 Restrictions on Multiple Repetitive Canned Cycle

Programmed commands

- Program memory

Programs using G70, G71, G72, or G73 must be stored in the program memory. The use of the mode in which programs stored in the program memory are called for operation enables these programs to be executed in other than the MEM mode. Programs using G74, G75, or G76 need not be stored in the program memory.

- Blocks in which data related to a multiple repetitive canned cycle is specified

The addresses P, Q, X, Z, U, W, and R should be specified correctly for each block.

In a block in which G70, G71, G72, or G73 is specified, the following functions cannot be specified:

- Custom macro calls
  (simple call, modal call, and subprogram call)

- Blocks in which data related to a target figure is specified

In the block which is specified by address P of a G71, G72 or G73, G00 or G01 code in group 01 should be commanded. If it is not commanded, alarm PS0065 is generated.

In blocks with sequence numbers between those specified at P and Q in G70, G71, G72, and G73, the following commands can be specified:

- Dwell (G04)
- G00, G01, G02, and G03
  When a circular interpolation command (G02, G03) is used, there must be no radius difference between the start point and end point of the arc. If there is a radius difference, the target finishing figure may not be recognized correctly, resulting in a cutting error such as excessive cutting.
- Custom macro branch and repeat command
  The branch destination must be between the sequence numbers specified at P and Q, however. High-speed branch specified by bits 1 and 4 of parameter No. 6000 is invalid. No custom macro call (simple, modal, or subprogram call) cannot be specified.
- Direct drawing dimension programming command and chamfering and corner R command
  Direct drawing dimension programming and chamfering and corner R require multiple blocks to be specified. The block with the last sequence number specified at Q must not be an intermediate block of these specified blocks.

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.

When #1 = 2500 is executed using a custom macro, 2500.000 is assigned to #1. In such a case, P#1 is equivalent to P2500.

Relation with other functions

- Manual intervention

While a multiple repetitive canned cycle (G70 to G76) is being executed, it is possible to stop the cycle and to perform manual intervention.
The setting of manual absolute on or off is effective for manual operation.

- Interruption type macro

Any interruption type macro program cannot be executed during execution of a multiple repetitive canned cycle.

- Program restart and tool retract and recover

These functions cannot be executed in a block in a multiple repetitive canned cycle.

- Axis name and second auxiliary functions

Even if address U, V, W, or A is used as an axis name or second auxiliary function, data specified at address U, V, W, or A in a G71 to G73 or G76 block is assumed to be that for the multiple repetitive canned cycle.

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70, G71, G72, G73) and specify the cancel command (G40) outside the blocks (from the block specified with P to the block specified with Q) specifying a target finishing figure.

6.5 CANNED CYCLE FOR DRILLING

Canned cycles for drilling make it easier for the programmer to create programs. With a canned cycle, a frequently-used machining operation can be specified in a single block with a G function; without canned cycles, more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.
Table 6.5 (a) lists canned cycles for drilling.

Table 6.5 (a) Canned cycles for drilling

G code	Drilling operation (-Z direction)	Operation in the bottom hole position	Retraction operation (-Z direction)	Applications
G80	------	------	------	Canceling
G81	Cutting feed	------	Rapid traverse	Drilling, Spot drilling
G82	Cutting feed	Dwell	Rapid traverse	Drilling, Counter boring
G83	Cutting feed/intermittent	------	Rapid traverse	Peck drilling cycle
G83.1	Cutting feed/ intermittent	------	Rapid traverse	High-speed peck drilling cycle
G84	Cutting feed	Dwell → Spindle CCW	Cutting feed	Tapping
G84.2	Cutting feed	Dwell → Spindle CCW	Cutting feed	Rigid tapping
G85	Cutting feed	------	Cutting feed	Boring
G89	Cutting feed	Dwell	Cutting feed	Boring

Explanation

The canned cycle for drilling consists of the following six operation sequences.

Operation 1 ..... Positioning of X and Z axis (Another axis may be targeted.)
Operation 2 ..... Rapid traverse up to point R level
Operation 3 ..... Hole machining
Operation 4 ..... Operation at the bottom of a hole
Operation 5 ..... Retraction to point R level
Operation 6 ..... Rapid traverse up to the initial level

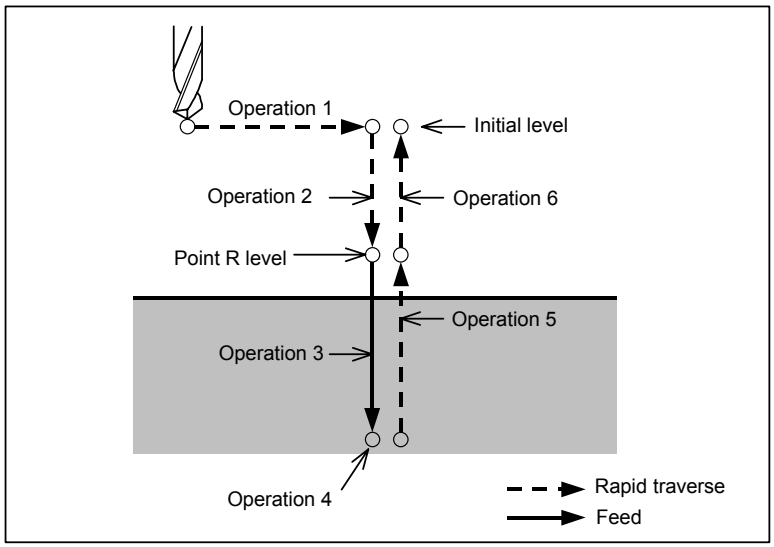


Fig. 6.5 (a) Operation sequence of canned cycle for drilling

- Positioning plane

A positioning plane is determined by plane selection with G17, G18, and G19.
The axes other than the drilling axis are used as positioning axes.

- Drilling axis

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.
The basic axis (X, Y, or Z) that does not exist on the positioning plane or its parallel axis is used as the drilling axis.
The axis address of the drilling axis specified in the same block as the G codes (G81 to G89) determines whether a basic axis or one of parallel axes is used as the drilling axis.
If the axis address of the drilling axis is not specified, the basic axis is used as the drilling axis.

Table 6.5 (b) Positioning plane and drilling axis

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

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

- Example

Suppose parameter No. 1022 is set so that U, V, and W are the parallel axes of X, Y, and Z, respectively.

G17 G81 ..... Z _ _: Drilling axis is Z axis.
G17 G81 ..... W _ _: Drilling axis is W axis.
G18 G81 ..... Y _ _: Drilling axis is Y axis.
G18 G81 ..... V _ _: Drilling axis is V axis.
G19 G81 ..... X _ _: Drilling axis is X axis.
G19 G81 ..... U _ _: Drilling axis is U axis.

G17, G18, and G19 may be specified in a block in which G73 to G89 are not present.

> ⚠ **CAUTION**
> Before switching between drilling axes, cancel canned cycles.

> **NOTE**
> The Z-axis can always be used as the drilling axis by setting FXY (bit 0 of parameter No.5101). When FXY is 0, the Z-axis is always used as the drilling axis.

- Specification of point R

In the Series 0i command format, the distance from the initial level to point R is specified using an incremental value during specification of point R.

In the Series 10/11 command format, the specification method depends on RAB (bit 6 of parameter No. 5102). When RAB = 0, an incremental value is always used for specification. When RAB = 1 for G code system A, an absolute value is used for specification. When RAB = 1 for G code system B, C, an absolute value is used in G90 mode while an incremental value is used in G91 mode.

<table>
<tr><th colspan="4">Series 10/11 command format</th><th>Series 0i command format</th></tr>
<tr><td colspan="3">Parameter RAB (No.5102#6) = 1</td><td>RAB=0</td><td rowspan="4">Incremental</td></tr>
<tr><td>G code system A</td><td colspan="2">G code system B,C</td><td rowspan="3">Incremental</td></tr>
<tr><td rowspan="2">Absolute</td><td>G90</td><td>G91</td></tr>
<tr><td>Absolute</td><td>Incremental</td></tr>
</table>

- Diameter/radius programming

The diameter/radius specification of canned cycles for drilling R command in the series 10/11 command format can be matched with the diameter/radius specification of the drilling axis by setting RDI (bit 7 of parameter No.5102) to 1.

- P

In the following G codes, dwell operation differs between Series 10/11 and Series 10/11.

Operation of this CNC using the Series 10/11 format
In G83, G83.1, G84, and G84.2, dwelling is performed only when address P is specified in a block.
Operation of Series 10/11
In G83 and G83.1, dwelling is not performed.
In G84 and G84.2, dwelling with address P can be performed by setting DWL (bit 1 of parameter No.6200). Address P is modal data.

- Q

Address Q is always specified by using an incremental value during specification of a radius.

- Feedrate for G85 and G89

In G85 and G89, the feedrate from point Z to point R is double the cutting feedrate. For Series 10/11, it is the same as the cutting feedrate.

- Drilling mode

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

- Return point level G98/G99

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.

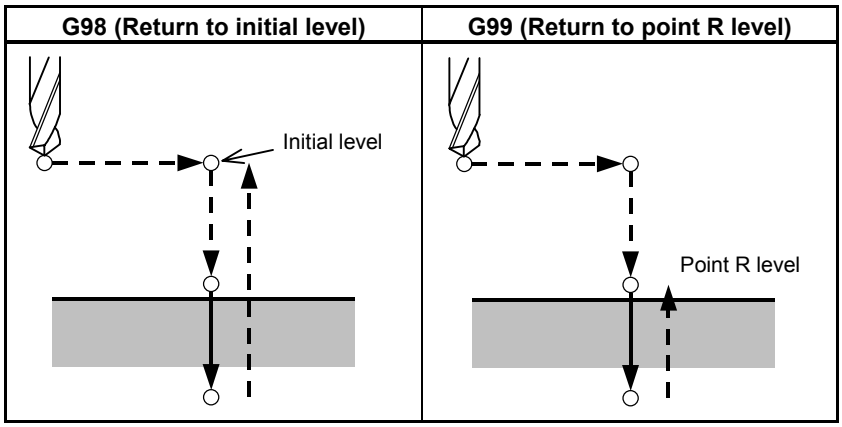


Fig. 6.5 (b) Initial level and point R level

- Number of repeats

To repeat drilling for equally-spaced holes, specify the number of repeats in L_.
L 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 L	The maximum command value = 9999

When L0 is specified, drilling data is just stored without drilling being performed.

NOTE
For L, specify an integer of 0 or 1 to 9999.

- C axis clamp

The M code for C axis clamp can be specified in the Series 0*i* command format, but it cannot be specified in the Series 10/11 command format.

- Disabling the Series 10/11 format

The Series 10/11 command format can be disabled only during a canned cycle for drilling by setting F0C (bit 3 of parameter No.5102) to 1. However, the repetitive count must be specified by address L.

> ⚠ **CAUTION**
> If F16 (bit 3 of parameter No.5102) is set to 1, the settings of RAB (bit 6 of No.5102) and RDI (bit 7 of No.5102) are disabled, and operation when RAB=0 and RDI=0 is performed.

- Cancel

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

Group 01 G codes (Example)

G00 : Positioning (rapid traverse)
G01 : Linear interpolation
G02 : Circular interpolation (CW) or helical interpolation (CW)
G03 : Circular interpolation (CCW) or helical interpolation (CCW)

- Symbols in figures

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

- - ►	Positioning (rapid traverse G00)
——►	Cutting feed (linear interpolation G01)
P	Dwell

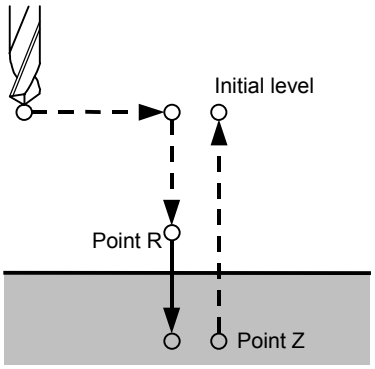
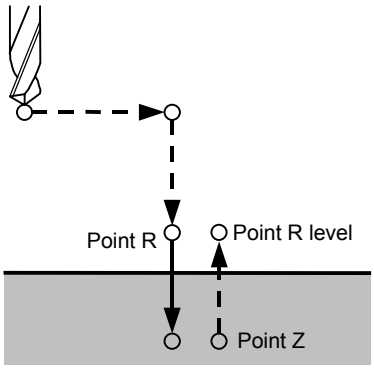
6.5.1 Drilling Cycle, Spot Drilling Cycle (G81)

The normal drilling cycle is used. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

G81 X_ Y_ Z_ R_ F_ L_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)

G81 (G98 mode)	G81 (G99 mode)
	

Explanation

- Operations

Rapid traverse to the point R level is performed after positioning of the X- and Y- axes.
Then, drilling is performed from point R level to point Z.
Escaping moves in rapid traverse.

- Spindle rotation

Before specifying G81, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

If the G81 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G81 is specified. This cancels G81.

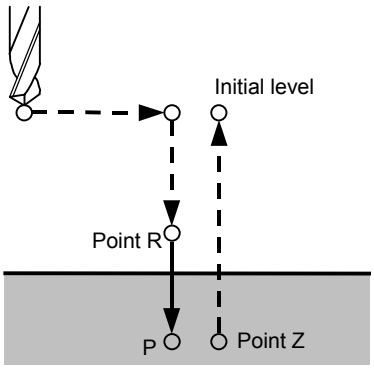
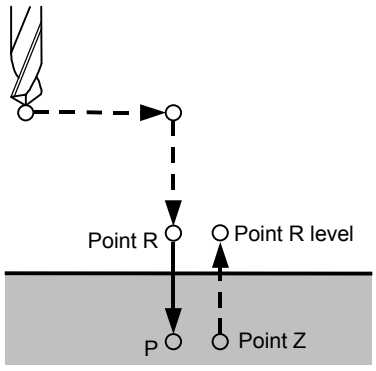
6.5.2 Drilling Cycle, Counter Boring (G82)

The normal drilling cycle is used. Cutting feed is performed to the bottom of the hole, dwelling is performed at the bottom, and then escaping from the bottom is performed in rapid traverse.
The accuracy of the hole depth is improved.

Format

G82 X_ Y_ Z_ R_ P_ F_ L_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)

G81 (G98 mode)	G81 (G99 mode)
	

Explanation

- Operations

Rapid traverse to the point R level is performed after positioning of the X- and Y- axes.
Then, drilling is performed from point R level to point Z.
Dwelling is performed at the bottom of the hole and then escaping is performed in rapid traverse.

- Spindle rotation

Before specifying G82, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

If the G82 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

P must be specified in a block in which drilling is instructed. Otherwise, data is not stored as modal data.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G82 is specified. This cancels G82.

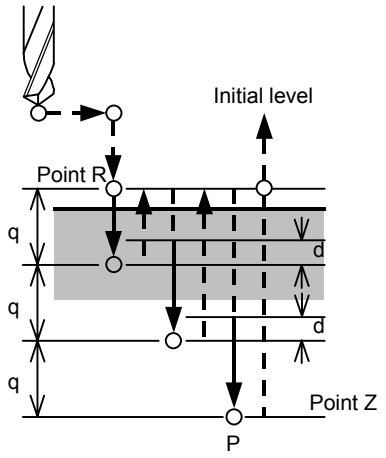
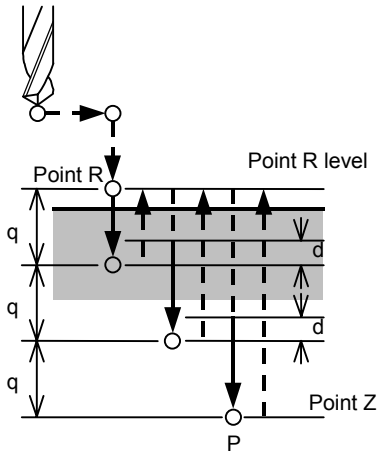
6.5.3 Peck Drilling Cycle (G83)

Peck drilling is performed.
Cutting feed is performed intermittently to the bottom of the hole while chips are discharged.

Format

G83 X_ Y_ Z_ R_ P_ Q_ F_ L_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time
Q_ : Depth of cut for each cutting feed
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)

G83 (G98 mode)	G83 (G99 mode)
	

Explanation

- Operations

Q indicates the depth of cut for each operation and is specified by an incremental value.
In the second and later cutting operations, rapid traverse is changed to cutting feed at the point distance "d" back from the previously drilled position. "d" is set in parameter No. 5115.
A positive value must be specified for Q. A negative value is ignored.

- Spindle rotation

Before specifying G83, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

If the G83 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

Dwelling is performed only when address P is specified in a block.

- Q

Q must be specified in a block in which drilling is instructed. Otherwise, data is not stored as modal data.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G83 is specified. This cancels G83.

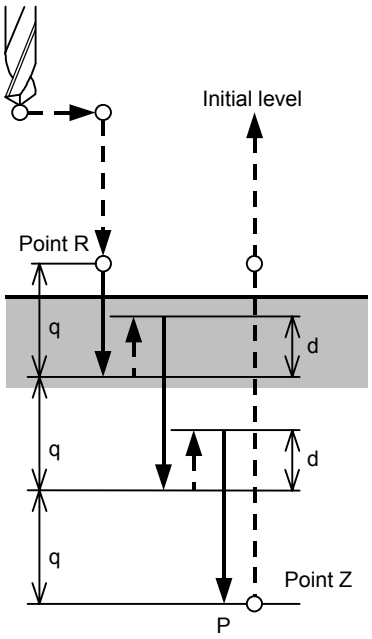
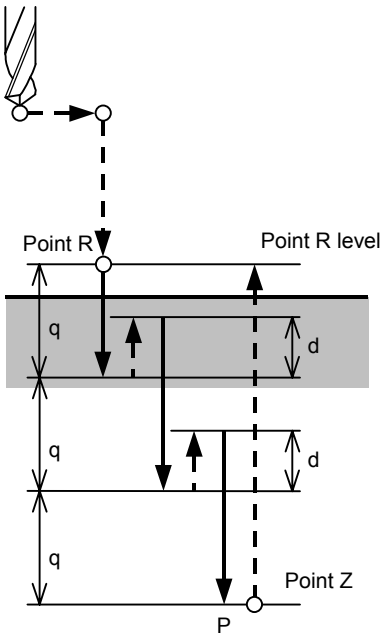
6.5.4 High-speed Peck Drilling Cycle (G83.1)

This cycle performs high-speed peck drilling. It performs cutting feed intermittently while discharging chips.

Format

G83.1 X_ Y_ Z_ R_ P_ Q_ F_ L_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time
Q_ : Depth of cut for each cutting feed
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed)

G83.1 (G98 mode)	G83.1 (G99 mode)
	

Explanation

- Operations

Since intermittent feed in the Z-axis direction makes discharge of chips easier and allows the fine setting of the escaping amount, efficient machining can be performed.
Escape amount d is set in parameter No. 5114.
Escaping moves in rapid traverse.

- Spindle rotation

Before specifying G83.1, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

If the G83.1 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

Dwelling is performed only when address P is specified in a block.

- Q

Q must be specified in a block in which drilling is instructed. Otherwise, data is not stored as modal data.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G83.1 is specified. This cancels G83.1.

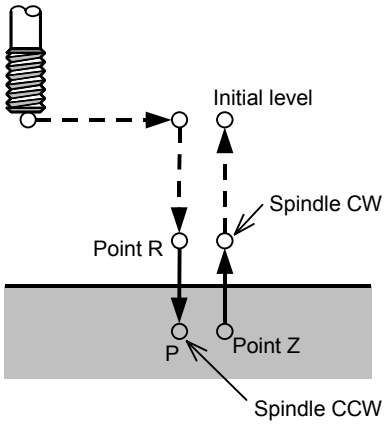
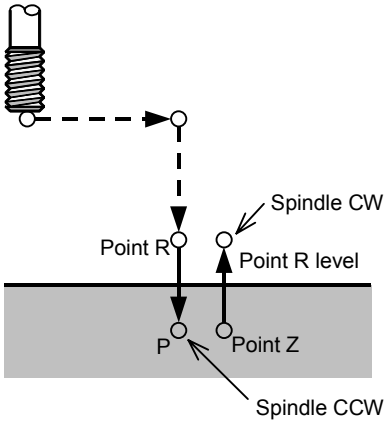
6.5.5 Tapping Cycle (G84)

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

Format

G84 X_ Y_ Z_ R_ P_ F_ L_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)

G84 (G98 mode)	G84 (G99 mode)
	

Explanation

- Operations

Tapping is performed by rotating the spindle clockwise.

> ⚠ **CAUTION**
> Feedrate override is ignored during tapping. In addition, applying feed hold does not stop the machine until return operation is completed.

- Spindle rotation

Before specifying G84, use an auxiliary function (M code) to rotate the spindle.
When drilling in which the distance from the hole position and initial level to the point R level is short is continuously performed, the spindle may not reach the normal speed by the time cutting operation for the hole is ready to be performed. In this case, reserve a time by inserting dwelling by G04 before each drilling operation without specifying repetitive count L.
Since this consideration may not be required depending on the machine type, refer to the manual issued by the machine tool builder.

- Auxiliary function

If the G84 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

Dwelling is performed only when address P is specified in a block.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G84 is specified. This cancels G84.

NOTE
Set M5T (bit 6 of parameter No. 5101) to specify whether the spindle stop command (M05) is specified before the command for rotating the spindle in the forward or reverse direction (M03 or M04) is specified.
For details, refer to the manual issued by the machine tool builder.

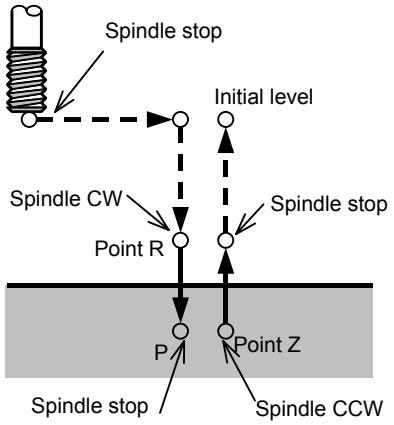
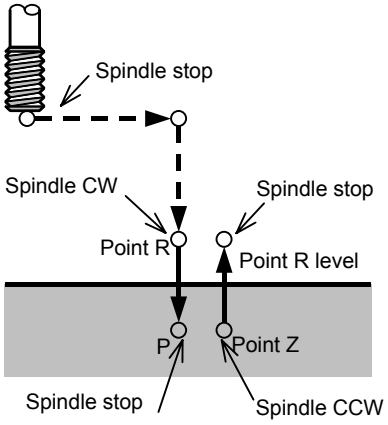
6.5.6 Tapping Cycle (G84.2)

Controlling the spindle motor in the same way as a servo motor executes the high-speed tapping cycle.

Format

G84.2 X_ Y_ Z_ R_ P_ F_ L_ S_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)
S_ : Spindle speed

G84.2 (G98 mode)	G84.2 (G99 mode)
	

A G code cannot discriminate between the front face tapping cycle and side face tapping cycle using Series 10/11 format commands. The drilling axis is determined by plane selection (G17, G18, or G19). Specify the plane selection that becomes equivalent to the front face tapping cycle or side face tapping cycle as appropriate. (When bit 0 (FXY) of parameter No. 5101 is set to 0, the Z-axis is used as the drilling axis. When the bit is set to 1, place selection is as follows.)

Plane selection	Drilling axis
G17 Xp-Yp plane	Zp
G18 Zp-Xp plane	Yp
G19 Yp-Zp plane	Xp

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

Explanation

- Operations

The tool is positioned along the X- and Y-axes, then moves to the point R level in rapid traverse.
Tapping is performed from the point R level to point Z, after which the spindle stops and the tool dwells. Then, the spindle starts reverse rotation, the tool is retracted to the point R level, and the spindle stops. After that, when G98 is specified, the tool moves to the initial level in rapid traverse.
During tapping, the feedrate override and spindle override are assumed to be 100%. For retraction (operation 5), however, a fixed override of up to 2000% can be applied by setting bit 4 (DOV) of parameter No. 5200, bit 3 (OVU) of parameter No. 5201, and parameter No. 5211.

- Thread lead

In the feed per minute mode, feedrate ÷ spindle speed = thread lead.
In the feed per rotation mode, feedrate = thread lead.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling. If the drilling axis is changed in the rigid mode, alarm PS0206 is issued.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

Dwelling is performed only when address P is specified in a block.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G84.2 is specified. This cancels G84.2.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

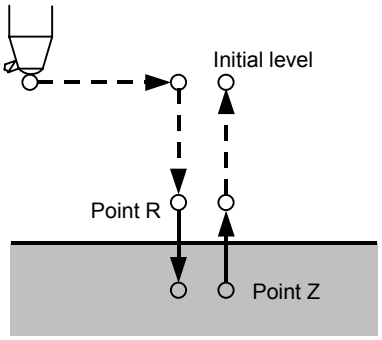
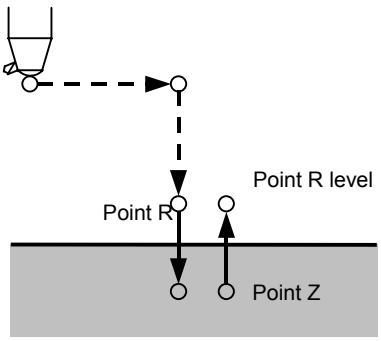
6.5.7 Boring Cycle (G85)

This cycle is used to bore a hole.

Format

G85 X_ Y_ Z_ R_ F_ L_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)

G85 (G98 mode)	G85 (G99 mode)
	

Explanation

- Operations

Rapid traverse to the point R level is performed after positioning of the X- and Y- axes.
Then, drilling is performed from point R level to point Z.
After reaching point Z, return to point R in cutting feed.

- Spindle rotation

Before specifying G85, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

If the G85 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G85 is specified. This cancels G85.

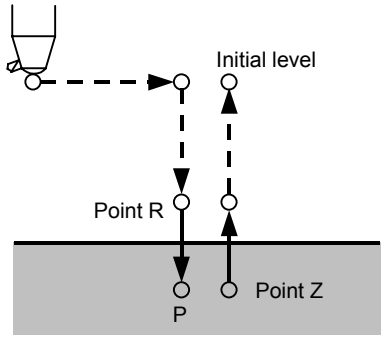
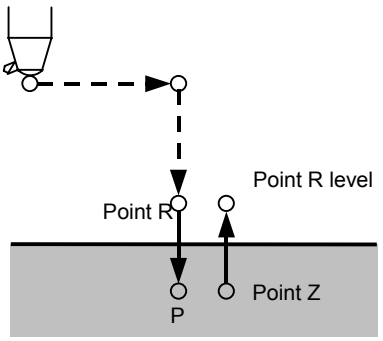
6.5.8 Boring Cycle (G89)

This cycle is used to bore a hole.

Format

G89 X_ Y_ Z_ R_ P_ F_ L_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
L_ : Number of repeats (When it is needed.)

G89 (G98 mode)	G89 (G99 mode)
	

Explanation

- Operations

This is the same as G85, but dwelling is performed at the bottom of the hole.

- Spindle rotation

Before specifying G89, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

If the G89 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

P must be specified in a block in which drilling is instructed. Otherwise, data is not stored as modal data.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G89 is specified. This cancels G89.

6.5.9 Canned Cycle for Drilling Cancel (G80)

G80 cancels canned cycle for drilling.

Format

G80 ;

Explanation

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

6.5.10 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, 6 in Fig. 6.5 (a).
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. If there is a remaining repetitive count at the end of operation 6, the operation is stopped by feed hold. If there is no remaining repetitive count, the operation is stopped in the single block stop state.

- 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 6.

- Override

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

7 AXIS CONTROL FUNCTIONS

Chapter 7, "AXIS CONTROL FUNCTIONS", consists of the following sections:

7.1 POLYGON TURNING (G50.2, G51.2).....339
7.2 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6).....345

7.1 POLYGON TURNING (G50.2, G51.2)

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

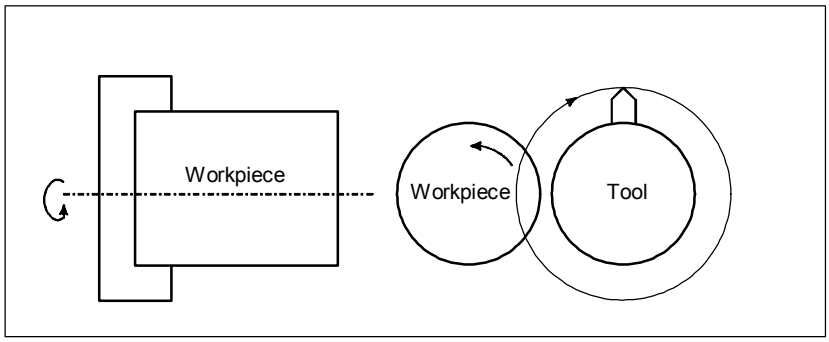


Fig. 7.1 (a) Polygon turning

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

As the tool rotary axis, one of the following can be used:

- CNC controlled axis (servo axis)
- Second spindle (Two serial spindles are connected.)

Polygonal machining performed using a servo axis as the tool rotary axis is referred to as polygon turning. Polygonal machining performed using the second spindle as the tool rotary axis is referred to as polygon turning with two spindles.

Function name	Workpiece axis	Tool rotary axis
Polygon turning	Spindle (Either an analog spindle or serial spindle is usable. However, a detector equivalent to a position coder is required.)	Servo axis
Polygon turning with two spindles	Spindle (Serial spindle)	Spindle (Serial spindle)

Explanation

A CNC controlled axis (servo axis) is assigned to the tool rotary axis. This rotary axis of tool is called Y-axis in the following description. As the workpiece axis (spindle), either a serial spindle or analog spindle can be used.

The Y-axis is controlled by the G51.2 command, so that the ratio of the rotation speeds of the spindle (previously specified by S-command) and the tool becomes the specified ratio.

When simultaneous start is specified by G51.2, the one-rotation signal sent from the position codes set on the spindle is detected. After one-rotation signal detection, the Y-axis is controlled using the rotation ratio of the spindle and Y-axis specified by P and Q. So, a position coder needs to be attached to the spindle. This control will be maintained until the polygon turning cancel command is executed (G50.2). Polygon turning is cancelled by any of the following in addition to the G50.2 command:

(1) Power off
(2) Emergency stop
(3) Servo alarm
(4) Reset (external reset signal ERS, reset/rewind signal RRW, and RESET key on the MDI panel)
(5) Occurrence of alarms PS0217 to PS0221, PS0314, and PS05018

NOTE

1 Before polygon turning, reference position return operation on the Y-axis needs to be specified to determine the rotation start position of the tool. This reference position return operation is performed by detecting a deceleration limit as in the case of manual reference position return operation. (By setting bit 7 (PLZ) of parameter No. 7600, reference position return operation can be performed without detecting a deceleration limit.)
2 The rotation direction on the Y-axis is determined by the sign of Q, and is not affected by the rotation direction of the position coder.
3 Among the current position display of the Y axis, the display for the machine coordinate value (MACHINE) changes from a range of 0 to the amount of movement per revolution as the Y axis moves. Absolute and relative coordinate values are not updated. So, when specifying an absolute programming for the Y-axis after polygon turning mode cancellation, set a workpiece coordinate system after reference position return operation.

NOTE

4 For the Y-axis engaged in polygon turning, jog feed and handle feed are disabled.
5 For the Y-axis not engaged in polygon turning, a move command can be specified as in the case of other controlled axes.
6 The Y-axis engaged in polygon turning is not counted in the number of simultaneously controlled axes.
7 One workpiece must be machined using a fixed spindle speed until the workpiece is finished.
8 Polygon turning with two spindles cannot be used at the same time.
9 G50.2 is the G code for suppressing buffering.

⚠ **CAUTION**

1 During polygon turning, threading cannot be performed.
2 For the Y-axis engaged in synchronous operation, the signals below are valid or invalid:
Signals valid for the Y-axis
- Machine lock
- Servo-off

Signals invalid for the Y-axis
- Feed hold
- Interlock
- Override
- Dry run
(At dry run time, however, the one-rotation signal is not awaited in a G51.2 block.)

Format

G50.2 Polygon turning cancel

G51.2 P_ Q_ ; Start of polygon turning

P,Q: Rotation ratio of spindle and Y-axis

Specify range:

P: Integer from 1 to 999

Q: Integer from -999 to -1 or from 1 to 999

When Q is a positive value, Y-axis makes positive rotation.

When Q is a negative value, Y-axis makes negative rotation.

NOTE

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

Example

```
G00 X100. 0 Z20.0 S1000.0 M03 ;
                          (Workpiece rotation speed 1000 (min-1))
G51.2 P1 Q2 ;             (Tool rotation start (tool rotation speed
                          2000 (min-1))
G01 X80.0 F10.0 ;         (X-axis in-feed)
G04 X2.0 ;                (Waiting 2 seconds)
G00 X100.0 ;              (X-axis escape)
G50.2 ;                   (Tool rotation stop)
M05 S0 ;                  (Spindle stop )
```

- Principle of polygon turning

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).

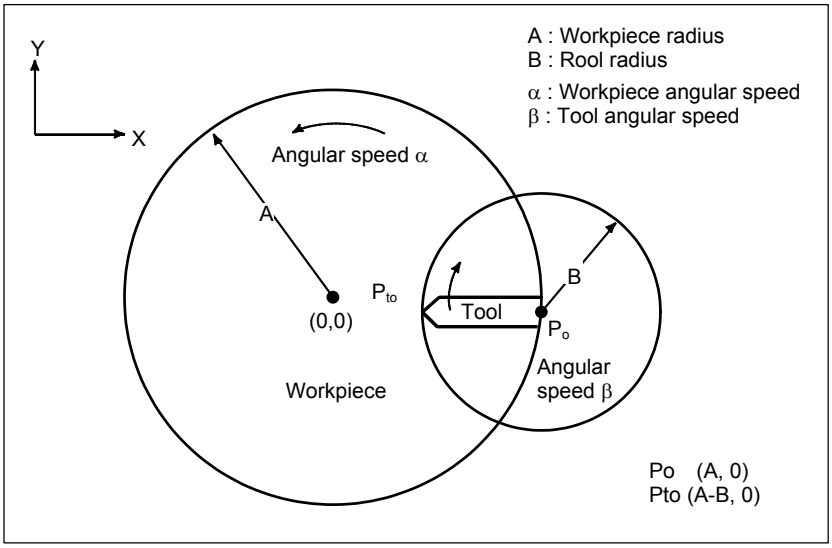


Fig. 7.1 (b) Principle of polygon turning

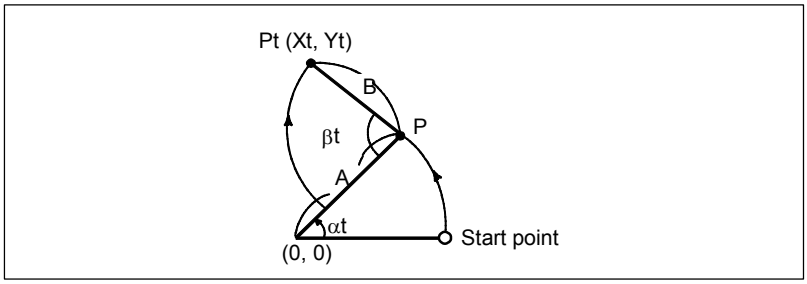


Fig. 7.1 (c) Tool nose position

In this case, the tool nose position Pt (Xt, Yt) after time t is expressed by equations 1 and 2:

$Xt=A\cos\alpha t-B\cos(\beta-\alpha)t$ (Equation 1)

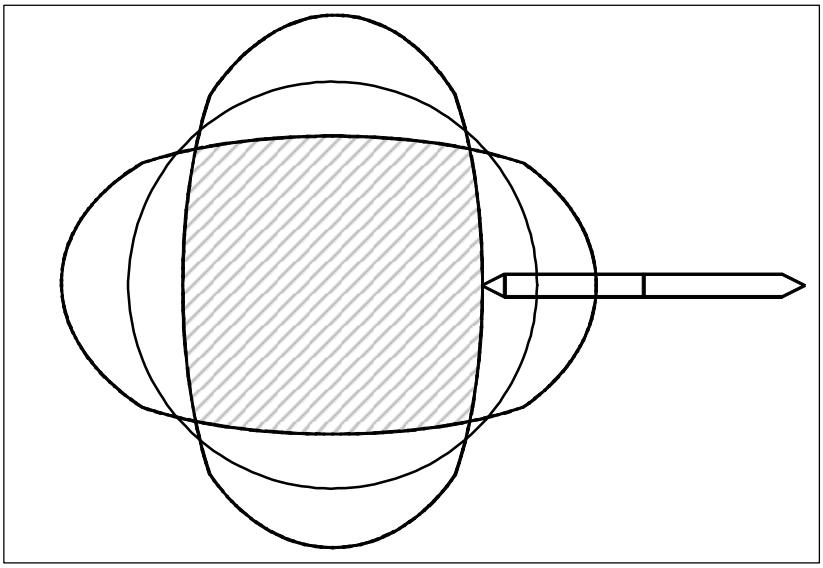
$Yt=A\sin\alpha t+B\sin(\beta-\alpha)t$ (Equation 2)

Assuming that the rotation ration of workpiece to tool is 1:2, namely, $\beta=2\alpha$, equations 1 and 2 are modified as follows:

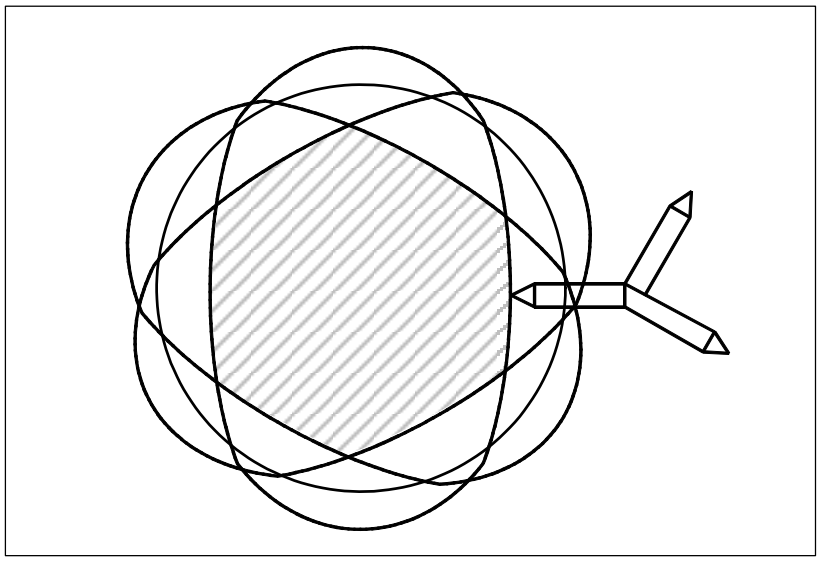
$Xt=A\cos\alpha t-B\cos\alpha t=(A-B)\cos\alpha t$ (Equation 1)'

$Xt=A\sin\alpha t+B\sin\alpha t=(A+B)\sin\alpha t$ (Equation 2)'

These equations indicate 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. 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
For the maximum rotation speed of the tool, see the instruction manual supplied with the machine. Do not specify a spindle speed higher than the maximum tool speed or a ratio to the spindle speed that results in a speed higher than the maximum tool speed.

7.2 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6)

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

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

Format

G51.4 P_ Q_ (L_) ; Synchronous control start (L is omissible.)
G50.4 Q_ ; Synchronous control cancel

P: Synchronous master axis ID number
Q: Synchronous slave axis ID number

L: Parking command
1: Master parking (slave parking cancel)
2: Slave parking (master parking cancel)
0: No parking (parking cancel)
(When L is omitted, the specification of L0 is assumed.)

G51.5 P_ Q_ ; Composite control start
G50.5 P_ Q_ ; Composite control cancel

P: Composite axis 1 ID number
Q: Composite axis 2 ID number

G51.6 P_ Q_ ; Superimposed control start
G50.6 Q_ ; Superimposed control cancel

P: Superimposed master axis ID number
Q: Superimposed slave axis ID number

As an ID number, set a unique value for identifying each axis in parameter No. 12600 for both of P and Q.

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

Explanation

Synchronous control

Synchronous control is performed with the G51.4/G50.4 commands, instead of simultaneously controlled axis selection signals.

Parameter setting examples for a 2-path system

- Parameter No.12600

	Path 1	Path 2
X	101	201
Z	102	202

- Parameter No.8180

	Path 1	Path 2
X	0	0
Z	0	102

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Operation
N10 M100 ;	M100 ;	Synchronization between paths 1 and 2
N20 G51.4 P102 Q202 ;		Start of Z1-Z2 synchronous control
N30 M101 ;	M101 ;	Synchronization between paths 1 and 2
N40 G00 Z100.;	Synchronous slave movement	Z1-Z2 synchronous control
N50 M102 ;	M102 ;	Synchronization between paths 1 and 2
N60 G50.4 Q202 ;		Cancellation of Z1-Z2 synchronous control
N70 M103 ;	M103 ;	Synchronization between paths 1 and 2

- Start of synchronous control

N20 G51.4 P102 Q202 : Start of synchronous control with the Z1-axis being a master axis and the Z2-axis being a slave axis

- Cancellation of synchronous control

N60 G50.4 Q202 : Cancellation of synchronous control with the Z1-axis being a master axis and the Z2-axis being a slave axis

- Parking

Use the L command to specify the start and cancellation of the parking of synchronous axes.
If the L command is omitted or if the L0 command is issued, the parking of both synchronous master axis and synchronous slave axis is canceled.

- Parameter check

If the axis number corresponding to the P specified with G51.4 is not set in slave axis parameter No. 8180, alarm PS5339 is issued.

Composite control

Composite control is performed with the G51.5/G50.5 commands, instead of composite control axis selection signals.

Parameter setting examples for a 2-path system

- Parameter No.12600

	Path 1	Path 2
X	101	201
Z	102	202

- Parameter No.8183

	Path 1	Path 2
X	0	101
Z	0	102

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Operation
N10 M100 ;	M100 ;	Synchronization between paths 1 and 2
N20 G51.5 P101 Q201 ; N30 G51.5 P102 Q202 ;		Start of X1-X2 composite control Start of Z1-Z2 composite control
N40 M101 ;	M101 ;	Synchronization between paths 1 and 2
N50 G00 X 100. Z100.;	Composite movement	X1-X2 and Z1-Z2 composite control
N60 M102 ;	M102 ;	Synchronization between paths 1 and 2
N70 G50.5 P101 Q201 ; N80 G50.5 P102 Q202 ;		Cancellation of X1-X2 composite control Cancellation of Z1-Z2 composite control
N90 M103 ;	M103 ;	Synchronization between paths 1 and 2

- Start of composite control

N20 G51.5 P101 Q201 : Start of composite control on the X1- and X2-axes
N30 G51.5 P102 Q202 : Start of composite control on the Z1- and Z2-axes

- Cancellation of composite control

N70 G50.5 P101 Q201 : Cancellation of composite control on the X1- and X2-axes
N80 G50.5 P102 Q202 : Cancellation of composite control on the Z1- and Z2-axes

- Parameter check

If the composite control axis corresponding to the P or Q specified with G51.5/G50.5 is not set in parameter No. 8183, alarm PS5339 is issued.

Superimposed control

Superimposed control is performed with the G51.6/G50.6 commands, instead of superimposed control axis selection signals.

Parameter setting examples for a 2-path system

- Parameter No.12600

	Path 1	Path 2
X	101	201
Z	102	202

- Parameter No.8186

	Path 1	Path 2
X	0	0
Z	0	0

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Operation
N10 M100 ;	M100 ;	Synchronization between paths 1 and 3
N20 G51.6 P102 Q202 ;		Start of Z1-Z2 superimposed control
N30 M101 ;	M101 ;	Synchronization between paths 1 and 3
N40 G00 Z100.;	G00 Z-200.;	Z1-Z2 superimposed control (Z+100 superimposed on Z2)
N50 M102 ;	M102 ;	Synchronization between paths 1 and 3
N60 G50.6 Q202 ;		Cancellation of Z1-Z2 superimposed control
N70 M103 ;	M103 ;	Synchronization between paths 1 and 3

- Start of superimposed control

N20 G51.6 P102 Q202 : Start of superimposed control with the Z1-axis being a master axis and the Z2-axis being a slave axis

- Cancellation of superimposed control

N60 G50.6 Q202 : Cancellation of superimposed control with the Z1-axis being a master axis and the Z2-axis being a slave axis

- Parameter check

If the axis number corresponding to the P specified with G51.6 is not set in superimposed slave axis parameter No. 8186, alarm PS5339 is issued.

NOTE

1 If G codes (G50.4/G50.5/G50.6) for canceling synchronous, composite, and superimposed control with program commands are issued for axes under synchronous, composite, and superimposed control with DI signals, synchronous, composite, and superimposed control is canceled.

2 If the synchronous control axis selection signal, composite control axis selection signal, and superimposed control axis selection signal are changed from '1' to '0' for axes under synchronous, composite, and superimposed control with program commands, synchronous, composite, and superimposed control is canceled.

8 2-PATH CONTROL FUNCTION

Chapter 8, "2-PATH CONTROL FUNCTION", consists of the following sections:

8.1 OVERVIEW 350
8.2 WAITING FUNCTION FOR PATHS 351
8.3 COMMON MEMORY BETWEEN EACH PATH.....352
8.4 SPINDLE CONTROL BETWEEN EACH PATH.....354
8.5 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL 355
8.6 BALANCE CUT (G68, G69).....358

8.1 OVERVIEW

The 2-path control function is intended to perform two types of machining simultaneously and independently. The function applies to lathes and automatic lathes with which two tool posts operate simultaneously to machine one workpiece.

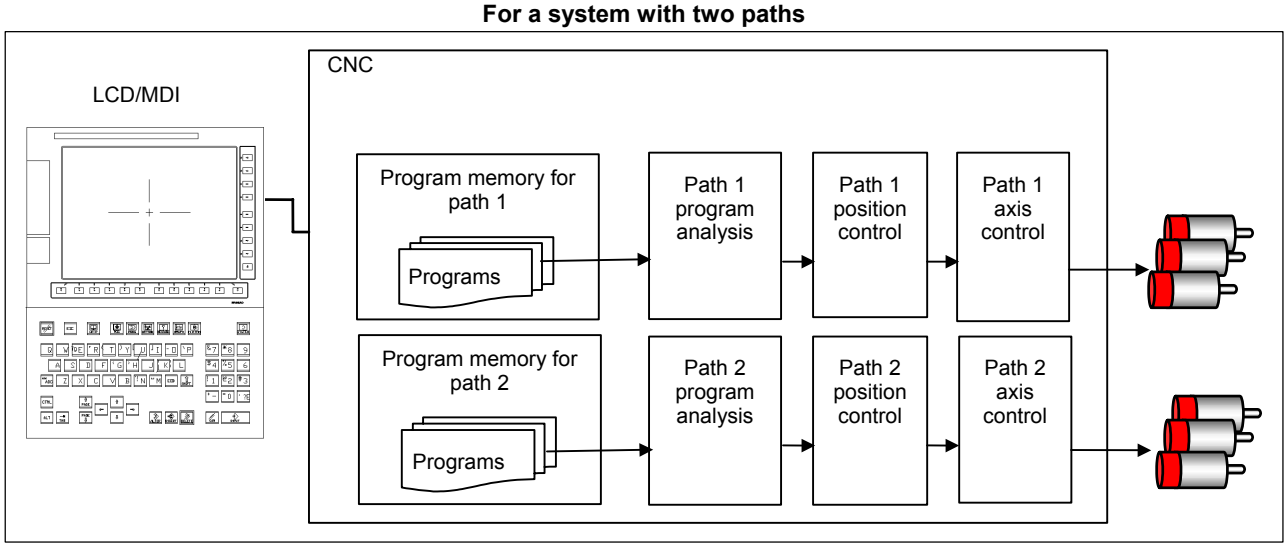
To control two paths to machine one workpiece simultaneously, the machining program for each path is stored in the program memory for the path. In automatic operation, this function selects the program for path 1 and that for path 2 from the program memory for the relevant path. When the paths are activated, the programs selected for the relevant tool posts are executed simultaneously and independently.

To make tool posts 1 and 2 synchronous during machining, the waiting function can be used.

Other functions specific to 2-path control that include the following functions can also be used: Interference check for each path, balance cutting, synchronous/composite/superimposed control, spindle control between each path, and common memory between each path.

Only one LCD/MDI is provided for two paths. The path selection signal is used to switch LCD/MDI operation or display between paths 1 and 2.

For a system with two paths



8.2 WAITING FUNCTION FOR PATHS

Overview

Control based on M codes is used to cause one path to wait for the other during machining. When an M code for waiting is specified in a block for one path during automatic operation, the another path waits for the same M code to be specified before staring the execution of the next block.
A range of M codes used as M codes for waiting is to be set in the parameters (Nos. 8110 and 8111) beforehand. Waiting can be ignored using a signal.

Format

Mm ;
m: Number of an M code for waiting

Explanation

⚠ **CAUTION**
1 An M code for waiting must always be specified in a single block.
2 Unlike other M codes, the M code for waiting is not output to the PMC.
3 If the operation of a single path is required, the M code for waiting need not be deleted. By using the signal to specify that waiting be ignored (NOWT), the M code for waiting in a machining program can be ignored. For details, refer to the manual supplied by the machine tool builder.
4 If using a waiting M code in 1 block multiple-M code command, be sure to specify it as the first M code.

8.3 COMMON MEMORY BETWEEN EACH PATH

Overview

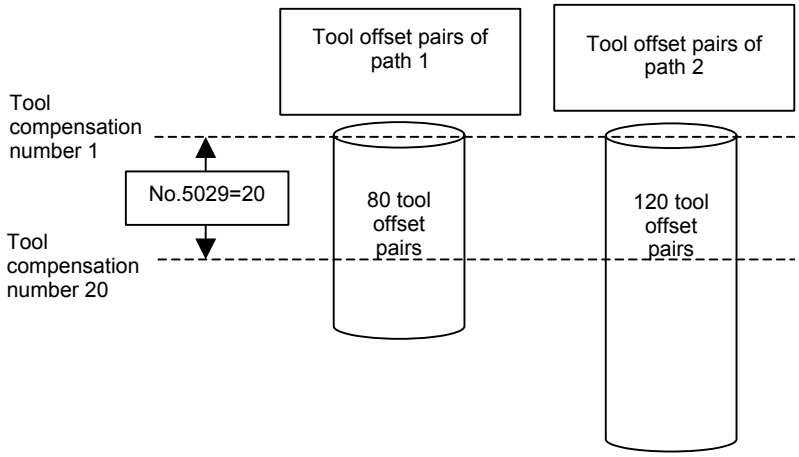
In a 2-path system, this function enables data within the specified range to be accessed as data common to both paths. The data includes tool compensation memory and custom macro common variables.

Explanation

The path common memory function enables the following operations.

- Tool compensation memory

Part or all of tool compensation memory for individual paths can be used as common data by setting parameter No. 5029.

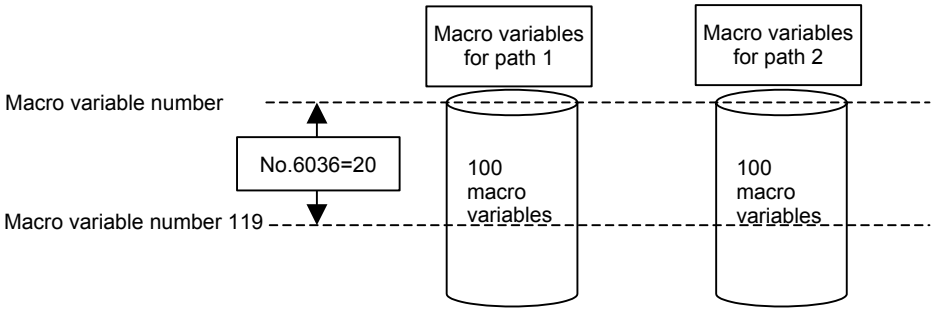


NOTE

1 The same unit for tool compensation (bits 0 and 1 of parameter No. 5042) must be set for both paths.
2 Set a value less than the number of tool compensation values for each path for parameter No. 5029.
3 When the setting of parameter No. 5029 exceeds the number of tool compensation values for each path, the smaller of the numbers of tool compensation values for both paths is used as a common number.
4 For details, refer to the relevant manual of the machine tool builder.

- Custom macro common variables

All or part of custom macro common variables #100 to #199 and #500 to #999 can be used as common data by setting parameters No. 6036 (#100 to #199) and 6037 (#500 to #999).



NOTE

If the value of parameter No. 6036 or 6037 exceeds the maximum number of custom macro common variables, the maximum number of custom macro common variables is assumed.

8.4 SPINDLE CONTROL BETWEEN EACH PATH

Overview

This function allows a workpiece attached to one spindle to be machined simultaneously with two tool posts and each of two workpieces attached to each of two spindles to be machined simultaneously with each of two tool posts.

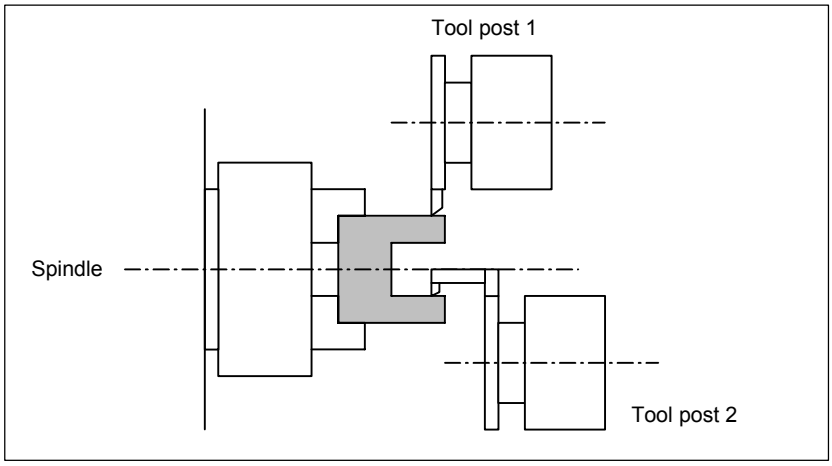


Fig. 8.4 (a) Application to a lathe with one spindle and two tool posts

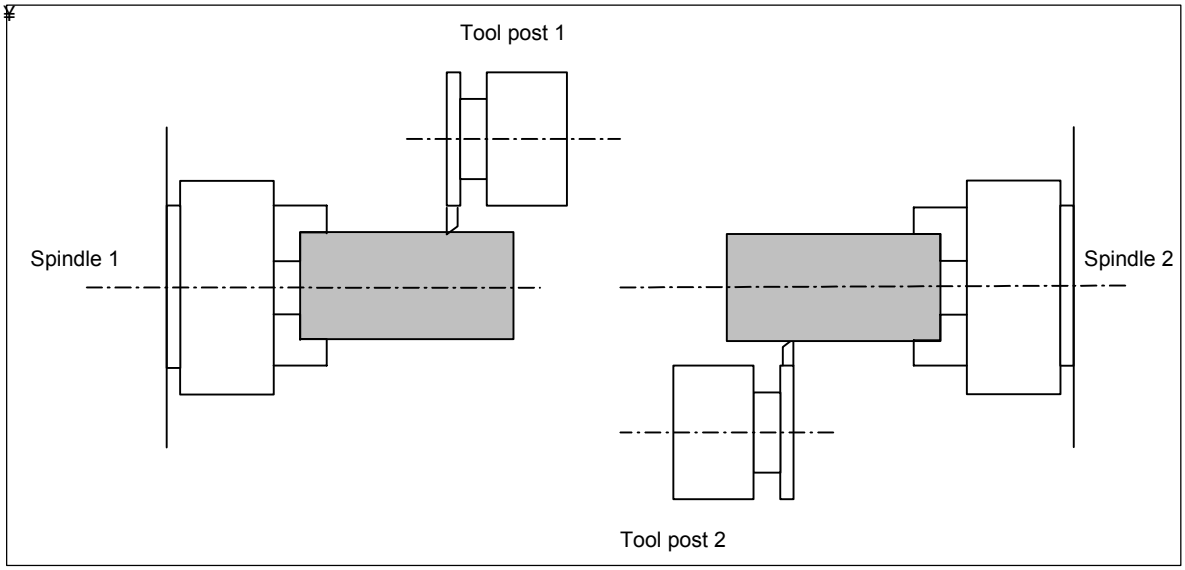


Fig. 8.4 (b) Application to a lathe with two spindles and two tool posts

The spindle belonging to each path can generally be controlled by programmed commands for the path. With path spindle command selection signals, programmed commands for any path can control the spindle belonging to any path.

NOTE
For the method of spindle command selection, refer to the relevant manual of the machine tool builder.

8.5 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL

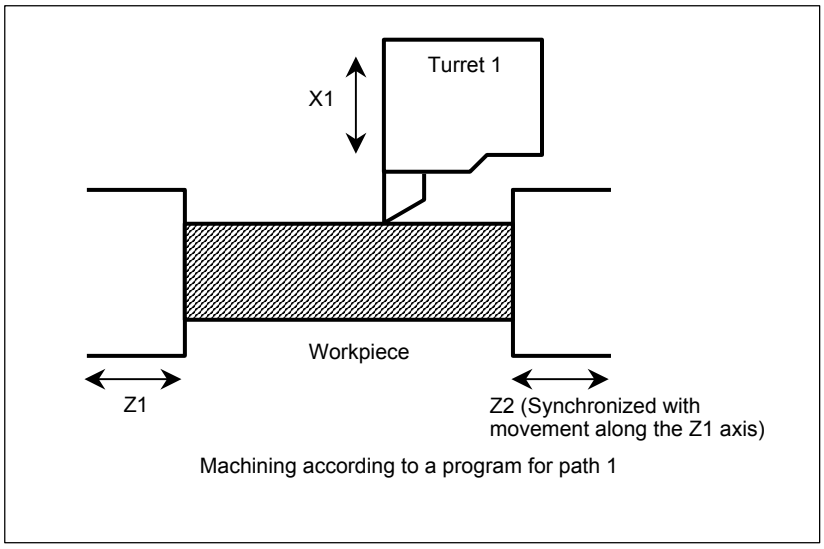
Overview

In 2-path control, the synchronous control function, composite control function, and superimposed control function enable synchronous control, composite control, and superimposed control in a single path system or between 2-path systems.

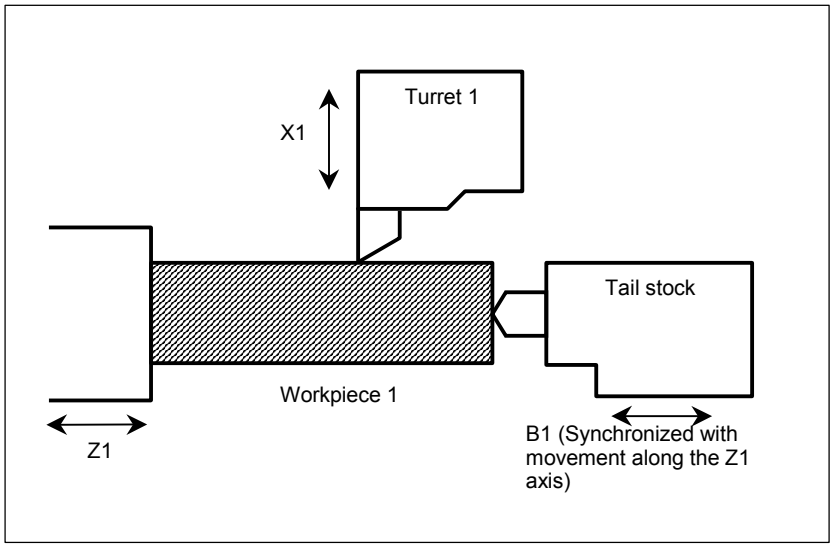
Explanation

- Synchronous control

- Synchronizes movement along an axis of one system with that along an axis of the another path.
  Example)
  Synchronizing movement along the Z1 (master) and Z2 (slave) axes



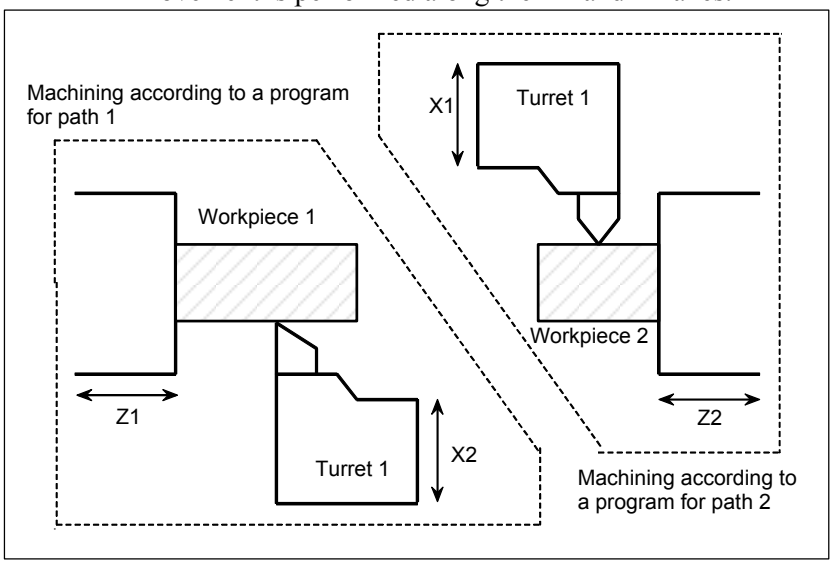
- Synchronizes movement along an axis of one path with that along another axis of the same path.
  Example)
  Synchronizing movement along the Z1 (master) and B1 (slave) axes



- Composite control

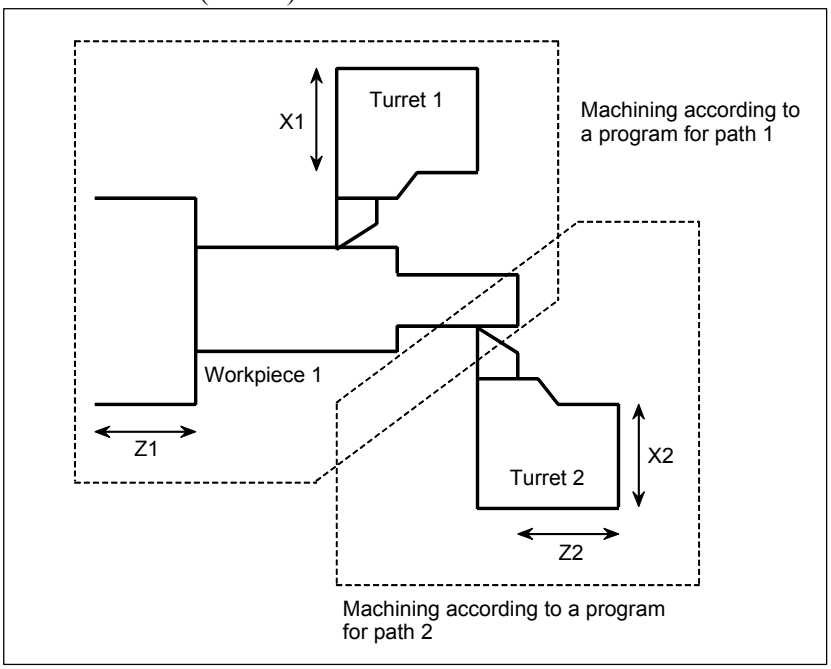
- Exchanges the move commands for different axes of different path.

  Example)
  Exchanging the commands for the X1 and X2 axes
  → Upon the execution of a command programmed for path 1, movement is performed along the X2 and Z1 axes.
  Upon the execution of a command programmed for path 2, movement is performed along the X1 and Z2 axes.



- Superimposed control

- Provides a move command of an axis for a different axis in another path.
  Example)
  Providing the Z2 (slave) axis with a move command specified for the Z1 (master) axis

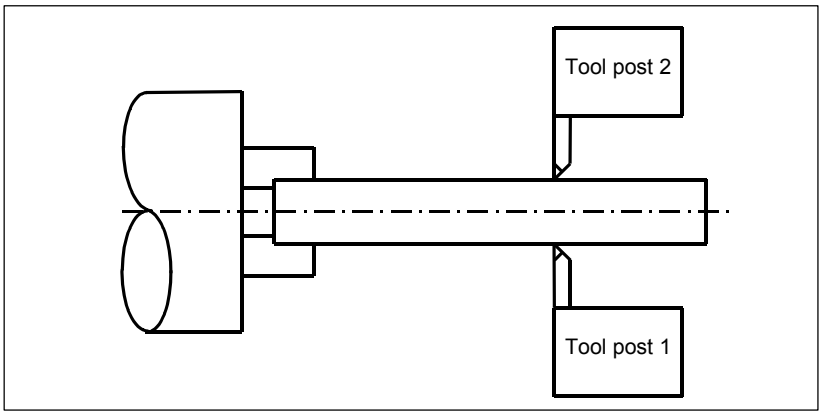


NOTE
The method used to specify synchronous, composite, or superimposed control varies with the machine tool builder. For details, refer to the manual supplied by the machine tool builder.

8.6 BALANCE CUT (G68, G69)

Overview

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



Format

```
G68 ; Balance cut mode on
G69 ; Balance cut mode cancel
```

Explanation

When G68 is specified in the programs for tool posts 1 and 2, the balance cut mode is set to on. When G69 is specified, the balance cut mode is canceled.
When G68 or G69 is specified for either tool post, the tool post waits until G68 or G69 is specified for the other tool post.

In the balance cut mode, balance cutting is performed when a move command in cutting feed is specified for both tool posts.
In balance cutting, the tool posts start moving simultaneously in each block in which a move command in cutting feed is specified.
Specify G68 or G69 in a single block.

NOTE
1 Balance cutting is not performed in dry run or machine lock state. G68 or G69 specified for one tool post is synchronized with G68 or G69 specified for the other tool post, however.
2 In the balance cut mode, G68 specified for one tool post is not synchronized with G68 specified for the other tool post. In the balance cut cancel mode, G69 specified for one tool post is not synchronized with G69 specified for the other tool post.
3 Balance cutting is not performed in a block in which 0 is specified for the travel distance.
4 Balance cutting is not performed when rapid traverse is specified.

Caution

⚠ CAUTION

1 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. Override and interlock can be applied independently to both tool posts. The settings for both tool posts that are related to override and interlock must also be the same to perform balance cutting.
2 After feed hold is applied during execution of balance cutting for both tool posts, balance cutting is not performed at the restart. Balance cutting is performed when the next move command is executed for both tool posts.

NOTE

1 Time delay before the pulse distribution of both tool posts is started is 2 msec or shorter.
2 Overlap is invalid. In the balance cut mode, synchronization is established at the start of each move block in which cutting is specified, so movement can momentarily stop.
3 In the balance cut mode, continuous thread cutting overlap is also invalid. Perform continuous thread cutting in the balance cut cancel mode.
4 To establish synchronization of the pulse distribution in a block in which thread cutting is specified, the same position coder must be selected.
5 The cancel mode (G69) is unconditionally set by a reset.
6 When the option "mirror image for double turret" is selected, the balance cut function cannot be used. To use the option “mirror image for double turret”, set bit 0 (NVC) of parameter No. 8137 to 0 to disable the balance cut function.

III. OPERATION

1 DATA INPUT/OUTPUT

By using the memory card interface on the left side of the display, information written in a memory card is read into the CNC and information is written from the CNC to a memory card.
The following types of data can be input and output:
1. Y-axis offset data

The above data can be input and output on the screens used for displaying and setting the data and on the ALL IO screen.

Chapter 1, "DATA INPUT/OUTPUT", consists of the following sections:

1.1 INPUT/OUTPUT ON EACH SCREEN 364
1.1.1 Inputting and Outputting Y-axis Offset Data 364
1.1.1.1 Inputting Y-axis offset data 364
1.1.1.2 Outputting Y-axis Offset Data.....365
1.2 INPUT/OUTPUT ON THE ALL IO SCREEN 366
1.2.1 Inputting and Outputting Y-axis Offset Data 367

1.1 INPUT/OUTPUT ON EACH SCREEN

Data can be input to and output from the Y-axis offset screens.

1.1.1 Inputting and Outputting Y-axis Offset Data

1.1.1.1 Inputting Y-axis offset data

Y-axis offset data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. The Y-axis offset data that is registered in the memory and has a corresponding data number is replaced with data input by this operation.

Inputting Y-axis offset data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key .
4 Press the continuous menu key  several times until soft key [Y OFFSET] appears.
5 Press soft key [Y OFFSET] to display the Y-axis offset data. screen.
6 Press soft key [(OPRT)].
7 Press the continuous menu key  several times until soft key [F INPUT] appears.
8 Press soft key [F INPUT].
9 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.
10 Press soft key [EXEC].
This starts reading the Y-axis offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input, press soft key [CANCEL].

1.1.1.2 Outputting Y-axis Offset Data

Y-axis offset data is output in a output format from the memory of the CNC to a memory card.

Outputting Y-axis offset data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for outputting.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key .
4 Press the continuous menu key  several times until soft key [Y OFFSET] appears.
5 Press soft key [Y OFFSET] to display the Y-axis offset data. screen.
6 Press soft key [(OPRT)].
7 Press the continuous menu key  several times until soft key [F OUTPUT] appears.
8 Press soft key [F OUTPUT].
9 Type the file name that you want to output.
If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
10 Press soft key [EXEC].
This starts outputting the Y-axis offset data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

1.2 INPUT/OUTPUT ON THE ALL IO SCREEN

Just by using the ALL IO screen, you can input and output Y-axis offset data and tool offset / 2nd geometry data.

The following explains how to display the ALL IO screen:

Displaying the ALL IO screen (for 8.4/10.4-inch display unit)

Procedure

1 Press function key 

.

2 Press the continuous menu key  several times until soft key [ALL IO] is displayed..

3 Press soft key [ALL IO] to display the ALL IO screen.

The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

1.2.1 Inputting and Outputting Y-axis Offset Data

With the lathe system, Y-axis offset data can be input and output using the ALL IO screen.

Inputting Y-axis offset data (for 8.4/10.4-inch display unit)

Procedure

1 On the ALL IO screen, press the continuous menu key ▷ several times until soft key [Y OFFSET] is displayed.
2 Press soft key [Y OFFSET].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [N INPUT].
6 Set the name of the file that you want to input.
Type a file name, and press soft key [F NAME].
If the input file name is omitted, default file name "TOOLOFST.TXT" is assumed.
7 Press soft key [EXEC].
This starts reading the Y-axis offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input, press soft key [CANCEL].

Outputting Y-axis offset data (for 8.4/10.4-inch display unit)

Procedure

1 On the ALL IO screen, press the continuous menu key ▷ several times until soft key [Y OFFSET] is displayed.
2 Press soft key [Y OFFSET].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [F OUTPUT].
6 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
7 Press soft key [EXEC].
This starts outputting the Y-axis offset data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press soft key [CANCEL].

2 SETTING AND DISPLAYING DATA

Chapter 2, "SETTING AND DISPLAYING DATA", consists of the following sections:

2.1 SCREENS DISPLAYED BY FUNCTION KEY  369

2.1.1 Setting and Displaying the Tool Offset Value 370
2.1.2 Direct Input of Tool Offset Value 374
2.1.3 Direct Input of Tool Offset Value Measured B.....376
2.1.4 Counter Input of Offset value.....379
2.1.5 Setting the Workpiece Coordinate System Shift Value380
2.1.6 Setting the Y-Axis Offset 382
2.1.7 Chuck and Tail Stock Barriers 385

2.1 SCREENS DISPLAYED BY FUNCTION KEY 

Press function key  to display or set tool compensation values and other data.
This section explains the display and setting of the following data items:

1. Tool offset value
2. Workpiece coordinate system shift value
3. Y-axis offset value
4. Chuck and tail stock barriers

For the display and setting of data other than the above, refer to "USER'S MANUAL (Common to Lathe System/Machining Center System)" (B-64304EN).

2.1.1 Setting and Displaying the Tool Offset Value

Dedicated screens are provided for displaying and setting tool offset values and tool nose radius compensation values.
Whether to use tool geometry and wear compensation can be selected using bit 6 (NGW) of parameter No. 8136; whether to use tool nose radius compensation can be selected using bit 7 (NCR) of parameter No. 8136. (0: Use the function./1: Does not use the function.)

Setting and displaying the tool offset value and the tool nose radius compensation value

Procedure

1 Press function key .

When using a 2-path system, select, in advance, a path for which a tool offset value is to be set, by using the path selection switch.

2 Press chapter selection soft key [OFFSET] or press function key  several times until the tool compensation screen is displayed.

Different screens are displayed depending on whether tool geometry offset, wear offset, or neither is applied.

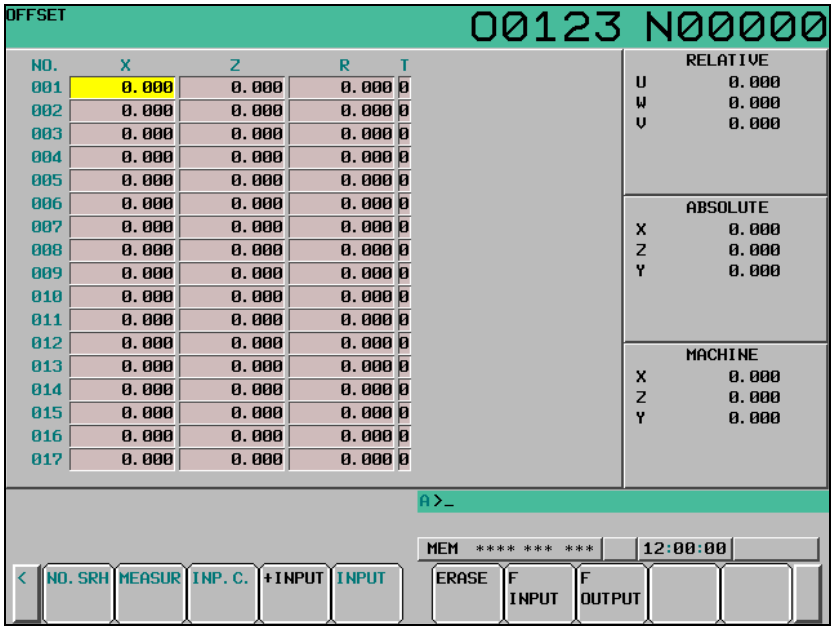


Fig. 2.1.1 (a) When tool geometry/wear offset is not used (10.4-inch)

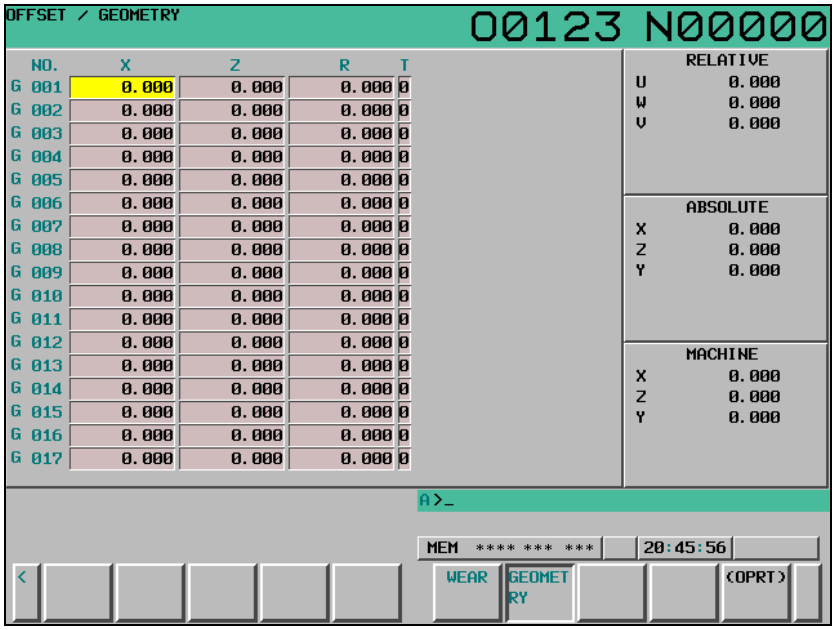


Fig. 2.1.1 (b) With tool geometry offset (10.4-inch)

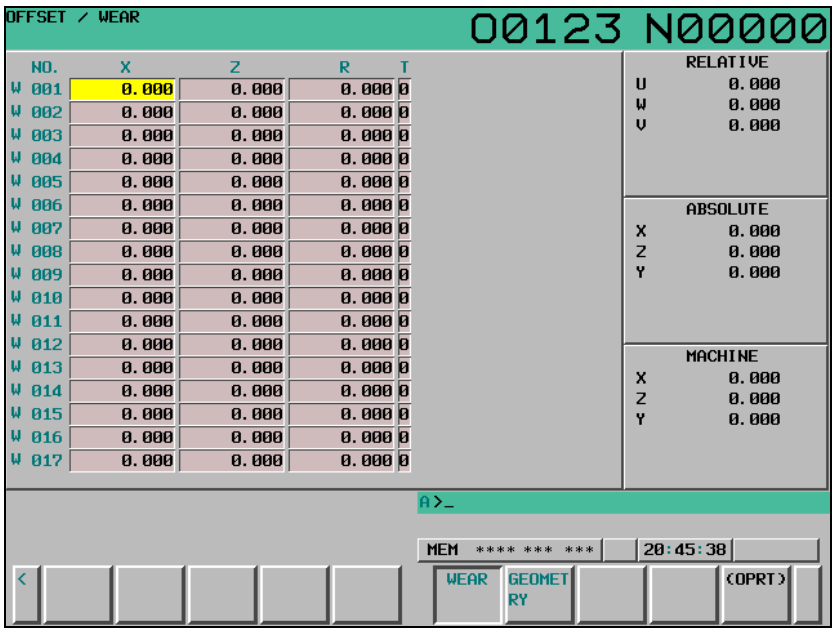


Fig. 2.1.1 (c) With tool wear offset (10.4-inch)

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].

4 To set a compensation value, enter a value and press soft key [INPUT]. To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [+INPUT].
T (TIP) is the number of the imaginary tool nose.
T may be specified on the geometry compensation screen or on the wear compensation screen.

When tool nose radius compensation is not used (bit 7 (NCR) of parameter No. 8136 is set to 1), neither radius nor T (TIP) is displayed.

Explanation

- 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 III-8 “Data Input/Output” in the User’s Manual (Common to Lathe System/Machining Center System).
Tool length compensation values can be set using the following functions described in subsequent subsections: direct input of tool offset value measured, direct input of tool offset value measured B, and counter input of offset value.

- Number of tool compensation values

Up to 64 (1-path system) or 200 (2-path system) tool compensation value sets are available.
When the function for 64 (1-path system) or 200 (2-path system) tool compensation value sets is not used (bit 5 (NDO) of parameter No. 8136 to 1), up to 32 tool compensation value sets are available.

For each set, tool geometry offset can be distinguished from the tool wear offset. (When bit 6 (NGW) of parameter No. 8136 is set to 0)

- Disabling entry of compensation values

In some cases, tool wear compensation or tool geometry compensation values cannot be input because of the settings in bits 0 (WOF) and 1 (GOF) of parameter No.3290. The number of the first tool offset amount of which entry is to be disabled can be set for parameter No. 3294 and the number of tool offset amounts following the start number can be set for parameter No. 3295 to disable entry of tool offset amounts within the specified range from the MDI.
Consecutive input values are set as follows:

1) When values are input for offset numbers, starting from one for which input is not inhibited to one for which input is inhibited, a warning is issued and values are set only for those offset numbers for which input is not inhibited.
2) When values are input for offset numbers, starting from one for which input is inhibited to one for which input is not inhibited, a warning is issued and no values are set.

- Displaying radius and T (TIP)

When tool nose radius compensation is not used according to the setting, neither radius nor T (TIP) is displayed. (Bit 7 (NCR) of parameter No. 8136 is set to 1.)

- Changing offset values during automatic operation

When offset values have been changed during automatic operation, bits 4 (LGT) and 6 (LWM) of parameter No.5002 can be used for specifying whether new offset values become valid in the next move command or in the next T code command.

Fig. 2.1.1 (a)

LGT	LWM	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

2.1.2 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 nose position of a tool actually used as an offset value.

Direct input of tool offset value

Procedure

- 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.

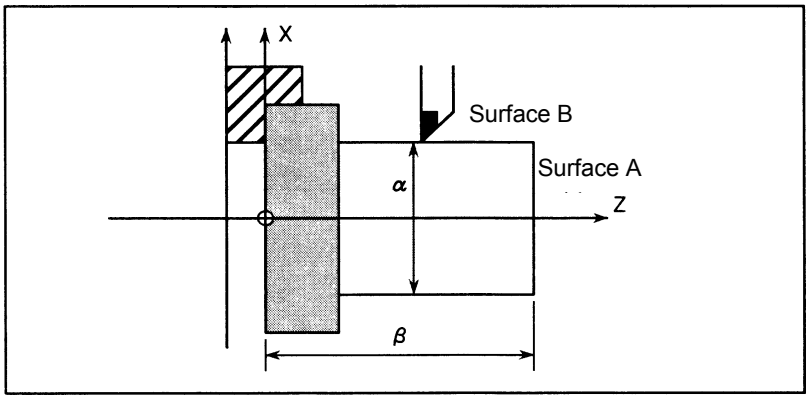


Fig. 2.1.2 (a)

2 Release the tool in X-axis direction only, without moving Z-axis and stop the spindle.

3 Measure distance β from the origin 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:

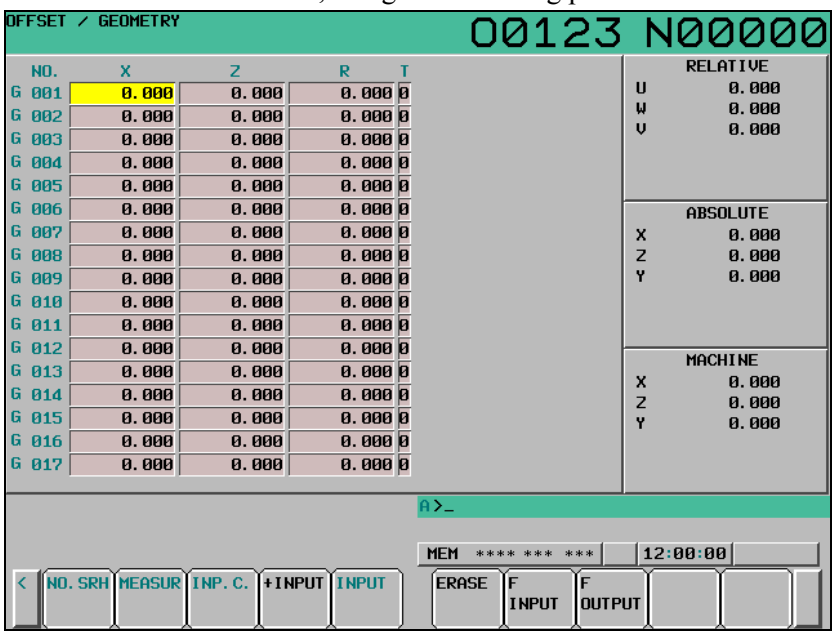


Fig. 2.1.2 (b) Tool offset screen (10.4-inch)

3-1 Press the function key  or the soft key [OFFSET] to display the tool offset screen. If geometry compensation values and wear offset values are separately specified, display the screen for either of them.
3-2 Move the cursor to the set offset number using cursor keys.
3-3 Press the address key  to be set.
3-4 Key in the measured value (β).
3-5 Press the soft key [MESURE].
The difference between measured value β and the 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 α=69.0 when the coordinate value of surface B in the diagram above is 70.0, set 69.0 [MEASURE] at offset No. 2.
In this case, 1.0 is set as the X-axis offset value to offset No. 2.

Explanation

- Offset values for a program created in diameter programming

Enter diameter values for the offset values for axes for which diameter programming is used.

- Tool geometry offset value and tool wear offset value

If measured values are set on the tool geometry offset screen, all offset values become geometry offset values and all wear offset values are set to 0. If measured values are set on the tool wear offset screen, the differences between the measured offset values and the current wear offset values become the new offset values.

- Release of both axes

When the record button is provided on the machine side, the tool can be released in the directions of the two axes by setting bit 2 (PRC) of parameter No. 5005 or using the position record signal. For details on the position record signal, refer to the manual issued by the machine tool builder.

2.1.3 Direct Input of Tool Offset Value Measured B

Explanation

- Basic procedure to set tool offset value

To use the tool setter function for a one–turret/two–spindle lathe, first specify the spindle to be measured, using the S2TLS (G040.5) (spindle measurement select) signal.

(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) Select manual handle mode or manual continuous feed mode and set the tool compensation value write mode select signal GOQSM to "1". The LCD display is automatically changed to the tool offset screen (geometry), and the "OFST" indicator starts blinking in the status indication area at the bottom of the screen, which indicates that the tool compensation value writing mode is ready. When the tool setter function for a one–turret/two–spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes 1.

> ⚠ **CAUTION**
> After this, it is impossible to switch the S2TLS (spindle measurement selection) signal until the GOQSM (offset write mode) signal becomes 0.

(3) Select a tool to be measured.

(4) When the cursor does not coincide with the tool offset number desired to be set, move the cursor to the desired offset number using the page key and cursor key.
The cursor can also be coincided with the tool offset number desired to be set automatically by the tool offset number input signals (when parameter QNI(No.5005#5)=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 tool compensation value writing signals to input to be CNC.
The following tool compensation amount write signals are set up according to the setting of the bit 3 (TS1)of parameter No. 5004.
When the parameter is 0: +MIT1, –MIT1, +MIT2, –MIT2
When the parameter is 1: +MIT1 only
If the tool compensation value writing signal is set to "1":
   i) The axis is interlocked in this direction and its feed is stopped.
   ii) 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 values are set by operations (5) and (6).
(8) Repeat operations (3) to (7) for all necessary tools.
(9) Set the tool compensation value writing mode signal GOQSM to "0".
The writing mode is canceled and the blinking "OFST" indicator light goes off.
When the tool setter function for a one–turret/two–spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal for the spindle being measured becomes 0.

- Basic procedure to set workpiece coordinate shift value

To use the tool setter function for a one–turret/two–spindle lathe, first specify the spindle to be measured, using the S2TLS <G040.5> (spindle measurement select) signal.

(1) Set the tool geometry offset values of each tool in advance.
(2) Execute manual reference position return.
By executing manual reference position return, the machine coordinate system is established.
The workpiece coordinate system shift amount is computed based on the machine coordinate system of the tool.
(3) Set the workpiece coordinate system shift amount writing mode select signal WOQSM to "1".
The LCD display automatically switches to the workpiece shifting screen, the "WFST" indicator starts blinking at the status indicator area in the bottom of the screen, which indicates that the workpiece coordinate system shift amount writing mode is ready.
When the tool setter function for a one–turret/two–spindle lathe is in use, the workpiece coordinate system screen is selected, and the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes 1.

> ⚠ **CAUTION**
> After this, it is impossible to switch the S2TLS (spindle measurement selection) signal until the WOQSM (offset write mode) signal becomes 0.

(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.5020) in advance.
The tool offset number can also be set automatically by setting the tool offset number input signal (with parameter QNI(No.5005#5)=1).
(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 using manual handle feed.
When the tool edge contacts the end face of the workpiece, input the workpiece coordinate system shift amount signal WOSET.
The workpiece coordinate system shift amount on the Z–axis is automatically set.

(8) Release the tool.
(9) Set the workpiece coordinate system shift amount write mode select signal WOQSM to “0”.
The writing mode is canceled and the blinking “WSFT” indicator light goes off.
When the tool setter function for a one–turret/two–spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes 0.

2.1.4 Counter Input of Offset value

By moving the tool until it reaches the desired reference position, the corresponding tool offset value can be set.

Counter input of offset value

Procedure

1 Manually move the reference tool to the reference position.
2 Reset the relative coordinates along the axes to 0.
3 Move the tool for which offset values are to be set to the reference position.
4 Select the tool offset screen. Move the cursor to the offset value to be set using cursor keys.

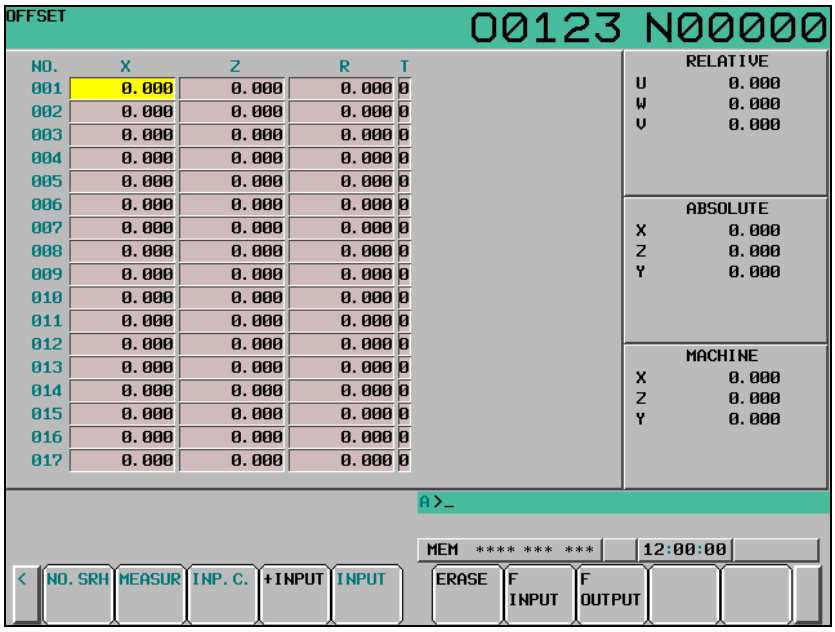


Fig. 2.1.4 (a) Tool offset screen (10.4-inch)

5 Press address key [X] (or [Z]) and the soft key [INP.C.].

Explanation

- Geometry offset and wear offset

When the above operations are performed on the tool geometry offset screen, tool geometry offset values are input and tool wear offset values do not change.
When the above operations are performed on the tool wear offset screen, tool wear offset values are input and tool geometry offset values do not change.

2.1.5 Setting the Workpiece Coordinate System Shift Value

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.
When a T series system is used, the workpiece coordinate system shift screen is displayed.

Setting the workpiece coordinate system shifting amount

Procedure

1 Press function key .

2 Press the continuous menu key  several times until the screen with soft key [W.SHFT] is displayed.

3 Press soft key [W.SHFT].

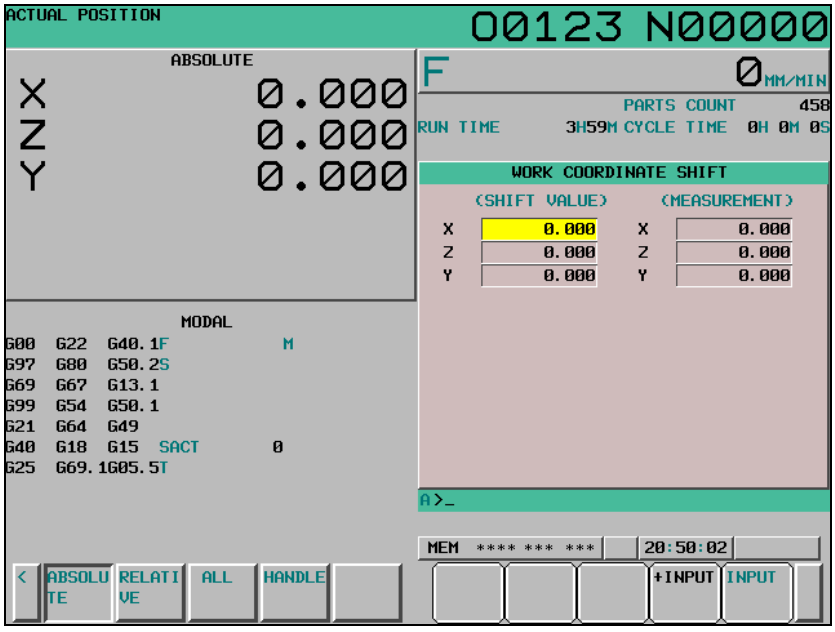


Fig. 2.1.5 (a) Workpiece coordinate system shift screen (10.4-inch)

4 Move the cursor using cursor keys to the axis along which the coordinate system is to be shifted.

5 Enter the shift value and press soft key [INPUT].

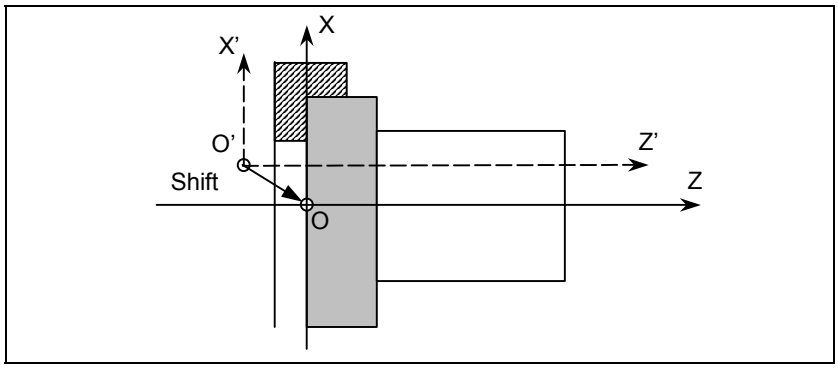


Fig. 2.1.5 (b)

Explanation

- 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.

- Position record signal

When bit 2 (PRC) of parameter No. 5005 is 1, the absolute coordinates when the position record signal is ON are recorded for calculation of the shift amount.

Example

When the actual position of the reference point is X=121.0 (diameter), Z=69.0 with respect to the workpiece origin but it should be X=120.0, Z=70.0, set the following shift values:

Shit value setting: X=1.0, Z=-1.0

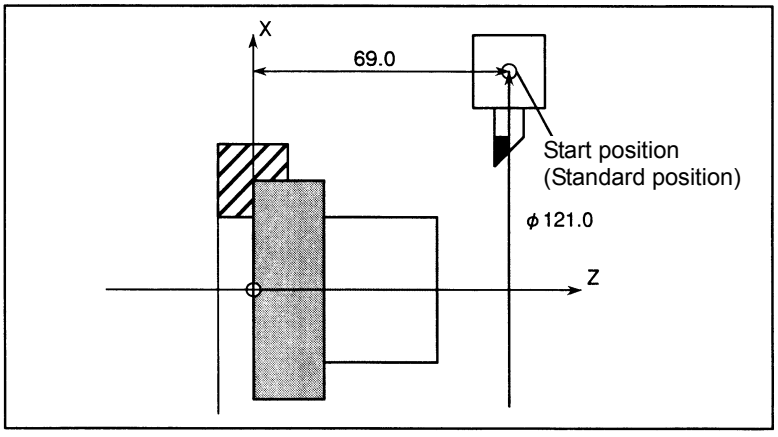


Fig. 2.1.5 (c)

2.1.6 Setting the Y-Axis Offset

Tool position offset values along the Y-axis can be set. Counter input of offset values is also possible.
For the Y-axis, no tool offset value can be directly input.
Whether the Y-axis offset is to be used can be selected using bit 1 (YOF) of parameter No. 8132. (0: Does not use the Y-axis offset./1: Uses the Y-axis offset.)
When the Y-axis is not used according to the setting, the screen is not also displayed.

Procedure for setting the tool offset value of the Y axis

Procedure

1 Press function key .

2 Press the continuous menu key  several times until the screen with soft key [Y OFFSET] is displayed.

3 Press soft key [Y OFFSET]. The Y-axis offset screen is displayed.

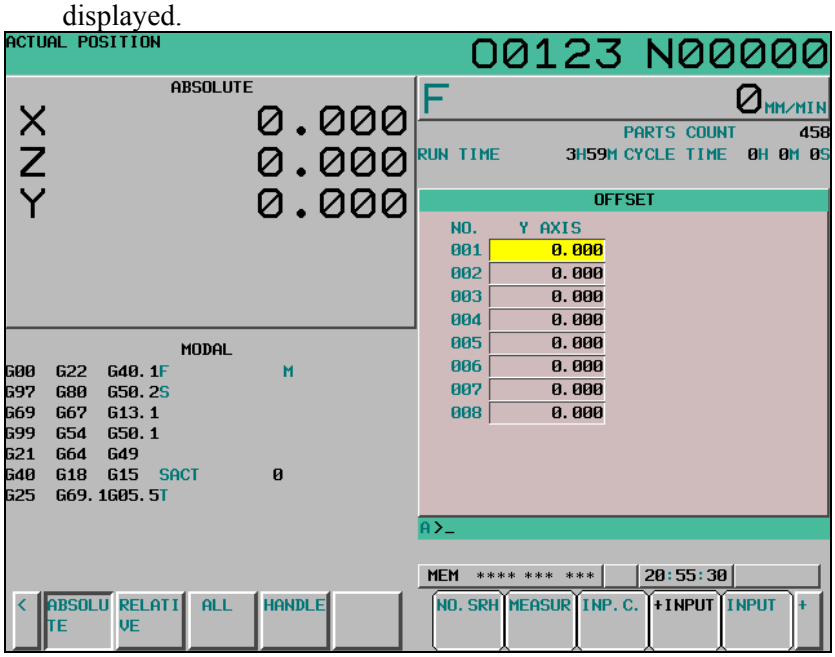


Fig. 2.1.6 (a) Y-axis offset screen (10.4-inch)

3-1 When the [GEOMETRY] soft key is pressed, Y-axis tool geometry compensation data is displayed. Press the [WEAR] soft key to switch the screen display to the display of tool wear compensation data.

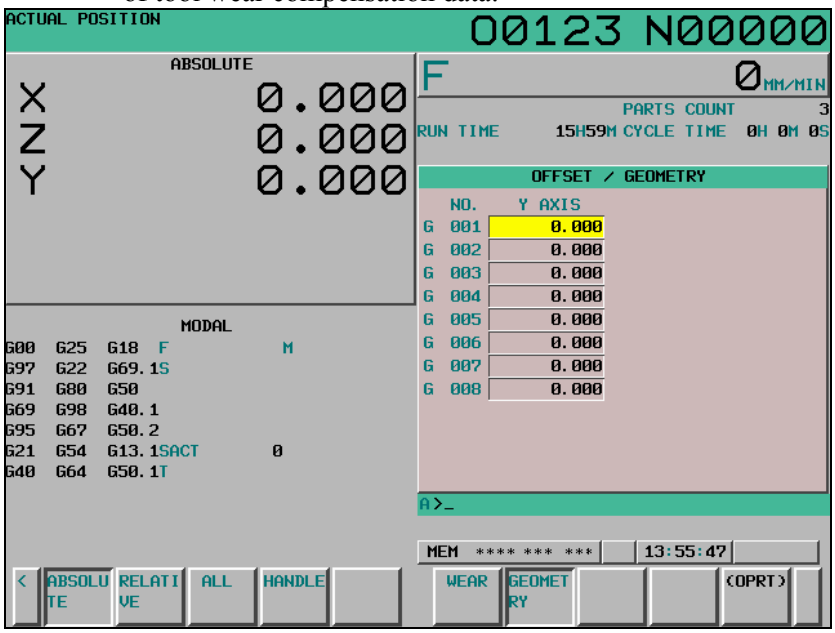


Fig. 2.1.6 (b) Y-axis offset screen (tool geometry) (10.4-inch)

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.
- Type the offset number and press soft key [NO.SRH].

5 Enter the offset value.

6 Press soft key [INPUT]. The offset value is set and displayed.

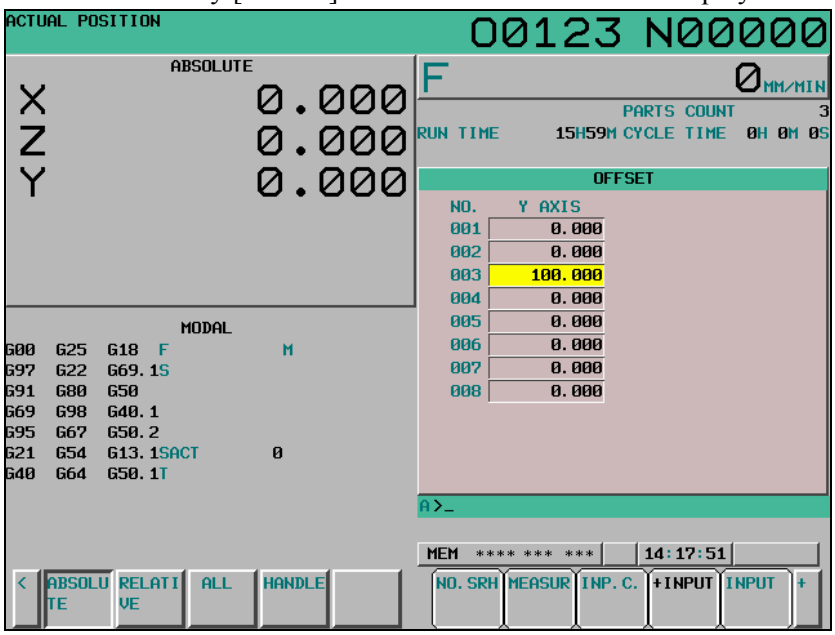


Fig. 2.1.6 (c) Y-axis offset screen (input) (10.4-inch)

Procedure for counter input of the offset value

Procedure

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.
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 [& Y] key, then press soft key [INP.C.].
Relative coordinate Y (or V) is now set as the offset value.

2.1.7 Chuck and Tail Stock Barriers

The chuck and tail stock barrier function prevents damage to the machine by checking whether the tool nose fouls either the chuck or tail stock. Specify an area into which the tool may not enter (entry-inhibition area). This is done using the special setting screen, according to the shapes of the chuck and tail stock. If the tool nose should enter the set area during a machining operation, this function stops the tool and outputs an alarm message. The tool can be cleared from the area only by retracting it in the direction opposite to that in which the tool entered the area.
Whether the chuck and tail stock barrier function is to be used can be selected using bit 1 (BAR) of parameter No. 8134. (0: Does not use the function./1: Uses the function.)
When the function is not used, the screen is not also displayed.

Setting the chuck and tail stock barriers

Procedure

- Setting the shapes of the chuck and tail stock

1 Press function key .

2 Press the continuous menu key . Then, press chapter selection soft key [BARRIER].

3 Pressing the page key  or  toggles the display between the chuck barrier setting screen and tail stock barrier setting screen.

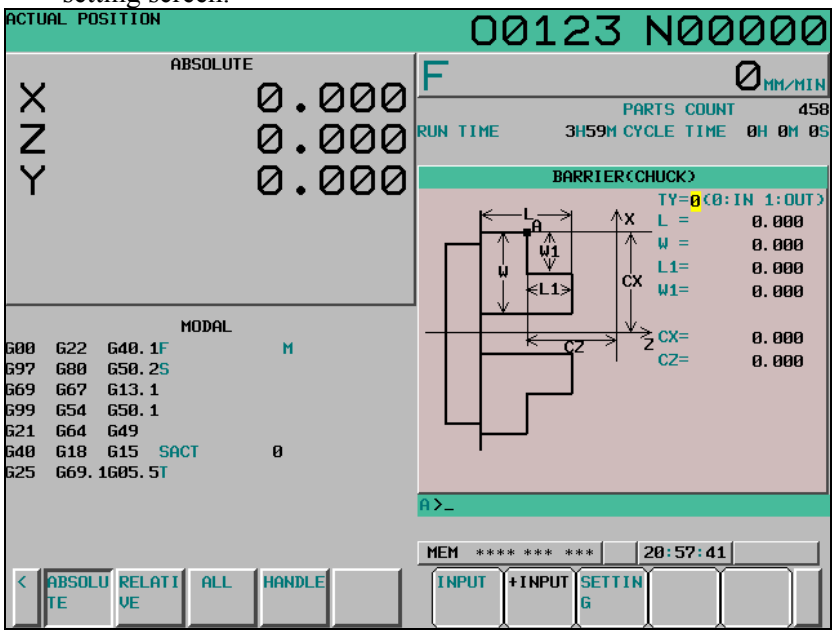


Fig. 2.1.7 (a) Chuck barrier setting screen (10.4-inch)

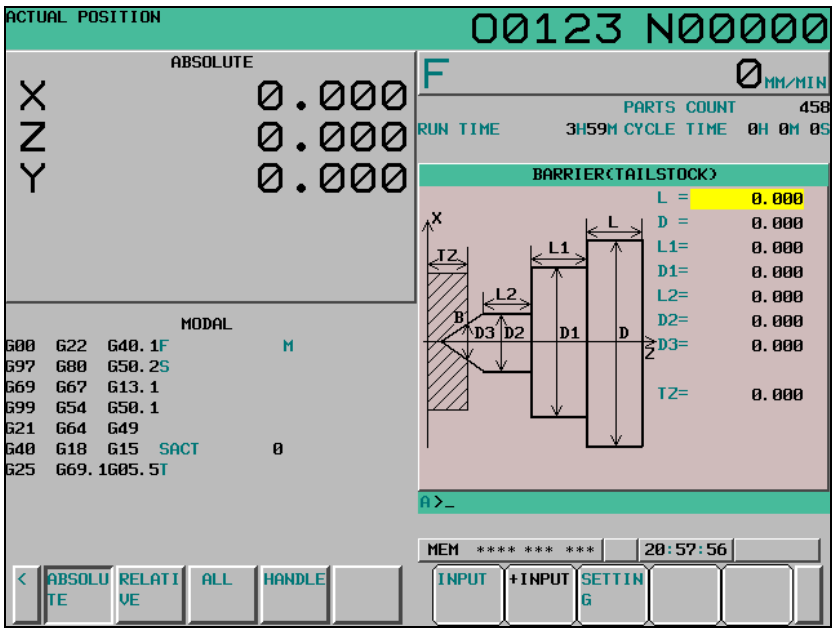


Fig. 2.1.7 (b) Tail stock barrier setting screen (10.4-inch)

4 Position the cursor to each item defining the shape of the chuck or tail stock, enter the corresponding value, then press soft key [INPUT]. The value is set. Pressing soft key [+INPUT] after a value has been entered adds the entered value to the current value, the new setting being the sum of the two values.
Items CX and CZ, both on the chuck barrier setting screen, and item TZ on the tail stock barrier setting screen can also be set in another way. Manually move the tool to the desired position, then press soft key [SETTING] to set the coordinate(s) of the tool in the workpiece coordinate system. If a tool having an offset other than 0 is manually moved to the desired position with no compensation applied, compensate for the tool offset in the set coordinate system. Items other than CX, CZ, and TZ cannot be set by using soft key [SETTING].
Example

When an alarm is issued, the tool stops before the entry-inhibition area if bit 7 (BFA) of parameter No. 1300 is set to 1. If bit 7 (BFA) of parameter No. 1300 is set to 0, the tool stops at a more inside position than the specified figure because the CNC and machine system stop with some delay in time.
For safety, therefore, set an area a little larger than the determined area. The distance between the boundaries of these two areas, L, is calculated from the following equation, based on the rapid traverse rate.

$$L = (\text{Rapid traverse rate}) \times \frac{1}{7500}$$

When the rapid traverse rate is 15 m/min, for example, set an area having a boundary 2 mm outside that of the determined area.
The shapes of the chuck and tail stock can be set using parameters Nos. 1330 to 1348

NOTE
Set G23 mode before attempting to specify the shapes of the chuck and tail stock.

- Reference position return

Return the tool to the reference position along the X- and Z-axes.
The chuck-tail stock barrier function becomes effective only once reference position return has been completed after power on.
When an absolute position detector is provided, reference position return need not always be performed. The positional relationship between the machine and the absolute position detector, however, must be determined.

- G22/G23

When G22 (stored stroke limit on) is specified, the chuck and tail stock area becomes an entry-inhibition area. When G23 (stored stroke limit off) is specified, the entry-inhibition area is canceled.
Even if G22 is specified, the entry-inhibition area for the tail stock can be disabled by issuing a tail stock barrier signal. When the tail stock is pushed up against a workpiece or separated from the workpiece by using the auxiliary functions, PMC signals are used to enable or disable the tail stock setting area.

Table 2.1.7 (a)

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

G22 is selected when the power is turned on. Using G23, bit 7 of parameter No. 3402, however, it can be changed to G23.

Explanation

- Setting the shape of the chuck barrier

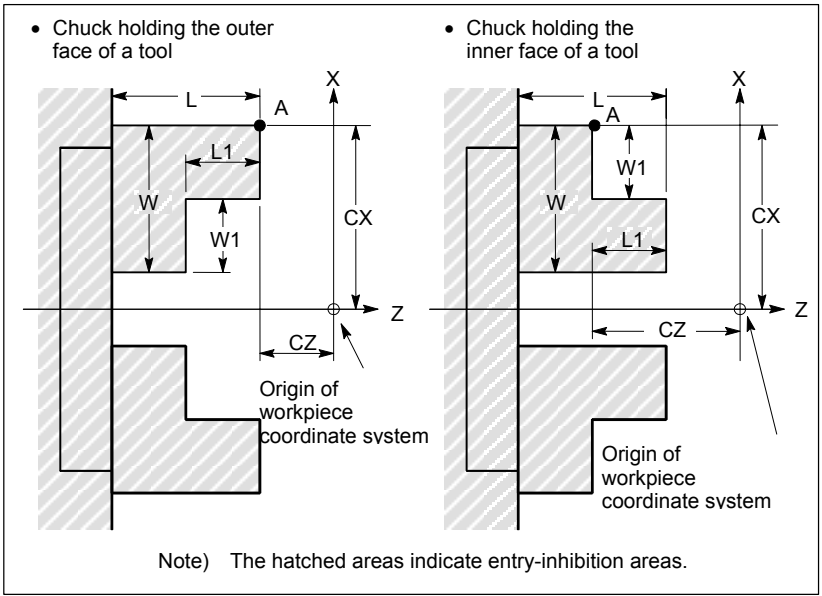


Fig. 2.1.7 (c)

Table2.1.7 (b)

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

TY : Selects a chuck type, based on its shape. Specifying 0 selects a chuck that holds the inner face of a tool. Specifying 1 selects a chuck that holds the outer face of a tool. A chuck is assumed to be symmetrical about its Z-axis.

CX, CZ :

Specify the coordinates of a chuck position, point A, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. The unit of data is indicated in Table 2.1.7(c).

⚠ CAUTION

Whether diameter programming or radius programming is used for the axis determines the programming system. When diameter programming is used for the axis, use diameter programming to enter data for the axis.

Table 2.1.7 (c)

Increment system	Unit of data		Valid data range
	IS-A	IS-B	
Metric input	0.001 mm	0.0001 mm	-99999999 to +99999999
Inch input	0.0001 inch	0.00001 inch	-99999999 to +99999999

L, L1, W, W1 : Define the figure of a chuck. The unit of data is indicated in Table 2.1.7(c).

> ⚠ **CAUTION**
> Always specify W and W1 in radius. When radius programming is used for the Z-axis, specify L and L1 in radius.

- Setting the shape of a tail stock barrier

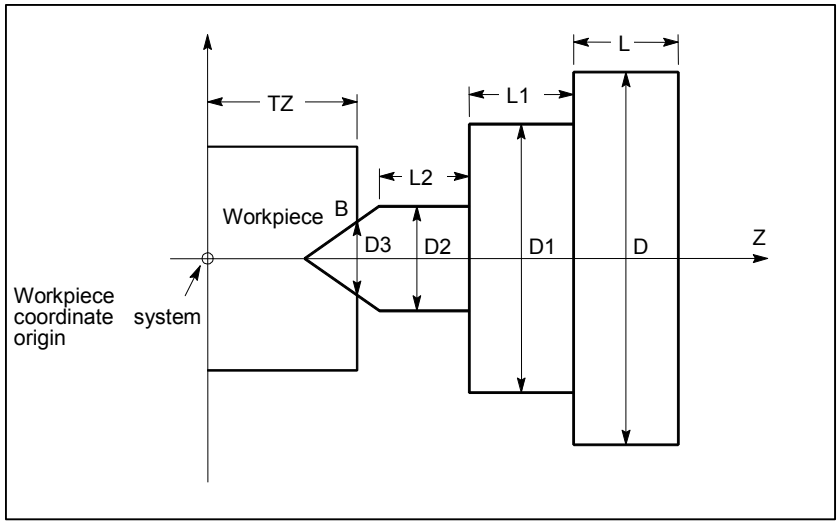


Table 2.1.7 (d)

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

TZ : Specifies the Z coordinate of the chuck position, point B, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. The unit of data is indicated in Table 2.1.7(c). A tail stock is assumed to be symmetrical about its Z-axis.

> **⚠ CAUTION**
> Whether diameter programming or radius programming is used for the Z-axis determines the programming system.

L, L1, L2, D, D1, D2, D3 :
Define the figure of a tail stock. The valid data range is indicated in Table 2.1.7(c).

> **⚠ CAUTION**
> Always specify D, D1, D2, and D3 in diameter programming. When radius programming is used for the Z-axis, specify L, L1, and L2 in radius.

- Setting the entry-inhibition area for the tail stock tip

The tip angle of the tail stock is 60 degrees. The entry-inhibition area is set around the tip, assuming the angle to be 90 degrees, as shown below.

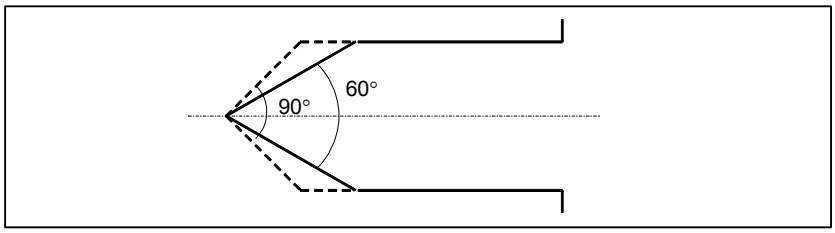


Fig. 2.1.7 (d)

Limitation

- Correct setting of an entry-inhibition area

If an entry-inhibition area is incorrectly set, it may not be possible to make the area effective. Avoid making the following settings:

- L ≤ L1 or W ≤ W1 in the chuck-shape settings.
- D2 ≤ D3 in the tail stock-shape settings.
- A chuck setting overlapping that of the tail stock.

- Retraction from the entry-inhibition area

If the tool enters the entry-inhibition area and an alarm is issued, switch to manual mode, retract the tool manually, then reset the system to release the alarm. In manual mode, the tool can be moved only in the opposite direction to that in which the tool entered the area.

The tool cannot be moved in the same direction (further into the area) as it was traveling when the tool entered the area.

When the entry-inhibition areas for the chuck and tail stock are enabled, and the tool is already positioned within those areas, an alarm is issued when the tool moves.

When the tool cannot be retracted, change the setting of the entry-inhibition areas, such that the tool is outside the areas, reset the system to release the alarm, then retract the tool. Finally, reinstall the original settings.

- Coordinate system

An entry-inhibition area is defined using the workpiece coordinate system. Note the following.

<1> When the workpiece coordinate system is shifted by means of a command or operation, the entry-inhibition area is also shifted by the same amount.

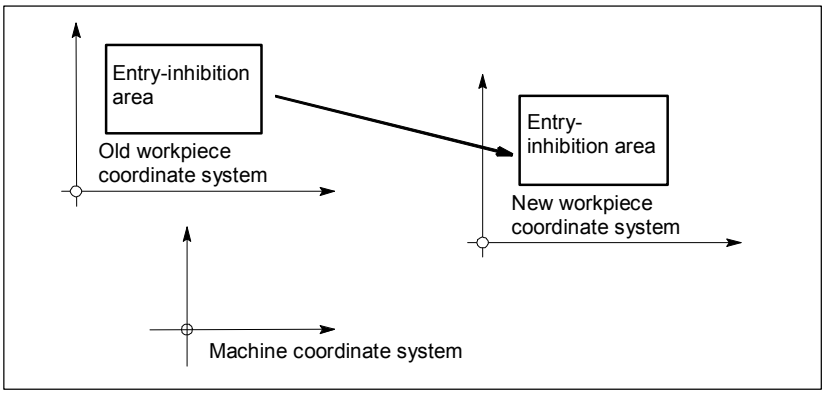


Fig. 2.1.7 (e)

Use of the following commands and operations will shift the workpiece coordinate system.

Commands:

G54 to G59, G52, G50 (G92 in G code system B or C)

Operations:

Manual handle interruption, change in offset relative to the workpiece origin, change in tool offset (tool geometry offset), operation with machine lock, manual operation with machine absolute signal off

<2> When the tool enters an entry-inhibition area during automatic operation, set the manual absolute signal, *ABSM, to 0 (on), then manually retract the tool from the area. If this signal is 1, the distance the tool moves in manual operation is not counted in the tool coordinates in the workpiece coordinate system. This results in the state where the tool can never be retracted from the entry-inhibition area.

- Stored stroke check 2/3

When both stored stroke check 2/3 and the chuck tail stock barrier function are provided, the barrier takes priority over the stored stroke check. Stored stroke check 2/3 is ignored.

APPENDIX

A PARAMETERS

This manual describes all parameters indicated in this manual.
For those parameters that are not indicated in this manual and other parameters, refer to the parameter manual.

Appendix A, "PARAMETERS", consists of the following sections:

A.1 DESCRIPTION OF PARAMETERS 396
A.2 DATA TYPE.....447
A.3 STANDARD PARAMETER SETTING TABLES 448

A.1 DESCRIPTION OF PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
0001							FCV	

[Input type] Setting input
[Data type] Bit path

 # 1 FCV Program format
0: Series 0 standard format
(This format is compliant with the Series 0*i*-C.)
1: Series 10/11 format

NOTE
1 Programs created in the Series 10/11 program format can be used for operation on the following functions:
1 Subprogram call M98,M198
2 Thread cutting with equal leads G32 (T series)
3 Canned cycle G90, G92, G94 (T series)
4 Multiple repetitive canned cycle G71 to G76 (T series)
5 Drilling canned cycle
G80 to G89 (T series)
2 When the program format used in the Series 10/11 is used for this CNC, some limits may add. Refer to the User's Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
1013	IESPx						ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

 # 0 ISA
 # 1 ISC Increment system of each axis

Increment system	#1 ISCx	#0 ISAx
IS-A	0	1
IS-B	0	0
IS-C	1	0

 # 7 IESP When the least input increment is C (IS-C), the function to allow to set the larger value to the parameter of the speed and the acceleration:
0: Not used.
1: Used.

As for the axis which set this parameter when the least input increment is C (IS-C), the larger value can be set to the parameter of the speed and the acceleration.
The valid data ranges of these parameters are indicated in the table of velocity and angular velocity parameters in (C) of the standard parameter setting tables and the table of acceleration and angular acceleration parameters in (D).
When this function is made effective, the digit number below the decimal point of the parameter on input screen is changed. The digit number below the decimal point decreases by one digit in case of the least input increment C (IS-C).

1022 **Setting of each axis in the basic coordinate system**

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, tool radius/tool nose radius compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotation axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1031	Reference axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

The unit of some parameters common to all axes such as those for dry run feedrate and one-digit F code feed may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.

Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

1290	Distance between two opposite tool posts in mirror image

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the distance between two opposite tool posts in mirror image.

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA							

[Input type] Setting input
[Data type] Bit path

 # 7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (T series), or a chuck/tail stock barrier (T series) alarm is issued:

0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

1330	Profile of a chuck

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1

Select a chuck figure.

0 : Chuck which holds a workpiece on the inner surface
1 : Chuck which holds a workpiece on the outer surface

1331	Dimensions of the claw of a chuck (L)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L) of the claw of the chuck.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1332	Dimensions of the claw of a chuck (W)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the width (W) of the claw of the chuck.

NOTE
Specify this parameter by using a radius value at all times.

1333	Dimensions of the claw of a chuck (L1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L1) of the claw of the chuck.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1334 **Dimensions of the claw of a chuck (W1)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the width (W1) of the claw of the chuck.

NOTE
Specify this parameter by using a radius value at all times.

1335 **X coordinate of a chuck (CX)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the chuck position (X coordinate) in the workpiece coordinate system.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1336	Z coordinate of a chuck (CZ)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the chuck position (Z coordinate) in the workpiece coordinate system.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1341	Length of a tail stock (L)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L) of the tail stock.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1342	Diameter of a tail stock (D)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1343	Length of a tail stock (L1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L1) of the tail stock.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1344	Diameter of a tail stock (D1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D1) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1345	Length of a tail stock (L2)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L2) of the tail stock.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1346	Diameter of a tail stock (D2)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D2) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1347	Diameter of the hole of a tail stock (D3)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D3) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1348	Z coordinate of a tail stock (TZ)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the tail stock position (Z coordinate) in the workpiece coordinate system.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0			LRP	

[Input type] Parameter input
[Data type] Bit path

 # 1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.

 # 4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0
1403				ROC				

[Input type] Parameter input
[Data type] Bit path

 # 4 ROC In the threading cycles G92 and G76, rapid traverse override for retraction after threading is finished is:
0: Effective
1: Not effective (Override of 100%)

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

1466	Feedrate for retraction in threading cycle G92, G76 or G76.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
When threading cycle G92, G76 or G76.7 is specified, retraction is performed after threading. Set a feedrate for this retraction.

NOTE
When this parameter is set to 0 or bit 1 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx	JGLx				CTLx

[Input type] Parameter input
[Data type] Bit axis

 # 0 CTLx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

 # 4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)

 # 5 THLx Acceleration/deceleration in threading cycles
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)
As the time constant and FL feedrate, however, the settings of parameter Nos. 1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
[Data type] Bit path

 # 0 CFR For retraction after threading in the threading cycles G92, G76 and G76.7:

0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL feedrate (parameter No. 1627).

1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

NOTE
If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse before look-ahead interpolation is disabled.

1626 **Acceleration/deceleration time constant in threading cycles for each axis**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627	FL rate for acceleration/deceleration in threading cycles for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set an FL feedrate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

3032	Allowable number of digits for the T code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8
Set the allowable numbers of digits for the M, S, and T codes.
When 0 is set, the allowable number of digits is assumed to be 8.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Input type] Parameter input
[Data type] Bit path

 # 0 WOF Setting the tool offset value (tool wear offset) by MDI key input is:
0: Not disabled.
1: Disabled. (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)

NOTE
The tool offset set in the parameter WOF is followed even if geometric compensation and wear compensation are not specified.

 # 1 GOF Setting the tool geometry offset value by MDI key input is:
0: Not disabled.
1: Disabled. (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled
3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to Tool compensation count - 1

When the modification of tool offset values by MDI key input is to be disabled using bit 0 (WOF) of parameter No.3290 and bit 1 (GOF) of parameter No.3290, parameter Nos.3294 and 3295 are used to set the range where such modification is disabled. In parameter No.3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No.3295, set the number of such values. In the following cases, however, none of the tool offset values may be modified:

- When 0 or a negative value is set in parameter No.3294
- When 0 or a negative value is set in parameter No.3295
- When a value greater than the maximum tool offset number is set in parameter No.3294

In the following case, a modification to the values ranging from the value set in parameter No.3294 to the maximum tool offset number is disabled:

When the value of parameter No.3294 added to the value of parameter No.3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI panel, the warning "WRITE PROTECT" is issued.

[Example]

When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:

- Bit 1 (GOF) of parameter No.3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No.3290 = 1 (to disable tool wear offset value modification)
- Parameter No.3294 = 51
- Parameter No.3295 = 10

If the setting of bit 0 (WOF) of parameter No.3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB						DPI

[Input type] Parameter input
[Data type] Bit path

 # 0 DPI When a decimal point is omitted in an address that can include a decimal point
0: The least input increment is assumed. (Normal decimal point input)
1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point input)

 # 6 GSB The G code system is set.
 # 7 GSC

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR			G91			G01

[Input type] Parameter input
[Data type] Bit path

 # 0 G01 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

 # 3 G91 When the power is turned on or when the control is cleared
0: G90 mode (absolute command)
1: G91 mode (incremental command)

 # 6 CLR Reset button on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the User's Manual.

 # 7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
3405			DDP	CCR				

[Input type] Parameter input
[Data type] Bit path

 # 4 CCR Addresses used for chamfering
0: Address is “I”, “J”, or “K”.
In direct drawing dimension programming, addresses ",C", ",R", and ",A" (with comma) are used in stead of "C", "R", and "A".
1: Address is “C”.
Addresses used for direct drawing dimension programming are "C", "R", and "A" without comma.

NOTE
If this bit (CCR) is set to 0, the function for changing the compensation direction by specifying I, J, or K in a G01 block in the tool nose radius compensation mode cannot be used.
If this bit (CCR) is set to 1 when address C is used as an axis name, the chamfer function cannot be used.

 # 5 DDP Angle commands by direct drawing dimension programming
0: Normal specification
1: A supplementary angle is given.

	#7	#6	#5	#4	#3	#2	#1	#0
3453								CRD

[Input type] Setting input
[Data type] Bit path

 # 0 CRD If the functions of chamfering or corner R and direct drawing dimension programming are both enabled,
0: Chamfering or corner R is enabled.
1: Direct drawing dimension programming is enabled.
Specify which function is used when both the chamfering/corner R function and the drawing dimension programming function are enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP	LWM	LGC	LGT		LWT	LGN	

[Input type] Parameter input
[Data type] Bit path

 # 1 LGN Geometry offset number of tool offset
0: Is the same as wear offset number
1: Specifies the geometry offset number by the tool selection number

NOTE
This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

 # 2 LWT Tool wear compensation is performed by:
0: Moving the tool.
1: Shifting the coordinate system.

NOTE
This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

 # 4 LGT Tool geometry compensation
0: Compensated by the shift of the coordinate system
1: Compensated by the tool movement

NOTE
This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

 # 5 LGC When tool geometry compensation is based on coordinate shifting, the tool geometry offset is:
0: Not canceled by a command with offset number 0.
1: Canceled by a command with offset number 0.

NOTE
This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

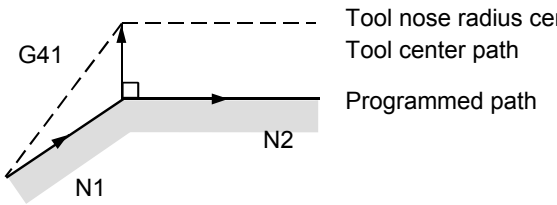
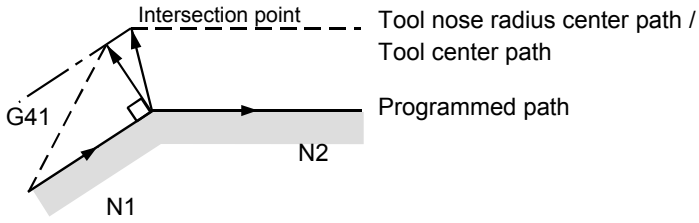
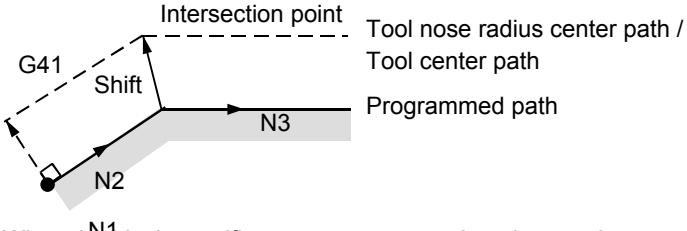
 # 6 LWM Tool offset operation based on tool movement is performed:
0: In a block where a T code is specified.
1: Together with a command for movement along an axis.

 # 7 WNP Imaginary tool tip number used for tool nose radius compensation, when the geometry/wear compensation function is equipped (bit 6 (NGW) of parameter No. 8136 is 0), is the number specified by:
0: Geometry offset number
1: Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5003	TGC						SUV	SUP

[Input type] Parameter input
[Data type] Bit path

 # 0 SUP
 # 1 SUV These bits are used to specify the type of startup/cancellation of cutter compensation or tool nose radius compensation.

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output. 
0	1	Type B	A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output. 
1	0 1	Type C	When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.  When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.

NOTE
When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS0*i*-TC is performed.

 # 7 TGC A tool geometry offset based on a coordinate shift is:
0: Not canceled by reset.
1: Canceled by reset.

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

	#7	#6	#5	#4	#3	#2	#1	#0
5004					TS1		ORC	

[Input type] Parameter input
[Data type] Bit path

 # 1 ORC The setting of a tool offset value is corrected as:
0: Diameter value
1: Radius value

> **NOTE**
> This parameter is valid only for an axis based on diameter specification. For an axis based on radius specification, specify a radius value, regardless of the setting of this parameter.

 # 3 TS1 For touch sensor contact detection with the function for direct input of offset value measured B (T series):
0: Four-contact input is used.
1: One-contact input is used.

	#7	#6	#5	#4	#3	#2	#1	#0
5005			QNI			PRC		

[Input type] Parameter input
[Data type] Bit path

 # 2 PRC For direct input of a tool offset value or workpiece coordinate system shift amount:
0: The PRC signal is not used.
1: The PRC signal is used.

 # 5 QNI With the function for direct input of offset value measured B, a tool compensation number is selected by:
0: Operation through the MDI panel by the operator (selection based on cursor operation).
1: Signal input from the PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
5006					LVC			

[Input type] Parameter input
[Data type] Bit

 # 3 LVC A tool offset (geometry/wear) based on a tool movement and wear offset based on a coordinate shift are:
0: Not canceled by reset.
1: Canceled by reset.

	#7	#6	#5	#4	#3	#2	#1	#0
5008					CNV		CNC	

[Input type] Parameter input
[Data type] Bit path

 # 1 CNC
 # 3 CNV These bits are used to select an interference check method in the cutter compensation or tool nose radius compensation mode.

CNV	CNC	Operation
0	0	Interference check is enabled. The direction and the angle of an arc are checked.
0	1	Interference check is enabled. Only the angle of an arc is checked.
1	-	Interference check is disabled.

For the operation taken when the interference check shows the occurrence of an reference (overcutting) , see the description of bit 5 (CAV) of parameter No. 19607.

NOTE
Checking of only the direction cannot be set.

	#7	#6	#5	#4	#3	#2	#1	#0
5009				TSD				GSC

[Input type] Parameter input
[Data type] Bit path

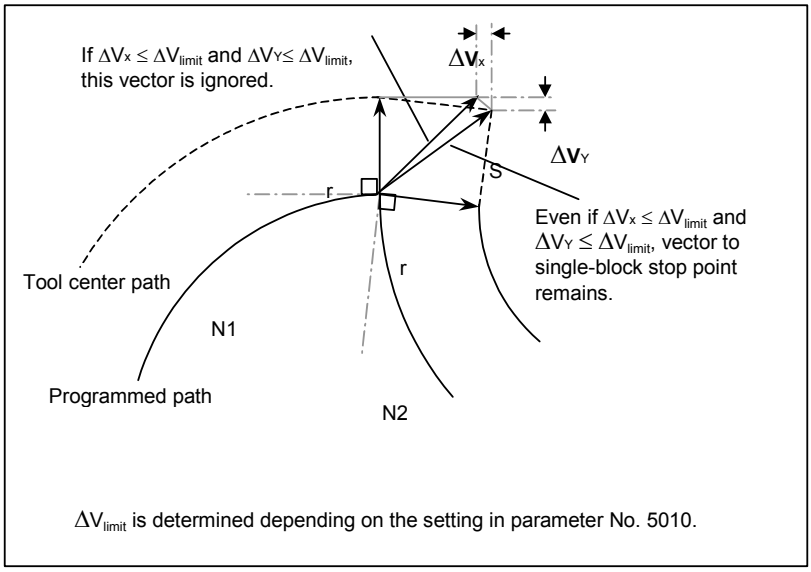
NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

 # 0 GSC When the function for direct input of offset value measured B (T series) is used, an offset write input signal is input from:
0: Machine side
1: PMC side
When the interlock function for each axis direction is enabled (when bit 3 (DIT) of parameter No. 3003 is set to 0), switching can also be made between input from the machine side and input from PMC side for the interlock function for each axis direction.

 # 4 TSD In the function for direct input of offset value measured B (T series), the movement direction determination specifications:
0: Do not apply.
1: Apply.
This parameter is valid when four-contact input is used (bit 3 (TS1) of parameter No. 5004 is set to 0).

5010	Limit for ignoring the small movement resulting from cutter or tool nose radius compensation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the tool moves around a corner in cutter compensation or tool nose radius compensation mode, the limit for ignoring the small travel amount resulting from compensation is set. This limit eliminates the interruption of buffering caused by the small travel amount generated at the corner and any change in feedrate due to the interruption.



ΔV_{limit} is determined depending on the setting in parameter No. 5010.

5020	Tool offset number used with the function for direct input of offset value measured B

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to number of tool compensation values
Set a tool offset number used with the function for direct input of offset value measured B (T series) (when a workpiece coordinate system shift amount is set). (Set the tool offset number corresponding to a tool under measurement beforehand.) This parameter is valid when automatic tool offset number selection is not performed (when bit 5 (QNI) of parameter No. 5005 is set to 0).

5024	Number of tool compensation values

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to number of tool compensation values

Set the maximum allowable number of tool compensation values used for each path.

Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system.

If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.

Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the number of compensation values usable for each path are specified, an alarm is issued.

For example, 100 tool compensation sets are used, 120 sets may be allocated to path 1 and 80 sets to path 2. All of 200 sets need not be used.

5028	Number of digits of an offset number used with a T code command

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Specify the number of digits of a T code portion that is used for a tool offset number (wear offset number when the tool geometry/wear compensation function is used).
When 0 is set, the number of digits is determined by the number of tool compensation values.
When the number of tool compensation values is 1 to 9: Lower 1 digit
When the number of tool compensation values is 10 to 99: Lower 2 digits
When the number of tool compensation values is 100 to 200: Lower 3 digits
Example :

When an offset number is specified using the lower 2 digits of a T code, set 2 in parameter No. 5028.
Txxxxxx yy
xxxxxx : Tool selection
yy : Tool offset number

NOTE
A value longer than the setting of parameter No. 3032 (allowable number of digits of a T code) cannot be set.

5029	Number of tool compensation value memories common to paths

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to number of tool compensation values

When using memories common to paths, set the number of common tool compensation values in this parameter.
Ensure that the setting of this parameter does not exceed the number of tool compensation values set for each path (parameter No. 5024).
[Example 1]
When parameter No. 5029 = 10, parameter No. 5024 (path 1) = 15, and parameter No. 5024 (path 2) = 30 in a 2-path system, tool compensation numbers 1 to 10 of all paths are made common.
[Example 2]
When parameter No. 5029 = 20 and the other conditions are the same as for Example 1, tool compensation numbers 1 to 15 are made common.

NOTE
1 Ensure that the setting of parameter No. 5029 does not exceed the number of tool compensation values for each path (parameter No. 5024). If the setting of parameter No. 5029 exceeds the number of compensation values of a path, the least of the numbers of compensation values in all paths is made common.
2 When 0 or a negative value is set, memories common to paths are not used.

	#7	#6	#5	#4	#3	#2	#1	#0
5040								OWD

[Input type] Parameter input
[Data type] Bit path

 # 0 OWD In radius programming (bit 1 (ORC) of parameter No. 5004 is set to 1),
0: Tool offset values of both geometry compensation and wear compensation are specified by radius.
1: Tool offset value of geometry compensation is specified by radius and tool offset value of wear compensation is specified by diameter, for an axis of diameter programming.

NOTE
This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No. 8136 is 0).

	#7	#6	#5	#4	#3	#2	#1	#0
5042							OFC	OFA

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

 # 0 OFA
 # 1 OFC These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFC	OFA	Unit	Valid data range
0	1	0.01mm	±9999.99mm
0	0	0.001mm	±9999.999mm
1	0	0.0001mm	±9999.9999mm

For inch input

OFC	OFA	Unit	Valid data range
0	1	0.001inch	±999.999inch
0	0	0.0001inch	±999.9999inch
1	0	0.00001inch	±999.99999inch

5043	Axis number for which Y-axis offset is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the number of an axis for which the tool offset is corrected.
If 0 or a value beyond the valid data range is set, the Y-axis offset is applied to the Y-axis of the basic three axes. If setting is made for the X- or Z-axis of the basic three axes, the standard tool offset for the X- or Z-axis is not used, and only the Y-axis offset is used.

	#7	#6	#5	#4	#3	#2	#1	#0
5101						RTR		FXY

[Input type] Parameter input
[Data type] Bit path

 # 0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:
0: In case of the Drilling canned cycle:
Z-axis at all times.
In case of the Grinding canned cycle:
Z-axis at all times.
1: Axis selected by the program

NOTE
1 In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 10/11 format.
2 When this parameter is 1, the drilling axis determined by plane selection (G17/G18/G19) in the drilling canned cycle in the T series 10/11 format. Therefore, the Y-axis is required to specify G17/G19.

 # 2 RTR G83 and G87
0: Specify a high-speed peck drilling cycle
1: Specify a peck drilling cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5102	RDI	RAB			F0C	QSR		

[Input type] Parameter input
[Data type] Bit path

 # 2 QSR Before a multiple repetitive canned cycle (G70 to G73) (T series) is started, a check to see if the program contains a block that has the sequence number specified in address Q is:
0: Not made.
1: Made.
When 1 is set in this parameter and the sequence number specified in address Q is not found, the alarm (PS0063) is issued and the canned cycle is not executed.

 # 3 F0C When the Series 10/11 format is used (with bit 1 (FCV) of parameter No.0001 set to 1), a canned drilling cycle is specified using :
0: Series 10/11 format
1: Series 0 format. However, the number of repetitions is specified using address L.

 # 6 RAB When a canned drilling cycle using the Series 10/11 format is specified (with bit 1 (FCV) of parameter No. 0001 set to 1 and bit 3 (F0C) of parameter No. 5102 set to 0), address R specifies:
0: Increment command.
1: Absolute command with G code system A. With G code system B or C, G90 and G91 are followed.

 # 7 RDI When a canned drilling cycle using the Series 10/11 format is specified (with bit 1 (FCV) of parameter No. 0001 set to 1 and bit 3 (F0C) of parameter No. 5102 set to 0), address R is based on:
0: Radius specification.
1: Diameter/radius specification of the drilling axis.

	#7	#6	#5	#4	#3	#2	#1	#0
5104						FCK		

[Input type] Parameter input
[Data type] Bit path

 # 2 FCK In a multiple repetitive canned cycle (G71/G72) (T series), the machining profile is:
0: Not checked.
1: Checked.
The target figure specified by G71 or G72 is checked for the following before machining operation:

- If the start point of the canned cycle is less than the maximum value of the machining profile even when the plus sign is specified for a finishing allowance, the alarm (PS0322) is issued.
- If the start point of the canned cycle is greater than the minimum value of the machining profile even when the minus sign is specified for a finishing allowance, the alarm (PS0322) is issued.
- If an unmonotonous command of type I is specified for the axis in the cutting direction, the alarm (PS0064 or PS0329) is issued.
- If an unmonotonous command is specified for the axis in the roughing direction, the alarm (PS0064 or PS0329) is issued.
- If the program does not include a block that has a sequence number specified by address Q, the alarm (PS0063) is issued. This check is made, regardless of bit 2 (QSR) of parameter No. 5102.
- If a command (G41/G42) on the blank side in tool nose radius compensation is inadequate, the alarm (PS0328) is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5105					M5T	RF2	RF1	SBC

[Input type] Parameter input
[Data type] Bit path

 # 0 SBC In each of a drilling canned cycle, chamfering/corner rounding cycle, and optional-angle chamfering/corner rounding (T series) cycle:
0: A single block stop is not carried out.
1: A single block stop is carried out.

 # 1 RF1 In a multiple repetitive canned cycle (G71/G72) (T series) of type I, roughing is:
0: Performed.
1: Not performed.

NOTE
When a roughing allowance ($\Delta i/\Delta k$) is specified using the Series 10/11 program format, roughing is performed, regardless of the setting of this parameter.

 # 2 RF2 In a multiple repetitive canned cycle (G71/G72) (T series) of type II, roughing is:
0: Performed.
1: Not performed.

> **NOTE**
> When a roughing allowance (Δi/Δk) is specified using the Series 10/11 program format, roughing is performed, regardless of the setting of this parameter.

 # 3 M5T When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G88):
0: M05 is output before output of M04 or M03.
1: M05 is not output before output of M04 or M03.

> **NOTE**
> 1 This parameter is equivalent to bit 6 (M5T) of parameter No. 5101 of the FS0*i*-C.
> 2 For the T series, the logical level (0/1) is opposite to that of the FS0*i*-C.

	#7	#6	#5	#4	#3	#2	#1	#0
5106								GFX

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 GFX When grinding canned cycle option is specified, the G71, G72, G73, or G74 command is:
0: A multiple repetitive canned cycle (T series) command.
1: A grinding canned cycle command.

5110	M code for C-axis clamping in a drilling canned cycle

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999998
This parameter sets an M code for C-axis clamping in a drilling canned cycle.

5111 **Dwell time when C-axis unclamping is specified in drilling canned cycle**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767
[Unit of data]

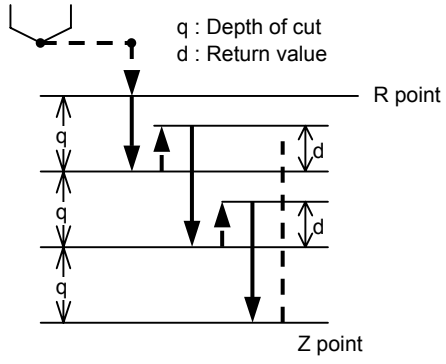
Increment system	IS-A	IS-B	IS-C	Unit
	10	1	0.1	msec

(The increment system does not depend on whether inch input or metric input is used.)
This parameter sets the dwell time when C-axis unclamping is specified in a drilling canned cycle.

5114 **Return value of high-speed peck drilling cycle**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the return value in high-speed peck drilling cycle.

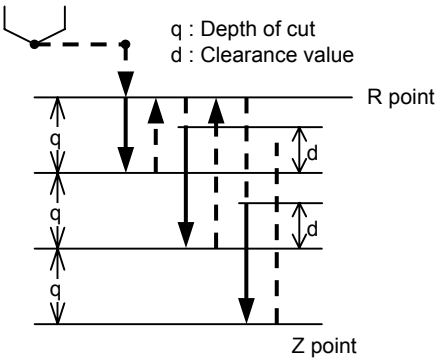
G83 (T series, when the parameter RTR (No.5101#2) is set to 0)



5115 **Clearance value in a peck drilling cycle**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a clearance value in a peck drilling cycle.

G83 (T series, when the parameter RTR (No.5101#2) is set to 1)



5130 **Cutting value (chamfering value) in thread cutting cycles G92 and G76**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] 0.1
[Valid data range] 0 to 127
This parameter sets a cutting value (chamfering value) in the thread cutting cycle (G76) of a multiple repetitive canned cycle (T series) and in the thread cutting cycle (G92) of a canned cycle.
Let L b a lead. Then, a cutting value range from 0.1L to 12.7L is allowed.
To specify a cutting value of 10.0L, for example, specify 100 in this parameter.

5131 **Cutting angle in thread cutting cycles G92 and G76**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] Degree
[Valid data range] 1 to 89
This parameter sets the cutting angle in the thread cutting cycle (G76) of a multiple repetitive canned cycle (T series) and in the thread cutting cycle (G92) of a canned cycle.
When 0 is set, an angle of 45 degrees is specified.

5132 **Depth of cut in multiple repetitive canned cycles G71 and G72**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the depth of cut in multiple repetitive canned cycles G71 and G72 (T series).
This parameter is not used with the Series 10/11 program format.

NOTE
Specify a radius value at all times.

5133 **Escape in multiple repetitive canned cycles G71 and G72**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the escape in multiple repetitive canned cycles G71 and G72 (T series).

NOTE
Specify a radius value at all times.

5134 **Clearance value in multiple repetitive canned cycles G71 and G72**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a clearance value up to the cutting feed start point in multiple repetitive canned cycles (G71/G72) (T series).

NOTE
Specify a radius value at all times.

5135	Retraction distance in the multiple repetitive canned cycle G73 (second axis on the plane)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a retraction distance along the second axis on the plane in the multiple repetitive canned cycle G73 (T series). This parameter is not used with the Series 10/11 program format.

NOTE
Specify a radius value at all times.

5136	Retraction distance in the multiple repetitive canned cycle G73 (first axis on the plane)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a retraction distance along the first axis on the plane in the multiple repetitive canned cycle G73 (T series). This parameter is not used with the Series 10/11 program format.

NOTE
Specify a radius value at all times.

5137	Number of divisions in the multiple repetitive canned cycle G73

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Cycle
[Valid data range] 1 to 99999999
This parameter sets the number of divisions in the multiple repetitive canned cycle G73 (T series).
This parameter is not used with the Series 10/11 program format.

5139 **Return in multiple repetitive canned cycles G74 and G75**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the return in multiple repetitive canned cycles G74 and G75 (T series).

NOTE
Specify a radius value at all times.

5140 **Minimum depth of cut in the multiple repetitive canned cycle G76**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a minimum depth of cut in the multiple repetitive canned cycle G76 (T series) so that the depth of cut does not become too small when the depth of cut is constant.

NOTE
Specify a radius value at all times.

5141 **Finishing allowance in the multiple repetitive canned cycle G76**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the finishing allowance in multiple repetitive canned cycle G76 (T series).

NOTE
Specify a radius value at all times.

5142	Repetition count of final finishing in multiple repetitive canned cycle G76

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Cycle
[Valid data range] 1 to 99999999

This parameter sets the number of final finishing cycle repeats in the multiple repetitive canned cycle G76 (T series).
When 0 is set, only one final finishing cycle is executed.

5143	Tool nose angle in multiple repetitive canned cycle G76

[Input type] Parameter input
[Data type] Byte path
[Unit of data] Degree
[Valid data range] 0, 29, 30, 55, 60, 80

This parameter sets the tool nose angle in multiple repetitive canned cycle G76 (T series).
This parameter is not used with the Series 10/11 program format.

5145	Allowable value 1 in multiple repetitive canned cycles G71 and G72

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

If a monotonous command of type I or II is not specified for the axis in the roughing direction, the alarm (PS0064 or PS0329) is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71 and G72 cycles can be executed even in a program including an unmonotonous figure.

Example)

Suppose that a G71 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the plus direction along the Z-axis is specified in a target figure program, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

NOTE

A check for a monotonous figure is made at all times during G71 and G72 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71or G72 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked.

Note that no alarm is issued when an allowable value is set.

Use a radius value to set this parameter at all times.

5146	Allowable value 2 in multiple repetitive canned cycles G71 and G72

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to cut of depth

If a monotonous command of type I is not specified for the axis in the cutting direction, the alarm (PS0064 or PS0329) is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71 and G72 cycles can be executed even in a program including an unmonotonous figure.
The allowable value is clamped to the depth of cut specified by a multiple repetitive canned cycle.
Example)

Suppose that a G71 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the minus direction along the X-axis is specified in a target figure program for moving from the bottom of cutting to the end point, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

NOTE
A check for a monotonous figure is made at all times during G71 and G72 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71 or G72 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked.
Note that no alarm is issued when an allowable value is set.
Use a radius value to set this parameter at all times.

5176 **Grinding axis number in Traverse Grinding Cycle(G71)**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Traverse Grinding Cycle(G71).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5177 **Grinding axis number of Traverse direct constant-size Grinding cycle(G72)**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Traverse direct constant-size Grinding cycle(G72).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5178 **Grinding axis number of Oscillation Grinding Cycle(G73)**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Oscillation Grinding Cycle(G73).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5179	Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5200		FHD	PCP	DOV				G84

[Input type] Parameter input
[Data type] Bit path

 # 0 G84 Method for specifying rigid tapping:
0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

 # 4 DOV Override during extraction in rigid tapping:
0: Invalidated
1: Validated (The override value is set in parameter No.5211.)

 # 5 PCP Address Q is specified in a tapping cycle/rigid tapping:
0: A high-speed peck tapping cycle is assumed.
1: A peck tapping cycle is assumed.

NOTE
In a tapping cycle, this parameter is valid when bit 6 (PCT) of parameter No. 5104 is 1. When bit 6 (PCT) of parameter No. 5104 is 0, a (high-speed) peck tapping cycle is not assumed.

 # 6 FHD Feed hold and single block in rigid tapping:
0: Invalidated
1: Validated

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU	TDR		

[Input type] Parameter input
[Data type] Bit path

 # 2 TDR Cutting time constant in rigid tapping:
0: Uses a same parameter during cutting and extraction (Parameter Nos. 5261 through 5264)
1: Not use a same parameter during cutting and extraction
Parameter Nos. 5261 to 5264: Time constant during cutting
Parameter Nos. 5271 to 5274: Time constant during extraction

 # 3 OVU The increment unit of the override parameter (No.5211) for tool rigid tapping extraction is:
0: 1%
1: 10%

 # 4 OV3 A spindle speed for extraction is programmed, so override for extraction operation is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5202		OVE						

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

 # 6 OVE The specification range of extraction override command (address J) by rigid tapping program specification is:
0: 100% to 200%.
1: 100% to 2000%.

NOTE
1 To enable the extraction override command (address J) by program specification, set bit 4 (OV3) of parameter No.5201 to 1.
2 When this parameter is set to 1, the operation equivalent to that of the FS0*i*-C is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS		RFF		

[Input type] Parameter input
[Data type] Bit path

 # 2 RFF In rigid tapping, feed forward is:
0: Disabled.
1: Enabled. (Recommended)

As the standard setting, set 1.
At the same time, set the parameter for the advanced preview feed forward coefficient for the tapping axis and the parameter for the advance preview feed forward coefficient for the spindle so that these values match.

- Advanced preview feed forward coefficient for the tapping axis: Parameter No.2092
  (or parameter No.2144 if the cutting/rapid traverse feed forward function is enabled (bit 4 of parameter No.2214 is set to 1))
- Advanced preview feed forward coefficient for the spindle: Parameter No.4344

NOTE
This parameter is valid when a serial spindle is used.

 # 4 OVS In rigid tapping, override by the feedrate override select signal and cancellation of override by the override cancel signal is:
0: Disabled.
1: Enabled.
When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5209								RTX

[Input type] Parameter input
[Data type] Bit path

 # 0 RTX In rigid tapping in a T series, the tapping axis is:
0: Selected by selecting a plane.
1: Always assumed to be the Z-axis for G84 or the X-axis for G88.

NOTE
This parameter becomes invalid when bit 1 (FCV) of parameter No.0001 is set to 1, and rigid tapping is specified using the Series10/11 format.

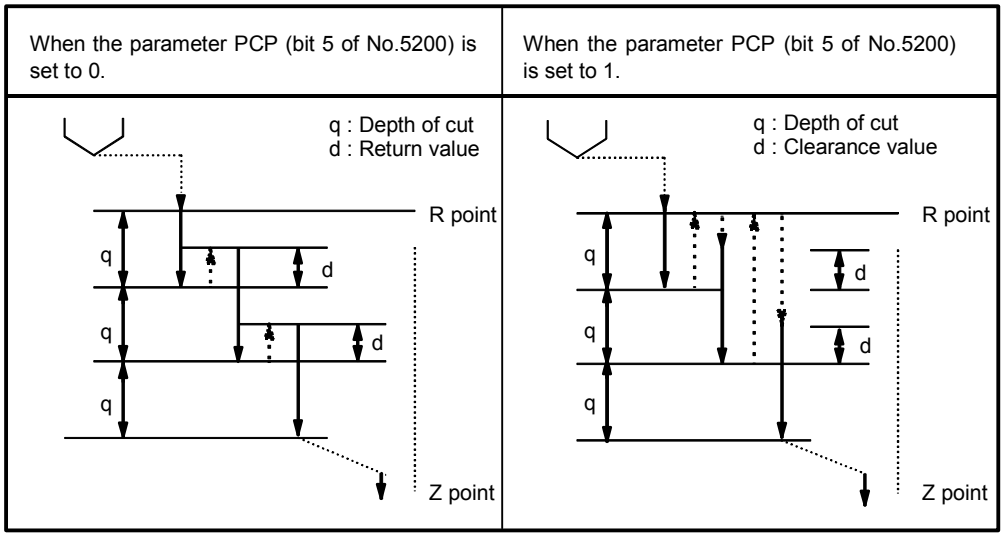
5211	Override value during rigid tapping extraction

[Input type] Parameter input
[Data type] Word path
[Unit of data] 1% or 10%
[Valid data range] 0 to 200
The parameter sets the override value during rigid tapping extraction.

NOTE
The override value is valid when bit 4 (DOV) of parameter No.5200 is set to 1. When bit 3 (OVU) of parameter No.5201 is set to 1, the unit of set data is 10%. An override of up to 200% can be applied to extraction.

5213	Return or clearance in peck tapping cycle

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the drilling axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the escape value of a high-speed peck tapping cycle or the clearance value of a peck tapping cycle.



NOTE
1 In a tapping cycle, this parameter is valid when bit 6 (PCT) of parameter No. 5104 is 1.
2 For the diameter axis, set this parameter using the diameter value.

5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)
5244	Maximum spindle speed in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 9999
Spindle position coder gear ratio
1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999
Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.
Set the same value for both parameter No.5241 and parameter No.5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200 will be issued.

5321	Spindle backlash in rigid tapping (first-stage gear)
5322	Spindle backlash in rigid tapping (second-stage gear)
5323	Spindle backlash in rigid tapping (third-stage gear)
5324	Spindle backlash in rigid tapping (fourth-stage gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] -9999 to 9999
Each of these parameters is used to set a spindle backlash.

	#7	#6	#5	#4	#3	#2	#1	#0
5450						PLS		PDI

[Input type] Parameter input
[Data type] Bit path

 # 0 PDI When the second axis on the plane in the polar coordinate interpolation mode is based on radius specification:
0: Radius specification is used.
1: Diameter specification is used.

 # 2 **PLS** The polar coordinate interpolation shift function is:
0: Not used.
1: Used.
This enables machining using the workpiece coordinate system with a desired point which is not the center of the rotation axis set as the origin of the coordinate system in polar coordinate interpolation.

5460 **Axis (linear axis) specification for polar coordinate interpolation**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of linear axis to execute polar interpolation.

5461 **Axis (rotation axis) specification for polar coordinate interpolation**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of rotation axis to execute polar interpolation.

5463 **Automatic override tolerance ratio for polar coordinate interpolation**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100
Typical setting: 90% (treated as 90% when set to 0)
Set the tolerance ratio of the fastest cutting feedrate to the speed of the rotation axis during automatic override of polar coordinate interpolation.

5464 **Compensation for error on hypothetical axis of polar coordinate interpolation**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(For IS-B, -999999.999 to +999999.999)
This parameter is used to set the error if the center of the rotation axis on which polar coordinate interpolation is performed is not on the X-axis.
If the setting of the parameter is 0, regular polar coordinate interpolation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
6000				HGO			MGO	

[Input type] Parameter input
[Data type] Bit path

 # 1 MGO When a GOTO statement for specifying custom macro control is executed, a high-speed branch to 20 sequence numbers executed from the start of the program is:
0: A high-speed branch is not caused to n sequence numbers from the start of the executed program.
1: A high-speed branch is caused to n sequence numbers from the start of the program.

 # 4 HGO When a GOTO statement in a custom macro control command is executed, a high-speed branch to the 30 sequence numbers immediately before the executed statement is:
0: Not made.
1: Made.

	#7	#6	#5	#4	#3	#2	#1	#0
6240	IGA							AE0

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

 # 0 AE0 Measurement position arrival is assumed when the automatic tool compensation signals XAE1 and XAE2 <X004.0, 1> (T series) or the automatic tool length measurement signals XAE1, XAE2, and XAE3 <X004.0, .1, .2> (M series) are:
0: 1.
1: 0.

 # 7 IGA Automatic tool compensation (T series) is:
0: Used.
1: Not used.

6241	Feedrate during measurement of automatic tool compensation (T series) (for the XAE1 and GAE1 signals)
6242	Feedrate during measurement of automatic tool compensation (T series) (for the XAE2 and GAE2 signals)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
These parameters set the relevant feedrate during measurement of automatic tool compensation (T series) .

NOTE
When the setting of parameter No. 6242 or 6243 is 0, the setting of parameter No. 6241 is used.

6251	γ value on the X axis during automatic tool compensation (T series)
6252	γ value on the Z axis during automatic tool compensation (T series)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the relevant γ value during automatic tool compensation (T series).

NOTE
Set a radius value regardless of whether diameter or radius programming is specified.

6254	ε value on the X axis during automatic tool compensation (T series)
6255	ε value on the Z axis during automatic tool compensation (T series)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the relevant ε value during automatic tool compensation (T series).

NOTE
Set a radius value regardless of whether diameter or radius programming is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
8103								MWT

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

 # 0 MWT As the signal interface for the waiting M code:
0: The path individual signal interface is used.
1: The path common signal interface is used.
This parameter can be selected only when 2-path control is used.

8110	Waiting M code range (minimum value)
8111	Waiting M code range (maximum value)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 ,100 to 99999999
A range of M code values can be set by specifying a minimum waiting M coder value (parameter No. 8110) and a maximum waiting M code value (parameter No. 8111).
(parameter No. 8110) ≤ (waiting M code) ≤ (parameter No. 8111)
Set 0 in these parameters when the waiting M code is not used.

	#7	#6	#5	#4	#3	#2	#1	#0
8132							YOF	

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

 # 1 YOF Y-axis offset is:
0: Not Used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP			SSC

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

 # 0 SSC Constant surface speed control is:
0: Not Used.
1: Used.

 # 3 MSP Multi-spindle is:
0: Not Used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8134						CCR	BAR	

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

 # 1 BAR Chuck and tail stock barrier function (T series) is:
0: Not Used.
1: Used.

NOTE
1 The chuck and tail stock barrier function is provided only for the T series.
2 When the chuck and tail stock barrier function is selected, stored stroke limits 2 and 3 cannot be used.

That is, this parameter also specifies whether to use stored stroke limits 2 and 3 as shown below.

BAR Stored stroke limits 2 and 3 are:
0: Used.
1: Not Used.

 # 2 CCR Chamfering / corner R is:
0: Not Used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8136		NGW						

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

 # 6 NGW Tool geometry/wear compensation (T series) is:
0: Used.
1: Not Used.

	#7	#6	#5	#4	#3	#2	#1	#0
19607		NAA	CAV			CCC		

[Input type] Parameter input
[Data type] Bit path

 # 2 CCC In the cutter compensation/tool nose radius compensation mode, the outer corner connection method is based on:
0: Linear connection type.
1: Circular connection type.

 # 5 CAV When an interference check finds that interference (overcutting) occurred:
0: Machining stops with the alarm (PS0041).
(Interference check alarm function)
1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)
For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

 # 6 NAA When the interference check avoidance function considers that an avoidance operation is dangerous or that a further interference to the interference avoidance vector occurs:
0: An alarm is issued.
When an avoidance operation is considered to be dangerous, the alarm (PS5447) is issued.
When a further interference to the interference avoidance vector is considered to occur, the alarm (PS5448) is issued.
1: No alarm is issued, and the avoidance operation is continued.

> ⚠ **CAUTION**
> When this parameter is set to 1, the path may be shifted largely. Therefore, set this parameter to 0 unless special reasons are present.

19625	Number of blocks to be read in the cutter compensation/tool nose radius compensation mode

[Input type] Setting input
[Data type] Byte path
[Valid data range] 3 to 8

This parameter sets the number of blocks to be read in the cutter compensation/tool nose radius compensation mode. When a value less than 3 is set, the specification of 3 is assumed. When a value greater than 8 is set, the specification of 8 is assumed. As a greater number of blocks are read, an overcutting (interference) forecast can be made with a command farther ahead. However, the number of blocks read and analyzed increases, so that a longer block processing time becomes necessary.

Even if the setting of this parameter is modified in the MDI mode by stopping in the cutter compensation/tool nose radius compensation mode, the setting does not become valid immediately. Before the new setting of this parameter can become valid, the cutter compensation/tool noise radius compensation mode must be canceled, then the mode must be entered again.

A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit	0 or 1	
Bit machine group		
Bit path		
Bit axis		
Bit spindle		
Byte	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Byte machine group		
Byte path		
Byte axis		
Byte spindle		
Word	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
Word machine group		
Word path		
Word axis		
Word spindle		
2-word	0 to ±999999999	Some parameters handle these types of data as unsigned data.
2-word machine group		
2-word path		
2-word axis		
2-word spindle		
Real	See the Standard Parameter Setting Tables.	
Real machine group		
Real path		
Real axis		
Real spindle		

NOTE

1 Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
2 For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
3 For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
4 For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
5 For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
6 The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

NOTE
1 Values are rounded up or down to the nearest multiples of the minimum data unit.
2 A valid data range means data input limits, and may differ from values representing actual performance.
3 For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value."

(A) Length and angle parameters (type 1)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
inch	IS-A	0.001	-99999.999 to +99999.999
	IS-B	0.0001	-99999.9999 to +99999.9999
	IS-C	0.00001	-9999.99999 to +9999.99999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
inch	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-A	0.01	0.0 to +999000.00
	IS-B	0.001	0.0 to +999000.000
	IS-C	0.0001	0.0 to +99999.9999
inch/min	IS-A	0.001	0.0 to +96000.000
	IS-B	0.0001	0.0 to +9600.0000
	IS-C	0.00001	0.0 to +4000.00000

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

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

(D)Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 $deg./sec^2$	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
$inch/sec^2$	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

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

B DIFFERENCES FROM SERIES 0*i*-C

Appendix B, "Differences from Series 0*i*-C", consists of the following sections:

B.1 SETTING UNIT 452
B.2 AUTOMATIC TOOL OFFSET 453
B.3 CIRCULAR INTERPOLATION 455
B.4 HELICAL INTERPOLATION 456
B.5 SKIP FUNCTION 457
B.6 MANUAL REFERENCE POSITION RETURN 459
B.7 WORKPIECE COORDINATE SYSTEM 461
B.8 LOCAL COORDINATE SYSTEM 462
B.9 Cs CONTOUR CONTROL 464
B.10 MULTI-SPINDLE CONTROL 465
B.11 SERIAL/ANALOG SPINDLE CONTROL 466
B.12 CONSTANT SURFACE SPEED CONTROL 467
B.13 SPINDLE POSITIONING 468
B.14 TOOL FUNCTIONS 470
B.15 TOOL COMPENSATION MEMORY 471
B.16 INPUT OF TOOL OFFSET VALUE MEASURED B 473
B.17 CUSTOM MACRO 474
B.18 INTERRUPTION TYPE CUSTOM MACRO 477
B.19 PROGRAMMABLE PARAMETER INPUT (G10) 478
B.20 ADVANCED PREVIEW CONTROL 479
B.21 MACHINING CONDITION SELECTION FUNCTION 481
B.22 AXIS SYNCHRONOUS CONTROL 482
B.23 ARBITRARY ANGULAR AXIS CONTROL 487
B.24 RUN HOUR AND PARTS COUNT DISPLAY 488
B.25 MANUAL HANDLE FEED 489
B.26 PMC AXIS CONTROL 491
B.27 EXTERNAL SUBPROGRAM CALL (M198) 496
B.28 SEQUENCE NUMBER SEARCH 497
B.29 STORED STROKE CHECK 498
B.30 STORED PITCH ERROR COMPENSATION 500
B.31 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION 501
B.32 RESET AND REWIND 502
B.33 MANUAL ABSOLUTE ON AND OFF 503
B.34 MEMORY PROTECTION SIGNAL FOR CNC PARAMETER 504
B.35 EXTERNAL DATA INPUT 505
B.36 DATA SERVER FUNCTION 507
B.37 POWER MATE CNC MANAGER 508
B.38 CHUCK/TAIL STOCK BARRIER 509

B.39 THREADING CYCLE RETRACT (CANNED CUTTING CYCLE/MULTIPLE REPETITIVE CANNED CUTTING CYCLE)....510
B.40 POLAR COORDINATE INTERPOLATION....511
B.41 PATH INTERFERENCE CHECK (2-PATH CONTROL)....513
B.42 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL(2-PATH CONTROL)....514
B.43 SUPERIMPOSED CONTROL (2-PATH CONTROL)....519
B.44 Y AXIS OFFSET....521
B.45 CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION....522
B.46 CANNED CYCLE FOR DRILLING....528
B.47 CANNED CYCLE /MULTIPLE REPETITIVE CANNED CYCLE....530
B.48 CANNED GRINDING CYCLE....531
B.49 MULTIPLE RESPECTIVE CANNED CYCLE FOR TURNING....532
B.50 CHAMFERING AND CORNER ROUNDING....537
B.51 DIRECT DRAWING DIMENSIONS PROGRAMMING....538

B.1 SETTING UNIT

B.1.1 Differences in Specifications

Function	Explanation
Diameter/radius specification in the move command for each axis	- Make a selection using bit 3 (DIAx) of parameter No. 1006. **Bit 3 (DIAx) of parameter No. 1006** The move command for each axis specifies: 0: Radius. 1: Diameter. With Series 0*i*-C, in order for an axis whose diameter is specified to travel the specified distance, it is necessary not only to set 1 in bit 3 (DIAx) of parameter No. 1006 but also to make either of the following two changes: - Reduce the command multiplier (CMR) to half. (The detection unit does not need to be changed.) - Reduce the detection unit to half, and double the flexible feed gear (DMR). With Series 0*i*-D, by contrast, just setting 1 in bit 3 (DIAx) of parameter No. 1006 causes the CNC to reduce the command pulses to half, eliminating the need to make the changes described above (when the detection unit is not changed). Note that, when the detection unit is reduced to half, both the CMR and DMR need to be doubled.

B.1.2 Differences in Diagnosis Display

None.

B.2 AUTOMATIC TOOL OFFSET

B.2.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Operation of the current offset for the measurement result	- Added to the current offset.	- Select whether to add or subtract, by using bit 6 (MDC) of parameter No. 6210. **Bit 6 (MDC) of parameter No. 6210** The measurement result of automatic tool length measurement (system M) or automatic tool compensation (system T) is: 0: Added to the current offset. 1: Subtracted from the current offset.
Setting of the feedrate for measurement	- Set the value in parameter No. 6241. This is a parameter common to the measuring position reached signals (XAE and ZAE).	- **Parameter No. 6241** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6242** This is a parameter for the measuring position reached signals (XAE2 and GAE2). **NOTE** When 0 is set in parameter No. 6242, the value in parameter No. 6241 becomes valid.
Setting of the γ value for the X axis	- Set the value in parameter No. 6251. This is a parameter common to the measuring position reached signals (XAE and ZAE).	- **Parameter No. 6251** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6252** This is a parameter for the measuring position reached signals (XAE2 and GAE2). **NOTE** When 0 is set in parameter No. 6252, the value in parameter No. 6251 becomes valid.
Setting of the ε value for the X axis	- Set the value in parameter No. 6254. This is a parameter common to the measuring position reached signals (XAE and ZAE).	- **Parameter No. 6254** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6255** This is a parameter for the measuring position reached signals (XAE2 and GAE2). **NOTE** When 0 is set in parameter No. 6255, the value in parameter No. 6254 becomes valid.

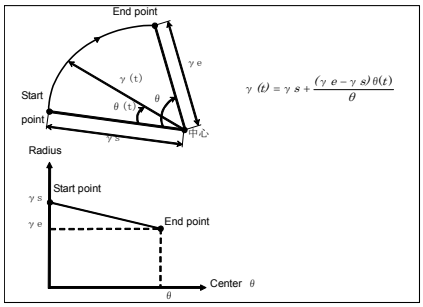
B.2.2 Differences in Diagnosis Display

None.

B.3 CIRCULAR INTERPOLATION

B.3.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Interpolation method when the arc end point is not on the arc</td><td colspan="2">If the difference between the radius values of the start point and end point of an arc is greater than the value set in parameter No. 3410, alarm PS0020 is issued. If the difference is smaller (the end point is not on the arc), circular interpolation is performed as follows.</td></tr>
<tr><td>- Circular interpolation is performed using the radius value of the start point and, when an axis reaches the end point, it is moved linearly.

<u>Parameter No. 3410</u>
In a circular interpolation command, set the limit allowed for the difference between the radius values of the start point and end point.</td><td>- Helical interpolation is performed as shown in the figure below.


In other words, the radius of the arc moves linearly according to the center angle θ(t). Specifying an arc where the arc radius of the start point differs from that of the end point enables helical interpolation. When performing helical interpolation, set a large value in parameter No. 3410 that specifies the limit for the arc radius difference.</td></tr>
</table>

B.3.2 Differences in Diagnosis Display

None.

B.4 HELICAL INTERPOLATION

B.4.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Specification of the feedrate	- Specify the feedrate along a circular arc. Therefore, the feedrate of the linear axis is as follows: $F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$	- Make a selection using bit 5 (HTG) of parameter No. 1403. 0: Same as left. 1: Specify a feedrate along the tool path including the linear axis. Therefore, the tangential velocity of the arc is expressed as follows: $F \times \frac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2+(\text{Length of linear axis})^2}}$ The velocity along the linear axis is expressed as follows: $F \times \frac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2+(\text{Length of linear axis})^2}}$ For details, refer to "HELICAL INTERPOLATION" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).
Helical cutting feedrate clamp	- Make a selection using bit 0 (HFC) of parameter No. 1404. 0: The feedrate of the arc and linear axes is clamped by parameter No. 1422 or No.1430. 1: The combined feedrate along the tool path including the linear axis is clamped by parameter No. 1422.	- Bit 0 (HFC) of parameter No. 1404 is not available. The feedrate of the arc and linear axes is clamped by parameter No. 1430.

B.4.2 Differences in Diagnosis Display

None.

B.5 SKIP FUNCTION

B.5.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Setting to enable the high-speed skip signal for normal skip (G31) when the multi-stage skip function is enabled</td><td>- Set 1 in bit 5 (SLS) of parameter No. 6200.

<table><tr><th rowspan="2">Multi-stage skip function</th><th rowspan="2">Command</th><th colspan="2">Parameter to decide on use of the high-speed skip signal</th></tr><tr><th>FS0i-C</th><th>FS0i-D</th></tr><tr><td>Disabled</td><td>G31 (normal skip)</td><td>HSS</td><td>HSS</td></tr><tr><td rowspan="2">Enabled</td><td>G31 (normal skip)</td><td>SLS</td><td>HSS</td></tr><tr><td>G31P1 to G31P4 (multi-stage skip)</td><td>SLS</td><td>SLS</td></tr></table></td><td>- Set 1 in bit 4 (HSS) of parameter No. 6200.</td></tr>
<tr><td>Target of acceleration/deceleration and servo system delay compensation</td><td>- Compensation is performed for the skip coordinates obtained when the high-speed skip signal is set to "1".</td><td>- Compensation is performed for the skip coordinates obtained when the skip or high-speed skip signal is set to "1".</td></tr>
<tr><td>Method of acceleration/deceleration and servo system delay compensation</td><td>- There are two ways to perform compensation, as follows.
[Compensating the value calculated from the cutting constant and servo constant]
Set 1 in bit 0 (SEA) of parameter No. 6201.
[Compensating the accumulated pulses and positional deviation due to acceleration/deceleration]
Set 1 in bit 1 (SEB) of parameter No. 6201.</td><td>- Bit 0 (SEA) of parameter No. 6201 is not available.
There is only one way to perform compensation, as follows.
[Compensating the accumulated pulses and positional deviation due to acceleration/deceleration]
Set 1 in bit 1 (SEB) of parameter No. 6201.</td></tr>
<tr><td>Skip cutting feedrate (normal skip)</td><td>- Feedrate specified by the F code in the program</td><td>- Depends on bit 1 (SFP) of parameter No. 6207. When 0 is set, the processing is the same as Series 0i-C.

<u>Bit 1 (SFP) of parameter No. 6207</u>
The feedrate during the skip function (G31) is:
0: Feedrate specified by the F code in the program.
1: Feedrate specified in parameter No. 6281.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Skip cutting feedrate (skip using the high-speed skip signal or multi-step skip)	- Feedrate specified by the F code in the program	- Depends on bit 2 (SFN) of parameter No. 6207. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 2 (SFP) of parameter No. 6207** When the skip function using the high-speed skip signal (1 is set in bit 4 (HSS) of parameter No. 6200) or the multi-step skip function is executed, the feedrate is: 0: Feedrate specified by the F code in the program. 1: Feedrate specified in parameter Nos. 6282 to 6285.
Axis to monitor to check whether the torque limit has been reached (torque limit skip)	- Depends on bit 3 (TSA) of parameter No. 6201. **Bit 3 (TSA) of parameter No. 6201** To check whether the torque limit has been reached, the torque limit skip function (G31 P99/98) monitors: 0: All axes. 1: Only the axis specified in the same block as G31 P99/98.	- Bit 3 (TSA) of parameter No. 6201 is not available. Only the axis specified in the same block as G31 P99/98 is monitored.
High-speed skip signal input for the G31 P99 command (torque limit skip)	As the skip signal for the G31 P99 command, the high-speed skip signal:	
	- Cannot be input.	- Can be input.
Setting of a positional deviation limit in the torque limit skip command (torque limit skip)	- No parameter is available dedicated to setting a positional deviation limit for the torque limit skip function.	- The value can be set in parameter No. 6287. **Parameter No. 6287** Set a positional deviation limit in the torque limit skip command for each axis.
When G31 P99/98 is specified without a torque limit being specified in advance (torque limit skip)	- The G31 P99/98 command is executed as is. (No alarm is issued.)	- Alarm PS0035 is issued.

B.5.2 Differences in Diagnosis Display

None.

B.6 MANUAL REFERENCE POSITION RETURN

B.6.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Conditions for performing manual reference position return during feed hold	Manual reference position return is performed when automatic operation is halted (feed hold) and when any of the following conditions is met: <Conditions> (1) Travel distance is remaining. (2) An auxiliary function (M, S, T, or B function) is being executed. (3) A dwell, canned cycle, or other cycle is in progress.	
	- Depends on bit 2 (OZR) of parameter No. 1800. [When OZR = 0] Alarm PS0091 occurs, and manual reference position return is not performed. [When OZR = 1] Manual reference position return is performed without issuing an alarm.	- Bit 2 (OZR) of parameter No. 1800 is not available. Alarm PS0091 occurs, and manual reference position return is not performed.
When inch/metric switch is done	- The reference position is lost. (The reference position is not established.)	- The reference position is not lost. (The reference position remains established.)
Reference position setting without dogs for all axes	- Set 1 in bit 1 (DLZ) of parameter No. 1002.	- Bit 1 (DLZ) of parameter No. 1002 is not available. Reference position setting without dogs (bit 1 (DLZx) of parameter No. 1005) is set for all axes.
Function that performs reference position setting without dogs two or more times when the reference position is not established in absolute position detection	- Not available.	- Depends on bit 4 (GRD) of parameter No. 1007. **Bit 4 (GRD) of parameter No. 1007** For the axis on which absolute values are detected, when correspondence between the machine position and the position by the absolute position detector is not completed, the reference position setting without dogs is: 0: Not performed two or more times. 1: Performed two or more times.

Function	Series 0*i*-C	Series 0*i*-D
Behavior when manual reference position return is started on a rotation axis with the deceleration dog pressed before a reference position is established	- [When bit 0 (RTLx) of parameter No. 1007 = 0] Movement is made at the rapid traverse feedrate until the grid is established. If the deceleration dog is released before the grid is established, one revolution is made at the rapid traverse feedrate, thus establishing the grid. Pressing the deceleration dog again establishes the reference position. [When bit 0 (RTLx) of parameter No. 1007 = 1] Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090.	- [Rotation axis type = A and bit 0 (RTLx) of parameter No. 1007 = 0] Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090. [Rotation axis type = A and bit 0 (RTLx) of parameter No. 1007 = 1] Movement is made at the rapid traverse feedrate until the grid is established. If the deceleration dog is released before the grid is established, one revolution is made at the rapid traverse feedrate, thus establishing the grid. Pressing the deceleration dog again establishes the reference position. [Rotation axis type = B] Does not depend on bit 0 (RTLx) of parameter No. 1007. Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090.
Reference position shift function	- Available only for the M series in Series 0*i*-C and earlier.	- Available for all series in Series 0*i*-D.
Reference position shift function setting	- The function is enabled for all axes by setting 1 in bit 2 (SFD) of parameter No. 1002.	- Bit 2 (SFD) of parameter No. 1002 is not available. Set bit 4 (SFDx) of parameter No. 1008 for each axis.
Setting of whether to preset the coordinate system upon high-speed manual reference position return	- Not available. The coordinate system is not preset.	- Depends on bit 1 (HZP) of parameter No. 1206. **Bit 1 (HZP) of parameter No.1206** Upon high-speed manual reference position return, the coordinate system is: 0: Preset. 1: Not preset (FS0i-C compatible specification).

B.6.2 Differences in Diagnosis Display

None.

B.7 WORKPIECE COORDINATE SYSTEM

B.7.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Change in absolute position display when the workpiece zero point offset value is changed	- Make a selection using bit 5 (AWK) of parameter No. 1201. **Bit 5 (AWK) of parameter No. 1201** When the workpiece zero point offset value is changed: 0: Changes the absolute position display when the program executes the block that is buffered next. 1: Changes the absolute position display immediately. In either case, the changed value does not take effect until the block that is buffered next.	- Bit 5 (AWK) of parameter No. 1201 is not available. The tool always behaves as when AWK is set to 1.

B.7.2 Differences in Diagnosis Display

None.

B.8 LOCAL COORDINATE SYSTEM

B.8.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Clearing of the local coordinate system after servo alarm cancellation	- The processing is determined by the settings of bit 5 (SNC) and bit 3 (RLC) of parameter No. 1202. **Bit 3 (RLC) of parameter No. 1202** Upon reset, the local coordinate system is: 0: Not canceled. 1: Canceled. **Bit 5 (SNC) of parameter No. 1202** After servo alarm cancellation, the local coordinate system is: 0: Cleared. 1: Not cleared. **NOTE** When the RLC bit of the parameter is set to 1, the local coordinate system is cleared, even if the SNC bit of the parameter is set to 1.	- The processing is determined by the settings of bit 7 (WZR) of parameter No. 1201, bit 3 (RLC) of parameter No. 1202, bit 6 (CLR) of parameter No. 3402, and bit 6 (C14) of parameter No. 3407. Bit 5 (SNC) of parameter No. 1202 is not available. **Bit 7 (WZR) of parameter No. 1201** If the CNC is reset by the reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 0, the G code of group number 14 (workpiece coordinate system) is: 0: Placed in the reset state. 1: Not placed in the reset state. **NOTE** When bit 6 (CLR) of parameter No. 3402 is set to 1, the processing depends on the setting of bit 6 (C14) of parameter No. 3407. **Bit 3 (RLC) of parameter No. 1202** Upon reset, the local coordinate system is: 0: Not canceled. 1: Canceled. **NOTE** - When bit 6 (CLR) of parameter No. 3402 is set to 0 and bit 7 (WZR) of parameter No. 1201 is set to 1, the local coordinate system is canceled, regardless of the setting of this parameter. - When bit 6 (CLR) of parameter No. 3402 is set to 1 and bit 6 (C14) of parameter No. 3407 is set to 0, the local coordinate system is canceled, regardless of the setting of this parameter. **Bit 6 (CLR) of parameter No. 3402** The reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal places the local coordinate system in: 0: Reset state. 1: Clear state. **Bit 6 (C14) of parameter No. 3407** If the CNC is reset by the reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 1, the G code of group number 14 (workpiece coordinate system) is: 0: Placed in the clear state. 1: Not placed in the clear state.

B.8.2 Differences in Diagnosis Display

None.

B.9 Cs CONTOUR CONTROL

B.9.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
In-position check when the Cs contour control mode is off	- The in-position check is not made.	- Make a selection using bit 2 (CSNs) of parameter No. 3729. **Bit 2 (CSNs) of parameter No. 3729** When the Cs contour control mode is off, the in-position check is: 0: Made. 1: Not made. When 1 is set in this parameter, the processing is the same as Series 0i-C.

B.9.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Position error display for Cs contour control	For the first spindle, diagnosis display No. 418 is used. For the second spindle, diagnosis display No. 420 is used.	For both the first and second spindles, diagnosis display No. 418 (spindle) is used.

B.10 MULTI-SPINDLE CONTROL

B.10.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Number of gear stages for each spindle	- The first spindle has four stages. Set the maximum spindle speeds for the individual gears in parameter Nos. 3741 to 3744, respectively. - The second spindle has two stages. Set the maximum spindle speeds for the individual gears in parameter No. 3811 and 3812.	- Both the first and second spindles each have four stages. Set the maximum spindle speeds for the individual gears in parameter Nos. 3741 to 3744, respectively. (The data type of parameter Nos. 3741 to 3744 is spindle.)
Spindle override when the override function is used for each axis in multi-spindle control type C	When the override function is used for each axis in multi-spindle control type C, the following spindle override specifications apply during the tapping cycle mode (G84 or G88) or threading mode (G32, G92, or G76).	
	- No function is available to clamp spindle override to 100%. (It does not depend on bit 6 (TSO) of parameter No. 3708.) Modify the ladder code as necessary.	- Depends on bit 6 (TSO) of parameter No. 3708. **Bit 6 (TSO) of parameter No. 3708** During the threading or tapping cycle, spindle override is: 0: Disabled (clamped to 100%). 1: Enabled.

B.10.2 Differences in Diagnosis Display

None.

B.11 SERIAL/ANALOG SPINDLE CONTROL

B.11.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Spindle number of the analog spindle	- When one serial spindle and one analog spindle are simultaneously controlled in one path (serial/analog spindle control), the spindle number of the analog spindle is as follows.	
	Third spindle	Second spindle For details about the parameters and other settings, refer to "SERIAL/ANALOG SPINDLE CNOTROL" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).

B.11.2 Differences in Diagnosis Display

None.

B.12 CONSTANT SURFACE SPEED CONTROL

B.12.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Constant surface speed control with no position coder	- This is an optional function for the T series. It is not available with the M series.	- This is a basic function for both M series and T series. It can be used by enabling constant surface speed control (setting 1 in bit 0 (SSC) of parameter No. 8133) and setting 1 in bit 2 (PCL) of parameter No. 1405.
	- Using bit 0 (PSSCL) of parameter No. 1407, select whether to enable or disable the axis feedrate clamp in feed per revolution when the spindle speed is clamped by the maximum spindle speed set in parameter No. 3772. **Bit 0 (PSSCL) of parameter No. 1407** In constant surface speed control with no position coder, when the spindle speed is clamped by the maximum spindle speed parameter, the axis feedrate in feed per revolution is: 0: Not clamped. 1: Clamped. When 1 is set in this parameter, select the spindle to be used for feed per revolution by using the position coder selection signal. (To use the position coder selection signal requires enabling multi-spindle control.)	- Bit 0 (PSSCL) of parameter No. 1407 is not available. The axis feedrate is always clamped. Using the position coder selection signal, select the spindle to be used for feed per revolution. (To use the position coder selection signal requires enabling multi-spindle control.)

B.12.2 Differences in Diagnosis Display

None.

B.13 SPINDLE POSITIONING

B.13.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Display unit of machine coordinates on the spindle positioning axis	- Pulses	- Make a selection using bit 0 (DMD) of parameter No. 4959. **Bit 0 (DMD) of parameter No. 4959** A machine coordinate on the spindle positioning axis is displayed in: 0: Degrees. 1: Pulses.
Spindle positioning using the second spindle	- Not available.	- Spindle positioning using the second spindle is possible when multi-spindle control is enabled.
Number of M codes for specifying the spindle positioning angle	- Make a selection using bit 6 (ESI) of parameter No. 4950. **Bit 6 (ESI) of parameter No. 4950** Select the specification of spindle positioning. (Bit) 0: Standard specification. 1: Extended specification. When the extended specification is selected, the number of M codes for specifying the spindle positioning angle can be changed from 6 to any number in the range of 1 to 255, depending on the setting of parameter No. 4964.	- Regardless of the setting of bit 6 (ESI) of parameter No. 4950, the setting of parameter No. 4964 takes effect.
Rapid traverse rate unit for spindle positioning	- Selecting the extended specification by setting 1 in bit 6 (ESI) of parameter No. 4950 extends the upper limit of the rapid traverse rate for spindle positioning from 240000 to 269000 (unit: 10 degrees/min).	- Make a selection using bit 6 (ESI) of parameter No. 4950. **Bit 6 (ESI) of parameter No. 4950** Select the rapid traverse rate unit for spindle positioning (bit spindle). 0: Not increased by a factor of 10. (Unit: degrees/min) 1: Increased by a factor of 10. (Unit: 10 degrees/min)
Rapid traverse rate for spindle orientation in the case of an analog spindle	- The feedrate set in parameter No. 1420 takes effect.	- The feedrate set in parameter No. 1428 takes effect. When 0 is set in parameter No. 1428, the value set in parameter No. 1420 takes effect.

B.13.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Diagnosis data indicating the spindle positioning sequence status (spindle)	- None.	- Diagnosis No.1544
Diagnosis data indicating the clamp/unclamp sequence status (servo)	- None.	- Diagnosis No.5207

B.14 TOOL FUNCTIONS

B.14.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Specification of a G code of the 00 group other than G50 (T series) and a T code in the same block</td><td>- Not allowed.</td><td>- Not allowed.
Specifying a G code in this way causes alarm PS0245.</td></tr>
<tr><td>Number of digits of an offset number in a T code command</td><td>- Set the value in bit 0 (LD1) of parameter No. 5002.</td><td>- Bit 0 (LD1) of parameter No. 5002 is not available.
Use parameter No. 5028.</td></tr>
<tr><td rowspan="2">Method of wear compensation</td><td colspan="2">- When 1 is set in bit 2 (LWT) and bit 4 (LGT) of parameter No. 5002, the method of wear compensation is as follows.</td></tr>
<tr><td>Compensation with tool movement</td><td>Compensation with coordinate shift</td></tr>
<tr><td>Offset cancellation by reset</td><td colspan="2">- Select the cancellation operation using bit 3 (LVC) of parameter No. 5006 and bit 7 (TGC) of parameter No. 5003.
<table>
<tr><th colspan="2" rowspan="2">Compensation method</th><th colspan="4">Parameter</th></tr>
<tr><th>LVC="0"
TGC="0"</th><th>LVC="1"
TGC="0"</th><th>LVC="0"
TGC="1"</th><th>LVC="1"
TGC="1"</th></tr>
<tr><td rowspan="2">Tool movement</td><td>Wear compensation
Geometry compensation</td><td>×</td><td>○
(When axis is moved)</td><td>×</td><td>○
(When axis is moved)</td></tr>
<tr><td rowspan="2">Coordinate shift</td><td>Wear compensation</td><td>×</td><td>○</td><td>×</td><td>○</td></tr>
<tr><td>Geometry compensation</td><td>×</td><td>×</td><td>*</td><td>○</td></tr>
</table>
○: Canceled ×: Not canceled

The operation marked by "*" differs between Series 0i-C and Series 0i-D.
Series 0i-C: × (Not canceled)
Series 0i-D: ○ (Canceled)</td></tr>
</table>

B.14.2 Differences in Diagnosis Display

None.

B.15 TOOL COMPENSATION MEMORY

B.15.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Unit and range of tool compensation values</td><td>- The unit and range of tool compensation values are determined by the setting unit.</td><td>- Set the unit and range using bit 0 (OFA) and bit 1 (OFC) of parameter No. 5042.

<u>Bit 0 (OFA) and bit 1 (OFC) of parameter No. 5042</u>
Select the setting unit and range of tool offset values.

Metric input
<table>
<tr><th>OFC</th><th>OFA</th><th>Unit</th><th>Range</th></tr>
<tr><td>0</td><td>1</td><td>0.01mm</td><td>±9999.99mm</td></tr>
<tr><td>0</td><td>0</td><td>0.001mm</td><td>±9999.999mm</td></tr>
<tr><td>1</td><td>0</td><td>0.0001mm</td><td>±9999.9999mm</td></tr>
</table>
Inch input
<table>
<tr><th>OFC</th><th>OFA</th><th>Unit</th><th>Range</th></tr>
<tr><td>0</td><td>1</td><td>0.001inch</td><td>±999.999inch</td></tr>
<tr><td>0</td><td>0</td><td>0.0001inch</td><td>±999.9999inch</td></tr>
<tr><td>1</td><td>0</td><td>0.00001inch</td><td>±999.99999inch</td></tr>
</table></td></tr>
<tr><td>Automatic conversion of tool compensation values upon inch/metric switch</td><td>- Make a selection using bit 0 (OIM) of parameter No. 5006.

<u>Bit 0 (OIM) of parameter No. 5006</u>
Upon inch/metric switch, automatic conversion of tool compensation values is:
0: Not performed.
1: Performed.
If the setting of this parameter is changed, set the tool compensation data again.</td><td>- Bit 0 (OIM) of parameter No. 5006 is not available.
Tool compensation values are always converted automatically.</td></tr>
</table>

Function	Series 0*i*-TTC	Series 0*i*-D
Number of tool compensation values for each axis during 2-path control	- Up to 64 tool compensation values can be used per path.	- Up to 128 tool compensation values can be used per system. Using parameter No. 5024 whose data type is path, set the number of tool compensation values to be assigned to each path. **NOTE** It is possible to increase to 200 tool compensation values by the option.
Tool compensation memory sharing during 2-path control	- Set this item using bit 5 (COF) of parameter No. 8100. All tool compensation memories can be shared by the paths. Note that it is not allowed to share only part of the memories. **Bit 5 (COF) of parameter No. 8100** Paths 1 and 2: 0: Do not share tool compensation memories. 1: Share tool compensation memories.	- Set this item using parameter No. 5029. The number of tool compensation memories to be shared can be set arbitrarily.

B.15.2 Differences in Diagnosis Display

None.

B.16 INPUT OF TOOL OFFSET VALUE MEASURED B

B.16.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Setting of the X and Z axes	- It is necessary to set the X axis as the first axis and the Z axis as the second axis.	- It is necessary to set the X axis as the X axis of the basic three axes (set 1 in parameter No. 1022) and the Z axis as the Z axis of the basic three axes (set 3 in parameter No. 1022).
Relationship with arbitrary angular axis control	- By setting 1 in bit 3 (QSA) of parameter No. 5009, the function can be used together with arbitrary angular axis control.	- Cannot be used together with arbitrary angular axis control. The correct value cannot be set for an angular axis under arbitrary angular axis control.
Relationship with composite control	- By setting bit 0 (MXC), bit 1 (XSI), and bit 2 (ZSI) of parameter No. 8160 as appropriate for the machine configuration, the function can be used together with composite control.	- Cannot be used together with composite control. The correct value cannot be set for a composite axis under composite control.

B.16.2 Differences in Diagnosis Display

None.

B.17 CUSTOM MACRO

B.17.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Keep-type common variable (#500 to #999)	- The default value is <null>.	- The default value is 0.
	- The Series 0i-D function (described at right) is not available.	- The range specified by parameter Nos. 6031 and 6032 can be made write-protected (read-only).
System variable to read machine coordinates #5021 to #5025	- Machine coordinates are always read in machine units (output units).	- Machine coordinates are always read in input units. Example) When the setting unit is IS-B, the input unit is the inch, the machine unit is the millimeter, and the coordinate value of the X axis (first axis) is as follows: Machine coordinate = 30.000 (mm) Since the value of #5021 is read in input units (inches), #5021 is 1.1811.
Logical operations in an if statement	- Logical operations can be used by setting 1 in bit 0 (MLG) of parameter No. 6006. **Bit 0 (MLG) of parameter No. 6006** In an if statement in a custom macro, logical operations: 0: Cannot be used. (P/S alarm No. 114 is issued.) 1: Can be used.	- Bit 0 (MLG) of parameter No. 6006 is not available. Logical operations can always be used.
Behavior of the GOTO statement when a sequence number is not found at the start of the block	- The command after the sequence number of the block (to the right of the sequence number) is executed.	- If a move command is specified before the sequence number (left side), alarm PS0128 is issued. If no move command is specified before the sequence number (left side), a block containing a sequence number is executed from the beginning.
	* Use a sequence number at the start of a block.	
Behavior of "GOTO 0" when there is a sequence number	- The program jumps to the block containing the sequence number.	- No jump occurs. Alarm PS1128 is issued.
	* Do not use a sequence number.	
When another NC command is found in a G65 block or in an M code block where a macro is called by an M code Example) G01 X100. G65 P9001 ;	- In a program like the one shown in the example, G01 changes the G code group to 01, while the move command X100. is not executed. X100. is regarded as an argument of G65.	- A program like the one shown in the example cannot be executed. Alarm PS0127 is issued. A G65 code or an M code that calls a macro must be specified at the beginning of a block (before all other arguments).

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Behavior when subprogram call using an M code and subprogram call using an T code are done</td><td colspan="2">- When the machine is run under the conditions and program described below:
[Conditions]
・ Subprogram call by T code is enabled (bit 5 (TCS) of parameter No. 6001 is set to 1).
・ The M code that calls subprogram No. 9001 is M06 (parameter No. 6071 is set to 6).
[Program]
O0001;
T100; (1)
M06 T200; (2)
T300 M06; (3)
M30;
%</td></tr>
<tr><td>In FS0i-C, blocks (1) to (3) of the program causes the machine to behave as follows:
1) Calls and executes O9000.
2) Outputs T200 and waits for FIN. Upon receipt of the FIN signal, the machine calls and executes O9001.
3) Outputs T300 and waits for FIN. Upon receipt of the FIN signal, the machine calls and executes O9001.</td><td>In FS0i-D, blocks (1) to (3) of the program causes the machine to behave as follows:
1) Calls and executes O9000.
2) Issues alarm PS1091.
3) Issues alarm PS1091 (when the program is run with block (2) deleted).</td></tr>
<tr><td rowspan="2">Block containing "M98 Pxxxx" or "M99" without any addresses other than O, N, P, and L</td><td>- Using bit 4 (NPS) of parameter No. 3450, it is possible to select whether the block is treated as an NC statement or a macro statement.

<u>Bit 4 (NPS) of parameter No. 3450</u>
0: Treated as a single-block NC statement without movement. (Single block stop is performed.)
1: Treated as a macro statement. (Single block stop is not performed.)</td><td>- Bit 4 (NPS) of parameter No. 3450 is not available. The block is always treated as a macro statement. (Single block stop is not performed.)</td></tr>
<tr><td colspan="2">* For details about macro and NC statements, refer to Section 16.4, "MACRO AND NC STATEMENTS", in "USER'S MANUAL" (B-64304EN).</td></tr>
<tr><td>Subprogram and macro calls</td><td colspan="2">- The call nesting level differs as follows.
<table>
<tr><th rowspan="2">Model
Call method</th><th colspan="2">Series 0i-C</th><th colspan="2">Series 0i-D</th></tr>
<tr><th>Independent nesting level</th><th>Total</th><th>Independent nesting level</th><th>Total</th></tr>
<tr><td>Macro call (G65/G66)</td><td>4 in all</td><td rowspan="2">(G65/G66/M98)
8 in all</td><td>5 in all</td><td rowspan="2">(G65/G66/M98)
15 in all</td></tr>
<tr><td>Subprogram call (M98)</td><td>4</td><td>10</td></tr>
</table></td></tr>
<tr><td>Local variable clear operation by reset</td><td>- Make a selection using bit 7 (CLV) of parameter No. 6001.

<u>Bit 7 (CLV) of parameter No. 6001</u>
When reset, the local variables in the custom macro are:
0: Cleared to <null>.
1: Not cleared.</td><td>- Bit 7 (CLV) of parameter No. 6001 is not available.
Local variables are always cleared to <null> when reset.</td></tr>
</table>

B.17.2 Differences in Diagnosis Display

None.

B.17.3 Miscellaneous

Series 0*i*-D allows you to customize the specifications related to the maximum and minimum variable values and accuracy by using bit 0 (F0C) of parameter No. 6008. When 1 is set in bit 0 (F0C) of parameter No. 6008, the specifications are the same as Series 0*i*-C. For details, refer to Section 16, "CUSTOM MACRO", in "USER'S MANUAL" (B-64304EN).

B.18 INTERRUPTION TYPE CUSTOM MACRO

B.18.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Interruption type custom macro in DNC operation	- Not available.	- Available.
Program restart	- When an interruption type custom macro is executed during return operation in dry run after search operation invoked by program restart:	
	The interruption type custom macro is executed after all axes have restarted.	Alarm DS0024 is issued.

B.18.2 Differences in Diagnosis Display

None.

B.19 PROGRAMMABLE PARAMETER INPUT (G10)

B.19.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Parameter input mode setting	- Specify G10 L50.	- Specify G10 L52.

B.19.2 Differences in Diagnosis Display

None.

B.20 ADVANCED PREVIEW CONTROL

B.20.1 Differences in Specifications

Differences common to advanced preview control, AI advanced preview control, and AI contour control

Function	Series 0*i*-C	Series 0*i*-D
Function name	Some function names have been changed as follows.	
	- Automatic corner deceleration	- Speed control based on the feedrate difference on each axis
	- Arc radius-based feedrate clamp	- Speed control with acceleration in circular interpolation
Setting to enable bell-shaped acceleration/deceleration in rapid traverse	- Setting 1 in bit 6 (RBL) of parameter No. 1603 enables bell-shaped acceleration/deceleration in rapid traverse.	- Bit 6 (RBL) of parameter No. 1603 is not available. Bell-shaped acceleration/deceleration in rapid traverse is enabled by setting the time constant of bell-shaped acceleration/deceleration after interpolation in rapid traverse in parameter No. 1621 or the acceleration change time of bell-shaped acceleration/deceleration before interpolation in rapid traverse in parameter No. 1672.
Selection of acceleration/deceleration before interpolation in rapid traverse or acceleration/deceleration after interpolation in rapid traverse	- The combination of bit 1 (AIR) of parameter No. 7054 and bit 1 (LRP) of parameter No. 1401 determines acceleration/deceleration before interpolation or acceleration/deceleration after interpolation.	- Bit 1 (AIR) of parameter No. 7054 is not available. The combination of bit 5 (FRP) of parameter No. 19501 and bit 1 (LRP) of parameter No. 1401 determines acceleration/deceleration before interpolation or acceleration/deceleration after interpolation. For details, refer to "PARAMETER MANUAL" (B-64310EN).
Setting of acceleration for look-ahead linear acceleration/deceleration before interpolation	- Set acceleration by specifying the maximum cutting feedrate for linear acceleration/deceleration before interpolation in parameter No. 1770 and the time to elapse before reaching the maximum cutting feedrate for linear acceleration/deceleration before interpolation in parameter No. 1771.	- Parameter Nos. 1770 and 1771 are not available. In parameter No. 1660, set the maximum permissible cutting feedrate for acceleration/deceleration before interpolation for each axis.

Function	Series 0*i*-C	Series 0*i*-D
Time constant setting of linear/bell-shaped acceleration/deceleration after interpolation in cutting feed common to all axes	- Set the value in parameter No. 1768.	- Parameter No. 1768 is not available. Set the time constant for each axis in parameter No. 1769.
Time constant setting of exponential acceleration/deceleration after interpolation in cutting feed for each axis	- Set the value in parameter No. 1762. (To set the value for linear or bell-shaped acceleration/deceleration, use parameter No. 1769.)	- Parameter No. 1762 is not available. Set the value in parameter No. 1769. (Use parameter No. 1769 for any acceleration/deceleration type - linear, bell-shaped, or exponential.)
Automatic corner deceleration based on angle difference	- Setting 0 in bit 4 (CSD) of parameter No. 1602 enables the function. Set the lower limit speed in parameter No. 1777 and the critical angle between the two blocks in parameter No. 1779.	- Automatic corner deceleration based on angle difference is not available. Therefore, bit 4 (CSD) of parameter No. 1602 and parameter Nos. 1777 and 1779 are not available.
Permissible speed difference common to all axes for automatic corner deceleration based on angle difference (speed control based on the feedrate difference on each axis)	- Set the value in parameter No. 1780.	- Parameter No. 1780 is not available. Set the permissible speed difference for each axis in parameter No. 1783.
Setting of arc radius-based feedrate clamp (speed control with acceleration in circular interpolation)	- Set the upper limit of the feedrate and the corresponding arc radius value in parameter Nos. 1730 and 1731, respectively.	- Parameter Nos. 1730 and 1731 are not available. Set the permissible acceleration for each axis in parameter No. 1735.
Setting of the maximum cutting feedrate common to all axes	- Set the value in parameter No. 1431.	- Parameter No. 1431 is not available. Set the maximum cutting feedrate for each axis in parameter No. 1432.
Rapid traverse block overlap	- Disabled in the advanced preview control .	- Enabled only when acceleration/deceleration after interpolation is used in the advanced preview control.

B.20.2 Differences in Diagnosis Display

None.

B.21 MACHINING CONDITION SELECTION FUNCTION

B.21.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Parameters set by "acceleration/deceleration before interpolation" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1770] Maximum cutting feedrate in linear acceleration/deceleration before interpolation [Parameter No. 1771] Time before the maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1770) is reached	- The following parameters are set according to the precision level: [Parameter No. 1660] Maximum permissible cutting feedrate in acceleration/deceleration before interpolation on each axis (Series 0*i*-D does not have parameter Nos. 1770 and 1771.)
Parameter 1 set by "permissible acceleration" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1730] Upper limit of the feedrate by arc radius-based feedrate clamp [Parameter No. 1731] Arc radius corresponding to the upper limit of the feedrate by arc radius-based feedrate clamp (parameter No. 1730)	- The following parameters are set according to the precision level: [Parameter No. 1735] Permissible acceleration in speed control with acceleration in circular interpolation (Series 0*i*-D does not have parameter Nos. 1730 and 1731. Also, "arc radius-based feedrate clamp" has been renamed "speed control with acceleration in circular interpolation".)

B.21.2 Differences in Diagnosis Display

None.

B.22 AXIS SYNCHRONOUS CONTROL

B.22.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Function name	- Quick synchronous control	- Axis synchronous control
Setting to perform synchronous operation all the time	- Not available.	- Depends on bit 5 (SCA) of parameter No. 8304 for the slave axis. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 5 (SCA) of parameter No. 8304** In axis synchronous control: 0: Synchronous operation is performed when the axis synchronous control selection signal SYNCx or axis synchronous control manual feed selection signal SYNCJx for the slave axis is set to "1". 1: Synchronous operation is performed all the time. Synchronous operation is performed regardless of the setting of the SYNCx or SYNCJx signal.
Setting to move multiple slave axes in synchronism with the master axis	- Not available.	- Available. This is possible by setting the same master axis number in parameter No. 8311 for the multiple slave axes.
Setting of the same name for the master and slave axes	- The same name cannot be set for the master and slave axes.	- The same name can be set for the master and slave axes. In that case, however, automatic operation cannot be performed in normal operation; only manual operation is allowed. (No alarm is caused even if an attempt to perform automatic operation is made.)
Setting of axes for which to perform simple synchronous control (axis synchronous control)	T - The setting method of parameter No. 8311 is different from that used for the M series. See Series 0*i*-C Connection Manual (Function) for details. M - The master axis number set in parameter No. 8311 must be smaller than the slave axis number.	- The master axis number set in parameter No. 8311 may or may not be smaller than the slave axis number. - The setting method of parameter No. 8311 for the M series of Series 0*i*-C is always used.

Function	Series 0*i*-C	Series 0*i*-D
Synchronization error check based on positional difference	- Not available.	- The servo positional difference between the master and slave axes is monitored, and alarm DS0001 is issued if the difference exceeds the limit value set in parameter No. 8323 for the slave axis. At the same time, the signal for indicating a positional difference error alarm for axis synchronous control SYNER<F403.0> is output. Parameter No. 8313 is not available. Regardless of the number of pairs, set the limit value in parameter No. 8323. - The data range of parameter No. 8323 is as follows: [Data range] 0 to 999999999
Synchronization error check based on machine coordinates	- Not available.	- The machine coordinates of the master and slave axes are compared and, if the difference is greater than the value set in parameter No. 8314 for the slave axis, alarm SV0005 is issued and the motor is stopped immediately. - The data range of parameter No. 8314 is as follows: [Data range] 0 or positive 9 digits of the minimum unit of data. (For IS-B, 0.0 to +999999.999)
Setting of synchronization establishment	- Synchronization establishment is not available.	- Synchronization establishment is enabled by setting 1 in bit 7 (SOF) of parameter No. 8303 for the slave axis. (Bit 7 (SOF) of parameter No. 8301 is not available. Regardless of the number of pairs, set 1 in bit 7 (SOF) of parameter No. 8303.)
Timing of synchronization establishment	- Synchronization establishment is not available.	- Synchronization establishment is performed when: 1. Power is turned on when the absolute position detector is used. 2. Manual reference position return operation is performed. 3. The state of servo position control is changed from off to on. (This occurs when emergency stop, servo alarm, servo off, etc. is canceled. Note, however, that synchronization establishment is not performed at the time of axis removal cancellation.)

Function	Series 0*i*-C	Series 0*i*-D
Maximum compensation for synchronization	- Synchronization establishment is not available.	- Set the value in parameter No. 8325 for the slave axis. If the compensation amount exceeds the values set in this parameter, alarm SV0001 occurs. (Parameter No. 8315 is not available. Regardless of the number of pairs, set the value in parameter No. 8325.) - The data unit and data range of parameter No. 8325 are as follows: [Data unit] Machine unit [Data range] 0 or positive 9 digits of the minimum unit of data. (For IS-B, 0.0 to +999999.999)
Automatic setting for grid position matching	- Automatic setting for grid position matching is not available.	- Set 1 in bit 0 (ATE) of parameter No. 8303 for the slave axis to enable automatic setting for grid position matching. (Bit 0 (ATE) of parameter No. 8302 is not available. Regardless of the number of pairs, set the value in bit 0 (ATE) of parameter No. 8303.) - Set 1 in bit 1 (ATS) of parameter No. 8303 for the slave axis to start automatic setting for grid position matching. (Bit 1 (ATS) of parameter No. 8302 is not available. Regardless of the number of pairs, set the value in bit 1 (ATS) of parameter No. 8303.)
Difference between the master axis reference counter and slave axis reference counter obtained through automatic setting for grid positioning	- Automatic setting for grid position matching is not available.	- Set the value in parameter No. 8326 for the slave axis. (Parameter No. 8316 is not available. Regardless of the number of pairs, set the value in parameter No. 8326.)
Time from the servo preparation completion signal SA <F000.6> being set to 1 until torque difference alarm detection is started	- Torque difference alarm detection is not available.	- Set the value in parameter No. 8327 for the slave axis. (Parameter No. 8317 is not available. Regardless of the number of pairs, set the value in parameter No. 8327.)

Function	Series 0*i*-C	Series 0*i*-D
Setting to use the external machine coordinate system shift function for the slave axis	- Not available.	- Bit 3 (SSE) of parameter No. 8302 is not available. By setting 1 in bit 7 (SYE) of parameter No. 8304 for the slave axis, the slave axis is shifted as well when an external machine coordinate system shift is set for the corresponding master axis. This parameter is used individually for each slave axis.
Setting to prevent slave axis movement from being added to the actual feedrate display	- Not available. Slave axis movement is always added to the actual feedrate display.	- Bit 7 (SMF) of parameter No. 3105 is not available. Setting 0 in bit 2 (SAF) of parameter No. 8303 prevents slave axis movement from being added to the actual feedrate display. (Note that the meaning of the value is the opposite from bit 7 (SMF) of parameter No. 3105.) This parameter is used individually for each slave axis.
Change of the synchronization state during a program command	- Specify an M code that is not to be buffered. Using this M code, change the input signal - SYNCx<G138> or SYNCJx<G140> - from the PMC side.	- Specify an M code that changes the synchronization state (parameter No. 8337 or 8338). By changing the input signal - SYNCx<G138> or SYNCJx<G140> - from the PMC side using this M code, it is possible to change the synchronization state during a program command. **Parameter No. 8337** Specify an M code that changes synchronous operation to normal operation. **Parameter No. 8338** Specify an M code that changes normal operation to synchronous operation.
Automatic slave axis parameter setting	- This function is enabled by setting 1 in bit 4 (TRP) of parameter No. 12762 for the master axis.	- Bit 4 (TRP) of parameter No. 12762 is not available. This function is enabled by setting 1 in bit 4 (SYP) of parameter No. 8303 for the master and slave axes.

T

Function	Series 0*i*-C	Series 0*i*-D
Number of pairs for synchronous operation	- One pair (two pairs for the M series)	- Two pairs (also two pairs for the M series)

Function	Series 0*i*-C	Series 0*i*-D
Synchronous operation during manual operation	- Synchronous operation is not available in jog, handle, or manual incremental feed.	- Setting axis synchronous control manual feed selection signal SYNCJx to 1 enables synchronous operation even in jog, handle, or manual incremental feed.

B.22.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Positional difference between the master and slave axes	- This item is displayed in diagnosis No. 540 for the master axis when the number of synchronized axis pairs is one or in diagnosis No. 541 for the master axis when the number of synchronized axis pairs is two.	- This item is displayed in diagnosis No. 3500 for the slave axis. (Regardless of the number of pairs, the item is displayed in diagnosis No. 3500.)

B.23 ARBITRARY ANGULAR AXIS CONTROL

B.23.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Angular and perpendicular axes when an invalid value is set in parameter No. 8211 or 8212</td><td colspan="2">
<table>
<tr><th></th><th colspan="2">Series 0i-C</th><th colspan="2">Series 0i-D</th></tr>
<tr><th></th><th>Angular axis</th><th>Perpendicular axis</th><th>Angular axis</th><th>Perpendicular axis</th></tr>
<tr><td>T series</td><td>X axis (1st axis)</td><td>Z axis (2nd axis)</td><td>X-axis of the basic three axes (axis with 1 set in parameter No. 1022)</td><td>Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)</td></tr>
</table>
</td></tr>
<tr><td>Reference position return completion signal ZP for the perpendicular axis moved with the angular axis <Fn094, Fn096, Fn098, Fn100></td><td>- Select the signal using bit 3 (AZP) of parameter No. 8200.
When the bit is set to 0, ZP is not set to "0". (The signal is not cleared.)
When the bit is set to 1, ZP is set to "0". (The signal is cleared.)</td><td>- Bit 3 (AZP) of parameter No. 8200 is not available.
ZP is always set to "0". (The signal is cleared.)</td></tr>
<tr><td>When an angular axis is specified individually in machine coordinate system selection (G53) during arbitrary angular axis control</td><td>- Select the perpendicular axis operation using bit 6 (A53) of parameter No. 8201.
When the bit is set to 0, the perpendicular axis is also moved.
When the bit is set to 1, only the angular axis is moved.</td><td>- Bit 6 (A53) of parameter No. 8201 is not available.
Only the angular axis is always moved.</td></tr>
<tr><td>G30 command during arbitrary angular axis control</td><td>- Select the operation using bit 0 (A30) of parameter No. 8202.
When the bit is set to 0, the operation is for the perpendicular coordinate system.
When the bit is set to 1, the operation is for the angular coordinate system.</td><td>- Bit 0 (A30) of parameter No. 8202 is not available.
The operation is always for the angular coordinate system.</td></tr>
</table>

B.23.2 Differences in Diagnosis Display

None.

B.24 RUN HOUR AND PARTS COUNT DISPLAY

B.24.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Data range of the M code that counts the number of machined parts	**Parameter No. 6710** The data range of the M code that counts the number of machined parts is as follows.	
	- 0 to 255	- 0 to 99999999 (8 digits)
Data range of the number of parts required	**Parameter No. 6713** The data range of the number of parts required is as follows.	
	- 0 to 9999	- 0 to 999999999 (9 digits)
Data range of the number and total number of parts machined	**Parameter No. 6711** Number of parts machined The data range is as follows.	**Parameter No. 6712** Total number of parts machined
	- 0 to 99999999 (8 digits)	- 0 to 999999999 (9 digits)
Data range of the power-on period, time during automatic operation, cutting time, input signal TMRON on time, and one automatic operation time	**Parameter No. 6750** Integrated value of power-on period **Parameter No. 6752** Integrated value of time during automatic operation **Parameter No. 6754** Integrated value of cutting time **Parameter No. 6756** Integrated value of time when input signal TMRON (G053.0) is on **Parameter No. 6758** Integrated value of one automatic operation time The data range is as follows.	
	- 0 to 99999999 (8 digits)	- 0 to 999999999 (9 digits)

B.24.2 Differences in Diagnosis Display

None.

B.25 MANUAL HANDLE FEED

B.25.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Handle pulses exceeding the rapid traverse rate	If manual handle feed exceeding the rapid traverse rate is specified, whether to ignore or accumulate handle pulses exceeding the rapid traverse feedrate can be set as follows.	
	- Depends on bit 4 (HPF) of parameter No. 7100. The amount of pulses to be accumulated is set in parameter No. 7117.	- Bit 4 (HPF) of parameter No. 7100 is not available. Whether to ignore or accumulate excess handle pulses is determined by the amount to be accumulated that is set in parameter No. 7117. [When parameter No. 7117 = 0] Ignored. [When parameter No. 7117 > 0] Accumulated in the CNC without being ignored.
Permissible amount of pulses for manual handle feed	- The value range of parameter No. 7117 is 0 to 99999999 (8 digits).	- The value range of parameter No. 7117 is 0 to 999999999 (9 digits).
Number of manual pulse generators used	- Set the value in parameter No. 7110.	- Parameter No. 7110 is not available. Up to two generators can be used without setting the parameter.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="4">Value range of the magnification parameter for manual handle feed</td><td>- For parameter Nos. 7113, 7131, and 12350, magnification ranges from 1 to 127.
For parameter Nos. 7114, 7132, and 12351, magnification ranges from 1 to 1000.</td><td>- For parameter Nos. 7113, 7114, 7131, 7132, 12350, and 12351, magnification ranges from 1 to 2000.</td></tr>
<tr><td><u>Parameter No. 7113</u>
Magnification when manual handle feed amount selection signals MP1 = 0 and MP2 = 1</td><td><u>Parameter No. 7114</u>
Magnification when manual handle feed amount selection signals MP1 = 1 and MP2 = 1</td></tr>
<tr><td colspan="2">[When bit 5 (MPX) of parameter No. 7100 = 0]
Magnification common to all the generators in the path
[When bit 5 (MPX) of parameter No. 7100 = 1]
Magnification used by the first generator in the path</td></tr>
<tr><td><u>Parameter No. 7131</u>
Magnification when manual handle feed amount selection signals MP21 = 0 and MP22 = 1</td><td><u>Parameter No. 7132</u>
Magnification when manual handle feed amount selection signals MP21 = 1 and MP22 = 1</td></tr>
<tr><td colspan="2">When bit 5 (MPX) of parameter No. 7100 is set to 1, the magnification used by the second generator in the path applies.</td></tr>
<tr><td rowspan="2"><u>Parameter No. 12350</u>
Magnification when per-axis manual handle feed amount selection signals MP1 = 0 and MP2 = 1</td><td><u>Parameter No. 12351</u>
Magnification when per-axis manual handle feed amount selection signals MP1 = 1 and MP2 = 1</td></tr>
</table>

B.25.2 Differences in Diagnosis Display

None.

B.26 PMC AXIS CONTROL

B.26.1 Differences in Specifications

Differences common to 1-path control and 2-path control

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Relationship with synchronous control (synchronous control of synchronous/composite control)</td><td>- PMC axis control can be applied for any axis other than a synchronous slave axis.</td><td>- PMC axis control cannot be applied for any axis under synchronous control.</td></tr>
<tr><td>Relationship with the feed-forward and advanced preview feed-forward functions</td><td>- Enable or disable the functions by using bit 7 (NAH) of parameter No. 1819, bit 3 (G8C) of parameter No. 8004, and bit 4 (G8R) of parameter No. 8004 in combination.</td><td>- Neither the feed-forward nor advanced preview feed-forward function is available for an axis under PMC axis control.
Bit 3 (G8C) and bit 4 (G8R) of parameter No. 8004 are not available.</td></tr>
<tr><td>Data range of rapid traverse rate for rapid traverse (00h), 1st to 4th reference position return (07h to 0Ah), and machine coordinate system selection (20h)</td><td>- The data range is as follows.
<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Valid data range</th><th rowspan="2">Unit of data</th></tr>
<tr><th>IS-A, IS-B</th><th>IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Millimeter machine</td><td>30 to 15000</td><td>30 to 12000</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>30 to 6000</td><td>30 to 4800</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>30 to 15000</td><td>30 to 12000</td><td>deg/min</td></tr>
</table></td><td>- 1 to 65535
The data unit is as follows.
<table>
<tr><th colspan="2" rowspan="2"></th><th>Data unit</th><th rowspan="2">Unit</th></tr>
<tr><th>IS-A to IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Metric machine</td><td>1</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>0.1</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>1</td><td>deg/min</td></tr>
</table></td></tr>
<tr><td>Data range of total moving distance for rapid traverse (00h), cutting feed - feed per minute (01h), cutting feed - feed per revolution (02h), and skip - feed per minute (03h)</td><td>- The data range is as follows.
<table>
<tr><th>Input increment</th><th>IS-B</th><th>IS-C</th><th>Unit</th></tr>
<tr><td>mm unit input
deg unit input</td><td>±99999.999</td><td>±9999.9999</td><td>mm
deg</td></tr>
<tr><td>inch unit input</td><td>±9999.9999</td><td>±999.99999</td><td>inch</td></tr>
</table></td><td>- The data range is as follows.
<table>
<tr><th>IS-A</th><th>IS-B,IS-C</th></tr>
<tr><td>-99999999 to 99999999 (8 digits)</td><td>-999999999 to 999999999 (9 digits)</td></tr>
</table>
The data unit is the minimum setting unit for the corresponding axis. (See the table below.)
<table>
<tr><th>Setting unit</th><th>Minimum data unit</th></tr>
<tr><td>IS-A</td><td>0.01</td></tr>
<tr><td>IS-B</td><td>0.001</td></tr>
<tr><td>IS-C</td><td>0.0001</td></tr>
</table></td></tr>
<tr><td>Data range of cutting feedrate for rapid traverse (01h) and skip - feed per minute (03h)</td><td>- 1 to 65535
The specified feedrate must be within the range shown in the table below.
<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Valid data range</th><th rowspan="2">Unit of data</th></tr>
<tr><th>IS-B</th><th>IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Millimeter machine</td><td>1 to 100000</td><td>0.1 to 12000.0</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>0.01 to 4000.00</td><td>0.01 to 480.000</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>1 to 100000</td><td>0.1 to 12000.0</td><td>deg/min</td></tr>
</table></td><td>- 1 to 65535</td></tr>
<tr><td>Function to increase the specification unit by a factor of 200 for continuous feed (06h)</td><td>- Not available.</td><td>- By setting 1 in bit 2 (JFM) of parameter No. 8004, it is possible to increase the specification unit by a factor of 200.

<u>Bit 2 (JFM) of parameter No. 8004</u>
Set the specification unit of feedrate data for specifying the continuous feed command for PMC axis control.
<table>
<tr><th>Increment system</th><th>Bit 2 (JFM) of No. 8004</th><th>Millimeter input (mm/min)</th><th>Inch input (inch/min)</th><th>Rotation axis (min^{-1})</th></tr>
<tr><td rowspan="2">IS-B</td><td>0</td><td>1</td><td>0.01</td><td>0.00023</td></tr>
<tr><td>1</td><td>200</td><td>2.00</td><td>0.046</td></tr>
<tr><td rowspan="2">IS-C</td><td>0</td><td>0.1</td><td>0.001</td><td>0.000023</td></tr>
<tr><td>1</td><td>20</td><td>0.200</td><td>0.0046</td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Maximum feedrate for continuous feed (06h)</td><td colspan="2">
- When an override of 254% is applied
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr>
<tr><td>1 time</td><td>166458
mm/min</td><td>1664.58
inch/min</td><td>16645
mm/min</td><td>166.45
inch/min</td></tr>
<tr><td>10 times</td><td>1664589
mm/min</td><td>16645.89
inch/min</td><td>166458
0mm/min</td><td>1664.58
inch/min</td></tr>
</table>
- When override is canceled
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr>
<tr><td>1 time</td><td>65535
mm/min</td><td>655.35
inch/min</td><td>6553
mm/min</td><td>65.53
inch/min</td></tr>
<tr><td>10 times</td><td>655350
mm/min</td><td>6553.50
inch/min</td><td>65535
mm/min</td><td>655.35
inch/min</td></tr>
</table>
</td><td>
- When an override of 254% is applied
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input
(mm/min)</th><th>Inch input
(inch/min)</th><th>Metric input
(mm/min)</th><th>Inch input
(inch/min)</th></tr>
<tr><td>1 time</td><td>166458</td><td>1664.58</td><td>16645</td><td>166.46</td></tr>
<tr><td>10 times</td><td>999000</td><td>16645.89</td><td>99900</td><td>1664.58</td></tr>
<tr><td>200 times</td><td>999000</td><td>39330.0</td><td>99900</td><td>3933.0</td></tr>
</table>
- When override is canceled
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input
(mm/min)</th><th>Inch input
(inch/min)</th><th>Metric input
(mm/min)</th><th>Inch input
(inch/min)</th></tr>
<tr><td>1 time</td><td>65535</td><td>655.35</td><td>6553</td><td>65.53</td></tr>
<tr><td>10 times</td><td>655350</td><td>6553.5</td><td>65535</td><td>655.35</td></tr>
<tr><td>200 times</td><td>999000</td><td>39330.0</td><td>999000</td><td>3933.0</td></tr>
</table>
</td></tr>
<tr><td>Minimum unit of feedrate for the speed command (10h)</td><td colspan="2">The minimum unit of feedrate is given by the expressions shown below. The value must be specified as an integer. No finer value may be specified.
A calculation is made according to IS-B.
Fmin: Minimum feedrate unit
P: Number of pulses per revolution of a detector for speed feedback</td></tr>
<tr><td></td><td>- Fmin = P ÷ 7500 (mm/min)</td><td>- Fmin = P ÷ 1000 (mm/min)</td></tr>
<tr><td>Speed specification in the speed command (10h)</td><td colspan="2">A speed is specified according to the expressions shown below.
A calculation is made according to IS-B.
F: Speed command (integer)
N: Servo motor speed (min^{-1})
P: Number of pulses per revolution of a detector for speed feedback</td></tr>
<tr><td></td><td>- F = N × P ÷ 7500 (mm/min)</td><td>- F = N × P ÷ 1000 (mm/min)</td></tr>
<tr><td>Setting range of torque data for torque control (11h)</td><td>- The setting range is as follows.
<table>
<tr><th>Valid data range</th><th>Unit</th></tr>
<tr><td>-99999999 to +99999999</td><td>0.0000 1 Nm</td></tr>
</table>
</td><td>- The setting range is as follows.
<table>
<tr><th>Valid data range</th><th>Unit</th></tr>
<tr><td>-999999999 to +999999999 (9 digits)</td><td>0.0000 1 Nm</td></tr>
</table>
</td></tr>
<tr><td>Note on executing an absolute command from the program for an axis subject to PMC axis control during automatic operation</td><td colspan="2">- [For Series 0i-D]
When you switch to PMC axis control to execute a move command during automatic operation and then switch back to NC axis control to execute an absolute command from the program for the moved axis, that PMC command needs to be executed using a non-buffering M code.

For example, when an absolute command is executed in a N40 block after PMC control is applied to Y axis, as in the example below, PMC axis control needs to be executed in a non-buffering M code (N20 block).

O0001 ;
N10 G94 G90 G01 X20. Y30. F3000 ;
N20 M55 ; → Executes PMC axis control for the Y axis.
N30 X70. ;
N40 Y50. ;
N50 M30 ;

Execute PMC axis control as follows.
1. After the output of the auxiliary function strobe signal MF for M55, start PMC axis control.
2. Upon completion of PMC axis control, input the completion signal FIN for M55.
- [For Series 0i-C]
Control does not need to be executed using a non-buffering M code.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Acceleration/deceleration control for an axis synchronized with external pulses using external pulse synchronization (0Bh, 0Dh to 0Fh)	- Depends on bit 2 (SUE) of parameter No. 8002. **Bit 2 (SUE) of parameter No. 8002** With the external pulse synchronization command for PMC axis control, the acceleration/deceleration of the axis synchronized with external pulses is: 0: Controlled (exponential acceleration/deceleration). 1: Not controlled.	- Bit 2 (SUE) of parameter No. 8002 is not available. The acceleration/deceleration of the axis synchronized with external pulses is controlled (exponential acceleration/deceleration).
Inch/metric conversion for a linear axis controlled only by PMC axis control	- Depends on bit 0 (PIM) of parameter No. 8003. **Bit 0 (PIM) of parameter No. 8003** When the axis controlled only by PMC axis control (see parameter No. 1010) is a linear axis, inch/metric input: 0: Influences the axis. 1: Does not influence the axis.	- Bit 0 (PIM) of parameter No. 8003 is not available. Parameter No. 1010 is not available, either. For a linear axis controlled only by PMC axis control, set rotation axis type B (set 1 in both bit 1 and bit 0 of parameter No. 1006) to avoid the influence of inch/metric input.
Setting to change all axes to CNC axes or PMC axes	- Depends on bit 1 (PAX) of parameter No. 8003. **Bit 1 (PAX) of parameter No. 8003** When 0 is set as the number of CNC control axes (parameter No. 1010), all axes are changed to: 0: CNC axes. 1: PMC axes.	- Bit 1 (PAX) of parameter No. 8003 is not available. Parameter No. 1010 is not available, either. There is no parameter to change all axes to PMC axes.
If the PMC issues an axis control command for an axis when the tool is waiting for the auxiliary function completion signal after moving that axis according to a move command and an auxiliary function specified from the CNC side	- Depends on bit 0 (CMV) of parameter No. 8004. **Bit 0 (CMV) of parameter No. 8004** If the PMC issues an axis control command for an axis when the tool is waiting for the auxiliary function completion signal after moving that axis according to a move command and an auxiliary function specified from the CNC side: 0: Alarm PS0130 is issued. 1: The axis control command from the PMC side is executed.	- Bit 0 (CMV) of parameter No. 8004 is not available. The axis control command from the PMC side is executed.
If the CNC issues a command for an axis when that axis is being moved by the axis control command from the PMC side	- Depends on bit 1 (NMT) of parameter No. 8004. **Bit 1 (NMT) of parameter No. 8004** If the CNC issues a command for an axis when that axis is being moved by the axis control command from the PMC side: 0: Alarm PS0130 is issued. 1: A command that does not involve moving the axis is executed without an alarm.	- Bit 1 (NMT) of parameter No. 8004 is not available. A command that does not involve moving the axis is executed without an alarm. (If the command involves moving the axis, alarm PS0130 is issued.)

Function	Series 0*i*-C	Series 0*i*-D
Setting of diameter/radius specification for the amount of travel and feedrate when diameter programming is specified for a PMC-controlled axis	- This item is determined by using bit 7 (NDI) of parameter No. 8004 and bit 1 (CDI) of parameter No. 8005 in combination.	- Bit 7 (NDI) of parameter No. 8004 is not available. The item is determined by bit 1 (CDI) of parameter No. 8005. **Bit 1 (CDI) of parameter No. 8005** In PMC axis control, when diameter programming is specified for a PMC-controlled axis: 0: The amount of travel and feedrate are each specified with a radius. 1: The amount of travel is specified with a diameter while the feedrate is specified with a radius.
Individual output of the auxiliary function	- Depends on bit 7 (MFD) of parameter No. 8005. **Bit 7 (MFD) of parameter No. 8005** The individual output of the auxiliary function for PMC axis control function is: 0: Disabled. 1: Enabled.	- Bit 7 (MFD) of parameter No. 8005 is not available. The individual output of the auxiliary function for PMC axis control function is enabled.
Function to exert position control for the speed command (10h)	- Depends on bit 4 (EVP) of parameter No. 8005. **Bit 4 (EVP) of parameter No. 8005** The speed of PMC axis control is specified by: 0: Speed command. 1: Position command.	- Depends on bit 4 (EVP) of parameter No. 8005. Note that, for the EVP=1 setting to take effect, 1 must be set in bit 2 (VCP) of parameter No. 8007. **Bit 2 (VCP) of parameter No. 8007** The speed command in PMC axis control is: 0: FS10/11 type. 1: FS0 type.
In-position check for an axis controlled only by PMC axis control	- Depends on bit 2 (IPA) of parameter No. 8006. **Bit 2 (IPA) of parameter No. 8006** In the case of an axis controlled only by PMC axis control (see parameter No. 1010), in-position check is: 0: Performed when no move command is specified for the PMC axis. 1: Always not performed.	- Bit 2 (IPA) of parameter No. 8006 is not available. Parameter No. 1010 is not available, either. The check is performed when no move command is specified for the PMC axis. Otherwise, the processing is determined by bit 6 (NCI) of parameter No. 8004. **Bit 6 (NCI) of parameter No. 8004** When the PMC-controlled axis is decelerated, in-position check is: 0: Performed. 1: Not performed.

Function	Series 0*i*-C	Series 0*i*-D
No in-position check signal for a PMC-controlled axis and no in-position check signals for individual axes	- Depends on bit 0 (NIS) of parameter No. 8007. **Bit 0 (NIS) of parameter No. 8007** For in-position check for a PMC axis, the no in-position check signal NOINPS<G023.5> and no in-position check signals for individual axes NOINP1<G359> to NOINP5<G359> are: 0: Disabled. 1: Enabled.	- Bit 0 (NIS) of parameter No. 8007 is not available. The no in-position check signal NOINPS<G023.5> and no in-position check signals for individual axes NOINP1<G359> to NOINP5<G359> are disabled for in-position check for a PMC axis.
Minimum speed for rapid traverse override in PMC axis control	- Set the value in parameter No. 8021.	- Parameter No. 8021 is not available. The minimum speed for rapid traverse override cannot be set.

Differences regarding 2-path control

Function	Series 0*i*-C	Series 0*i*-D
Relationship with composite control	- PMC axis control can also be applied to axes subject to composite control.	- PMC axis control cannot be applied to axes subject to composite control.
Setting when groups A to D in the path 2 is used.	- 1 (group A) to 4 (group D) are set in parameter No. 8010 for the path 2.	- 5 (group A for the path 2) to 8 (group D for the path 2) are set in the axis parameter No. 8010 controlled in the path 2. **Parameter No. 8010** Specify the DI/DO group to be used to specify a command for each PMC-controlled axis.

B.26.2 Differences in Diagnosis Display

None.

B.27 EXTERNAL SUBPROGRAM CALL (M198)

B.27.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Address P format when calling a subprogram on the memory card (file number specification/program number specification)	- Depends on bit 2 (SBP) of parameter No. 3404. **Bit 2 (SBP) of parameter No. 3404** In the external device subprogram call M198, address P is specified using: 0: File number. 1: Program number.	- To call a subprogram, the program number must always be specified in address P. When calling a subprogram on the memory card, the processing is not dependent on the setting of bit 2 (SBP) of parameter No. 3404.
Multiple call alarm	If a subprogram called by an external subprogram call specifies a further external subprogram call, the following alarms are issued, respectively:	
	- Alarm PS0210	- Alarm PS1080
External subprogram call in MDI mode	- Enabled.	- Depends on bit 1 (MDE) of parameter No. 11630. **Bit 1 (MDE) of parameter No. 11630** In MDI mode, an external device subprogram call (M198 command) is: 0: Disabled. (Alarm PS1081 is issued.) 1: Enabled.

B.27.2 Differences in Diagnosis Display

None.

B.28 SEQUENCE NUMBER SEARCH

B.28.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Return from a subprogram to the calling program's block that has a specified sequence number
Sequence number search when (M99 Pxxxxx) is executed</td><td>- The calling program is searched from the beginning, and control is returned to the first block found to have sequence number Nxxxxx.</td><td>- The calling program is searched in a forward direction from the block that called the subprogram, and control is returned to the first block found to have sequence number Nxxxxx.
If the specified sequence number is not found, the calling program is searched from the beginning, and control is returned to the first block found to have sequence number Nxxxxx.</td></tr>
<tr><td colspan="2">Example) Main program
O0001 ;
N100 ; (1)
N100 ; (2)
M98 P9001 ;
N100 ; (3)
N100 ; (4)
M30 ;
Sub program
O9001 ;
M99 P100 ;
- [For Series 0i-C]
Control is returned to block (1).
- [For Series 0i-D]
Control is returned to block (3).

⚠ WARNING
Be sure to avoid writing two or more identical sequence numbers in a program. Doing so may cause the search to find unintended blocks.</td></tr>
</table>

B.28.2 Differences in Diagnosis Display

None.

B.29 STORED STROKE CHECK

B.29.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Stored stroke check immediately following powering on	- This function is always enabled for all axes.	- It is possible to select whether to enable or disable the function on an axis-by-axis basis using bit 0 (DOT) of parameter No. 1311. **Bit 0 (DOT) of parameter No. 1311** The stored stroke limit check immediately following powering on is: 0: Disabled. 1: Enabled. **NOTE** This function stores machine coordinates using software and therefore imposes a burden on the system. Disable the function for those axes that do not require it. Movements made while the power is off are not reflected on the machine coordinate system immediately after powering on.
	- Machine coordinates are set upon powering on. Absolute and relative coordinates are not set. (They are set when the absolute position detector is provided.)	- Machine coordinates are set upon powering on. Absolute and relative coordinates are set based on these machine coordinates.
Y and J address specification using G22	Not available.	- Available for both the T series and M series.
Overtravel alarm	- Stored stoke check 2 does not support bit 7 (BFA) of parameter No. 1300. Therefore, if an interference alarm occurs, the tool stops after entering the prohibited area. This makes it necessary to make the prohibited area slightly larger than actually necessary.	- Stored stoke check 2 also supports bit 7 (BFA) of parameter No. 1300. Setting 1 in BFA allows the tool to stop before entering the prohibited area, thus eliminating the need to make the prohibited area slightly larger than actually necessary. **Bit 7 (BFA) of parameter No. 1300** If a stored stoke check 1, 2, or 3 alarm occurs, if an interference alarm occurs with the inter-path interference check function (T series), or if an alarm occurs with chuck/tail stock barrier (T series), the tool stops: 0: After entering the prohibited area. 1: Before entering the prohibited area.

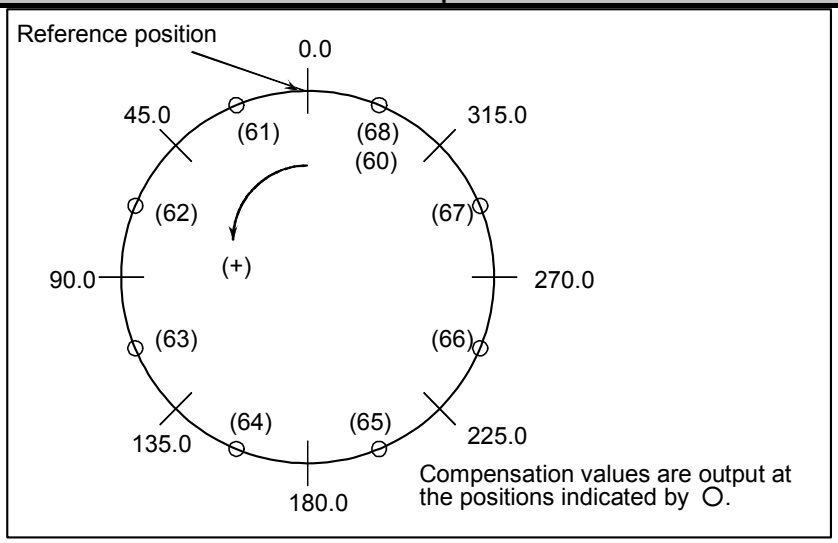
Function	Series 0*i*-C	Series 0*i*-D
Operation continuation after automatic alarm cancellation when a soft OT1 alarm is issued during the execution of an absolute command in automatic operation	- When the operation is resumed, the tool moves the remaining travel distance of the block that caused the soft OT. Therefore, the program can be continued if the tool is moved through manual intervention beyond the remaining travel distance.	- When the operation is resumed, the tool moves toward the end point of the block that caused the soft OT, causing another soft OT and making it impossible to continue the program. For details, refer to "STORED STROKE CHECK 1" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN).

B.29.2 Differences in Diagnosis Display

None.

B.30 STORED PITCH ERROR COMPENSATION

B.30.1 Differences in Specifications

Function	Explanation
Value of parameter No. 3621 for the setting of a rotary axis (type A)	 - Amount of movement per rotation: 360° - Interval between pitch error compensation positions: 45° - Number of the compensation position of the reference position: 60 In the above case, the values of the parameters are as follows.

Parameter	Series 0*i*-C	Series 0*i*-D
No. 3620: Number of the compensation position of the reference position	60	60
No. 3621: Smallest compensation position number	60	61
No. 3622: Largest compensation position number	68	68
No. 3623: Compensation magnification	1	1
No. 3624: Interval between compensation positions	45000	45000
No. 3625: Amount of movement per rotation	360000	360000

The value of parameter No. 3621 is as follows.
Series 0*i*-C
= Number of the compensation position of the reference position (parameter No. 3620)
Series 0*i*-D
= Number of the compensation position of the reference position (parameter No. 3620) + 1

B.30.2 Differences in Diagnosis Display

None.

B.31 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION

B.31.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Behavior of the manual screen erasure function ("<CAN> + function key") when an alarm is issued	- When an alarm is issued (including one associated with the other path), the manual screen erasure function is enabled. ("<CAN> + function key" erases the screen.)	- When an alarm is issued (including one associated with the other path), the manual screen erasure function is disabled. ("<CAN> + function key" does not erase the screen.)
Redisplay of the screen upon mode switching	- When the operation mode is switched while the screen is erased:	
	The screen is not redisplayed. (The screen remains erased.)	The screen is redisplayed.
Function key input when the screen is erased or displayed	- Select the behavior using bit 2 (NFU) of parameter No. 3209. **Bit 2 (NFU) of parameter No. 3209** When a function key is pressed to erase or display the screen for the screen erasure or automatic screen erasure function, the screen change using a function key is: 0: Performed. 1: Not performed.	- Bit 2 (NFU) of parameter No. 3209 is not available. The tool always behaves as when 1 is set in bit 2 (NFU) of parameter No. 3209.
Time before the automatic screen erasure function starts	- Set the value in parameter No.3123.	
	The value range is 1 to 255 (minutes).	The value range is 1 to 127 (minutes).

B.31.2 Differences in Diagnosis Display

None.

B.32 RESET AND REWIND

B.32.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Modal data when reset during the execution of a block	- If reset occurs during the execution of a block, the states of the modal G codes and modal addresses (N, F, S, T, M, etc.) specified in that block are handled as follows.	
	Maintained.	Not maintained. The states return to those of the modal data specified in the preceding blocks. (The modal data is updated after the specified block is fully executed.) Example) If reset occurs before positioning is completed in the N2 block in the program shown below, the T code and offset return to the data of the preceding tool (T0101) data. N1 G00 X120. Z0. T0101 ; ; N2 G00 X180. Z20. T0202 ; ;
Information in a block that is pre-read when a reset is made during an automatic operation (contents of the buffer)	- The information in the block may or may not be held depending on whether MDI mode is in progress. **In MDI mode** The information in the block is held. **In modes other than MDI mode** The information in the block is not held.	- The information in the block is not held regardless whether MDI mode is in progress.

B.32.2 Differences in Diagnosis Display

None.

B.33 MANUAL ABSOLUTE ON AND OFF

B.33.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Absolute coordinates during automatic tool compensation change	- If tool compensation is automatically changed when the manual absolute signal *ABSM(Gn006.2) is set to 1, absolute coordinates are handled as follows.	
	Absolute coordinates are not changed.	Absolute coordinates are changed according to the amount of tool compensation resulting from the coordinate shift.

B.33.2 Differences in Diagnosis Display

None.

B.34 MEMORY PROTECTION SIGNAL FOR CNC PARAMETER

B.34.1 Differences in Specifications

Function	Series 0*i*-TTC	Series 0*i*-D
Memory protection signal for CNC parameter KEYP, KEY1 to KEY4 <G046.0, G046.3 to G046.6>	- The signal is different for each path.	- The signal is common to all paths.
Parameter to enable the KEYP signal	- Enable or disable the signal using bit 7 (PK5) of parameter No. 3292. This is a bit path parameter.	- Enable or disable the signal using bit 0 (PKY) of parameter No. 3299. This is a bit system common parameter.

B.34.2 Differences in Diagnosis Display

None.

B.35 EXTERNAL DATA INPUT

B.35.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Number of external alarm messages and message length	- [Number of messages that can be set at a time] Up to 4 messages [Length of a message] Up to 32 characters	- [Number of messages that can be set at a time] Depends on bit 1 (M16) of parameter No. 11931. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 1 (M16) of parameter No. 11931** The maximum number of external alarm messages or external operator messages that can be displayed in connection with external data input or external messages is: 0: 4. 1: 16. [Length of a message] Up to 32 characters
Display format of external alarm messages	- [Alarm numbers that can be sent] 0 to 999 [How to distinguish these numbers from general alarm numbers] Add 1000 to the number sent	- Depends on bit 0 (EXA) of parameter No. 6301. **Bit 0 (EXA) of parameter No. 6301** Select the external alarm message specification. 0: The alarm numbers that can be sent range from 0 to 999. The CNC displays an alarm number, with 1000 added to the number following the character string "EX". 1: The alarm numbers that can be sent range from 0 to 4095. The CNC displays an alarm number, with the character string "EX" added in front of it.
Number of external operator messages and message length	- Depends on bit 0 (OM4) of parameter No. 3207. **Bit 0 (OM4) of parameter No. 3207** The external operator message screen can display: 0: Up to 256 characters in up to 1 message. 1: Up to 64 characters in up to 4 messages.	- Bit 0 (OM4) of parameter No. 3207 is not available. [Number of messages that can be set at a time] Depends on bit 1 (M16) of parameter No. 11931. Select either up to 4 or 16 messages. [Length of a message] 256 characters or less

Function	Series 0*i*-C	Series 0*i*-D
Display format of external operator messages	- [Message numbers that can be sent] 0 to 999 [How to distinguish these numbers from alarm and other numbers] **Messages from 0 to 99** The message is displayed on the screen along with the number. The CNC adds 2000 to this number for distinction. **Messages from 100 to 999** Only the message is displayed on the screen without the number.	- Depends on bit 1 (EXM) of parameter No. 6301. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 1 (EXM) of parameter No. 6301** Select the external operator message specification. 0: The message numbers that can be sent range from 0 to 999. A message from 0 to 99 is displayed on the screen along with the number. The CNC adds 2000 to this number for distinction. As for the messages from 100 to 999, only the message is displayed on the screen without the number. 1: The message numbers that can be sent range from 0 to 4095. A message from 0 to 99 is displayed on the screen along with the number. The CNC adds the character string "EX" in front of the number. As for the messages from 100 to 4095, only the message is displayed on the screen without the number.
Data range of external operator message numbers	**Parameter No. 6310** The data range of external operator message numbers is as follows.	
	- 0 to 1000	- 0 to 4096
When an external program number search is done with 0 set as the program number	- An alarm is not issued; the search is not done, either.	- Alarm DS0059 is issued.
Input of an external tool offset for an invalid function compensation value	- The input is ignored without issuing an alarm.	- Alarm DS1121 is issued.
Number of history messages for external operator messages and message length	- Make a selection using bit 7 (MS1) and bit 6 (MS0) of parameter No. 3113 in combination.	- Bit 7 (MS1) and bit 6 (MS0) of parameter No. 3113 are not available. [Number of history messages] Up to 32 [History message length of a message] Up to 256 characters

B.35.2 Differences in Diagnosis Display

None.

B.36 DATA SERVER FUNCTION

B.36.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Memory operation mode	- The memory operation mode is not supported.	- In the memory operation mode, the following operations can be performed for a program registered with the data server: 1. Select the program on the data server as the main program and run it in the memory mode. 2. Call a subprogram or custom macro in the same directory as the main program on the data server. 3. Edit the program, including inserting, deleting, and replacing words.
Simultaneous call from two paths	In a 2-path system, a simultaneous external subprogram call (M198) of a data server program from both paths is:	
	- Allowed under the following conditions. [Storage mode] Both paths must use the same work directory. [FTP mode] Both paths must use the same connection host.	- Not allowed. Use the subprogram/custom macro call for the memory operation mode instead.

B.36.2 Differences in Diagnosis Display

None.

B.37 POWER MATE CNC MANAGER

B.37.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
4-slave display function	- By setting 1 in bit 0 (SLV) of parameter No. 0960, it is possible to split the screen into four windows, enabling up to four slaves to be displayed. **Bit 0 (SLV) of parameter No. 0960** When Power Mate CNC Manager is selected, the screen: 0: Displays one slave. 1: Is split into four windows, enabling up to four slaves to be displayed.	- Bit 0 (SLV) of parameter No. 0960 is not available. One slave is always displayed. When there is more than one slave, you switch the active slave by using the relevant soft key.

B.37.2 Differences in Diagnosis Display

None.

B.38 CHUCK/TAIL STOCK BARRIER

B.38.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Overtravel alarm	- Bit 7 (BFA) of parameter No. 1300 is not supported. Therefore, if an interference alarm occurs, the tool stops after entering the prohibited area. This makes it necessary to make the prohibited area slightly larger than actually necessary.	- Bit 7 (BFA) of parameter No. 1300 is supported. Setting 1 in BFA allows the tool to stop before entering the prohibited area, thus eliminating the need to make the prohibited area slightly larger than actually necessary. **Bit 7 (BFA) of parameter No. 1300** If a stored stoke check 1, 2, or 3 occurs, if an interference alarm occurs with the inter-path interference check function (T series), or if an alarm occurs with chuck/tail stock barrier (T series), the tool stops: 0: After entering the prohibited area. 1: Before entering the prohibited area.

B.38.2 Differences in Diagnosis Display

None.

B.39 THREADING CYCLE RETRACT (CANNED CUTTING CYCLE/MULTIPLE REPETITIVE CANNED CUTTING CYCLE)

B.39.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Return position after chamfering in multiple repetitive threading cycle (G76)	- The tool returns to the start point of the current cycle. For example, if it is the nth cycle, the tool returns to the position where the nth cut has been made.	- The tool returns to the start point of the threading cycle. This means that the tool returns to the position where it was before cutting, no matter how many cycles it has undergone.
Retraction after chamfering	- The specifications are as follows. [Acceleration/deceleration type] Acceleration/deceleration after interpolation for threading is used. [Time constant] The time constant for threading (parameter No. 1626) is used. [Feedrate] The feedrate set in parameter No. 1466 is used.	- Depends on bit 0 (CFR) of parameter No. 1611. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 0 (CFR) of parameter No. 1611** In threading cycle G92 or G76, retraction after threading uses: 0: Type of acceleration/deceleration after interpolation for threading, together with the threading time constant (parameter No.1626) and the feedrate set in parameter No.1466. 1: Type of acceleration/deceleration after interpolation for rapid traverse, together with the rapid traverse time constant and the rapid traverse rate.

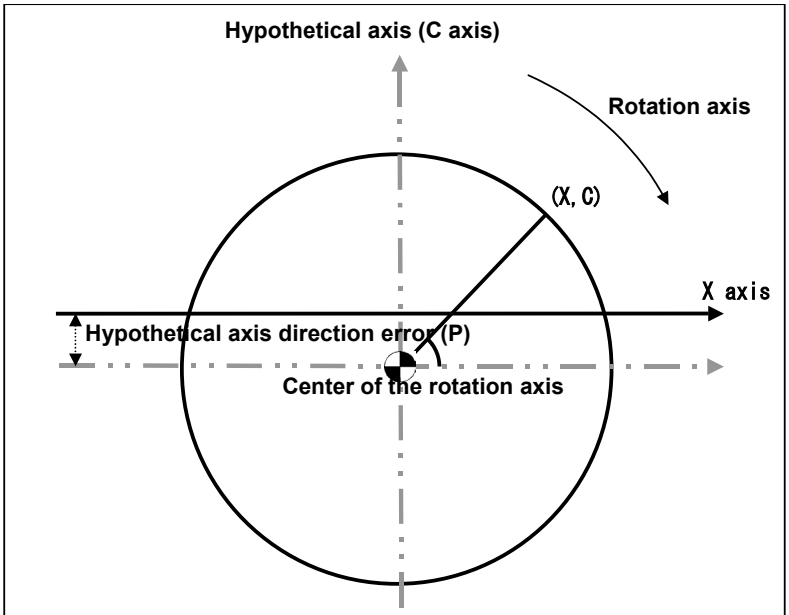
B.39.2 Differences in Diagnosis Display

None.

B.40 POLAR COORDINATE INTERPOLATION

B.40.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Coordinate system shift during polar coordinate interpolation (polar coordinate interpolation shift function)	- Not available.	- Enable or disable the function using bit 2 (PLS) of parameter No. 5450. **Bit 2 (PLS) of parameter No. 5450** The polar coordinate interpolation shift function is: 0: Not used. 1: Used. This enables machining using the workpiece coordinate system with a desired point which is not the center of the rotation axis set as the origin of the coordinate system in polar coordinate interpolation. For details, refer to "POLAR COORDINATE INTERPOLATION" in "USER'S MANUAL (LATHE SYSTEM)" (B-64304EN-1).

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Hypothetical axis direction compensation during polar coordinate interpolation</td><td colspan="2">- If the first axis of the plane is in a hypothetical axis direction relative to the center of the rotation axis, i.e. the center of the rotation axis is not on the X axis, the hypothetical axis direction compensation function in polar coordinate interpolation mode performs polar coordinate interpolation while taking the error into consideration. Set the error value in parameter No. 5464.


(X,C) X-C plane point (The center of the rotation axis is the origin o the X-C plane.)
X X axis coordinate value in the X-C plane
C Hypothetical axis coordinate value in the X-C plane
P Hypothetical axis direction error
(Set the value in parameter No. 5464.)</td></tr>
<tr><td></td><td>- This function is not available.</td><td>- This function is available.</td></tr>
<tr><td>Maximum cutting feedrate and feedrate clamp during polar coordinate interpolation</td><td>- Set the value in parameter No. 5462. When the value is 0, the feedrate is clamped by parameter No. 1422.</td><td>- Parameter No. 5462 is not available. Set the value in parameter No. 1430.</td></tr>
<tr><td>Automatic override and automatic feedrate clamp during polar coordinate interpolation</td><td>- Enable or disable the function using bit 1 (AFC) of parameter No. 5450.

<u>Bit 1 (AFC) of parameter No. 5450</u>
In the polar coordinate interpolation mode, automatic override and automatic feedrate clamp are:
0: Not performed.
1: Performed.</td><td>- Bit 1 (AFC) of parameter No. 5450 is not available.
Automatic override and automatic feedrate clamp are always performed.</td></tr>
</table>

B.40.2 Differences in Diagnosis Display

None.

B.41 PATH INTERFERENCE CHECK (2-PATH CONTROL)

B.41.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Interference alarm	- Bit 7 (BFA) of parameter No. 1300 is not supported. Therefore, if an interference alarm occurs, the tool stops after entering the prohibited area. This makes it necessary to make the prohibited area slightly larger than actually necessary.	- Bit 7 (BFA) of parameter No. 1300 is supported. Setting 1 in BFA allows the tool to stop before entering the prohibited area, thus eliminating the need to make the prohibited area slightly larger than actually necessary. **Bit 7 (BFA) of parameter No. 1300** If a stored stoke check 1, 2, or 3 alarm occurs, if an interference alarm occurs with the inter-path interference check function (T series), or if an alarm occurs with chuck/tail stock barrier (T series), the tool stops: 0: After entering the prohibited area. 1: Before entering the prohibited area.

B.41.2 Differences in Diagnosis Display

None.

B.42 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (2-PATH CONTROL)

B.42.1 Differences in Specifications

Function	Series 0*i*-TTC	Series 0*i*-D
Axis synchronous control (Series 0*i*-C: Quick synchronous control)	- Adding synchronous or composite control disables simple synchronous control.	- Adding synchronous or composite control does not disable simple synchronous control. - The master and slave axes used for axis synchronous control cannot be used for synchronous control. - Composite control is available for the master axis used for axis synchronous control, while it is not available for the slave axis.
Feed forward function and cutting/rapid traverse change function for synchronous and composite axes of another path	- Make a selection using bit 1 (SVF) of parameter No. 8165. **Bit 1 (SVF) of parameter No. 8165** In synchronous or composite control, the feed forward function and cutting/rapid traverse change function for synchronous and composite axes of another path are: 0: Disabled. 1: Enabled.	- Bit 1 (SVF) of parameter No. 8165 is not available. The tool always behaves as when SVF is set to 1. (The feed forward function and cutting/rapid traverse change function are enabled for synchronous and composite axes of another path.)
Move command when neither synchronous nor composite control is in effect	- Not prohibited.	- Make a selection using bit 7 (NUMx) of parameter No. 8163. **Bit 7 (NUMx) of parameter No. 8163** When neither synchronous nor composite control is in effect, specifying the move command for an axis that is set with this parameter is: 0: Not prohibited. 1: Prohibited. (Alarm PS0353 is issued.)

Function	Series 0*i*-TTC	Series 0*i*-D
Behavior when an alarm is issued in relation to synchronous or composite control	- Both paths are placed in the feed hold state.	- Make a selection using bit 0 (MPA) of parameter No. 8168. **Bit 0 (MPA) of parameter No. 8168** If an alarm is issued in relation to synchronous, composite, or superposition control: 0: Both paths are placed in the feed hold state. 1: Only the path including axes related to synchronous, composite, or superposition control is placed in the feed hold state. For example, when synchronous control is exerted in one path, only the path that caused the alarm is placed in the feed hold state. The handling of the other path depends on the setting of bit 1 (IAL) of parameter No. 8100.
Behavior when overtravel occurs for an axis under synchronous or composite control	- The synchronous or composite control mode is canceled.	- Make a selection using bit 5 (NCS) of parameter No. 8160. **Bit 5 (NCSx) of parameter No. 8160** If overtravel occurs for an axis under synchronous, composite, or superposition control, the synchronous, composite, or superposition control mode is: 0: Canceled. 1: Not canceled.
Switch between synchronous control axis selection signal and composite control axis selection signal during automatic operation	- The signals can be switched at any time.	- Use an M code command. Specify a waiting M code (M code without buffering) before and after the M code. When synchronous or composite control is exerted in one path, specify an M or other code without buffering before and after the M code that starts or cancels the control so as to prohibit the look-ahead operation.

Synchronous control

Item	Series 0*i*-TTC	Series 0*i*-D
G28 when the master axis is parking	- When the reference position of the slave axis is not established, the machine coordinates are moved to the coordinates set in parameter No. 1240, completing the reference position return.	- When the reference position of the slave axis is not established, alarm PS0354 occurs.
Update of the workpiece coordinates and relative coordinates of the slave axis under synchronous control	- Make a selection using bit 4 (SPN) of parameter No. 8164. **Bit 4 (SPN) of parameter No. 8164** The workpiece coordinates and relative coordinates of the slave axis under synchronous control are: 0: Updated. 1: Not updated.	- Bit 4 (SPN) of parameter No. 8164 is not available. The tool always behaves as when SPNx is set to 0 (coordinates are updated).

Item	Series 0*i*-TTC	Series 0*i*-D
Out-of-synchronization detection when synchronous control is exerted in one path (1 is set in bit 1 (SER) of parameter No. 8162)	- Out-of-synchronization detection is not performed.	- Out-of-synchronization detection is performed.
Manual handle interruption amount or mirror image mode for the master axis	- Always reflected on the slave axis.	- Select whether to reflect the amount or mode on the slave axis, using bit 5 (SMIx) of parameter No. 8163. **Bit 5 (SMIx) of parameter No. 8163** During synchronous control, the manual handle interruption amount or mirror image mode for the master axis is: 0: Reflected on the slave axis. 1: Not reflected on the slave axis.
Automatic setting of a workpiece coordinate system for the slave axis at the end of synchronous control	- A workpiece coordinate system is not automatically set for the slave axis.	- Make a selection using bit 6 (SPVx) of parameter No. 8167. **Bit 6 (SPVx) of parameter No. 8167** At the end of synchronous control, a workpiece coordinate system for the slave axis is: 0: Not automatically set. 1: Automatically set. The workpiece coordinate system to be set is determined by the machine coordinate values and the workpiece coordinate values of the reference points of the individual axes defined by parameter No. 1250.

Composite control

Item	Series 0*i*-TTC	Series 0*i*-D
G28 during composite control	- When the reference position of the composite axis of the other path is not established, the machine coordinates are moved to the coordinates set in parameter No. 1240, completing the reference position return.	- When the reference position of the composite axis of the other path is not established, alarm PS0359 occurs.
Composite control for the Cs contour axis reference position return command when composite control is exerted for Cs contour axes	- Select whether to use the composite function of the Cs contour axis reference position return command, by using bit 1 (CZMx) of parameter No. 8161. **Bit 1 (CZMx) of parameter No. 8161** When composite control is exerted for Cs contour axes, the composite control function for the Cs contour axis reference position return command is: 0: Not used. 1: Used.	- Bit 1 (CZMx) of parameter No. 8161 is not available. The tool always behaves as when CZMx is set to 1 (composite control is used).

Item	Series 0*i*-TTC	Series 0*i*-D
Manual handle interruption for composite axes	- Disabled.	- Enable or disable the interruption using bit 6 (MMIx) of parameter No. 8163. **Bit 6 (MMIx) of parameter No. 8163** During composite control, manual handle interruption for composite axes is: 0: Enabled. 1: Disabled.
Current position display during composite control (absolute/relative coordinates)	- Make a selection using bit 0 (MDXx) of parameter No. 8163. **Bit 0 (MDXx) of parameter No. 8163.** During composite control, the current position display (absolute/relative coordinates) shows: 0: Coordinate values of the local path. 1: Coordinate values of the mate path.	- Bit 0 (MDXx) of parameter No. 8163 is not available. The coordinate values of the local path are always displayed.
G53 during composite control	- Make a selection using bit 2 (CPMx) of parameter No. 8165. **Bit 2 (CPMx) of parameter No. 8165.** During composite control, machine coordinate system selection (G53) is: 0: Disabled. 1: Enabled. (The travel distance is calculated so that the machine moves according to the machine coordinate system selection signal of the mate path.)	- Bit 2 (CPMx) of parameter No. 8165 is not available. The tool always behaves as when CPMx is set to 1. (G53 is enabled.)
Constant acceleration/deceleration of acceleration time for acceleration/deceleration in rapid traverse for an axis subject to composite control (bit 4 (RPT) of parameter No. 1603)	- Make a selection using bit 0 (NLSx) of parameter No. 8167. **Bit 0 (NLSx) of parameter No. 8167** Constant acceleration/deceleration of acceleration time for acceleration/deceleration in rapid traverse for an axis subject to composite control (bit 4 (RPT) of parameter No. 1603) is: 0: Enabled. 1: Disabled.	- Bit 0 (NLSx) of parameter No. 8167 is not available. The tool always behaves as when NLSx is set to 1. (Constant acceleration/deceleration of acceleration time is enabled.)
Machine coordinates during composite control	- The coordinate values of the local path are displayed.	- Make a selection using bit 0 (MDMx) of parameter No. 8169. **Bit 0 (MDMx) of parameter No. 8169** The machine coordinates displayed during composite control are: 0: Coordinate values of the local path. 1: Machine coordinate values of the mate path.
Reading of machine coordinates (#5021 and later) during composite control	- The coordinate values of the local path are read.	- Make a selection using bit 1 (MVMx) of parameter No. 8169. **Bit 1 (MVMx) of parameter No. 8169** The machine coordinates (#5021 and later) that are read during composite control are: 0: Machine coordinate values of the local path. 1: Machine coordinate values of the mate path.

Item	Series 0*i*-TTC	Series 0*i*-D
Rapid traverse feedrate during composite control	- The rapid traverse feedrate of the specified axis is used.	- Make a selection using bit 2 (MRFx) of parameter No. 8169. **Bit 2 (MRFx) of parameter No. 8169** The rapid traverse feedrate used during composite control is: 0: Rapid traverse feedrate of the specified axis. 1: Rapid traverse feedrate of the moving axis.

B.42.2 Differences in Diagnosis Display

Item	Series 0*i*-TTC	Series 0*i*-D
Synchronization error value display for each axis	- Displayed in parameter No. 8182.	- Displayed in diagnosis No. 3502.

B.43 SUPERIMPOSED CONTROL (2-PATH CONTROL)

B.43.1 Differences in Specifications

Function	Series 0*i*-TTC	Series 0*i*-D
Axis synchronous control (Series 0*i*: Quick synchronous control)	- Adding superimposed control disables simple synchronous control.	- Adding superimposed control does not disable simple synchronous control. - The same axis can be used as the master axis for axis synchronous control and the master axis for superimposed control.
Feed hold when an alarm occurs with respect to superimposed control	- Both paths are placed in the feed hold state.	- Make a selection using bit 0 (MPA) of parameter No. 8168. **Bit 0 (MPA) of parameter No. 8168** The axis movement in-progress signal <Fn102> or axis movement direction signal <Fn106> for the slave axis during superimposed control: 0: Places both paths in the feed hold state. 1: Places only the path including axes related to superposition control in the feed hold state. (For example, when superposition control is exerted in one path, only the path that caused the alarm is placed in the feed hold state.)
Reference position return of the slave axis during superimposed control	- Not available.	- Not available. Alarm PS0363 occurs.
Multiple slave axes	- Superimposed control cannot be exerted when there are multiple slave axes and one master axis.	- Superimposed control can be exerted when there are multiple slave axes and one master axis.
Axis movement in-progress signal and axis movement direction signal for the slave axis during superimposed control	- State output is performed according to the result of adding superimposed move pulses.	- Make a selection using bit 4 (AXS) of parameter No. 8160. **Bit 4 (AXS) of parameter No. 8160** The axis movement in-progress signal <Fn102> or axis movement direction signal <Fn106> for the slave axis during superimposed control: 0: Performs state output according to the result of adding superimposed move pulses. 1: Performs state output according to the result of moving the individual axes, regardless of superimposed move pulses.

Function	Series 0*i*-TTC	Series 0*i*-D
Axis overtravel during superimposed control	- The superimposed control mode is canceled.	- Make a selection using bit 5 (NCS) of parameter No. 8160. **Bit 5 (NCS) of parameter No. 8160** If overtravel occurs for an axis under synchronous, composite, or superposition control, the synchronous, composite, or superposition control mode is: 0: Canceled. 1: Not canceled.
Switch between superimposed control axis selection signals during automatic operation	- The signals can be switched at any time. Note that both the master and slave axes must be stopped.	- Use an M code command. Specify a waiting M code (M code without buffering) before and after the M code. When superimposed control is exerted in one path, specify an M or other code without buffering before and after the M code that starts or cancels the control so as to prohibit the look-ahead operation.

B.43.2 Differences in Diagnosis Display

None.

B.44 Y AXIS OFFSET

B.44.1 Differences in Specifications

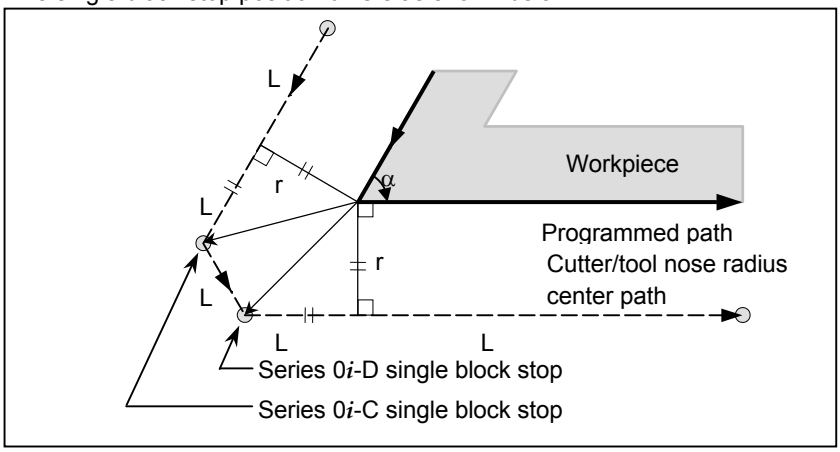
Function	Series 0*i*-C	Series 0*i*-D
Number of the axis for which the Y axis offset is used	- Make a selection using bit 7 (Y03) of parameter No. 5004. **Bit 7 (Y03) of parameter No. 5004** The Y axis offset is used for: 0: 4th axis. 1: 3rd axis.	- Make a selection using parameter No. 5043. When 0 or a value outside the data range is set, the Y axis offset is used for the Y axes of the basic three axes (X, Y, and Z).

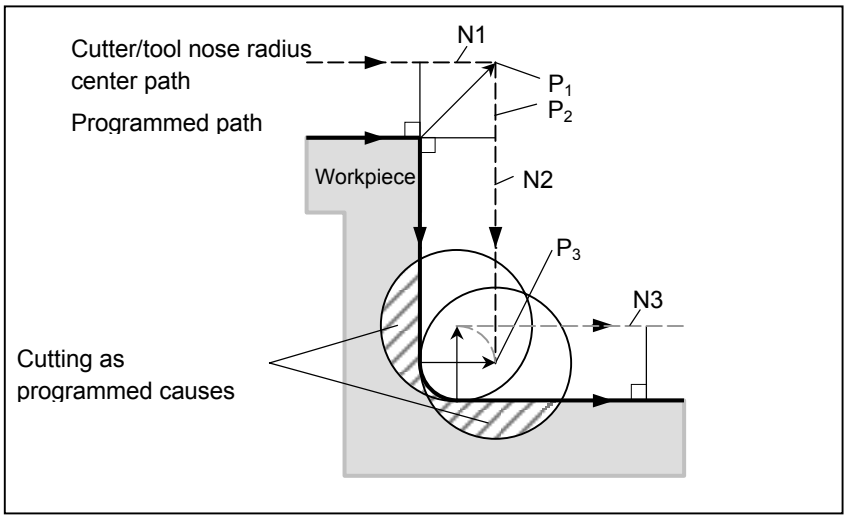
B.44.2 Differences in Diagnosis Display

None.

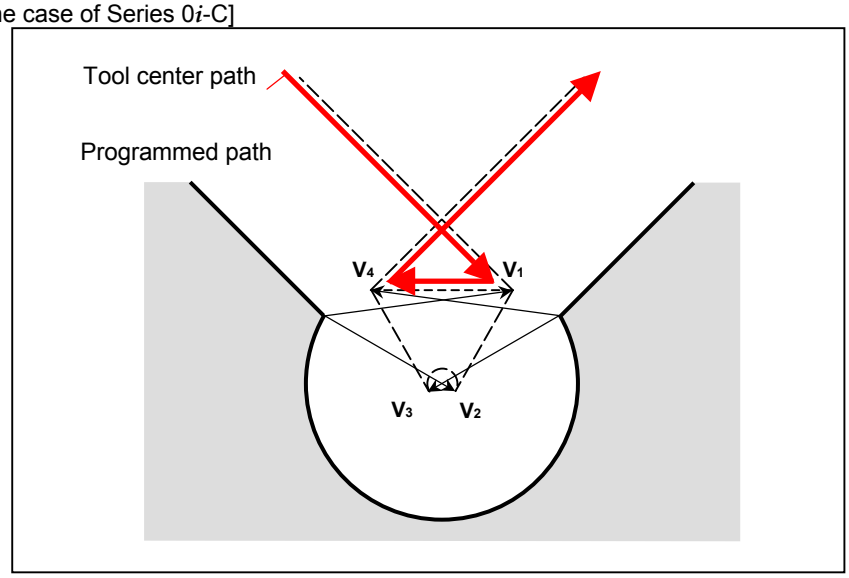
B.45 CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION

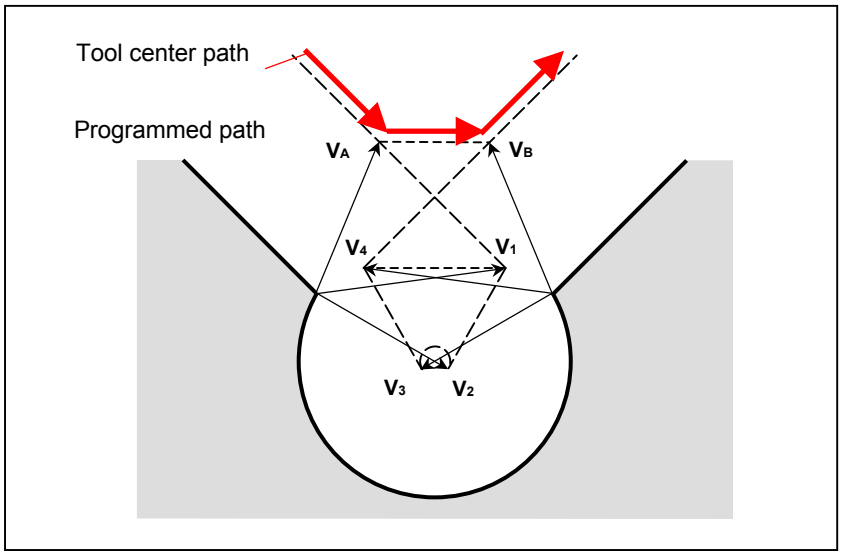
B.45.1 Differences in Specifications

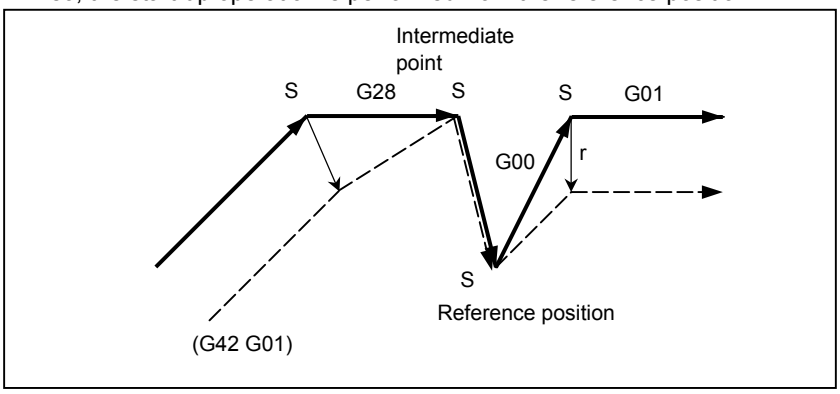
Function	Series 0*i*-C	Series 0*i*-D
Cutter compensation/tool nose radius compensation	- In Series 0*i*-D, the cutter compensation C (M series) and tool-nose radius compensation (T series) functions of Series 0*i*-C are collectively referred to as cutter compensation/tool nose radius compensation.	
Corner circular interpolation (G39)	- Not available.	- Available. It is included in cutter compensation/tool nose radius compensation. Since corner circular interpolation (G39) is always enabled, bit 2 (G39) of parameter No. 5008 is not available.
Cutter compensation/tool nose radius compensation in MDI operation	- Neither cutter compensation C nor tool nose radius compensation is available in MDI operation.	- Cutter compensation/tool nose radius compensation is also available in MDI operation.
Single block stop position during the cutter compensation/tool nose radius compensation mode	- The single block stop position differs as shown below. 	
Function to change the compensation direction intentionally (IJ type vector, KI type vector, and JK type vector)	- Not available.	- At the start of or during the cutter compensation/tool nose radius compensation mode, specify I, J, or K in a G00 or G01 block. This makes the compensation vector at the end point of the block perpendicular to the direction specified by I, J, or K. This way, you can change the compensation direction intentionally.

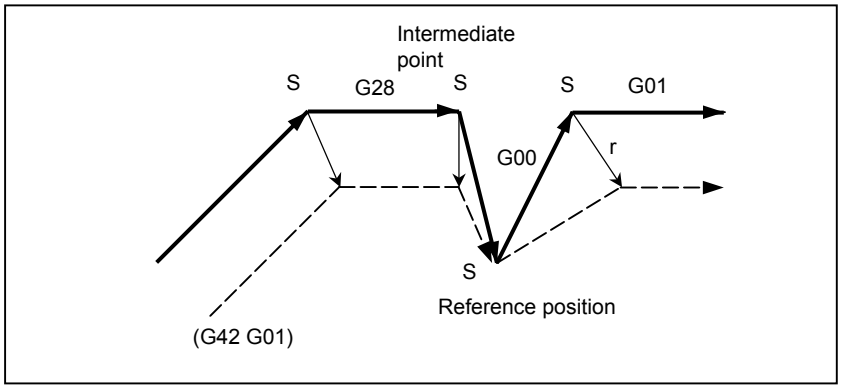
<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Stop position upon an overcutting alarm</td><td colspan="2">- If the specified radius value for circular interpolation is smaller than that for cutter compensation/tool nose radius compensation, as in the example below, performing compensation inwardly through cutter compensation/tool nose radius compensation causes overcutting, generating an alarm and stopping the tool. The stop position differs.

</td></tr>
<tr><td></td><td colspan="2">[When single block stop occurs in the preceding block in Series 0i-C]
Since the tool moves until it reaches the end point of the block (P_3 in the figure), overcutting may result.
[When single block stop does not occur in the preceding block in Series 0i-C]
The tool stops immediately after executing the block (P_2 in the figure).
[In the case of Series 0i-D]
Since the tool stops at the start point of the block (P_1 in the figure), regardless of the single block state, overcutting can be prevented.</td></tr>
<tr><td>Single block stop in a block created internally for cutter compensation/tool nose radius compensation</td><td>- Not available.</td><td>- Depends on bit 0 (SBK) of parameter No. 5000.

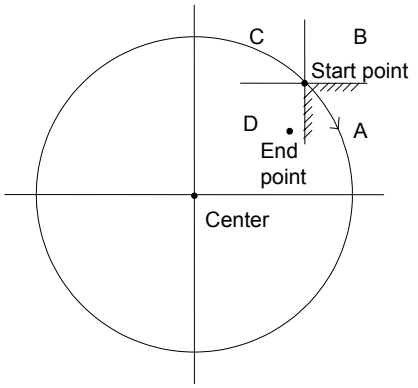
<u>Bit 0 (SBK) of parameter No. 5000</u>
In a block created internally for cutter compensation/tool nose radius compensation, single block stop is:
0: Not performed.
1: Performed.
This parameter is used to check a program including cutter compensation/tool nose radius compensation.</td></tr>
</table>

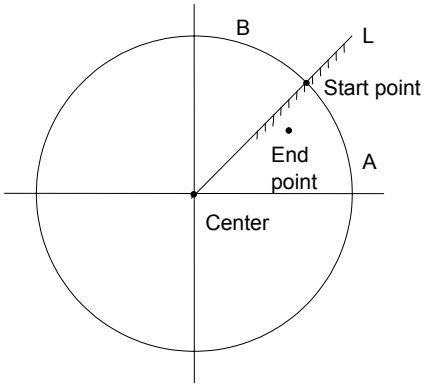
<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Setting to disable interference checking and to delete interfering vectors</td><td>- Set 1 in bit 0 (CNI) of parameter No. 5008.
In the example below, an interference check is made on the vectors inside V_1 and V_4, and the interfering vectors are deleted. As a result, the tool center path is from V_1 to V_4.</td><td>- Not available.
(Bit 0 (CNI) of parameter No. 5008 is not available.)
To prevent overcutting, the interference check avoidance function (bit 5 (CAV) of parameter No. 19607) is used.
In the example below, interference occurs between V_1 and V_4 and between V_2 and V_3. Therefore, vectors V_A and V_B are created. The tool center path is from V_A to V_B.</td></tr>
<tr><td colspan="3">[In the case of Series 0i-C]


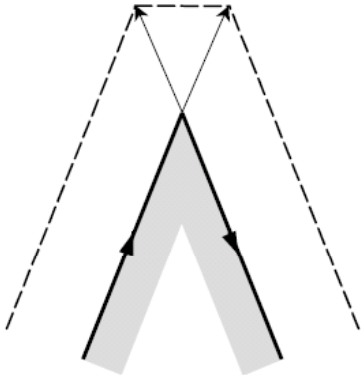
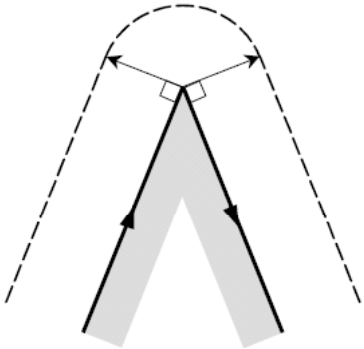
[In the case of Series 0i-D]

</td></tr>
</table>

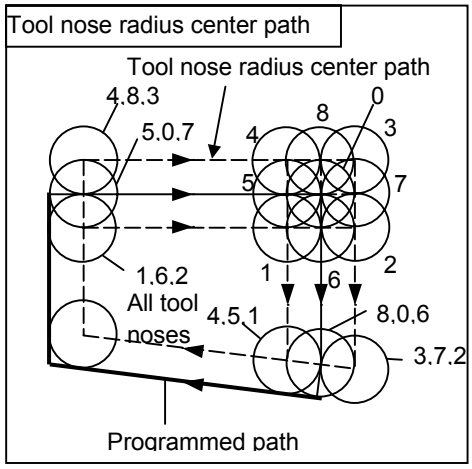
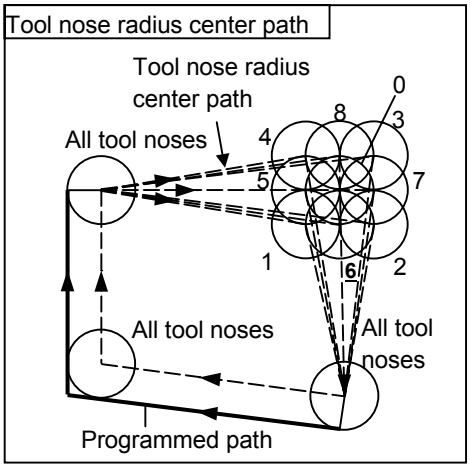
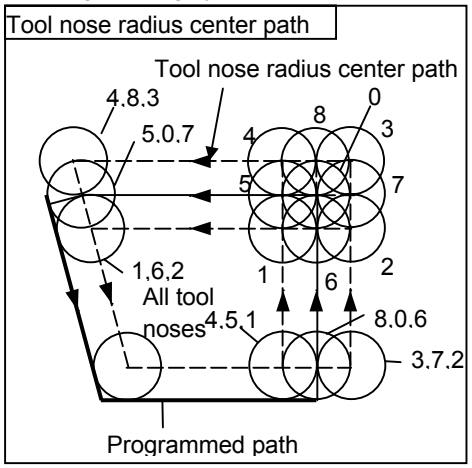
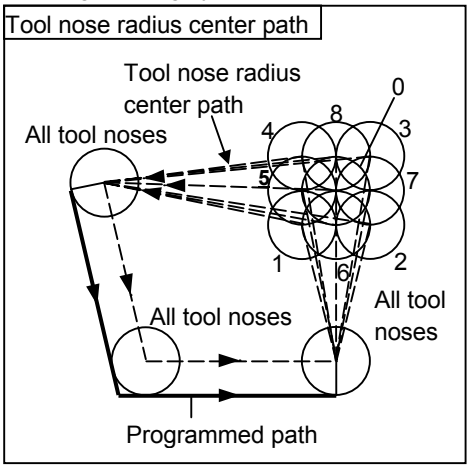
<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Number of blocks to be read in the cutter compensation/tool nose radius compensation mode</td><td>- Always 3 blocks</td><td>- The number can be set in parameter No. 19625. The specifiable range is 3 to 8 blocks.
If the parameter is not set (0 is set), the same number as Series 0i-C (3 blocks) is assumed.</td></tr>
<tr><td>When circular interpolation is specified that causes the center to coincide with the start or end point during the cutter compensation/tool nose radius compensation mode</td><td>- Alarm PS0038 is issued, and the tool stops at the end point of the block preceding the circular interpolation block.</td><td>- Alarm PS0041 is issued, and the tool stops at the start point of the block preceding the circular interpolation block.</td></tr>
<tr><td rowspan="2">Behavior when automatic reference position return is specified during the cutter compensation/tool nose radius compensation mode</td><td>- Depends on bit 2 (CCN) of parameter No. 5003.</td><td>- Bit 2 (CCN) of parameter No. 5003 is not available. The tool always behaves as when CCN is set to 1.</td></tr>
<tr><td colspan="2">[When CCN = 0]
The offset vector is canceled when the tool moves to the middle point.
Also, the start-up operation is performed from the reference position.


[When CCN = 1 or for Series 0i-D]
The offset vector is not canceled when the tool moves to the middle point; it is canceled when the tool moves to the reference position.
Also, the tool moves from the reference position to the next intersection point.

</td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Travel distance judgment method for circular interpolation in cutter compensation/tool nose radius compensation</td><td>- Depends on bit 5 (QCR) of parameter No. 5008.</td><td>- Bit 5 (QCR) of parameter No. 5008 is not available. The tool always behaves as when QCR is set to 1.</td></tr>
<tr><td>[When QCR = 0]


If the end point is on side A when viewed from the start point, the travel distance is small. If it is on side B, C, or D, the tool has traveled almost one round.</td><td>[When QCR = 1 or for Series 0i-D]


If the end point is on side A of line L connecting the start point and center, the travel distance is small. If it is on side B, the tool has traveled almost one round.</td></tr>
<tr><td rowspan="2">Compensation vector connection method when the tool travels around an external corner during the cutter compensation/tool nose radius compensation mode</td><td>- Connected by linear interpolation.</td><td>- Depends on bit 2 (CCC) of parameter No. 19607.</td></tr>
<tr><td>[When CCC = 0 or for Series 0i-C]
Connect vectors by linear interpolation
</td><td>[When CCC = 1]
Connect vectors by circular interpolation
</td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Virtual tool tip direction and plane selection</td><td>- Virtual tool tip directions 1 to 8 can be used only for the G18 (Z-X) plane. When the virtual tool tip direction is 0 or 9, compensation can be performed for the G17 and G19 planes as well.</td><td>- All virtual tool tip directions can be used for the G17, G18, and G19 planes.</td></tr>
<tr><td rowspan="4">Tool nose radius center path for tool nose radius compensation in a canned cycle (G90 or G94)</td><td>- [Outer surface turning/boring cycle (G90)]

</td><td>- [Outer surface turning/boring cycle (G90)]

</td></tr>
<tr><td>- [Edge cutting cycle (G94)]

</td><td>- [Edge cutting cycle (G94)]

</td></tr>
<tr><td colspan="2">* Numbers 0 to 8 in the figure are virtual tool tip numbers.</td></tr>
<tr><td>Start-up/cancellation type of tool nose radius compensation</td><td>- The start-up/cancellation type cannot be set.</td><td>- Depends on bit 0 (SUP) and bit 1 (SUV) of parameter No. 5003. When SUV and SUP are respectively set to 0 and 1 (type B), the processing is the same as Series 0i-C.</td></tr>
</table>

B.45.2 Differences in Diagnosis Display

None.

B.46 CANNED CYCLE FOR DRILLING

B.46.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
M05 output in a tapping cycle	- Make a selection using bit 6 (M5T) of parameter No. 5101. **Bit 6 (M5T) of parameter No. 5101** When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G74 with the M series, or G84/G88 with the T series): 0: M05 is not output before output of M04 or M03. 1: M05 is output before output of M04 or M03.	- Make a selection using bit 3 (M5T) of parameter No. 5105. **Bit 3 (M5T) of parameter No. 5105** When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G74 with the M series, or G84/G88 with the T series): 0: M05 is output before output of M04 or M03. 1: M05 is not output before output of M04 or M03. **NOTE** This parameter corresponds to bit 6 (M5T) of parameter No. 5101 of Series 0*i*-C. With the T series, the logic of the values 0 and 1 is opposite from that of Series 0*i*-C.
Behavior when K0 is specified for the number of repetitions K	- Make a selection using bit 5 (K0E) of parameter No. 5102. **Bit 5 (K0E) of parameter No. 5102** When K0 is specified in a drilling canned cycle (G80 to G89): 0: One drilling operation is performed. 1: Drilling operation is not performed, and only drilling data is stored.	- Make a selection using bit 4 (K0D) of parameter No. 5105 for both T series and M series. **Bit 4 (K0D) of parameter No. 5105** When K0 is specified in a drilling canned cycle (G80 to G89): 0: Drilling operation is not performed, and only drilling data is stored. 1: One drilling operation is performed. **NOTE** With the T series, the logic of the values 0 and 1 is opposite from that of bit 5 (K0E) of parameter No. 5102 of Series 0*i*-C.
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **Bit 1 (NRF) of parameter No. 3700** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.

Function	Series 0*i*-C	Series 0*i*-D
Retraction in a boring cycle (G85, G89)	- Select the retraction operation using bit 1 (BCR) of parameter No. 5104. **Bit 1 (BCR) of parameter No. 5104** The retraction operation in a boring cycle is performed: at 0: Cutting feedrate In this case, the cutting feedrate of the retraction operation can be multiplied by the override value set in parameter No. 5121. The override value range is 100% to 2000%. 1: Rapid traverse rate In this case, rapid traverse override is also enabled.	- Bit 1 (BCR) of parameter No. 5104 is not available. The retraction operation is always performed at the cutting feedrate. In this case, the cutting feedrate of the retraction operation can be multiplied by the override value set in parameter No. 5149. The override value range is 1% to 2000%.
Clearance value in a peck drilling cycle	- Set the value in parameter No. 5114.	- Set the value in parameter No. 5115.
Drilling axis in the Series 10/11 format	- Y axis cannot be used as a drilling axis. P/S alarm No. 028 is issued.	- Y axis can be used as a drilling axis.

B.46.2 Differences in Diagnosis Display

None.

B.47 CANNED CYCLE /MULTIPLE REPETITIVE CANNED CYCLE

B.47.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Machining plane	- The plane on which the canned cycle is performed is always the ZX plane.	- The plane on which the canned cycle ca be selected arbitrarily (including a parallel axis). Note that, with G code system A, an axis whose name is U, V, or W cannot be set as a parallel axis.
Address R setting unit (Address I, J, or K for the Series 10/11 format)	- The setting unit common to all axes is used.	- The setting unit applies to a different axis depending on the machining plane and the command. Second axis of the axes comprising the machining plane for G90 and G92 First axis of the axes comprising the machining plane for G94
Application of tool nose radius compensation	- Refer to Section 4.1.5, "CANNED CYCLE AND TOOL NOSE RADIUS COMPENSATION" in "USER'S MANUAL (T SERIES)" (B-64304EN-1). The differences in specifications are detailed.	
Inch threading by address E (Series 10/11 format)	- Threading is performed as the lead threading command of address F.	- Inch threading is performed.
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **Bit 1 (NRF) of parameter No. 3700** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.

B.47.2 Differences in Diagnosis Display

None.

B.48 CANNED GRINDING CYCLE

B.48.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Grinding axis specification	- The grinding axis is always the Z axis.	- Set the grinding axes for the individual canned grinding cycles in parameter Nos. 5176 to 5179. If the same axis number as the cutting axis is specified in any of these parameters, or if a canned grinding cycle is executed when 0 is set, alarm PS0456 is issued.
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **Bit 1 (NRF) of parameter No. 3700** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.
Exclusive control against the multiple respective canned cycle (standard function)	- When the grinding canned cycle option is specified, the multiple respective canned cycle (standard function) cannot be used.	- When the grinding canned cycle option is specified, select whether to use the multiple respective canned cycle (standard function) or grinding canned cycle, by using bit 0 (GFX) of parameter No. 5106. **Bit 0 (GFX) of parameter No. 5106** When the grinding canned cycle option is specified, the G71, G72, G73, and G74 commands are intended for: 0: Multiple respective canned cycle. 1: Grinding canned cycle.

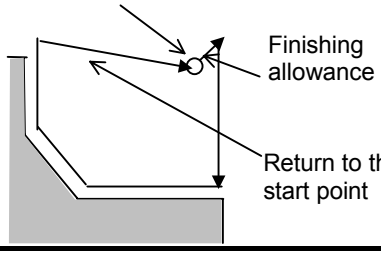
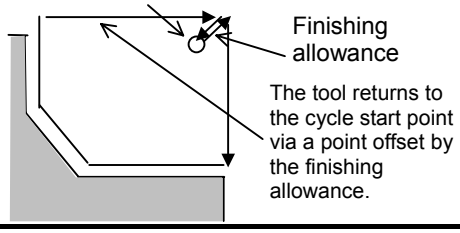
B.48.2 Differences in Diagnosis Display

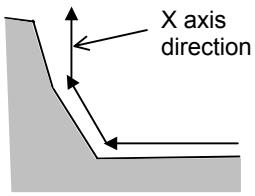
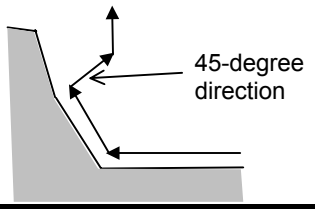
None.

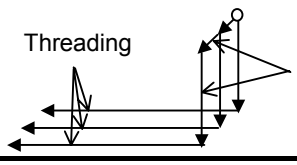
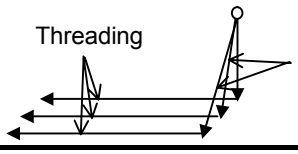
B.49 MULTIPLE RESPECTIVE CANNED CYCLE FOR TURNING

B.49.1 Differences in Specifications

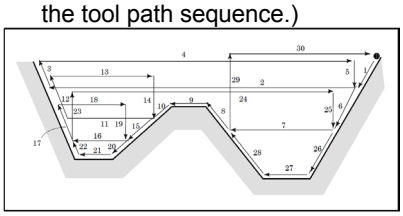
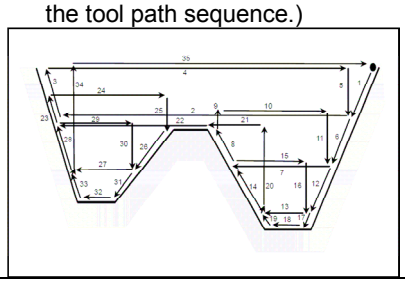
Differences common to the Series 0 standard format and Series 10/11 format

Function	Series 0*i*-C	Series 0*i*-D
Specifiable plane	- The cycle can be specified for a Z-X plane, with the X axis set as the first axis and the Z axis set as the second axis.	- The cycle can be specified for an arbitrary plane selected with the basic three axes and their parallel axes.
Specification for a plane including a parallel axis	- Not allowed.	- For G code system A, the cycle can be specified when the name of the parallel axis is other than U, V, or W. (To use U, V, or W as an axis name is not allowed for G code system A.)
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **Bit 1 (NRF) of parameter No. 3700** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.
Cycle start point return path when the finishing allowance is specified in G71 or G72	- The tool returns directly to the cycle start point. Cycle start point  	- The tool returns to the cycle start point via a point offset by the finishing allowance. Cycle start point 

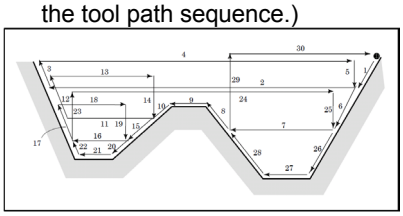
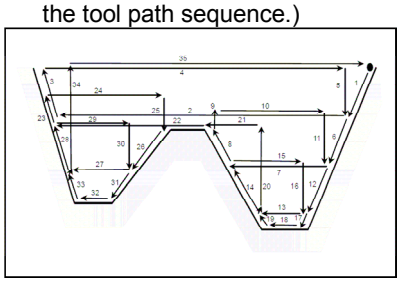
Function	Series 0*i*-C	Series 0*i*-D
Monotonous increase/decrease check in G71/G72 type I (multiple respective canned cycle for turning)	- Depends on bit 1 (MRC) of parameter No. 5102. **Bit 1 (MRC) of parameter No. 5102** When any target figure other than monotonous increase or decrease is specified in a multiple respective canned cycle for turning (G71 or G72): 0: An alarm is not issued. 1: Alarm PS0064 is issued.	- Bit 1 (MRC) of parameter No. 5102 is not available. If monotonous increase or decrease is not specified for the first axis direction of the plane, alarm PS0064 is issued. If monotonous increase or decrease is not specified for the second axis direction of the plane, alarm PS0329 is issued. Note that, by setting a permissible amount in parameter Nos. 5145 and 5146, it is possible to prevent the alarm from occurring, even if the monotonous increase/decrease condition is not met, as long as the permissible amount is not exceeded.
Monotonous increase/decrease check in G71/G72 type II (multiple respective canned cycle for turning II)	- Not checked. Bit 1 (MRC) of parameter No. 5102 does not take effect for multiple respective canned cycle for turning II (type II).	- Always checked. If monotonous increase or decrease is not specified for the first axis direction of the plane, alarm PS0064 is issued. Note that, by setting a permissible amount in parameter No. 5145, it is possible to prevent the alarm from occurring, even if the monotonous increase/decrease condition is not met, as long as the permissible amount is not exceeded.
Roughing after start point return by G71 or G72	- Not performed.	- [Multiple respective canned cycle for turning I (type I)] Depends on bit 1 (RF1) of parameter No. 5105. [Multiple respective canned cycle for turning II (type II)] Depends on bit 2 (RF2) of parameter No. 5105.
	Bit 1 (RF1) of parameter No. 5105 In the multiple repetitive canned cycle (T series) (G71/G72) of type I, roughing is: 0: Performed. 1: Not performed.	**Bit 2 (RF2) of parameter No. 5105** In the multiple repetitive canned cycle (T series) (G71/G72) of type II, roughing is: 0: Performed. 1: Not performed.
Retraction operation at the bottom of a hole in G71/G72 type II (multiple respective canned cycle for turning II)	- The tool retracts in the X axis direction after chamfering.  	- After chamfering, the tool first retracts in the 45-degree direction and then in the second axis direction of the plane. 

Function	Series 0*i*-C	Series 0*i*-D
G70 to G76 commands during the tool nose radius compensation mode	- [G70 command] Tool nose radius compensation is performed. [G71 to G73 commands] While tool nose radius compensation is not performed, it is possible to apply tool nose radius compensation partially by setting bit 4 (RFC) of parameter No. 5102. **Bit 4 (RFC) of parameter No. 5102** For a G71 or G72 semi-finished shape or a G73 cutting pattern, tool nose radius compensation is: 0: Not performed. 1: Performed. [G74 to G76 commands] Tool nose radius compensation is not performed.	- Bit 4 (RFC) of parameter No. 5102 is not available. [G70 to G73 commands] Tool nose radius compensation is performed. [G74 to G76 commands] Tool nose radius compensation is not performed.
Positioning in G70 to G76 cycle operations	- Non-linear type positioning is always used, regardless of the setting of bit 1 (LRP) of parameter No. 1401.	- [Start point return by G70] Non-linear type positioning is always used. [Other positioning operations] Depends on bit 1 (LRP) of parameter No. 1401.
T code specified in the same block as G74 or G75	- Invalid	- Valid
Chamfering and corner R commands and direct drawing dimension programming command for a target figure program	- Cannot be specified.	- Can be specified. Note that the last block of the target figure program must not be in the middle of the chamfering, corner R, or direct drawing dimension programming command.
Approach to the threading start point in G76	- Approach by two cycles 	- Approach by one cycle 

Differences regarding the Series 0 standard format

Function	Series 0*i*-C	Series 0*i*-D
Pocketing path in G71/G72 type II (multiple respective canned cycle for turning II)	- The tool moves from one pocket to another for each cut. (The numbers in the figure represent the tool path sequence.) 	- The tool completes one pocketing process before proceeding to cut the next pocket. (The numbers in the figure represent the tool path sequence.) 
Limitation on the number of pockets in G71/G72 type II (multiple respective canned cycle for turning II)	- Up to 10 pockets can be specified. Specifying 11 or more pockets causes alarm PS0068.	- Not limited.
Number of divisions in G73	- The number of divisions is also 2 for the R1 command. For R2 and subsequent commands, the number of divisions specified by R applies.	- The number of divisions specified by R applies.

Differences regarding the Series 10/11 format

Function	Series 0*i*-C	Series 0*i*-D
Pocketing path in G71/G72 type II (multiple respective canned cycle for turning II)	- Depends on bit 2 (P15) of parameter No. 5103. [When P15 = 0] The tool moves from one pocket to another for each cut. (The numbers in the figure represent the tool path sequence.)  [When P15 = 1] The tool completes one pocketing process before proceeding to cut the next pocket. (See the figure at right.)	- Bit 2 (P15) of parameter No. 5103 is not available. The tool completes one pocketing process before proceeding to cut the next pocket. (The numbers in the figure represent the tool path sequence.) 
Limitation on the number of pockets in G71/G72 type II (multiple respective canned cycle for turning II)	- Depends on bit 2 (P15) of parameter No. 5103. [When P15 = 0] Up to 10 pockets can be specified. Specifying 11 or more pockets causes alarm PS0068. [When P15 = 1] Not limited.	- Bit 2 (P15) of parameter No. 5103 is not available. Not limited.

Function	Series 0*i*-C	Series 0*i*-D
Specification of finishing allowance in G71/G72	- Not allowed. The finishing allowance is ignored if specified.	- Allowed.
Number of divisions in G73	- The number of divisions is also 2 for the D1 command. For D2 and subsequent commands, the number of divisions specified by D applies.	- The number of divisions specified by D applies.
Address E command in G76	- Threading is performed as the lead threading command of address F.	- Inch threading is performed.

B.49.2 Differences in Diagnosis Display

None.

B.50 CHAMFERING AND CORNER ROUNDING

B.50.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Chamfering and corner rounding commands for a plane other than the Z-X plane	- Not available. Alarm PS0212 is issued.	- Available. The commands can be specified for any plane, even one that includes a parallel axis.
Single block operation	- [Chamfering] Single block stop is not performed at the start point of an inserted chamfering block. [Corner rounding] Single block stop is performed at the start point of an inserted corner rounding block.	- [Common to chamfering and corner rounding] Whether to perform single block stop at the start point of an inserted block depends on bit 0 (SBC) of parameter No. 5105. **Bit 0 (SBC) of parameter No. 5105** In a drilling canned cycle, chamfer cycle/corner rounding (T series) or optional angle chamfering/corner rounding cycle (M series): 0: Single block stop is not performed. 1: Single block stop is performed.

B.50.2 Differences in Diagnosis Display

None.

B.51 DIRECT DRAWING DIMENSIONS PROGRAMMING

B.51.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Specification of the direct drawing dimension programming command for a plane other than the Z-X plane	- P/S alarm No. 212 is issued.	- No alarm is issued. The command can be specified for a plane other than the Z-X plane.
When two or more blocks not to be moved exist between consecutive commands that specify direct input of drawing dimensions	- No alarm is issued.	- Alarm PS0312 is issued.

B.51.2 Differences in Diagnosis Display

None.

INDEX

<Number>

2-PATH CONTROL FUNCTION 349

<A>

ADDRESSES AND SPECIFIABLE VALUE RANGE FOR Series 10/11 PROGRAM FORMAT 249
ADVANCED PREVIEW CONTROL 479
ARBITRARY ANGULAR AXIS CONTROL..... 487
AUTOMATIC TOOL OFFSET 453
AUTOMATIC TOOL OFFSET (G36, G37)..... 244
AXIS CONTROL FUNCTIONS 338
AXIS SYNCHRONOUS CONTROL 482

<B>

BALANCE CUT (G68, G69)..... 358
Boring Cycle (G85)..... 333
Boring Cycle (G89)..... 335

<C>

CANNED CYCLE 250
CANNED CYCLE (G90, G92, G94) 38
CANNED CYCLE /MULTIPLE REPETITIVE CANNED CYCLE 530
Canned Cycle and Tool Nose Radius Compensation..... 55,268
Canned Cycle Cancel (G80)..... 127
CANNED CYCLE FOR DRILLING..... 99,315,528
Canned Cycle for Drilling Cancel (G80) 113,337
CANNED GRINDING CYCLE..... 531
CANNED GRINDING CYCLE (FOR GRINDING MACHINE)..... 131
CHAMFERING AND CORNER R 145
CHAMFERING AND CORNER ROUNDING..... 537
Chuck and Tail Stock Barriers 385
CHUCK/TAIL STOCK BARRIER 509
CIRCULAR INTERPOLATION 455
COMMON MEMORY BETWEEN EACH PATH..... 352
COMPENSATION FUNCTION..... 161
CONSTANT LEAD THREADING (G32) 29
CONSTANT SURFACE SPEED CONTROL..... 467
CONTINUOUS THREADING..... 34
CORNER CIRCULAR INTERPOLATION (G39)..... 242
Counter Input of Offset value..... 379
Cs CONTOUR CONTROL..... 464
CUSTOM MACRO..... 474
CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION..... 522

<D>

DATA INPUT/OUTPUT 363
DATA SERVER FUNCTION 507
DATA TYPE..... 447
DEFINITION OF WARNING, CAUTION, AND NOTE s-2
DESCRIPTION OF PARAMETERS..... 396
DETAILS OF TOOL NOSE RADIUS COMPENSATION..... 185
DIFFERENCES FROM SERIES 0*i*-C 450
DIRECT DRAWING DIMENSION PROGRAMMING..... 155
DIRECT DRAWING DIMENSIONS PROGRAMMING..... 538
Direct Input of Tool Offset Value 374
Direct Input of Tool Offset Value Measured B 376
Direction of Imaginary Tool Nose 171
Drilling Cycle, Counter Boring (G82)..... 323
Drilling Cycle, Spot Drilling Cycle (G81) 321

<E>

End Face Peck Drilling Cycle (G74)..... 86,301
End Face Turning Cycle (G94) 50,262
EXTERNAL DATA INPUT 505
EXTERNAL SUBPROGRAM CALL (M198) 496
Extraction override..... 128

<F>

Face cutting cycle..... 50,262
Finishing Cycle (G70)..... 82,297
Front Boring Cycle (G85) / Side Boring Cycle (G89) .. 112
Front Drilling Cycle (G83)/Side Drilling Cycle (G87) . 103
FRONT FACE RIGID TAPPING CYCLE (G84) / SIDE FACE RIGID TAPPING CYCLE (G88)..... 116
Front Tapping Cycle (G84) / Side Tapping Cycle (G88) 106
FUNCTIONS TO SIMPLIFY PROGRAMMING 37

<G>

GENERAL 3,13
GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL 7
GENERAL WARNINGS AND CAUTIONS s-3

<H>

HELICAL INTERPOLATION 456
High-speed Peck Drilling Cycle (G83.1) 327
How to Use Canned Cycles 266
How to Use Canned Cycles (G90, G92, G94) 53

<I>

Imaginary Tool Nose 169
INPUT OF TOOL OFFSET VALUE MEASURED B 473
INPUT/OUTPUT ON EACH SCREEN 364
INPUT/OUTPUT ON THE ALL IO SCREEN 366
Inputting and Outputting Y-axis Offset Data 364,367
Inputting Y-axis offset data 364
Interference Check 228
Interference check alarm function 232
Interference check avoidance function 234
INTERPOLATION FUNCTION 20
INTERRUPTION TYPE CUSTOM MACRO 477

<L>

LOCAL COORDINATE SYSTEM 462

<M>

MACHINING CONDITION SELECTION FUNCTION 481
MANUAL ABSOLUTE ON AND OFF 503
MANUAL HANDLE FEED 489
MANUAL REFERENCE POSITION RETURN 459
MEMORY OPERATION USING SERIES 10/11 FORMAT 248
MEMORY PROTECTION SIGNAL FOR CNC PARAMETER 504
MIRROR IMAGE FOR DOUBLE TURRET (G68, G69) 153
Miscellaneous 476
MULTIPLE REPETITIVE CANNED CYCLE 272
MULTIPLE REPETITIVE CANNED CYCLE (G70-G76) 59
MULTIPLE RESPECTIVE CANNED CYCLE FOR TURNING 532
MULTIPLE THREADING 35
Multiple Threading Cycle (G76) 90,305
MULTI-SPINDLE CONTROL 465

<N>

NOTES ON READING THIS MANUAL 9
Notes on Tool Nose Radius Compensation 182
NOTES ON VARIOUS KINDS OF DATA 9

<O>

Offset 164
OFFSET 14
Offset Number 163
Offset Number and Offset Value 173
Operation to be performed if an interference is judged to occur 232
Oscillation Direct Constant-Size Grinding Cycle (G74) 142
Oscillation Grinding Cycle (G73) 139
Outer Diameter / Internal Diameter Drilling Cycle (G75) 88,303
Outer Diameter/Internal Diameter Cutting Cycle (G90) 39,251
Outputting Y-axis Offset Data 365
Override during Rigid Tapping 128
Override signal 130
Overview 185
OVERVIEW 350
OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42) 168

<P>

PARAMETERS 395
PATH INTERFERENCE CHECK (2-PATH CONTROL) 513
Pattern Repeating (G73) 79,294
Peck Drilling Cycle (G83) 325
Peck Rigid Tapping Cycle (G84 or G88) 122
PMC AXIS CONTROL 491
POLAR COORDINATE INTERPOLATION 511
POLAR COORDINATE INTERPOLATION (G12.1, G13.1) 21
POLYGON TURNING (G50.2, G51.2) 339
POWER MATE CNC MANAGER 508
Precautions to be Taken by Operator 114,337
PREPARATORY FUNCTION (G FUNCTION) 15

Prevention of Overcutting Due to Tool Nose Radius Compensation....224
PROGRAMMABLE PARAMETER INPUT (G10)....478

<R>

RESET AND REWIND....502
Restrictions on Canned Cycles....57,270
Restrictions on Multiple Repetitive Canned Cycle....313
Restrictions on Multiple Repetitive Canned Cycle (G70-G76)....97
RIGID TAPPING....115
RUN HOUR AND PARTS COUNT DISPLAY....488

<S>

SAFETY PRECAUTIONS....s-1
SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION....501
SCREENS DISPLAYED BY FUNCTION KEY ....369
SEQUENCE NUMBER SEARCH....497
SERIAL/ANALOG SPINDLE CONTROL....466
SETTING AND DISPLAYING DATA....368
Setting and Displaying the Tool Offset Value....370
Setting the Workpiece Coordinate System Shift Value....380
Setting the Y-Axis Offset....382
SETTING UNIT....452
SKIP FUNCTION....457
SPINDLE CONTROL BETWEEN EACH PATH....354
SPINDLE POSITIONING....468
STANDARD PARAMETER SETTING TABLES....448
Stock Removal in Facing (G72)....74,289
Stock Removal in Turning (G71)....60,273
STORED PITCH ERROR COMPENSATION....500
STORED STROKE CHECK....498
Straight cutting cycle....39,251
Straight threading cycle....43,255
SUBPROGRAM CALLING....249
SUPERIMPOSED CONTROL (2-PATH CONTROL))....519
SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (2-PATH CONTROL)....514
SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6)....345
SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL....355

<T>

T Code for Tool Offset....163
Taper cutting cycle....41,51,253,264
Taper threading cycle....47,259
Tapping Cycle (G84)....329
Tapping Cycle (G84.2)....331
Threading Cycle (G92)....43,255
THREADING CYCLE RETRACT (CANNED CUTTING CYCLE/MULTIPLE REPETITIVE CANNED CUTTING CYCLE)....510
TOOL COMPENSATION MEMORY....471
TOOL FUNCTIONS....470
Tool Geometry Offset and Tool Wear Offset....162
Tool Movement in Offset Mode....196
Tool Movement in Offset Mode Cancel....217
Tool Movement in Start-up....190
Tool Nose Radius Compensation for Input from MDI....240
TOOL OFFSET....162
Tool Selection....163
Traverse Direct Constant-Size Grinding Cycle (G72)....136
Traverse Grinding Cycle (G71)....133

<V>

VARIABLE LEAD THREADING (G34)....33

<W>

WAITING FUNCTION FOR PATHS....351
WARNINGS AND CAUTIONS RELATED TO HANDLING....s-9
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING....s-6
WARNINGS RELATED TO DAILY MAINTENANCE....s-12
WORKPIECE COORDINATE SYSTEM....461
Workpiece Position and Move Command....175

<Y>

Y Axis Offset....167
Y AXIS OFFSET....521
Y axis offset (arbitrary axes)....167

Revision Record

FANUC Series 0*i*-MODEL D/Series 0*i* Mate-MODEL D USER'S MANUAL (For Lathe System) (B-64304EN-1)

Edition	Date	Contents	Edition	Date	Contents
01	Jun., 2008	———————————————			