FANUC Series 0*i*-MODEL F Plus

For Machining Center System

OPERATOR'S MANUAL

B-64694EN-2/01

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

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

The products in this manual are manufactured under strict quality control. However, when a serious accident or loss is predicted due to a failure of the product, pay careful attention to safety.

In this manual, we endeavor to include all pertinent matters.
There are, however, a very large number of operations that must not or cannot be performed, and if the manual contained them all, it would be enormous in volume.
It is, therefore, requested to assume that any operations that are not explicitly described as being possible are "not possible".

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.

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**
Used if a danger resulting in the death or serious injury of the user is expected to occur if he or she fails to observe the approved procedure.

⚠ **CAUTION**
Used if a danger resulting in the minor or moderate injury of the user or equipment damage is expected to occur if he or she fails to observe the approved procedure.

NOTE
Used if a supplementary explanation not related to any of DANGER, WARNING, and CAUTION is to be indicated.

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

GENERAL WARNINGS AND CAUTIONS

⚠ **WARNING**

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

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

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

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

5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

⚠ CAUTION

1 Immediately after switching on the power, do not touch any of the keys on the MDI unit until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI unit are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

2 The OPERATOR'S MANUAL and programming manual supplied with a CNC unit provide an overall description of the machine's functions. Note that the-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.

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

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

NOTE

Programs, parameters, and macro variables are stored in non-volatile 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 non-volatile 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.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a personal computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

⚠ **WARNING**

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

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

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

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

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.

⚠ **WARNING**

7 **Tool post interference check**
A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user. After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8 **Same address command in same block**
The G code or M code including the same address cannot be commanded on the same block. If you use the same address, it may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user. Command on separate block.（About address P, refer to the appendix “List of functions include address P in the program command”）

⚠ **CAUTION**

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

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

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

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

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

WARNINGS AND CAUTIONS RELATED TO HANDLING

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

⚠ **WARNING**

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

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

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

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

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

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

⚠ **WARNING**

7 **Software operator's panel and menu switches**

Using the software operator's panel and menu switches, in combination with the MDI unit, 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 unit 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 unit problem. So, when the motors must be stopped, use the emergency stop button instead of the RESET key to ensure security.

⚠ **CAUTION**

1 **Manual intervention**

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

2 **Feed hold, override, and single block**

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

3 **Dry run**

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

4 **Program editing**

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

5 **PS alarm**

When a PS alarm is occurred during executing a blolck, the axis movement of the block is continued to the end of block. After finishing the executing the block, the axis movement stops.

WARNINGS RELATED TO DAILY MAINTENANCE

⚠ **WARNING**

1 **Memory backup battery replacement**

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

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

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

NOTE

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

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

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

Refer to the Section “Method of replacing battery” in the OPERATOR’S MANUAL (Common to Lathe System/Machining Center System) 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

I. GENERAL

1 GENERAL 3
1.1 NOTES ON READING THIS MANUAL 6
1.2 NOTES ON VARIOUS KINDS OF DATA 6

II. PROGRAMMING

1 GENERAL 9
1.1 TOOL FIGURE AND TOOL MOTION BY PROGRAM 9

2 PREPARATORY FUNCTION (G FUNCTION) 10

3 INTERPOLATION FUNCTION 14
3.1 THREADING (G33) 14

4 COORDINATE VALUE AND DIMENSION 16
4.1 POLAR COORDINATE COMMAND (G15, G16) 16

5 FUNCTIONS TO SIMPLIFY PROGRAMMING 21
5.1 CANNED CYCLE FOR DRILLING 21
5.1.1 High-Speed Peck Drilling Cycle (G73) 26
5.1.2 Left-Handed Tapping Cycle (G74) 28
5.1.3 Fine Boring Cycle (G76) 30
5.1.4 Drilling Cycle, Spot Drilling (G81) 32
5.1.5 Drilling Cycle Counter Boring Cycle (G82) 33
5.1.6 Peck Drilling Cycle (G83) 35
5.1.7 Small-Hole Peck Drilling Cycle (G83) 37
5.1.8 Tapping Cycle (G84) 41
5.1.9 Boring Cycle (G85) 43
5.1.10 Boring Cycle (G86) 44
5.1.11 Back Boring Cycle (G87) 46
5.1.12 Boring Cycle (G88) 48
5.1.13 Boring Cycle (G89) 50
5.1.14 Canned Cycle Cancel for Drilling (G80) 51
5.1.15 Example for Using Canned Cycles for Drilling 52
5.1.16 Reducing of Waiting Time of Spindle Speed Arrival in the Canned Cycle for Drilling 54
5.2 CANNED CYCLE OVERLAP FOR DRILLING 55
5.3 RIGID TAPPING 60
5.3.1 Rigid Tapping (G84) 61
5.3.2 Left-Handed Rigid Tapping Cycle (G74) 65
5.3.3 Peck Rigid Tapping Cycle (G84 or G74) 69
5.3.4 Canned Cycle Cancel (G80) 73
5.3.5 Override during Rigid Tapping 73

5.3.5.1 Extraction override ... 73
5.3.5.2 Override signal ... 75
5.4 OPTIONAL CHAMFERING AND CORNER R ... 76
5.5 INDEX TABLE INDEXING FUNCTION ... 79
5.6 IN-FEED CONTROL (FOR GRINDING MACHINE) ... 81
5.7 CANNED GRINDING CYCLE (FOR GRINDING MACHINE) ... 84
5.7.1 Plunge Grinding Cycle (G75) ... 86
5.7.2 Direct Constant-Dimension Plunge Grinding Cycle (G77) ... 89
5.7.3 Continuous-feed Surface Grinding Cycle (G78) ... 92
5.7.4 Intermittent-feed Surface Grinding Cycle (G79) ... 95

6 COMPENSATION FUNCTION ... 97
6.1 TOOL LENGTH COMPENSATION SHIFT TYPES ... 97
6.2 AUTOMATIC TOOL LENGTH MEASUREMENT (G37) ... 104
6.3 TOOL OFFSET (G45 TO G48) ... 107
6.4 OVERVIEW OF CUTTER COMPENSATION (G40-G42) ... 112
6.5 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42) ... 117
6.5.1 Imaginary Tool Nose ... 117
6.5.2 Direction of Imaginary Tool Nose ... 119
6.5.3 Offset Number and Offset Value ... 120
6.5.4 Workpiece Position and Move Command ... 121
6.5.5 Notes on Tool Nose Radius Compensation ... 127
6.6 DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION ... 128
6.6.1 Overview ... 128
6.6.2 Tool Movement in Start-up ... 132
6.6.3 Tool Movement in Offset Mode ... 137
6.6.4 Tool Movement in Offset Mode Cancel ... 155
6.6.5 Prevention of Overcutting Due to Tool Radius / Tool Nose Radius Compensation ... 162
6.6.6 Interference Check ... 165
6.6.6.1 Operation to be performed if an interference is judged to occur ... 168
6.6.6.2 Interference check alarm function ... 169
6.6.6.3 Interference check avoidance function ... 171
6.6.7 Tool Radius / Tool Nose Radius Compensation for Input from MDI ... 176
6.7 VECTOR RETENTION (G38) ... 178
6.8 CORNER CIRCULAR INTERPOLATION (G39) ... 179
6.9 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10) ... 181
6.10 COORDINATE SYSTEM ROTATION (G68, G69) ... 183

7 MEMORY OPERATION USING Series 15 PROGRAM FORMAT ... 190

8 HIGH-SPEED CUTTING FUNCTIONS ... 192
8.1 SELECT SETTING PATTERN OF FINE SURFACE SETTING BY PROGRAM COMMAND ... 192

III. OPERATION

1 SETTING AND DISPLAYING DATA ... 195
1.1 SCREENS DISPLAYED BY FUNCTION KEY ... 195
1.1.1 Setting and Displaying the Tool Compensation Value ... 195

1.1.2 Tool Length Measurement 199
1.1.3 Fine surface setting 202

APPENDIX

A LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND 207
A.1 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE 207
A.2 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE 210

I. GENERAL

1 GENERAL

This manual consists of the following parts:

About this manual

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

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, 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
Describes supplementary materials.

NOTE
1 This manual describes the functions that can operate in the machining center system path control type. For other functions not specific to the lathe system, refer to the Operator's Manual (Common to Lathe System/Machining Center System) (B-64694EN).
2 Some functions described in this manual may not be applied to some products. For detail, refer to the DESCRIPTIONS manual (B-64692EN).
3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-64700EN). Parameters are used to set functions and operating conditions of a CNC machine tool, and frequently-used values in advance. Usually, the machine tool builder factory-sets parameters so that the user can use the machine tool easily.
4 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

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

Model name	Abbreviation		
FANUC Series 0*i*-MF Plus	0*i*-MF Plus	Series 0*i*-F Plus	Series 0*i*

NOTE
1 For an explanatory purpose, the following descriptions may be used according to the CNC model :
- 0*i*-MF Plus : Machining center system (M series)

NOTE
2 Some functions described in this manual may not be applied to some products. For details, refer to the Descriptions (B-64692EN).

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*-F Plus

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

Table 1 (a) Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64692EN	
CONNECTION MANUAL (HARDWARE)	B-64693EN	
CONNECTION MANUAL (FUNCTION)	B-64693EN-1	
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64694EN	
OPERATOR'S MANUAL (For Lathe System)	B-64694EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64694EN-2	*
MAINTENANCE MANUAL	B-64695EN	
PARAMETER MANUAL	B-64700EN	
Programming		
Macro Executor PROGRAMMING MANUAL	B-63943EN-2	
Macro Compiler PROGRAMMING MANUAL	B-66263EN	
C Language Executor PROGRAMMING MANUAL	B-63943EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64513EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-63993EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN	
DeviceNet Board CONNECTION MANUAL	B-64043EN	
FL-net Board CONNECTION MANUAL	B-64163EN	
CC-Link Board CONNECTION MANUAL	B-64463EN	
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	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64483EN-2	

Related manuals of SERVO MOTOR αi/βi series

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

Table 1 (b) Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR αi-B series FANUC AC SERVO MOTOR αi series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR αi-B / βi-B series DESCRIPTIONS	B-65452EN
FANUC AC SERVO MOTOR βi-B series FANUC AC SERVO MOTOR βi series DESCRIPTIONS	B-65302EN
FANUC SERVO AMPLIFIER αi series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER βi series DESCRIPTIONS	B-65322EN
FANUC AC SERVO MOTOR αi series FANUC AC SPINDLE MOTOR αi series FANUC SERVO AMPLIFIER αi series MAINTENANCE MANUAL	B-65285EN
FANUC AC SERVO MOTOR βi series FANUC AC SPINDLE MOTOR βi series FANUC SERVO AMPLIFIER βi series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR αi series FANUC AC SERVO MOTOR βi series FANUC LINEAR MOTOR LiS series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR DiS series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR αi/βi series, BUILT-IN SPINDLE MOTOR Bi series PARAMETER MANUAL	B-65280EN

The above servo motors and the corresponding spindles can be connected to the CNC covered in this manual.

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 NOTES ON READING THIS MANUAL

⚠ **CAUTION**

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

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

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

1.2 NOTES ON VARIOUS KINDS OF DATA

⚠ **CAUTION**

Machining programs, parameters, offset data, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a personal computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.

II. PROGRAMMING

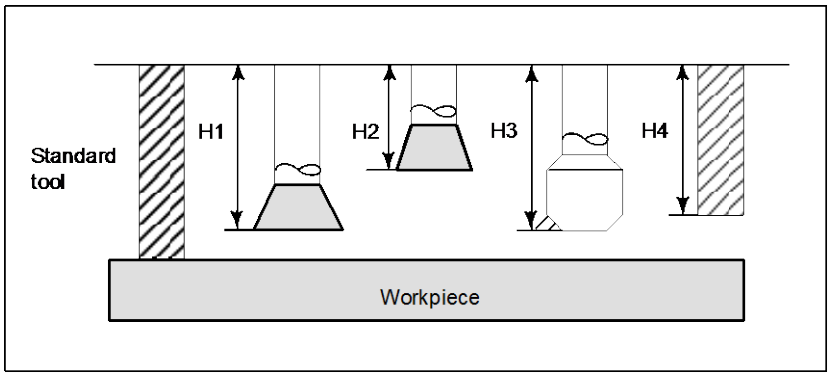
1 GENERAL

1.1 TOOL FIGURE AND TOOL MOTION BY PROGRAM

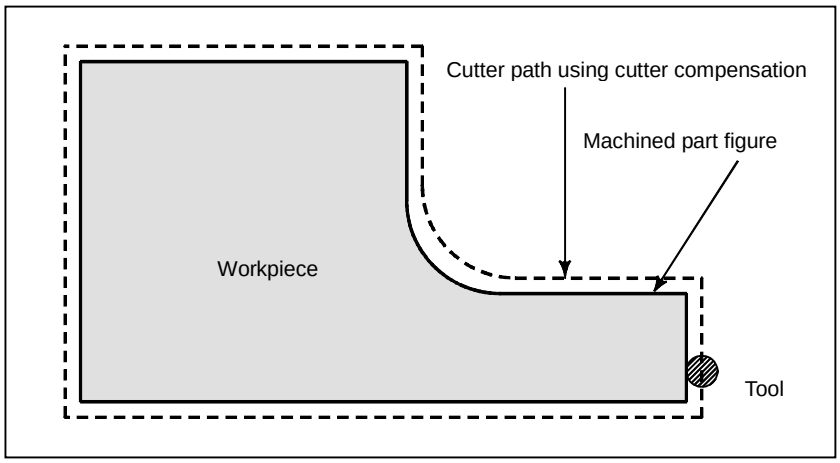
Explanation

- Machining using the end of cutter - Tool length compensation function

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 Chapter, “Setting and Displaying Data” in OPERATOR’S MANUAL (Common to Lathe System / Machining Center System)(B-64694EN)), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation (See Section, “Tool Length Compensation” in OPERATOR’S MANUAL (Common to Lathe System / Machining Center System) (B-64694EN)).



- Machining using the side of cutter - Cutter compensation function



Because a cutter has a radius, the center of the cutter path goes around the workpiece with the cutter radius deviated.
If radius of cutters are stored in the CNC (See Chapter, “Setting and Displaying Data” in OPERATOR’S MANUAL (Common to Lathe System / Machining Center System) (B-64694EN)), the tool can be moved by cutter radius apart from the machining part figure. This function is called cutter compensation (See Chapter, “Compensation Function”).

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_ ;
    :
```

Explanation

1. When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◪ as indicated in Table 2 .
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by bit 7 (G23) of parameter No. 3402. However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting bit 0 (G01) of parameter No. 3402.
   (5) The user can select G90 or G91 by setting bit 3 (G91) of parameter No. 3402.
   When G code system B or C is used in the lathe system, setting bit 3 (G91) of parameter No. 3402 determines which code, either G90 or G91, is effective.
   (6) In the machining center system, the user can select G17, G18, or G19 by setting bits 1 (G18) and 2 (G19) of parameter No. 3402.
2. G codes of 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, “IMPROPER G-CODE” occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle for drilling.
6. G codes are indicated by group.
7. The group of G60 is switched according to the setting of the bit 0 (MDL) of parameter No. 5431. (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

Table 2 G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G04	00	Dwell	
G04.1		G code preventing buffering	
G05		AI contour control (high-precision contour control compatible command)	
G05.1		AI contour control	
G05.4		HRV3 on/off	
G07.1	00	Cylindrical interpolation	
G08		AI contour control (advanced preview control compatible command)	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G11		Programmable data input mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20 (G70)	06	Input in inch	
G21 (G71)		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G25	19	Spindle speed fluctuation detection off	
G26		Spindle speed fluctuation detection on	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G28.2		In-position check disable reference position return	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.2		In-position check disable 2nd, 3rd, or 4th reference position return	
G31		Skip function	
G31.8		EGB-axis skip	
G33	01	Threading	
G37	00	Automatic tool length measurement	
G38		Tool radius/tool nose radius compensation : preserve vector	
G39		Tool radius/tool nose radius compensation : corner circular interpolation	
G40	07	Tool radius/tool nose radius compensation : cancel	
G41		Tool radius/tool nose radius compensation : left	
G42		Tool radius/tool nose radius compensation : right	
G40.1	18	Normal direction control cancel mode	
G41.1		Normal direction control on : left	
G42.1		Normal direction control on : right	
G43	08	Tool length compensation +	
G44		Tool length compensation -	
G43.7		Tool offset	
G45	00	Tool offset : increase	
G46		Tool offset : decrease	
G47		Tool offset : double increase	
G48		Tool offset : double decrease	
G49 (G49.1)	08	Tool length compensation cancel	

Table 2 G code list

G code	Group	Function
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G50.4	00	Cancel synchronous control
G50.5		Cancel composite control
G50.6		Cancel superimposed control
G51.4		Start synchronous control
G51.5		Start composite control
G51.6		Start superimposed control
G52	00	Local coordinate system setting
G53		Machine coordinate system setting
G53.1		Tool axis direction control
G53.2		Selecting a machine coordinate system with feedrate
G53.6		Tool center point retention type tool axis direction control
G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call A
G66.1		Macro modal call B
G67		Macro modal call A/B cancel
G68	16	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G69		Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G68.2		Tilted working plane indexing
G68.3		Tilted working plane indexing by tool axis direction
G68.4		Tilted working plane indexing (incremental multi-command)
G72.1	00	Figure copying (rotary copy)
G72.2		Figure copying (linear copy)
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G75	01	Plunge grinding cycle
G76	09	Fine boring cycle
G77	01	Plunge direct sizing/grinding cycle
G78		Continuous-feed surface grinding cycle
G79		Intermittent-feed surface grinding cycle
G80	09	Canned cycle cancel Electronic gear box : synchronization cancellation
G80.4	34	Electronic gear box: synchronization cancellation
G81.4		Electronic gear box: synchronization start
G80.5	24	Electronic gear box 2 pair: synchronization cancellation
G81.5		Electronic gear box 2 pair: synchronization start
G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start

Table 2 G code list

G code	Group	Function
G81.1	00	High precision oscillation
G82	09	Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS15 format)
G84.3		Left-handed rigid tapping cycle (FS15 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
◤ G90	03	Absolute programming
◤ G91		Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92		Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G93	05	Inverse time feed
◤ G94		Feed per minute
G95		Feed per revolution
G96	13	Constant surface speed control
◤ G97		Constant surface speed control cancel
G96.1	00	Spindle indexing execution (waiting for completion)
G96.2		Spindle indexing execution (not waiting for completion)
G96.3		Spindle indexing completion check
G96.4		SV speed control mode ON
◤ G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G107	00	Cylindrical interpolation
◤ G160	20	In-feed control cancel
G161		In-feed control

3 INTERPOLATION FUNCTION

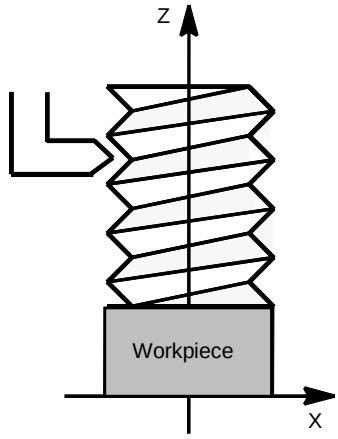
3.1 THREADING (G33)

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

Format

G33IP_ F_ ;

F : Long axis direction lead



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 1-turn 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.

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 3.1 (a) lists the ranges for specifying the thread lead.

Table 3.1 (a) Ranges of lead sizes that can be specified

	Least command increment	Command value range of the lead
Metric input	0.001 mm	F1 to F50000 (0.01 to 500.00mm)
	0.0001 mm	F1 to F50000 (0.01 to 500.00mm)
Inch input	0.0001 inch	F1 to F99999 (0.0001 to 9.9999inch)
	0.00001 inch	F1 to F99999 (0.0001 to 9.9999inch)

NOTE

1 The threading spindle speed which executes threading is limited as follows :
0 < spindle speed ≤ (Maximum cutting feedrate of threading axis(per rev.))
/ (Thread lead (length per rev.))
Spindle speed : min^{-1}
Thread lead : mm or inch
Maximum cutting feedrate : mm/min or inch/min ; maximum command-specified feedrate for feed-per-minute mode or maximum feedrate that is determined based on mechanical restrictions including those related to motors, whichever is smaller

2 Cutting feedrate override is not applied to the converted feedrate in all machining process from rough cutting to finish cutting. The feedrate is fixed at 100%

3 The converted feedrate is limited by the upper feedrate specified.

4 Feed hold is disabled during threading. Pressing the feed hold key during threading causes the machine to stop at the end point of the next block after threading (that is, after the G33 mode is terminated)

5 The leads of a thread are generally incorrect, due to automatic acceleration and deceleration. Thus distance allowances must be made to the extent in the program. Refer “INCORRECT THREADED LENGTH” in the OPERATOR’S MANUAL (Common to Lathe System/Machining Center System) (B-64694EN).

Limitation

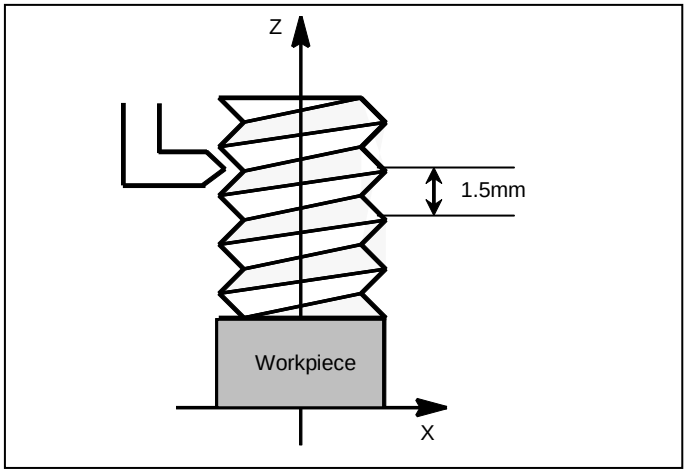
- Tool Retract and Recover

When the major axis for threading is specified as the retraction axis, retraction is not performed. In this case, after a block that does not specify threading is executed, an alarm PS0429, “ILLEGAL COMMAND IN G10.6” is issued and the tool stops.

Example

Threading at a lead of 1.5mm

```
G33 Z10. F1.5;
```



4 COORDINATE VALUE AND DIMENSION

4.1 POLAR COORDINATE COMMAND (G15, G16)

The end point coordinate value can be input in polar coordinates (radius and angle).
The plus direction of the angle is counterclockwise of the selected plane first axis + direction, and the minus direction is clockwise.
Both radius and angle can be commanded in either absolute or incremental programming (G90, G91).

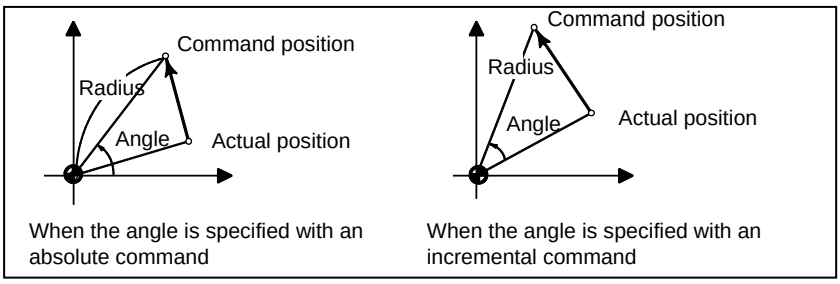
Format

```
Gxx Gyy G16;  Starting the polar coordinate command
              (polar coordinate mode)
G00 IP_ ;
 :            Polar coordinate command
 :
G15;          Canceling the polar coordinate command
              (polar coordinate mode)
```

G16	:	Polar coordinate command
G15	:	Polar coordinate command cancel
Gxx	:	Plane selection of the polar coordinate command (G17, G18 or G19)
Gyy	:	Center selection of the polar coordinate command (G90 or G91) G90 specifies the origin of the program coordinate system as the origin of the polar coordinate system, from which a radius is measured. G91 specifies the current position as the origin of the polar coordinate system, from which a radius is measured.
IP_	:	Specifying the addresses of axes constituting the plane selected for the polar coordinate system, and their values First axis : radius of polar coordinate Second axis : angle of polar coordinate

- **Setting the origin of the program coordinate system as the origin of the polar coordinate system**

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

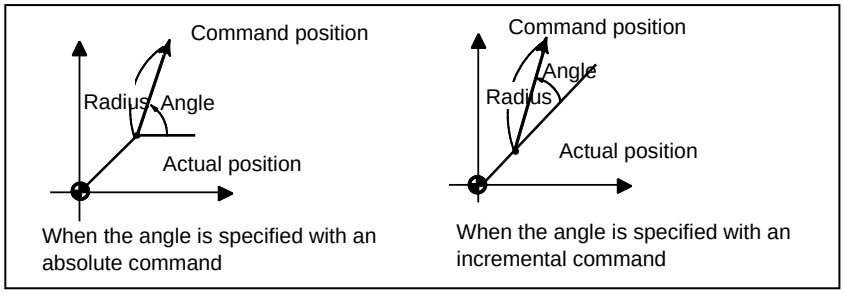


When the angle is specified with an absolute command

When the angle is specified with an incremental command

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

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



- Operation of which the address in the selected plane 1st axis (radius) or 2nd axis (angle) is omitted

The behavior depends on bit 5 (PCC) of parameter No. 10351.
(PCC = 0 (FS0*i* specification), PCC = 1 (FS16*i* compatible specification))

The origin of the polar coordinate system

The origin of the polar coordinate system is decided according to Table 4.1 (a).

Table 4.1 (a) The origin of the polar coordinate system is decided

<table>
<tr><th colspan="2"></th><th>PCC = 0</th><th>PCC = 1</th></tr>
<tr><td colspan="2">When G16 has been commanded</td><td colspan="2" rowspan="3">The origin of the program coordinate system

However, when the modal is G91 and there is the address of the selected plane 1st axis (radius), the origin of the polar coordinate system is the current position.

[Example] G16 G91 G00 X20.0 Y30.0</td></tr>
<tr><td colspan="2">When Polar coordinate command has been commanded after reset (*1)</td></tr>
<tr><td colspan="2">When the selected plane has been changed (G17,G18,G19)</td></tr>
<tr><td colspan="2">When the modal is G90 and there is the address of the selected plane 1st axis (radius)</td><td colspan="2">The origin of the program coordinate system</td></tr>
<tr><td colspan="2">When the modal is G91 and there is the address of the selected plane 1st axis (radius)</td><td colspan="2">The current position</td></tr>
<tr><td rowspan="2">When there is not the address of the selected plane 1st axis (radius) and there is the address of the selected plane 2nd axis (angle)</td><td>When the origin of the polar coordinate system before this command is the origin of the program coordinate system (*2)</td><td>The origin of the program coordinate system</td><td rowspan="2">The origin of the program coordinate system</td></tr>
<tr><td>When the origin of the polar coordinate system before this command is the current position (*3)</td><td>The current position
In addition, the radius becomes 0. Therefore, the axis doesn't move by this command.</td></tr>
<tr><td colspan="2">When there is not the address of the selected plane 1st axis (radius) and the selected plane 2nd axis (angle)</td><td colspan="2">The origin of the polar coordinate system is not decided because this command is not regarded as Polar coordinate command.</td></tr>
</table>

*1 This means that Polar coordinate command is continued after reset in the polar coordinate command mode.
This operation can use at reset state (bit 6 (CLR) of parameter No. 3402 is 0).
[Example]
G16 G90 G00 X100.0 Y45.0
:
RESET
G91 Y60.0 ..... Polar coordinate command is continued after reset.

*2 This means the following.
(1) G16 or the selected plane 1st axis (radius) in G90 is commanded.
(2) The origin of the program coordinate system is set to the origin of the polar coordinate.
(3) Thereafter, the selected plane 2nd axis (angle) is commanded without the address of the selected plane 1st axis (radius).
[Example]
G16 ..... The origin of the polar coordinate system is the origin of the program coordinate system.
G91 Y60.0 ..... There is not the address of the selected plane 1st axis (radius) and there is the address of the selected plane 2nd axis (angle).

*3 This means the following.
(1) The selected plane 1st axis (radius) in G91 is commanded.
(2) The current position is set to the origin of the polar coordinate.
(3) Thereafter, the selected plane 2nd axis (angle) is commanded without the address of the selected plane 1st axis (radius).

[Example]
G16
G91 X30.0 Y30.0 ..... The origin of the polar coordinate system is the current position.
G90 Y40.0 ..... There is not the address of the selected plane 1st axis (radius) and there is the address of the selected plane 2nd axis (angle).

The radius and angle

The radius and the angle at following cases are set according to Table 4.1 (b).
- When G16 has been commanded.
- When Polar coordinate command has been commanded after reset.
- When the selected plane has been changed (G17, G18, G19).

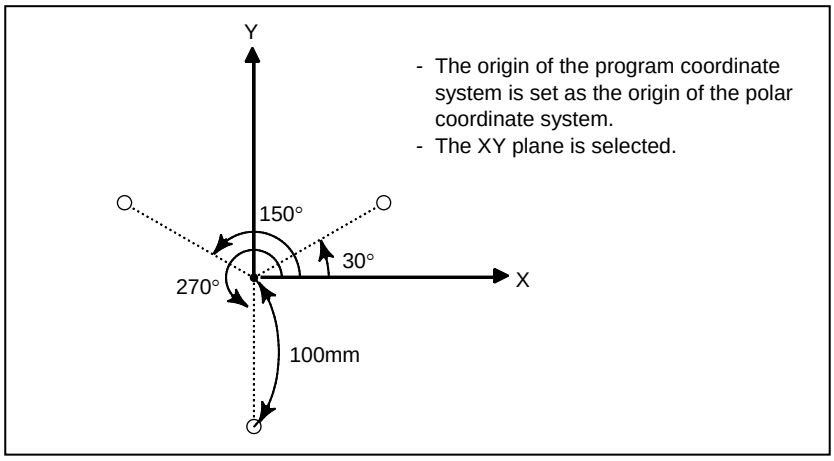
Table 4.1 (b) The radius and the angle

	PCC = 0	PCC = 1
When G16 has been commanded	The radius and the angle become 0. When the radius or the angle is commanded at the same time, the radius or the angle becomes the value specified in the command. [Example] G90 G00 X50.0 Y50.0 G16 ..... The radius = 0, the angle = 0. Y60.0 ..... The radius = 0, the angle = 60.0. Therefore, the axes move to (X 0.0, Y 0.0).	The radius and the angle are calculated from the current position. When the radius or the angle is commanded at the same time, the radius or the angle becomes the value specified in the command. [Example] G90 G00 X50.0 Y50.0 G16 ..... The radius = 70.710, the angle = 45.0. (from the current position (X 50.0, Y 50.0)) Y60.0 ..... The radius = 70.710, the angle = 60.0. Therefore, the axes move to (X 35.355, Y 61.237).
When Polar coordinate command has been commanded after reset		

	PCC = 0	PCC = 1
When the selected plane has been changed (G17,G18, G19)	The radius and the angle become 0. When the radius or the angle is commanded at the same time, the radius or the angle becomes the value specified in the command. [Example] G90 G16 G17 X100.0 Y30.0 ..... The radius = 100.0, the angle = 30.0. G19 Z40.0 .....The radius = 0, the angle = 40.0. Therefore, the axes move to (Y 0.0, Z 0.0).	The radius and the angle are succeeded. When the radius or the angle is commanded at the same time, the radius or the angle becomes the value specified in the command. [Example] G90 G16 G17 X100.0 Y30.0 ..... The radius = 100.0, the angle = 30.0. G19 Z40.0 .... The radius = 100.0, the angle = 40.0. Therefore, the axes move to (Y 76.604, Z 64.279).

Example

Bolt hole circle



- Specifying angles and a radius with absolute programmings

N1 G17 G90 G16 ; Specifying the polar coordinate command and selecting the XY plane
Setting the origin of the program coordinate system as the origin of the polar coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;
Specifying a distance of 100 mm and an angle of 30 deg
N3 Y150.0 ; Specifying a distance of 100 mm and an angle of 150 deg
N4 Y270.0 ; Specifying a distance of 100 mm and an angle of 270 deg
N5 G15 G80 ; Canceling the polar coordinate command

- Specifying angles with incremental programmings and a radius with absolute programmings

N1 G17 G90 G16; Specifying the polar coordinate command and selecting the XY plane
Setting the origin of the program coordinate system as the origin of the polar coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;
Specifying a distance of 100 mm and an angle of 30 deg
N3 G91 Y120.0 ; Specifying a distance of 100 mm and an angle of +120 deg
N4 Y120.0 ; Specifying a distance of 100 mm and an angle of +120 deg
N5 G15 G80 ; Canceling the polar coordinate command

Limitation

- Specifying a radius in the polar coordinate mode

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

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

Axes specified for the following commands are not considered part of the polar coordinate command. The command value is not converted by the polar coordinate command.

- Dwell (G04)
- Programmable data input (G10)
- Local coordinate system setting (G52)
- Workpiece coordinate system setting (G92)
- Machine coordinate system setting (G53)
- Stored stroke check (G22)
- Coordinate system rotation (G68)
- Scaling (G51)
- Tool retract and recover (G10.6)
- Workpiece coordinate system preset (G92.1)
- Figure copying (G72.1, G72.2)
- Cylindrical interpolation (G07.1, G107)
- Programmable mirror image (G51.1)

- Rotary axis

The polar coordinate command specify by the selected plane first axis and second axis. The polar coordinate command cannot be specified with the axis that is set as a rotation axis.

- Function with limitation when using simultaneously

There is a limitation when the following functions are used together with the polar coordinate command. For details of the limitations, refer to the explanation of each function.

- Retrace
- Inch/metric conversion

- Functions that cannot be used simultaneously

The following functions cannot be used together with the polar coordinate command.

- AI contour control
- Tilted working plane indexing
- Cs contour control
- Optional angle chamfering and corner rounding

NOTE

"Axes that are not considered part of a polar coordinate command in the polar coordinate mode", "Function with limitation when using simultaneously" and "Functions that cannot be used simultaneously" might be changed or added by adding new CNC function.

5 FUNCTIONS TO SIMPLIFY PROGRAMMING

5.1 CANNED CYCLE FOR DRILLING

Overview

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, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.
Table 5.1 (a) lists canned cycles for drilling.

Table 5.1 (a) Canned cycles for drilling

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

Explanation

A canned cycle for drilling consists of a sequence of six operations.

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

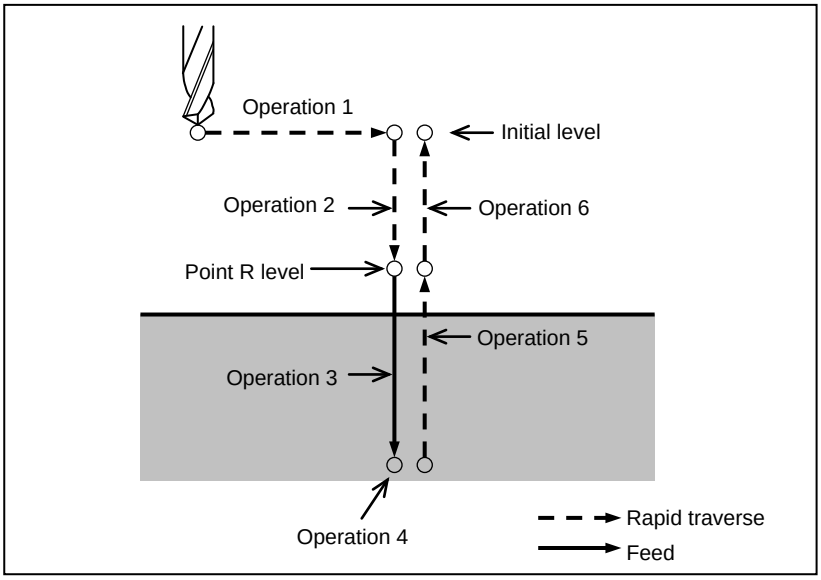


Fig. 5.1 (a) Operation sequence of canned cycle for drilling

- Positioning plane

The positioning plane is determined by plane selection code G17, G18, or G19.
The positioning axis is an axis other than the drilling axis.

- Drilling axis

Although canned cycles for drilling include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.
The drilling axis is a basic axis (X, Y, or Z) not used to define the positioning plane, or any axis parallel to that basic axis.
The axis (basic axis or parallel axis) used as the drilling axis is determined according to the axis address for the drilling axis specified in the same block as G codes G73 to G89.
If no axis address is specified for the drilling axis, the basic axis is assumed to be the drilling axis.

Table 5.1 (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 an axis parallel to the X axis
Yp: Y axis or an axis parallel to the Y axis
Zp: Z axis or an axis parallel to the Z axis

Example

Assume that the U, V and W axes be parallel to the X, Y, and Z axes respectively. This condition is specified by parameter No. 1022.

G17 G81 Z_ _ : The Z axis is used for drilling.
G17 G81 W_ _: The W axis is used for drilling.
G18 G81 Y_ _ : The Y axis is used for drilling.
G18 G81 V_ _ : The V axis is used for drilling.
G19 G81 X_ _ : The X axis is used for drilling.
G19 G81 U_ _ : The U axis is used for drilling.

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

> ⚠ **CAUTION**
> Switch the drilling axis after canceling a canned cycle for drilling.

> **NOTE**
> A bit 0 (FXY) of parameter No. 5101 can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

- Travel distance along the drilling axis G90/G91

The travel distance along the drilling axis varies for G90 and G91 as Fig. 5.1 (b):

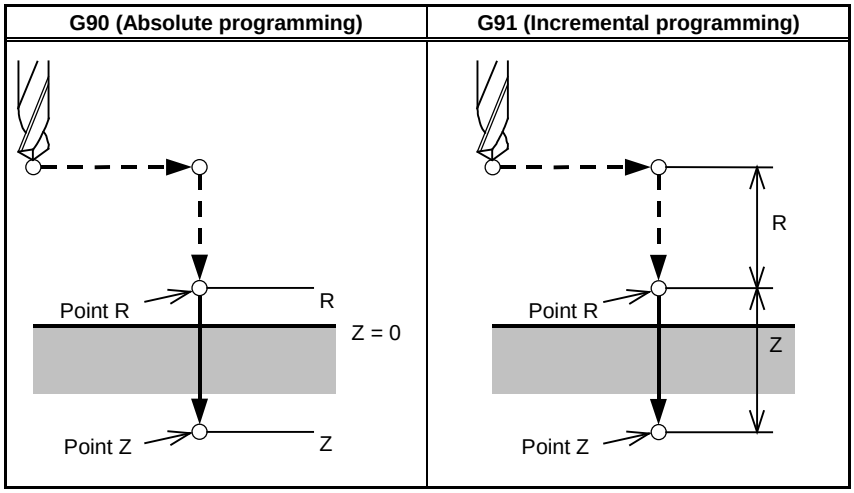


Fig. 5.1 (b) Absolute programming and incremental programming

- Drilling mode

G73, G74, G76, and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.

Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.

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

- Return point level G98/G99

When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The operations performed when G98 and G99 are specified are shown in Fig. 5.1 (c). 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.

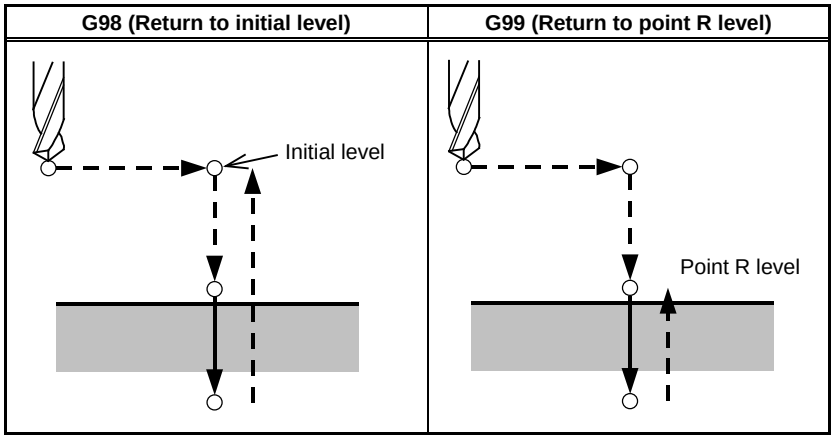


Fig. 5.1 (c) Initial level and point R level

- Clearance

Clearance is commanded with an address D with a comma. If the cycle is commanded without ",D" command, the clearance parameter will be valid. The cycles that can be commanded are as shown in Table 5.1 (c).
If clearance is not programmed, use parameterized clearance. ",D" should be commanded in the block where the drilling operation is performed. It is memorized as a modal command during canned cycle for drilling. Decimal point input is possible for the ",D" command.

Table 5.1 (c) List of canned cycle for drilling which clearance can be commanded

G code	Function	In the case of without ",D" command, the clearance parameter to be used
G73	High-speed peck drilling cycle	No.5114
G74, G84	High-speed peck rigid tapping cycle, Peck rigid tapping cycle	No.5213
G83	Peck drilling cycle	No.5115
	Small-hole peck drilling cycle	No.5174
G84.2	Rigid tapping cycle (FS15 format)	No.5213
G84.3	Left-handed rigid tapping cycle (FS15 format)	No.5213

- Repeat

To repeat drilling for equally-spaced holes, specify the number of repeats in K_.
K is effective only within the block where it is specified.
Specify the first hole position in incremental programming (G91).
If it is specified in absolute programming (G90), drilling is repeated at the same position.

Number of repeats K	The maximum command value = 9999

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

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

- Single block

If a drilling cycle is performed in a single block, the control unit stops at each of the end points of operations 1, 2, and 6 in Fig. 5.1 (a). This means that three starts are made to make a single hole. At the end points of operations 1 and 2, the feed hold lamp turns on and the control unit stops. If the repetitive count is not exhausted at the end point of operation 6, the control unit stops in the feed hold mode, and otherwise, stops in the single block stop mode. Note that G87 does not cause a stop at point R in G87. G88 causes a stop at point Z after a dwell.

- Cancel

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

Group 01 G codes

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

- Symbols in figures

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

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

5.1.1 High-Speed Peck Drilling Cycle (G73)

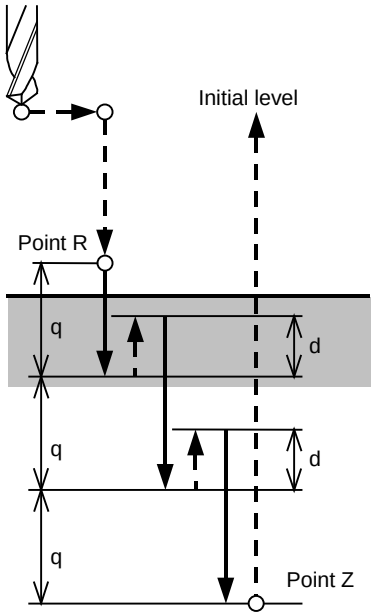
This cycle performs high-speed peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

Format

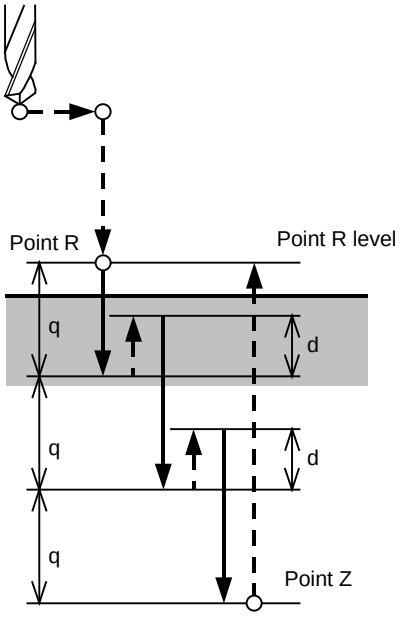
G73 X_ Y_ Z_ R_ Q_ ,D_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
Q_	:	Depth of cut for each cutting feed
,D_	:	Clearance
F_	:	Cutting feedrate
K_	:	Number of repeats (if required)

G73 (G98)



G73 (G99)



Explanation

- Operations

The high-speed peck drilling cycle performs intermittent feeding along the Z-axis. When this cycle is used, chips can be removed from the hole easily, and a smaller value can be set for retraction. This allows, drilling to be performed efficiently. Set the clearance, d, in ",D" command or parameter No. 5114.
The tool is retracted in rapid traverse.

- Spindle rotation

Before specifying G73, rotate the spindle using an auxiliary function (M code).

- Auxiliary function

When the G73 code and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Q

Specify Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G73 in a single block. Otherwise, G73 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                  Cause the spindle to start rotating.
G90 G99 G73 X300.0 Y-250.0 Z-150.0 R-100.0 Q15.0 F120 ;
                            Position, drill hole 1, then return to point R.
Y-550.0 ;                   Position, drill hole 2, then return to point R.
Y-750.0 ;                   Position, drill hole 3, then return to point R.
X1000.0 ;                   Position, drill hole 4, then return to point R.
Y-550.0 ;                   Position, drill hole 5, then return to point R.
G98 Y-750.0 ;               Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;      Return to the reference position
M5 ;                        Cause the spindle to stop rotating.
```

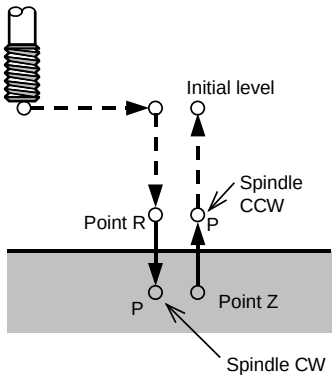
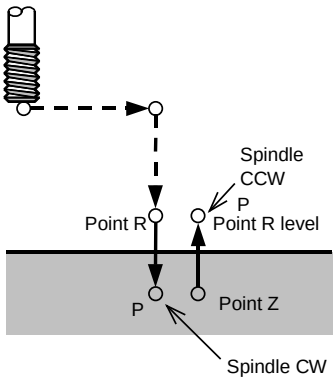
5.1.2 Left-Handed Tapping Cycle (G74)

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

Format

G74 X_ Y_ Z_ R_P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time
F_	:	Cutting feedrate
K_	:	Number of repeats (if required)

G74 (G98)	G74 (G99)
	

Explanation

- Operations

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

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

- Spindle rotation

Before specifying G74, use an auxiliary function (M code) to rotate the spindle counterclockwise.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation. In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G74 in a single block. Otherwise, G74 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M4 S100 ;	Cause the spindle to start rotating.
G90 G99 G74 X300.0 Y-250.0 Z-150.0 R-120.0 F120 ;	Position, tapping hole 1, then return to point R.
Y-550.0 ;	Position, tapping hole 2, then return to point R.
Y-750.0 ;	Position, tapping hole 3, then return to point R.
X1000.0 ;	Position, tapping hole 4, then return to point R.
Y-550.0 ;	Position, tapping hole 5, then return to point R.
G98 Y-750.0 ;	Position, tapping hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

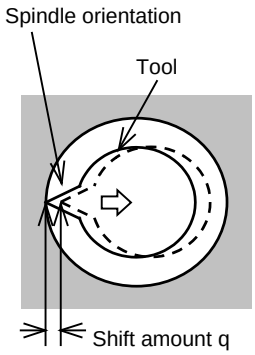
5.1.3 Fine Boring Cycle (G76)

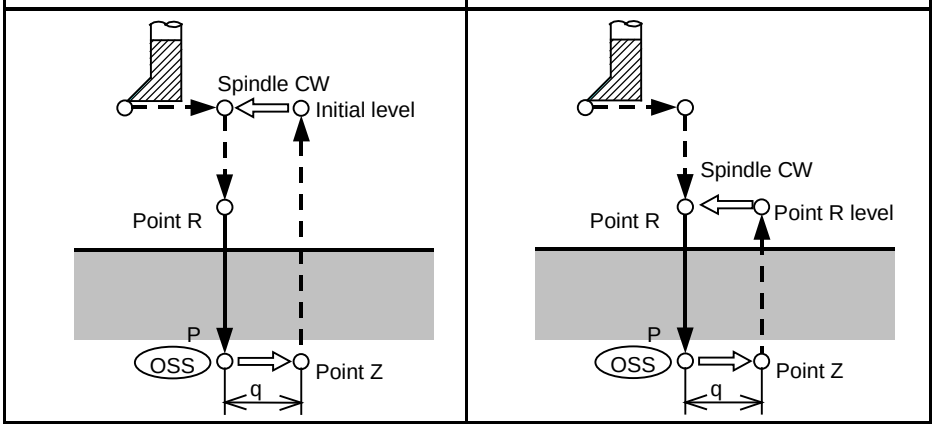
The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.

Format

G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

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





Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

When the G76 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P, Q

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in the parameter No.5148.
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.

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G76 in a single block. Otherwise, G76 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M3 S500 ;	Cause the spindle to start rotating.
G90 G99 G76 X300.0 Y-250.0	Position, bore hole 1, then return to point R.
Z-150.0 R-120.0 Q5.0	Orient at the bottom of the hole, then shift by 5 mm.
P1000 F120 ;	Stop at the bottom of the hole for 1 sec.
Y-550.0 ;	Position, drill hole 2, then return to point R.
Y-750.0 ;	Position, drill hole 3, then return to point R.
X1000.0 ;	Position, drill hole 4, then return to point R.
Y-550.0 ;	Position, drill hole 5, then return to point R.
G98 Y-750.0 ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

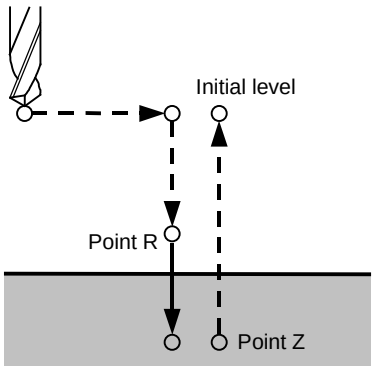
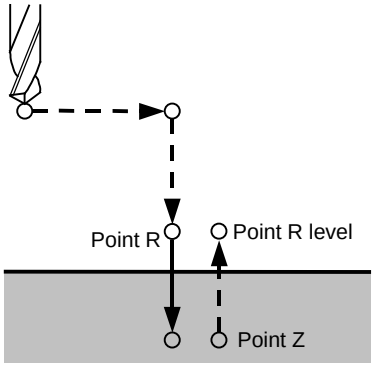
5.1.4 Drilling Cycle, Spot Drilling (G81)

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

Format

G81 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
F_	:	Cutting feedrate
K_	:	Number of repeats (if required)

G81 (G98)	G81 (G99)
	

Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
The tool is then retracted in rapid traverse.

- Spindle rotation

Before specifying G81, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G81 in a single block. Otherwise, G81 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G81 X300.0 Y-250.0 Z-150.0 R-100.0 F120 ;	Position, drill hole 1, then return to point R.
Y-550.0 ;	Position, drill hole 2, then return to point R.
Y-750.0 ;	Position, drill hole 3, then return to point R.
X1000.0 ;	Position, drill hole 4, then return to point R.
Y-550.0 ;	Position, drill hole 5, then return to point R.
G98 Y-750.0 ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

5.1.5 Drilling Cycle Counter Boring Cycle (G82)

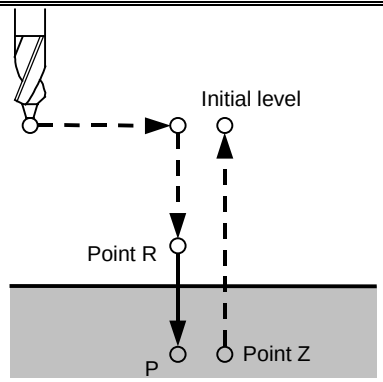
This cycle is used for normal drilling.
Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse.
This cycle is used to drill holes more accurately with respect to depth.

Format

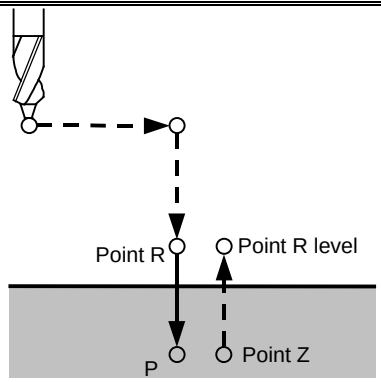
G82 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)

G82 (G98)



G82 (G99)



Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is then performed from point R to point Z.
When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.

- Spindle rotation

Before specifying G82, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G82 in a single block. Otherwise, G82 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G82 X300.0 Y-250.0 Z-150.0 R-100.0 P1000 F120 ;	
	Position, drill hole 1, and dwell for 1 sec at the bottom of the hole, then return to point R.
Y-550.0 ;	Position, drill hole 2, then return to point R.
Y-750.0 ;	Position, drill hole 3, then return to point R.
X1000.0 ;	Position, drill hole 4, then return to point R.
Y-550.0 ;	Position, drill hole 5, then return to point R.
G98 Y-750.0 ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

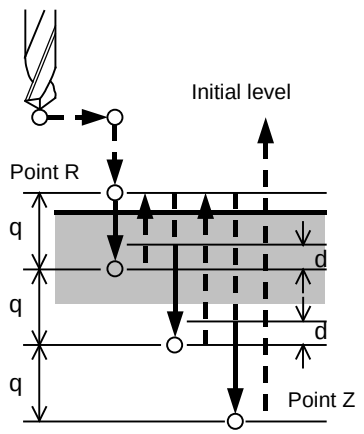
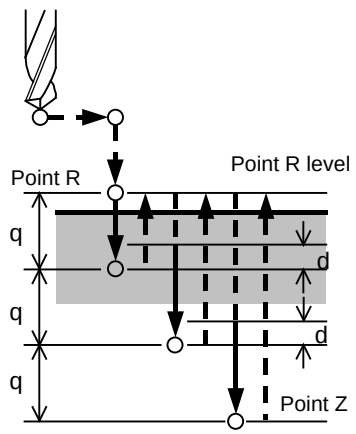
5.1.6 Peck Drilling Cycle (G83)

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

Format

G83 X_ Y_ Z_ R_ Q_ ,D_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
Q_	:	Depth of cut for each cutting feed
,D_	:	Clearance
F_	:	Cutting feedrate
K_	:	Number of repeats (if required)

G83 (G98)	G83 (G99)
	

Explanation

- Operations

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value.
In the second and subsequent cutting feeds, rapid traverse is performed up to a point that lies at a distance of d just before where the last drilling has ended. Then, the cutting feed is performed from the point again.
d is set in ",D" command or parameter No.5115.
Be sure to specify a positive value in Q. Negative values are ignored.

- Spindle rotation

Before specifying G83, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Q

Specify Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G83 in a single block. Otherwise, G83 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                    Cause the spindle to start rotating.
G90 G99 G83 X300.0 Y-250.0 Z-150.0 R-100.0 Q15.0 F120.0 ;
                              Position, drill hole 1, then return to point R.
Y-550.0 ;                     Position, drill hole 2, then return to point R.
Y-750.0 ;                     Position, drill hole 3, then return to point R.
X1000.0 ;                     Position, drill hole 4, then return to point R.
Y-550.0 ;                     Position, drill hole 5, then return to point R.
G98 Y-750.0 ;                 Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;        Return to the reference position
M5 ;                          Cause the spindle to stop rotating.
```

5.1.7 Small-Hole Peck Drilling Cycle (G83)

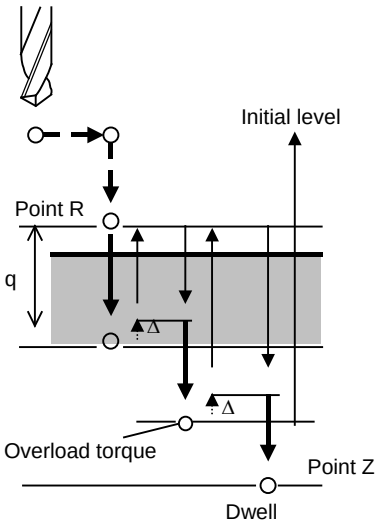
An arbor with the overload torque detection function is used to retract the tool when the overload torque detection signal (skip signal) is detected during drilling. Drilling is resumed after the spindle speed and cutting feedrate are changed. These steps are repeated in this peck drilling cycle.
The mode for the small–hole peck drilling cycle is selected when the M code in parameter No. 5163 is specified. The cycle can be started by specifying G83 in this mode. This mode is canceled when G80 is specified or when a reset occurs.

Format

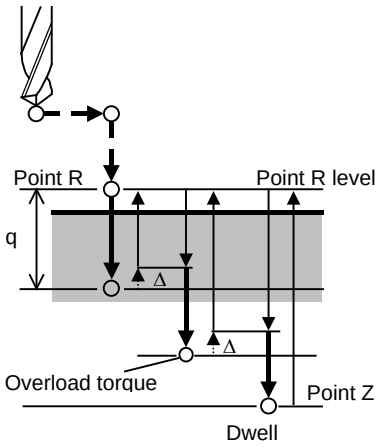
G83 X_ Y_ Z_ R_ Q_ ,D_ F_ I_ K_ P_ ;

X_ Y_	:	Hole position data
Z_	:	Distance from point R to the bottom of the hole
R_	:	Distance from the initial level to point R
Q_	:	Depth of each cut
,D_	:	Clearance
F_	:	Cutting feedrate
I_	:	Forward or backward traveling speed (same format as F above) (If this is omitted, the values in parameters Nos. 5172 and 5173 are assumed as defaults.)
K_	:	Number of times the operation is repeated (if required)
P_	:	Dwell time at the bottom of the hole (If this is omitted, P0 is assumed as the default.)

G83 (G98)



G83 (G99)



Δ: Initial clearance when the tool is retracted to point R and the clearance from the bottom of the hole in the second or subsequent drilling (",D" command or parameter No. 5174)
q: Depth of each cut
- - → Path along which the tool travels at the rapid traverse rate
——→ Path along which the tool travels at the programmed cutting feedrate
——→ Path along which the tool travels at the forward or backward rate specified with parameters during the cycle
(·····→)

Explanations

- Component operations of the cycle

* X- and Y-axis positioning
* Positioning at point R along the Z-axis
* Cutting along the Z-axis (first time, depth of cut Q, incremental)
  Retracting (bottom of hole → minimum clearance Δ, incremental)
  Retracting (bottom of hole + clearance Δ → point R, absolute)
  Forwarding (point R → bottom of hole + clearance Δ, absolute)
  Cutting (second and subsequent times, cut of depth Q + Δ, incremental)
* Dwell
* Return to point R along the Z-axis (or initial point) = end of cycle

Acceleration/deceleration during advancing and retraction is controlled according to the cutting feed acceleration/deceleration time constant.
When retraction is performed, the position is checked at point R.

- Specifying an M code

When the M code in parameter No. 5163 is specified, the system enters the mode for the small–hole peck drilling cycle.
This M code does not wait for FIN. Care must be taken when this M code is specified with another M code in the same block.

(Example) M03 M□□ ; → Waits for FIN.
M□□ M03 ; → Does not wait for FIN.

- Specifying a G code

When G83 is specified in the mode for the small-hole peck drilling cycle, the cycle is started.
This continuous–state G code remains unchanged until another canned cycle is specified or until the G code for canceling the canned cycle is specified. This eliminates the need for specifying drilling data in each block when identical drilling is repeated.

- Signal indicating that the cycle is in progress

In this cycle mode, the small-hole peck drilling cycle in progress signal is set to "1" at the start of point R positioning on the axis in the drilling direction after G83 is specified and positioning is performed to the specified hold position. This signal is set to "0" if another canned cycle is specified or if this mode is canceled with G80, a reset, or an emergency stop. For details, refer to the manual of the machine tool builder.

- Overload torque detection signal

A skip signal is used as the overload torque detection signal. The skip signal is effective while the tool is advancing or drilling and the tool tip is between points R and Z. (The signal causes a retraction). For details, refer to the manual of the machine tool builder.

NOTE
When receiving overload torque detect signal while the tool is advancing, the tool will be retracted (clearance Δ and to the point R), then advanced to the same target point as previous advancing.

- Changing the drilling conditions

In a single G83 cycle, drilling conditions are changed for each drilling operation (advance → drilling → retraction). Bits 1 and 2 of parameter OLS, NOL No. 5160 can be specified to suppress the change in drilling conditions.

1 Changing the cutting feedrate
The cutting feedrate programmed with the F code is changed for each of the second and subsequent drilling operations. In parameters Nos.5166 and 5167, specify the respective rates of change applied when the skip signal is detected and when it is not detected in the previous drilling operation.

Cutting feedrate = F × α

<First drilling> $\alpha=1.0$
<Second or subsequent drilling>
$\alpha=\alpha\times\beta\div100$, where β is the rate of change for each drilling operation
When the skip signal is detected during the previous drilling operation: β=b1% (parameter No.5166)
When the skip signal is not detected during the previous drilling operation: β=b2% (parameter No.5167)

If the rate of change in cutting feedrate becomes smaller than the rate specified in parameter No. 5168, the cutting feedrate is not changed.
The cutting feedrate can be increased up to the maximum cutting feedrate.

2 Changing the spindle speed
The spindle speed programmed with the S code is changed for each of the second and subsequent advances. In parameters Nos. 5164 and 5165, specify the rates of change applied when the skip signal is detected and when it is not detected in the previous drilling operation.

Spindle speed = S × γ

<First drilling> $\gamma=1.0$
<Second or subsequent drilling>
$\gamma=\gamma\times\delta\div100$, where δ is the rate of change for each drilling operation
When the skip signal is detected during the previous drilling operation: δ=d1% (parameter No.5164)
When the skip signal is not detected during the previous drilling operation: δ=d2% (parameter No.5165)

When the cutting feedrate reaches the minimum rate, the spindle speed is not changed. The spindle speed can be increased up to a value corresponding to the maximum value of S analog data.

- Advance and retraction

Advancing and retraction of the tool are not executed in the same manner as rapid-traverse positioning. Like cutting feed, the two operations are carried out as interpolated operations. Note that the tool life management function excludes advancing and retraction from the calculation of the tool life.

- Specifying address I

The forward or backward traveling speed can be specified with address I in the same format as address F, as shown below:

```
G83 I1000 ;   (without decimal point)
G83 I1000. ;  (with decimal point)
```

Both commands indicate a speed of 1000 mm/min.
Address I specified with G83 in the continuous-state mode continues to be valid until G80 is specified or until a reset occurs.

NOTE
If address I is not specified and parameter No.5172 (for backward) or No.5173 (for forward) is set to 0, the forward or backward travel speed is same as the cutting feedrate specified by F.

- Functions that can be specified

In this canned cycle mode, the following functions can be specified:

- Hole position on the X-axis, Y-axis, and additional axis
- Operation and branch by custom macro
- Subprogram (hole position group, etc.) calling
- Switching between absolute and incremental modes
- Coordinate system rotation
- Scaling (This command will not affect depth of cut Q or small clearance Δ.)
- Dry run
- Feed hold

- Single block

When single-block operation is enabled, drilling is stopped after each retraction. Also, a single block stop is performed by setting bit 0 (SBC) of parameter No. 5105.

- Feedrate override

The feedrate override function works during cutting, retraction, and advancing in the cycle.

- Custom macro interface

The number of retractions made during cutting and the number of retractions made in response to the overload signal received during cutting can be output to custom macro common variables (#100 to #149) specified in parameters Nos.5170 and 5171. Parameters Nos.5170 and 5171 can specify variable numbers within the range of #100 to #149.

Parameter No.5170: Specifies the number of the common variable to which the number of retractions made during cutting is output.

Parameter No.5171: Specifies the number of the common variable to which the number of retractions made in response to the overload signal received during cutting is output.

NOTE
The numbers of retraction output to common variables are cleared by G83 while small-hole peck drilling cycle mode.

- Positioning to hole position

When positioning the axes to hole position (axes X and Y when XY plane is used) in Small-hole peck drilling cycle, the machining time can be shortened by the spindle is not stopped.
This function is enabled by the parameter SPH(No.5108#6).

Limitation

- Subprogram call

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

Example

M03 S2000 ;	Cause the spindle to start rotating.
M□□ ;	Specifies the small-hole peck drilling cycle mode.
G90 G99 G83 X_ Y_ Z_ R_ Q_ F_ I_ K_ P_ ;	Specifies the small-hole peck drilling cycle.
X_ Y_ ;	Drills at another position.
:	
:	
G80 ;	Cancels the small-hole peck drilling cycle mode.

5.1.8 Tapping Cycle (G84)

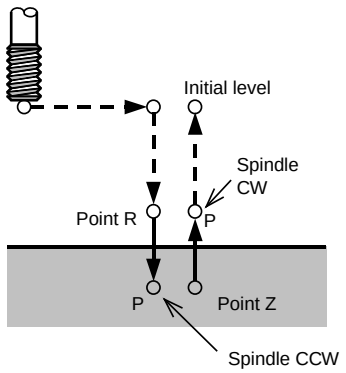
This cycle performs tapping.
In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format

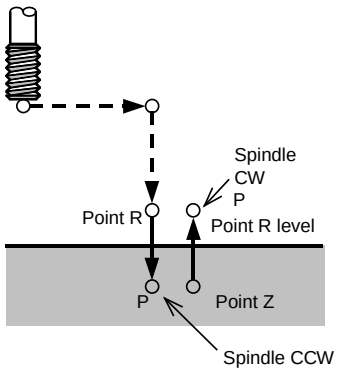
G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time
F_	:	Cutting feedrate
K_	:	Number of repeats (if required)

G84 (G98)



G84 (G99)



Explanation

- Operations

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.

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

- Spindle rotation

Before specifying G84, use an auxiliary function (M code) to rotate the spindle.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation. In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When the K is used to specify number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

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

Example

```
M3 S100 ;                    Cause the spindle to start rotating.
G90 G99 G84 X300.0 Y-250.0 Z-150.0 R-120.0 P300 F120.0 ;
                             Position, drill hole 1, then return to point R.
Y-550.0;                     Position, drill hole 2, then return to point R.
Y-750.0;                     Position, drill hole 3, then return to point R.
X1000.0;                     Position, drill hole 4, then return to point R.
Y-550.0;                     Position, drill hole 5, then return to point R.
G98 Y-750.0;                 Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;       Return to the reference position
M5 ;                         Cause the spindle to stop rotating.
```

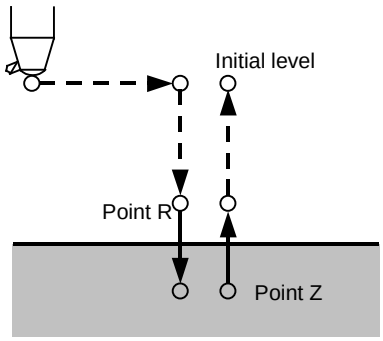
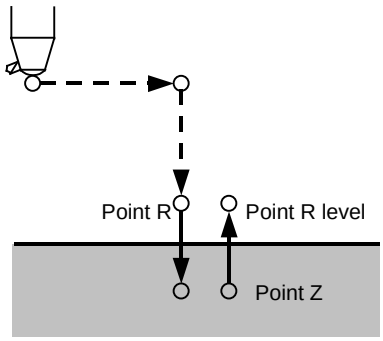
5.1.9 Boring Cycle (G85)

This cycle is used to bore a hole.

Format

G85 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)

G85 (G98)	G85 (G99)
	

Explanation

- Operations

After positioning along the X- and Y- axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
When point Z has been reached, cutting feed is performed to return to point R.

- Spindle rotation

Before specifying G85, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G85 in a single block. Otherwise, G85 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S100 ;                    Cause the spindle to start rotating.
G90 G99 G85 X300.0Y-250.0Z-150.0R-120.0F120.0;
                             Position, drill hole 1, then return to point R.
Y-550.0;                     Position, drill hole 2, then return to point R.
Y-750.0;                     Position, drill hole 3, then return to point R.
X1000.0;                     Position, drill hole 4, then return to point R.
Y-550.0;                     Position, drill hole 5, then return to point R.
G98 Y-750.0;                 Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;       Return to the reference position
M5 ;                         Cause the spindle to stop rotating.
```

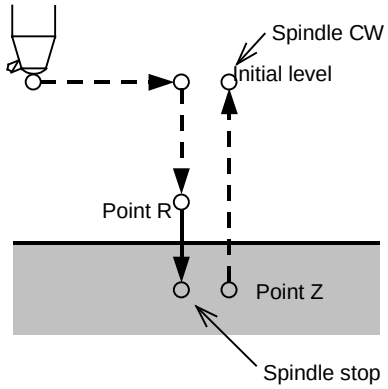
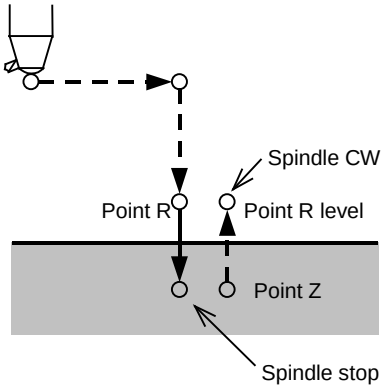
5.1.10 Boring Cycle (G86)

This cycle is used to bore a hole.

Format

G86 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)

G86 (G98)	G86 (G99)
	

Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.

- Spindle rotation

Before specifying G86, use an auxiliary function (M code) to rotate the spindle.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation.
In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G86 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G86 in a single block. Otherwise, G86 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                    Cause the spindle to start rotating.
G90 G99 G86 X300.0Y-250.0Z-150.0R-100.0F120.0;
                              Position, drill hole 1, then return to point R.
Y-550.0;                      Position, drill hole 2, then return to point R.
Y-750.0;                      Position, drill hole 3, then return to point R.
X1000.0;                      Position, drill hole 4, then return to point R.
Y-550.0;                      Position, drill hole 5, then return to point R.
G98 Y-750.0;                  Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;        Return to the reference position
M5 ;                          Cause the spindle to stop rotating.
```

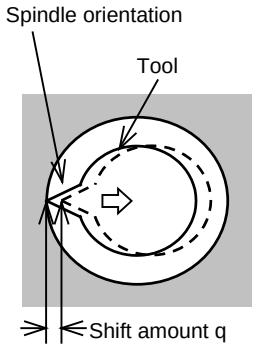
5.1.11 Back Boring Cycle (G87)

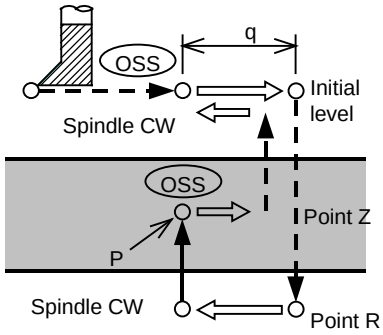
This cycle performs accurate boring.

Format

G87 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R
Q_	:	Shift amount at the bottom of a hole
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)



G87 (G98)	G87 (G99)
	Not used

Explanation

After positioning along the X- and Y-axes, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tool nose, positioning (rapid traverse) is performed to the bottom of the hole (point R).
The tool is then shifted in the direction of the tool nose and the spindle is rotated clockwise. Boring is performed in the positive direction along the Z-axis until point Z is reached.
At point Z, the spindle is stopped at the fixed rotation position again, the tool is shifted in the direction opposite to the tool nose, then the tool is returned to the initial level. The tool is then shifted in the direction of the tool nose and the spindle is rotated clockwise to proceed to the next block operation.

- Spindle rotation

Before specifying G87, use an auxiliary function (M code) to rotate the spindle.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation. In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G87 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P/Q

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in the parameter No. 5148.
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.

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G87 in a single block. Otherwise, G87 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M3 S500 ;	Cause the spindle to start rotating.
G90 G87 X300.0 Y-250.0	Position, bore hole 1.
Z-150.0 R-120.0 Q5.0	Orient at the initial level, then shift by 5 mm.
P1000 F120.0 ;	Stop at point Z for 1 s.
Y-550.0 ;	Position, drill hole 2.
Y-750.0 ;	Position, drill hole 3.
X1000.0 ;	Position, drill hole 4.
Y-550.0 ;	Position, drill hole 5.
Y-750.0 ;	Position, drill hole 6
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

5.1.12 Boring Cycle (G88)

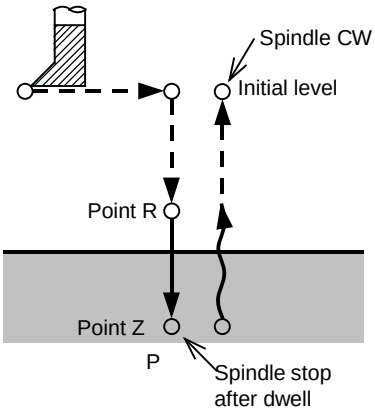
This cycle is used to bore a hole.

Format

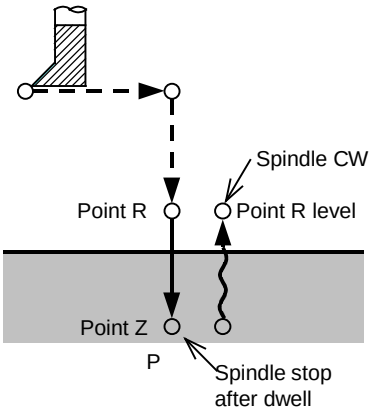
G88 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)

G88 (G98)



G88 (G99)



Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R. Boring is performed from point R to point Z.
When boring is completed, a dwell is performed at the bottom of the hole, then the spindle is stopped and enters the hold state. At this time, you can switch to the manual mode and move the tool manually. Any manual operations are available; it is desirable to finally retract the tool from the hole for safety, though.
At the restart of machining in the DNC operation or memory mode, the tool returns to the initial level or point R level according to G98 or G99 and the spindle rotates clockwise. Then, operation is restarted according to the programmed commands in the next block.

- Spindle rotation

Before specifying G88, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G88 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G88 in a single block. Otherwise, G88 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G88 X300.0 Y-250.0 Z-150.0 R-100.0 P1000 F120.0 ;	
	Position, drill hole 1, return to point R then stop at the bottom of the hole for 1 s.
Y-550.0 ;	Position, drill hole 2, then return to point R.
Y-750.0 ;	Position, drill hole 3, then return to point R.
X1000.0 ;	Position, drill hole 4, then return to point R.
Y-550.0 ;	Position, drill hole 5, then return to point R.
G98 Y-750.0 ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

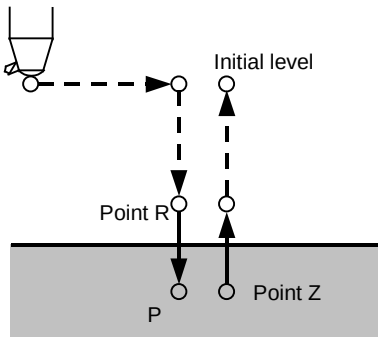
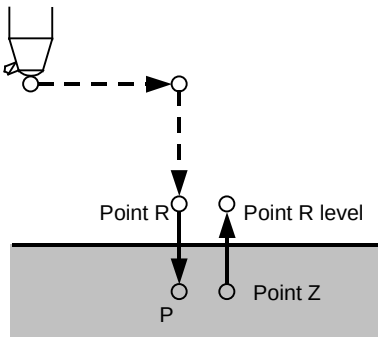
5.1.13 Boring Cycle (G89)

This cycle is used to bore a hole.

Format

G89 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)

G89 (G98)	G89 (G99)
	

Explanation

- Operations

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

- Spindle rotation

Before specifying G89, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G89 in a single block. Otherwise, G89 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S100 ;                    Cause the spindle to start rotating.
G90 G99 G89 X300.0 Y-250.0 Z-150.0 R-120.0 P1000 F120.0 ;
                             Position, drill hole 1, return to point R then stop at the bottom of the hole
                             for 1 s.
Y-550.0 ;                    Position, drill hole 2, then return to point R.
Y-750.0 ;                    Position, drill hole 3, then return to point R.
X1000.0 ;                    Position, drill hole 4, then return to point R.
Y-550.0 ;                    Position, drill hole 5, then return to point R.
G98 Y-750.0 ;                Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;       Return to the reference position
M5 ;                         Cause the spindle to stop rotating.
```

5.1.14 Canned Cycle Cancel for Drilling (G80)

G80 cancels canned cycles for drilling.

Format

G80 ;

Explanation

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

Example

```
M3 S100 ;                    Cause the spindle to start rotating.
G90 G99 G88 X300.0 Y-250.0 Z-150.0 R-120.0 F120.0 ;
                             Position, drill hole 1, then return to point R.
Y-550.0 ;                    Position, drill hole 2, then return to point R.
Y-750.0 ;                    Position, drill hole 3, then return to point R.
X1000.0 ;                    Position, drill hole 4, then return to point R.
Y-550.0 ;                    Position, drill hole 5, then return to point R.
G98 Y-750.0 ;                Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;       Return to the reference position, canned cycle cancel
M5 ;                         Cause the spindle to stop rotating.
```

5.1.15 Example for Using Canned Cycles for Drilling

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

Program example

```
;
N001  G92 X0 Y0 Z0;                                  Coordinate setting at reference position
N002  G90 G00 Z250.0 T11 M6;                         Tool change
N003  G43 Z0 H11;                                    Initial level, tool length compensation
N004  S30 M3;                                        Spindle start
N005  G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120;    Positioning, then #1 drilling
N006  Y-550.0;                                       Positioning, then #2 drilling and point R level return
N007  G98 Y-750.0;                                   Positioning, then #3 drilling and initial level return
N008  G99 X1200.0;                                   Positioning, then #4 drilling and point R level return
N009  Y-550.0;                                       Positioning, then #5 drilling and point R level return
N010  G98 Y-350.0;                                   Positioning, then #6 drilling and initial level return
N011  G00 X0 Y0 M5;                                  Reference position return, spindle stop
N012  G49 Z250.0 T15 M6;                             Tool length compensation cancel, tool change
N013  G43 Z0 H15;                                    Initial level, tool length compensation
N014  S20 M3;                                        Spindle start
N015  G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 P300 F70 ;  Positioning, then #7 drilling, point R level return
N016  G98 Y-650.0;                                   Positioning, then #8 drilling, initial level return
N017  G99 X1050.0;                                   Positioning, then #9 drilling, point R level return
N018  G98 Y-450.0;                                   Positioning, then #10 drilling, initial level return
N019  G00 X0 Y0 M5;                                  Reference position return, spindle stop
N020  G49 Z250.0 T31 M6;                             Tool length compensation cancel, tool change
N021  G43 Z0 H31;                                    Initial level, tool length compensation
N022  S10 M3;                                        Spindle start
N023  G85 G99 X800.0 Y-350.0 Z-153.0 R47.0 F50;      Positioning, then #11 drilling, point R level return
N024  G91 Y-200.0 K2;                                Positioning, then #12, 13 drilling, point R level
                                                     return
N025  G28 X0 Y0 M5;                                  Reference position return, spindle stop
N026  G49 Z0;                                        Tool length compensation cancel
N027  M0;                                            Program stop
```

Program using tool length offset and canned cycles

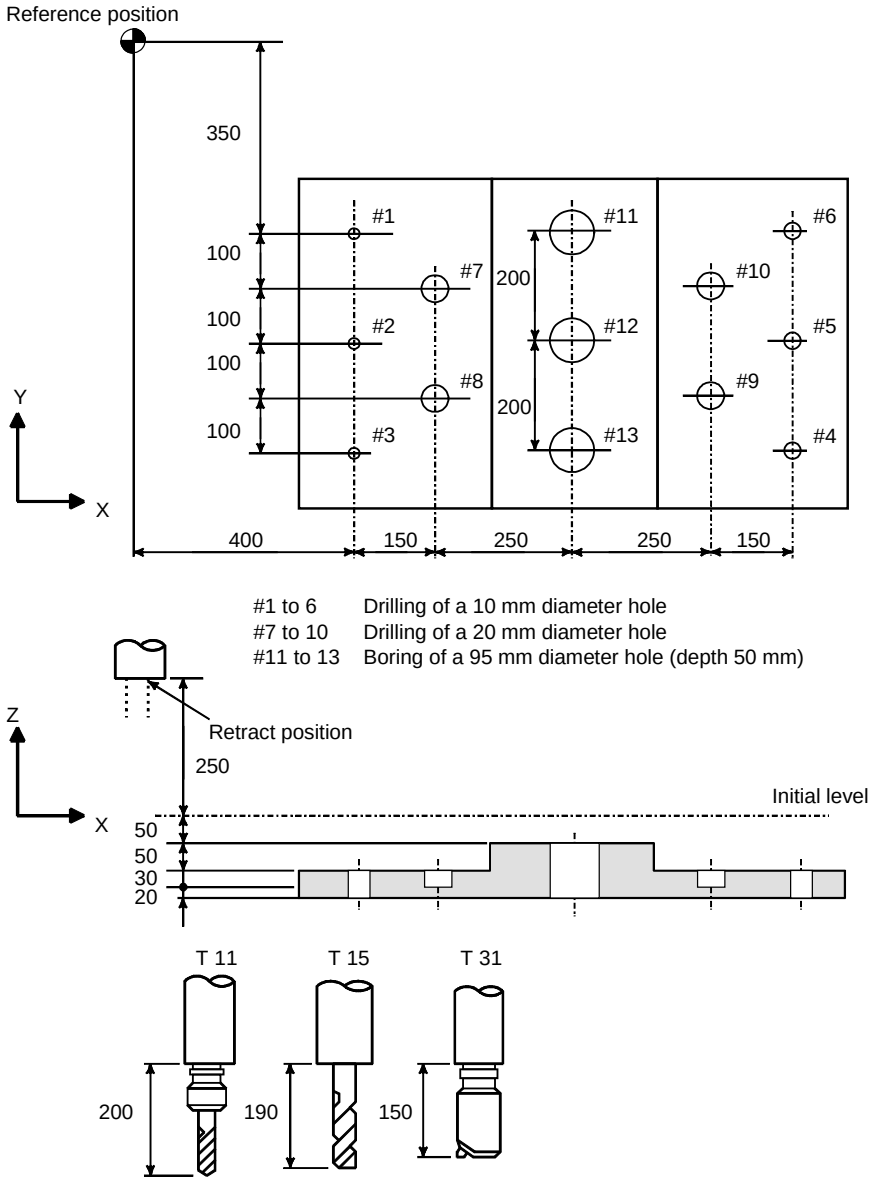


Fig. 5.1.15 (a) Example for using canned cycles for drilling

5.1.16 Reducing of Waiting Time of Spindle Speed Arrival in the Canned Cycle for Drilling

Overview

When bit 7 (SAC) of parameter No.11507 is set to 1, this function checks the spindle speed arrival signal SAR without waiting time that is set a parameter No.3740 at starting of drilling since the second times in canned cycle for drilling.
Also, this function is available rapid traverse to the initial lever and block overlap in rapid traverse of positioning to a next position of hole in canned cycle for drilling. These improvements reduce the cycle time.

Explanation

A canned cycle for drilling consists of a sequence of six operations.
Operation 1 Positioning of axes X and Y (including also another axis)
Operation 2 Rapid traverse up to point R level
Operation 3 Hole machining
Operation 4 Operation at the bottom of a hole
Operation 5 Retraction to point R level
Operation 6 Rapid traverse up to the initial point

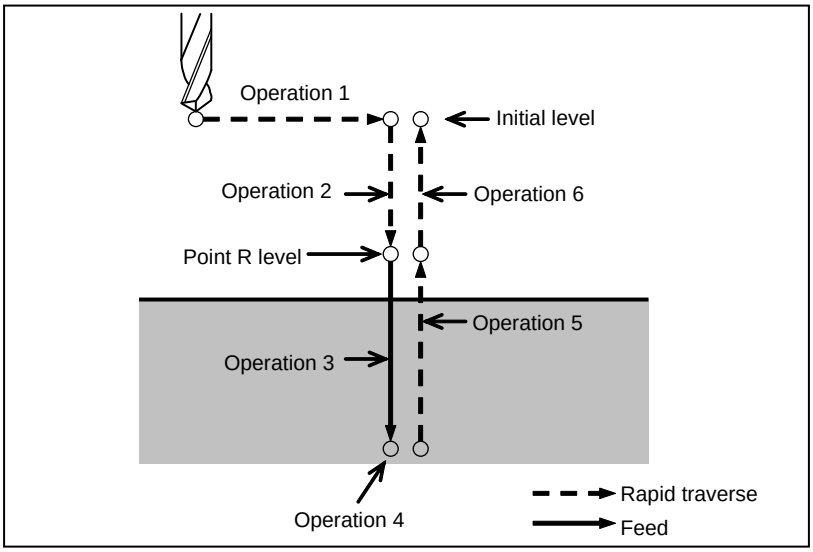


Fig. 5.1.16 (a) Operation sequence of canned cycle for drilling

When bit 7 (SAC) of parameter No.11507 is set to 0, the spindle speed arrival signal SAR is checked after waiting for elapsing time that is set parameter No.3740 for each drilling.
When bit 7 (SAC) of parameter No.11507 is set to 1, in drilling since the second times, the spindle speed arrival signal SAR is checked immediately that is set parameter No.3740 is not related.
However, when command and state are the following conditions, CNC is waiting for elapsing time that is set parameter No.3740 before checking the spindle speed arrival signal SAR.

- Canned cycle for drilling is canceled by G80 or G code of group 01.
- S code is commanded.
- G code of canned cycle for drilling is commanded which is different modal G code.
- The spindle speed arrival signal SAR becomes “0”.
- CNC becomes reset state.

Applied of speed-up of each command

Table of canned cycle for drilling (Series 0*i* format)

G code	Function	Reducing of waiting time for SAR	Block overlap in rapid traverse
G73	High-speed peck drilling cycle	available	available
G74	Left-hand tapping cycle Left-handed rigid tapping cycle	-	available
G76	Fine boring cycle	available	available
G81	Drilling cycle, spot drilling Cycle	available	available
G82	Drilling cycle, counter boring Cycle	available	available
G83	Peck drilling cycle	available	available
G84	Tapping cycle Rigid tapping cycle	-	available
G85	Boring cycle	available	available
G86	Boring cycle	available	available
G87	Back boring cycle	available	available
G88	Boring cycle	available	available
G89	Boring cycle	available	available

Table of canned cycle for drilling (Series 15 format)

G code	Function	Reducing of waiting time for SAR	Block overlap in rapid traverse
G84.2	Rigid tapping cycle	-	available
G84.3	Left-handed rigid tapping cycle	-	available

5.2 CANNED CYCLE OVERLAP FOR DRILLING

Overview

With this function, during the canned cycle mode for drilling, the command can overlap with the next block for the fixed time of the time constant of the acceleration/deceleration after interpolation.
This speeds up the operation of the canned cycle for drilling and shortens the cycle time.

Explanation

- Supported canned cycle for drilling

Table5.2 (a) Canned cycle for drilling shows canned cycle for drilling supported by this function.

Table5.2 (a) Canned cycle for drilling

G code	Drilling	Operation at the bottom of a hole	Retraction	Application
G73	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G81	Cutting feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Cutting feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	-	Rapid traverse	Peck drilling cycle
G85	Cutting feed	-	Cutting feed	Boring cycle
G86	Cutting feed	Spindle stop	Rapid traverse	Boring cycle
G89	Cutting feed	Dwell	Cutting feed	Boring cycle

- Operations in which overlap is valid in each cycle

High-Speed Peck Drilling Cycle (G73)

q	: Depth of cut
d	: Return value
Point A	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.)
Point B	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point C1, C2	: Overlap is valid. (When bit 4 (DQL) of parameter No.1681 is set to 1, overlap is invalid.) (When overlap is done at point C1 and point C2, the actual depth of cut is smaller than specified amount q.)
Point D1, D2	: Overlap is valid. (When bit 2 (DRV) of parameter No.1681 is set to 1, overlap is invalid.) (When overlap is done at point D1 and point D2, the actual return value is smaller than specified amount d.)
Point E	: Overlap is valid. (When bit 5 (DZL) of parameter No.1681 is set to 1, overlap is invalid.)
Point F	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point F, no overlap is done at point F.)

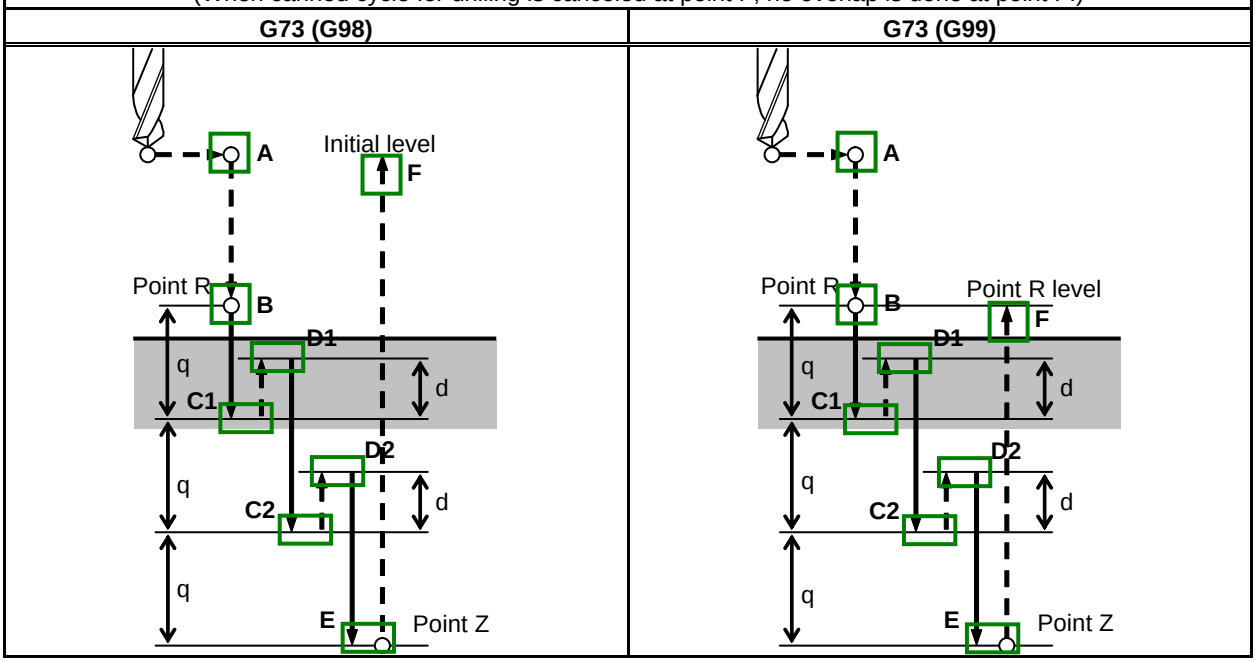


Fig. 5.2 (a) High-Speed Peck Drilling Cycle (G73)

Drilling Cycle, Spot Drilling (G81)	
Point A	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.)
Point B	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point C	: Overlap is valid. (When bit 5 (DZL) of parameter No.1681 is set to 1, overlap is invalid.)
Point D	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point D, no overlap is done at point D.)

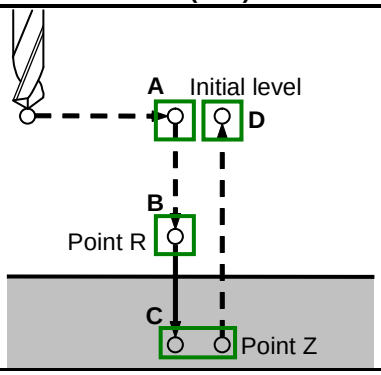
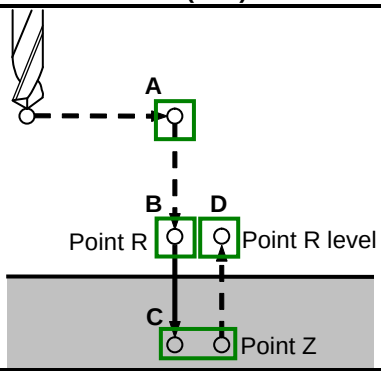
G81 (G98)	G81 (G99)
	

Fig.5.2 (b) Drilling Cycle, Spot Drilling (G81)

Drilling Cycle Counter Boring Cycle (G82)	
P	: Dwell time
Point A	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.)
Point B	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point C	: Overlap is valid. (When bit 5 (DZL) of parameter No.1681 is set to 1, overlap is invalid.) (When dwell is commanded, no overlap is done at point C.)
Point D	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point D, no overlap is done at point D.)

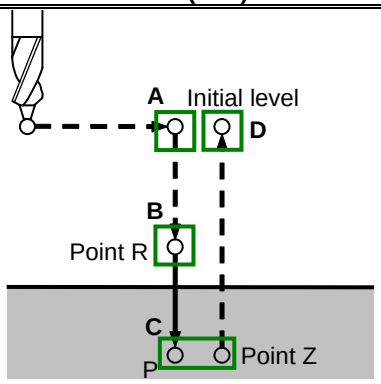
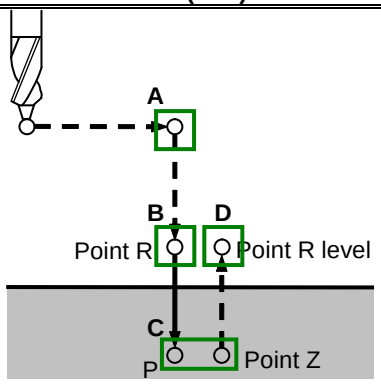
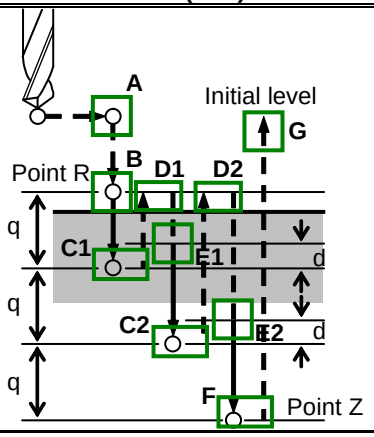
G82 (G98)	G82 (G99)
	

Fig.5.2 (c) Drilling Cycle Counter Boring Cycle (G82)

Peck Drilling Cycle (G83)

q	: Depth of cut
d	: In the second and subsequent cuts, rapid traverse changes to cutting feed at a position d before the end point of the last cut.
Point A	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.)
Point B	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point C1, C2	: Overlap is valid. (When bit 4 (DQL) of parameter No.1681 is set to 1, overlap is invalid.) (When overlap is done at point C1 and point C2, the actual depth of cut is smaller than specified amount q.)
Point D1, D2	: Overlap is valid. (When bit 2 (DRV) of parameter No.1681 is set to 1, overlap is invalid.) (When overlap is done at point D1 and point D2, the direction of movement is reversed before the tool reaches point R.)
Point E1, E2	: Overlap is valid. (When bit 3 (DFW) of parameter No.1681 is set to 1, overlap is invalid.)
Point F	: Overlap is valid. (When bit 5 (DZL) of parameter No.1681 is set to 1, overlap is invalid.)
Point G	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point G, no overlap is done at point G.)

G83 (G98)



G83 (G99)

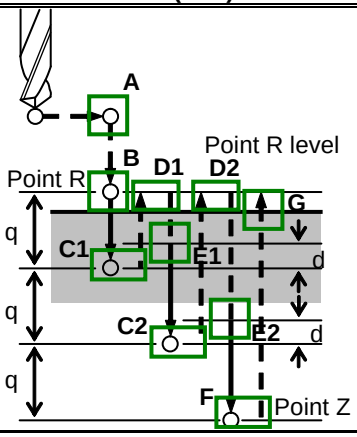


Fig.5.2 (d) Peck Drilling Cycle (G83)

Boring Cycle (G85)	
Point A	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.)
Point B	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point C	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point D	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point D, no overlap is done at point D.)
Point E	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 or DPS(No.1681#0) is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point E, no overlap is done at point E.)

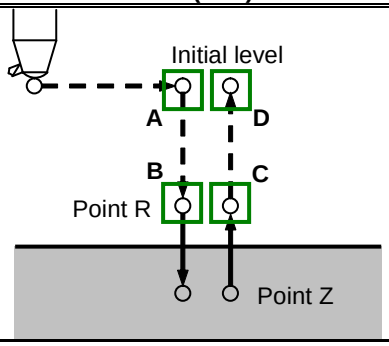
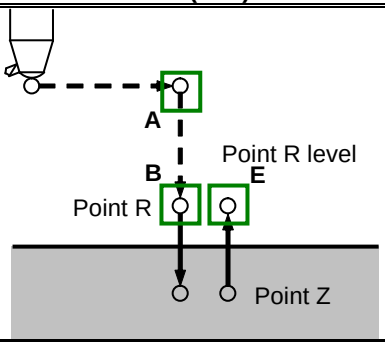
G85 (G98)	G85 (G99)
	

Fig.5.2 (e) Boring Cycle (G85)

Boring Cycle (G86)	
Point A	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.)
Point B	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)

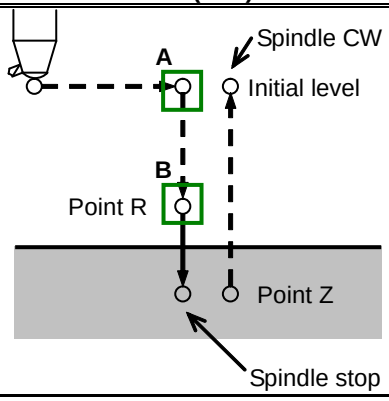
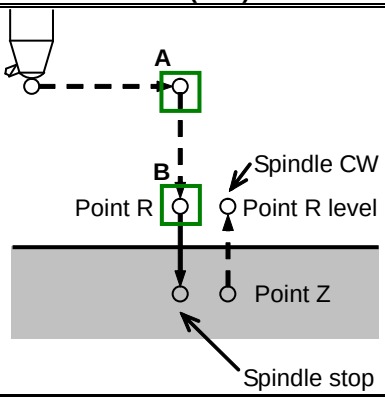
G86 (G98)	G86 (G99)
	

Fig.5.2 (f) Boring Cycle (G86)

Boring Cycle (G89)	
P	: Dwell time
Point A	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.)
Point B	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point C	: Overlap is valid. (When bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.)
Point D	: Overlap is valid. (When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point D, no overlap is done at point D.)
Point E	: Overlap is valid. (When bit 0 (DPS) or bit 1 (DRL) of parameter No.1681 is set to 1, overlap is invalid.) (When canned cycle for drilling is canceled at point E, no overlap is done at point E.)

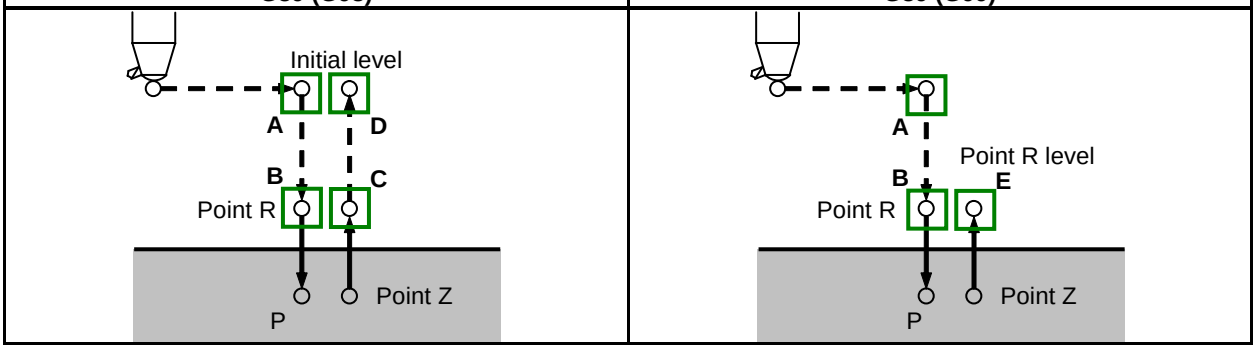


Fig.5.2 (g) Boring Cycle (G89)

5.3 RIGID TAPPING

The tapping cycle (G84) and left-handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode.
In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using auxiliary functions M03 (rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping.
In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle.
When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.
Rigid mode eliminates the need to use a floating tap required in the standard tapping mode, thus allowing faster and more precise tapping.

5.3.1 Rigid Tapping (G84)

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

Format

The rigid tapping mode can be specified using one of three methods.

- Specification of M29S_ in a tapping block.

G84 X_ Y_ Z_ R_ P_ F_ K_ M29 S_ ;

- Specification of M29S_ before the tapping block.

M29 S_ ;

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

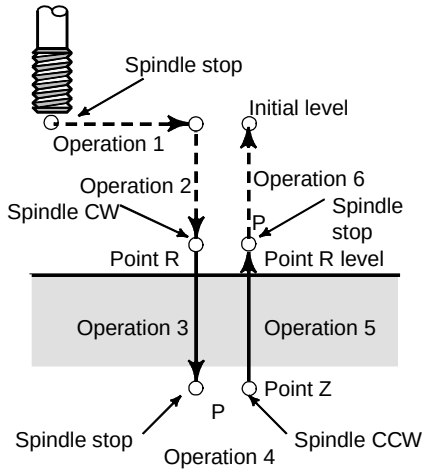
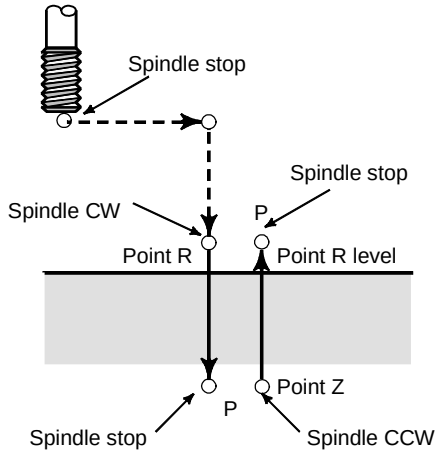
- Enabling rigid tapping to be performed without specifying M29S_ (bit 0 (G84) of parameter No. 5200 set to 1).

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of the hole and at point R when a return is made
F_ : Cutting feedrate
K_ : Number of repeats (if required)

G84.2 X_ Y_ Z_ R_ P_ F_ L_ ;

(Series 15 format)
L_ : Number of repeats (if required)

G84 (G98)	G84 (G99)
	

Explanation

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Tapping is performed from point R to point Z. When tapping is completed, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed.
While tapping is being performed, the feedrate override and spindle override are assumed to be 100%. Feedrate override can be enabled by setting, however.

- Thread lead

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

- Tool length compensation

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

- FANUC Series 15 format command

Rigid tapping can be performed using FANUC Series 15 format commands. The rigid tapping sequence (including data transfer to and from the PMC), Limitation, and the like are the same as described in this chapter.

- 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 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), 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 (G74).
When G84 (G74) 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 operation is performed during rigid tapping, rigid tapping is stopped and G code is set as Table 5.3.1 (a) according to bit 5 (CLR) of parameter No.3402, bit 1 (C09) of No.3407. In restarting operation with G84/G74, command G80 if necessary.

Table 5.3.1 (a)

CLR (bit 5 of parameter No.3402)	C09 (bit 1 of parameter No.3407)	G code
0	-	G84/G74
1	0	G80
1	1	G84/G74

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

Feed hold and single block are disabled in rigid tapping. When bit 6 (FHD) of parameter No.5200 is 1, feed hold and single block are enabled.

- Feed hold
  - Bit 6 (FHD) of parameter No.5200 = 0
    If feed hold is applied between Operation 3 and Operation 5, the feed hold lamp turns on immediately, but the tool decelerates and stops after moving to Operation 6. If feed hold is applied during Operation 1, Operation 2, and Operation 6, CNC becomes feed hold state and the tool decelerates and stops.
  - Bit 6 (FHD) of parameter No.5200 = 1
    If feed hold is applied between Operation 1 and Operation 6, CNC becomes feed hold state and the tool decelerates and stops.
- Single block
  - Bit 6 (FHD) of parameter No.5200 = 0
    Single block stop points are the end points of Operation 1, Operation 2, and Operation 6.
  - Bit 6 (FHD) of parameter No.5200 = 1
    Single block stop points are the end points of each Operation.

- Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle" in Operator’s manual (Common to Lathe system/Machining center system) / B-64694EN.
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 regardless of whether the rigid tap.

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 command

- 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 as follows (displayed on diagnosis data No. 451):
- For a serial spindle: 32,767 pulses per 8 ms

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

If a value exceeding the upper limit of cutting feedrate is specified, alarm PS0011 is issued.

- Unit of F command

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

- 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.

- 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 or G60 (when the bit 0 (MDL) of parameter No. 5431 is set to 1)) and G84 (G74) in a single block. Otherwise, G84 (G74) will be canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Program restart

A program cannot be restarted during rigid tapping.

- Subprogram call

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

- Constant surface speed control

If rigid tapping is commanded during constant surface speed control, alarm (PS0200), ”ILLEGAL S CODE COMMAND” is issued. Command rigid tapping after canceling constant surface speed control.

- Positioning by optimum accelerations

Positioning by optimum accelerations is disabled during rigid tapping.

Example

Z-axis feedrate 1000 mm/min
Spindle speed 1000 min^{-1}
Thread lead 1.0 mm

```
<Programming of feed per minute>
G94;                    Specify a feed-per-minute command.
G00 X120.0 Y100.0 ;     Positioning
M29 S1000 ;             Rigid mode specification
G84 Z-100.0 R-20.0 F1000 ;  Rigid tapping
<Programming of feed per revolution>
G95 ;                   Specify a feed-per-revolution command.
G00 X120.0 Y100.0 ;     Positioning
M29 S1000 ;             Rigid mode specification
G84 Z-100.0 R-20.0 F1.0 ;   Rigid tapping
```

5.3.2 Left-Handed Rigid Tapping Cycle (G74)

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

Format

The rigid tapping mode can be specified using one of three methods.

- Specification of M29S_ in a tapping block.

G74 X_ Y_ Z_ R_ P_ F_ K_ M29 S_ ;

- Specification of M29S_ before the tapping block.

M29 S_ ;

G74 X_ Y_ Z_ R_ P_ F_ K_ ;

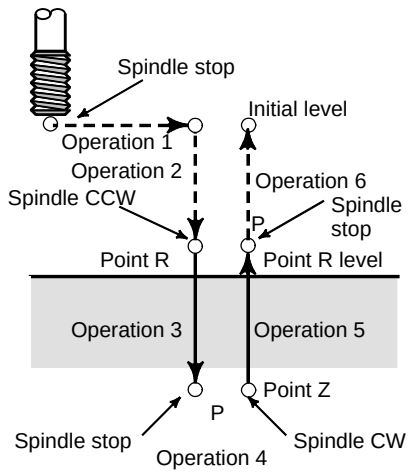
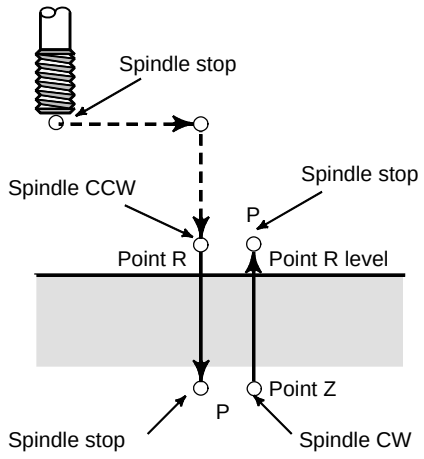
- Enabling rigid tapping to be performed without specifying M29S_ (bit 0 (G84) of parameter No. 5200 set to 1).

G74 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of the hole and at point R when return is made.
F_ : Cutting feedrate
K_ : Number of repeats (if required)

G84.3 X_ Y_ Z_ R_ P_ F_ L_ ;

(Series 15 format)
L_ : Number of repeats (if required)

G74 (G98)	G74 (G99)
	

Explanation

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Tapping is performed from point R to point Z. When tapping is completed, the spindle is stopped and a dwell is performed. The spindle is then rotated in the normal direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed.
While tapping is being performed, the feedrate override and spindle override are assumed to be 100%. Feedrate override can be enabled by setting, however.

- Thread lead

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

- Tool length compensation

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

- FANUC Series 15 format command

Rigid tapping can be performed using FANUC Series 15 format commands. The rigid tapping sequence (including data transfer to and from the PMC), Limitation, and the like are the same as described in this chapter.

- 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 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), 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 (G74).
When G84 (G74) 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 operation is performed during rigid tapping, rigid tapping is stopped and G code is set as Table 5.3.2 (a) according to bit 5 (CLR) of parameter No.3402, bit 1 (C09) of No.3407. In restarting operation with G84/G74, command G80 if necessary.

Table 5.3.2 (a)

CLR (bit 5 of parameter No.3402)	C09 (bit 1 of parameter No.3407)	G code
0	-	G84/G74
1	0	G80
1	1	G84/G74

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

Feed hold and single block are disabled in rigid tapping. When bit 6 (FHD) of parameter No.5200 is 1, feed hold and single block are enabled.

- Feed hold
  - Bit 6 (FHD) of parameter No.5200 = 0
    If feed hold is applied between Operation 3 and Operation 5, the feed hold lamp turns on immediately, but the tool decelerates and stops after moving to Operation 6. If feed hold is applied during Operation 1, Operation 2, and Operation 6, CNC becomes feed hold state and the tool decelerates and stops.
  - Bit 6 (FHD) of parameter No.5200 = 1
    If feed hold is applied between Operation 1 and Operation 6, CNC becomes feed hold state and the tool decelerates and stops.
- Single block
  - Bit 6 (FHD) of parameter No.5200 = 0
    Single block stop points are the end points of Operation 1, Operation 2, and Operation 6.
  - Bit 6 (FHD) of parameter No.5200 = 1
    Single block stop points are the end points of each Operation.

- 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 regardless of whether the rigid tap.

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 command

- Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm PS0200.
- 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 as follows (displayed on diagnosis data No. 451):

- For a serial spindle: 32,767 pulses per 8 ms

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 that exceeds the upper limit of cutting feedrate causes alarm PS0011.

- Unit of F command

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

- M29

Specifying an S command or axis movement between M29 and G84 causes alarm PS0203.
Then, specifying M29 in the tapping cycle causes alarm PS0204.

- 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 or G60 (when the bit 0 (MDL) of parameter No. 5431 is set to 1)) and G74 in a single block. Otherwise, G74 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.

- Constant surface speed control

If rigid tapping is commanded during constant surface speed control, alarm (PS0200), ”ILLEGAL S CODE COMMAND” is issued. Command rigid tapping after canceling constant surface speed control.

- Positioning by optimum accelerations

Positioning by optimum accelerations is disabled during rigid tapping.

Example

Z-axis feedrate	1000 mm/min
Spindle speed	1000 min^{-1}
Thread lead	1.0 mm
<Programming for feed per minute>	
G94 ;	Specify a feed-per-minute command.
G00 X120.0 Y100.0 ;	Positioning
M29 S1000 ;	Rigid mode specification
G74 Z-100.0 R-20.0 F1000 ;	Rigid tapping
<Programming for feed per revolution>	
G95 ;	Specify a feed-per-revolution command.
G00 X120.0 Y100.0 ;	Positioning
M29 S1000 ;	Rigid mode specification
G74 Z-100.0 R-20.0 F1.0 ;	Rigid tapping

5.3.3 Peck Rigid Tapping Cycle (G84 or G74)

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

Format

G84 (or G74) X_ Y_ Z_ R_ P_ Q_ ,D_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of the hole and at point R when a return is made
Q_	:	Depth of cut for each cutting feed
,D_	:	Clearance
F_	:	The cutting feedrate
K_	:	Number of repeats (if required)

G84.2 (or G84.3) X_ Y_ Z_ R_ P_ Q_ ,D_ F_ L_ ;

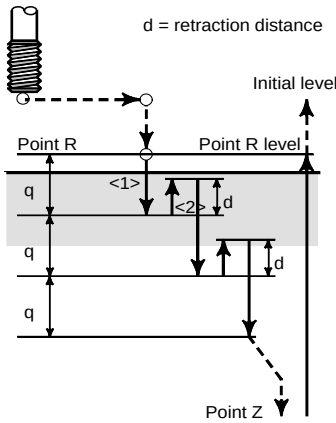
(Series 15 format)

L_	:	Number of repeats (if required)

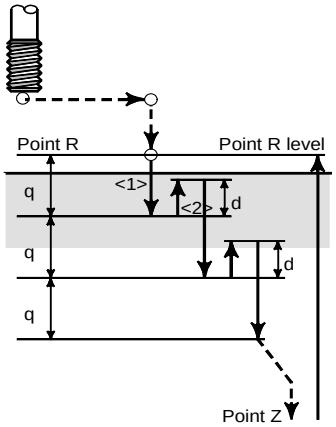
- High-speed peck tapping cycle (Bit 5 (PCP) of parameter No. 5200=0)
  - <1> The tool operates at a normal cutting feedrate. The normal time constant is used.
  - <2> Retraction can be overridden. The retraction time constant is used.

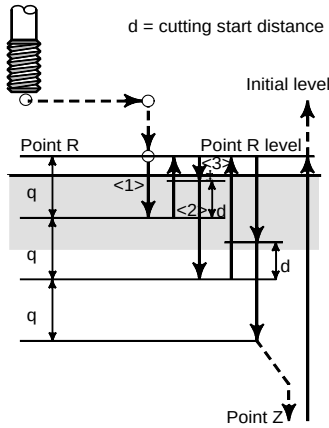
- Peck tapping cycle (Bit 5 (PCP) of parameter No. 5200=1)
  - <1> The tool operates at a normal cutting feedrate. The normal time constant is used.
  - <2> Retraction can be overridden. The retraction time constant is used.
  - <3> Retraction can be overridden. The normal time constant is used.

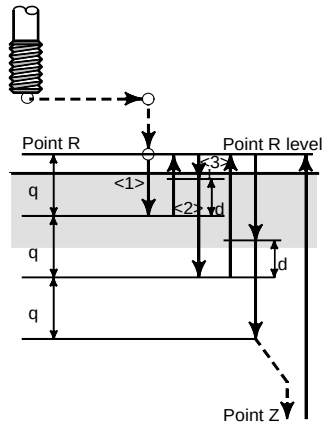
G84, G74 (G98)



G84, G74 (G99)







Explanation

- High-speed peck tapping cycle

After positioning along the X- and Y-axes, rapid traverse is performed to point R. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d. The bit 4 (DOV) of parameter No. 5200 specifies whether retraction can be overridden or not. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.
Set the retraction distance, d, in ",D" command or parameter No. 5213.

- Peck tapping cycle

After positioning along the X- and Y-axes, rapid traverse is performed to point R level. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then a return is performed to point R. The bit 4 (DOV) of parameter No. 5200 specifies whether the retraction can be overridden or not. The moving of cutting feedrate F is performed from point R to a position distance d from the end point of the last cutting, which is where cutting is restarted. For this moving of cutting feedrate F, the specification of the bit 4 (DOV) of parameter No. 5200 is also valid. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.
Set d (distance to the point at which cutting is started) in ",D" command or parameter No. 5213.

- 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 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), 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 (G74).
When G84 (G74) 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 operation is performed during rigid tapping, rigid tapping is stopped and G code is set as Table 5.3.3 (a) according to bit 6 (CLR) of parameter No.3402, bit 1 (C09) of No.3407. In restarting operation with G84/G74, command G80 if necessary.

Table 5.3.3 (a)

CLR (bit 6 of parameter No.3402)	C09 (bit 1 of parameter No.3407)	G code
0	-	G84/G74
1	0	G80
1	1	G84/G74

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

Feed hold and single block are disabled in rigid tapping. When bit 6 (FHD) of parameter No.5200 is 1, feed hold and single block are enabled.

- Feed hold
  - Bit 6 (FHD) of parameter No.5200 = 0
    If feed hold is applied during tapping, the feed hold lamp turns on immediately, but the tool decelerates and stops after retraction to initial level (G98) or point R level (G99). If feed hold is applied during positioning to tapping position, positioning from initial level to point R level, and retraction from point R level to initial level, CNC becomes feed hold state and the tool decelerates and stops.
  - Bit 6 (FHD) of parameter No.5200 = 1
    If feed hold is applied from positioning to tapping position to retraction to initial level (G98) (point R level (G99)), CNC becomes feed hold state and the tool decelerates and stops.
- Single block
  - Bit 6 (FHD) of parameter No.5200 = 0
    Single block stop points are the end points of positioning to tapping position, positioning from initial level to point R level, and retraction from point R level to initial level (G98) (point R level (G99)).
  - Bit 6 (FHD) of parameter No.5200 = 1
    Single block stop points are the end points of each operation.

- 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 regardless of whether the rigid tap.

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 command

- Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm PS0200.
- 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 as follows (displayed on diagnosis data No. 451):

- For a serial spindle: 32,767 pulses per 8 ms
  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 that exceeds the upper limit of cutting feedrate causes alarm PS0011.

- Unit of F command

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

- M29

Specifying an S command or axis movement between M29 and G84 causes alarm PS0203.
Then, specifying M29 in the tapping cycle causes alarm PS0204.

- 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 group 01 G code (G00 to G03 or G60 (when the bit 0 (MDL) of parameter No. 5431 is set to 1)) and G84 (G74) in the same block. If they are specified together, G84 (G74) is 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.

- Amount of return and cutting start distance

Set the amount of return and the cutting start distance (No. 5213) so that point R is not exceeded.

- Constant surface speed control

If rigid tapping is commanded during constant surface speed control, alarm (PS0200), ”ILLEGAL S CODE COMMAND” is issued. Command rigid tapping after canceling constant surface speed control.

- Positioning by optimum accelerations

Positioning by optimum accelerations is disabled during rigid tapping.

Example

Z-axis feedrate 1000 mm/min
Spindle speed 1000 min^{-1}
Thread lead 1.0 mm
Depth of cut 20.0 mm
Amount of retraction or cutting start distance 10.0 mm (set by parameter No.5213)

```
<Programming for feed per minute>
G94 ;                          Specify a feed-per-minute command.
G00 X120.0 Y100.0 ;            Positioning
M29 S1000 ;                    Rigid mode specification
G74 Z-100.0 R-20.0 Q20.0 F1000 ; Rigid tapping
<Programming for feed per revolution>
G95 ;                          Specify a feed-per-revolution command.
G00 X120.0 Y100.0 ;            Positioning
M29 S1000 ;                    Rigid mode specification
G74 Z-100.0 R-20.0 Q20.0 F1.0 ; Rigid tapping
```

5.3.4 Canned Cycle Cancel (G80)

The rigid tapping canned cycle is canceled. For how to cancel this cycle, see the Subsection 5.1.14, "Canned Cycle Cancel for Drilling (G80)."

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.

5.3.5 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

5.3.5.1 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%.

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

The override to be applied is determined according to the setting of parameters and that in the command as shown in the Table 5.3.5.1 (a).

Table 5.3.5.1 (a)

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%
	Outside the range between 100% to 200%	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 between 100% to 200% is assumed.

3 The maximum override is obtained using the following equation so that the spindle speed to which override at extraction is applied does 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.

5.3.5.2 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
  (When the second feedrate override signal turns “1”, the second feedrate override is applied to the feedrate to which feedrate override is applied.)
- 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 does 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.

5.4 OPTIONAL CHAMFERING AND CORNER R

Overview

Chamfering and corner R blocks can be inserted automatically between the following:

- Between linear interpolation and linear interpolation blocks
- Between linear interpolation and circular interpolation blocks
- Between circular interpolation and linear interpolation blocks
- Between circular interpolation and circular interpolation blocks

It is possible to use this function with AI contour control.

Format

, C_ Chamfering

, R_ Corner R

Explanation

When the above specification is added to the end of a block that specifies linear interpolation (G01) or circular interpolation (G02 or G03), a chamfering or corner R block is inserted.
Blocks specifying chamfering and corner R can be specified consecutively.

- Chamfering

After C, specify the distance from the hypothetical corner intersection to the start and end points. The hypothetical corner point is the corner point that would exist if chamfering were not performed.

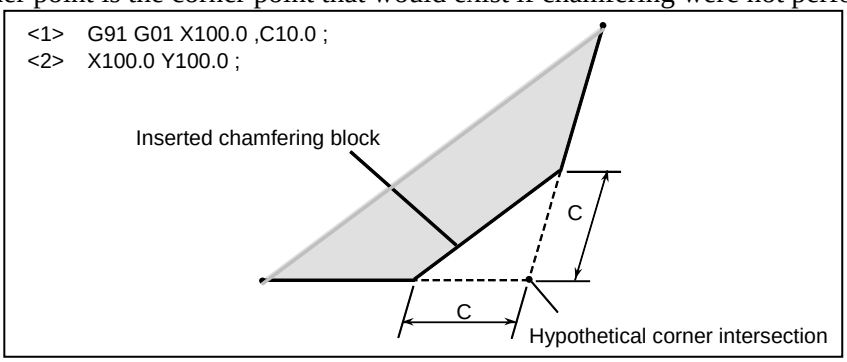


Fig. 5.4 (a) Chamfering

- Corner R

After R, specify the radius for corner R.

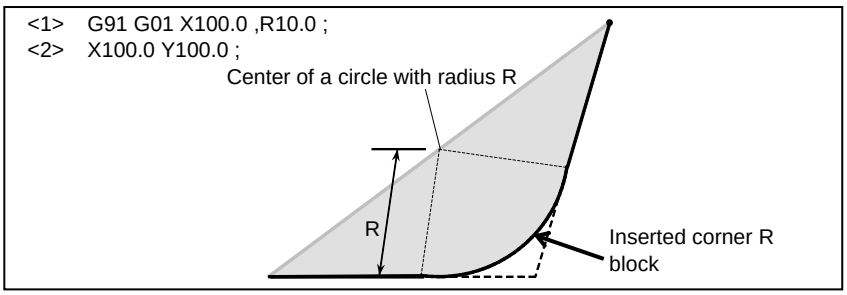


Fig. 5.4 (b) Corner R

Example

```
N001 G92 G90 X0 Y0 ;
N002 G00 X10.0 Y10.0 ;
N003 G01 X50.0 F10.0 ,C5.0 ;
N004 Y25.0 ,R8.0 ;
N005 G03 X80.0 Y55.0 R30.0 ,R8.0 ;
N006 G01 X50.0 ,R8.0 ;
N007 Y70.0 ,C5.0 ;
N008 X10.0 ,C5.0 ;
N009 Y10.0 ;
N010 G00 X0 Y0 ;
N011 M0;
```

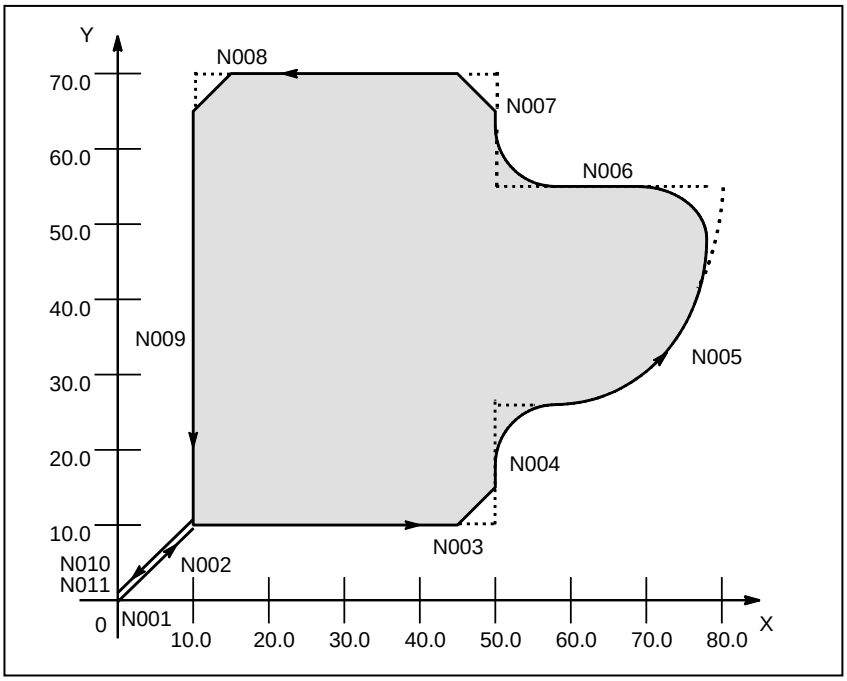


Fig.5.4 (c) Example

Limitation

- Invalid specification

Chamfering (,C) or corner R (,R) specified in a block other than a linear interpolation (G01) or circular interpolation (G02 or G03) block is ignored.

- Next block

A block specifying chamfering or corner R must be followed by a block that specifies a move command using linear interpolation (G01) or circular interpolation (G02 or G03). If the next block does not contain these specifications, alarm PS0051 is issued.
Between these blocks, however, only one block specifying G04 (dwell) can be inserted. The dwell is executed after execution of the inserted chamfering or corner R block.

- Exceeding the move range

If the inserted chamfering or corner R block causes the tool to go beyond the original interpolation move range, alarm PS0055 is issued.

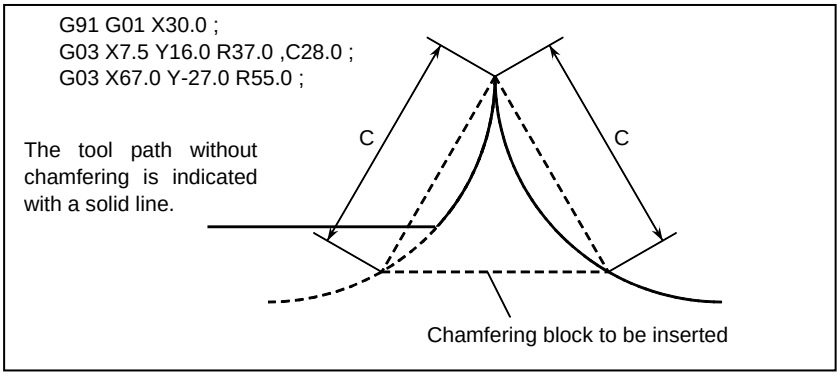


Fig 5.4 (d) Exceeding the move range

- Plane selection

A chamfering or corner R block is inserted only for a command to move the tool within the same plane.
Example:

When the U-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 U-axis and that along the Y-axis:

```
G17 U0 Y0
G00 U100.0 Y100.0
G01 U200.0 F100 ,C30.0
Y200.0
```

The following program causes alarm PS0055, however. (Because chamfering is specified in the block to move the tool along the X-axis, which is not on the selected plane)

```
G17 U0 Y0
G00 U100.0 Y100.0
G01 X200.0 F100 ,C30.0
Y200.0
```

The following program also causes alarm PS0055. (Because the block next to the chamfering command moves the tool along the X-axis, which is not on the selected plane)

```
G17 U0 Y0
G00 U100.0 Y100.0
G01 Y200.0 F100 ,C30.0
X200.0
```

If 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 is issued.

- Travel distance 0

When two linear interpolation operations are performed, the chamfering or corner R block is regarded as having a travel distance of zero if the angle between the two straight lines is within ±1°.
When linear interpolation and circular interpolation operations are performed, the corner R block is regarded as having a travel distance of zero if the angle between the straight line and the tangent to the arc at the intersection is within ±1°. When two circular interpolation operations are performed, the corner R block is regarded as having a travel distance of zero if the angle between the tangents to the arcs at the intersection is within ±1°.

- Macro executer

Optional angle chamfering and corner rounding specified in the execution macro is disabled.

- 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.

NOTE
1 When ",C" and ",R" are specified in the same block, the address specified last is valid.
2 If ",C" or ",R" is specified in a thread cutting command block, alarm PS0050 is issued.

5.5 INDEX TABLE INDEXING FUNCTION

By specifying indexing positions (angles) for one rotation axis (usually referred to as axis A, B, or C), the index table of the machining center can be indexed.
Before and after indexing, the index table is automatically unclamped or clamped.

NOTE
When bit 3 (IXC) of parameter No.8132 is 1, this function can be used.

Explanation

- Indexing position

Specify an indexing position with address A, B, or C.
The indexing position is specified by either of the following (depending on bit 4 of parameter G90 No.5500):
1. Absolute value only
2. Absolute or incremental value depending on the specified G code: G90 or G91

A positive value indicates an indexing position in the counterclockwise direction. A negative value indicates an indexing position in the clockwise direction.
The minimum indexing angle of the index table is the value set to parameter 5512. Only multiples of the least input increment can be specified as the indexing angle. If any value that is not a multiple is specified, an alarm PS1561 occurs. Decimal fractions can also be entered. When a decimal fraction is entered, the 1's digit corresponds to degree units.

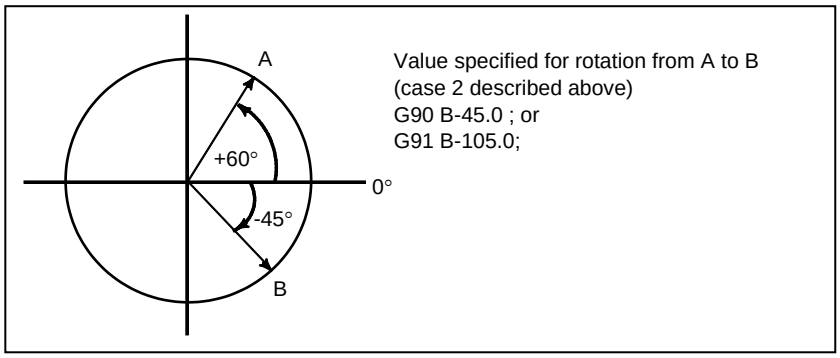


Fig.5.5 (a) Indexing position

- Direction and value of rotation

The direction of rotation and angular displacement are determined by either of the following two methods. Refer to the manual written by the machine tool builder to find out which method is applied.

1. Using the auxiliary function specified in parameter No. 5511 (Address) (Indexing position) (Miscellaneous function); Rotation in the negative direction (Address) (Indexing position); Rotation in the positive direction (No auxiliary functions are specified.)

   An angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° when bit 2 (ABS) of parameter No. 5500 specifies this option.

   For example, when G90 B400.0 (auxiliary function); is specified at a position of 0, the table is rotated by 40° in the negative direction.
2. Using no auxiliary functions

   By setting to bits 2 (ABS), 3 (INC), and 4 (G90) of parameter No. 5500, operation can be selected from the following two options.

   Select the operation by referring to the manual written by the machine tool builder.

   (1) Rotating in the direction in which an angular displacement becomes shortest

   This is valid only in absolute programming. A specified angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° when bit 2 (ABS) of parameter No. 5500 specifies this option.

   For example, when G90 B400.0; is specified at a position of 0, the table is rotated by 40° in the positive direction.

   (2) Rotating in the specified direction

   In the absolute programming, the value set in bit 2 (ABS) of parameter No. 5500 determines whether an angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360°.

   In the incremental programming, the angular displacement is not rounded down. For example, when G90 B720.0; is specified at a position of 0, the table is rotated twice in the positive direction, when the angular displacement is not rounded down.

- Feedrate

The table is always rotated around the control axis (a rotation axis) used for index table indexing (hereafter called the index table indexing axis) in the rapid traverse mode.

Dry runs cannot be executed for the index table indexing axis.

⚠ CAUTION

1 If a reset is made during indexing of the index table, a reference position return must be made before each time the index table is indexed subsequently.

2 For a path on which the index table indexing function is not to be used, disable the index table indexing function (set bit 0 (ITI) of parameter No. 5501 to 0).

NOTE

1 If an index table indexing axis and another controlled axis are specified in the same block either alarm PS1564 is issued or the command is executed, depending on bit 6 (SIM) of parameter No. 5500 and bit 0 (IXS) of parameter No. 5502.

2 The auxiliary function specifying a negative direction is processed in the CNC. The relevant M code signal and completion signal are sent between the CNC and the machine.

3 If a reset is made while waiting for completion of clamping or unclamping, the clamp or unclamp signal is cleared and the CNC exits the completion wait state.

- Index table indexing function and other functions

Table 5.5 (a) Index table indexing function and other functions

Item	Explanation
Relative position display	This value is rounded down when bit 1 of parameter REL No.5500 specifies this option.
Absolute position display	This value is rounded down when bit 2 (ABS) of parameter No. 5500 specifies this option.
Single direction positioning	Impossible to specify
2nd auxiliary function (B code)	Possible with any address other than B that of the index table indexing axis.
Operations during index table indexing axis movement	Unless otherwise processed by the machine, feed hold, interlock and emergency stop can be executed. Machine lock can be executed after indexing is completed.
SERVO OFF signal	Disabled The index table indexing axis is usually in the servo-off state.
Incremental commands for an index table indexing axis	The workpiece coordinate system and machine coordinate system must always agree with each other on the index table indexing axis (the workpiece zero point offset value is zero.).
Operations for an index table indexing axis	Manual operation is disabled in the JOG, INC, or HANDLE mode. A manual reference position return can be made. If the axis selection signal is set to zero during manual reference position return, movement is stopped and the clamp command is not executed.
Pole position detection function	This function cannot be used on an axis on which the pole position detection function is used.

5.6 IN-FEED CONTROL (FOR GRINDING MACHINE)

Overview

Each time an external signal is input when the machine is at a table swing end point, the machine makes a cut by a constant amount along the programmed profile on the specified YZ plane. This makes it possible to perform grinding and cutting in a timely manner and facilitating the grinding of a workpiece with a profile.

NOTE

This function is included in the option "Grinding function A" and "Grinding function B".
To use this function, any one of the above option is required.

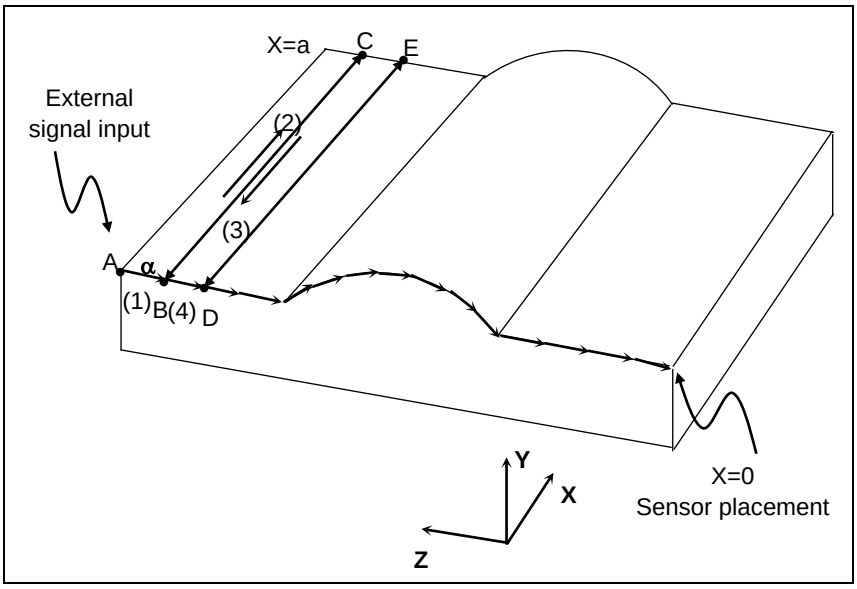


Fig. 5.6 (a)

For example, it is possible to machine a workpiece with a profile programmed with linear interpolation, circular interpolation, and linear interpolation on the YZ plane, such as that shown in the Fig. 5.6 (a).
A sensor is placed at an X = 0 position so that the external signal is input when the sensor detects the grinding wheel. When the program is started at point A, the machine is first placed in the state in which it waits for the input of the external signal.
Then, when the sensor detects the grinding wheel, the external signal is input, and the machine makes a cut by the constant amount α along the programmed profile on the specified YZ plane and moves to point B (operation (1)). The machine is then placed in the state in which it waits for the input of the external signal again, and performs a grinding operation along the X-axis. It grinds from point B to point C (operation (2)) and grinds back from point C to point B (operation (3)). When the machine returns to point B, the sensor detects the grinding wheel again, and the external signal is input, so that the machine makes a cut by the amount of α and moves to point D (operation (4)). At point D, the machine performs a grinding operation along the X-axis.
Afterwards, each time the external signal is input, the machine makes a cut by the amount of α along the profile program, so that the workpiece is machined to a profile such as that shown in the Fig. 5.6 (a).

Format

```
G161 R_ ;
  Profile program

G160 ;
```

NOTE
Always specify G160 and G161 in an independent block.
(Do not specify other G codes at the same time.)

Explanation

- G161 R_

This specifies an operation mode and the start of a profile program.
A depth of cut can be specified with R.

- Profile program

Program the profile of a workpiece on the YZ plane, using linear interpolation (G01) or circular interpolation (G02, G03). Multiple-block commands are possible.
When a profile program is started, the machine is placed in the state in which it waits for the input of the external signal. When the external signal is input in this state, the machine makes a cut by the depth of cut specified with R. Later, until the end point of the program, the machine makes a cut each time the external signal is input. If the final depth of cut is less than R, the remaining travel distance is assumed the depth of cut.
The feedrate is the one specified in the program with an F code. As in normal linear interpolation (G01) or circular interpolation (G02, G03), override can be applied.

- G160

This specifies the cancellation of an operation mode (end of a profile program).

Limitation

- G161 R_

If no value is specified with R or if the value specified with R is negative, alarm PS0230 is issued.

- Profile program

In a profile program, do not issue move commands other than those for linear interpolation (G01) and circular interpolation (G02, G03).

> ⚠ **CAUTION**
> If a move command other than those for linear interpolation (G01) and circular interpolation (G02, G03) is issued in a profile program, the specified depth of cut is not correct.

- Grinding operation

In this operation mode, a grinding operation that causes the machine to move to and from the grinding wheel cannot be specified in an NC program. Perform such an operation by PMC axis control etc.

- Block overlap

In this operation mode, block overlap is disabled.

- The external signal

The external signal is disabled when it is input before a profile program is started. Input the external signal after the start of a profile program. Also, even if the external signal is input during a cut, this is not accepted in the next cut. It is necessary to input the signal again after the end of the cut, when the machine is in the state in which it waits for the input of the external signal.

Example

```
O0001 ;
  :
N0 G161 R10.0 ;
N1 G91 G01 Z-70.0 F100 ;
N2 G19 G02 Z-80.0 R67.0 ;
N3 G01 Z-70.0 ;
N4 G160 ;
  :
```

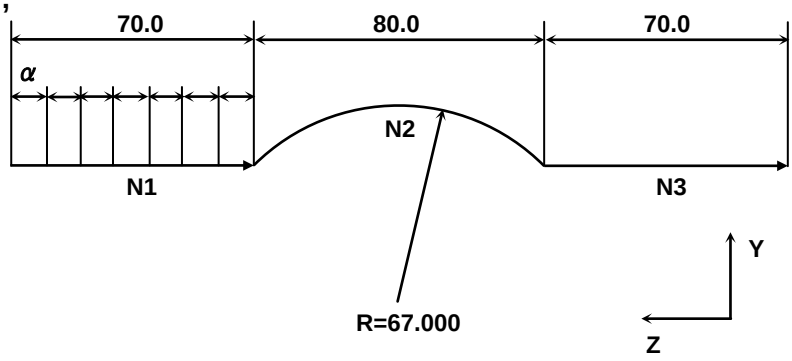


Fig. 5.6 (b)

The program above causes the machine to move by 10.000 along the machining profile in the Fig. 5.6 (b) each time the external signal is input.
α = Travel distance at each input of the external signal.
The feedrate is the one specified in the program with an F code.

Note

NOTE

If manual intervention is performed during in-feed control, the tool path after the manual intervention can be switched by setting the manual absolute switch to on or off as in normal linear/circular interpolation. When the manual absolute switch is on, the machine returns to the programmed path for an absolute command or for an incremental command with bit 1 (ABS) of parameter No. 7001 being 1.

5.7 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:

- Plunge grinding cycle (G75)
- Direct constant-dimension plunge grinding cycle (G77)
- Continuous-feed surface grinding cycle (G78)
- Intermittent-feed surface grinding cycle (G79)

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
Axis on which to make a dresser cut:	Dressing axis

During execution of a canned grinding cycle, the following functions cannot be used:

- Programmable mirror image
- Scaling
- Coordinate system rotation
- 3-dimensional coordinate conversion
- One-digit F code feed
- Tool length compensation

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.

⚠ WARNING

The G codes for canned grinding cycles G75, G77, G78, and G79 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 G75, G77, G78, and G79 needs to be specified. So, when switching to another axis move command from canned grinding cycles, 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 This function is included in the option "Grinding function A" and "Grinding function B".
To use this function, any one of the above option is required.

2 If the G code for a canned grinding cycle (G75, G77, G78, or G79) is specified, the canned grinding cycle is executed according to the values of I, J, K, α, R, F, and P preserved as modal data while the cycle is valid, even if a block specified later specifies none of G75, G77, G78, and G79.
Example:
G75 I_ J_ K_ α_ R_ F_ P_ ;
; ← The canned grinding cycle is executed even if an empty block is specified.
%

3 When switching from a canned cycle for drilling to a canned grinding cycle, specify G80 to cancel the canned cycle for drilling.

4 When switching from a canned grinding cycle to another axis move command, cancel the canned cycle according to the warning above.

5.7.1 Plunge Grinding Cycle (G75)

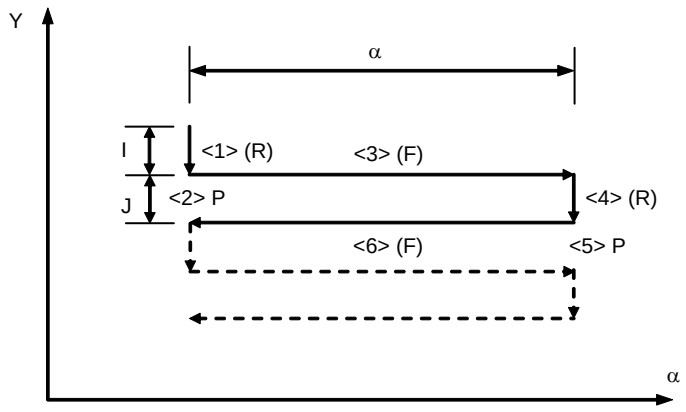
A plunge grinding cycle can be executed.

Format

G75 I_ J_ K_ α_ R_ F_ P_ L_ ;

I_	:	First depth of cut (The cutting direction depends on the sign.)
J_	:	Second depth of cut (The cutting direction depends on the sign.)
K_	:	Total depth of cut (The cutting direction depends on the sign.)
α_	:	Grinding range (The grinding direction depends on the sign.)
R_	:	Feedrate for I and J
F_	:	Feedrate for α
P_	:	Dwell time
L_	:	Grinding-wheel wear compensation number (during continuous dressing only)

G75
G83(G99)



NOTE
α is an arbitrary axis address on the grinding axis as determined with parameter No. 5176.

Explanation

A plunge grinding cycle consists of a sequence of six operations.
Operations <1> to <6> are repeated until the depth of cut reaches the total depth of cut specified with address K. For a single block, operations <1> to <6> are executed with a single cycle start.

- Operation sequence in a cycle

<1> Cutting with a grinding wheel

Makes a cut in the Y-axis direction with cutting feed by the amount specified as the first depth of cut I. The feedrate is the one specified with R.

<2> Dwell

Performs a dwell for the time specified with P.

<3> Grinding

Causes the machine to move with cutting feed by the amount specified with α. The grinding axis is specified with parameter No. 5176. The feedrate is the one specified with F. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis. The dressing axis is specified with parameter No. 5180.

<4> Cutting with a grinding wheel

Makes a cut in the Y-axis direction with cutting feed by the amount specified as the second depth of cut J. The feedrate is the one specified with R.

<5> Dwell

Performs a dwell for the time specified with P.

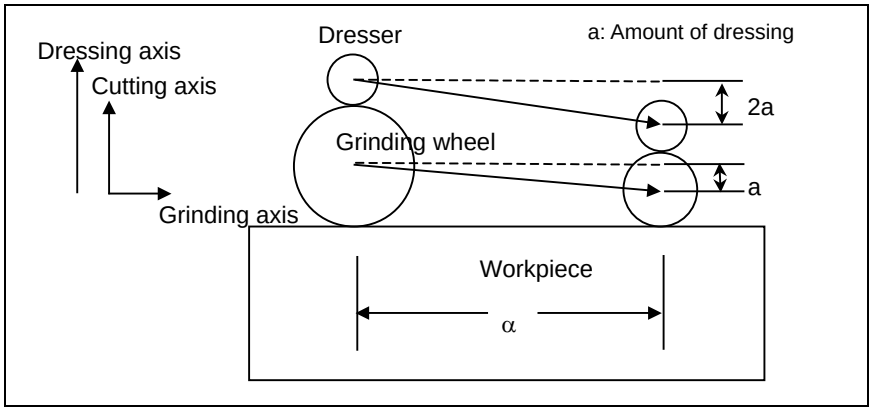
<6> Grinding (return direction)

Feeds the machine at the feedrate specified with F in the opposite direction by the amount specified with α. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis.

- Continuous dressing

If the continuous dressing function is enabled, the grinding-wheel cut and the dresser cut are continuously compensated for according to the dressing amount specified with L during the execution of grinding.
That is, continuous dressing is performed in each grinding operation in the sequence of operations in the cycle, resulting in simultaneous 3-axis interpolation with compensation in the cutting axis direction and compensation in the dressing axis direction simultaneous with movement along the grinding axis. At this time, the travel distance (compensation) along the cutting axis is equal to the specified dressing amount, and the travel distance along the dressing axis is equal to double the specified dressing amount (diameter).
For the dressing amount, specify an offset number with address L. Up to 400 offset numbers (L1 to L400) can be specified. Establish correspondence between compensation amounts and offset numbers, and set it in offset memory in advance, using the MDI unit.
No compensation operation is performed in the following cases:

The continuous dressing function is disabled.
L is not specified.
L0 is specified.



Limitations

- Cutting axis

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

- Grinding axis

To specify a grinding axis, set its axis number, which must be other than that of the cutting axis, in parameter No. 5176.

- Dressing axis

To specify a dressing axis, set its axis number, which must be other than those of the cutting axis and the grinding axis, in parameter No. 5180.

- α,I,J,K

α, I, J, and K commands are all incremental ones.
Spark-out (execution of movement in the grinding direction only) occurs in the following cases:

- I or J is not specified or I = J = 0
- K is not specified or K = 0

If I or J is not specified or if I = J = 0 is true, and K is not equal to 0, a grinding operation is performed infinitely.

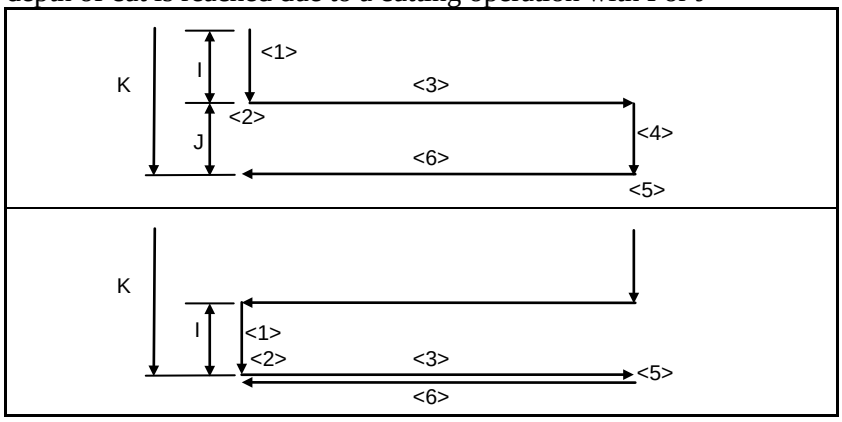
- Clearing

The data items I, J, K, α, R, F, and P in a canned cycle are modal information common to G75, G77, G78, and G79, so that once specified, they remain effective until specified anew. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G75, G77, G78, and G79 is specified. L is effective only in the block in which it is specified.

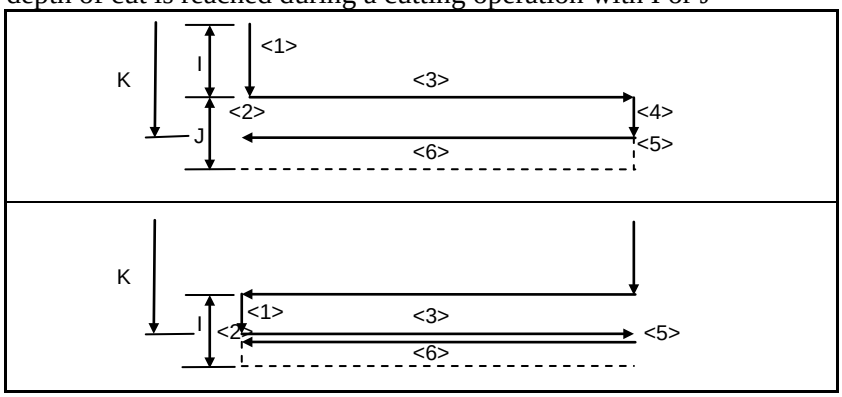
- Operation to be performed if the total depth of cut is reached

If, during cutting with I or J, the total depth of cut is reached, the cycle is ended after the subsequent operations in the sequence (up to <6>) are executed.
If this occurs, the depth of cut is equal to or less than the total depth of cut.

- If the total depth of cut is reached due to a cutting operation with I or J



- If the total depth of cut is reached during a cutting operation with I or J



NOTE
1 If I, J, and K have different signs, alarm PS0455 is issued.
2 If G75 is specified, but a grinding axis is not specified, alarm PS0455 is issued.
3 If any two of the cutting axis number, the grinding axis number, and the dressing axis number are the same, alarm PS0456 is issued.
4 While this cycle is effective, even if G90 (absolute command) is executed, the α, I, J, and K commands are incremental ones.

5.7.2 Direct Constant-Dimension Plunge Grinding Cycle (G77)

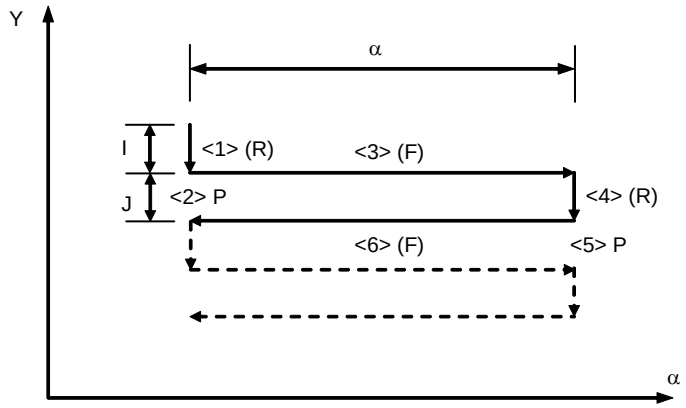
A direct constant-dimension plunge grinding cycle can be performed.

Format

G77 I_ J_ K_ α_ R_ F_ P_ L_ ;

I_	:	First depth of cut (The cutting direction depends on the sign.)
J_	:	Second depth of cut (The cutting direction depends on the sign.)
K_	:	Total depth of cut (The cutting direction depends on the sign.)
α_	:	Grinding range (The grinding direction depends on the sign.)
R_	:	Feedrate for I and J
F_	:	Feedrate for α
P_	:	Dwell time
L_	:	Grinding-wheel wear compensation number (during continuous dressing only)

G77
G83(G99)



NOTE
α is an arbitrary axis address on the grinding axis as determined with parameter No. 5177.

Explanation

A direct constant-dimension plunge grinding cycle consists of a sequence of six operations.
Operations <1> to <6> are repeated until the depth of cut reaches the total depth of cut specified with address K. For a single block, operations <1> to <6> are executed with a single cycle start.

- Operation sequence in a cycle

<1> Cutting with a grinding wheel

Makes a cut in the Y-axis direction with cutting feed by the amount specified as the first depth of cut I. The feedrate is the one specified with R.

<2> Dwell

Performs a dwell for the time specified with P.

<3> Grinding

Causes the machine to move with cutting feed by the amount specified with α. The grinding axis is specified with parameter No. 5177. The feedrate is the one specified with F. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis. The dressing axis is specified with parameter No. 5181.

<4> Cutting with a grinding wheel

Makes a cut in the Y-axis direction with cutting feed by the amount specified as the second depth of cut J. The feedrate is the one specified with R.

<5> Dwell

Performs a dwell for the time specified with P.

<6> Grinding (return direction)

Feeds the machine at the feedrate specified with F in the opposite direction by the amount specified with α. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis.

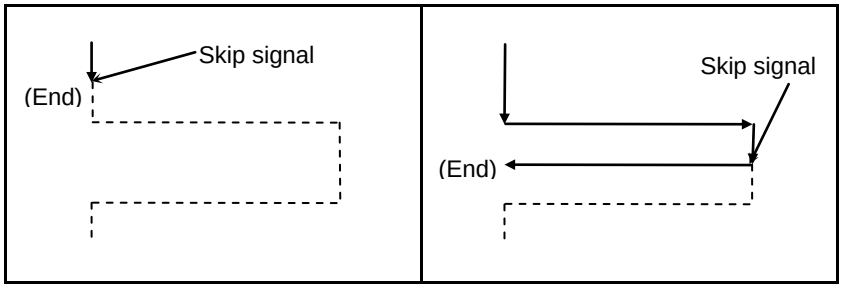
- Continuous dressing

If the continuous dressing function is enabled, the grinding-wheel cut and the dresser cut are continuously compensated for according to the dressing amount specified with L during the execution of grinding. For details, see Explanation of G75.

- Operation to be performed when a skip signal is input

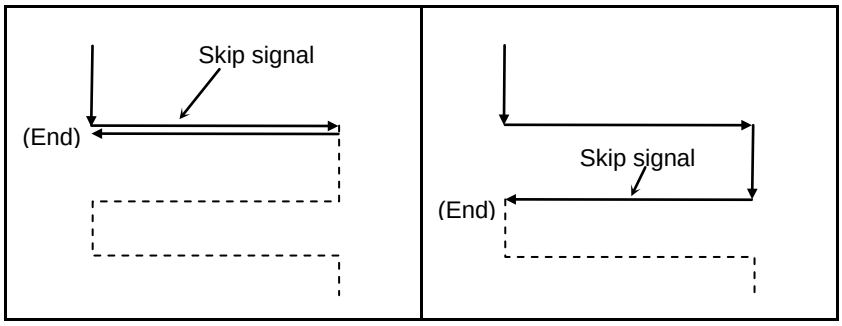
With G77, by inputting a skip signal in a cycle, it is possible to end the cycle after interrupting the current operation sequence (or after ending the current operation sequence).
The following shows the operation to be performed when a skip signal is input in each operation sequence.

- If operation <1> or <4> in the sequence (movement with I or J) is in progress, the machine immediately stops cutting and returns to the α coordinate, assumed at the start of the cycle.



- If operation <2> or <5> in the sequence (dwell) is in progress, the machine immediately cancels the dwell and returns to the α coordinates, assumed at the start of the cycle.

- If operation <3> or <6> in the sequence (grinding movement) is in progress, the machine returns to the α coordinate, assumed at the start of the cycle after the end of the α movement.



Limitations

- Cutting axis

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

- Grinding axis

To specify a grinding axis, set its axis number, which must be other than that of the cutting axis, in parameter No. 5177.

- Dressing axis

To specify a dressing axis, set its axis number, which must be other than those of the cutting axis and the grinding axis, in parameter No. 5181.

- α,I,J,K

α, I, J, and K commands are all incremental ones.
Spark-out (execution of movement in the grinding direction only) occurs in the following cases:

- I or J is not specified or I = J = 0
- K is not specified or K = 0

If I or J is not specified or if I = J = 0 is true, and K is not equal to 0, a grinding operation is performed infinitely.

- Clearing

The data items I, J, K, α, R, F, and P in a canned cycle are modal information common to G75, G77, G78, and G79, so that once specified, they remain effective until specified anew. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G75, G77, G78, and G79 is specified. L is effective only in the block in which it is specified.

- Operation to be performed if the total depth of cut is reached

The operation to be performed if the total depth of cut reaches during cutting with I or J is the same as that for G75. See Limitation on G75.

NOTE

1 If I, J, and K have different signs, alarm PS0455 is issued.
2 If G77 is specified, but a grinding axis is not specified, alarm PS0455 is issued.
3 If any two of the cutting axis number, the grinding axis number, and the dressing axis number are the same, alarm PS0456 is issued.
4 While this cycle is effective, even if G90 (absolute command) is executed, the α, I, J, and K commands are incremental ones.

5.7.3 Continuous-feed Surface Grinding Cycle (G78)

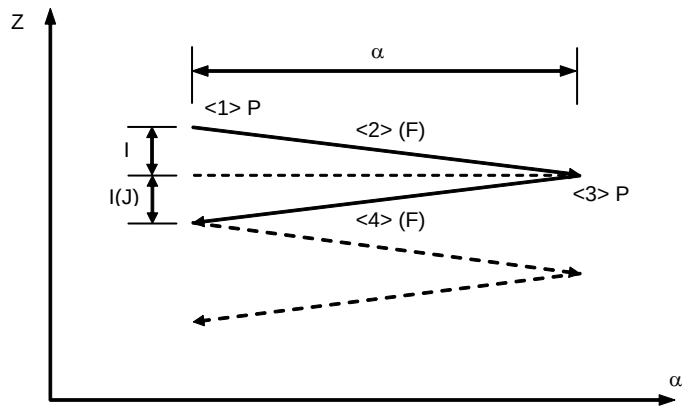
A continuous-feed surface grinding cycle can be performed.

Format

G78 I_ (J_) K_ α_ F_ P_ L_ ;

I_	:	First depth of cut (The cutting direction depends on the sign.)
J_	:	Second depth of cut (The cutting direction depends on the sign.)
K_	:	Total depth of cut (The cutting direction depends on the sign.)
α_	:	Grinding range (The grinding direction depends on the sign.)
F_	:	Feedrate
P_	:	Dwell time
L_	:	Grinding-wheel wear compensation number (during continuous dressing only)

G78
G83(G99)



NOTE
α is an arbitrary axis address on the grinding axis as determined with parameter No. 5178.

Explanation

A continuous-feed surface grinding cycle consists of a sequence of four operations.
Operations <1> to <4> are repeated until the depth of cut reaches the total depth of cut specified with address K. For a single block, operations <1> to <4> are executed with a single cycle start.

- Operation sequence in a cycle

<1> Dwell

Performs a dwell for the time specified with P.

<2> Cutting with a grinding wheel+Grinding

Performs cutting feed along the cutting axis (Z-axis) and the grinding axis at the same time. The travel distance (depth of cut) along the cutting axis is equal to the amount specified as the first depth of cut I, and the travel distance along the grinding axis is equal to the amount specified with α. The grinding axis is specified with parameter No. 5178. The feedrate is the one specified with F. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis. The dressing axis is specified with parameter No. 5182.

<3> Dwell

Performs a dwell for the time specified with P.

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

Performs cutting feed along the cutting axis (Z-axis) and the grinding axis at the same time. The travel distance (depth of cut) along the cutting axis is equal to the amount specified as the first depth of cut I, and the travel distance along the grinding axis is equal to the amount specified with α, with the direction being the opposite one. The feedrate is the one specified with F. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis.

- Continuous dressing

If the continuous dressing function is enabled, the grinding-wheel cut and the dresser cut are continuously compensated for according to the dressing amount specified with L during the execution of grinding. For details, see Explanation of G75.

Limitations

- Cutting axis

The cutting axis is the third controlled axis. By setting bit 0 (FXY) of parameter No. 5101, the axis can be switched with a plane selection command (G17, G18, or G19).

- Grinding axis

To specify a grinding axis, set its axis number, which must be other than that of the cutting axis, in parameter No. 5178.

- Dressing axis

To specify a dressing axis, set its axis number, which must be other than those of the cutting axis and the grinding axis, in parameter No. 5182.

- J

If J is not specified, J is regarded as being equal to I.
The J command is effective only in the block in which it is specified.

- α,I,J,K

α, I, J, and K commands are all incremental ones.
Spark-out (execution of movement in the grinding direction only) occurs in the following cases:
- I or J is not specified or I = J = 0
- K is not specified or K = 0

If I or J is not specified or if I = J = 0 is true, and K is not equal to 0, a grinding operation is performed infinitely.

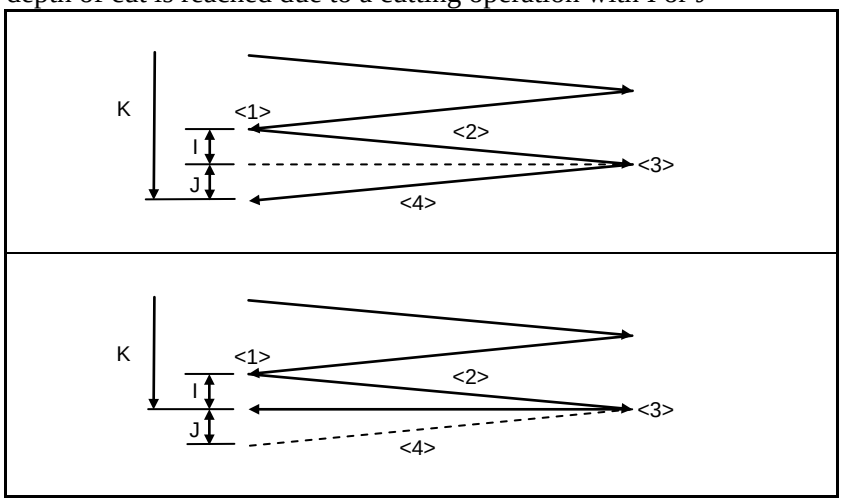
- Clearing

The data items I, K, α, R, F, and P in a canned cycle are modal information common to G75, G77, G78, and G79, so that once specified, they remain effective until specified anew. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G75, G77, G78, and G79 is specified. J, L is effective only in the block in which it is specified.

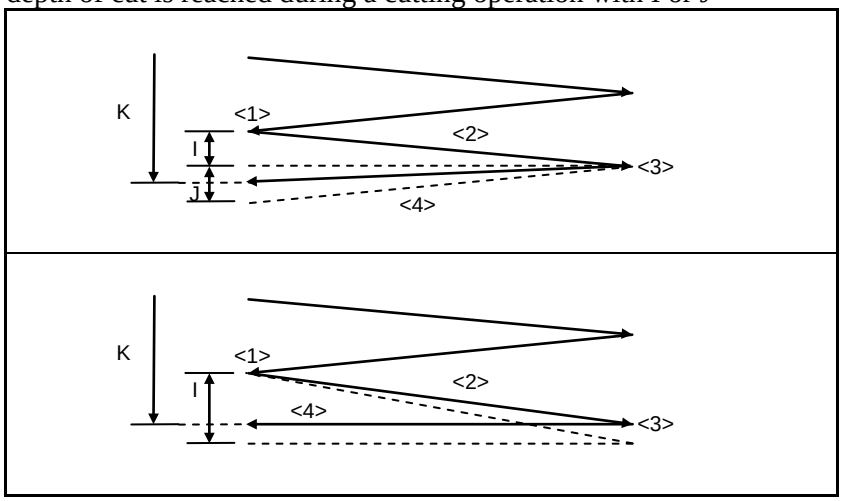
- Operation to be performed if the total depth of cut is reached

If, during cutting with I or J, the total depth of cut is reached, the cycle is ended after the subsequent operations in the sequence (up to <4>) are executed.
If this occurs, the depth of cut is equal to or less than the total depth of cut.

- If the total depth of cut is reached due to a cutting operation with I or J



- If the total depth of cut is reached during a cutting operation with I or J



NOTE

1 If I, J, and K have different signs, alarm PS0455 is issued.
2 If G78 is specified, but a grinding axis is not specified, alarm PS0455 is issued.
3 If any two of the cutting axis number, the grinding axis number, and the dressing axis number are the same, alarm PS0456 is issued.
4 While this cycle is effective, even if G90 (absolute command) is executed, the α, I, J, and K commands are incremental ones.

5.7.4 Intermittent-feed Surface Grinding Cycle (G79)

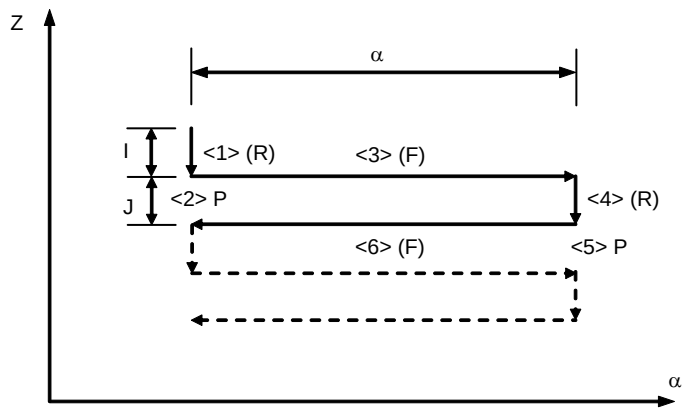
An intermittent-feed surface grinding cycle can be performed.

Format

G79 I_ J_ K_ α_ R_ F_ P_ L_ ;

I_	:	First depth of cut (The cutting direction depends on the sign.)
J_	:	Second depth of cut (The cutting direction depends on the sign.)
K_	:	Total depth of cut (The cutting direction depends on the sign.)
α_	:	Grinding range (The grinding direction depends on the sign.)
R_	:	Feedrate for I and J
F_	:	Feedrate for α
P_	:	Dwell time
L_	:	Grinding-wheel wear compensation number (during continuous dressing only)

G79
G83(G99)



NOTE
α is an arbitrary axis address on the grinding axis as determined with parameter No. 5179.

Explanation

An intermittent-feed surface grinding cycle consists of a sequence of six operations.
Operations <1> to <6> are repeated until the depth of cut reaches the total depth of cut specified with address K. For a single block, operations <1> to <6> are executed with a single cycle start.

- Operation sequence in a cycle

<1> Cutting with a grinding wheel

Makes a cut in the Z-axis direction with cutting feed by the amount specified as the first depth of cut I. The feedrate is the one specified with R.

<2> Dwell

Performs a dwell for the time specified with P.

<3> Grinding

Causes the machine to move with cutting feed by the amount specified with α. The grinding axis is specified with parameter No. 5179. The feedrate is the one specified with F. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis. The dressing axis is specified with parameter No. 5183.

<4> Cutting with a grinding wheel

Makes a cut in the Z-axis direction with cutting feed by the amount specified as the second depth of cut J. The feedrate is the one specified with R.

<5> Dwell

Performs a dwell for the time specified with P.

<6> Grinding (return direction)

Feeds the machine at the feedrate specified with F in the opposite direction by the amount specified with α. If L is specified when the continuous dressing function is enabled, dressing is performed with the cutting axis and the dressing axis.

- Continuous dressing

If the continuous dressing function is enabled, the grinding-wheel cut and the dresser cut are continuously compensated for according to the dressing amount specified with L during the execution of grinding. For details, see Explanation of G75.

Limitations

- Cutting axis

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

- Grinding axis

To specify a grinding axis, set its axis number, which must be other than that of the cutting axis, in parameter No. 5179.

- Dressing axis

To specify a dressing axis, set its axis number, which must be other than those of the cutting axis and the grinding axis, in parameter No. 5183.

- α,I,J,K

α, I, J, and K commands are all incremental ones.
Spark-out (execution of movement in the grinding direction only) occurs in the following cases:

- I or J is not specified or I = J = 0
- K is not specified or K = 0

If I or J is not specified or if I = J = 0 is true, and K is not equal to 0, a grinding operation is performed infinitely.

- Clearing

The data items I, J, K, α, R, F, and P in a canned cycle are modal information common to G75, G77, G78, and G79, so that once specified, they remain effective until specified anew. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G75, G77, G78, and G79 is specified. L is effective only in the block in which it is specified.

- Operation to be performed if the total depth of cut is reached

The operation to be performed if the total depth of cut reaches during cutting with I or J is the same as that for G75. See Limitation on G75.

NOTE

1 If I, J, and K have different signs, alarm PS0455 is issued.
2 If G79 is specified, but a grinding axis is not specified, alarm PS0455 is issued.
3 If any two of the cutting axis number, the grinding axis number, and the dressing axis number are the same, alarm PS0456 is issued.
4 While this cycle is effective, even if G90 (absolute command) is executed, the α, I, J, and K commands are incremental ones.

6 COMPENSATION FUNCTION

6.1 TOOL LENGTH COMPENSATION SHIFT TYPES

Overview

A tool length compensation operation can be performed by shifting the program coordinate system: The coordinate system containing the axis subject to tool length compensation is shifted by the tool length compensation value. A tool length compensation shift type can be selected with bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400. If no move command is specified together with the G43, G44, or G49 command, the tool will not move along the axis. If a move command is specified together with the G43, G44, or G49 command, the coordinate system will be shifted first, then the tool will move along the axis.

One of the following three methods is available, depending on the type of axis that can be subject to tool length compensation:

- Tool length compensation A
  Compensates the value of the tool length on the Z axis.
- Tool length compensation B
  Compensates the value of the tool length on one of the X, Y, and Z axis.
- Tool length compensation C
  Compensates the value of the tool length on a specified axis.

Format

- Tool length compensation A

G43 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the + side.

G44 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the - side.

G43 (or G44) : + (or -) side offset at which to start tool length compensation
H_ : Address specifying the tool length compensation value

- Tool length compensation B

G17 G43 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the + side.

G17 G44 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the - side.

G18 G43 Y_H_;
Shifts the coordinate system along the X axis by the compensation value, to the + side.

G18 G44 Y_H_;
Shifts the coordinate system along the X axis by the compensation value, to the - side.

G19 G43 X_H_;
Shifts the coordinate system along the Y axis by the compensation value, to the + side.

G19 G44 X_H_;
Shifts the coordinate system along the Y axis by the compensation value, to the - side.

G17 (or G18, G19) : Plane selection
G43 (or G44) : + (or -) side offset at which to start tool length compensation
H_ : Address specifying the tool length compensation value

- Tool length compensation C

G43 α_H_;

Shifts the coordinate system along a specified axis by the compensation value, to the + side.

G44 α_H_;

Shifts the coordinate system along a specified axis by the compensation value, to the - side.

G43 (or G44) : + (or -) side offset at which to start tool length compensation
α_ : Address of any one axis
H_ : Address specifying the tool length compensation value

- Tool length compensation cancel

G49; or H0; Tool length compensation cancel

G49 (or H0) : Tool length compensation cancel

Explanation

- Offset direction

If the tool length compensation value specified with an H code (and stored in offset memory) is G43, the coordinate system is shifted to the + side; if G44, to the - side. If the sign of the tool length compensation value is -, the coordinate system is shifted to the - side if G43 and to the + side if G44. G43 and G44 are modal G codes; they remain valid until another G code in the same group is used.

- Specifying a tool length compensation value

The tool length compensation value corresponding to the number (offset number) specified with an H code (and stored in offset memory) is used. The tool length compensation corresponding to the offset number 0 always means 0. It is not possible to set a tool length compensation value corresponding to H0.

- Compensation axis

Specify one of tool length compensation types A, B, and C, using bits 0 (TLC) and 1 (TLB) of parameter No. 5001.

- Specifying offset on two or more axes

Tool length compensation B enables offset on two or more axes by specifying offset axes in multiple blocks.

To perform offset on X and Y axes
G19 G43 H_; Performs offset on the X axis.
G18 G43 H_; Performs offset on the Y axis.

Tool length compensation C suppresses the generation of an alarm even if offset is performed on two or more axes at the same time, by setting bit 3 (TAL) of parameter No. 5001 to 1.

- Tool length compensation cancel

To cancel offset, specify either G49 or H0. Canceling offset causes the shifting of the coordinate system to be undone. If no move command is specified at this time, the tool will not move along the axis.

Limitation

- Operation to be performed at the start and cancellation of tool length compensation

When a tool length compensation shift type is used (bit 6 (TOS) of parameter No. 5006 = 1 or bit 2 (TOP) of parameter No. 11400 = 1), and if the start or cancellation of a tool length compensation or other command(*2) is specified in cutter compensation or other mode(*1), look-ahead of the subsequent blocks is not performed until the end of the block in which the start or cancellation is specified. Thus, the operation is as described below.

- In the block in which the start or cancellation is specified, deceleration to a stop is performed.
- Because look-ahead is not performed, the compensation vector of cutter compensation is vertical to the block immediately preceding the one in which the start or cancellation is specified. Thus, overcutting or undercutting may occur before or after this command.
- Until the completion of the block in which the start or cancellation is specified, the subsequent custom macros will not be executed.

*1 Look-ahead of blocks is not performed with the commands below.
- G code of group 07 other than G40
  (in each of cutter compensation (G41/G42) mode)

*2 The commands below are included:
- Tool length compensation (G43/G44)

<u>Example in which overcutting occurs in cutter compensation)</u>

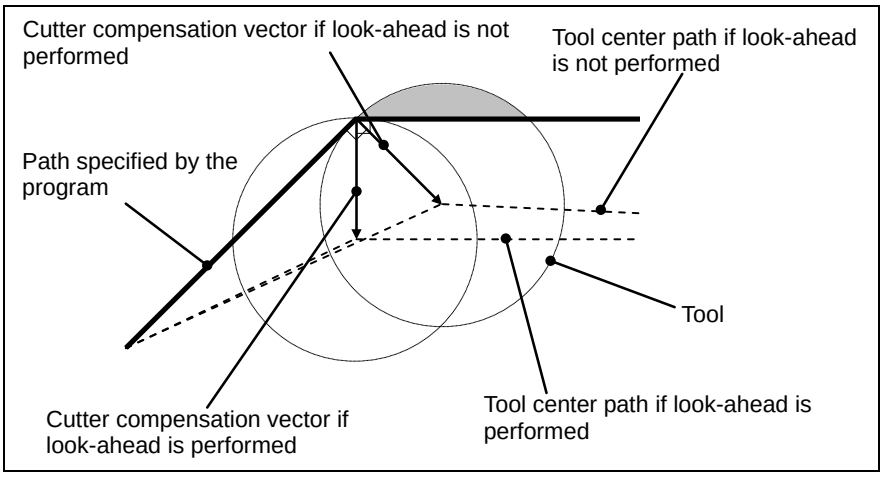
Overcutting may occur if tool length compensation is started or canceled in cutter compensation mode.

```
  :
G40 G49 G00 G90 X0 Y0 Z100. ;
N1 G42 G01 X10. Y10. F500 D1 ;     Start of cutter compensation
N2 G43 Z0. H2 ;                    Start of tool length compensation
N3 X100. ;
N4 Y100. ;
N5 X10. ;
N6 Y10. ;
N7 G49 Z100. ;                     Cancellation of tool length compensation
N8 #100=#5023 ;                    Custom macro command
N9 G40 X0 Y0 ;                     Cancellation of cutter compensation
  :
```

N2 contains G43 (start of tool length compensation) in cutter compensation (G42) mode and, therefore, look-ahead of N3 and subsequent blocks is not performed.

As a result,

- Deceleration to a stop is performed between N2 and N3.
- The cutter compensation vector at the end point of N1 is vertical to block N1.
  (Overcutting may occur.)
  If it is assumed that look-ahead is performed, the vector is vertical to the start point of N2, and no overcutting occurs.



N7 contains G49 (cancellation of tool length compensation) in G42 mode and, therefore, look-ahead of N8 and subsequent blocks is not performed.
As a result,

- Deceleration to a stop is performed at the end point of N7.
- The custom macro command in N8 is executed after the end of N7. This means that in this example, variable #100 will be the machine coordinate on the Z-axis at the end point position of N7. (Variable #5023: Machine coordinate on the third axis)
  If it is assumed that look-ahead is performed, N8 is executed at the point the look-ahead of N8 is performed, that is, before the end of N7, so that variable #100 will be a position before the end point of N7.
- The cutter compensation vector at the end point of N6 is vertical to block N6.
  (Overcutting or undercutting may occur.)

Example in which no overcutting occurs in cutter compensation (recommended)

Before cutter compensation mode, start tool length compensation.

```
  :
G40 G49 G00 G90 X0 Y0 Z100. ;
N1 G43 G01 Z100. F500 H2 ;      Start of tool length compensation
N2 G42 X10. Y10. D1 ;           Start of cutter compensation
N3 Z0 ;
N4 X100. ;
N5 Y100. ;
N6 X10. ;
N7 Y10. ;
N8 G40 X0 Y0 ;                  Cancellation of cutter compensation
N9 G49 Z100. ;                  Cancellation of tool length compensation
N10 #100=#5023 ;                Custom macro command
  :
```

N1 contains a command to start tool length compensation, but because the mode is not included in "*1" above, look-ahead of N2 and subsequent blocks is performed. As a result, the cutter compensation path can be determined correctly. In blocks N1 and N9, deceleration to a stop is not performed. The custom macro command in N10 is executed without waiting for the end of N9.

- Operation to be performed if the tool length compensation is changed in tool length compensation mode

When a tool length compensation shift type is used (bit 6 (TOS) of parameter No. 5006 = 1 or bit 2 (TOP) of parameter No. 11400 = 1), it is possible to select the operation to be performed if the tool length compensation is changed (*3) in cutter compensation or other mode(*1) and in tool length or other mode(*2), by using bit 1 (MOF) of parameter No. 5000.

- Bit 1 (MOF) of parameter No. 5000 = 0
  The tool moves along the axis by the change in tool length compensation.
- Bit 1 (MOF) of parameter No. 5000 = 1:
  After the tool length compensation is changed, movement by the change in tool length compensation is not performed until the absolute command for the compensation axis is executed.

*1 The commands below are included:
G code of group 07 other than G40
(in each of cutter compensation (G41/G42) mode)

*2 The commands below are included:
Tool length compensation (G43/G44)

*3 Changes in tool length compensation include:
- H code specified in a program (D code for the lathe system extended tool selection function)
- G43/G44 specified so that the direction of tool length compensation is changed
- Change in tool compensation made on the offset screen, with a G10 command, a system variable, a window function, etc. with bit 6 (EVO) of parameter No. 5001 being 1.
- Restoration of the tool length compensation vector temporarily canceled with G53, G28 or G30 during tool length compensation

Example in which the tool length compensation is changed with an H code)

The following explains the operation to be performed if the offset number is changed in tool length compensation mode.

```
    :
G40 G49 G00 G90 X0 Y0 Z100. ;
N1 G43 G01 Z100. F500 H2 ;      Start of tool length compensation
N2 G42 X10. Y10. D1 ;           Start of cutter compensation
N3 Z0 ;
N4 X100. ;
N5 Y100. ;
N6 H3 ;                         Tool length compensation (number) change
N7 X10. ;
N8 Y10. ;
N9 G91Z-5. ;                    Incremental command for the compensation axis
N10 G90 Z-5. ;                  Absolute command for the compensation axis
   :
```

In N6, a tool length compensation change (H code) is specified in cutter compensation (G42) mode and tool length compensation (G43) mode. The operation to be performed in this case is as described below, depending on the setting of bit 1 (MOF) of parameter No. 5000.

- Bit 1 (MOF) of parameter No. 5000 = 0:
  In block N6, the tool moves along the axis by the change in tool length compensation.
- Bit 1 (MOF) of parameter No. 5000 = 1:
  In block N6, no movement is performed.
  Block N9 contains an incremental command and, therefore, the movement by the tool length compensation change is not performed. The tool moves by the travel distance specified in the program (-5.000).
  Block N10 contains the absolute command for the compensation axis that is specified first after the tool length compensation change and, therefore, the tool length compensation change is reflected in this block.

Example in which the tool length compensation is overwritten during operation)

The following explains the operation to be performed if continuous operation is executed with the program below, with bit 6 (EVO) of parameter No. 5001 being 1, and tool compensation No. 2 is changed during the execution of N3.

```
      :
G40 G49 G00 G90 X0 Y0 Z100. ;
N1 G43 G01 Z100. F500 H2 ;      Start of tool length compensation
N2 G42 X10. Y10. D1 ;           Start of cutter compensation
N3 Z0 ;                         Change tool length compensation (No. 2) during execution
N4 X100. ;
N5 Y100. ;
N6 X10. ;
N7 Y10. ;
N8 G91Z-5. ;                    Incremental command for the compensation axis
N9 G90 Z-5. ;                   Absolute command for the compensation axis
  :
```

- Bit 1 (MOF) of parameter No. 5000 = 0:
  In N6 (first buffered block after the tool compensation is changed), the tool moves along the axis by the change in tool length compensation.
- Bit 1 (MOF) of parameter No. 5000 = 1:
  Block N6 is the first block after the tool compensation is changed, but this block does not contain a compensation axis command, and the movement by the change in tool length compensation is not performed.
  Block N8 contains a compensation axis command, but the command is an incremental one, and the movement by the change in tool length compensation is not performed. The tool moves by the travel distance specified in the program (-5.000).
  Block N9 contains the first absolute command for the compensation axis that is specified after the tool length compensation is changed and, therefore, the movement by the change in tool length compensation is performed in this block.

⚠ **CAUTION**

1 Specifying tool length compensation (a shift type) first and then executing an incremental programming causes the tool length compensation value to be reflected in the coordinates only, not in the travel distance of the machine; executing an absolute programming causes the tool length compensation value to be reflected in both the movement of the machine and the coordinates.
2 If a programmable mirror image is effective, the tool length compensation is applied in the specified direction.
3 No scaling magnification is applied to the tool length compensation value.
4 No coordinate system rotation is applied to the tool length compensation value. Tool length compensation is effective in the direction in which the offset is applied.
5 3-dimensional coordinate conversion is applied to tool length compensation. If tool length compensation is made effective to multiple axes, the tool length compensation must be canceled for one axis at a time.
6 With the WINDOW command, changing bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400 during automatic operation does not cause the tool length compensation type to be changed.
7 If offset has been performed on two or more axes with tool length compensation B, a G49 command causes the offset to be canceled on all axes; H0 causes the offset to be canceled only on the axis vertical to the specified plane.
8 If the tool length compensation value is changed by changing the offset number, this simply means that the value is replaced by a new tool length compensation value; it does not mean that a new tool length compensation value is added to the old tool length compensation.
9 If reference position return (G28 or G30) has been specified, tool length compensation is canceled for the axis specified at the time of positioning on the reference point; however, tool length compensation is not canceled for an un-specified axis. If reference position return has been specified in the same block as that containing tool length compensation cancel (G49), tool length compensation is canceled for both the specified and un-specified axes at the time of positioning on the mid-point.
10 With a machine coordinate system command (G53), tool length compensation is canceled for the axis specified at the time of positioning on the specified point.
11 The tool length compensation vector canceled by specifying G53, G28, or G30 during tool length compensation is restored as described below:
   - For tool length compensation types A and B, if bit 6 (EVO) of parameter No. 5001 is 1, the vector is restored in the block buffered next; for all of tool length compensation types A, B, and C, it is restored in a block containing an H, G43, or G44 command if parameter is 0.
12 When a tool length compensation shift type is used, if the start or cancellation of a tool length compensation or other command is specified tool radius · tool nose radius compensation mode, look-ahead is not performed. As a result, overcutting or undercutting may occur before or after the block in which the start or cancellation is specified. Thus, specify the start and cancellation of tool length compensation before the entry to tool radius · tool nose radius compensation mode or at a location where machining is not affected.

6.2 AUTOMATIC TOOL LENGTH MEASUREMENT (G37)

By issuing G37 the tool starts moving to the measurement position and keeps on moving till the approach end signal from the measurement device is output. Movement of the tool is stopped when the tool nose reaches the measurement position.
Difference between coordinate value when tool reaches the measurement position and coordinate value commanded by G37 is added to the tool length compensation amount currently used.

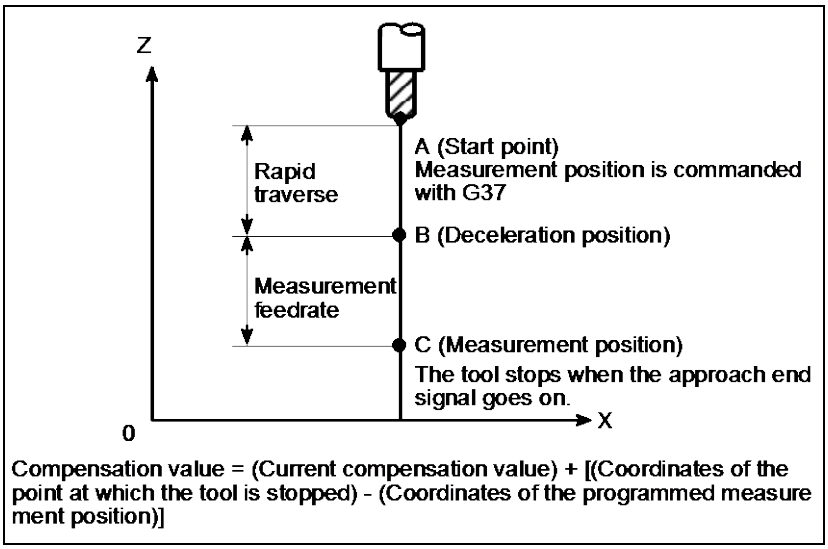


Fig. 6.2 (a) Automatic tool length measurement

Format

Hxx ;	Specifies an offset number for tool length compensation.
G90 G37 IP_ ;	Absolute programming G37 is valid only in the block in which it is specified. IP_ indicates the X-, Y- or Z-axis.

Explanation

- Setting the workpiece coordinate system

Set the workpiece coordinate system so that a measurement can be made after moving the tool to the measurement position. The coordinate system must be the same as the workpiece coordinate system for programming.

- Specifying G37

Specify the absolute coordinates of the correct measurement position.
Execution of this command moves the tool at the rapid traverse rate toward the measurement position, reduces the federate halfway, then continuous to move it until the approach end signal from the measuring instrument is issued. When the tool nose reaches the measurement position, the measuring instrument sends an approach end signal to the CNC which stops the tool.

- Changing the offset value

The difference between the coordinates of the position at which the tool reaches for measurement and the coordinates specified by G37 is added to the current tool length compensation value. (If bit 6 (MDC) of parameter No. 6210 is 1, it is subtracted.)

Offset value =

(Current offset value) + [(Coordinates of the position at which the tool reaches for measurement) - (Coordinates specified by G37)]

These offset values can be manually changed from MDI.

By setting bit 7(CCM) of parameter No.6210 to 1, the offset value can be calculated considering the actual offset amount. The actual offset amount is judged from the G08 group modal and the offset memory.

Offset value = - (Tool geometry offset) $\begin{pmatrix} +(\text{Current offset value}) & : G43 \\ -(\text{Current offset value}) & : G44 \\ 0 & : G49 \end{pmatrix}$

+ [(Coordinates of the position at which the tool reaches for measurement)
- (Coordinates specified by G37)]

- Alarm

When automatic tool length measurement is executed, the tool moves as shown in Fig. 6.2 (b). If the approach end signal turns 1 while the tool is traveling from point B to point C, an alarm occurs. Unless the approach end signal turns 1 before the tool reaches point F, the same alarm occurs. The alarm number is PS0080.

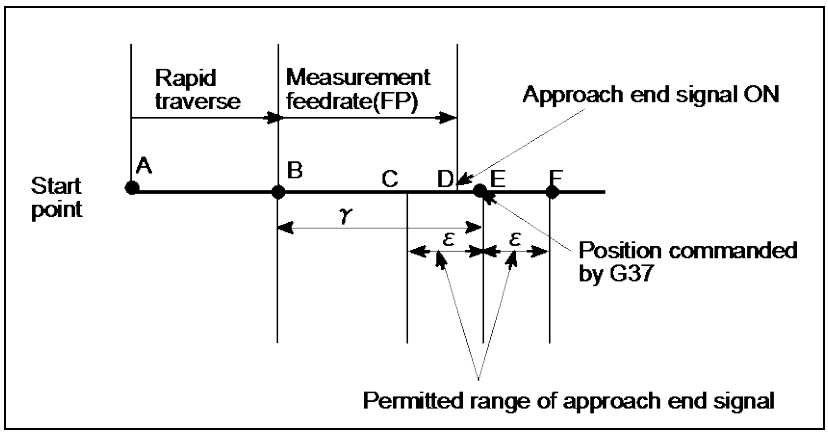


Fig. 6.2 (b) Tool movement to the measurement position

> ⚠ **CAUTION**
> When a manual movement is inserted into a movement at a measurement federate, return the tool to the position before the inserted manual movement for restart.

> **NOTE**
> 1 When an H code is specified in the same block as G37, an alarm is generated. Specify H code before the block of G37.
> 2 The measurement speed (FP), γ, and ε are set as parameters (FP: No. 6241, γ: No. 6251, ε: No. 6254) by the machine tool builder. Make settings so that e are always positive and γ are always greater than ε.
> 3 When tool offset memory A is used, the offset value is changed.
> When tool offset memory C is used, the tool wear compensation value for the H code is changed.

NOTE

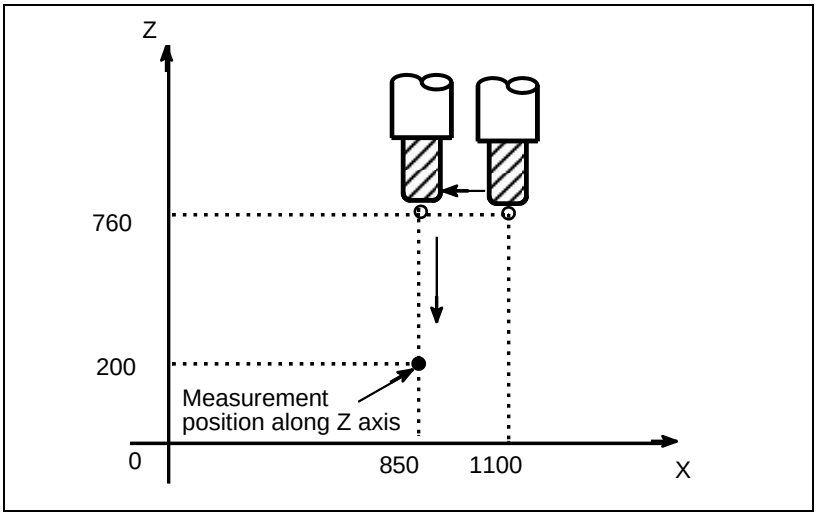
4 A delay or variation in detection of the measurement position arrival signal is 0 to 2 msec on the CNC side excluding the PMC side. Therefore, the measurement error is the sum of 2 msec and a delay or variation (including a delay or variation on the receiver side) in propagation of the skip signal on the PMC side, multiplied by the feedrate set in parameter No. 6241.

5 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.

Example

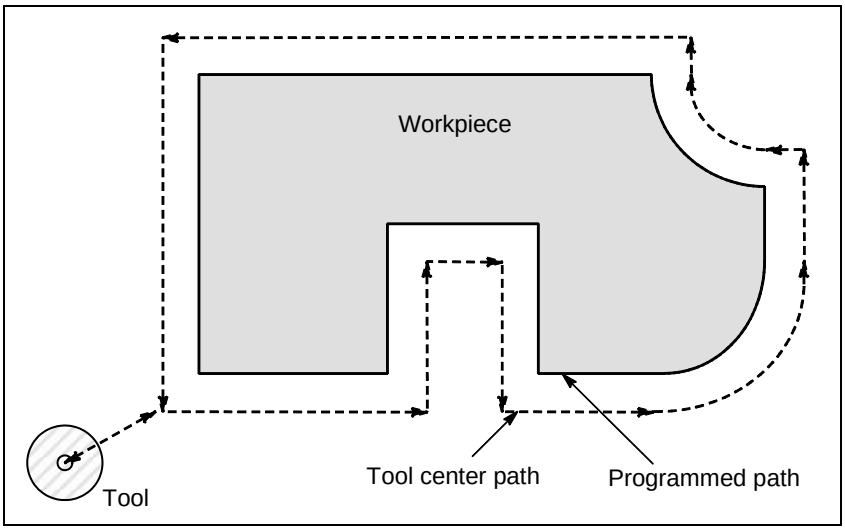
G92 Z760.0 X1100.0 ;	Sets a workpiece coordinate system with respect to the programmed absolute zero point.
G00 G90 X850.0 ;	Moves the tool to X850.0. That is the tool is moved to a position that is a specified distance from the measurement position along the Z-axis.
H01 ;	Specifies offset number 1.
G37 Z200.0 ;	Moves the tool to the measurement position.
G00 Z204.0 ;	Retracts the tool a small distance along the Z-axis.

For example, if the tool reaches the measurement position with Z198.0;, the compensation value must be corrected. Because the correct measurement position is at a distance of 200 mm, the compensation value is lessened by 2.0 mm (198.0 - 200.0 = -2.0).



6.3 TOOL OFFSET (G45 TO G48)

The programmed travel distance of the tool can be increased or decreased by a specified tool offset value or by twice the offset value.
The tool offset function can also be applied to an additional axis.



Format

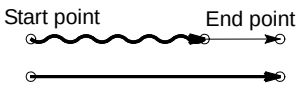
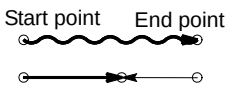
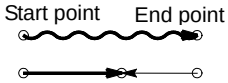
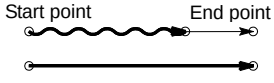
G45 IP_ D_ ;	Increase the travel distance by the tool offset value
G46 IP_ D_ ;	Decrease the travel distance by the tool offset value
G47 IP_ D_ ;	Increase the travel distance by twice the tool offset value
G48 IP_ D_ ;	Decrease the travel distance by twice the tool offset value
G45 to 48 :	One-shot G code for increasing or decreasing the travel distance
IP_ :	Command for moving the tool
D_	Code for specifying the tool offset value

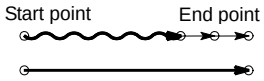
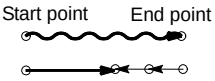
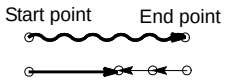
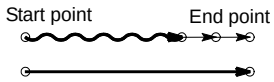
Explanation

- Increase and decrease

As shown in Table 6.3 (a), the travel distance of the tool is increased or decreased by the specified tool offset value.
In the absolute mode, the travel distance is increased or decreased as the tool is moved from the end point of the previous block to the position specified by the block containing G45 to G48.

Table 6.3 (a) Increase and decrease of the tool travel distance

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G45		
G46		

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G47		
G48		

Programmed movement distance
Tool offset value
Actual movement position

If a move command with a travel distance of zero is specified in the incremental programming (G91) mode, the tool is moved by the distance corresponding to the specified tool offset value.
If a move command with a travel distance of zero is specified in the absolute programming (G90) mode, the tool is not moved.

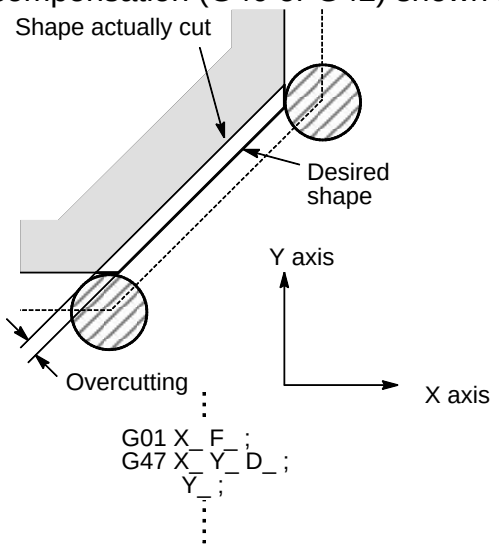
- Tool offset value

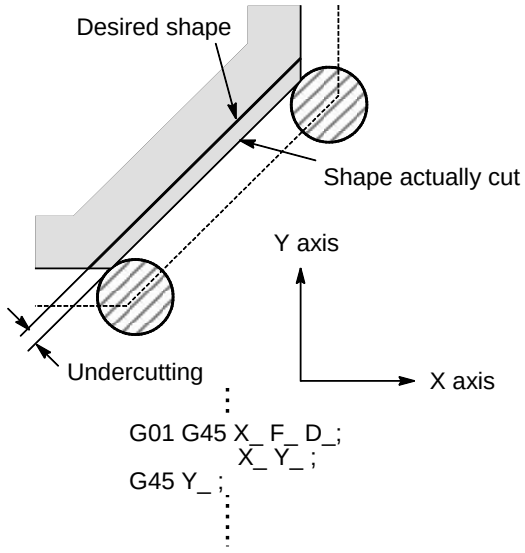
Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.
Tool offset values can be set within the following range:

D0 always indicates a tool offset value of zero.

⚠ **CAUTION**

1 When G45 to G48 is specified to n axes (n=1-6) simultaneously in a motion block, offset is applied to all n axes.
When the cutter is offset only for cutter radius or diameter in taper cutting, overcutting or undercutting occurs.
Therefore, use cutter compensation (G40 or G42) shown in II-6.4 or 6.6.

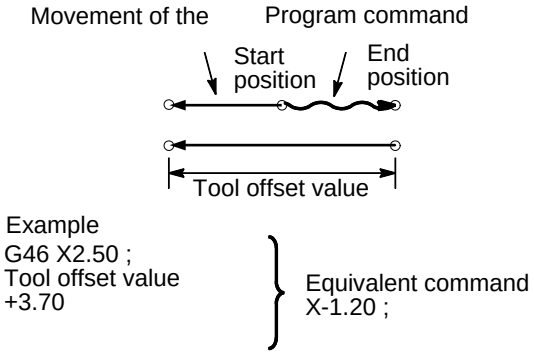




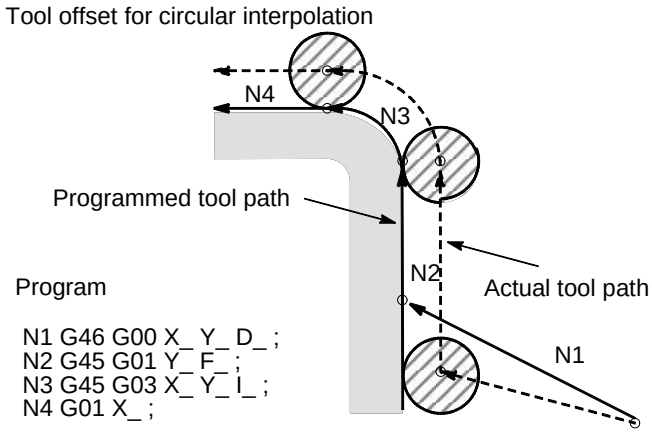
2 G45 to G48 (tool offset) must not be used in the G41 or G42 (cutter compensation) mode.

NOTE

1 When the specified direction is reversed by decrease, the tool moves in the opposite direction.



2 Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I, J and K by the parameter setting, providing that the coordinate system rotation be not specified at the same time. This function is provided for compatibility with the conventional CNC program without any cutter compensation. The function should not be used when a new CNC program is prepared.

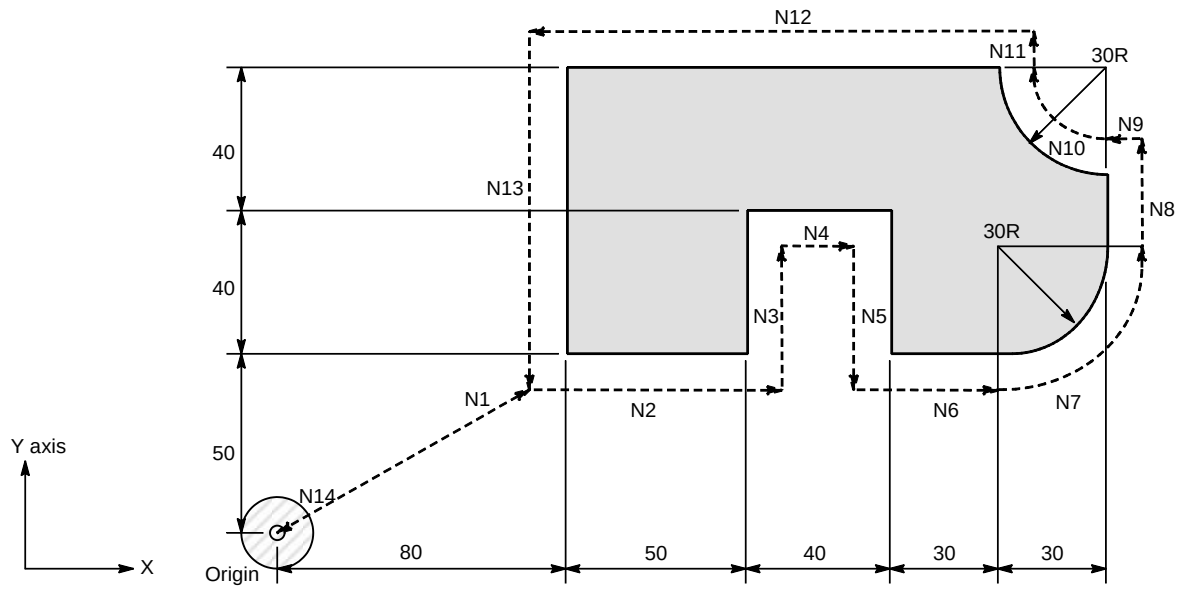


3 D code should be used in tool offset mode.

4 G45 to G48 are ignored in canned cycle mode. Perform tool offset by specifying G45 to G48 before entering canned cycle mode and cancel the offset after releasing the canned cycle mode.

Example

Program using tool offset



Tool diameter : ϕ20
Offset No. : 01
Tool offset value : +10.0

Program

```
N1  G91 G46 G00 X80.0 Y50.0 D01 ;
N2  G47 G01 X50.0 F120.0 ;
N3  Y40.0 ;
N4  G48 X40.0 ;
N5  Y-40.0 ;
N6  G45 X30.0 ;
N7  G45 G03 X30.0 Y30.0 J30.0 ;
N8  G45 G01 Y20.0 ;
N9  G46 X0 ;        (Decreases toward the positive direction for movement amount "0". The tool moves
                    in the -X direction by the offset value.)
N10 G46 G02 X-30.0 Y30.0 J30.0 ;
N11 G45 G01 Y0 ;    (Increase toward the positive direction for movement amount "0". The tool moves in
                    the +Y direction by the offset value.)
N12 G47 X-120.0 ;
N13 G47 Y-80.0 ;
N14 G46 G00 X-80.0 Y-50.0 ;
```

6.4 OVERVIEW OF CUTTER COMPENSATION (G40-G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 6.4 (a)).
To make an offset as large as the radius of the tool, CNC first creates an offset vector with a length equal to the radius of the tool (start-up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head positions to the center of the tool.
If a linear interpolation or circular interpolation command is specified after start-up, the tool path can be shifted by the length of the offset vector during machining.
To return the tool to the start point at the end of machining, cancel the cutter compensation mode.

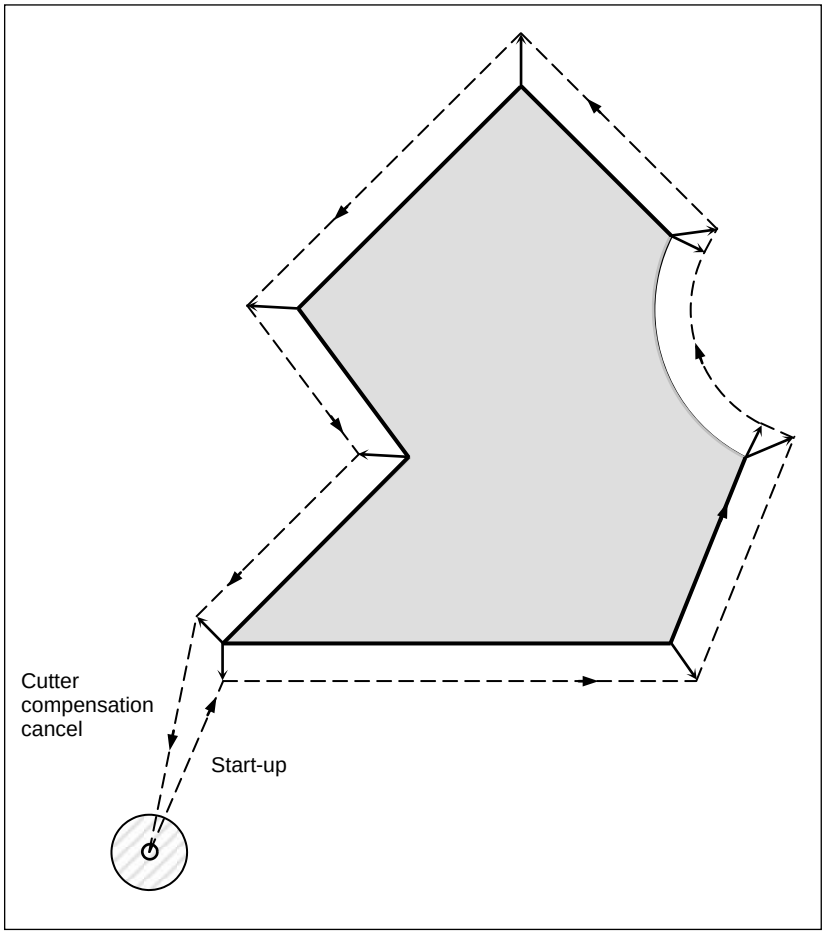


Fig. 6.4 (a) Outline of cutter compensation

Format

- Start up (cutter compensation start)

G00(or G01)G41(or G42) IP_D_;

G41 : Cutter compensation left (Group 07)
G42 : Cutter compensation right (Group 07)
IP_ : Command for axis movement
D_ : Code for specifying as the cutter compensation value (1-3 digits) (D code)

- Cutter compensation cancel (offset mode cancel)

G40 IP_;

G40 : Cutter compensation cancel (Group 07)
(Offset mode cancel)
IP_ : Command for axis movement

- Selection of the offset plane

Offset plane	Command for plane selection	IP_
XpYp	G17 ;	Xp_Yp_
ZpXp	G18 ;	Xp_Zp_
YpZp	G19 ;	Yp_Zp_

Explanation

- Offset cancel mode

At the beginning when power is applied the control is in the cancel mode. In the cancel mode, the vector is always 0, and the tool center path coincides with the programmed path.

- Start-up

When a cutter compensation command (G41 or G42, D code other than 0) is specified in the offset cancel mode, the CNC enters the offset mode.
Moving the tool with this command is called start-up.
Specify positioning (G00) or linear interpolation (G01) for start-up.
If circular interpolation (G02, G03) is specified, alarm PS0034, “ONLY G00/G01 ALLOWED IN STUP/EXT BLK” occurs.
For the start-up and subsequent blocks, the CNC prereads as many blocks as the number of preread blocks set in the parameter No. 19625.

- Offset mode

In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03).
If three or more blocks that move the tool cannot be read in offset mode, the tool may make either an excessive or insufficient cut.
If the offset plane is switched in the offset mode, alarm PS0037, “CAN NOT CHANGE PLANE IN G41/G42” occurs and the tool is stopped.

- Offset mode cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the CNC enters the offset cancel mode, and the action of this block is called the offset cancel.
1. G40 has been commanded.
2. 0 has been commanded as the offset number for cutter compensation (D code).

When performing offset cancel, circular arc commands (G02 and G03) are not available. If these commands are specified, alarm PS0034 is generated and the tool stops. In the offset cancel, the control executes the instructions in that block and the block in the cutter compensation buffer.
In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without reading the next block.
Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for cutter compensation.

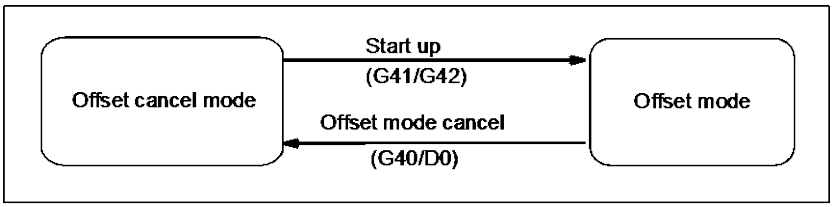


Fig. 6.4 (b) Changing the offset mode

- Change of the cutter compensation value

In general, the cutter compensation value shall be changed in the cancel mode, when changing tools. If the cutter compensation value is changed in offset mode, the vector at the end point of the block is calculated for the new cutter compensation value.

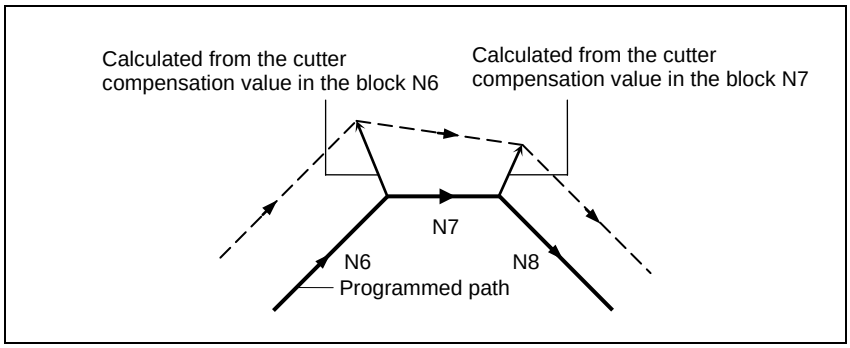


Fig. 6.4 (c) Changing the cutter compensation value

- Positive/negative cutter compensation value and tool center path

If the compensation value is negative (–), distribution is made for a figure in which G41's and G42's are all replaced with each other on the program. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vice versa.
Fig. 6.4 (d) shows one example.
Generally, the compensation value is programmed to be positive (+).
When a tool path is programmed as in <1>, if the compensation value is made negative (–), the tool center moves as in <2>, and vice versa. Consequently, the same program permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the compensation value.
Applicable if start-up and cancel is A type. (See the descriptions about the start-up of cutter compensation.)

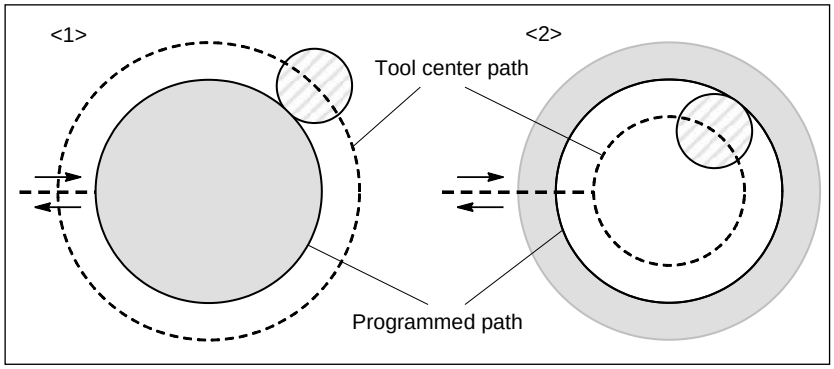


Fig. 6.4 (d) Tool center paths when positive and negative cutter compensation values are specified

- Cutter compensation value setting

Assign a cutter compensation values to the D codes on the MDI unit.

NOTE

The cutter compensation value for which the D code corresponds to 0 always means 0.
It is not possible to set the cutter compensation value corresponding to D0.

- Valid compensation value range

The valid range of values that can be set as a compensation value is either of the following, depending on the bits 3 (OFE), 2 (OFD), 1 (OFC), and 0 (OFA) of parameter No. 5042.

Valid compensation range (metric input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±9999.99 mm
0	0	0	0	±9999.999 mm
0	0	1	0	±9999.9999 mm
0	1	0	0	±9999.99999 mm
1	0	0	0	±999.999999 mm

Valid compensation range (inch input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±999.999 inch
0	0	0	0	±999.9999 inch
0	0	1	0	±999.99999 inch
0	1	0	0	±999.999999 inch
1	0	0	0	±99.9999999 inch

The compensation value corresponding to offset No. 0 always means 0. It is not possible to set the compensation value corresponding to offset No. 0.

- Offset vector

The offset vector is the two dimensional vector that is equal to the cutter compensation value assigned by D code. It is calculated inside the control unit, and its direction is up-dated in accordance with the progress of the tool in each block.
The offset vector is deleted by reset.

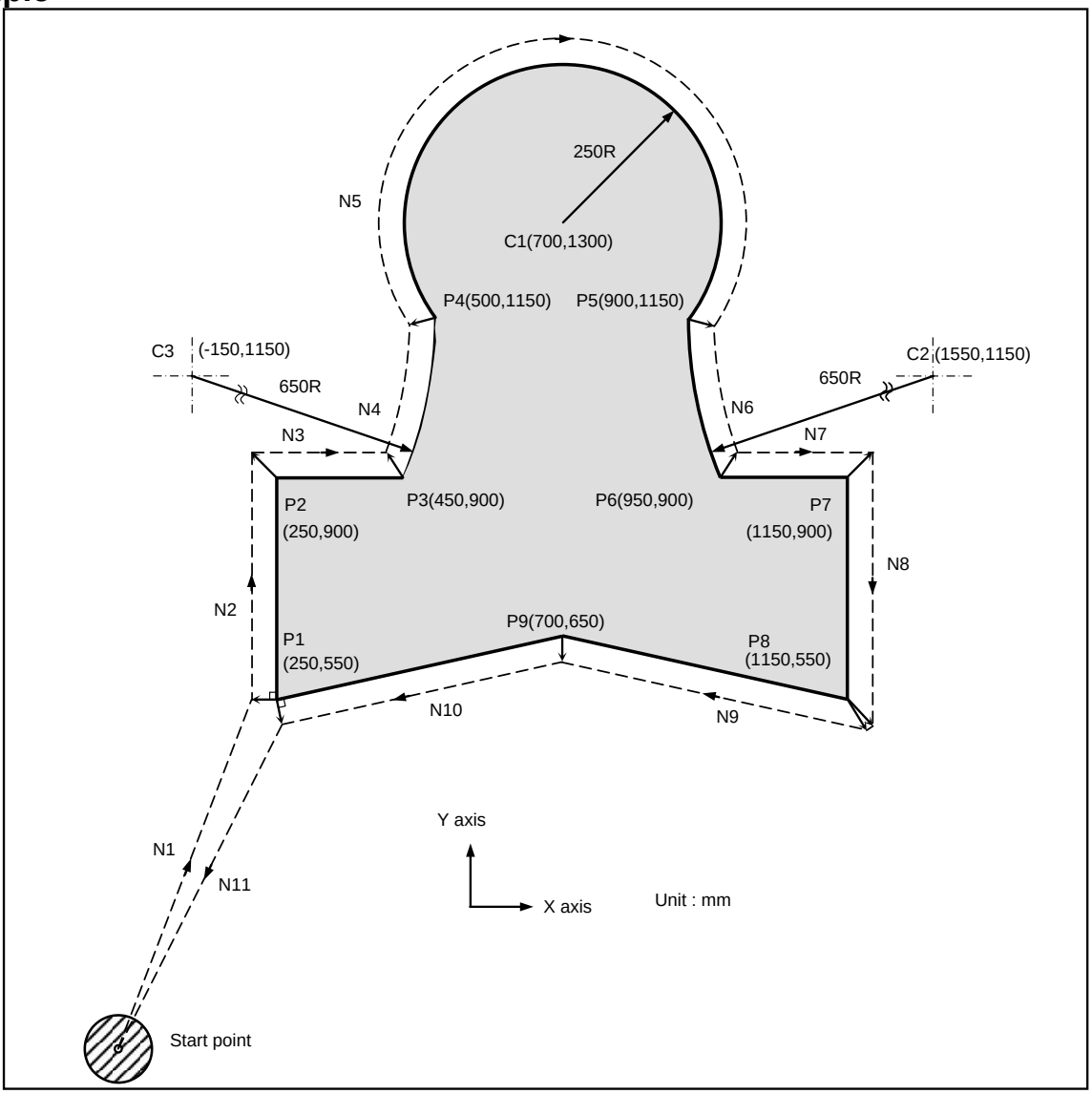
- Specifying a cutter compensation value

Specify a cutter compensation value with a number assigned to it. The number consists of 1 to 3 digits after address D (D code).
The D code is valid until another D code is specified. The D code is used to specify the tool offset value as well as the cutter compensation value.

- Plane selection and vector

Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane.
Compensation is not executed for the coordinate of a position which is not in the specified plane. The programmed values are used as they are.
In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.
The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, alarm PS0037 is displayed and the machine is stopped.

Example



G17 G92 X0.0 Y0.0 Z0.0 ;..... Specifies absolute coordinates.
The tool is positioned at the start point (X0, Y0, Z0).

N1 G90 G00 G41 D07 X250.0 Y550.0 ; .... Starts cutter compensation (start-up).
The tool is shifted to the left of the programmed path by the distance specified in D07.
In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15 mm).

N2 G01 Y900.0 F150 ;..... Specifies machining from P1 to P2.
N3 X450.0 ;..... Specifies machining from P2 to P3.
N4 G03 X500.0 Y1150.0 R650.0 ; ..... Specifies machining from P3 to P4.
N5 G02 X900.0 R-250.0 ;..... Specifies machining from P4 to P5.
N6 G03 X950.0 Y900.0 R650.0 ; ..... Specifies machining from P5 to P6.
N7 G01 X1150.0 ;..... Specifies machining from P6 to P7.
N8 Y550.0 ;..... Specifies machining from P7 to P8.
N9 X700.0 Y650.0 ;..... Specifies machining from P8 to P9.
N10 X250.0 Y550.0 ;..... Specifies machining from P9 to P1.
N11 G00 G40 X0 Y0.0 ; ..... Cancels the offset mode.
The tool is returned to the start point (X0.0, Y0.0, Z0.0).

6.5 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)

The tool nose radius compensation function automatically compensates for the errors due to the tool nose roundness.

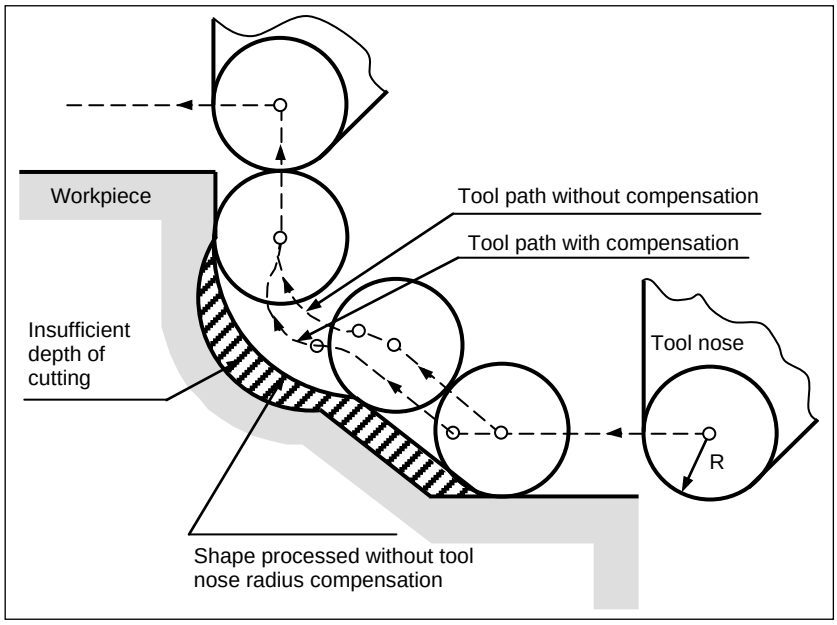


Fig. 6.5 (a) Tool path of tool nose radius compensation

6.5.1 Imaginary Tool Nose

The tool nose at position A in Fig. 6.5.1 (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. 6.5.1 (a).

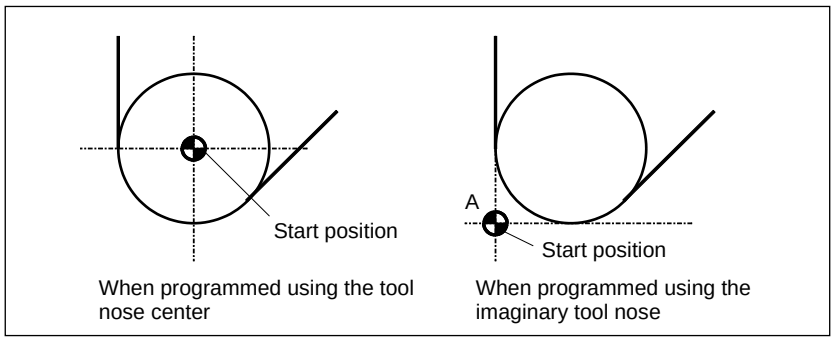


Fig. 6.5.1 (a) Tool nose radius center and imaginary tool nose

⚠ 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 tool nose radius center or the imaginary tool nose is compensated by the tool length compensation function.
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.

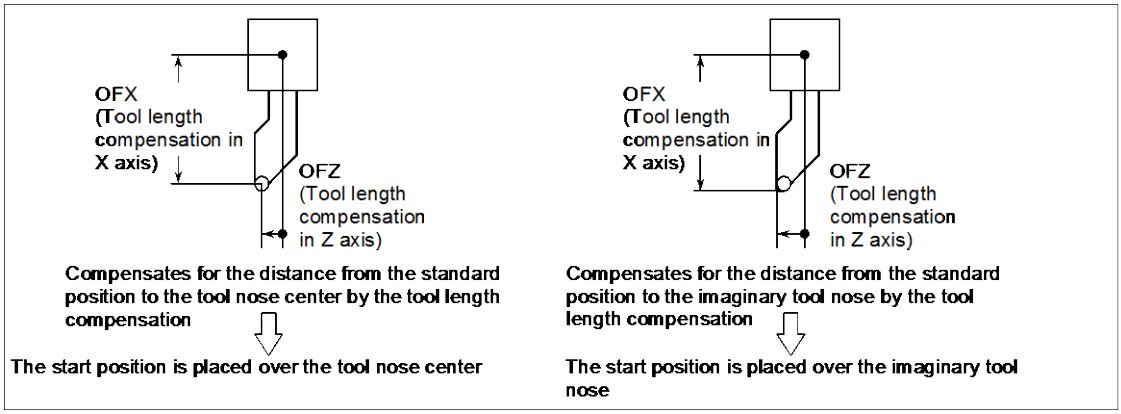


Fig. 6.5.1 (b) Tool length compensation when the turret center is placed over the start point

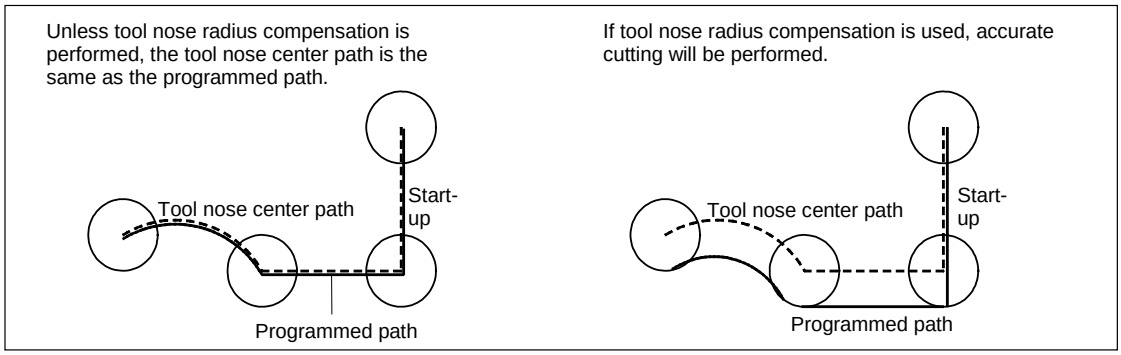


Fig. 6.5.1 (c) Tool path when programming using the tool nose center

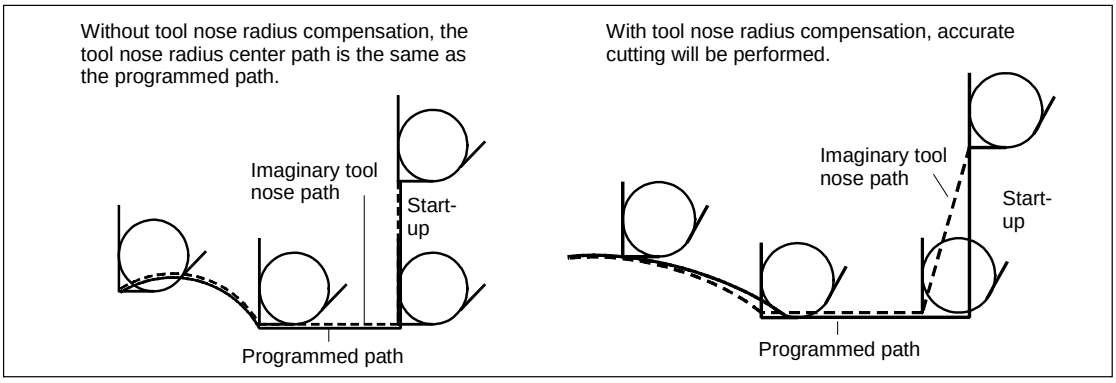


Fig. 6.5.1 (d) Tool path when programming using the imaginary tool nose

6.5.2 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. 6.5.2 (a) below together with their corresponding codes. Fig. 6.5.2 (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.

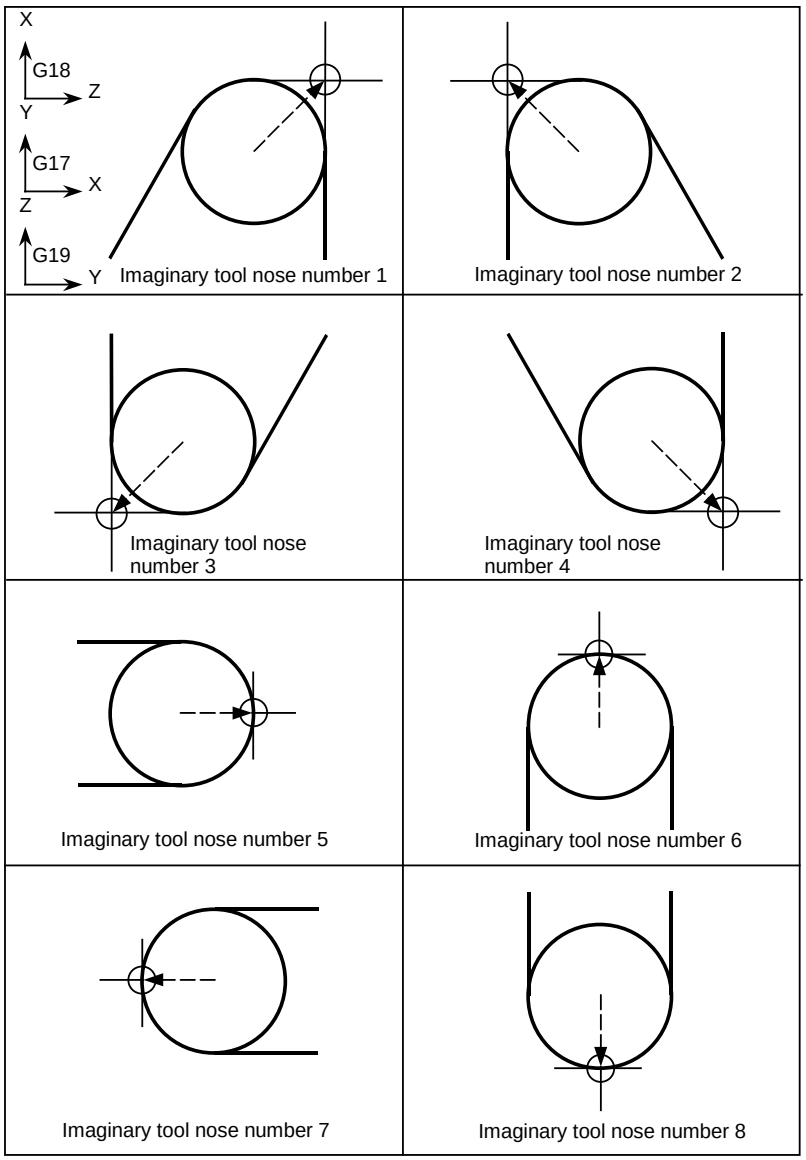
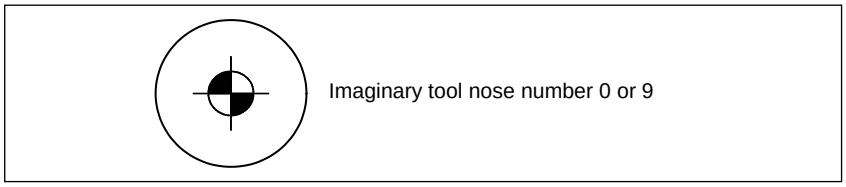


Fig. 6.5.2 (a) Direction of imaginary tool nose

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.



6.5.3 Offset Number and Offset Value

Explanation

- Offset number and offset value

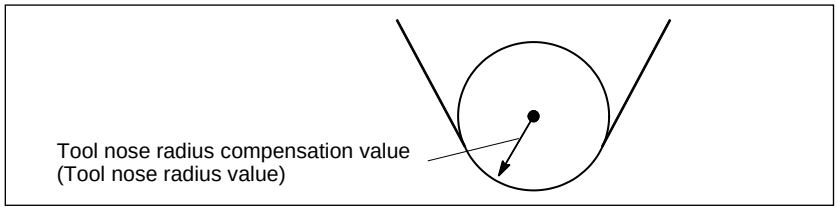


Table 6.5.3 (a) Offset number and offset value (example)

Offset number Up to 999 sets	(Tool compensation value)	(Direction of imaginary tool nose)
001	0.200	1
002	0.250	2
003	0.120	6
004	:	:
005	:	:
:	:	:

- Command of offset value

An offset number is specified with the D code.

- 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 3 (OFE), 2 (OFD), 1 (OFC), and 0 (OFA) of parameter No. 5042.

Table 6.5.3 (b) Valid compensation range (metric input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±9999.99 mm
0	0	0	0	±9999.999 mm
0	0	1	0	±9999.9999 mm
0	1	0	0	±9999.99999 mm
1	0	0	0	±999.999999 mm

Table 6.5.3 (c) Valid compensation range (inch input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±999.999 inch
0	0	0	0	±999.9999 inch
0	0	1	0	±999.99999 inch
0	1	0	0	±999.999999 inch
1	0	0	0	±99.9999999 inch

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

6.5.4 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.

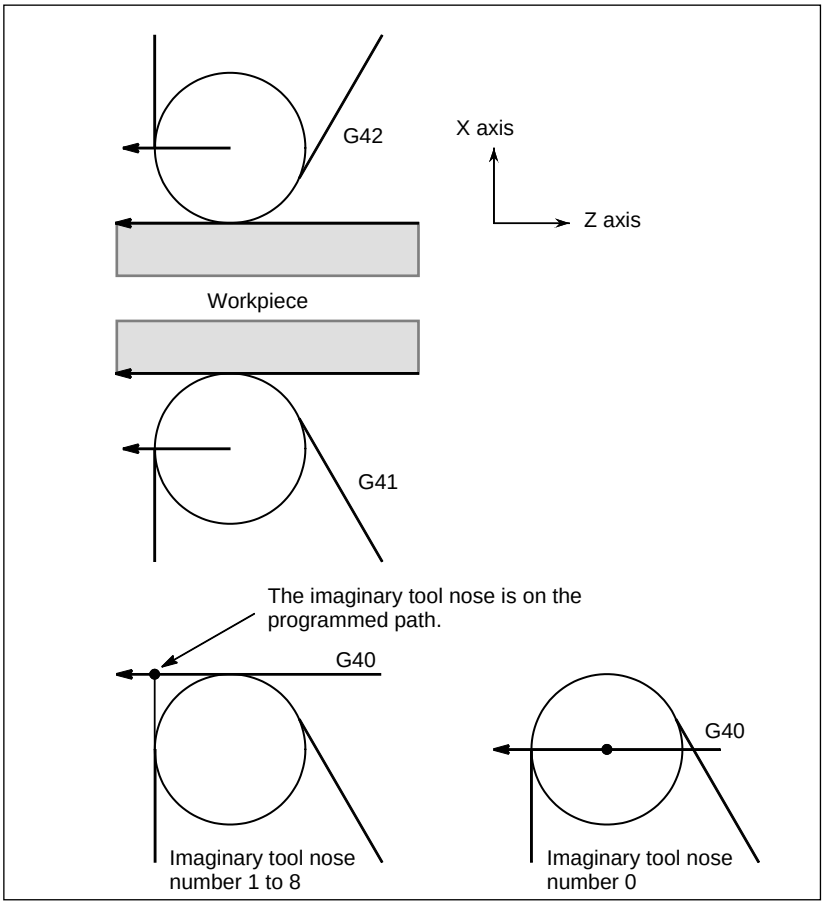


Fig. 6.5.4 (a) Workpiece position

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

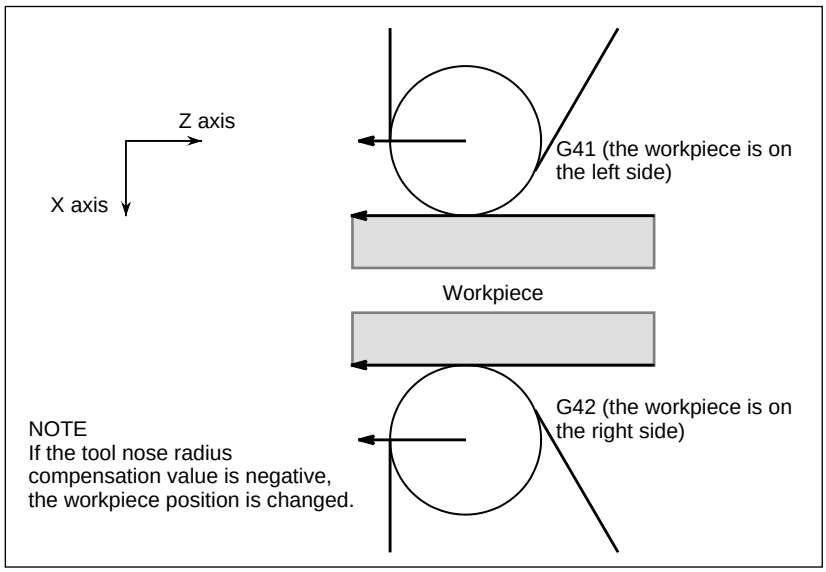


Fig. 6.5.4 (b) When the workpiece position is changed

G40, G41, and, G42 are modal.
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.
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 nose does not change. For a use in which the imaginary tool nose 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.

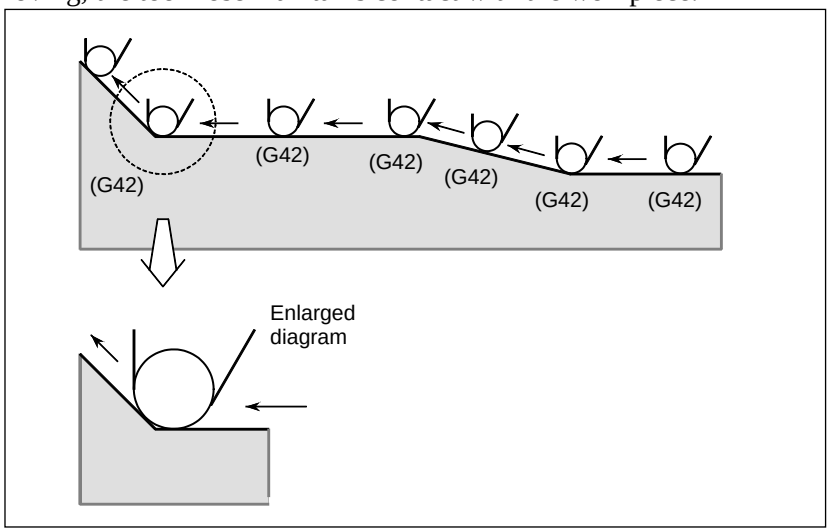


Fig. 6.5.4 (c) Tool movement when the workpiece position does not change

- 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.

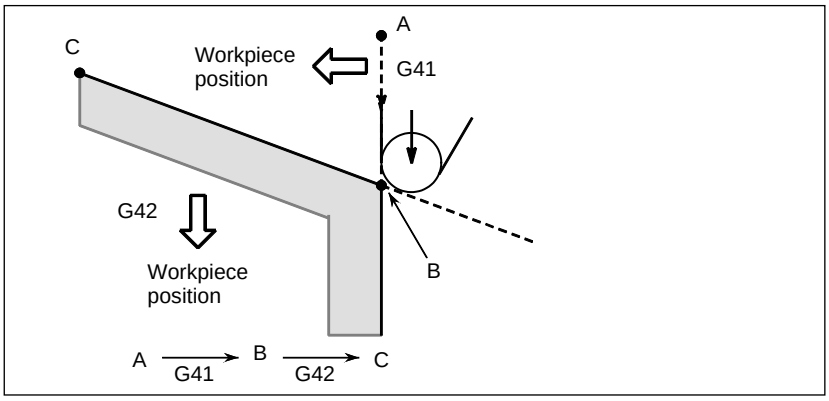


Fig. 6.5.4 (d) Tool movement when the workpiece position changes

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 example in the Fig. 6.5.4 (d), 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.

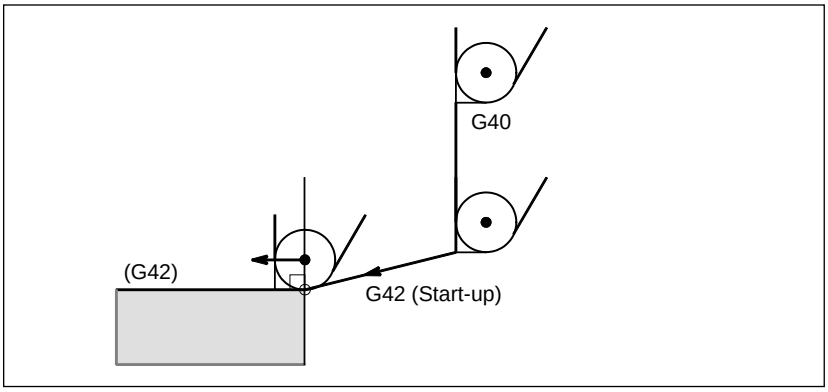


Fig. 6.5.4 (e) Start-up

- 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 point in the offset cancel block (G40) as shown below.

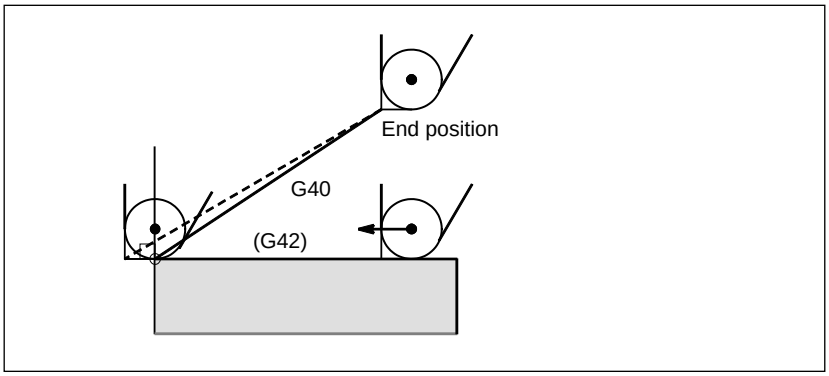


Fig. 6.5.4 (f) Offset cancel

- 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.

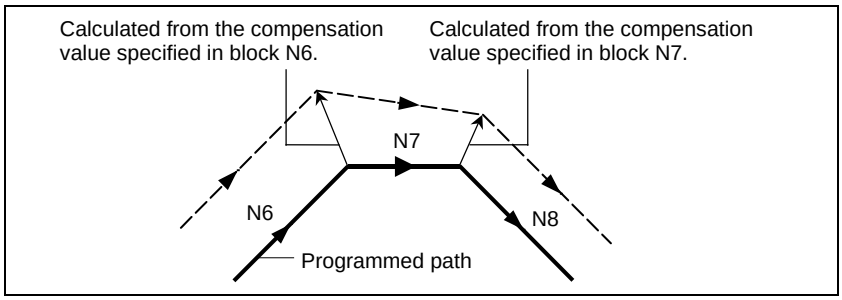


Fig. 6.5.4 (g) Changing the compensation value

- 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 point of the preceding block.

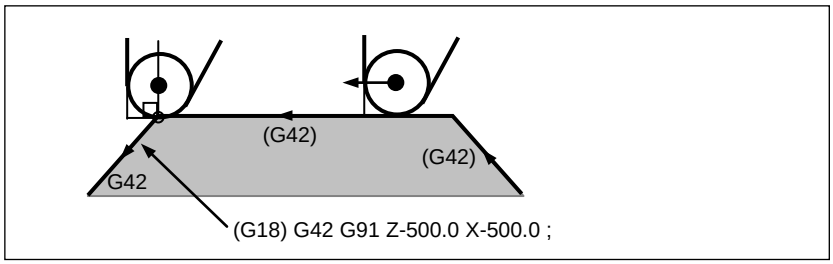


Fig. 6.5.4 (h) Specification of G41/G42 in G41/G42 mode

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 and Z canceling the tool nose radius compensation at the end of machining the first block in Fig. 6.5.4 (i), specify the following :

G40 X _ Z _ I _ K _ ;

where I and K are the direction of the workpiece in the next block, which is specified in incremental mode.

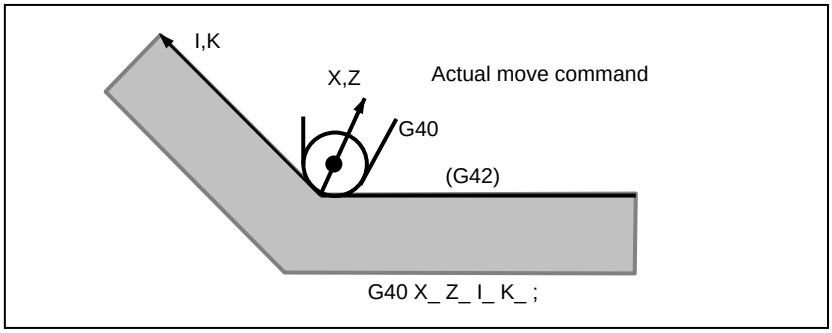


Fig. 6.5.4 (i) If I and K are specified in the same block as G40

Thus, this prevents the tool from overcutting, as shown in Fig. 6.5.4 (j).

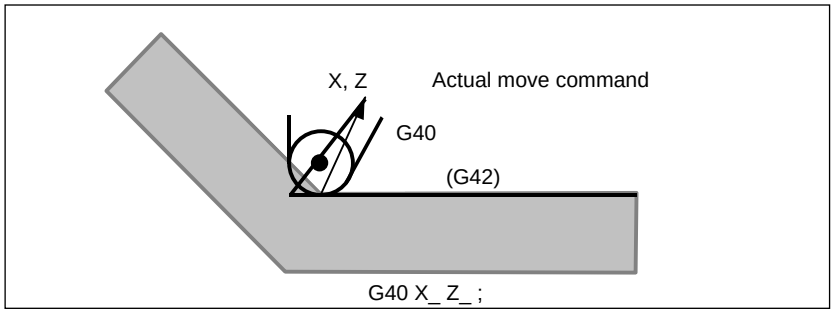


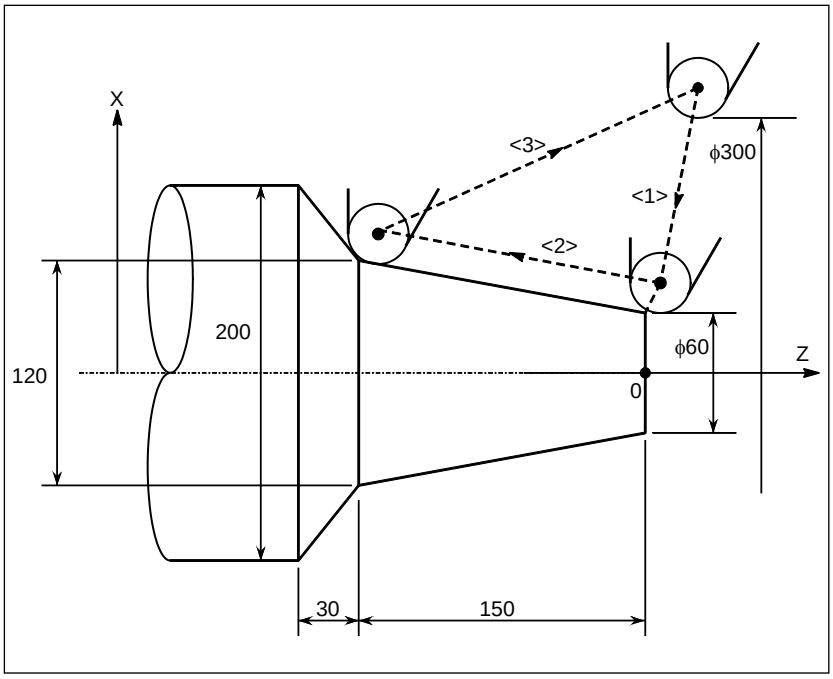
Fig. 6.5.4 (j) Case in which overcutting occurs in the same block as G40

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 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 Z-150.0 F10 ;**

<3> **G40 G00 X300.0 Z0 I40.0 K-30.0 ;**

6.5.5 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>	**G91 G01 X0.0 ;**	Feed distance of zero
<6>	**G90 ;**	G code only
<7>	**G10 L11 P01 R10.0 ;**	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.

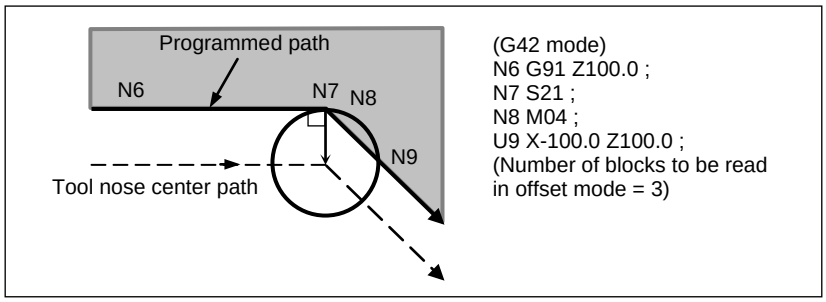


Fig. 6.5.5 (a)

Overcutting may, therefore, occur in the Fig. 6.5.5 (a).

- Tool nose radius compensation when chamfering is performed

Movement after compensation is shown Fig. 6.5.5 (b).

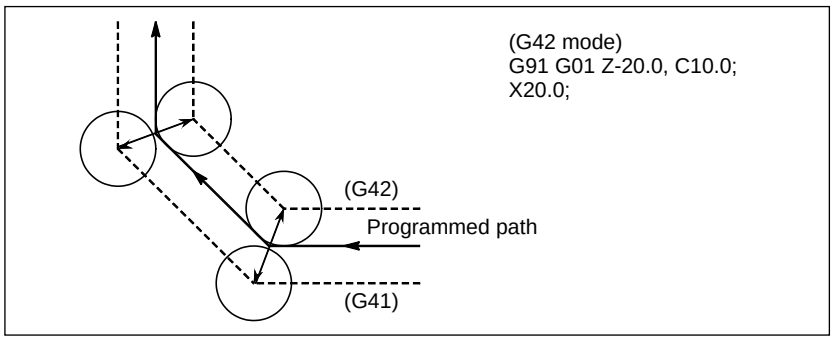


Fig. 6.5.5 (b)

- Tool nose radius compensation when a corner R is performed

Movement after compensation is shown Fig. 6.5.5 (c).

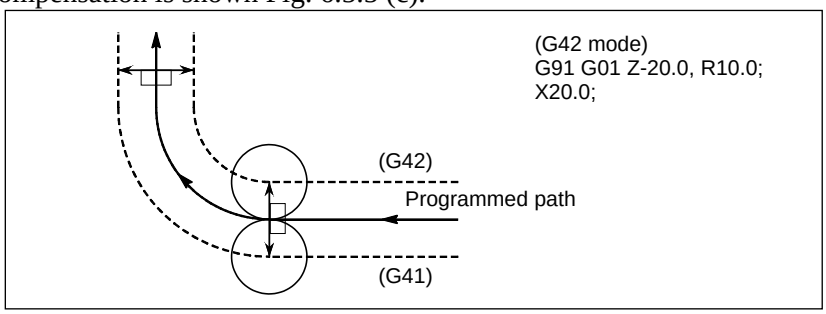


Fig. 6.5.5 (c)

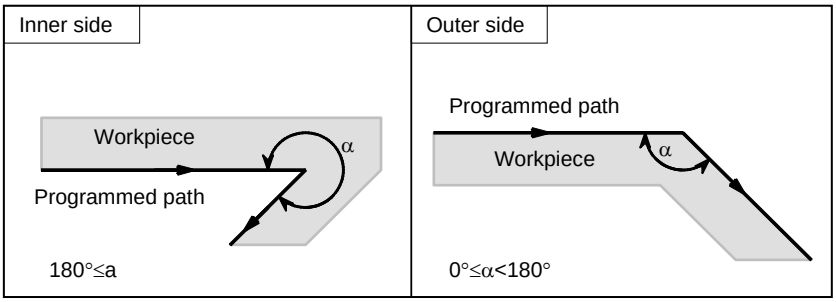
6.6 DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION

6.6.1 Overview

The following explanation focuses on the cutter compensation, but applies to the tool nose radius compensation as well.

- 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."

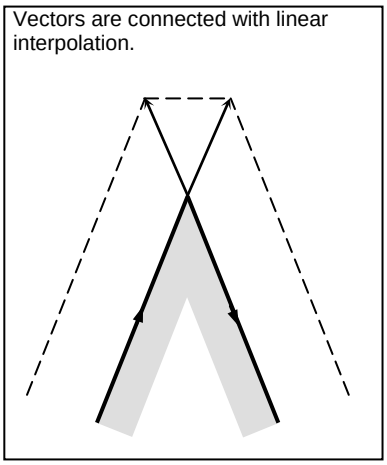


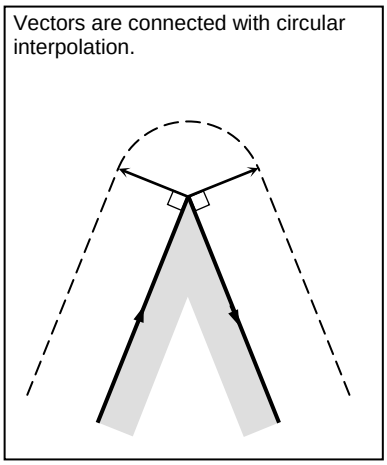
- Outer corner connection method

If the tool moves around an outer corner in cutter compensation mode, it is possible to specify whether to connect compensation vectors with linear interpolation or with circular interpolation, using bit 2 (CCC) of parameter No. 19607.

<1> Linear connection type [Bit 2 (CCC) of parameter No. 19607 = 0]

<2> Circular connection type [Bit 2 (CCC) of parameter No. 19607 = 1]





- Cancel mode

The cutter compensation enters the cancel mode under the following conditions. (The system may not enter the cancel mode depending on the machine tool.)

<1> Immediately after the power is turned on

<2> When the RESET key on the MDI unit is pushed

<3> After a program is forced to end by executing M02 or M30

<4> After the cutter compensation cancel command (G40) is exercised

In the cancel mode, the compensation vector is set to zero, and the path of the center of tool coincides with the programmed path. A program must end in cancel mode. If it ends in the cutter compensation mode, the tool cannot be positioned at the end point, and the tool stops at a location the compensation vector length away from the end point.

NOTE

The operation to be performed when a reset operation is performed during cutter compensation differs depending on bit 6 (CLR) of parameter No. 3402.

- If CLR is 0
  The system enters the reset state. G41/G42 are retained as the modal code of group 07, but to perform cutter compensation, an offset number (D code) must be specified again.
- If CLR is 1
  The system enters the clear state. The modal code of group 07 is G40, and to perform cutter compensation again, G41/G42 and an offset number (D code) must be specified.

- Start-up

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

<1> G41 or G42 is contained in the block, or has been specified to place the CNC in the cutter compensation mode.

<2> 0 < compensation number of cutter compensation ≤ maximum compensation number

<3> Positioning (G00) or linear interpolation (G01) mode

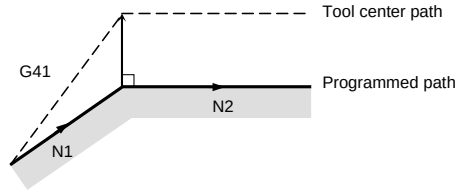
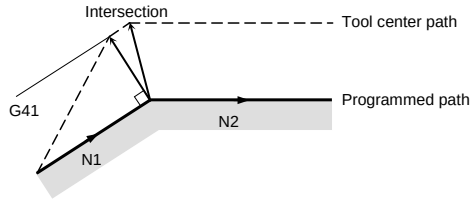
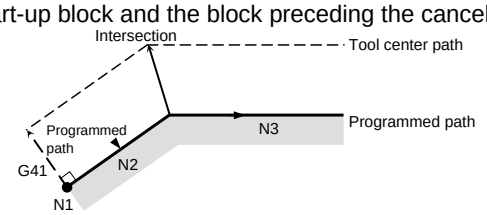
<4> A compensation plane axis command with a travel distance of 0 (except start-up type C) is specified.

If start-up is specified in circular interpolation (G02, G03) mode, alarm PS0034, “ONLY G00/G01 ALLOWED IN STUP/EXT BLK” will occur.

In a start-up block, workpiece coordinate system switching (G54 to G59) cannot be specified.

As a start-up operation, one of the three types A, B, and C can be selected by setting bit 0 (SUP) of parameter No. 5003 and bit 1 (SUV) of parameter No. 5003 appropriately. The operation to be performed if the tool moves around an inner side is of single type only.

Table 6.6.1 (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 radius · 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 cutter compensation mode

In cutter compensation mode, input commands are read from usually three blocks and up to eight blocks depending on the setting of parameter No. 19625 to perform intersection calculation or an interference check, described later, regardless of whether the blocks are with or without tool movement, until a cancel command is received.
To perform intersection calculation, it is necessary to read at least two blocks with tool movement. To perform an interference check, it is necessary to read at least three blocks with tool movement.

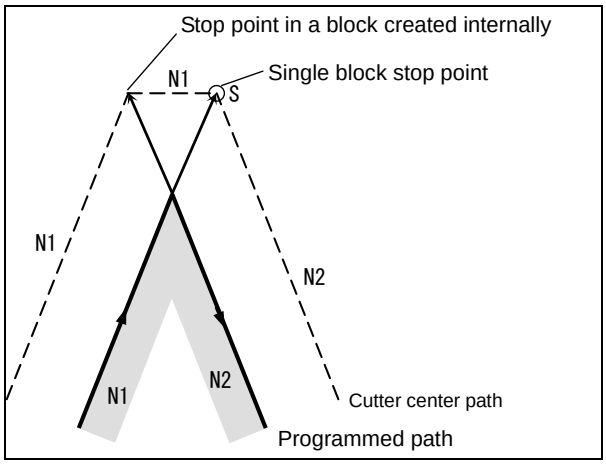
As the setting of parameter No. 19625, that is, the number of blocks to read, increases, it is possible to predict overcutting (interference) for up to more subsequent commands. Increases in blocks to read and analyze, however, cause reading and analysis to take more time.

- Ending (canceling) cutter compensation

In cutter compensation mode, cutter compensation is canceled if a block that satisfies at least either one of the following conditions is executed:
<1> G40 is specified.
<2> D00 is specified as the compensation number of cutter compensation.
If cutter compensation cancel is to be performed, it must not be by a circular command (G02, G03). Otherwise, an alarm will occur.
For a cancel operation, one of three types, A, B, and C, can be selected by appropriately setting bit 0 (SUP) of parameter No. 5003 and bit 1 (SUV) of parameter No. 5003. The operation to be performed if the tool turns around the inside is of a single type.

- Bit 0 (SBK) of parameter No. 5000

When bit 0 (SBK) of parameter No. 5000 is set to 1, a single block stop can be performed in a block created internally for cutter compensation.
Use this parameter to check a program including cutter compensation.



NOTE
When an auxiliary function (M code), spindle speed function (S code), tool function (T code), or second auxiliary function (B code) is specified in the N1 block in the figure above, FIN is not accepted if the tool stops at the stop point in a block created internally (excluding the single block stop point).

- Meaning of symbols

The following symbols are used in subsequent figures:

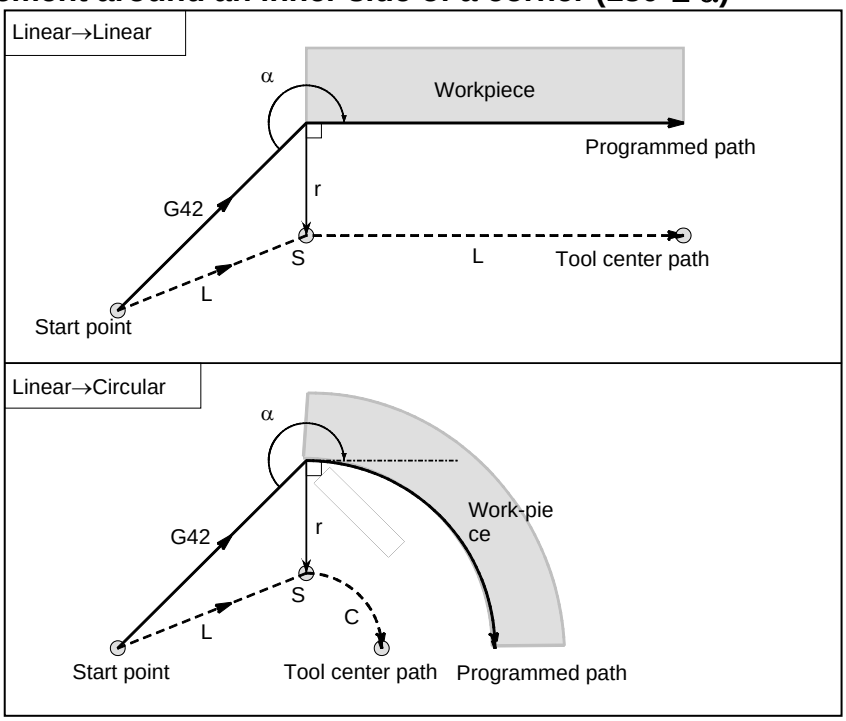
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves along an arc.
- r indicates the tool radius · 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.

6.6.2 Tool Movement in Start-up

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

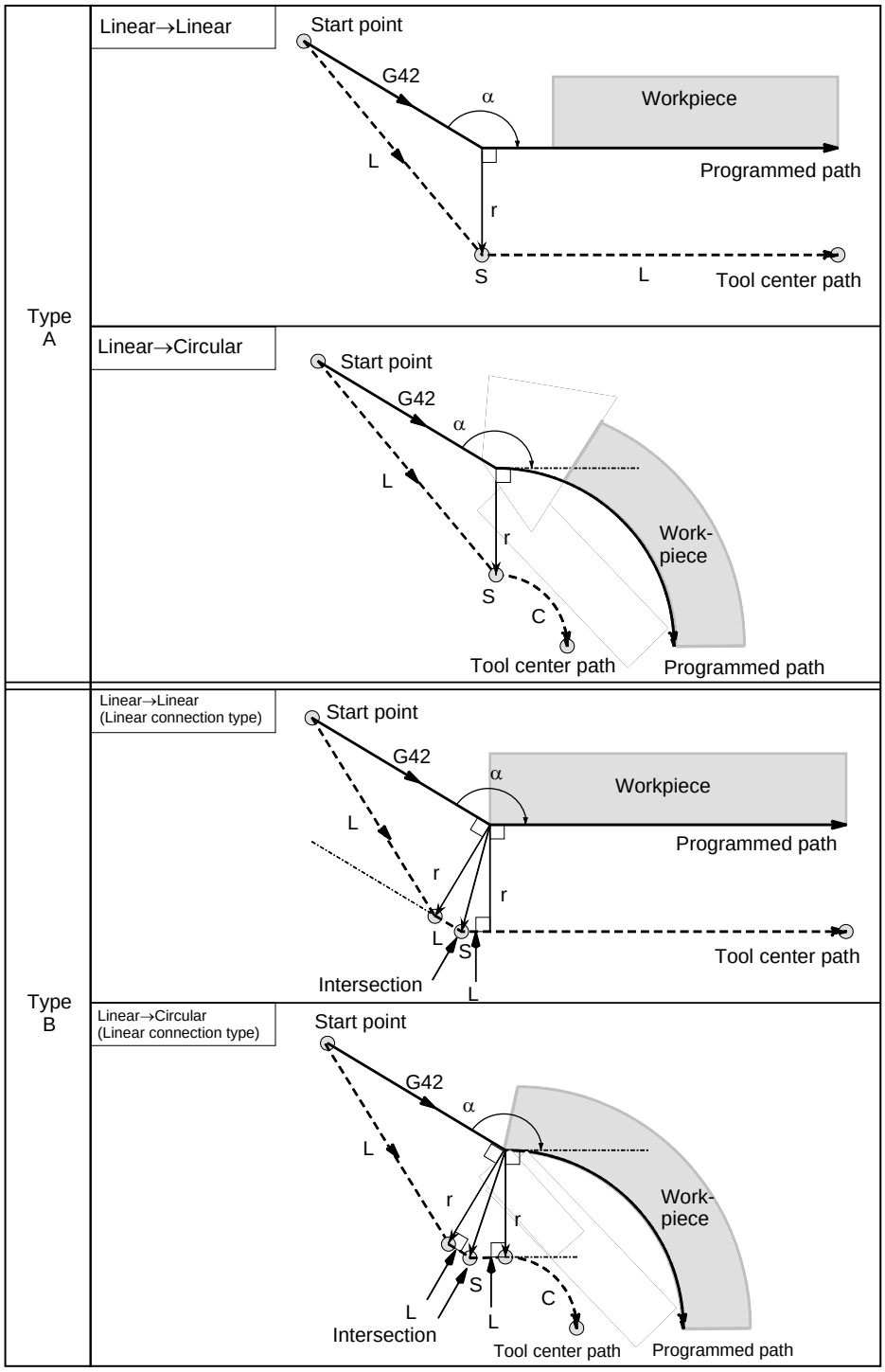
Explanation

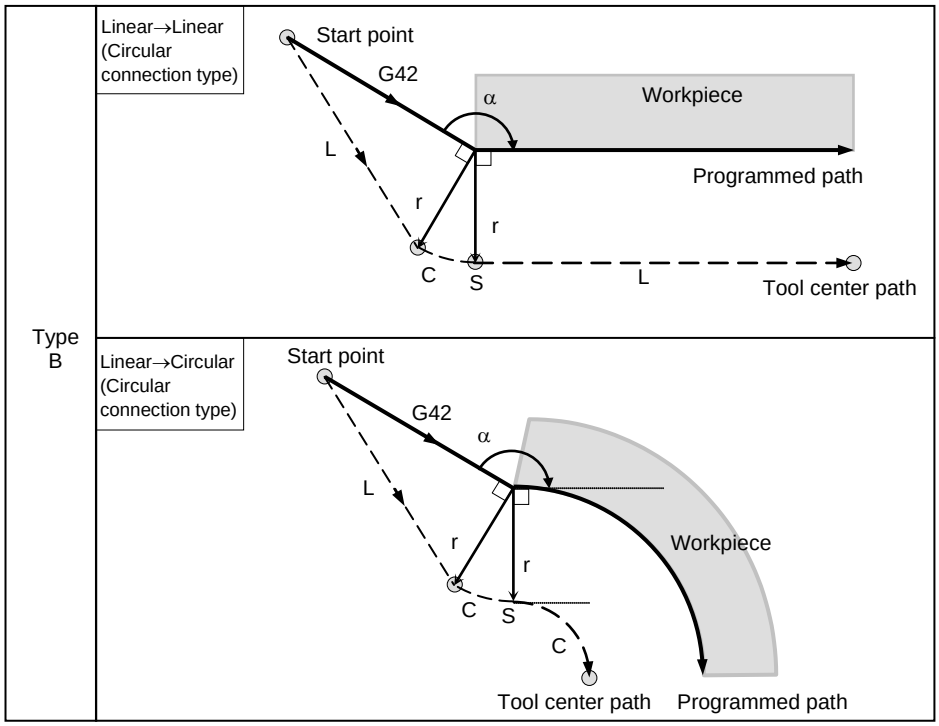
- **Tool movement around an inner side of a corner (180°≤ α)**



- **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°)**

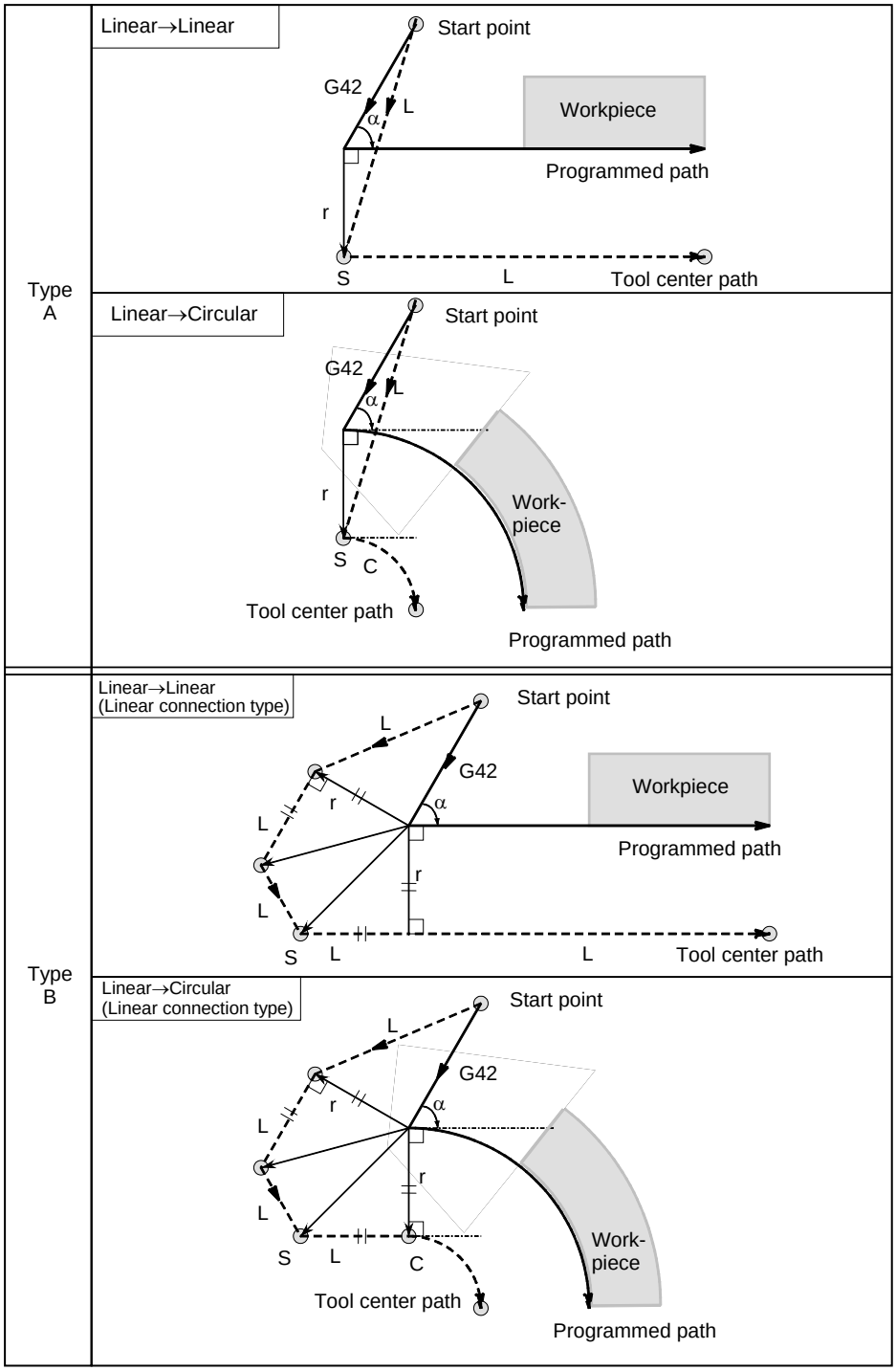
Tool path in start-up has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

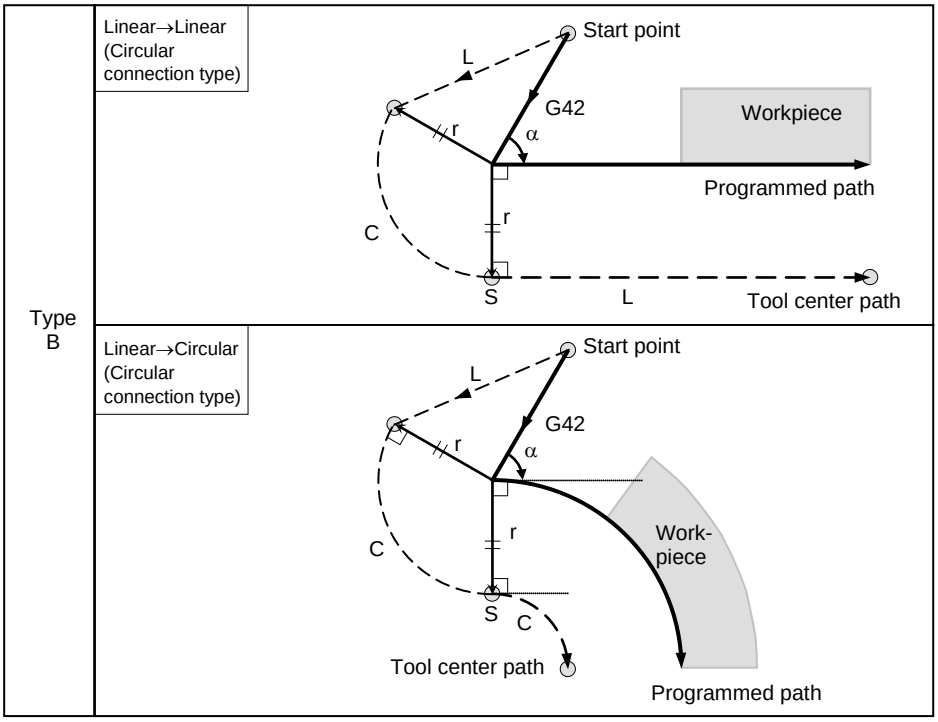



Type B
Linear→Linear (Circular connection type)
Start point
G42
α
Workpiece
L
Programmed path
r
r
C
S
L
Tool center path
Linear→Circular (Circular connection type)
Start point
G42
α
L
r
r
Workpiece
C
S
C
Tool center path
Programmed path

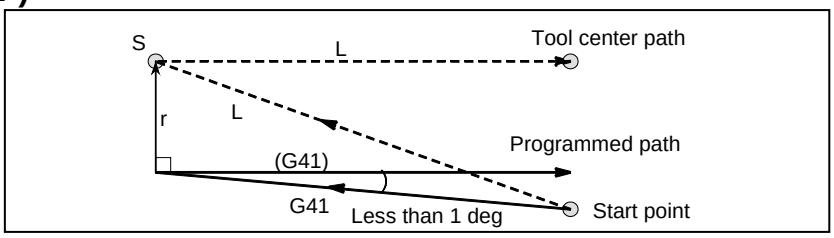
- **Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle (α<90°)**

Tool path in start-up has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.





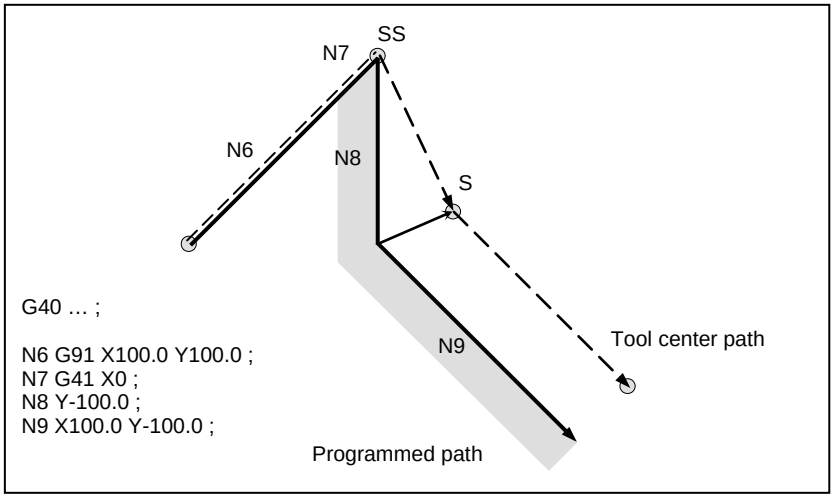
- **Tool movement around the outside linear → linear at an acute angle less than 1 deg (α<1°)**



- **A block without tool movement specified at start-up**

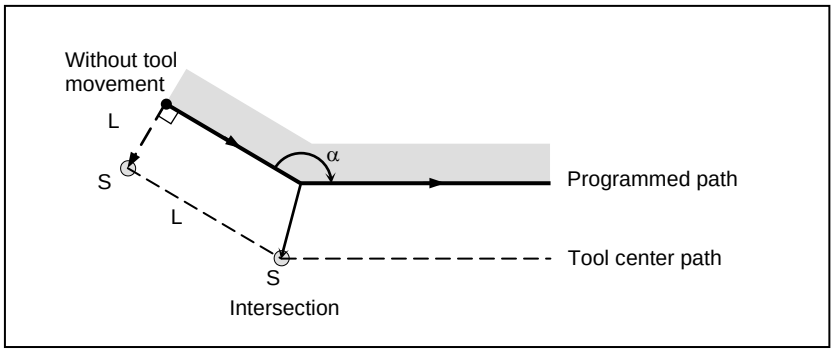
For type A and B

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.



6.6.3 Tool Movement in Offset Mode

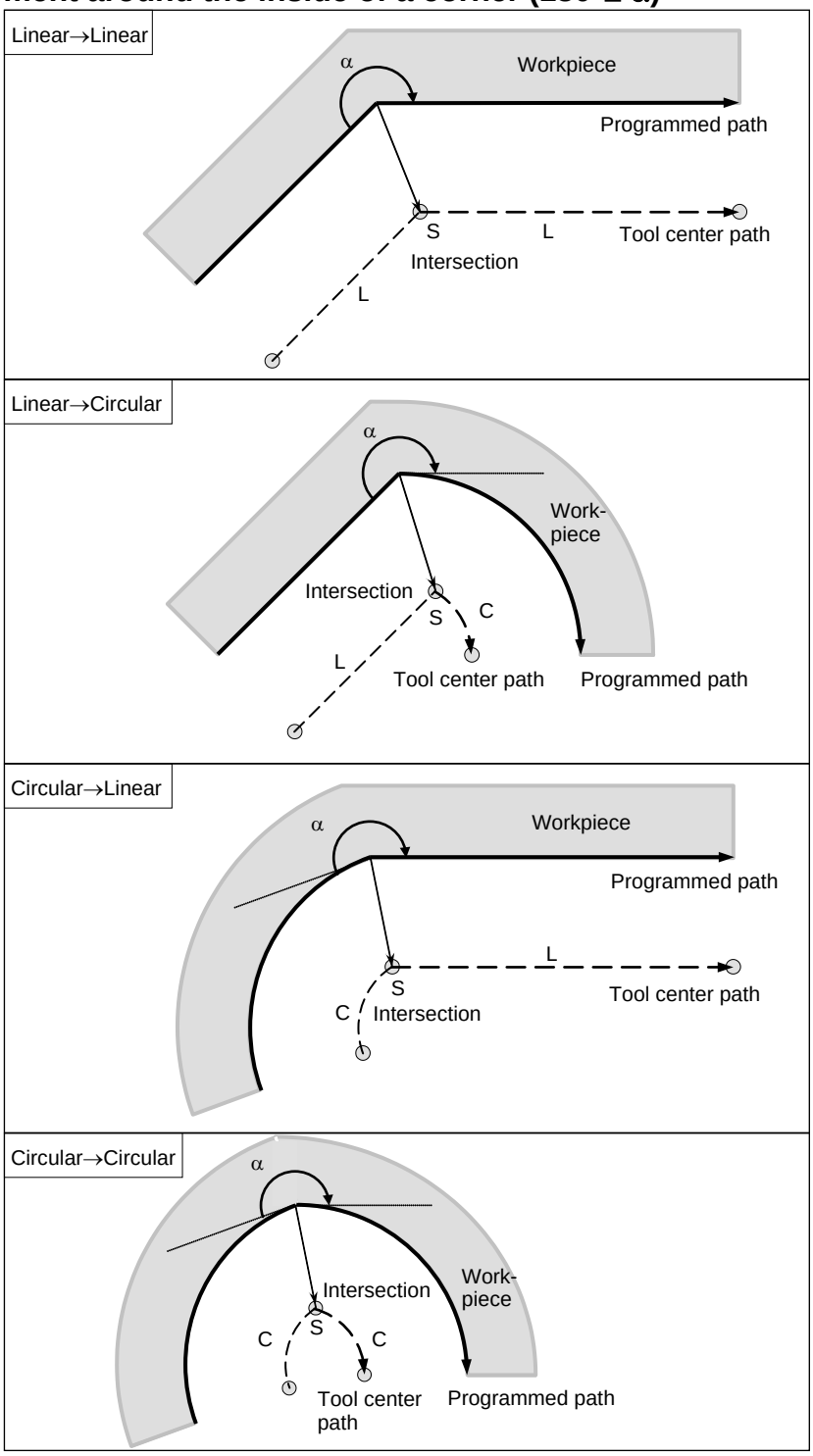
In offset mode, compensation is performed even for positioning commands, not to speak of linear and circular interpolations. To perform intersection calculation, it is necessary to read at least two blocks with tool movement. If, therefore, two or more blocks with tool movement cannot be read in offset 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) ≥ 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

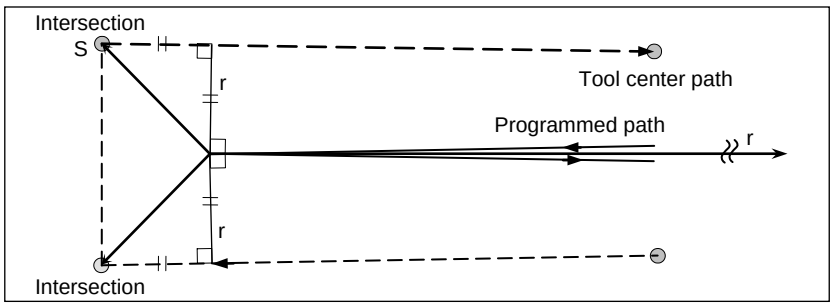
The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the interference check.

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)**

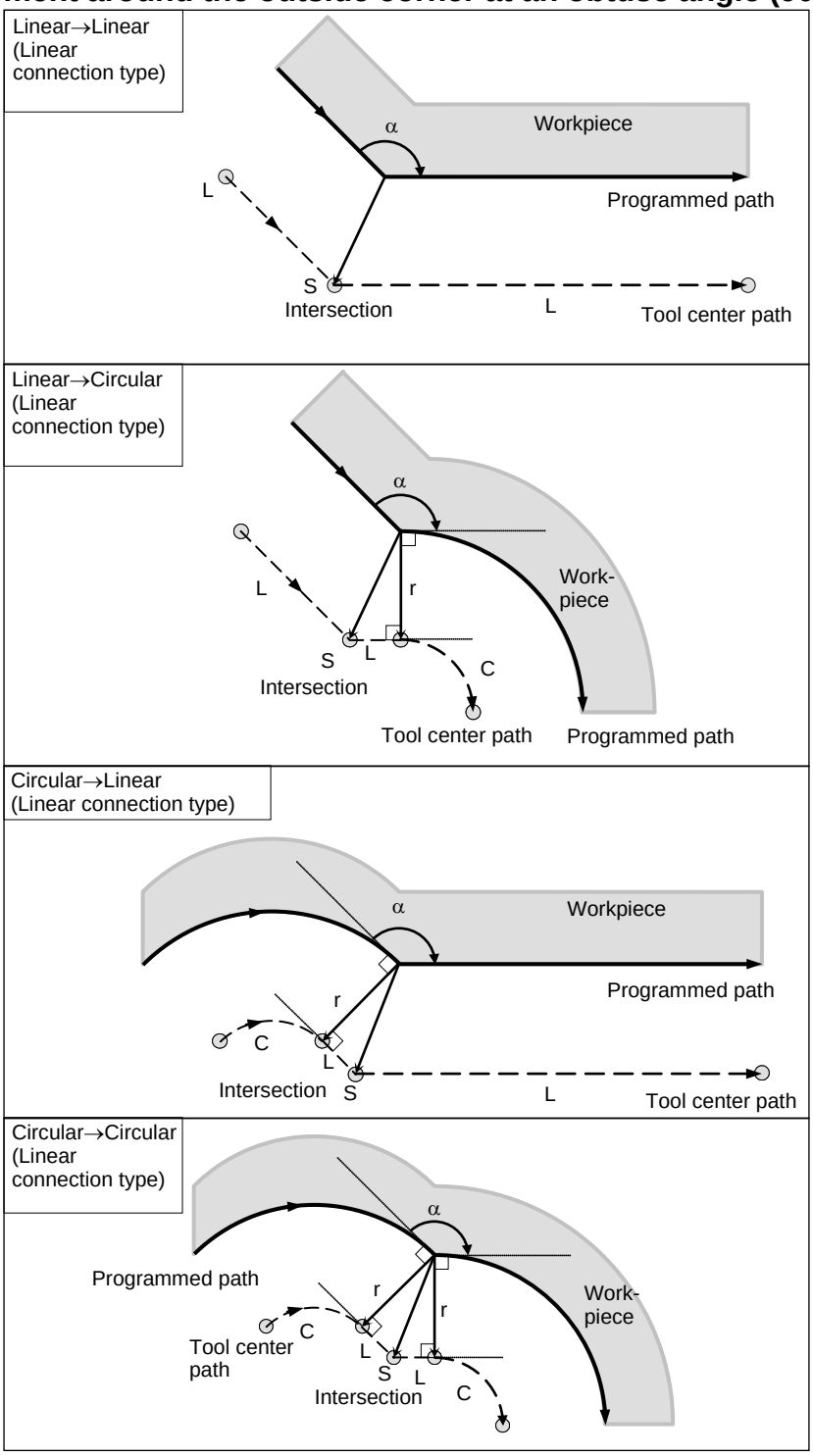


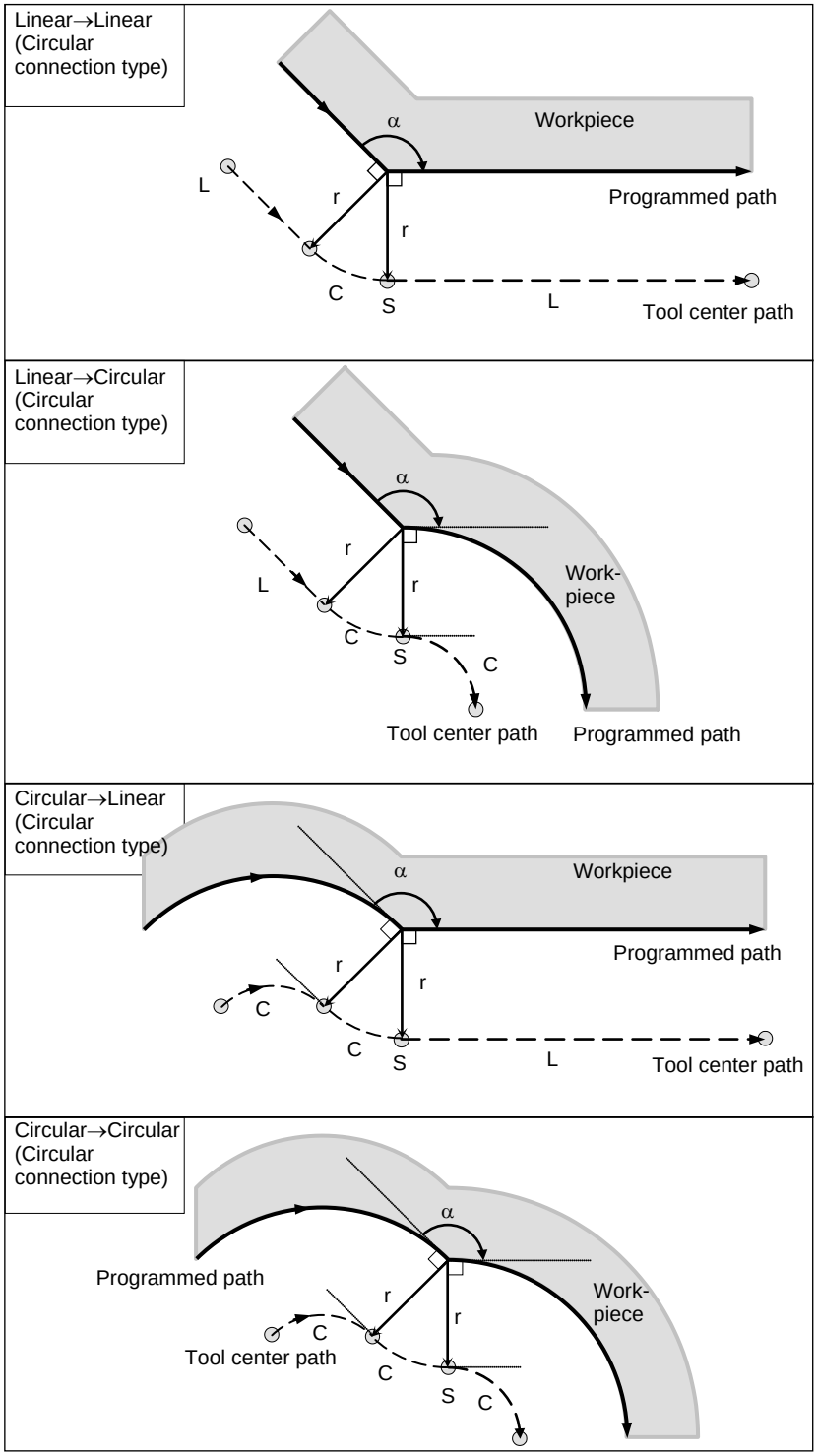
- **Tool movement around the inside (α≤1°) with an abnormally long vector, linear → linear**



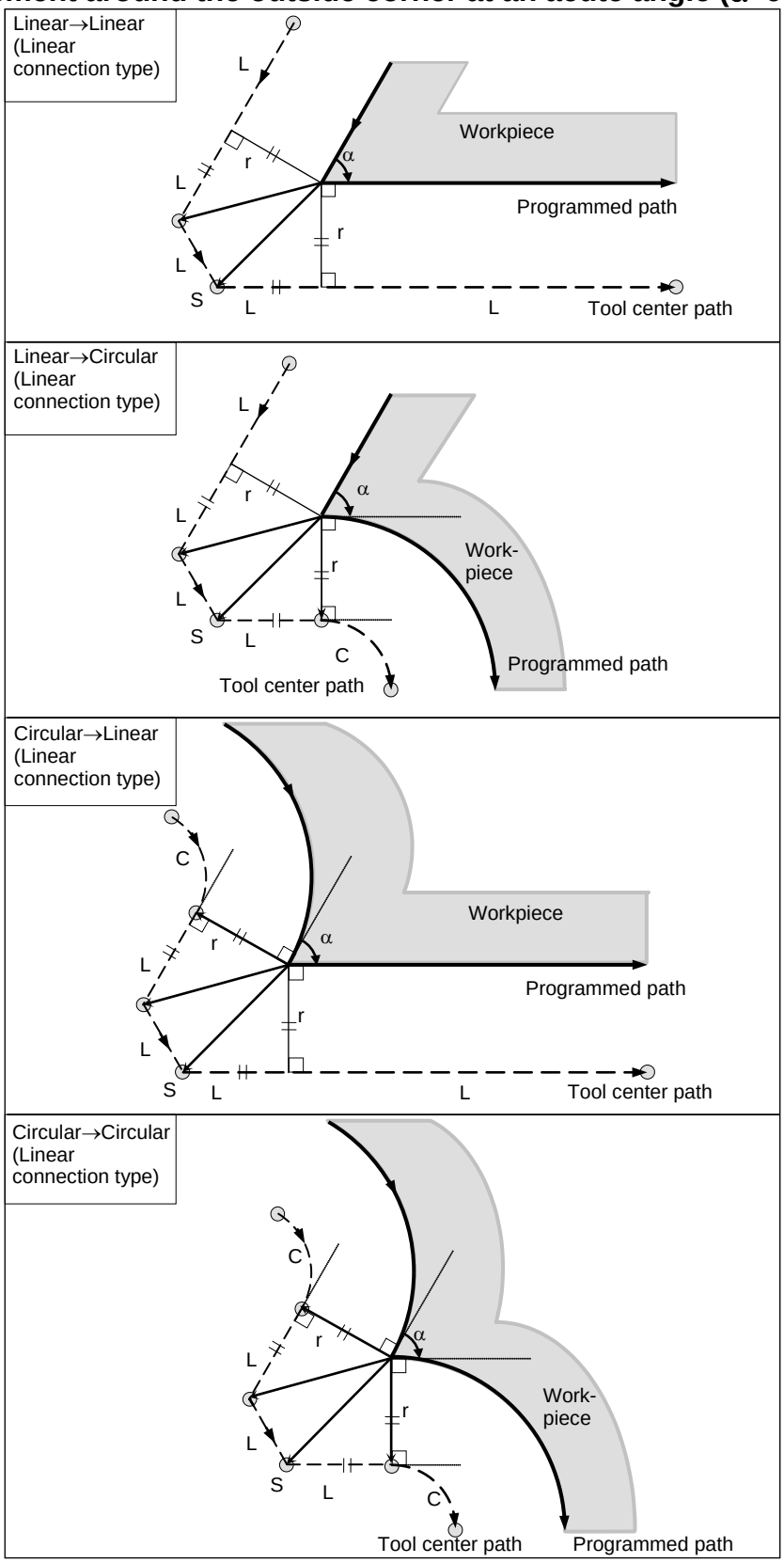
Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

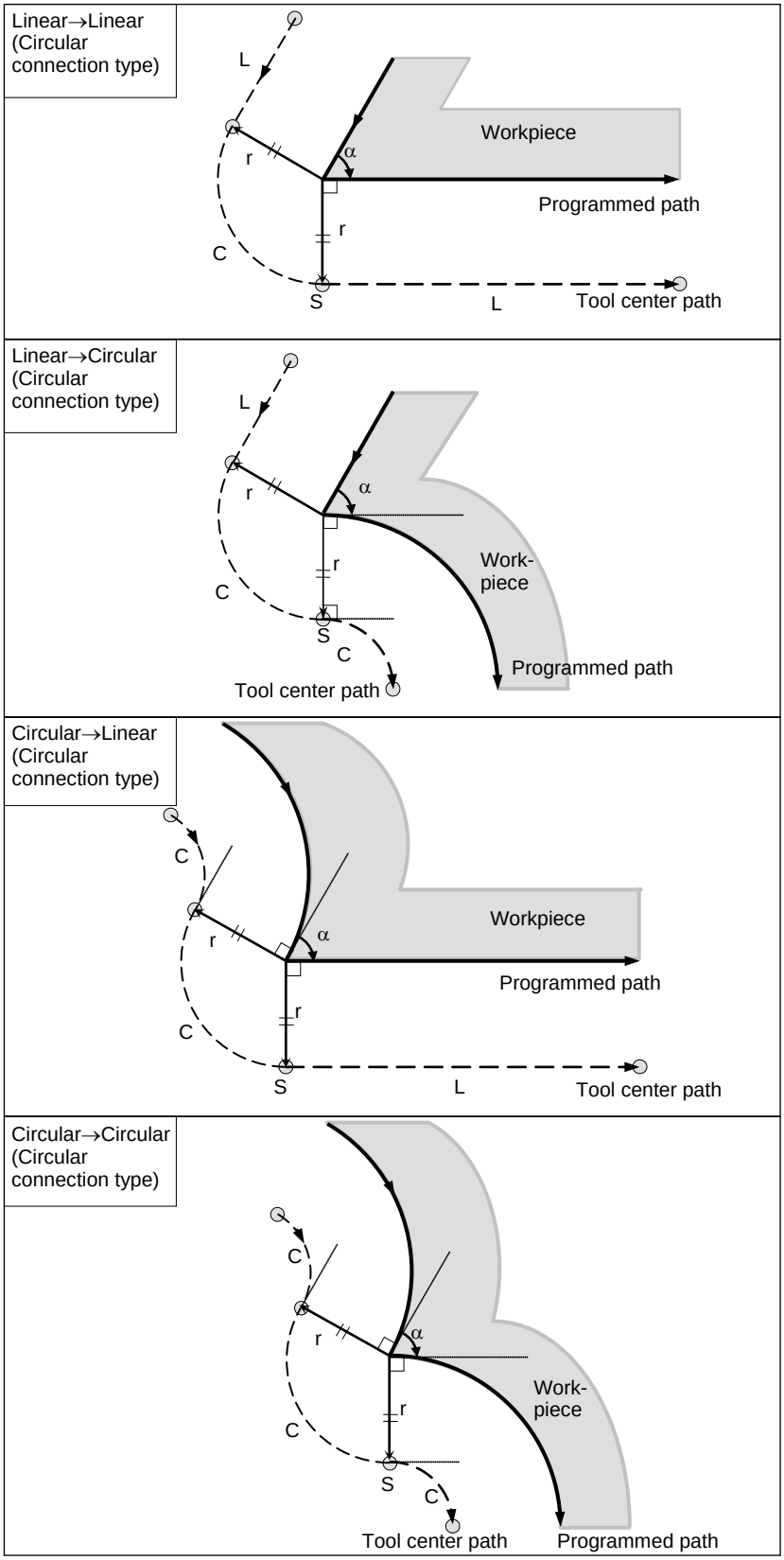
- **Tool movement around the outside corner at an obtuse angle (90°≤α<180°)**




Linear→Linear
(Circular
connection type)
α
Workpiece
L
r
Programmed path
r
C
S
L
Tool center path
Linear→Circular
(Circular
connection type)
α
r
L
r
Work-
piece
C
S
C
Tool center path
Programmed path
Circular→Linear
(Circular
connection type)
α
Workpiece
Programmed path
r
r
C
C
S
L
Tool center path
Circular→Circular
(Circular
connection type)
α
Programmed path
r
Work-
piece
C
r
Tool center path
C
S
C

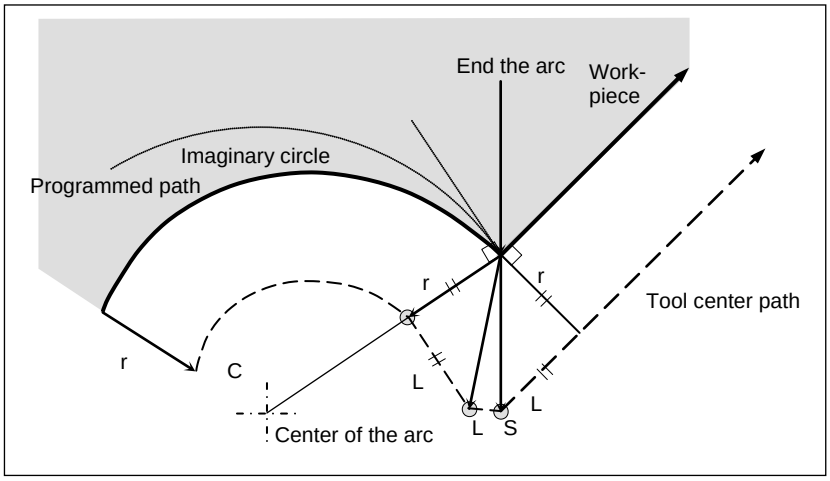
- **Tool movement around the outside corner at an acute angle (α<90°)**




Linear→Linear
(Circular
connection type)
L
r
α
Workpiece
Programmed path
r
C
S
L
Tool center path
Linear→Circular
(Circular
connection type)
L
r
α
r
Work-
piece
C
S
C
Programmed path
Tool center path
Circular→Linear
(Circular
connection type)
C
r
α
Workpiece
Programmed path
r
C
S
L
Tool center path
Circular→Circular
(Circular
connection type)
C
r
α
Work-
piece
r
C
S
C
Tool center path
Programmed path

- When it is exceptional
End point for the arc is not on the arc

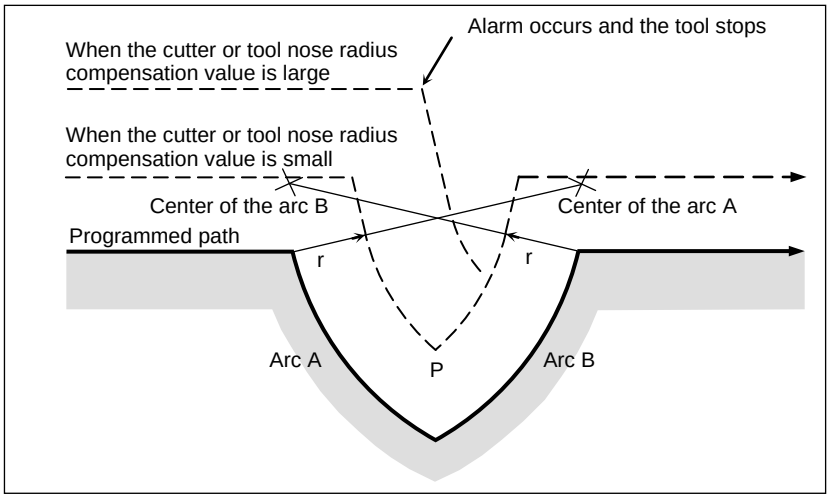
If the end of a line leading to an arc is not on the arc, the system assumes that the cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end point. Based on this assumption, the system creates a vector and carries out compensation. The same description applies to tool movement between two circular paths.



There is no inner intersection

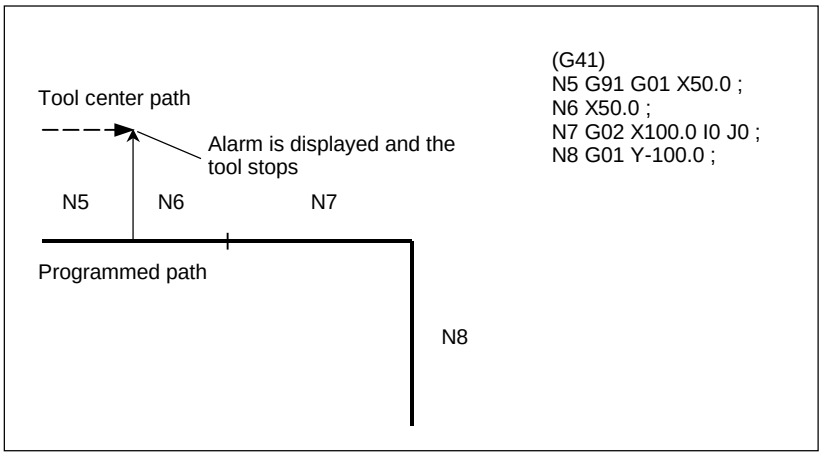
If the tool radius / tool nose radius compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool radius · tool nose radius compensation. When this is predicted, alarm PS0033, “NO INTERSECTION AT G41/G42” occurs at the end of the previous block and the tool is stopped.

For example, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool radius · 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 point

If the center of the arc is identical with the start point or end point, alarm PS0041, “INTERFERENCE IN G41/G42” is displayed, and the tool will stop at the start point of the preceding block of the arc.



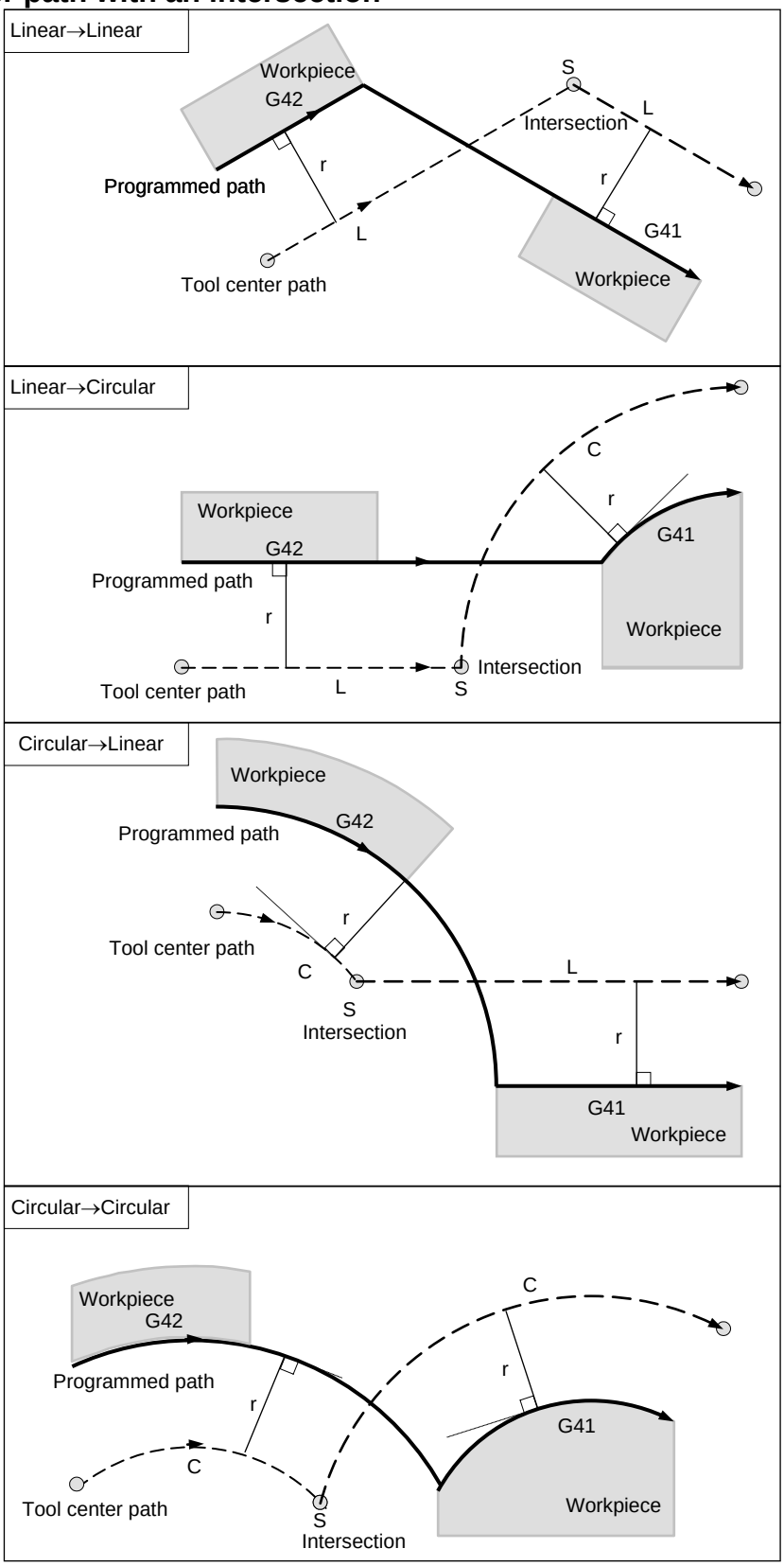
- Change in the offset direction in the offset mode

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

G code \ Sign of compensation	+	–
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

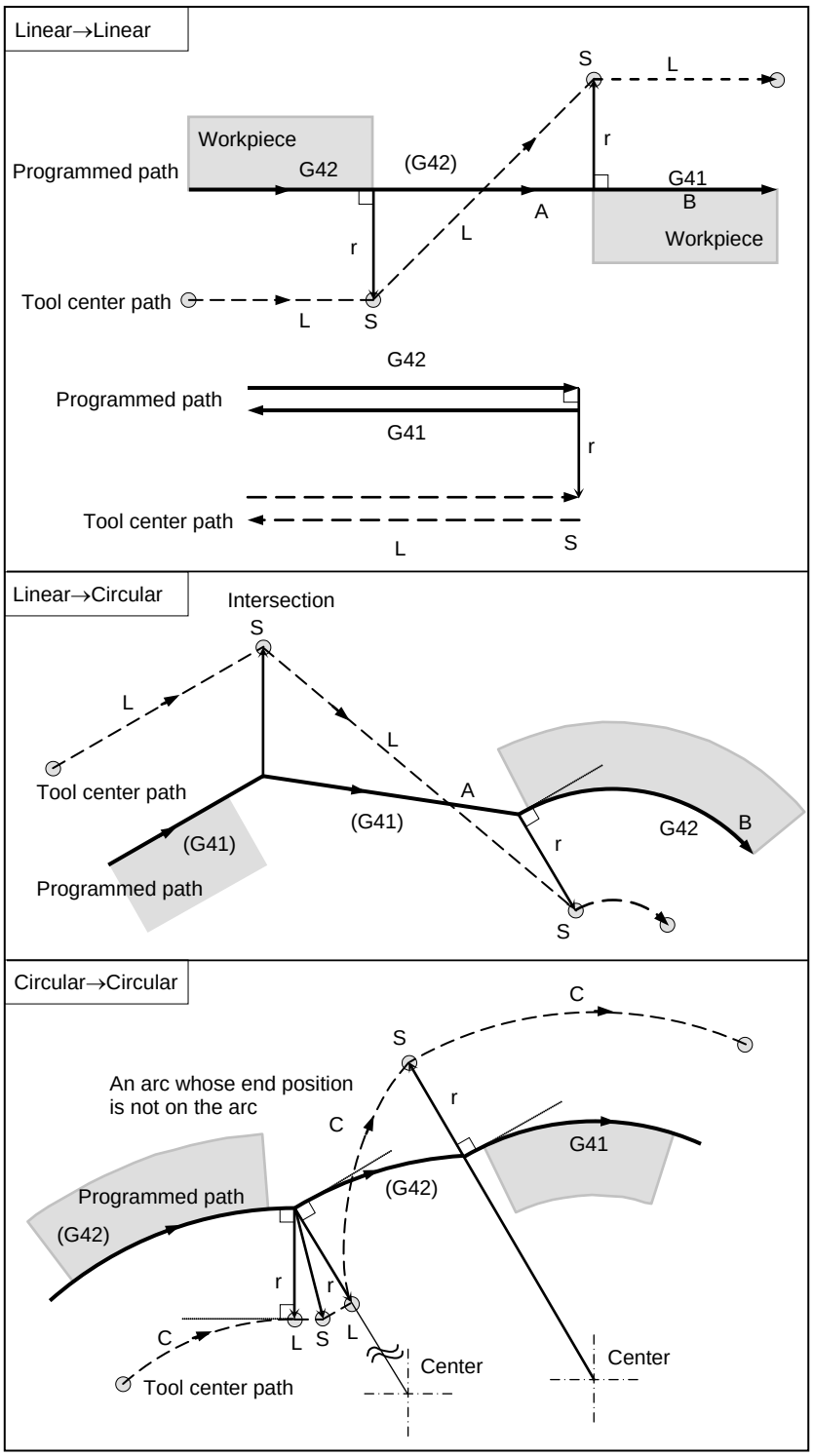
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block.
However, the change is not available in the start-up block and the block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

- Tool center path with an intersection



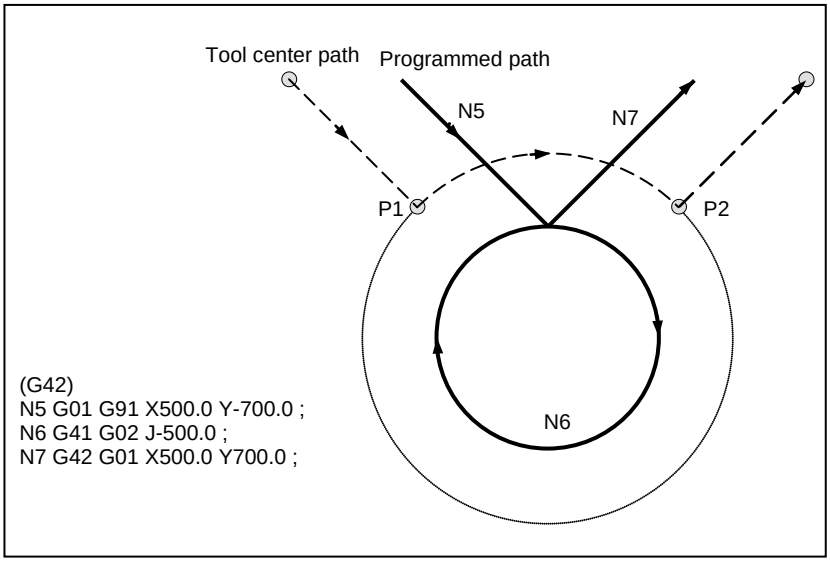
- Tool center path without an intersection

When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.



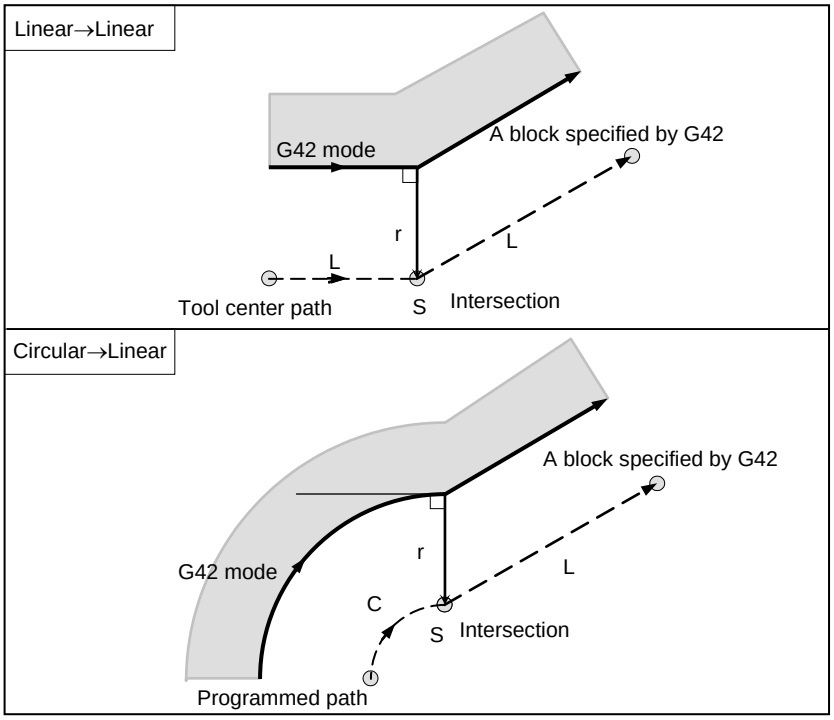
- The length of tool center path larger than the circumference of a circle

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.
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.



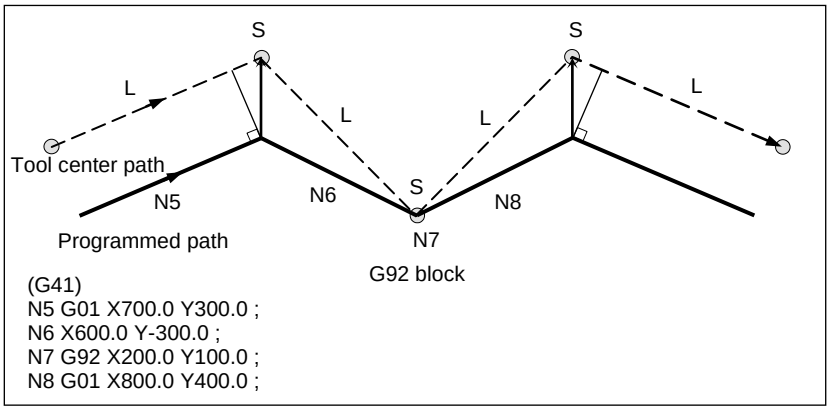
- Cutter compensation G code in the offset mode

The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining inner or outer side, by commanding the cutter compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct circular motion will not be obtained.
When the direction of offset is expected to be changed by the command of cutter compensation G code (G41, G42), see "Change in the offset direction in the offset mode".



- Command canceling the offset vector temporarily

During offset mode, if G92 (workpiece coordinate system setting) or G52 (local coordinate system setting) is commanded, the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.
In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled.
Also when restored to offset mode, the tool moves directly to the intersecting point.



Before specifying G28 (reference position return), G29 (movement from reference position), G30 (second, third, and fourth reference position return), and G53 (machine coordinate system selection) commands, cancel offset mode, using G40. If an attempt is made to specify any of the commands in offset mode, the offset vector temporarily disappears.

- If I, J, and K are specified in a G00/G01 mode block

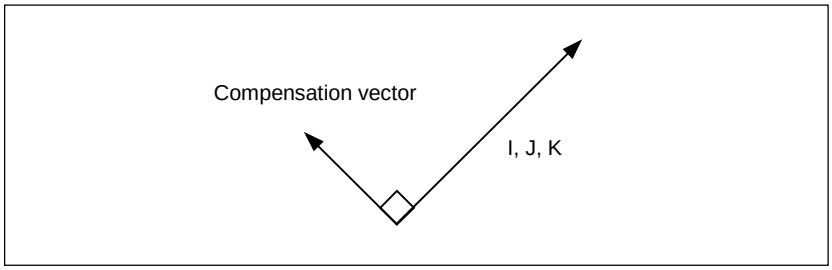
At the start of cutter compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This makes it possible to change the compensation direction intentionally.

IJ type vector (XY plane)

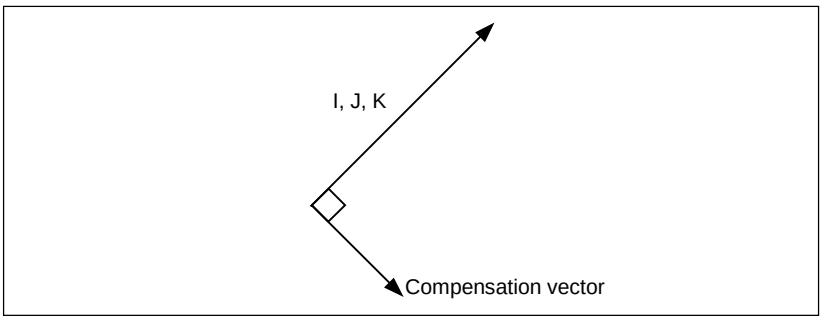
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) It is assumed that the compensation vector (IJ type vector) is the vector with a size equal to the compensation value and vertical to the direction specified by I and J, without performing intersection calculation on the programmed path. I and J can be specified both at the start of cutter compensation and in that mode. If they are specified at the start of compensation, any start-up type set in the appropriate parameter will be invalid, and an IJ type vector is assumed.

Offset vector direction

In G41 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the left side.



In G42 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the right side.

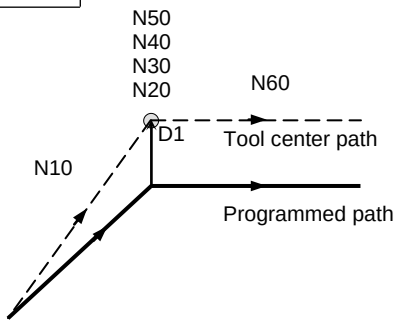


Example

If I and J are specified at the start of compensation (with tool movement)

```
      (G40)
N10 G91 G41 X100.0 Y100.0
      I1 D1 ;
N20 G04 X1000 ;
N30 G01 F1000 ;
N40 S300 ;
N50 M50 ;
N60 X150. ;
```

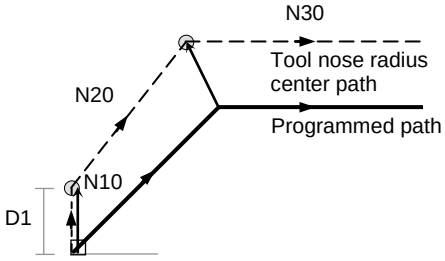
Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.



If I and J are specified at the start of compensation (without tool movement)

```
      (G40)
N10 G41 I1 D1 ;
N20 G91 X100. Y100. ;
N30 X150. ;
```

Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.

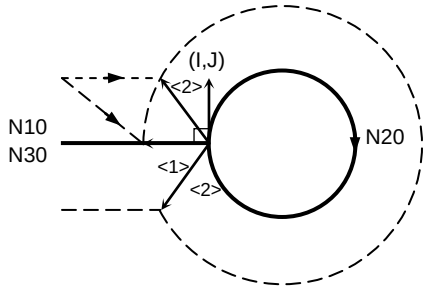


If I and J are specified at the start of compensation (with tool movement)

(G17 G41 G91 D1)
N10 G00 X150. J50. ;
N20 G02 I50. ;
N30 G00 X-150. ;

Note) In N10, a vector is specified with a size of D1 in the direction vertical to the Y axis, using J50.

<1> IJ type vector
<2> Vector determined with intersection calculation



Tool center path

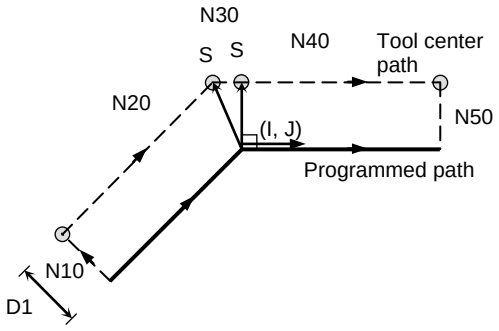
Programmed path

Path determined with intersection calculation

If I and J are specified in a block without tool movement in compensation mode

Start-up/cancel type C

N10 G41 D1 G01 F1000 ;
N20 G91 X100. Y100. ;
N30 I10. ;
N40 X150. ;
N50 G40 ;

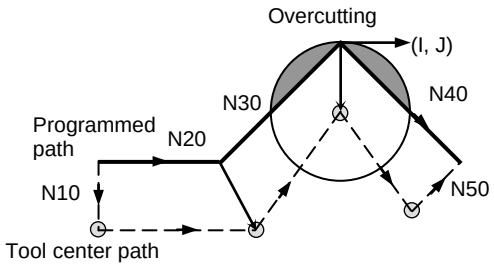


Limitation

If an IJ type vector is specified, tool interference may occur due to that vector alone, depending on the direction. If this occurs, no interference alarm will occur, or no interference avoidance will be performed. Overcutting may, therefore, result.

Start-up/cancel
Type C

N10 G42 D1 F1000 ;
N20 G91 X100. ;
N30 X100. Y-100. I10. ;
N40 X100. Y-100. ;
N50 G40 ;



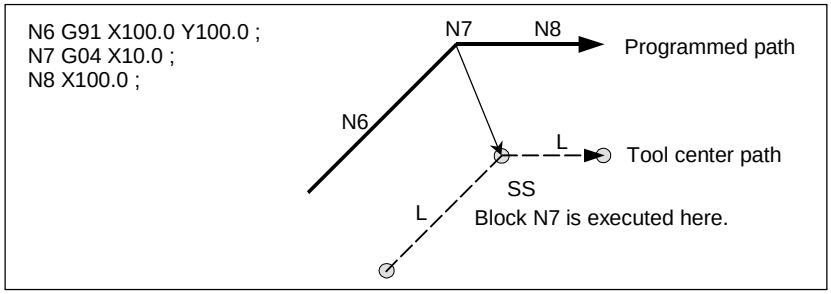
- A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

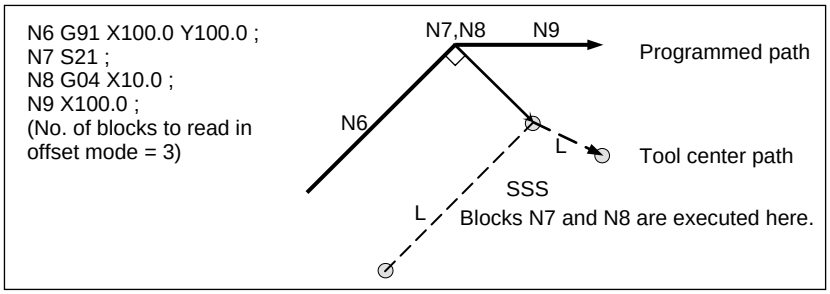
M05 ;	:	M code output
S21 ;	:	S code output
G04 X10.0 ;	:	Dwell
G22 X100000 ;	:	Machining area setting
G10 L11 P01 R10.0 ;	:	Cutter compensation value setting/changing
(G17) Z200.0 ;	:	Move command not included in the offset plane.
G90 ;, O10 ;, N20 ;	:	G, O, and N codes only
G91 X0.0 ;	:	Move distance is zero.

- A block without tool movement specified in offset mode

Unless the number of blocks without movement consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)) in offset mode, the vector and the tool center path will be as usual. This block is executed at the single block stop point.

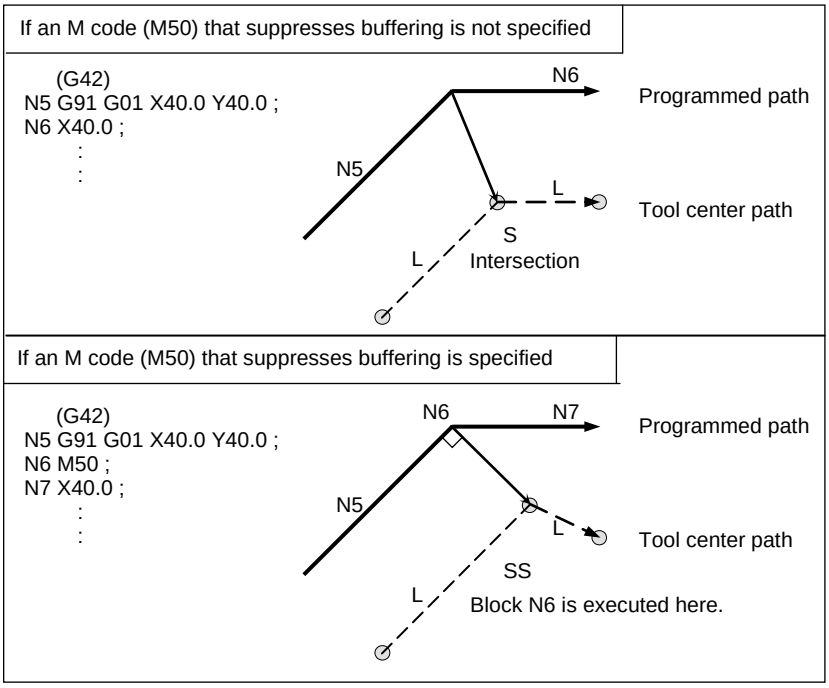


In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter No. 19625). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.



- If an M/G code that suppresses buffering is specified

If an M/G code that suppresses buffering is specified in offset mode, it is no longer possible to read and analyze subsequent blocks regardless of the number of blocks to read in offset mode, which is determined by parameter No. 19625. Then, intersection calculation and a interference check, described later, are no longer possible. If this occurs, overcutting may occur because a vertical vector is output in the immediately preceding block.

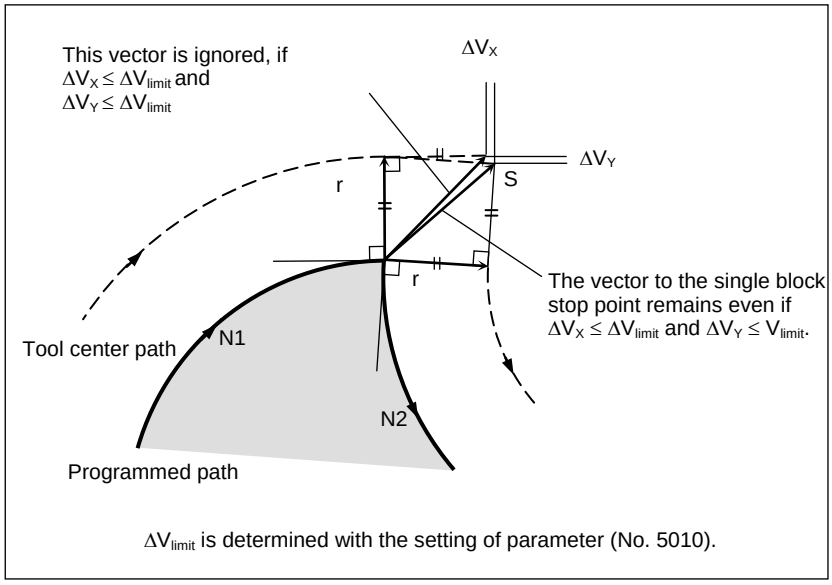


- Workpiece coordinate system or local coordinate system command in the offset mode

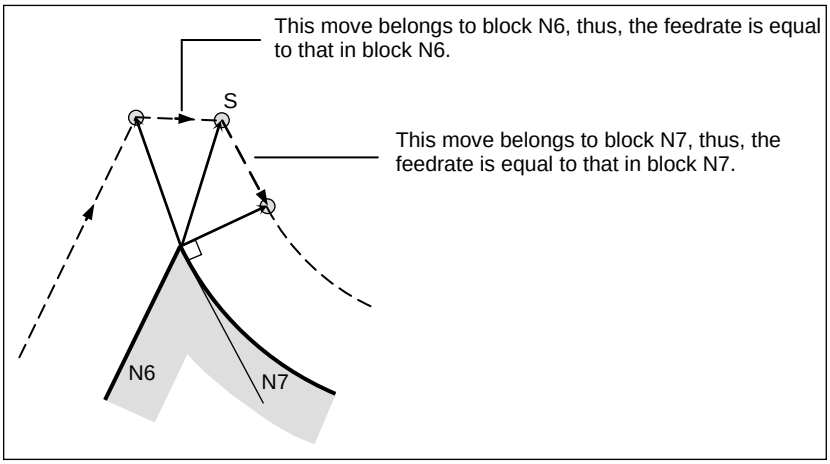
If the local coordinate system (G52) or workpiece coordinate system (G92) is specified in the cutter compensation (G41 or G42) mode, G52 or G92 is assumed to be a buffering masked G code. The subsequent blocks are not executed until the G52 or G92 block is executed.

- Corner movement

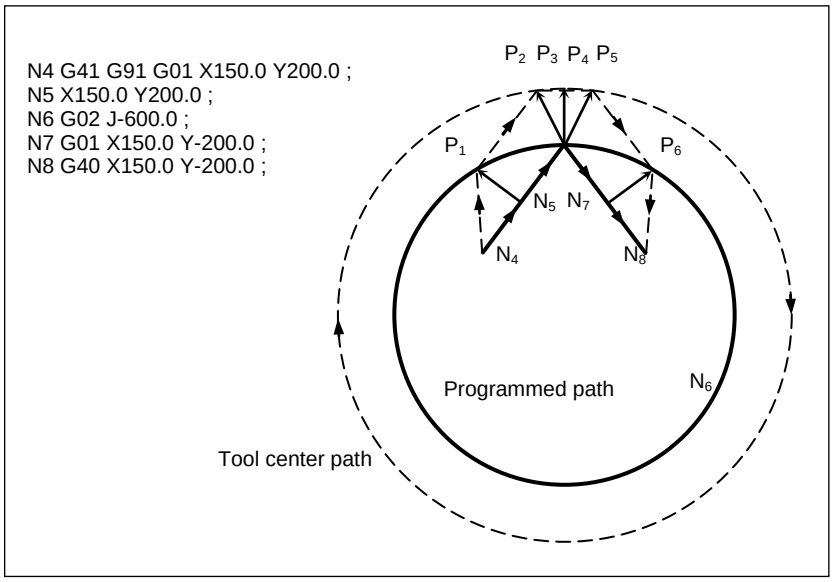
When two or more offset vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.
If these vectors almost coincide with each other (the distance of corner movement between the vectors is judged short due to the setting of parameter No. 5010), corner movement is not performed. In this case, the vector to the single block stop point takes precedence and remains, while other vectors are ignored. This makes it possible to ignore the very small movements arising from performing cutter 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.

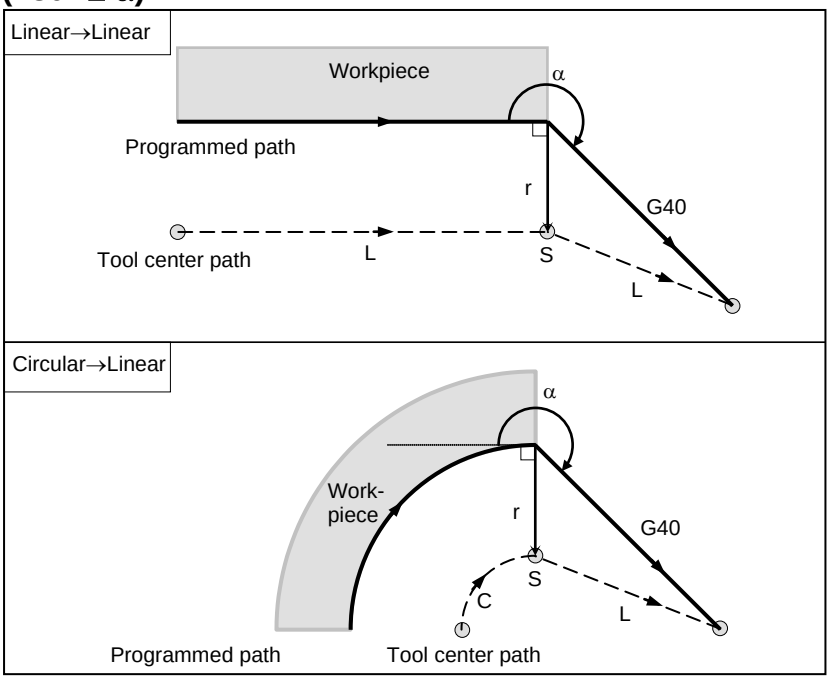
- Interruption of manual operation

For manual operation during the offset mode, see "Manual Absolute ON and OFF."

6.6.4 Tool Movement in Offset Mode Cancel

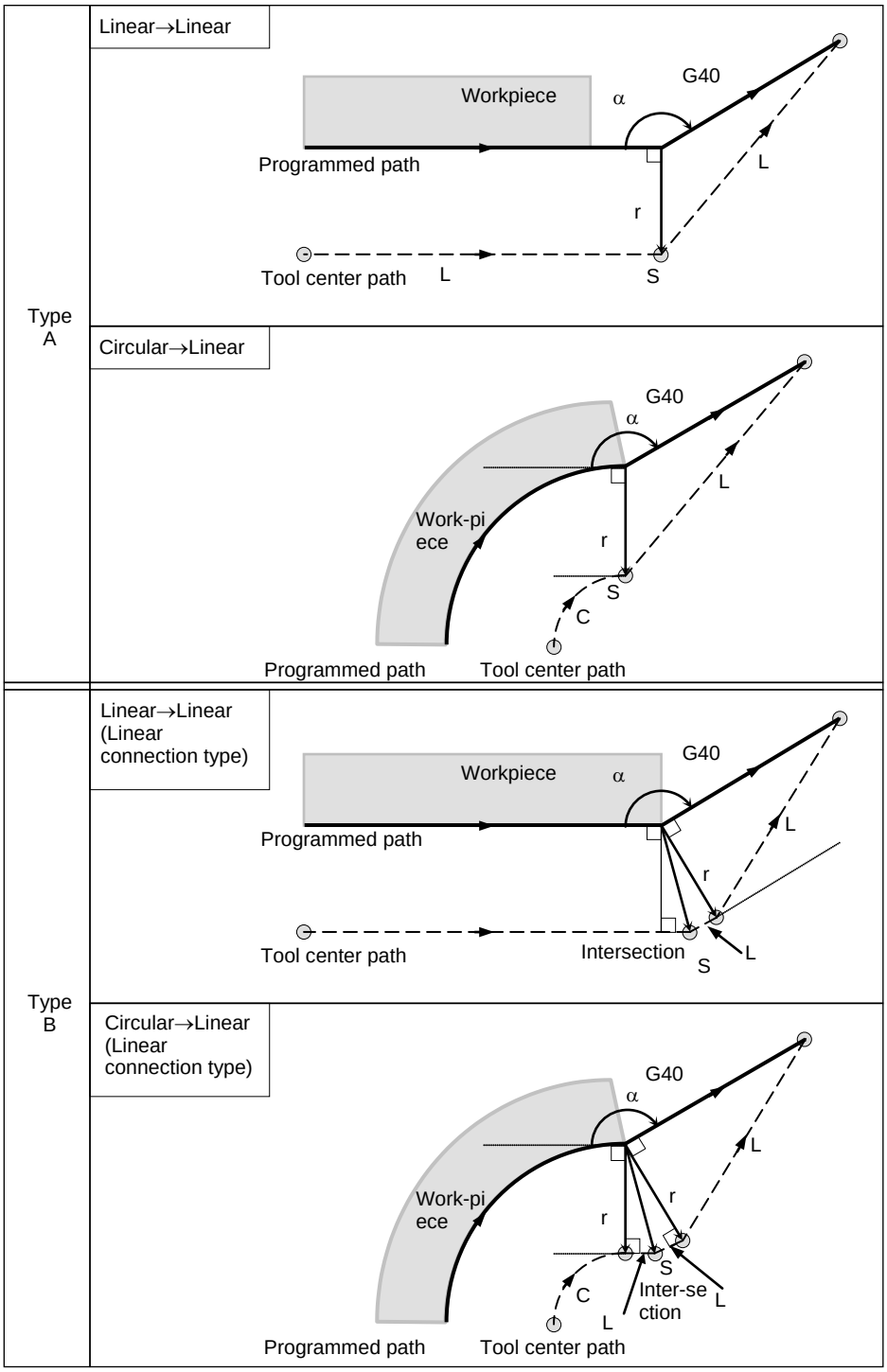
Explanation

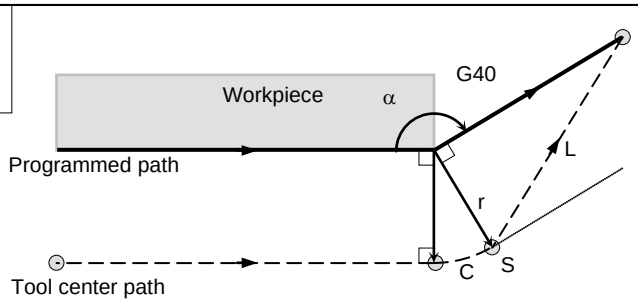
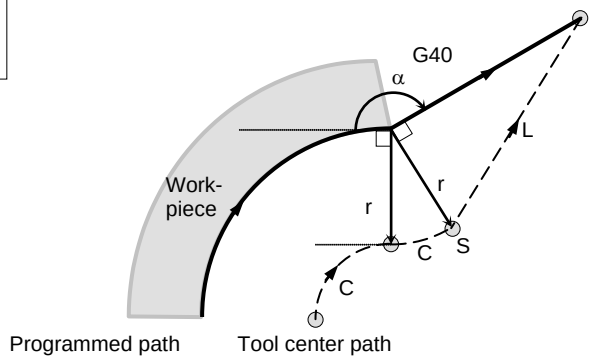
- **If the cancel block is a block with tool movement, and the tool moves around the inside (180° ≤ α)**



- **If the cancel block is a block with tool movement, and the tool moves around the outside at an obtuse angle (90° ≤ α < 180°)**

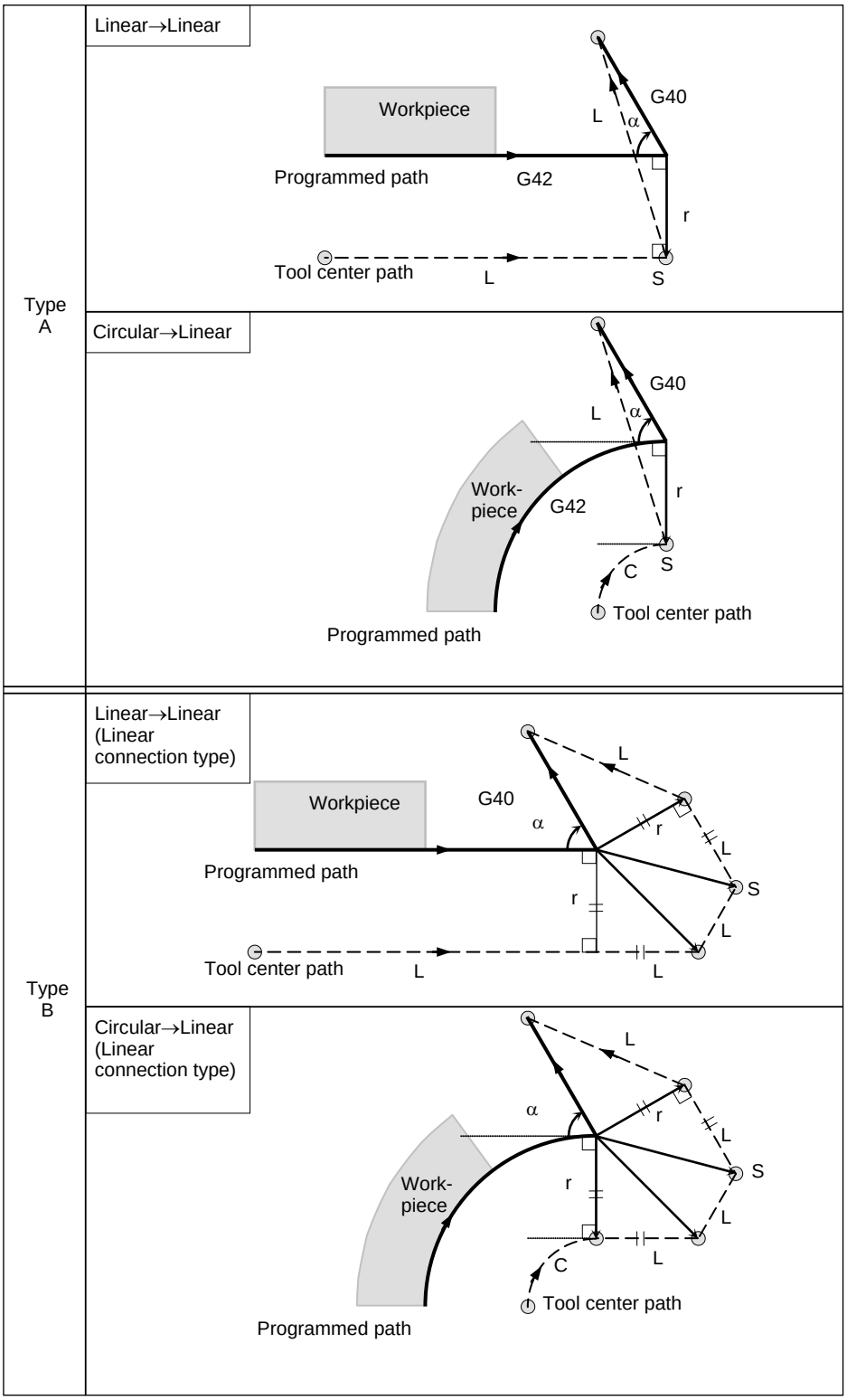
Tool path has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

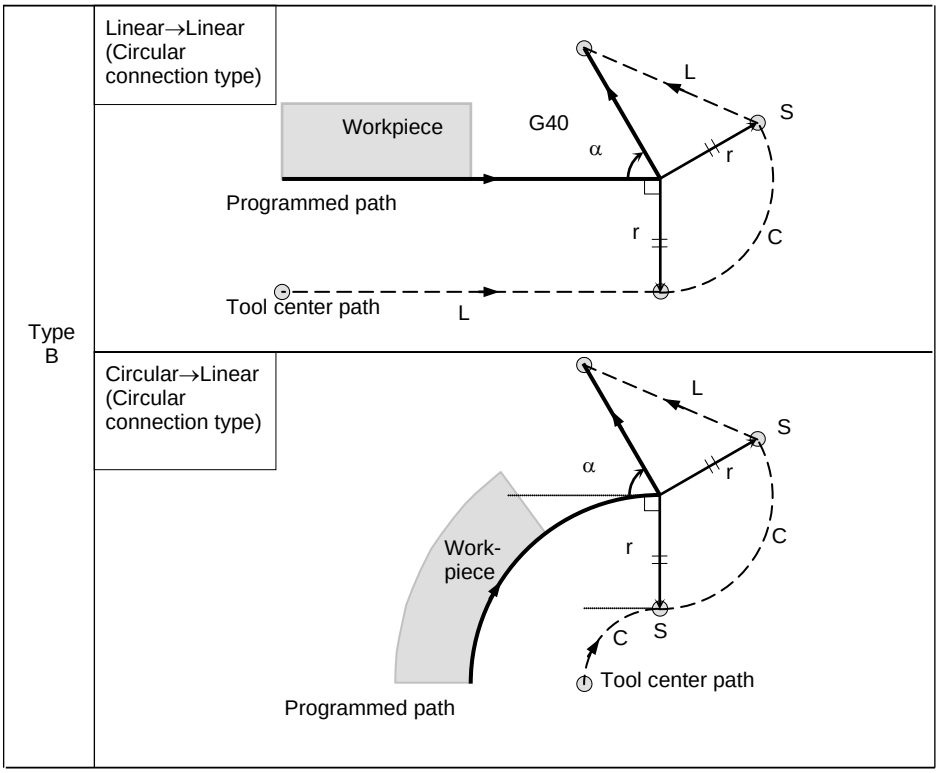


Type B	Linear→Linear (Circular connection type)	
	Circular→Linear (Circular connection type)	

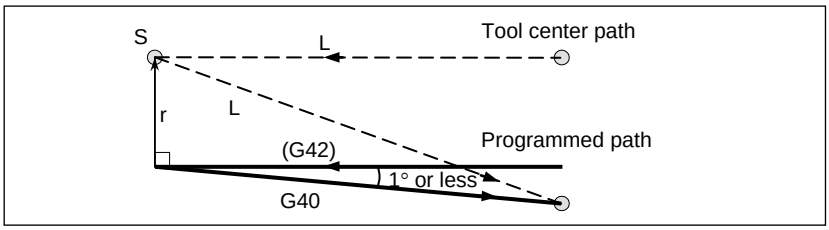
- **If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle (α<90°)**

Tool path has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.





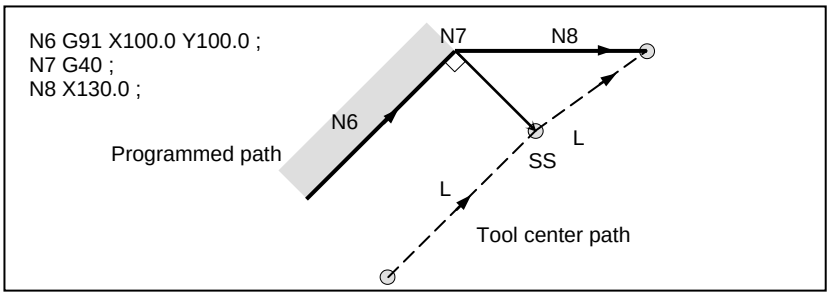
- **If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle of 1 deg or less in a linear → linear manner (α≤1°)**



- **A block without tool movement specified together with offset cancel**

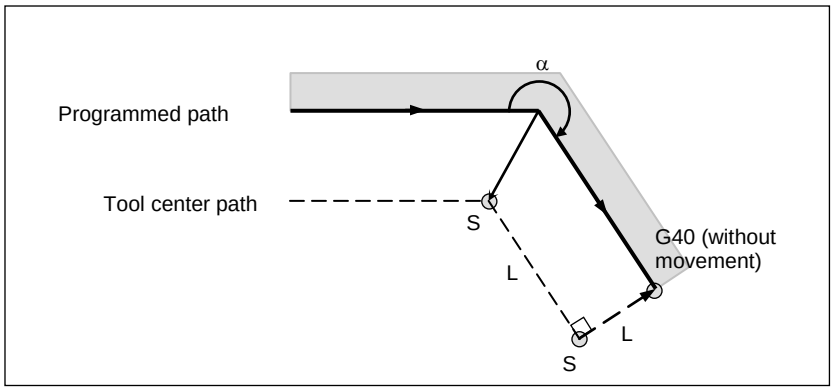
For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the tool radius · 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.



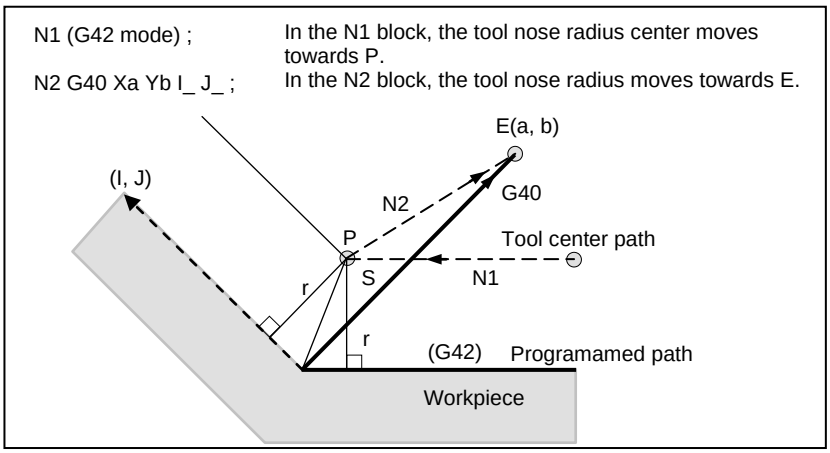
For type C

The tool shifts by the compensation value in the direction vertical to the block preceding the cancel block.

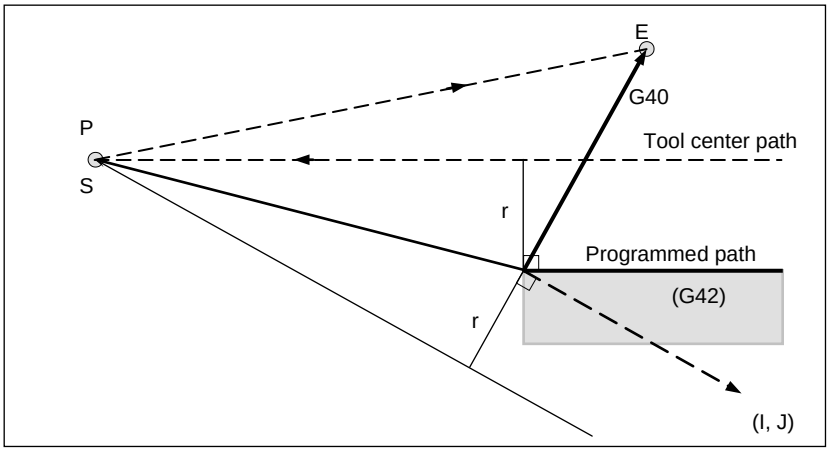


- Block containing G40 and I_J_K_ The previous block contains G41 or G42

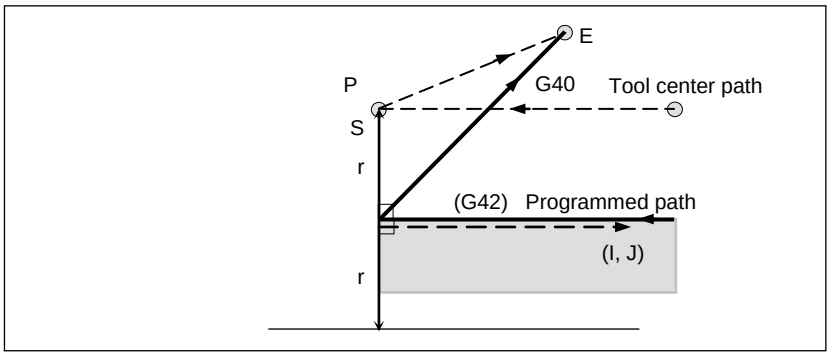
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end point determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified.

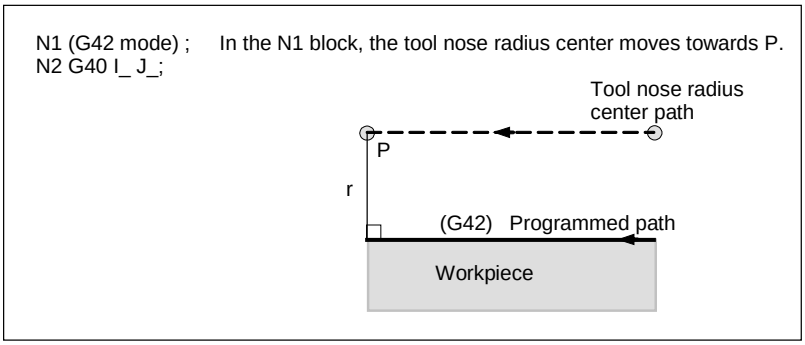


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.

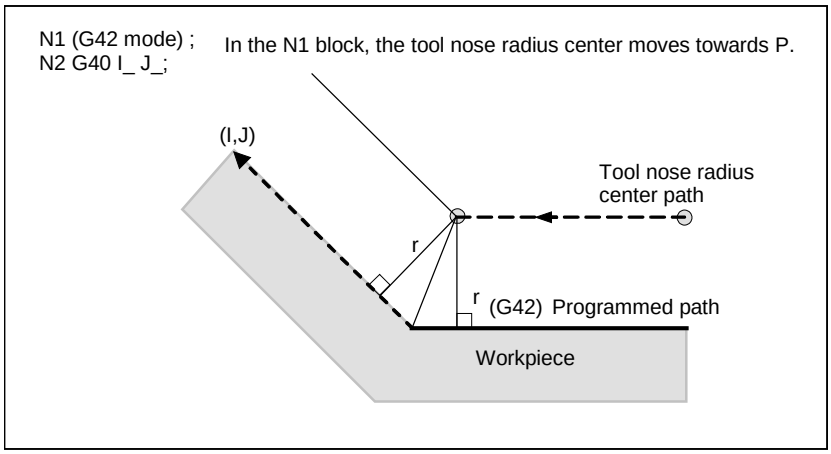


- Block containing G40 and I_J_K_ without tool movement specified The previous block contains G41 or G42

When bit 3 (GNI) of parameter No.5000 is set to 0, G40 block is considered no tool movement and the tool comes to the normal position to the previous block at the end of the previous block. (It differs from Series 16*i*/18*i*/21*i*, Series 0*i*-C.)



When bit 3 (GNI) of parameter No.5000 is set to 1, 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. (Series 16*i*/18*i*/21*i*, Series 0*i*-C compatible specification)

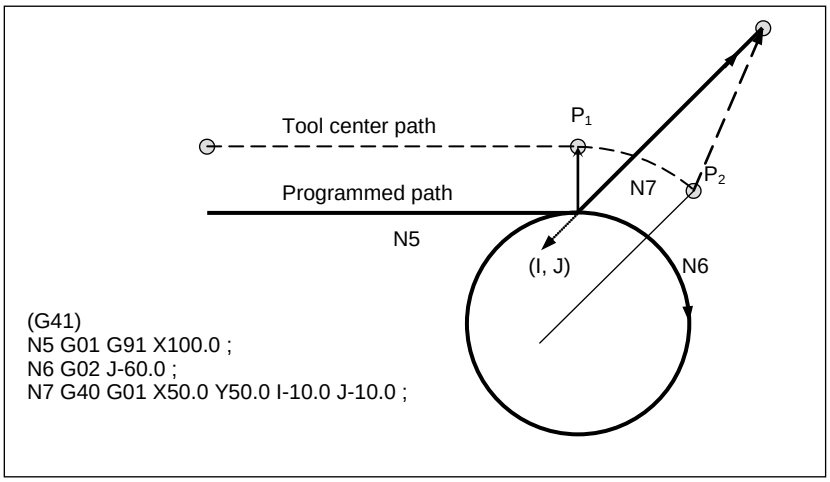


NOTE

When using the program used in Series 16*i*/18*i*/21*i* or Series 0*i*-C with the Series 0*i*-F/0*i*-D, set the bit 3 (GNI) of parameter No.5000 to 1.
Behavior is different when the bit 3 (GNI) of parameter No.5000 set to 1, and it may cause excessive cutting or remain uncut.

- 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.



6.6.5 Prevention of Overcutting Due to Tool Radius / Tool Nose Radius Compensation

Explanation

- Machining a groove smaller than the diameter of the tool

Since the cutter compensation forces the tool center path to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.

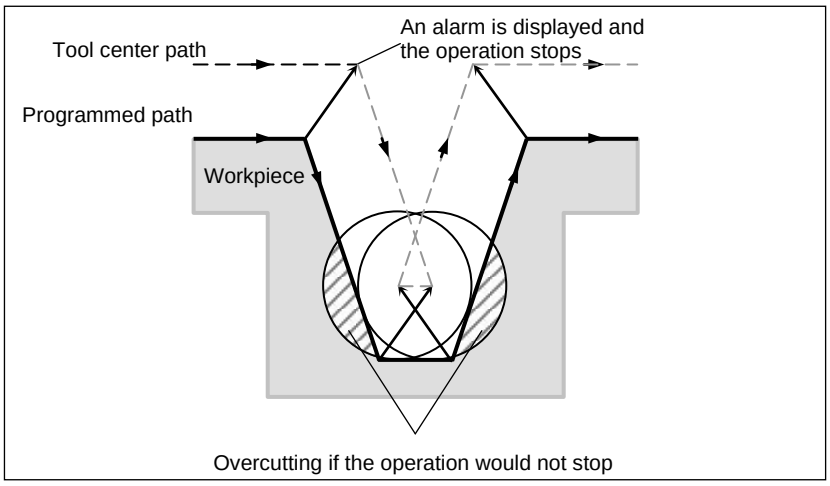


Fig. 6.6.5 (a) Machining a groove smaller than the diameter of the tool

- Machining a step smaller than the tool radius

For a figure in which a workpiece step is specified with an arc, the tool center path will be as shown in Fig. 6.6.5 (b). If the step is smaller than the tool radius, the tool center path usually compensated as shown in Fig. 6.6.5 (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.

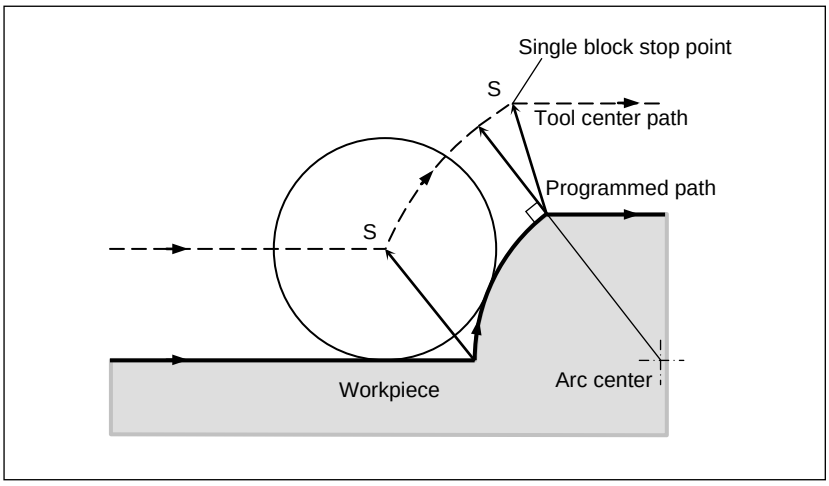


Fig. 6.6.5 (b) Machining a step larger than the tool radius

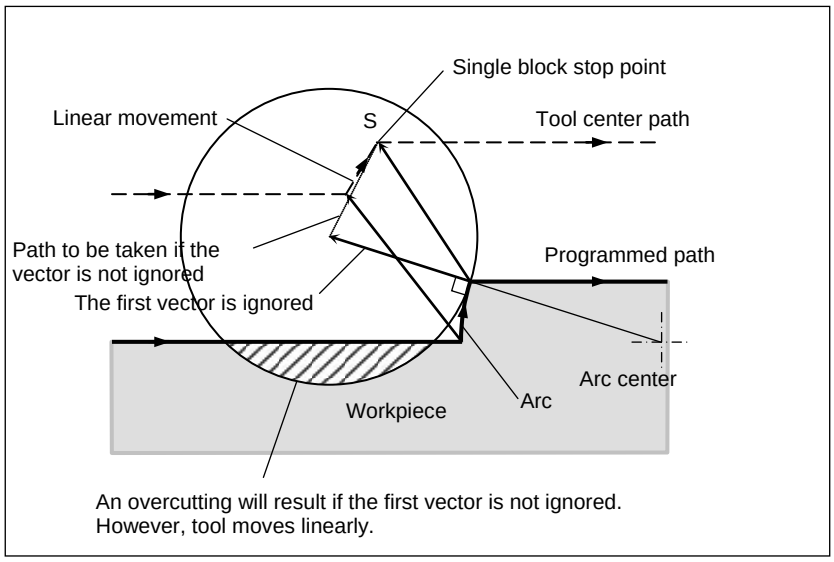


Fig. 6.6.5 (c) Machining a step smaller than the tool radius

- Starting compensation and cutting along the Z-axis

It is usually used such a method that the tool is moved along the Z axis after the cutter compensation (normally XY plane) is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.

Let us consider the program in the Fig. 6.6.5 (d), assuming the number of blocks to read in cutter compensation mode (parameter No. 19625) to be 3.

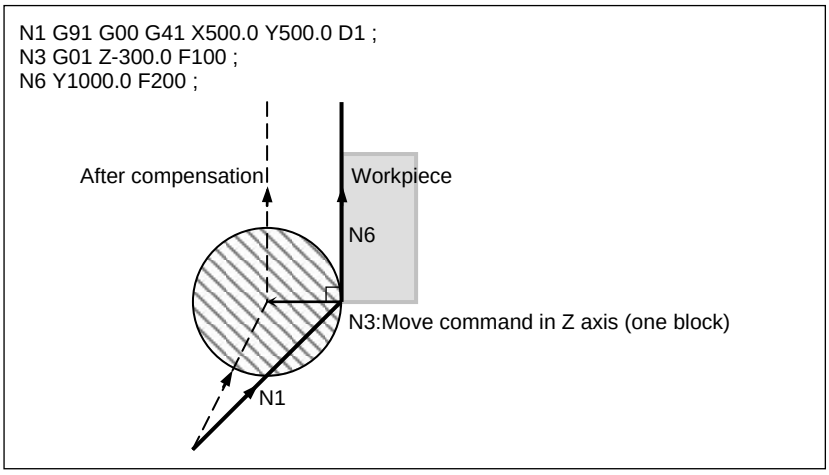


Fig. 6.6.5 (d)

In the program example in the Fig. 6.6.5 (d), when executing block N1, all blocks to N6 are also entered into the buffer storage. By the relationship among them, the correct compensation is performed as in the Fig. 6.6.5 (d).

Then, suppose that the block N3 (move command in Z axis) is divided into N3 and N5 in the Fig. 6.6.5 (e).

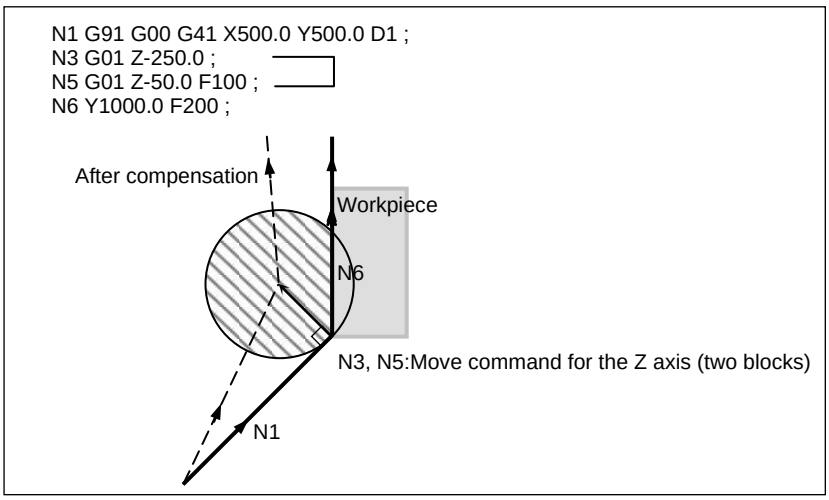


Fig. 6.6.5 (e)

At this time, because the number of blocks to read is 3, blocks up to N5 can be read at the start of N1 compensation, but block N6 cannot be read. As a result, compensation is performed only on the basis of the information in block N1, and a vertical vector is created at the end of the compensation start block. Usually, therefore, overcutting will result as shown in the Fig. 6.6.5 (e).

In such a case, it is possible to prevent overcutting by specifying a command with the exactly the same direction as the advance direction immediately before movement along the Z axis beforehand, after the tool is moved along the Z axis using the above rule.

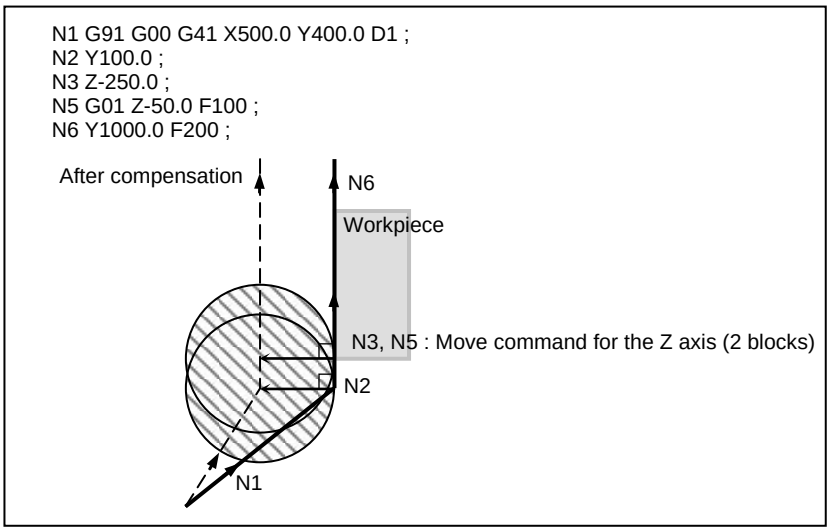


Fig. 6.6.5 (f)

As the block with sequence N2 has the move command in the same direction as that of the block with sequence N6, the correct compensation is performed.

Alternatively, it is possible to prevent overcutting in the same way by specifying an IJ type vector with the same direction as the advance direction in the start-up block, as in N1 G91 G00 G41 X500. Y500. I0 J1 D1;, after the tool has moved along the Z axis.

6.6.6 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

Explanation

- Condition under which an interference check is possible

To perform an interference check, it is necessary to read at least three blocks with tool movement. If, therefore, three or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as independent auxiliary function and dwell, are specified in succession, excessive or insufficient cutting may occur because an interference check fails. Assuming the number of blocks to read in offset mode, which is determined by parameter No. 19625, to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which an interference check is possible is

$(N - 3) \geq M$.

For example, if the maximum number of blocks to read in offset mode is 8, an interference check is possible even if up to five blocks without tool movement are specified. In this case, three adjacent blocks can be checked for interference, but any subsequent interference that may occur cannot be detected.

- Interference check method

Two interference check methods are available, direction check and circular angle check. Bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008 are used to specify whether to enable these methods.

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.

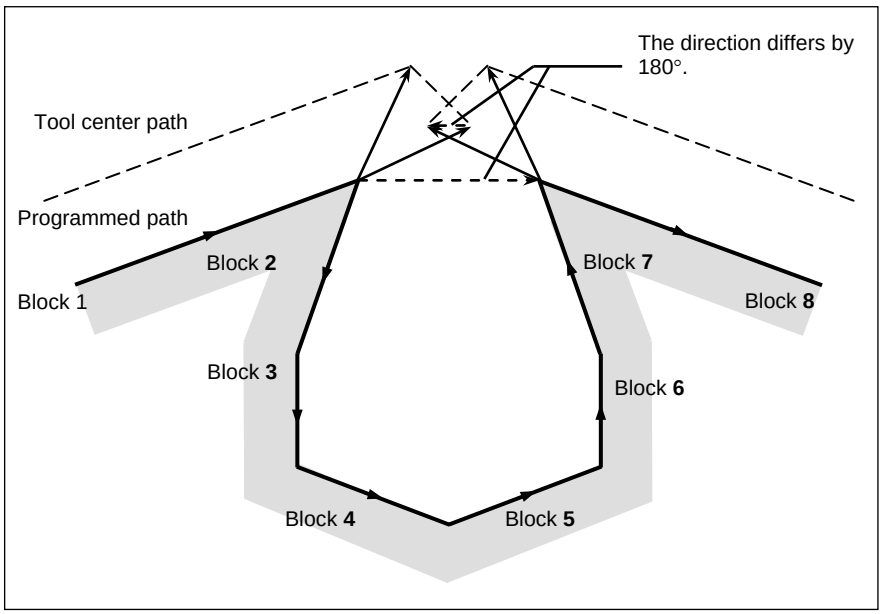
- Interference reference <1> (direction check)

Assuming the number of blocks to read during cutter compensation to be N, a check is first performed on the compensation vector group calculated in (block 1 - block 2) to be output this time and the compensation vector group calculated in (block N-1 - block N); if they intersect, they are judged to interfere. If no interference is found, a check is performed sequentially in the direction toward the compensation vector group to be output this time, as follows:

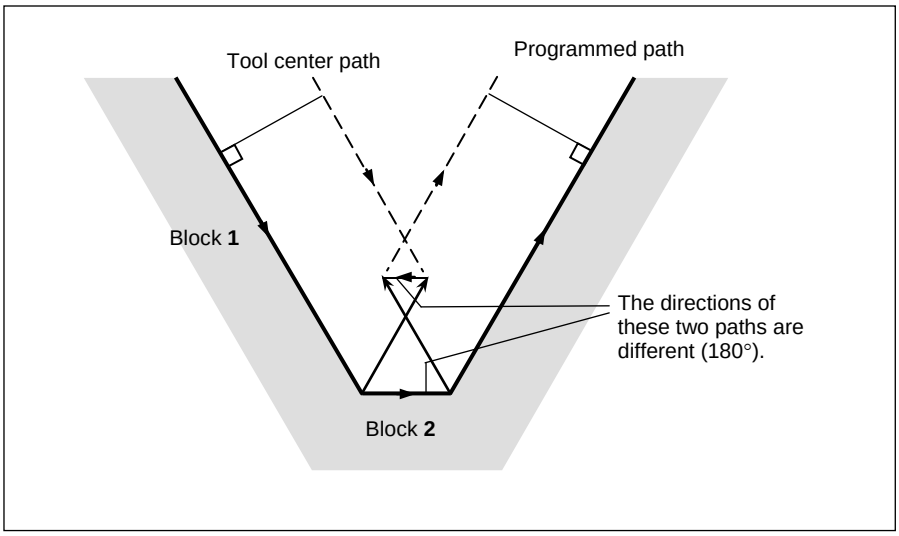
(Block 1 - block 2) and (block N-2 - block N-1)
(Block 1 - block 2) and (block N-3 - block N-2)
:
:
(Block 1 - block 2) and (block 2 - block 3)

Even if multiple number of compensation vector groups are generated, a check is performed on all pairs. The judgment method is as follows: For a check on the compensation vector group in (block 1 - block 2) and those in (block N-1 - block N), the direction vector from the specified (end point of block 1) to the (end point of block N-1) is compared with the direction vector from the (point resulting from adding the compensation vector to be checked to the end of block 1) to the (point resulting from adding the compensation vector to be checked to the end of block N-1), and if the direction is 90° or greater or 270° or less, they are judged to intersect and interfere. This is called a direction check.

Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 7 end-point vector)



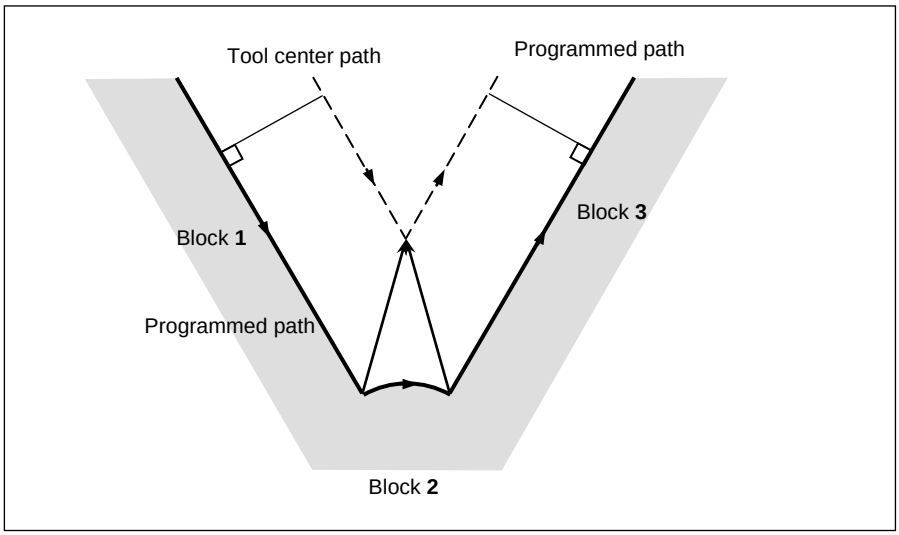
Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 2 end-point vector)



- Interference reference <2> (circular angle check)

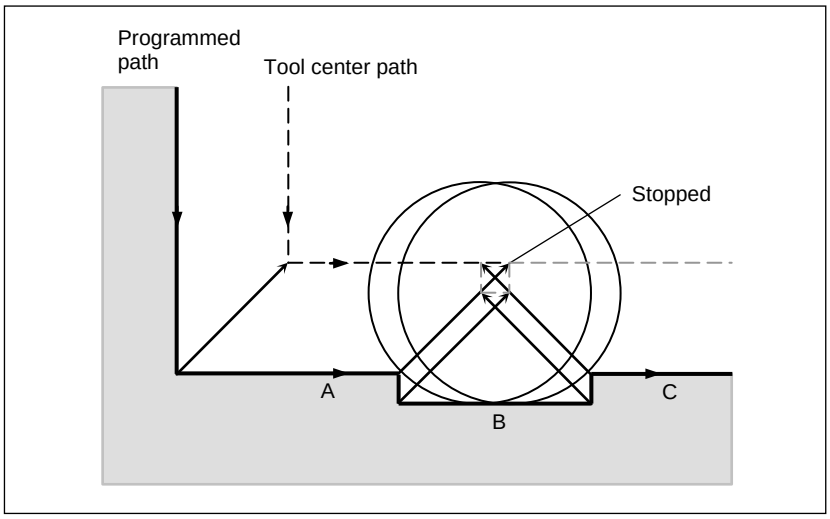
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed path and the circular angle of the start and end point of the post-compensation path, in addition to direction check <1>. If the difference is 180° or greater, the blocks are judged to interfere. This is called a circular angle check.

Example of <2> (if block 2 is circular and the start point of the post-compensation arc coincide with the end point)



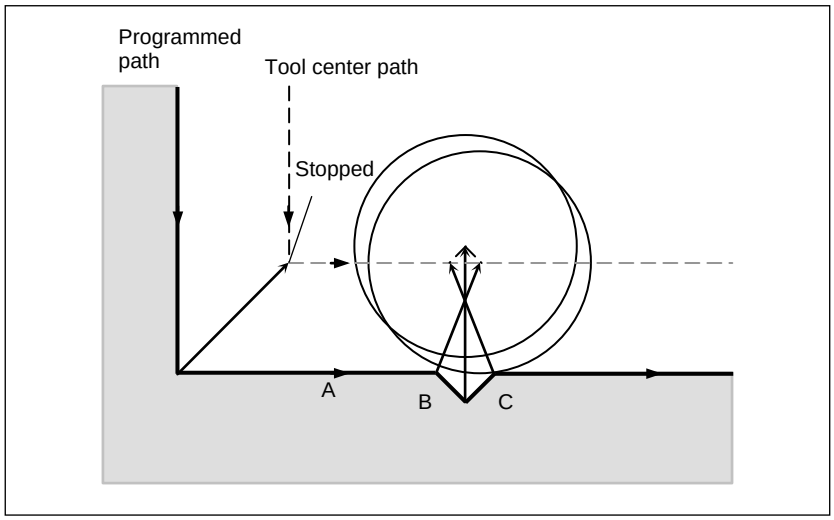
- When interference is assumed although actual interference does not occur

<1> Depression which is smaller than the tool radius · 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 cutter compensation, the tool stops and an alarm is displayed.

<2> Groove which is smaller than the tool radius · tool nose radius compensation value



Like <1>, an alarm is displayed because of the interference as the direction is reverse in block B.

6.6.6.1 Operation to be performed if an interference is judged to occur

The operation to be performed if an interference check judges that an interference (due to overcutting) occurs can be either of the following two, depending on the setting of bit 5 (CAV) of parameter No. 19607.

Parameter CAV	Function	Operation
0	Interference check alarm function	An alarm stop occurs before the execution of the block in which overcutting (interference) occurs.
1	Interference check avoidance function	The tool path is changed so that overcutting (interference) does not occur, and processing continues.

6.6.6.2 Interference check alarm function

- Interference other than those between adjacent three blocks

If the end-point vector of block 1 and the end-point vector of block 7 are judged to interfere as shown in the Fig. 6.6.6.2 (a), an alarm will occur before the execution of block 1 so that the tool stops. In this case, the vectors will not be erased.

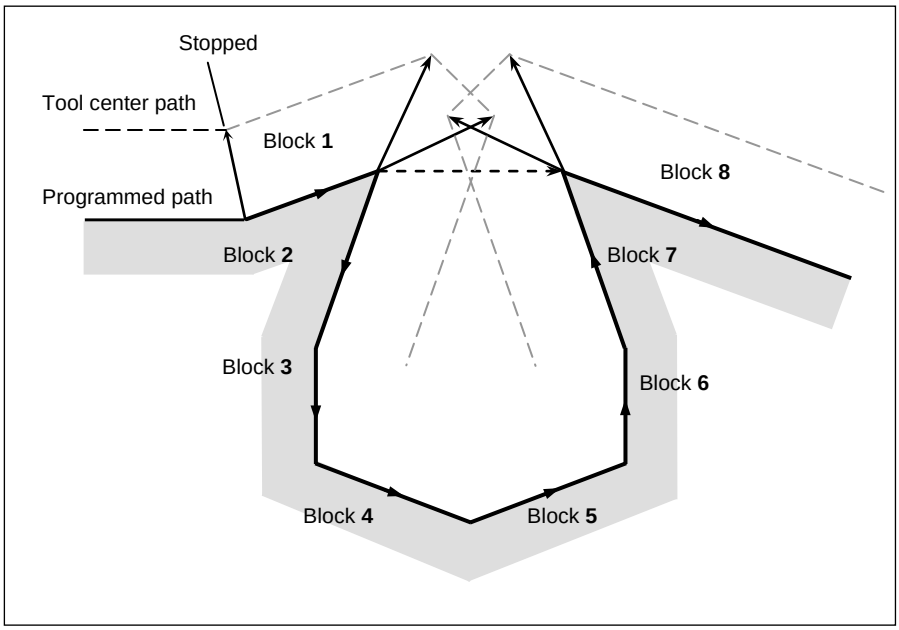


Fig. 6.6.6.2 (a)

- Interference between adjacent three blocks

If an interference is judged to occur between adjacent three blocks, the interfering vector, as well as any vectors existing inside of it, is erased, and a path is created to connect the remaining vectors. In the Example shown in the Fig. 6.6.6.2 (b), V_2 and V_5 interfere, so that V_2 and V_5 are erased, so are V_3 and V_4, which are inside of them, and V_1 is connected to V_6. The operation during this time is linear interpolation.

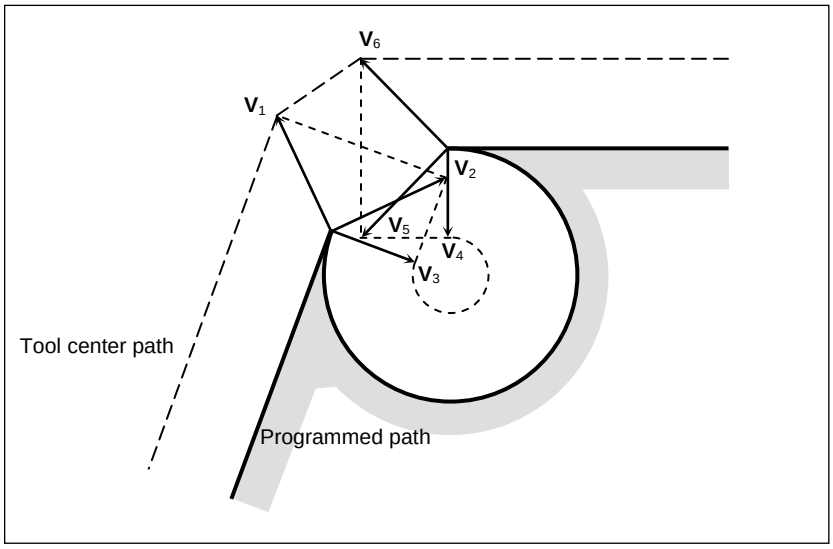


Fig. 6.6.6.2 (b)

If, after vector erasure, the last single vector still interferes, or if there is only one vector at the beginning and it interferes, an alarm will occur immediately after the start of the previous block (end point for a single block) and the tool stops. In the Example shown in the Fig. 6.6.6.2 (c), V_2 and V_3 interfere, but, even after erasure, an alarm will occur because the final vectors V_1 and V_4 interfere.

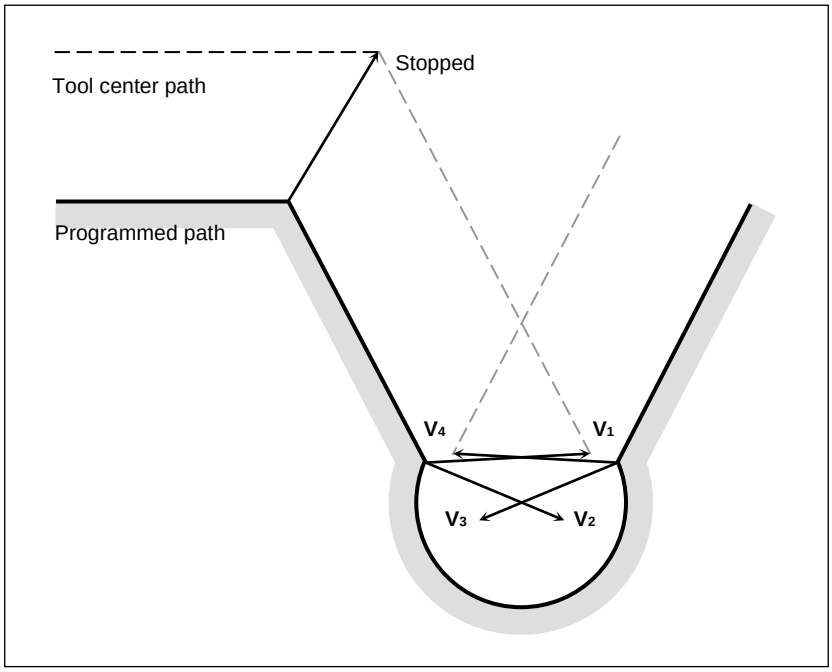


Fig. 6.6.6.2 (c)

6.6.6.3 Interference check avoidance function

Overview

If a command is specified which satisfies the condition under which the interference check alarm function generates an interference alarm, this function suppresses the generation of the interference alarm, but causes a new compensation vector to be calculated as a path for avoiding interference, thereby continuing machining. For the path for avoiding interference, insufficient cutting occurs in comparison with the programmed path. In addition, depending on the specified figure, no path for avoiding interference can be determined or the path for avoiding interference may be judged dangerous. In such a case, an alarm stop will occur. For this reason, it is not always possible to avoid interference for all commands.

- Interference avoidance method

Let us consider a case in which an interference occurs between the compensation vector between (block 1 - block 2) and the compensation vector between (block N-1 - block N). The direction vector from the end point of block 1 to the end point of block N-1 is called a gap vector. At this time, a post-compensation intersection vector between (block 1 - gap vector) and a post-compensation intersection vector between (gap vector - block N) is determined, and a path connecting them is created.

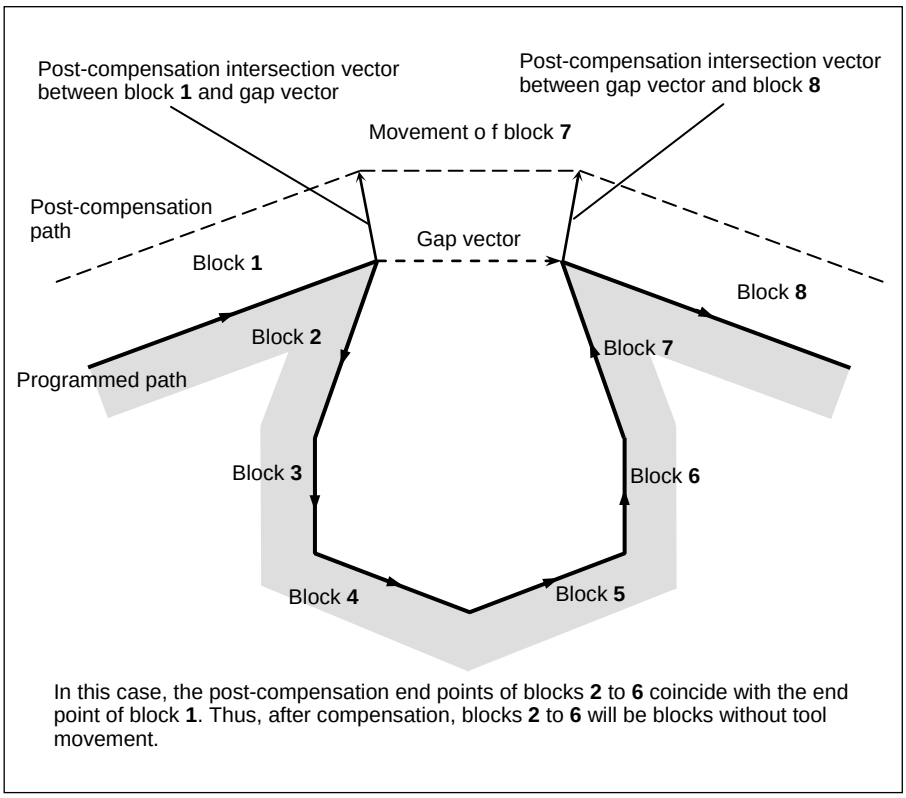


Fig. 6.6.6.3 (a)

If the post-compensation intersection vector of (block 1 - gap vector) and the post-compensation intersection vector of (gap vector - block N) further intersect, vector erasure is first performed in the same way as in "Interference between adjacent three blocks". If the last vectors that remains still intersects, the post-compensation intersection vector of (block 1 - block N) is re-calculated.

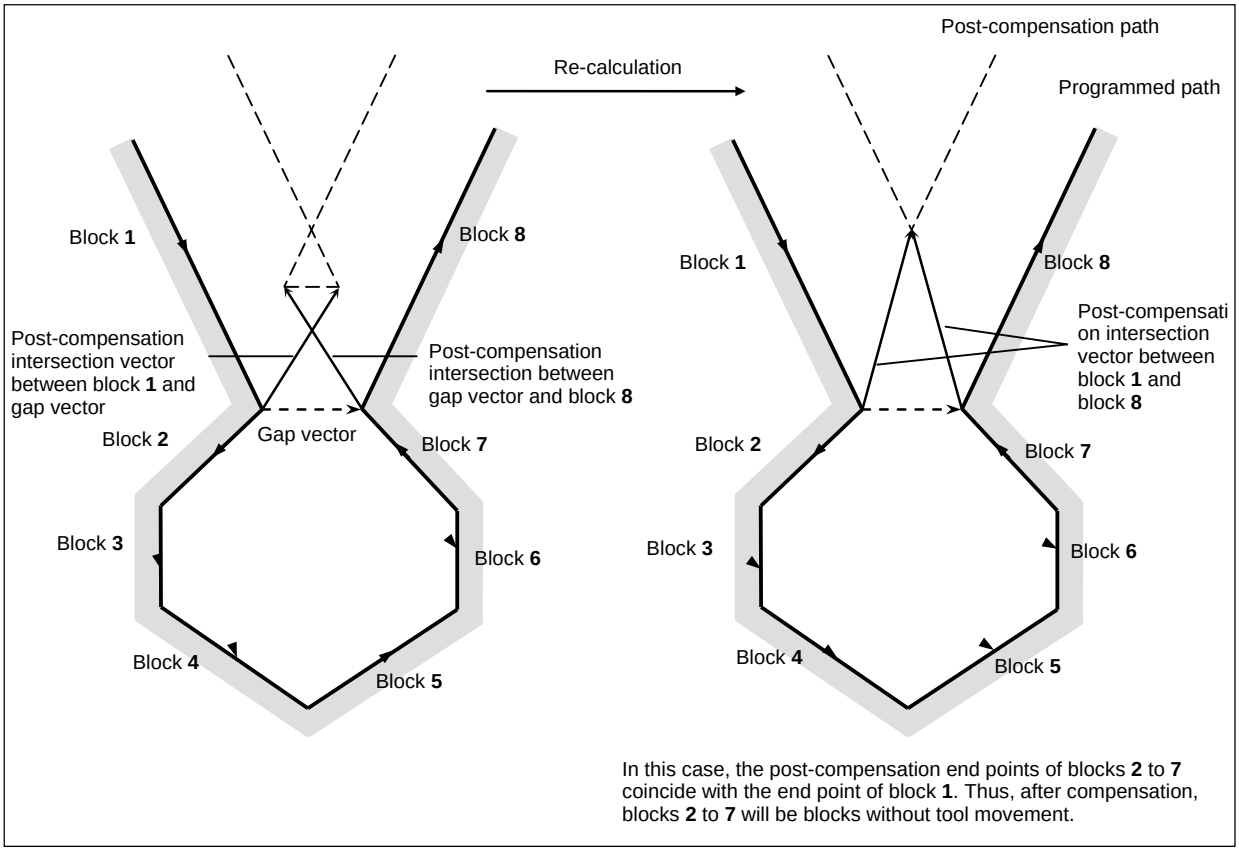


Fig. 6.6.6.3 (b)

If the tool radius/tool nose radius compensation value is greater than the radius of the specified arc as shown in the Fig. 6.6.6.3 (c), and a command is specified which results in compensation with respect to the inside of the arc, interference is avoided by performing intersection calculation with an arc command being assumed a linear one. In this case, avoided vectors are connected with linear interpolation.

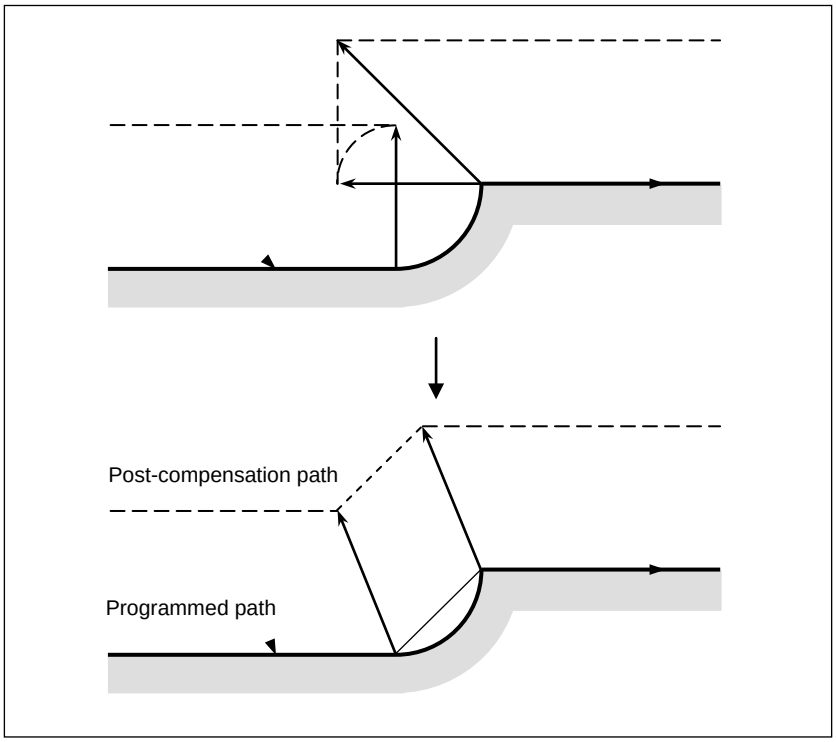


Fig. 6.6.6.3 (c)

- If no interference avoidance vector exists

If the parallel pocket shown in the Fig. 6.6.6.3 (d) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are parallel to each other, no intersection exists. In this case, an alarm PS5446, “NO AVOIDANCE AT G41/G42” will occur immediately before block 1 and the tool will stop.

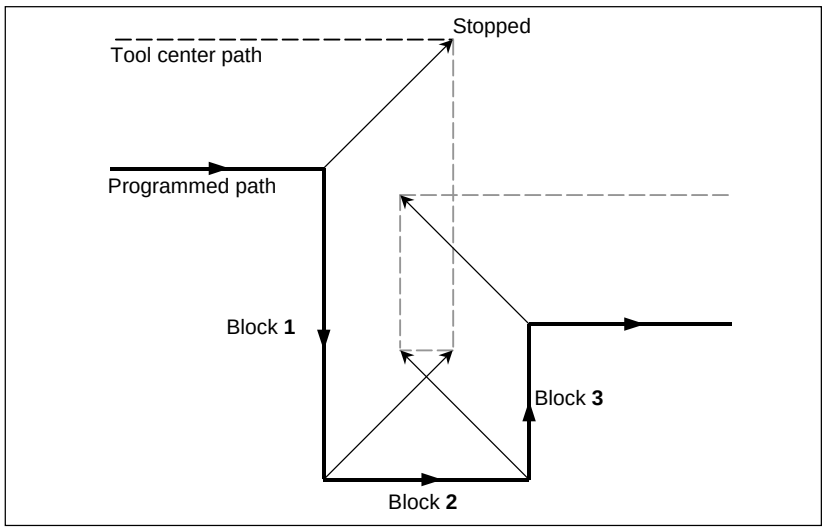


Fig. 6.6.6.3 (d)

If the circular pocket shown in the Fig. 6.6.6.3 (e) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are circular, no post-compensation intersection exists. In this case, an alarm PS5446 will occur immediately before block 1 and the tool will stop, as in the previous example.

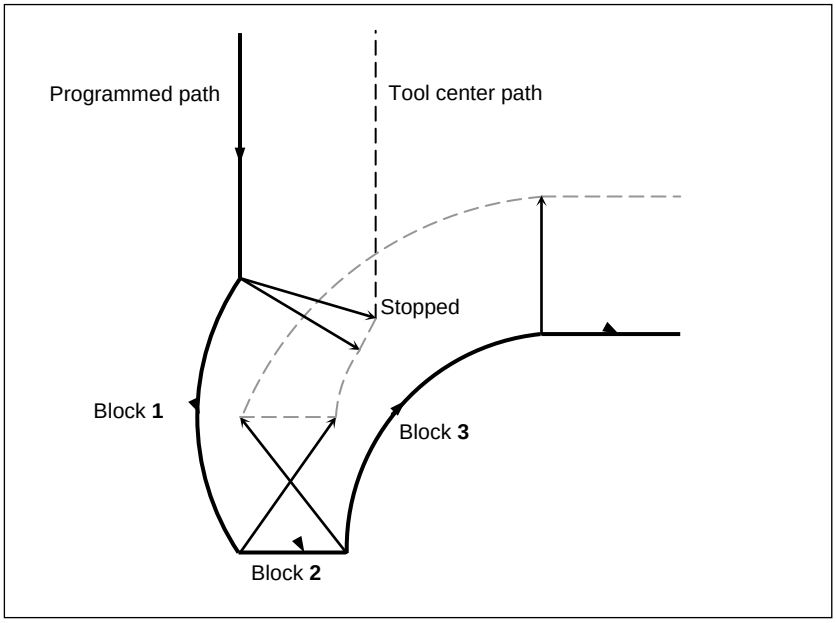


Fig. 6.6.6.3 (e)

- If it is judged dangerous to avoid interference

If the acute-angle pocket shown in the Fig. 6.6.6.3 (f) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the movement direction of the post-avoidance path extremely differs from the previously specified direction. If the post-avoidance path extremely differs from that of the original command (90° or greater or 270° or less), interference avoidance operation is judged dangerous; an alarm PS5447, “DANGEROUS AVOIDANCE AT G41/G42” will occur immediately before block 1 and the tool will stop.

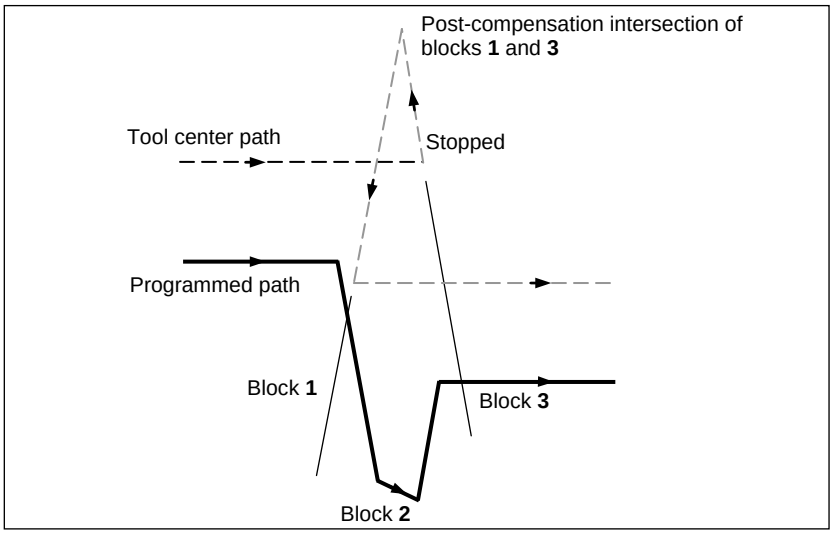


Fig. 6.6.6.3 (f)

If a pocket in which the bottom is wider than the top, such as that shown in the Fig. 6.6.6.3 (g), is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the relation between blocks 1 and 3 is judged an outer one, the post-avoidance path results in overcutting as compared with the original command. In such a case, interference avoidance operation is judge dangerous; an alarm PS5447 will occur immediately before block 1 and the tool will stop.

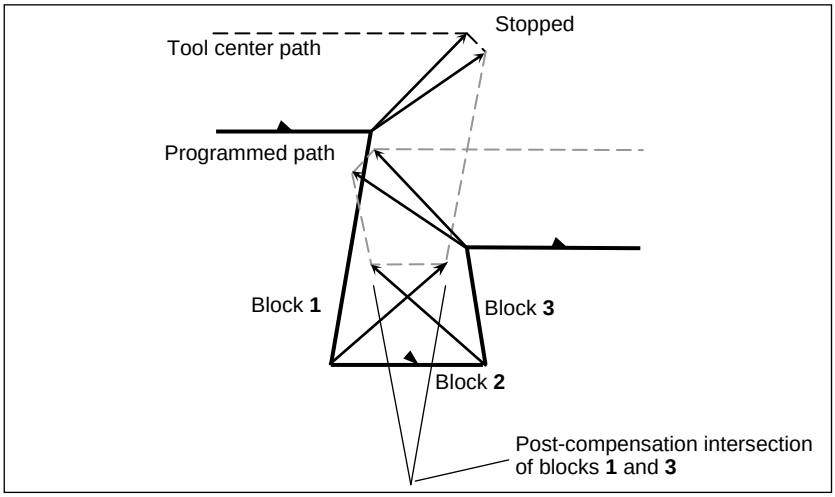


Fig. 6.6.6.3 (g)

- If further interference with an interference avoidance vector occurs

If the pocket shown in the Fig. 6.6.6.3 (h) is to be machined, if the number of blocks to read is 3, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, however, the end-point vector of block 3 that is to be calculated next further interferes with the previous interference avoidance vector.
If a further interference occurs to the interference avoidance vector once created and output, the movement in the block will not be performed; an alarm PS5448, “INTERFERENCE TO AVD. AT G41/G42” will occur immediately before the block and the tool will stop.

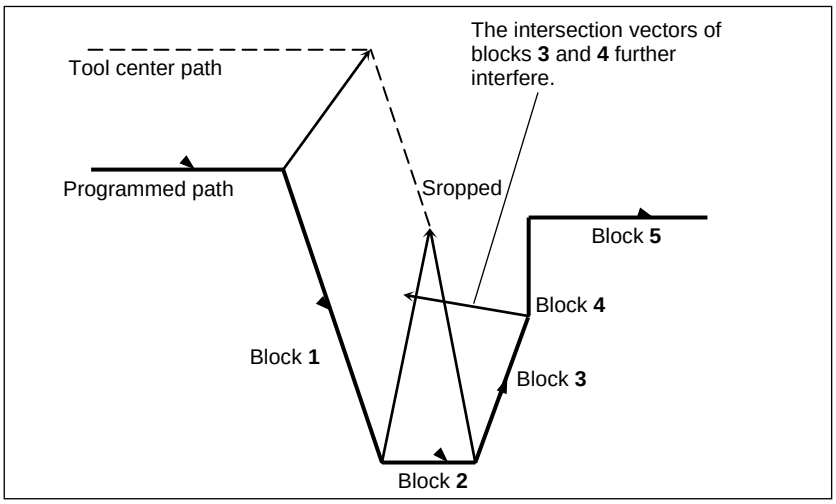


Fig. 6.6.6.3 (h)

NOTE

1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting bit 6 (NAA) of parameter No. 19607 appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", however, it is not possible to avoid an alarm regardless of the setting of this parameter.

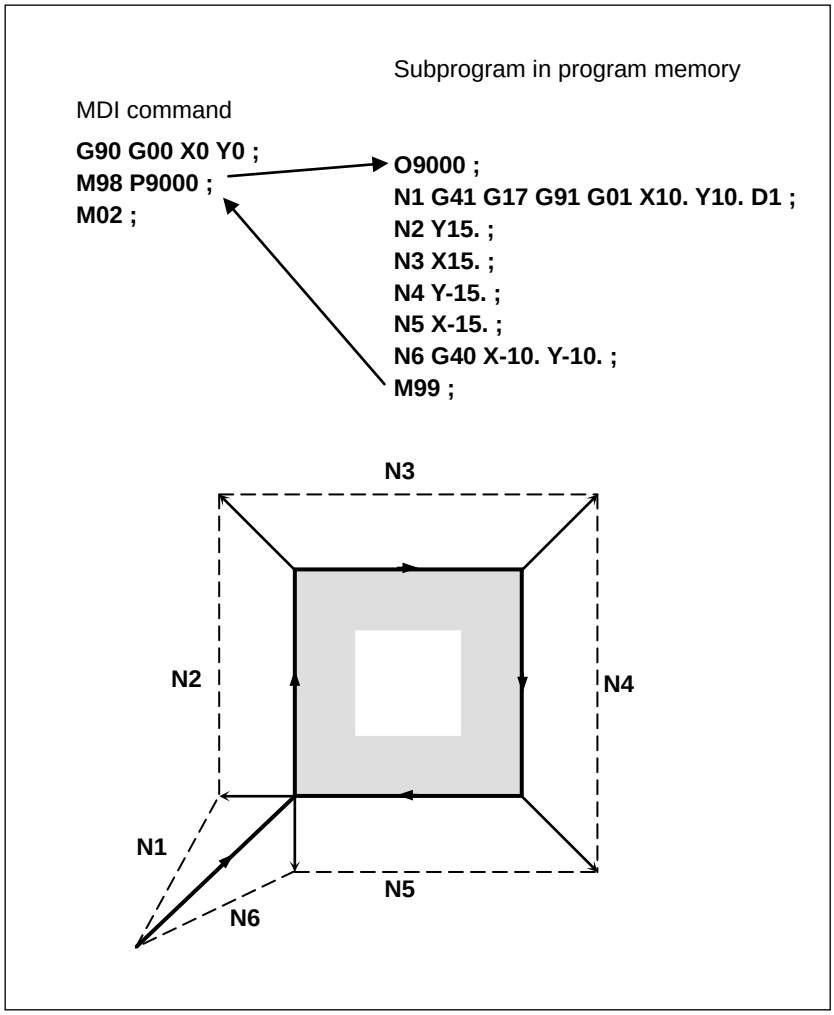
2 If a single block stop occurs during interference avoidance operation, and an operation is performed which differs from the original movement, such as manual intervention, MDI intervention, tool radius / tool nose radius compensation value change, intersection calculation is performed with a new path. If such an operation is performed, therefore, an interference may occur again although interference avoidance has been performed once.

6.6.7 Tool Radius / Tool Nose Radius Compensation for Input from MDI

Explanation

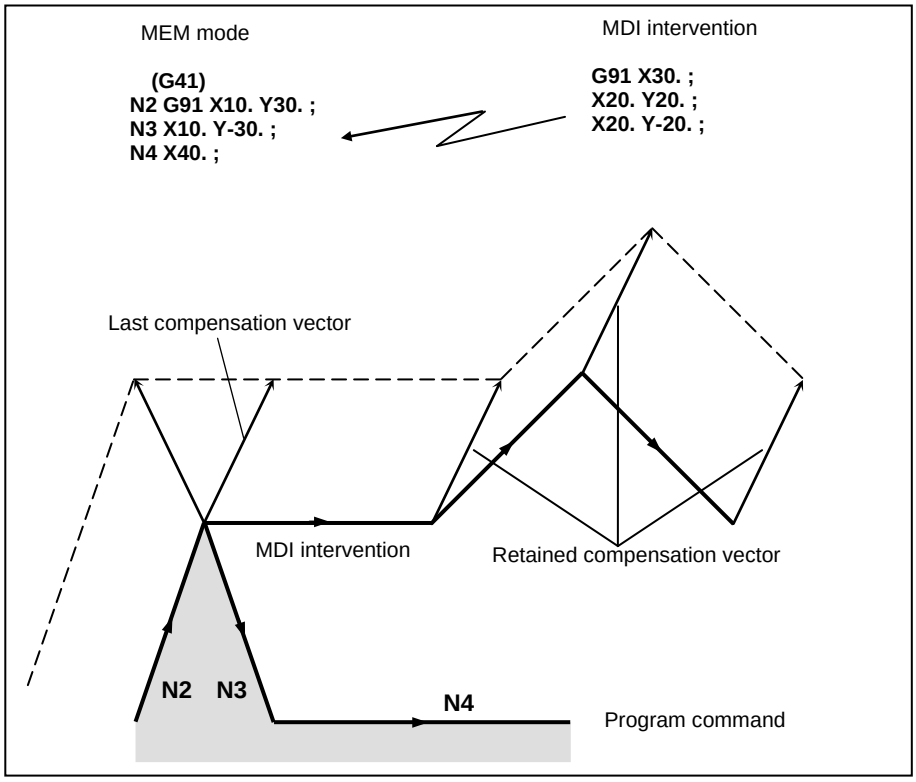
- MDI operation

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, cutter compensation does not perform intersection calculation, retaining the last compensation vector before the intervention.



NOTE

If tool radius / tool nose radius compensation is specified in MDI mode or MDI intervention, be sure to set bit 4(MCR) of parameter No.5008 to 0.
When bit 4 (MCR) of parameter No.5008 is set to 1, an alarm PS5257, "G41/G42 NOT ALLOWED IN MDI MODE" occurs.

6.7 VECTOR RETENTION (G38)

In tool radius / tool nose radius compensation, by specifying G38 in offset mode, it is possible to retain the compensation vector at the end point of the previous block, without performing intersection calculation.

Format

(In offset mode)
G38 IP_ ;

IP: Value specified for axial movement

Explanation

- Vector retention

By specifying the above command, a vector is created at the end point of the block immediately preceding the G38 block, vertical to that block. In the G38 block, the vertical vector output in the previous block is retained. G38 is a one-shot G code. With the next move command without a G38 command, the compensation vector is re-created.

Limitation

- Mode

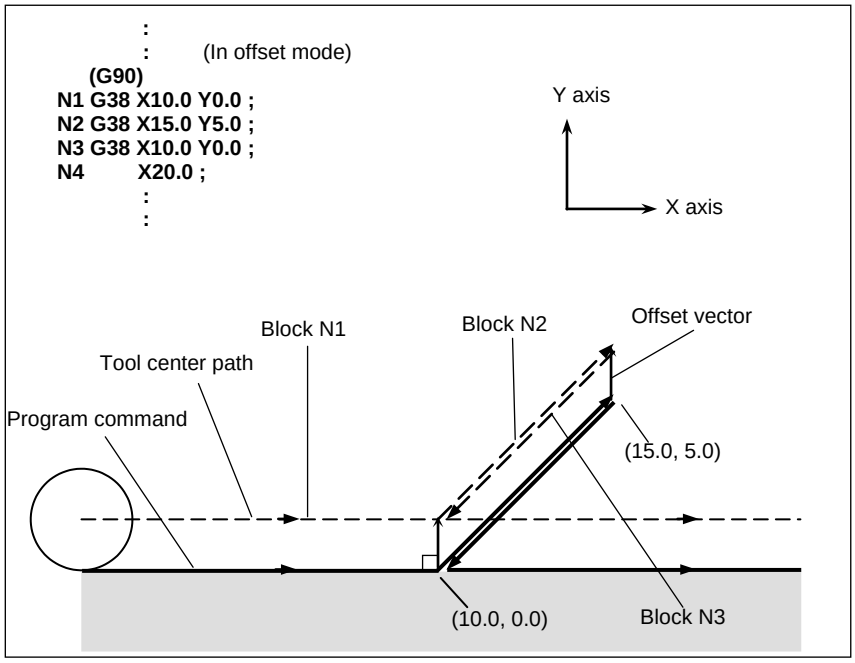
Specify G38 in either G00 or G01 mode. If it is specified in G02 or G03 (circular interpolation) mode, a radial error may occur at the start and end points.

- Start-up/cancel

In start-up/cancel, the operation is as described in Subsections, “Tool Movement in Start-up” and “Tool Movement in Offset Mode Cancel”. Thus, G38 cannot be specified in the following blocks:

1) Start-up command (G41 or G42) block
2) Cancel command (G40) block
3) Block immediately preceding the cancel command (G40) block

Example



6.8 CORNER CIRCULAR INTERPOLATION (G39)

By specifying G39 in offset mode during tool radius / 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 ;

$$\mathbf{G39}\left\{\begin{matrix}\mathbf{I_\ J_}\\ \mathbf{I_\ K_}\\ \mathbf{J_\ K_}\end{matrix}\right\}\ \mathbf{;}$$

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.
If it is specified, alarm PS5445 ”CAN NOT COMMAND MOTION IN G39” is issued..

- Inner corner

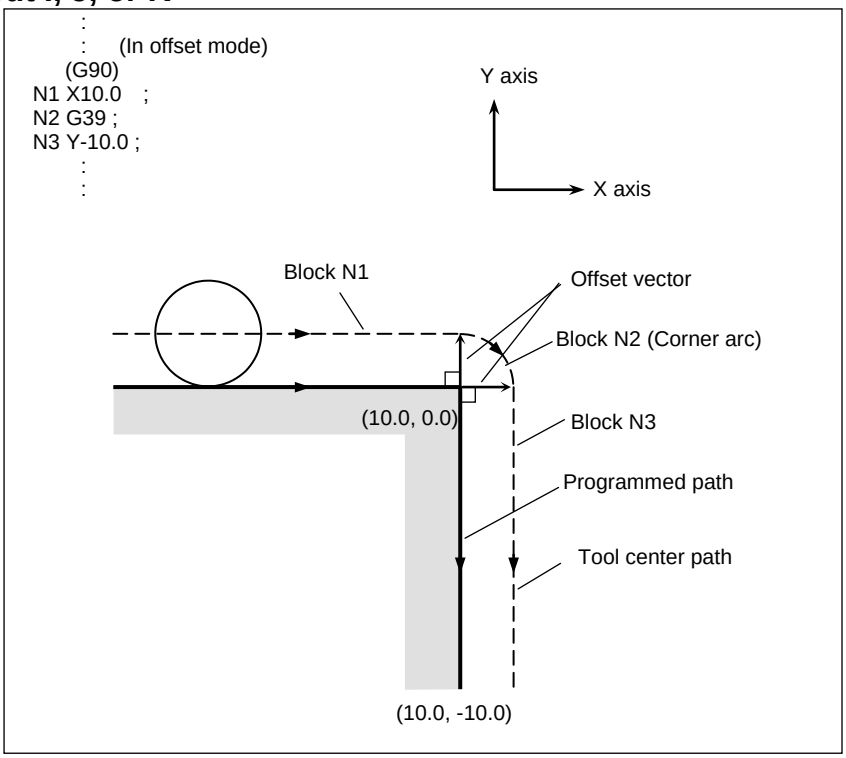
In an inner corner block, G39 cannot be specified. Otherwise, overcutting will occur.

- Corner arc velocity

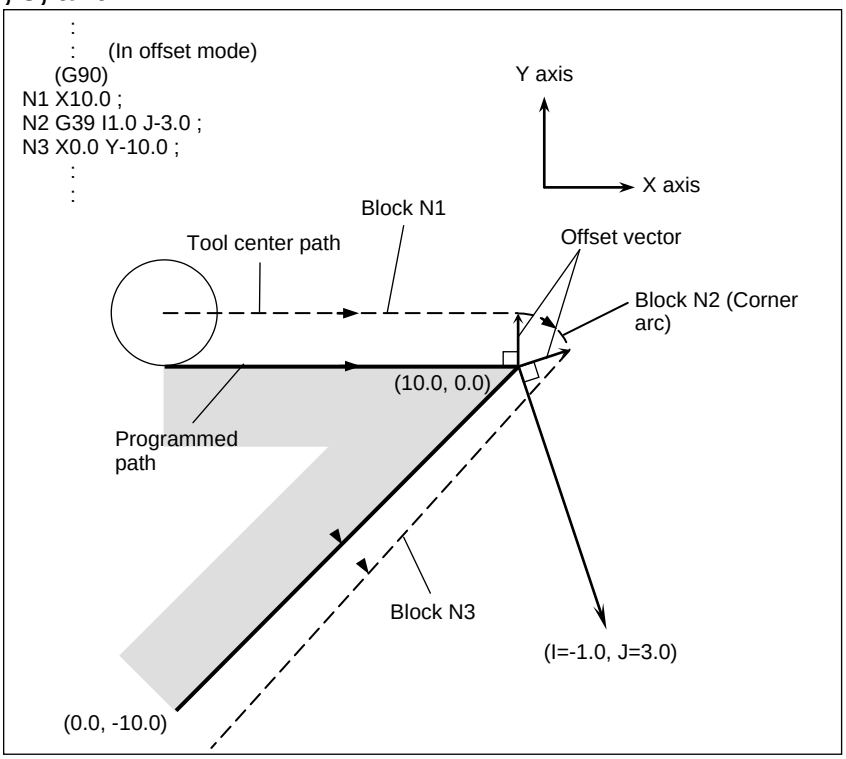
If a corner arc is specified with G39 in G00 mode, the corner arc block velocity will be that of the F command previously specified. If G39 is specified in a state in which no F command has never been specified, the velocity of the corner arc block will be that specified with parameter No. 1411.

Example

- **G39 without I, J, or K**



- **G39 with I, J, and K**



6.9 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Tool compensation values include tool geometry compensation values and tool wear compensation (Fig. 6.9 (a)).

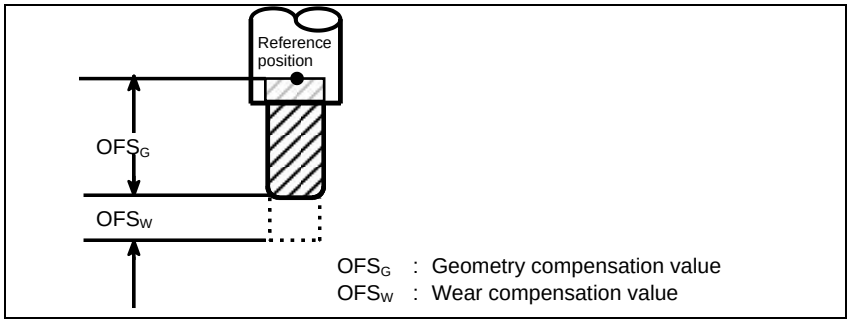


Fig. 6.9 (a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the MDI unit (see section III-1.1.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, cutter compensation, or the tool offset.
Two types of tool compensation memories are available according to the compensation value configuration: tool compensation memory A and C. One of the types can be selected.

Explanation

- Tool compensation memory A

In tool compensation memory A, memory for geometry compensation and memory for wear compensation are not distinguished from each other. So, the sum of geometry compensation and wear compensation values is to be set in the compensation memory. Moreover, no distinction is made between memory for cutter compensation (for D code) and memory for tool length compensation (for H code).

Setting example

Compensation number	Compensation value (geometry+wear)	Common to D code/H code
001	10.000	For D code
002	20.000	For D code
003	100.000	For H code
:	:	:

- Tool compensation memory C

In tool compensation memory C, memory for geometry compensation and memory for wear compensation are prepared separately. So, geometry compensation values and wear compensation values can be set separately. Moreover, memory for cutter compensation (for D code) and memory for tool length compensation (for H code) are prepared separately.

Setting example

Compensation number	D code		H code	
	For geometry compensation	For wear compensation	For geometry compensation	For wear compensation
001	10.000	0.100	100.000	0.100
002	20.000	0.200	200.000	0.300
:	:	:	:	:

- Unit and valid range of tool compensation values

A unit and valid range of tool offset values can be selected from the following by parameter setting:

Unit and valid range of tool compensation values (metric input)

OFE	OFD	OFC	OFA	Unit	Valid range
0	0	0	1	0.01mm	±9999.99mm
0	0	0	0	0.001mm	±9999.999mm
0	0	1	0	0.0001mm	±9999.9999mm
0	1	0	0	0.00001mm	±9999.99999mm
1	0	0	0	0.000001mm	±999.999999mm

Unit and valid range of tool compensation values (inch input)

OFE	OFD	OFC	OFA	Unit	Valid range
0	0	0	1	0.001inch	±999.999inch
0	0	0	0	0.0001inch	±999.9999inch
0	0	1	0	0.00001inch	±999.99999inch
0	1	0	0	0.000001inch	±999.999999inch
1	0	0	0	0.0000001inch	±99.9999999inch

- Number of tool compensation data items

The number of tool compensation data items used by the entire system varies from one machine to another. Refer to the relevant manual of the machine tool builder.

Format

The format for programming depends on the type of tool compensation memory.

For tool compensation memory A

```
G10 L11 P_ R_ Q_ ;
    P_      :  Tool compensation number
    R_      :  Tool compensation value
    Q_      :  Imaginary tool nose number
```

For tool compensation memory C

```
G10 L_ P_ R_ Q_ ;
    L_  : Type of compensation memory
          L10 :  Geometry compensation value corresponding to an H code
          L11 :  Wear compensation value corresponding to an H code
          L12 :  Geometry compensation value corresponding to a D code
          L13 :  Wear compensation corresponding to a D code
          L110 : Geometry compensation value corresponding to a D code (for corner R
                 offset)
          L111 : Wear compensation corresponding to a D code (for corner R offset)
    P_  : Tool compensation number
    R_  : Tool compensation value
    Q_  : Imaginary tool nose number
```

By specifying G10, a tool compensation value can be set or modified.
When G10 is specified by absolute input (G90), the specified value is used as the new tool compensation value.
When incremental input (G91) is used, a specified value added to the tool compensation value currently set is used as the new tool compensation value.

NOTE
1 Address R follows the increment system for tool offset values.
2 If L is omitted for compatibility with the conventional CNC format, or L1 is specified, the same operation as when L11 is specified is performed.
3 Set a imaginary tool nose number when the cutter compensation function is specified and a imaginary tool nose direction is used.

6.10 COORDINATE SYSTEM ROTATION (G68, G69)

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation.

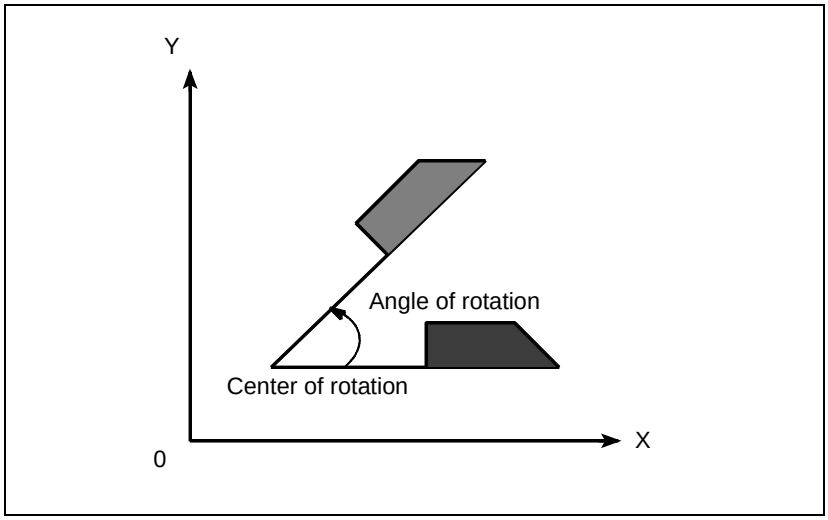


Fig. 6.10 (a) Coordinate system rotation

Format

```
{G17}
{G18}G68 α_β_ R_ ;      Start rotation of a coordinate system.
{G19}
  :                     Coordinate system rotation mode
  :                     (The coordinate system is rotated.)
G69 ;                   Coordinate system rotation cancel command
```

G17 (G18 or G19) : Select the plane in which contains the figure to be rotated.

α_β_ : Absolute programming for two of the X_, Y_, and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68

R_ : Specifies the rotation angle which of the counterclockwise is positive.
Bit 0 (RIN) of parameter No. 5400 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).
Least input increment : 0.001 deg
Valid data range : -360000 to 360000

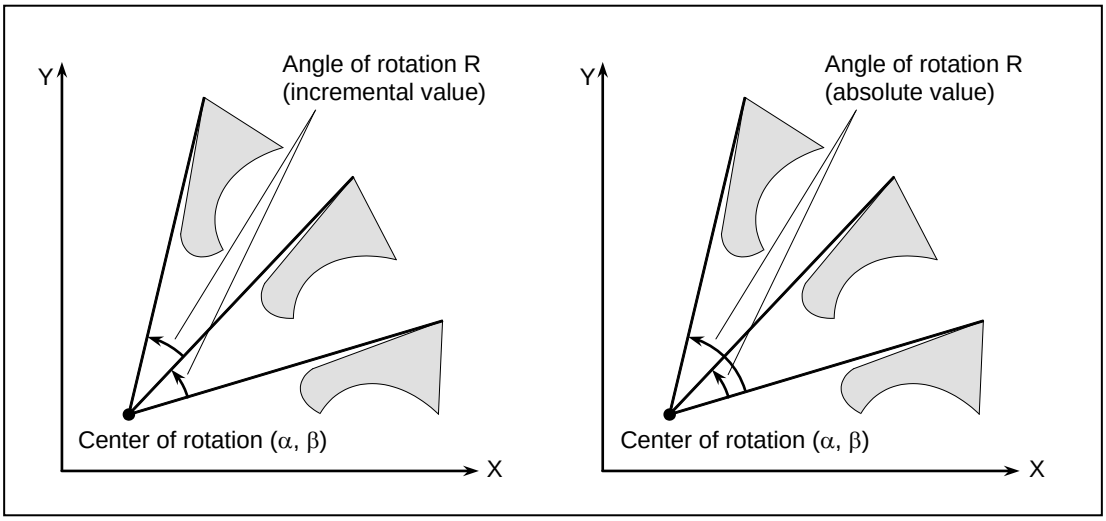


Fig. 6.10 (b) Center and angle of rotation

NOTE
When a decimal fraction is used to specify angular displacement (R_), the 1's digit corresponds to degree units.

Explanation

- G code for selecting a plane: G17,G18 or G19

The G code for selecting a plane (G17,G18,or G19) can be specified before the block containing the G code for coordinate system rotation (G68). G17, G18 or G19 must not be designated in the mode of coordinate system rotation.

- Incremental programming in coordinate system rotation mode

The center of rotation for an incremental programming programmed after G68 but before an absolute programming is the tool position when G68 was programmed (Fig. 6.10 (e)).

- Center of rotation

When α_β_ is not programmed, the tool position when G68 was programmed is assumed as the center of rotation.

- Angular displacement

When R_ is not specified, the value specified in parameter No. 5410 is assumed as the angular displacement.
To specify angular displacement (R_) in 0.00001 degrees (one hundred-thousandth), set bit 0 (FRD) of parameter No. 11630 to 1. In this case, angular displacement R is specified within the range of -36000000 to 36000000.

- Coordinate system rotation cancel command

The G code used to cancel coordinate system rotation (G69) may be specified in a block in which another command is specified.

- Tool compensation

Tool radius/tool nose radius compensation, tool length compensation, tool offset, and other compensation operations are executed after the coordinate system is rotated.

- Relationship with 3-dimensional coordinate conversion (G68, G69)

Both coordinate system rotation and 3-dimensional coordinate conversion use the same G codes: G68 and G69. The G code with I, J, and K is processed as a command for 3-dimensional coordinate conversion. The G code without I, J, and K is processed as a command for two-dimensional coordinate system rotation.

Limitation

- Commands related to reference position return and the coordinate system

In coordinate system rotation mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling coordinate system rotation mode. If any of them is specified during coordinate system rotation mode, alarm PS0412, “AN UNUSABLE G CODE WAS USED”, will occur.

- Incremental programming

The first move command after the coordinate system rotation cancel command (G69) must be specified with absolute values. If an incremental move command is specified, correct movement will not be performed.

- Specifying during AI contour control mode or 3-dimensional coordinate system conversion mode

Coordinate system rotation can always be specified for the M series.

- Note on the specification of one axis in coordinate system rotation

With the parameter below, a move position in the case where one axis is specified in the absolute mode can selected. If two axes are specified, a movement is made to the same position, regardless of the setting of the parameter.

Bit 5 (AX1) of parameter No. 11600

If one axis is specified in the absolute mode when the coordinate system rotation mode is set:

0: The specified position is first calculated in the coordinate system before rotation then the coordinate system is rotated.

1: The coordinate system is first rotated then a movement is made to the specified position in the rotated coordinate system. (FS16*i*/18*i*/21*i*-compatible specification)

This parameter changes the handling of coordinates on axes not specified, so that a position to be reached by movement changes.

(Example)

```
G90 G0 X0 Y0
G01 X10. Y10. F6000
G68 X0 Y0 R45.... ..... Specifies coordinate system rotation.
Y14.142..................... Specifies one axis ....(1)
G69
```

When bit 5 (AX1) of parameter No. 11600= 0:

The specified position is calculated in the coordinate system (XY) before rotation then the coordinate system is rotated. So, with the specification of (1), the position on the unspecified X axis is X10, and the specified position is (X10,Y14.142). Next, a movement is made to the move position (X-2.929,Y17.071) obtained by 45° rotation.

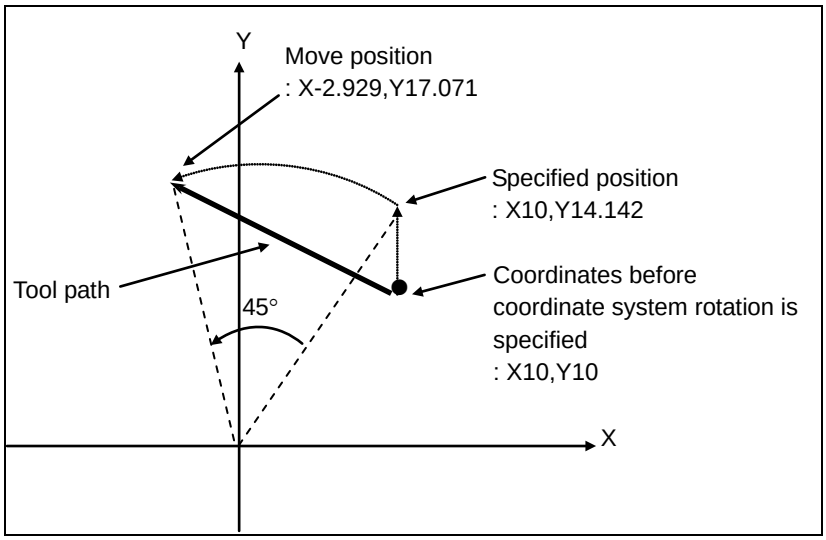


Fig. 6.10 (c) Specification of one axis (AX1=0)

When bit 5 (AX1) of parameter No. 11600= 1:

With the specification of (1), coordinates (X10,Y10) before coordinate system rotation are converted to coordinates (X'14.142,Y'0) in the coordinate system (X'Y') obtained by 45° rotation. Next, a movement is made to the specified position (X'14.142,Y'14.142), that is, the move position (X0,Y20).

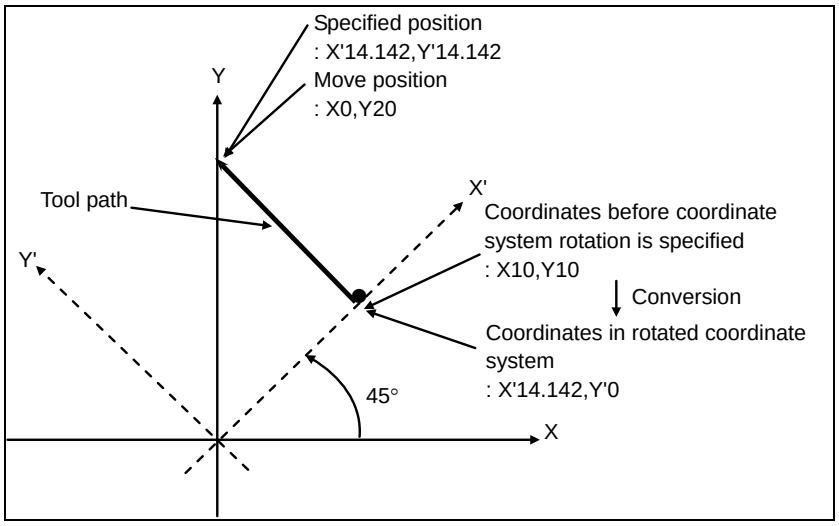


Fig. 6.10 (d) Specification of one axis (AX1=1)

Explanation

- Absolute/Incremental position commands

```
N1 G92 X-500.0 Y-500.0 G69 G17 ;
N2 G68 X700.0 Y300.0 R60.0 ;
N3 G90 G01 X0 Y0 F200 ;
    (G91X500.0Y500.0)
N4 G91 X1000.0 ;
N5 G02 Y1000.0 R1000.0 ;
N6 G03 X-1000.0 I-500.0 J-500.0 ;
N7 G01 Y-1000.0 ;
N8 G69 G90 X-500.0 Y-500.0 M02 ;
```

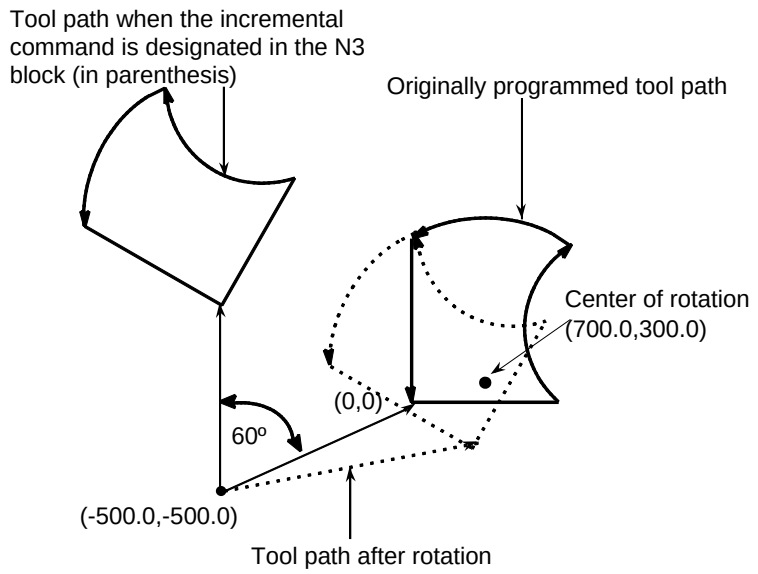


Fig. 6.10 (e) Absolute/incremental programming during coordinate system rotation

- Cutter compensation and coordinate system rotation

It is possible to specify G68 and G69 in cutter compensation mode.
The rotation plane must coincide with the plane of cutter compensation.

```
N1 G92 X0 Y0 G69 G01 ;
N2 G42 G90 X1000 Y1000 F1000 D01 ;
N3 G68 R-30000 ;
N4 G91 X2000 ;
N5 G03 Y1000 R1000 J500 ;
N6 G01 X-2000 ;
N7 Y-1000 ;
N8 G69 G40 G90 X0 Y0 M30 ;
```

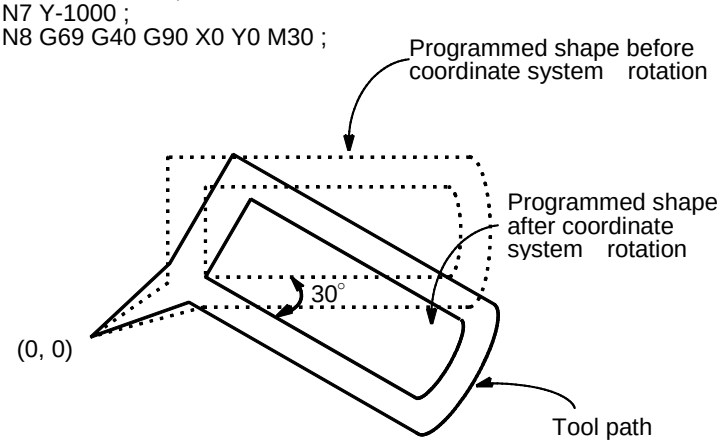


Fig. 6.10 (f) Cutter compensation and coordinate system rotation

- Scaling and coordinate system rotation

If a coordinate system rotation command is executed in the scaling mode (G51 mode), the coordinate value (a, b) of the rotation center will also be scaled, but not the rotation angle (R). When a move command is issued, the scaling is applied first and then the coordinates are rotated.
A coordinate system rotation command (G68) should not be issued in cutter compensation mode (G41, G42) on scaling mode (G51). The coordinate system rotation command should always be specified prior to setting the cutter compensation mode.

1. When the system is not in cutter compensation mode, specify the commands in the following order :
   G51 ; Scaling mode start
   G68 ; Coordinate system rotation mode start
   :
   G69 ; Coordinate system rotation mode cancel
   G50 ; Scaling mode cancel

2. When the system is in cutter compensation, specify the commands in the following order (Fig. 6.10 (g)) :
   (cutter compensation cancel)
   G51 ; Scaling mode start
   G68 ; Coordinate system rotation start
   :
   G41 ; Cutter compensation mode start
   :

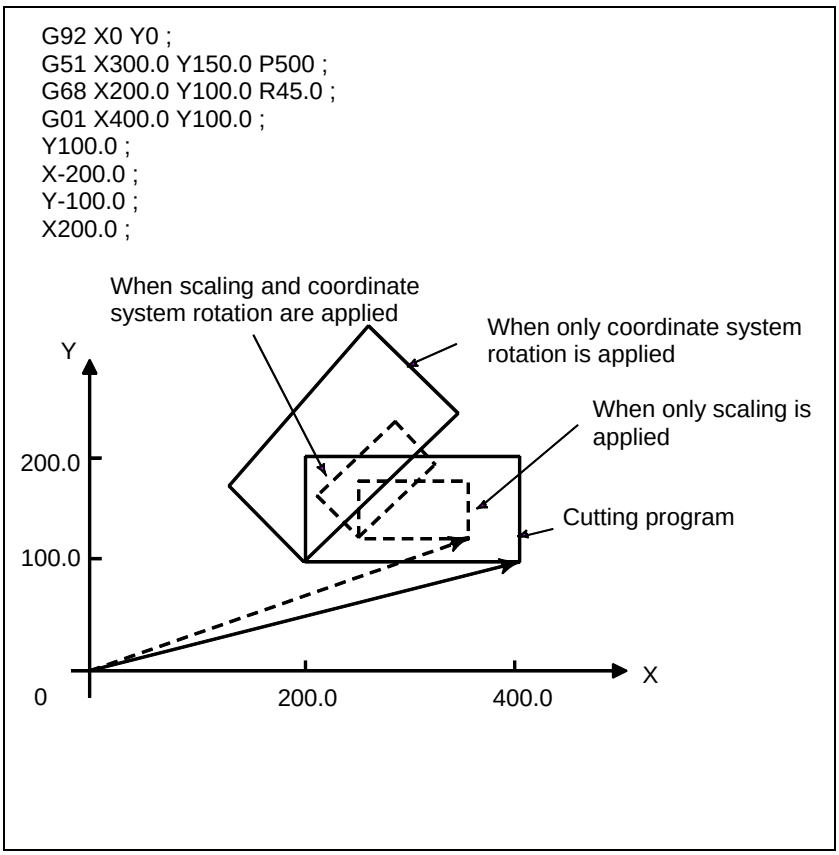


Fig. 6.10 (g) Scaling and coordinate system rotation in cutter compensation mode

- Repetitive commands for coordinate system rotation

It is possible to store one program as a subprogram and recall subprogram by changing the angle.

Sample program for when parameter RIN (No. 5400#0) is set to 1.
The specified angular displacement is treated as an absolute or incremental value depending on the specified G code (G90 or G91).

```
    G92 X0 Y0 G69 G17;
    G01 F200 H01 ;
    M98 P2100 ;
    M98 P072200 ;
    G00 G90 X0 Y0 M30 ;

O 2200 G68 X0 Y0 G91 R45.0 ;
    G90 M98 P2100 ;
    M99 ;

O 2100 G90 G01 G42 X0 Y-10.0 ;
    X4.142 ;
    X7.071 Y-7.071 ;
    G40 ;
    M99 ;
```

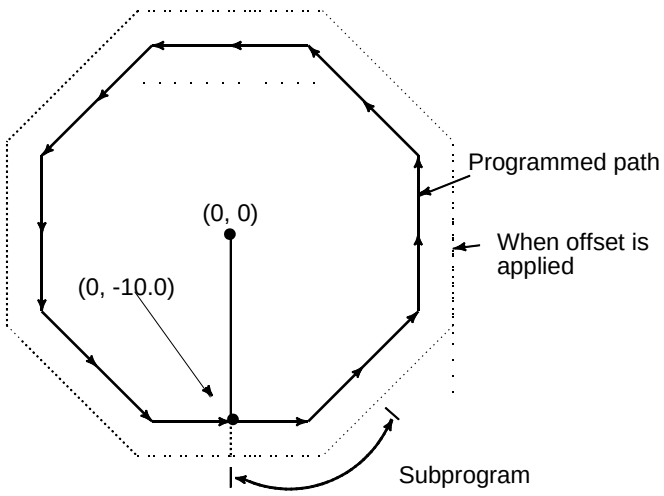


Fig. 6.10 (h) Coordinate system rotation command

7 MEMORY OPERATION USING Series 15 PROGRAM FORMAT

Overview

Memory operation of the program registered in Series 15 program format is possible by setting the setting bit 1 (FCV) of parameter No. 0001 to 1.

Explanation

Data formats for tool radius compensation, subprogram call, and canned cycles are different between the Series 0*i*-F Plus and Series 15. The Series 15 program formats can be processed for memory operation. Other data formats must comply with the Series 0*i*-F Plus. When a value out of the specified range for the Series 0*i*-F Plus is registered, an alarm occurs.

NOTE

1 Registration to memory and memory operation are possible only for the functions available in Series 0*i*-F Plus.
2 Do not change the setting of this parameter (bit 1 (FCV) of parameter No. 0001) during memory operation. Change the setting of this parameter in the reset state.
3 For rigid tapping, refer to "5.3 RIGID TAPPING".
4 Refer to a usual format for the limitations and notes of the Series15 program format.

- Address for the tool radius compensation offset number

Offset numbers are specified by address D in the Series 15.
When an offset number is specified by address D, the modal value specified by address H is replaced with the offset number specified by address D.

- Subprogram call

If a subprogram number of more than four digits is specified, the four low-order digits are regarded as the subprogram number.
If no repeat count is specified, 1 is assumed.

Table 7 (a) Subprogram call program format

CNC	Program format
Series 15	**M98 P**○○○○ **L**○○○○ **;** P : Subprogram number L : Repetition count (1 to 9999)
Series 0*i*-F Plus	**M98 P**○○○ □□□□ **;** Repetition count Subprogram number (1 to 9999)

If, however, the custom macro is enabled (bit 5 (NMC) of parameter No.8135 is 0), both formats can be used.

- Address for the canned cycle repetition count for drilling

The Series 15 and Series 0*i*-F Plus use different addresses for the canned cycle repetition count for drilling as listed in Table 7 (b).

Table 7 (b) Address for the canned cycle repetition count for drilling

CNC	Address
Series 15	L
Series 0*i*-F Plus	K

8 HIGH-SPEED CUTTING FUNCTIONS

8.1 SELECT SETTING PATTERN OF FINE SURFACE SETTING BY PROGRAM COMMAND

Overview

In parameters related to high-quality machining, operators can select an effective setting pattern from the three kinds of parameter settings (called setting patterns) machine tool builders' made patterns depending on the machining conditions. For these reasons, this function makes it easier to set parameters related to high-quality machining and to choose parameters according to the machining conditions.

In this chapter, explain about select setting pattern by program command. Please refer to “FINE SURFACE SETTING” for “PATTERN SELECT” screen (selection for setting patterns) in Fine surface setting.

Format

G08.1P_ ;

P_ : Setting pattern
P1 : “Pattern 1”
P2 : “Pattern 2”
P3 : “Pattern 3”

Ex.) When commanding G08.1P3, “Pattern 3” is selected as an effective setting pattern

G05.1 Q1 R_ ;

R_ : level (1 to 10)

G05 P10000 R_ ;

R_ : level (1 to 10)

G08 P1 R_ ;

R_ : level (1 to 10)

Explanation

Setting pattern selection by G08.1 command

By performing G08.1 command during automatic operation, an effective setting pattern is selected. The effective setting pattern is specified with address P.

Setting pattern selection by G05.1, G05, G08 command

A setting pattern can be selected by the G code of Machining condition selection function (G05.1, G05, G08 commands). This enables to use the machining programs without any changes, which has been used on the machines with Machining condition selection function available.

III. OPERATION

1 SETTING AND DISPLAYING DATA

1.1 SCREENS DISPLAYED BY FUNCTION KEY

Press function key to display or set tool compensation values and other data.

This section describes how to display or set the following data:

1. Tool compensation value
2. Tool length measurement
3. Tool offset value

1.1.1 Setting and Displaying the Tool Compensation Value

Tool offset values, tool length compensation values, and cutter compensation values are specified by D codes or H codes in a program. Compensation values corresponding to D codes or H codes are displayed or set on the screen.

Procedure for setting and displaying the tool compensation value (for 8.4/10.4-inch display unit)

Procedure

1 Press function key .

For the two-path control, select the path for which tool compensation values are to be displayed with the tool post selection switch.

2 Press chapter selection soft key [OFFSET] or press function key several times until the tool compensation screen is displayed.

The screen varies according to the type of tool compensation memory.

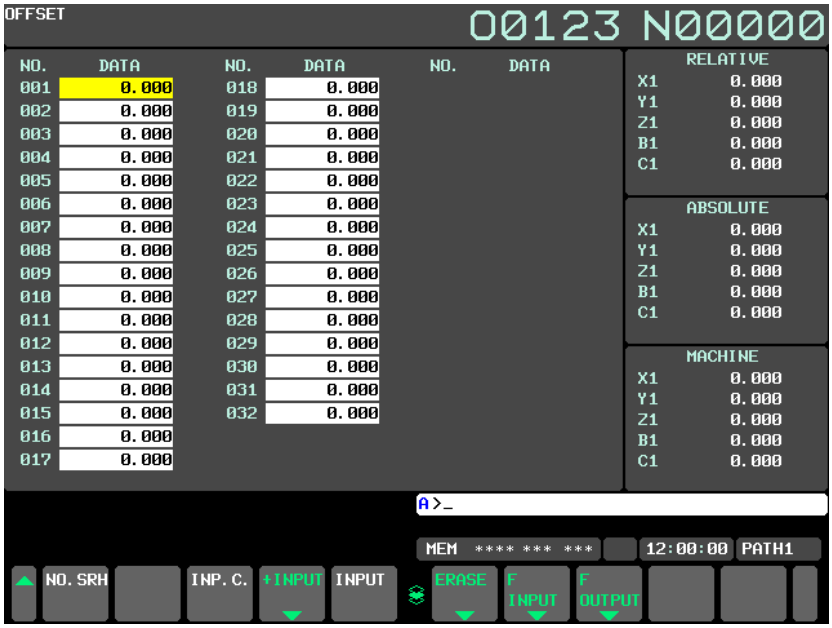


Fig. 1.1.1 (a) Tool compensation memory A (10.4-inch display unit)

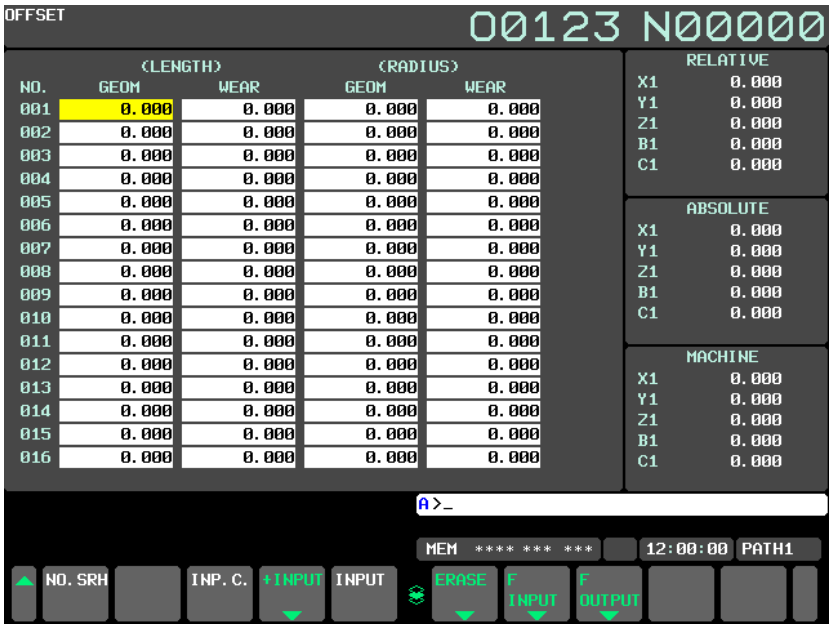


Fig. 1.1.1 (b) Tool compensation memory C (10.4-inch display unit)

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].

4 To set a compensation value, enter a value and press soft key [INPUT].
To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [+INPUT]. Or, enter a new value and press soft key [INPUT].

Procedure for setting and displaying the tool compensation value (for 15/19-inch display unit)

Procedure

1 Press function key .

For the two-path control, select the path for which tool compensation values are to be displayed with the tool post selection switch.

2 Press vertical soft key [OFFSET] for the chapter selection key.

Or, press function key several times until the tool compensation screen is displayed.

The screen varies according to the type of tool compensation memory.

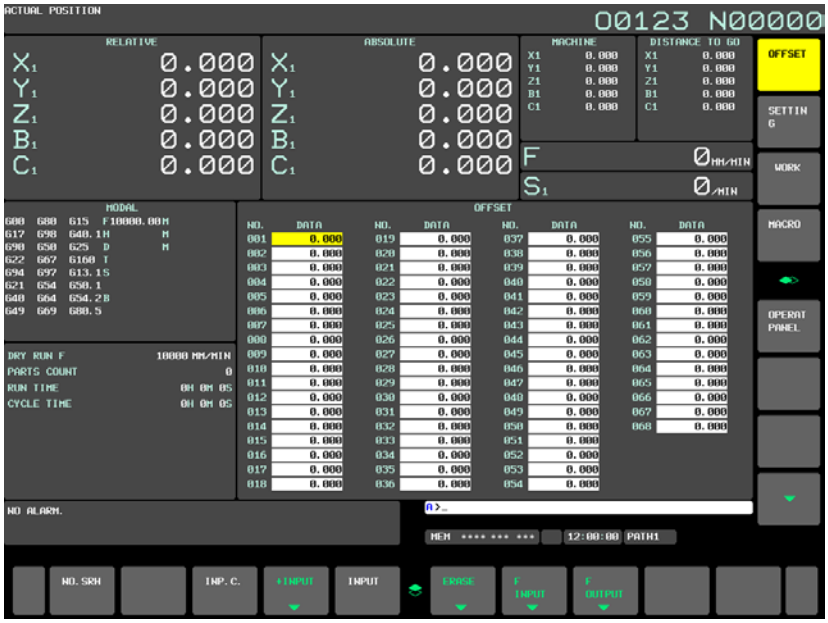


Fig.1.1.1 (c) Tool compensation memory A (15-inch display unit)

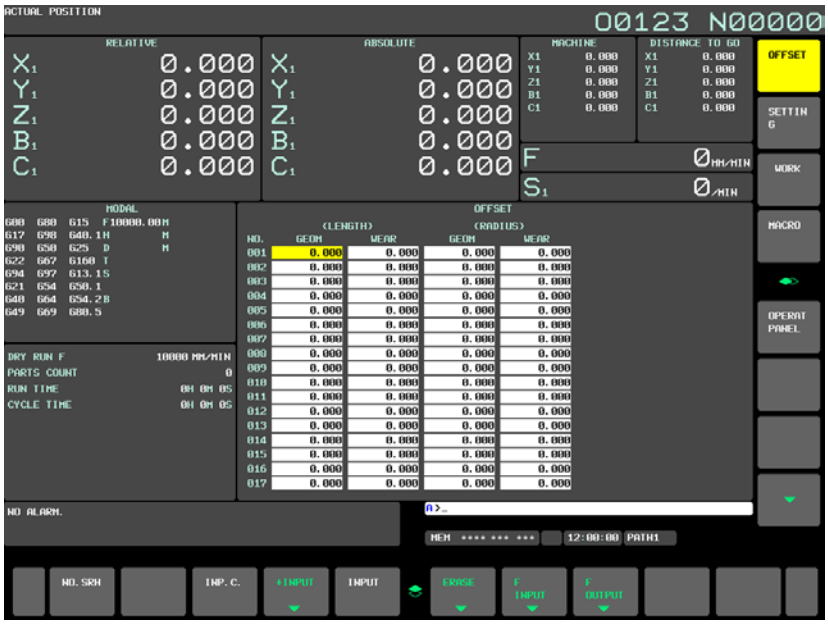


Fig. 1.1.1 (d) Tool compensation memory C (15-inch display unit)

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press horizontal soft key [NO.SRH].

4 To set a compensation value, enter a value and press horizontal 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 horizontal soft key [+INPUT]. Or, enter a new value and press horizontal soft key [INPUT].

Explanation

- Decimal point input

A decimal point can be used when entering a compensation value.

- Other setting method

An external input/output device can be used to input or output a tool offset value. See Chapter, “DATA INPUT/OUTPUT” in OPERATOR’S MANUAL (Common to Lathe System/Machining Center) (B-64694EN). A tool length compensation value can be set by measuring the tool length as described in the next subsection.

- Tool compensation memory

There are tool compensation memories A and C, which are classified as follows:

Tool compensation memory A

D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated the same.

Tool compensation memory C

D codes and H codes are treated differently. Tool geometry compensation and tool wear compensation are treated differently.

- Disabling entry of compensation values

The entry of compensation values may be disabled by setting bit 0 (WOF) and bit 1 (GOF) of parameter No.3290 (not applied to tool compensation memory A).

And then, the input of tool compensation values from the MDI can be inhibited for a specified range of offset numbers. The first offset number for which the input of a value is inhibited is set in parameter No. 3294. The number of offset numbers, starting from the specified first number, for which the input of a value is inhibited is set in parameter No. 3295.

Consecutive input values are set as follows:

1) When compensation values are input consecutively from offset numbers for which the input of values is enabled to offset numbers for which the input of values is inhibited, a warning is issued, but the compensation values in the range of the offset numbers for which the input of values is enabled are set.
2) When compensation values are input consecutively from offset numbers for which the input of values is inhibited to offset numbers for which the input of values is enabled, a warning is issued and the compensation values are not set.

1.1.2 Tool Length Measurement

The length of the tool can be measured and registered as the tool length compensation value by moving the reference tool and the tool to be measured until they touch the specified position on the machine.
The tool length can be measured along the X-, Y-, or Z-axis.
You can select measurement based on either relative coordinates or machine coordinates with bit 2 (CIM) of parameter No.11405.

Procedure for tool length measurement based on relative coordinates (for 8.4/10.4-inch display unit)

1 Use manual operation to move the reference tool until it touches the specified position on the machine (or workpiece.)

2 Press function key [POS] several times until the current position display screen with relative coordinates is displayed.

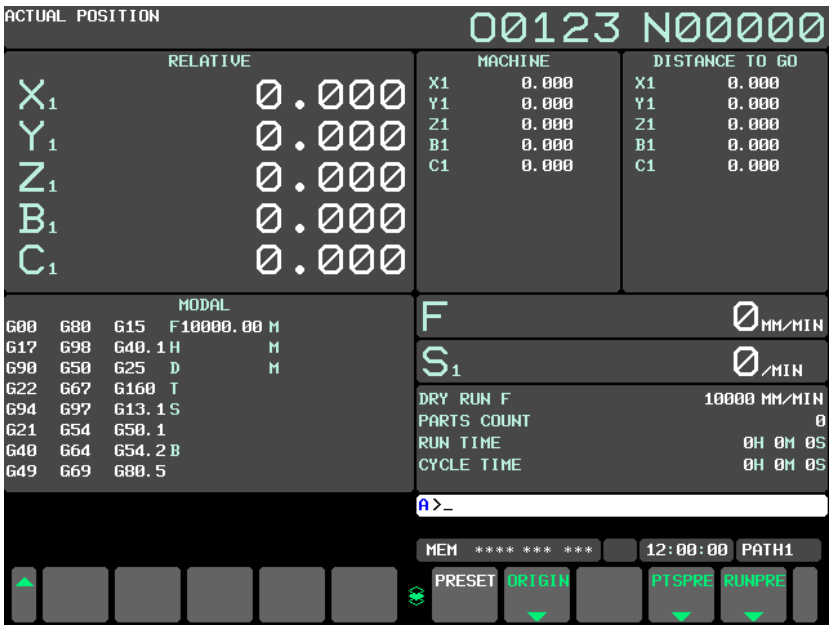


Fig. 1.1.2 (a) Current position display screen (10.4-inch display unit)

3 Reset the relative coordinate for the Z-axis to 0.

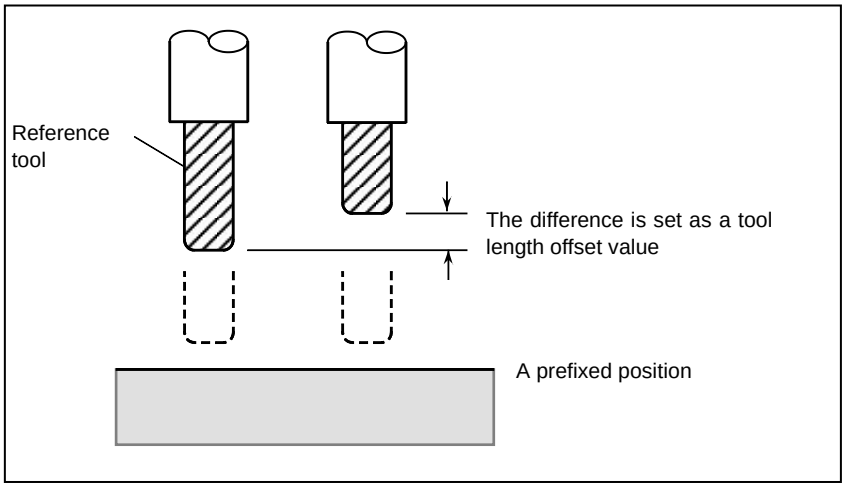
4 Press function key [OFS/SET] several times until the tool compensation screen is displayed.

5 Use manual operation to move the tool to be measured until it touches the same specified position. The difference between the length of the reference tool and the tool to be measured is displayed in the relative coordinates on the screen.

6 Move the cursor to the compensation number for the target tool (the cursor can be moved in the same way as for setting tool compensation values).

7 Press the address key [Z]. If either [X] or [Y] key is depressed instead of [Z] key, the X or Y axis relative coordinate value is input as an tool length compensation value.

8 Press the soft key [INP.C.]. The Z axis relative coordinate value is input and displayed as an tool length compensation value.



Procedure for tool length measurement based on relative coordinates (for 15/19-inch display unit)

1 Use manual operation to move the reference tool until it touches the specified position on the machine (or workpiece.)

2 Press function key [POS] to display the overall position display screen.

Fig. 1.1.2 (b) Current position display screen (15-inch display unit)

3 Reset the relative coordinate for the Z-axis to 0.

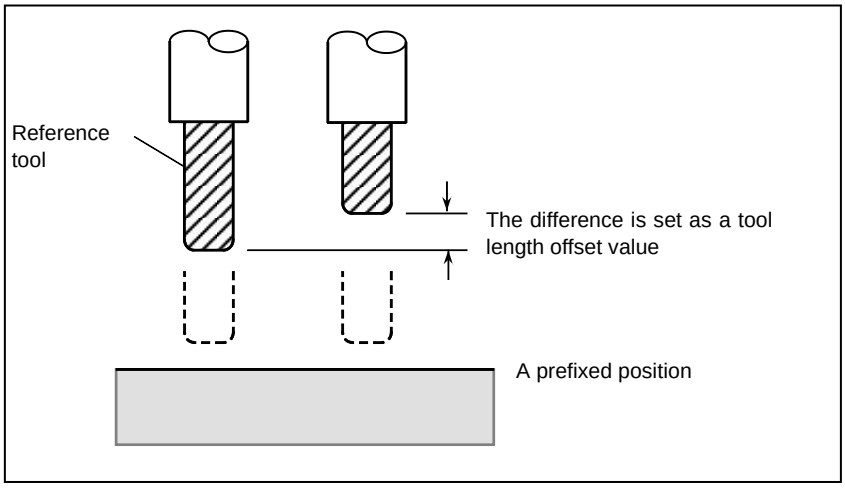
4 Press function key [OFS/SET] several times until the tool compensation screen is displayed.

5 Use manual operation to move the tool to be measured until it touches the same specified position. The difference between the length of the reference tool and the tool to be measured is displayed in the relative coordinates on the screen.

6 Move the cursor to the compensation number for the target tool (the cursor can be moved in the same way as for setting tool compensation values).

7 Press the address key [Z]. If either [X] or [Y] key is depressed instead of [Z] key, the X or Y axis relative coordinate value is input as an tool length compensation value.

8 Press the horizontal soft key [INP.C.]. The Z axis relative coordinate value is input and displayed as an tool length compensation value.



Procedure for tool length measurement based on machine coordinates

1 Press function key [POS] several times until the current position display screen with machine coordinates is displayed.

2 Press function key [OFS/SET] several times until the tool compensation screen is displayed.

3 Use manual operation to move the tool to be measured until it touches the same specified position.

4 Move the cursor to the compensation number for the target tool (the cursor can be moved in the same way as for setting tool compensation values).

5 Press the address key [Z]. If either [X] or [Y] key is depressed instead of [Z] key, the X or Y axis machine coordinate value is input as a value of “OFFSET”.

6 Press the soft key [INP.M.]. The value of “ADJUST” and “OFFSET” is displayed in warning area.

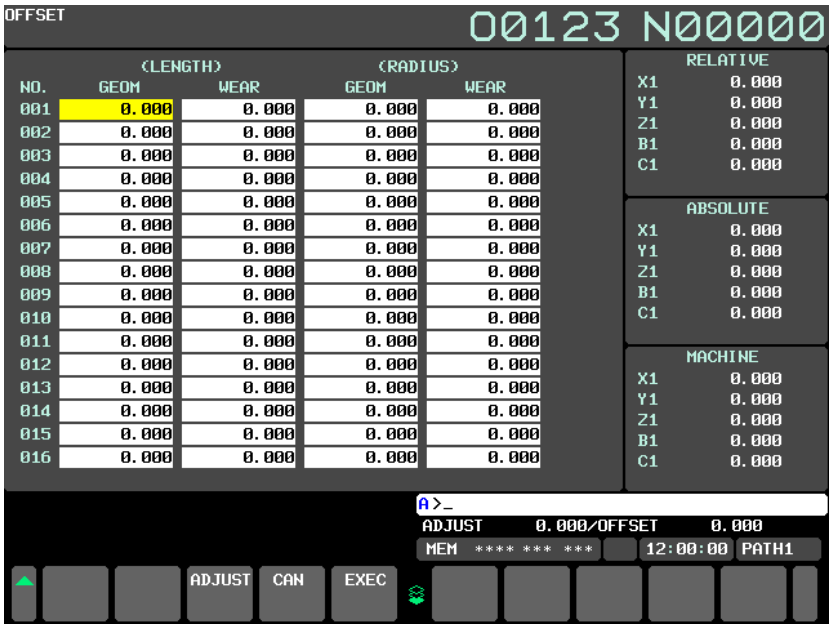


Fig. 1.1.2 (c) Setting screen of machine input (10.4-inch display unit)

7 Type the value of “ADJUST” and press the soft key [ADJUST]. The value of “ADJUST” and “OFFSET” is input.

8 Press the soft key [EXEC]. The value of “OFFSET” is input and displayed as a tool length compensation value.

1.1.3 Fine surface setting

In parameters related to high-quality machining, operators can select an effective setting pattern from the three kinds of parameter settings (called setting patterns) beforehand made patterns depending on the machining conditions. For these reasons, this function makes it easier to set parameters related to high-quality machining and to choose parameters according to the machining conditions.
Two screens are available in Fine surface setting; “FINE SURFACE SETTING” screen, and “PATTERN SELECT” screen. Setting patterns are prepared in “FINE SURFACE SETTING” screen and is selected in “PATTERN SELECT” screen. In this chapter, explain about “PATTERN SELECT” screen (selection for setting patterns). Please refer to CONNECTION MANUAL (FUNCTION)(B-64693EN-1) for “FINE SURFACE SETTING” screen (preparation for setting patterns).

Display of “PATTERN SELECT” screen

1 Press function key [OFS SET]

2 Press continuous menu key [▷] several times, soft key [MCNSET SELECT] is displayed.

3 Press soft key [MCNSET SELECT], and “PATTERN SELECT” screen is displayed.

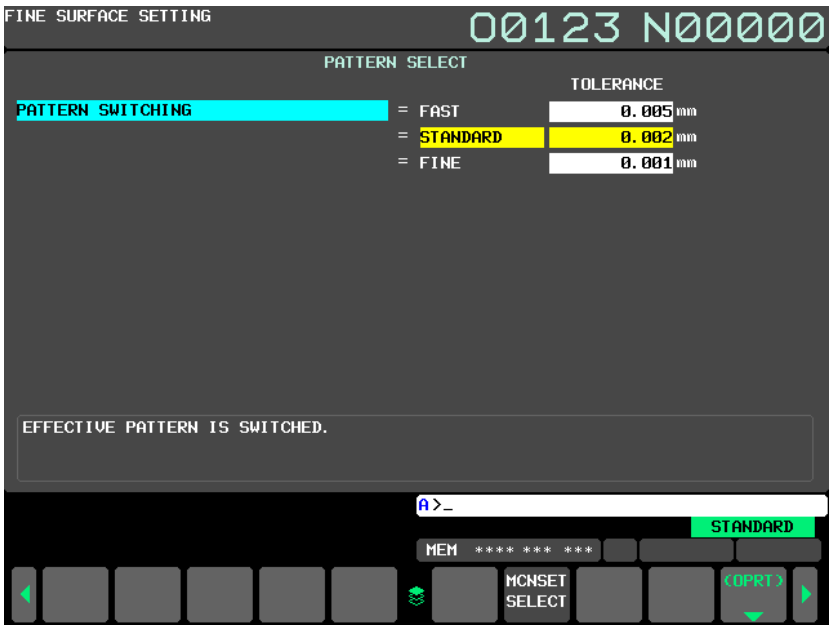


Fig. 1.1.3 (a) “PATTERN SELECT” screen

Operation in “PATTERN SELECT” screen

Setting of tolerance

1 Press cursor keys ← ↑ ↓ → to move cursor to the tolerance to be set.

2 Input setting data in key input buffer, and press input key INPUT or soft key [INPUT].

Fig. 1.1.3 (b) “PATTERN SELECT” screen (Input tolerance setting data)

NOTE
Setting of tolerance is not necessary if Smooth tolerance control+ is not used.

NOTE
1 When Smooth tolerance control+ is invalid, the item of tolerance is not displayed.
2 When the setting parameter is protected with the 8-Level data protection function, setting value cannot input in “PATTERN SELECT” screen. When setting value input in this state, a warning “WRITE PROTECT” is issued.
3 Setting data must be input on “PATTERN SELECT” screen during emergency stop or in MDI mode with automatic operation stopped. When performing input operation in other mode, the warning “SWITCH TO MDI MODE” is issued.
4 “PATTERN SELECT” screen is not displayed in a loader control path when Loader control function is valid.

Select setting pattern

1 Press cursor keys ← ↑ ↓ → to move cursor to the setting pattern to be enabled.
2 Press soft key [OPRT].
3 Press soft key [SELECT], and selected setting pattern is turned to effective.

Fig. 1.1.3 (c) "PATTERN SELECT" screen (Setting pattern selection)

Setting pattern selection can also be selected by program command. Please refer to "SELECT SETTING PATTERN OF FINE SURFACE SETTING BY PROGRAM COMMAND" for select method of setting pattern selection by program command.

NOTE
1 Setting data must be input on "PATTERN SELECT" screen during emergency stop or in MDI mode with automatic operation stopped. When performing input operation in other mode, the warning "SWITCH TO MDI MODE" is issued.
2 "PATTERN SELECT" screen is not displayed in a loader control path when Loader control function is valid.

APPENDIX

A LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND

A.1 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE

The function including address P in the argument of G code is shown below.

Function name	Machining center system	Lathe system G code system			Reference item
		A	B	C	
Dwell	G04	G04	G04	G04	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
G code preventing buffering	G04.1	G04.1	G04.1	G04.1	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
AI contour control II(high-precision contour control/AI high-precision contour control compatible command)	G05	G05	G05	G05	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
AI contour control (advanced preview control compatible command)	G08	G08	G08	G08	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable data input	G10	G10	G10	G10	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable parameter input	G10	G10	G10	G10	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable internal data change	G10.8	G10.8	G10.8	G10.8	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Machine configuration selecting function	G10.8	G10.8	G10.8	G10.8	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle speed fluctuation detection	G26	G26	G26	G26	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Reference position return	G30	G30	G30	G30	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Multi-step skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Torque limit skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Continuous high-speed skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Skip function for EGB axis	G31.8	G31.8	G31.8	G31.8	CONNECTION MANUAL(FUNCTION)
Wheel wear compensation	G41	G41	G41	G41	OPERATOR'S MANUAL(For Machining Center System) II. PROGRAMMING

Function name	Machining center system	Lathe system G code system A	B	C	Reference item
Scaling	G51	-	G51	G51	CONNECTION MANUAL(FUNCTION) 13.13 OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Polygon turning	G51.2	G51.2	G51.2	G51.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
		(G251)	(G251)	(G251)	
Synchronous, Composite and Superimposed control by program command	G51.4,	G51.4,	G51.4,	G51.4,	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
	G51.5,	G51.5,	G51.5,	G51.5,	
	G51.6,	G51.6,	G51.6,	G51.6,	
	G50.5	G50.5	G50.5	G50.5	
Flexible path axis assignment	G52.1,	G52.1,	G52.1,	G52.1,	CONNECTION MANUAL(FUNCTION)
	G52.2,	G52.2,	G52.2,	G52.2,	
	G52.3	G52.3	G52.3	G52.3	
High-speed G53 function	G53	G53	G53	G53	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Workpiece coordinate system	G54	G54	G54	G54	CONNECTION MANUAL(FUNCTION)
	(G54.1)	(G54.1)	(G54.1)	(G54.1)	OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Custom macro	G65,	G65,	G65,	G65,	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
	G66,	G66,	G66,	G66,	
	G66.1	G66.1	G66.1	G66.1	
Execution macro Note1)	G65,	G65,	G65,	G65,	Macro Executor PROGRAMMING MANUAL
	G66,	G66,	G66,	G66,	
	G66.1	G66.1	G66.1	G66.1	
	Note1)	Note1)	Note1)	Note1)	
Pattern Data Input	G65	G65	G65	G65	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Balanced cutting	-	G68	G68	G68	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System)
Tilted working plane indexing	G68.2	G68.2	G68.2	G68.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Tilted working plane indexing by tool axis direction	G68.3,	G68.3,	G68.3,	G68.3,	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Tilted working plane indexing (incremental multi-command)	G68.4	G68.4	G68.4	G68.4	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Multiple repetitive cycles	G70.7 to G76.7	G70 to G76	G70 to G76	G72 to G78	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
Figure copying	G72.1,	G72.1,	G72.1,	G72.1,	OPERATOR'S MANUAL(COMMON TO T/M SERIES)
	G72.2	G72.2	G72.2	G72.2	
Canned cycle	G74, G76	-	-	-	OPERATOR'S MANUAL(For Machining Center System)
Canned cycle for	G75,	G72,	G72,	G73,	OPERATOR'S MANUAL(For Lathe

Function name	Machining center system	Lathe system G code system A	B	C	Reference item
grinding	G77,	G74	G74	G75	System) II. PROGRAMMING
	G78,				
	G79				
Canned cycle	G82 to G84,	G82 to G85,	G82 to G85,	G82 to G85,	OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
	G87 to G89	G87 to G89,	G87 to G89,	G87 to G89,	
		G83.5,	G83.5,	G83.5,	
		G83.6,	G83.6,	G83.6,	
		G87.5,	G87.5,	G87.5,	
		G87.6	G87.6	G87.6	
Electronic gear box	G81	G81	G81	G81	CONNECTION MANUAL(FUNCTION)
Electronic gear box 2 pair	G81.5	G81.5	G81.5	G81.5	CONNECTION MANUAL(FUNCTION)
High-speed peck drilling cycle	-	G83.1	G83.1	G83.1	OPERATOR'S MANUAL(For Lathe System)
Rigid tapping cycle (FS15-T format)	G84.2	G84.2	G84.2	G84.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
Left-handed rigid tapping cycle(FS15-T format)	G84.3	-	-	-	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Machining Center System)
Constant surface speed control	G96	G96	G96	G96	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle Indexing Function	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle control with servo motor	G96.4	G96.4	G96.4	G96.4	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING

Note 1) : Arbitrary G code is optional with the following compilation parameter.
No.9013 to No.9022,No.9034,No.9045 to No.9047,No.9129 to No.9137

A.2 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE

The function including address P in the argument of M and S code is shown below.

Function name	M code format	related parameters	Reference item
Waiting M codes	M_P_	No.8110,No.8111, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Waiting M codes of high-speed Type	M_P_	No.8114,No.8115, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Waiting function by specifying start point	M_P_L_IP	STW(No.8101#1), No.8110,No.8111, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
(Custom macro) Subprogram	M98P_		OPERATOR'S MANUAL(COMMON to T/M series) II. PROGRAMMING
Program end	M99P_		
External subprogram call	M198P_	SBP(No.3404#2)	OPERATOR'S MANUAL(COMMON to T/M series) III. OPERATION
Custom macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_	MAA(No.6009#2)	OPERATOR'S MANUAL(COMMON to T/M series) II. PROGRAMMING
Execution macro Subprogram call	M98P_		Macro Executor PROGRAMMING MANUAL
Subprogram call for user program	M_P_		
Execution macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_		
Multi-spindle	S_P_	MPP(No.3703#3), No.3781	CONNECTION MANUAL(FUNCTION)

INDEX

<A>

AUTOMATIC TOOL LENGTH MEASUREMENT (G37) 104

<B>

Back Boring Cycle (G87)....46
Boring Cycle (G85)....43
Boring Cycle (G86)....44
Boring Cycle (G88)....48
Boring Cycle (G89)....50

<C>

Canned Cycle Cancel (G80)....73
Canned Cycle Cancel for Drilling (G80) 51
CANNED CYCLE FOR DRILLING....21
CANNED CYCLE OVERLAP FOR DRILLING....55
CANNED GRINDING CYCLE (FOR GRINDING MACHINE)....84
COMPENSATION FUNCTION....97
Continuous-feed Surface Grinding Cycle (G78)....92
COORDINATE SYSTEM ROTATION (G68, G69)...183
COORDINATE VALUE AND DIMENSION....16
CORNER CIRCULAR INTERPOLATION (G39)....179

<D>

DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION....128
Direct Constant-Dimension Plunge Grinding Cycle (G77) 89
Direction of Imaginary Tool Nose 119
Drilling Cycle Counter Boring Cycle (G82) 33
Drilling Cycle, Spot Drilling (G81) 32

<E>

Example for Using Canned Cycles for Drilling 52
Extraction override....73

<F>

Fine Boring Cycle (G76)....30
Fine surface setting 202
FUNCTIONS TO SIMPLIFY PROGRAMMING....21

<G>

GENERAL 3,9

<H>

HIGH-SPEED CUTTING FUNCTIONS....192
High-Speed Peck Drilling Cycle (G73)....26

<I>

Imaginary Tool Nose....117
INDEX TABLE INDEXING FUNCTION 79
IN-FEED CONTROL (FOR GRINDING MACHINE) .81
Interference Check 165
Interference check alarm function....169
Interference check avoidance function....171
Intermittent-feed Surface Grinding Cycle (G79) 95
INTERPOLATION FUNCTION....14

<L>

Left-Handed Rigid Tapping Cycle (G74) 65
Left-Handed Tapping Cycle (G74) 28
LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE....207
LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE....210
LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND 207

<M>

MEMORY OPERATION USING Series 15 PROGRAM FORMAT....190

<N>

NOTES ON READING THIS MANUAL 6
Notes on Tool Nose Radius Compensation....127
NOTES ON VARIOUS KINDS OF DATA 6

<O>

Offset Number and Offset Value 120
Operation to be performed if an interference is judged to occur....168
OPTIONAL CHAMFERING AND CORNER R 76
Override during Rigid Tapping....73
Override signal....75
Overview....128
OVERVIEW OF CUTTER COMPENSATION (G40-G42)....112
OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42) 117

<P>

Peck Drilling Cycle (G83) 35
Peck Rigid Tapping Cycle (G84 or G74)....69
Plunge Grinding Cycle (G75) 86
POLAR COORDINATE COMMAND (G15, G16)....16
PREPARATORY FUNCTION (G FUNCTION) 10
Prevention of Overcutting Due to Tool Radius / Tool Nose Radius Compensation....162

<R>

Reducing of Waiting Time of Spindle Speed Arrival in the Canned Cycle for Drilling 54
RIGID TAPPING....60
Rigid Tapping (G84)....61

<S>

SAFETY PRECAUTIONS s-1
SCREENS DISPLAYED BY FUNCTION KEY 195
SELECT SETTING PATTERN OF FINE SURFACE SETTING BY PROGRAM COMMAND 192

SETTING AND DISPLAYING DATA.....195
Setting and Displaying the Tool Compensation Value.195
Small-Hole Peck Drilling Cycle (G83) 37

<*T*>

Tapping Cycle (G84).....41
THREADING (G33).....14
TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10) 181
TOOL FIGURE AND TOOL MOTION BY PROGRAM 9
TOOL LENGTH COMPENSATION SHIFT TYPES ...97
Tool Length Measurement 199
Tool Movement in Offset Mode.....137
Tool Movement in Offset Mode Cancel.....155
Tool Movement in Start-up 132
TOOL OFFSET (G45 TO G48) 107
Tool Radius / Tool Nose Radius Compensation for Input from MDI 176

<*V*>

VECTOR RETENTION (G38).....178

<*W*>

Workpiece Position and Move Command.....121

REVISION RECORD

Edition	Date	Contents
01	Feb., 2019	

B-64694EN-2/01

B-64694EN-2/01