

FANUC Series 30*i*-MODEL B
FANUC Series 31*i*-MODEL B
FANUC Series 32*i*-MODEL B

For Machining Center System
OPERATOR'S MANUAL

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

The products in this manual are controlled based on Japan's "Foreign Exchange and Foreign Trade Law". The export of Series 30*i*-B, Series 31*i*-B5 from Japan is subject to an export license by the government of Japan. Other models in this manual may also be subject to export controls.

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 have tried as much as possible to describe all the various matters.

However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities.

Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible".

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.

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

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

CONTENTS

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

DEFINITION OF WARNING, CAUTION, AND NOTE

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

WARNING

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

CAUTION

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

NOTE

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

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

GENERAL WARNINGS AND CAUTIONS

WARNING

- 1 Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 2 Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 3 Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 4 When using a tool compensation function, thoroughly check the direction and amount of compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 6 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.
- 7 The OPERATOR'S MANUAL and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.
- 8 Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.

**CAUTION**

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

NOTE

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

 **WARNING****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.

7 Tool post interference check

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

8 Absolute/incremental mode

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

9 Plane selection

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

10 Torque limit skip

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

11 Programmable mirror image

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

12 Compensation function

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

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 numeric command

When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid.

Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4 Manual handle feed

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

5 Disabled override

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

6 Origin/preset operation

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

7 Workpiece coordinate system shift

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

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

 **WARNING****8 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.

9 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.

10 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.

11 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.

12 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.

13 Cutter and tool nose radius compensation in MDI mode

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

14 Program editing


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

WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1 **Memory backup battery replacement**

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

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

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

NOTE

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

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


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

Refer to the Section "Method of replacing battery" in the 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 α i series Maintenance Manual for details of the battery replacement procedure.

 **WARNING****3 Fuse replacement**

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

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


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

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

TABLE OF CONTENTS

SAFETY PRECAUTIONS	S-1
DEFINITION OF WARNING, CAUTION, AND NOTE	s-1
GENERAL WARNINGS AND CAUTIONS.....	s-2
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING	s-3
WARNINGS AND CAUTIONS RELATED TO HANDLING.....	s-5
WARNINGS RELATED TO DAILY MAINTENANCE.....	s-7
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	15
3.1 INVOLUTE INTERPOLATION (G02.2, G03.2).....	15
3.1.1 Automatic Speed Control for Involute Interpolation.....	19
3.1.2 Helical Involute Interpolation (G02.2, G03.2)	20
3.1.3 Involute Interpolation on Linear Axis and Rotary Axis (G02.2, G03.2)	21
3.2 THREADING (G33)	23
3.3 CONTINUOUS THREADING.....	24
3.4 MULTIPLE THREADING	24
3.5 CIRCULAR THREAD CUTTING B (G2.1,G3.1).....	26
3.6 GROOVE CUTTING BY CONTINUOUS CIRCLE MOTION (G12.4, G13.4).....	30
4 COORDINATE VALUE AND DIMENSION	41
4.1 POLAR COORDINATE COMMAND (G15, G16).....	41
5 FUNCTIONS TO SIMPLIFY PROGRAMMING	44
5.1 CANNED CYCLE FOR DRILLING.....	44
5.1.1 High-Speed Peck Drilling Cycle (G73).....	48
5.1.2 Left-Handed Tapping Cycle (G74)	50
5.1.3 Fine Boring Cycle (G76).....	51
5.1.4 Drilling Cycle, Spot Drilling (G81)	53
5.1.5 Drilling Cycle Counter Boring Cycle (G82)	54
5.1.6 Peck Drilling Cycle (G83).....	56
5.1.7 Small-Hole Peck Drilling Cycle (G83)	58
5.1.8 Tapping Cycle (G84).....	62
5.1.9 Boring Cycle (G85).....	64
5.1.10 Boring Cycle (G86).....	65
5.1.11 Back Boring Cycle (G87).....	67

5.1.12	Boring Cycle (G88).....	69
5.1.13	Boring Cycle (G89).....	71
5.1.14	Canned Cycle Cancel for Drilling (G80).....	72
5.1.15	Example for Using Canned Cycles for Drilling	73
5.2	IN-POSITION CHECK SWITCHING FOR DRILLING CANNED CYCLE	74
5.3	RIGID TAPPING	88
5.3.1	Rigid Tapping (G84)	88
5.3.2	Left-Handed Rigid Tapping Cycle (G74).....	92
5.3.3	Peck Rigid Tapping Cycle (G84 or G74).....	96
5.3.4	Canned Cycle Cancel (G80).....	99
5.3.5	Override during Rigid Tapping	99
5.3.5.1	Extraction override	99
5.3.5.2	Override signal	100
5.4	OPTIONAL CHAMFERING AND CORNER R.....	102
5.5	INDEX TABLE INDEXING FUNCTION.....	105
5.6	IN-FEED CONTROL (FOR GRINDING MACHINE).....	107
5.7	CANNED GRINDING CYCLE (FOR GRINDING MACHINE).....	109
5.7.1	Plunge Grinding Cycle (G75).....	111
5.7.2	Direct Constant-Dimension Plunge Grinding Cycle (G77).....	114
5.7.3	Continuous-feed Surface Grinding Cycle (G78).....	117
5.7.4	Intermittent-feed Surface Grinding Cycle (G79).....	120
5.8	MULTIPLE REPETITIVE CYCLE (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7, G76.7).....	123
5.8.1	Stock Removal in Turning (G71.7).....	124
5.8.2	Stock Removal in Facing (G72.7).....	135
5.8.3	Pattern Repeating (G73.7).....	139
5.8.4	Finishing Cycle (G70.7).....	142
5.8.5	End Face Peck Drilling Cycle (G74.7).....	146
5.8.6	Outer Diameter / Internal Diameter Drilling Cycle (G75.7)	148
5.8.7	Multiple Threading Cycle (G76.7).....	150
5.8.8	Restrictions on Multiple Repetitive Cycle (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7, and G76.7).....	155
6	COMPENSATION FUNCTION.....	157
6.1	TOOL LENGTH COMPENSATION SHIFT TYPES	157
6.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)	164
6.3	TOOL OFFSET (G45 TO G48).....	167
6.4	OVERVIEW OF CUTTER COMPENSATION (G40-G42).....	172
6.5	OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)	177
6.5.1	Imaginary Tool Nose.....	177
6.5.2	Direction of Imaginary Tool Nose	179
6.5.3	Offset Number and Offset Value.....	180
6.5.4	Workpiece Position and Move Command.....	180
6.5.5	Notes on Tool Nose Radius Compensation.....	185
6.6	DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION.....	187
6.6.1	Overview	187
6.6.2	Tool Movement in Start-up	191
6.6.3	Tool Movement in Offset Mode.....	197
6.6.4	Tool Movement in Offset Mode Cancel.....	215
6.6.5	Prevention of Overcutting Due to Tool Radius / Tool Nose Radius Compensation.....	221
6.6.6	Interference Check	224

6.6.6.1	Operation to be performed if an interference is judged to occur	227
6.6.6.2	Interference check alarm function	228
6.6.6.3	Interference check avoidance function	229
6.6.7	Tool Radius / Tool Nose Radius Compensation for Input from MDI.....	235
6.7	VECTOR RETENTION (G38).....	237
6.8	CORNER CIRCULAR INTERPOLATION (G39).....	238
6.9	3-DIMENSIONAL TOOL COMPENSATION (G40, G41).....	240
6.10	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	243
6.11	COORDINATE SYSTEM ROTATION (G68, G69).....	246
6.12	GRINDING WHEEL WEAR COMPENSATION	253
6.13	ACTIVE OFFSET VALUE CHANGE FUNCTION BASED ON MANUAL FEED	258
6.14	ROTARY TABLE DYNAMIC FIXTURE OFFSET.....	261
6.15	TOOL AXIS DIRECTION TOOL LENGTH COMPENSATION.....	267
6.15.1	Control Point Compensation of Tool Length Compensation Along Tool Axis...271	
6.16	SPINDLE UNIT COMPENSATION, NUTATING ROTARY HEAD TOOL LENGTH COMPENSATION	275
7	MEMORY OPERATION USING Series 15 PROGRAM FORMAT	279
7.1	MULTIPLE REPETITIVE CYCLE	280
7.1.1	Stock Removal in Turning (G71.7).....	281
7.1.2	Stock Removal in Facing (G72.7).....	293
7.1.3	Pattern Repeating (G73.7).....	297
7.1.4	Finishing Cycle (G70.7).....	300
7.1.5	End Face Peck Drilling Cycle (G74.7).....	304
7.1.6	Outer Diameter / Internal Diameter Drilling Cycle (G75.7)	305
7.1.7	Multiple Threading Cycle (G76.7).....	307
7.1.8	Restrictions on Multiple Repetitive Cycle	312
8	AXIS CONTROL FUNCTIONS.....	314
8.1	PARALLEL AXIS CONTROL.....	314
9	GAS CUTTING MACHINE	319
9.1	TOOL OFFSET B	319
9.2	CONER CONTROL BY FEED RATE	322
9.3	AUTOMATIC EXACT STOP CHECK	324
9.4	AXIS SWITCHING.....	327
9.5	GENTLE CURVE CUTTING.....	330
9.6	GENTLE NORMAL DIRECTION CONTROL.....	332
9.6.1	Linear Distance Setting	333
III. OPERATION		
1	SETTING AND DISPLAYING DATA.....	337
1.1	SCREENS DISPLAYED BY FUNCTION KEY 	337
1.1.1	Setting and Displaying the Tool Compensation Value	337
1.1.2	Tool Length Measurement	342
1.1.3	Tool Length/Workpiece Origin Measurement	345
1.1.4	Setting and Displaying the Rotary Table Dynamic Fixture Offset	364

1.1.5	Input of Tool Offset Value Measured B.....	367
1.1.6	Spindle Unit Compensation, Nutating Rotary Head Tool Length Compensation.....	367

APPENDIX

A	PARAMETERS.....	373
A.1	DESCRIPTION OF PARAMETERS.....	373
A.2	DATA TYPE.....	420
A.3	STANDARD PARAMETER SETTING TABLES.....	421

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

Lists parameters.

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-64484EN).
- 2 Some functions described in this manual may not be applied to some products. For detail, refer to the DESCRIPTIONS manual (B-64482EN).
- 3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-64490EN). 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 models indicated in the table below.

In the text, the abbreviations indicated below may be used.

Model name	Abbreviation	
FANUC Series 30 <i>i</i> -B	30 <i>i</i> -B	Series 30 <i>i</i>
FANUC Series 31 <i>i</i> -B	31 <i>i</i> -B	Series 31 <i>i</i>
FANUC Series 31 <i>i</i> -B5	31 <i>i</i> -B5	
FANUC Series 32 <i>i</i> -B	32 <i>i</i> -B	Series 32 <i>i</i>

NOTE

- 1 Unless otherwise noted, the model names 31*i*-B, 31*i*-B5, and 32*i*-B are collectively referred to as 30*i*. However, this convention is not necessarily observed when item 3 below is applicable.
- 2 Some functions described in this manual may not be applied to some products. For details, refer to the Descriptions (B-64482EN).

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 30*i*- MODEL B Series 31*i*- MODEL B Series 32*i*- MODEL B

The following table lists the manuals related to Series 30*i*-B, Series 31*i*-B, Series 32*i*-B. This manual is indicated by an asterisk(*).

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64482EN	
CONNECTION MANUAL (HARDWARE)	B-64483EN	
CONNECTION MANUAL (FUNCTION)	B-64483EN-1	
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64484EN	
OPERATOR'S MANUAL (For Lathe System)	B-64484EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64484EN-2	*
MAINTENANCE MANUAL	B-64485EN	
PARAMETER MANUAL	B-64490EN	
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>i</i> (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE <i>i</i> (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE <i>i</i> (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN-1	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64483EN-2	

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

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

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR αi series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR αi series DESCRIPTIONS	B-65272EN
FANUC AC SERVO MOTOR βi series DESCRIPTIONS	B-65302EN
FANUC AC SPINDLE MOTOR βi series DESCRIPTIONS	B-65312EN
FANUC SERVO AMPLIFIER αi series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER βi series DESCRIPTIONS	B-65322EN
FANUC SERVO MOTOR αis series FANUC SERVO MOTOR αi series FANUC AC SPINDLE MOTOR αi series FANUC SERVO AMPLIFIER αi series MAINTENANCE MANUAL	B-65285EN
FANUC SERVO MOTOR βis series FANUC AC SPINDLE MOTOR βi series FANUC SERVO AMPLIFIER βi series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR αi series FANUC AC SERVO MOTOR βi series FANUC LINEAR MOTOR $L i S$ series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR $D i S$ series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR $\alpha i/\beta i$ series, BUILT-IN SPINDLE MOTOR $B i$ series PARAMETER MANUAL	B-65280EN

The above servo motors and the corresponding spindles can be connected to the CNC covered in this manual. In the αi SV, αi SP, αi PS, and βi SV series, however, they can be connected only to 30 i -B-compatible versions. In the βi SVSP series, they cannot be connected.

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 an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.
- 2 In the header field of each page of this manual, a chapter title is indicated so that the reader can reference necessary information easily.
By finding a desired title first, the reader can reference necessary parts only.
- 3 This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
If a particular combination of operations is not described, it should not be attempted.

1.2 NOTES ON VARIOUS KINDS OF DATA

CAUTION

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

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

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

1.1 TOOL FIGURE AND TOOL MOTION BY PROGRAM9

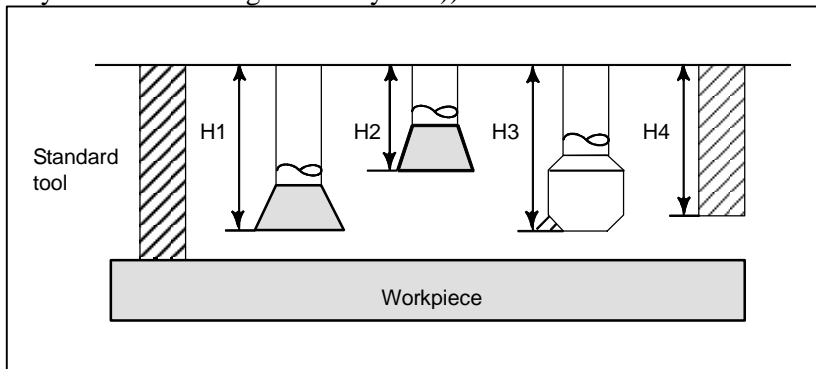
1.1 TOOL FIGURE AND TOOL MOTION BY PROGRAM

Explanation

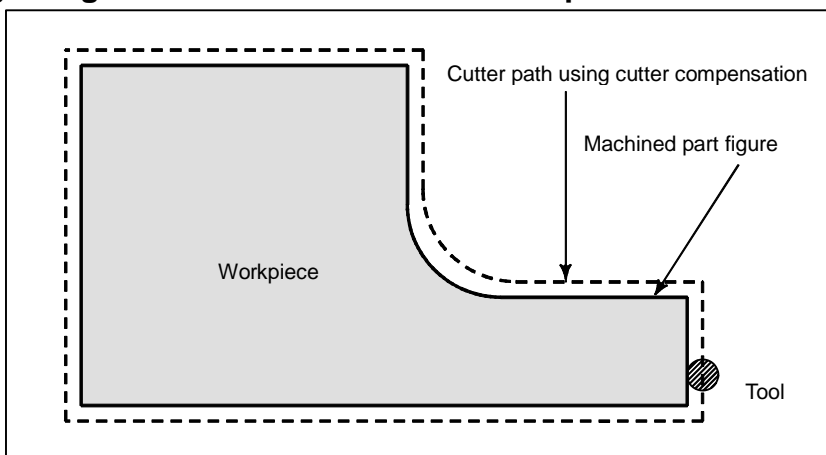
- Machining using the end of cutter - Tool length compensation function

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)), 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)).



- 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)), 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_ ;
      X_ ;
G00   Z_ ;
      X_ ;
G01   X_ ;
      :
    
```

} G01 is effective in this range.

} G00 is effective in this range.

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 (a).
 - (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 other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010 occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a 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 (a) 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	
G02.1, G03.1		Circular thread cutting B CW/CCW	
G02.2, G03.2		Involute interpolation CW/CCW	
G02.3, G03.3		Exponential interpolation CW/CCW	
G02.4, G03.4		3-dimensional coordinate system conversion CW/CCW	
G04	00	Dwell	
G05		AI contour control (high-precision contour control compatible command), High-speed cycle machining, High-speed binary program operation	
G05.1		AI contour control / Nano smoothing / Smooth interpolation	
G05.4		HRV3, 4 on/off	
G06.2	01	NURBS interpolation	
G07	00	Hypothetical axis interpolation	
G07.1 (G107)		Cylindrical interpolation	
G08		AI contour control (advanced preview control compatible command)	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G10.9		Programmable switching of diameter/radius specification	
G11	Programmable data input mode cancel		
G12.1	21	Polar coordinate interpolation mode	
G13.1		Polar coordinate interpolation cancel mode	
G12.4	00	Groove cutting by continuous circle motion (CW)	
G13.4		Groove cutting by continuous circle motion (CCW)	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis Yp: Y axis or its parallel axis Zp: Z axis or its parallel axis
G17.1		Plane conversion function	
G18		ZpXp plane selection	
G19		YpZp plane selection	
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	
G30.1		Floating 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
G34	Variable lead threading		
G35	Circular threading CW		
G36	Circular threading CCW		

2. PREPARATORY FUNCTION
(G FUNCTION)

PROGRAMMING

B-64484EN-2/03

Table 2 (a) G code list

G code	Group	Function	
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 3-dimensional cutter compensation : cancel	
G41		Tool radius/tool nose radius compensation : left 3-dimensional cutter compensation : left	
G42		Tool radius/tool nose radius compensation : right 3-dimensional cutter compensation : right	
G41.2		3-dimensional cutter compensation : left (type 1)	
G41.3		3-dimensional cutter compensation : leading edge offset	
G41.4		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)	
G41.5		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)	
G41.6		3-dimensional cutter compensation : left (type 2)	
G42.2		3-dimensional cutter compensation : right (type 1)	
G42.4		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)	
G42.5		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)	
G42.6		3-dimensional cutter compensation : right (type 2)	
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 -	
G43.1		Tool length compensation in tool axis direction	
G43.3		Nutating rotary head tool length compensation	
G43.4		Tool center point control (type 1)	
G43.5		Tool center point control (type 2)	
G43.7		Tool offset	
G44.1		Tool offset conversion	
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	
G44.9	27	Spindle unit compensation	
G49.9		Spindle unit compensation cancel	
G50	11	Scaling cancel	
G51		Scaling	
G50.1	22	Programmable mirror image cancel	
G51.1		Programmable mirror image	
G50.2	31	Polygon turning cancel	
G51.2		Polygon turning	
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		Local coordinate system setting	
G53		Machine coordinate system setting	
G53.1		Tool axis direction control	
G53.6		Tool center point retention type tool axis direction control	

Table 2 (a) G code list

G code	Group	Function
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
G54.2	23	Rotary table dynamic fixture offset
G54.4	33	Workpiece setting error compensation
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)
G70.7	00	Finishing cycle
G71.7		Outer surface rough machining cycle
G72.7		End rough machining cycle
G73.7		Closed loop cutting cycle
G74.7		End cutting off cycle
G75.7		Outer or inner cutting off cycle
G76.7		Multiple threading cycle
G72.1		Figure copying (rotary copy)
G72.2		Figure copying (linear copy)
G73		09
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
G81.1	00	Chopping function/High precision oscillation function

Table 2 (a) G code list

G code	Group	Function
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
G112	21	Polar coordinate interpolation mode
G113		Polar coordinate interpolation mode cancel
G160	20	In-feed control cancel
G161		In-feed control

3 INTERPOLATION FUNCTION

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

3.1 INVOLUTE INTERPOLATION (G02.2, G03.2).....	15
3.2 THREADING (G33).....	23
3.3 CONTINUOUS THREADING.....	24
3.4 MULTIPLE THREADING.....	24
3.5 CIRCULAR THREAD CUTTING B (G2.1,G3.1).....	26
3.6 GROOVE CUTTING BY CONTINUOUS CIRCLE MOTION (G12.4, G13.4).....	30

3.1 INVOLUTE INTERPOLATION (G02.2, G03.2)

Overview

Involute curve machining can be performed by using involute interpolation. Cutter compensation can be performed. Involute interpolation eliminates the need for approximating an involute curve with minute segments or arcs, and continuous pulse distribution is ensured even in high-speed operation of small blocks. Accordingly, high-speed operation can be performed smoothly. Moreover, machining programs can be created more easily, and the size of machining programs can be reduced.

In involute interpolation, the following two types of feedrate override functions are automatically executed, and a favorable cutting surface can be formed with high precision. (Automatic speed control function for involute interpolation)

- Override in cutter compensation mode
- Override in the vicinity of basic circle

Format

Involute interpolation on the Xp-Yp plane

G17 G02.2 Xp_ Yp_ I_ J_ R_ F_ ;

G17 G03.2 Xp_ Yp_ I_ J_ R_ F_ ;

Involute interpolation on the Zp-Xp plane

G18 G02.2 Zp_ Xp_ K_ I_ R_ F_ ;

G18 G03.2 Zp_ Xp_ K_ I_ R_ F_ ;

Involute interpolation on the Yp-Zp plane

G19 G02.2 Yp_ Zp_ J_ K_ R_ F_ ;

G19 G03.2 Yp_ Zp_ J_ K_ R_ F_ ;

Where,

G02.2 : Involute interpolation (clockwise)

G03.2 : Involute interpolation (counterclockwise)

G17/G18/G19 : Xp-Yp/Zp-Xp/Yp-Zp plane selection

Xp_ : X-axis or an axis parallel to the X-axis (specified in a parameter)

Yp_ : Y-axis or an axis parallel to the Y-axis (specified in a parameter)

Zp_ : Z-axis or an axis parallel to the Z-axis (specified in a parameter)

I_, J_, K_ : Center of the base circle for an involute curve viewed from the start point

R_ : Base circle radius

F_ : Cutting feedrate

Explanation

Involute curve machining can be performed by using involute interpolation. Involute interpolation ensures continuous pulse distribution even in high-speed operation in small blocks, thus enabling smooth and high-speed machining. Moreover, machining programs can be created more easily, and the size of machining programs can be reduced.

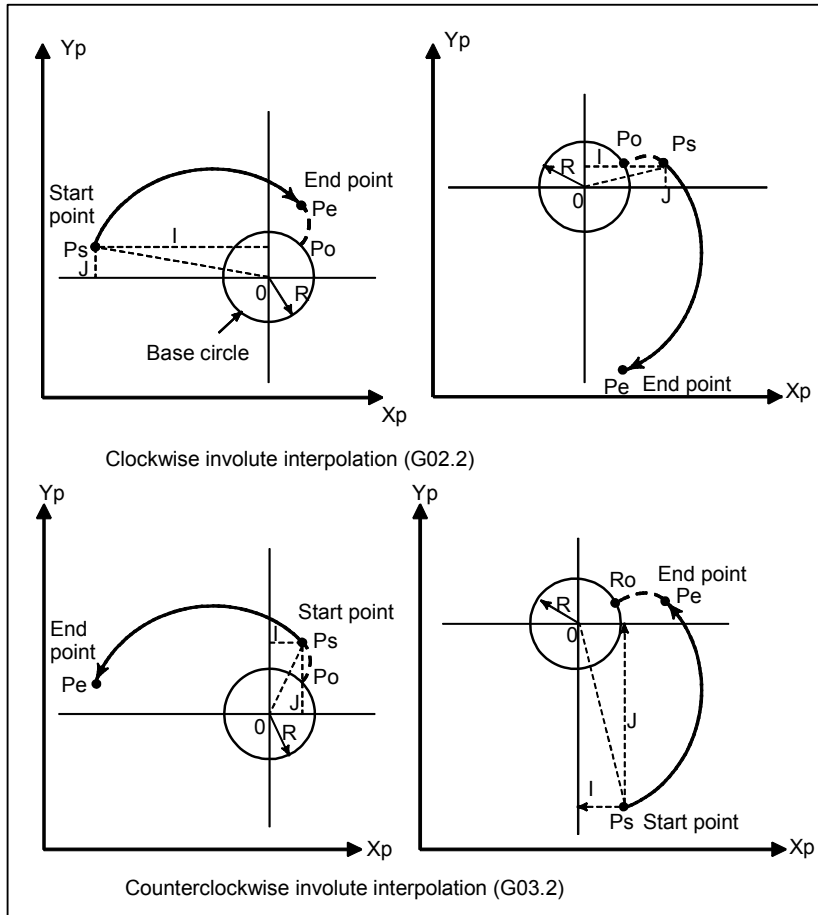


Fig. 3.1 (a) Actual movement

- Involute curve

An involute curve on the X-Y plane is defined as follows ;

$$X(\theta) = R [\cos \theta + (\theta - \theta_0) \sin \theta] + X_0$$

$$Y(\theta) = R [\sin \theta - (\theta - \theta_0) \cos \theta] + Y_0$$

where,

X_0, Y_0 : Coordinates of the center of a base circle

R : Base circle radius

θ_0 : Angle of the start point of an involute curve

θ : Angle of the point where a tangent from the current position to the base circle contacts the base circle

$X(\theta), Y(\theta)$: Current position on the X-axis and Y-axis

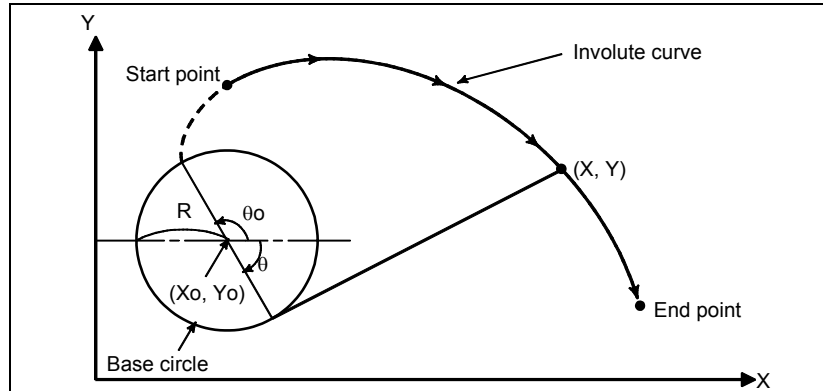


Fig. 3.1 (b) Involute curve

Involute curves on the Z-X plane and Y-Z plane are defined in the same way as an involute curve on the X-Y plane.

- Start point and end point

The end point of an involute curve is specified using address Xp, Yp, or Zp. An absolute value or incremental value is used to specify an Xp, Yp, or Zp value. When using an incremental value, specify the coordinates of the end point viewed from the start point of the involute curve.

When no end point is specified, alarm PS0241 is issued.

If the specified start point or end point lies within the base circle, alarm PS0242 is issued. The same alarm is issued if cutter compensation C causes the offset vector to enter the base circle. Be particularly careful when applying an offset to the inside of an involute curve.

- Base circle specification

The center of a base circle is specified with I, J, and K, corresponding to X, Y, and Z. The value following I, J, or K is a vector component defined when the center of the base circle is viewed from the start point of the involute curve; this value must always be specified as an incremental value, regardless of the G90/G91 setting. Assign a sign to I, J, and K according to the direction.

If I, J, and K are all left unspecified, or I0, J0, K0 is specified, alarm PS0241 or PS0242 is issued.

If R is not specified, or $R \leq 0$, alarm PS0241 or PS0242 is issued.

- Choosing from two types of involute curves

When only a start point and I, J, and K data are given, two types of involute curves can be created. One type of involute curve extends towards the base circle, and the other extends away from the base circle. When the specified end point is closer to the center of the base circle than the start point, the involute curve extends toward the base circle. In the opposite case, the involute curve extends away from the base circle.

- Feedrate

The cutting feedrate specified in an F code is used as the feedrate for involute interpolation. The feedrate along the involute curve (feedrate along the tangent to the involute curve) is controlled to satisfy the specified feedrate.

- Plane selection

As with circular interpolation, the plane to which to apply involute interpolation can be selected using G17, G18, and G19.

- Cutter compensation

Cutter compensation can be applied to involute curve machining. As with linear and circular interpolation, G40, G41, and G42 are used to specify cutter compensation.

G40: Cutter compensation cancel

G41: Cutter compensation left

G42: Cutter compensation right

First, a point of intersection with a segment or an arc is approximated both at the start point and at the end point of the involute curve. An involute curve passing through the two approximated points of intersection with the start point and end point becomes the tool center path.

Before selecting the involute interpolation mode, specify G41 or G42, cancel involute interpolation, and then specify G40. G41, G42, and G40 for cutter compensation cannot be specified in the involute interpolation mode.

- Automatic speed control

Cutting precision can be improved by automatically overriding the programmed feedrate during involute interpolation. See a subsequent subsection, "Automatic Speed Control for Involute Interpolation."

- Specifiable G codes

The following G codes can be specified in involute interpolation mode:

G04 : Dwell

G10 : Programmable data input

G17 : X-Y plane selection

G18 : Z-X plane selection

G19 : Y-Z plane selection

G65 : Macro call

G66 : Macro modal call

G67 : Macro modal call cancel

G90 : Absolute programming

G91 : Incremental programming

- Modes that allow involute interpolation specification

Involute interpolation can be specified in the following G code modes:

G41 : Cutter compensation left

G42 : Cutter compensation right

G51 : Scaling

G51.1 : Programmable mirror image

G68 : Coordinate rotation

- End point error

As shown below the end point may not be located on an involute curve that passes through the start point (Fig. 3.1 (c)).

When an involute curve that passes through the start point deviates from the involute curve that passes through the end point by more than the value set in parameter No. 5610, alarm PS0243 is issued.

If there is an end point error, the programmed feedrate changes by the amount of error.

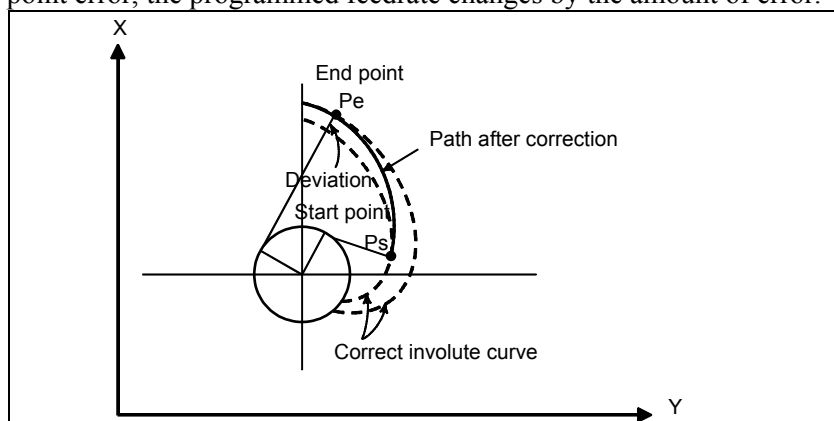


Fig. 3.1 (c) End point error in counterclockwise involute interpolation (G03.2)

3.1.1 Automatic Speed Control for Involute Interpolation

This function automatically overrides the programmed feedrate in two different ways during involute interpolation. With this function, a favorable cutting surface can be formed with high precision.

- Override in cutter compensation mode
- Override in the vicinity of basic circle

- Override in cutter compensation mode

When cutter compensation is applied to involute interpolation, control is exercised in ordinary involute interpolation so that the tangential feedrate on the tool-center path always keeps the specified feedrate.

Under the control, the actual cutting feedrate (feedrate around the perimeter of the tool (cutting point) on the path specified in the program) changes because the curvature of the involute curve changes every moment.

If the tool is offset in the inward direction of the involute curve in particular, the actual cutting feedrate becomes higher than the specified feedrate as the tool gets nearer to the base circle.

For smooth machining, it is desirable to control the actual cutting feedrate so that the feedrate keeps the specified feedrate. This function calculates an appropriate override value for the ever-changing curvature of the involute curve in the involute interpolation mode after cutter compensation. The function also controls the actual cutting feedrate (tangential feedrate at the cutting point) so that it always keeps the specified feedrate.

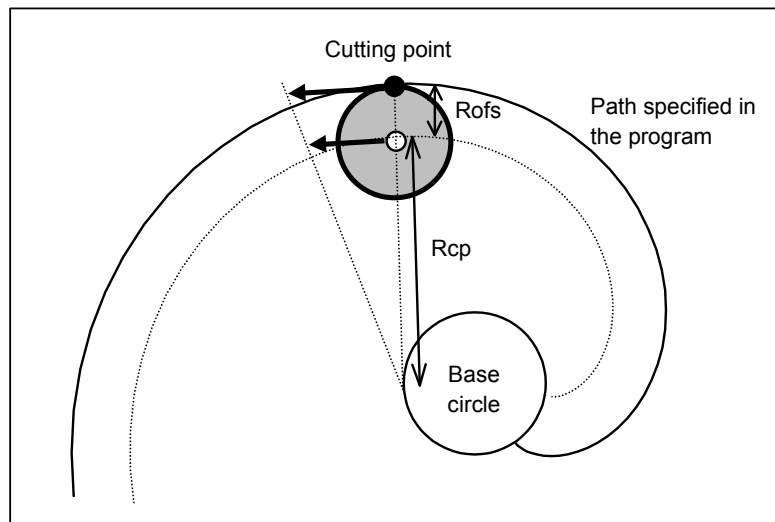


Fig. 3.1 (d) Override for inward offset by cutter compensation

Inward offset

$$\text{OVR} = \text{Rcp}/(\text{Rcp} + \text{Rofs}) \times 100$$

Outward offset

$$\text{OVR} = \text{Rcp}/(\text{Rcp} - \text{Rofs}) \times 100$$

where,

Rcp : Radius of curvature at the center of the tool of the involute curve passing through the center of the tool

Rofs : Radius of the cutter

- Clamping the override

The lower limit of override is specified in parameter No. 5620 so that the override for inward offset by cutter compensation or the override in the vicinity of the basic circle will not bring the speed of the tool center to zero in the vicinity of the basic circle.

The lower limit of override (OVR1o) is specified in parameter No. 5620 so that the inward offset will not reduce the speed of the tool center to a very low level in the vicinity of the basic circle.

Accordingly, the feedrate is clamped but does not fall below the level determined by the programmed feedrate and the lower limit of override (OVR1o).

The outward offset may increase the override to a very high level, but the feedrate will not exceed the maximum cutting feedrate.

- Clamping the acceleration in the vicinity of basic circle

If the acceleration calculated from the radius of curvature of the involute curve exceeds a value specified in the corresponding parameter, the tangential velocity is controlled so that the actual acceleration will not exceed the value specified in the parameter. Because the acceleration is always limited to a constant level, efficient velocity control can be performed for each machine. Because smooth velocity control can be performed continuously, impacts in machining in the vicinity of the basic circle can be reduced.

To calculate the acceleration, the radius of curvature of the involute curve and the tangential velocity are substituted into the following formula of circular acceleration:

Acceleration = $F \times F/R$

F: Tangential velocity

R: Radius of curvature

The maximum permissible acceleration is specified in parameter No. 1735.

If the calculated acceleration exceeds the maximum permissible acceleration, the feedrate is clamped to the level calculated by the following expression:

$$\text{Clamp level} = \sqrt{\text{Radius of curvature} \times \text{Maximum permissible acceleration}}$$

If the calculated clamp level falls below the lower limit of feedrate, the lower limit of feedrate becomes the clamp level. The lower limit of feedrate is specified in parameter No. 1732.

3.1.2 Helical Involute Interpolation (G02.2, G03.2)

As with arc helical involute interpolation, this function performs helical involute interpolation on the two axes involute interpolation and on up to four other axes simultaneously.

Format

Helical involute interpolation in Xp-Yp plane

G17 $\left\{ \begin{array}{l} \text{G02.2} \\ \text{G03.2} \end{array} \right\} \text{Xp_Yp_I_J_R_}\alpha\text{_}\beta\text{_}\gamma\text{_}\delta\text{_F_};$

Helical involute interpolation in Zp-Xp plane

G18 $\left\{ \begin{array}{l} \text{G02.2} \\ \text{G03.2} \end{array} \right\} \text{Zp_Xp_K_I_R_}\alpha\text{_}\beta\text{_}\gamma\text{_}\delta\text{_F_};$

Helical involute interpolation in Yp-Zp plane

G19 $\left\{ \begin{array}{l} \text{G02.2} \\ \text{G03.2} \end{array} \right\} \text{Yp_Zp_J_K_R_}\alpha\text{_}\beta\text{_}\gamma\text{_}\delta\text{_F_};$

$\alpha, \beta, \gamma, \delta$: Optional axis other than the axes of involute interpolation. Up to four axes can be specified.

3.1.3 Involute Interpolation on Linear Axis and Rotary Axis (G02.2, G03.2)

By performing involute interpolation in the polar coordinate interpolation mode, involute cutting can be carried out. Cutting is performed along an involute curve drawn in the plane formed by a linear axis and a rotary axis.

Format

If the linear axis is the X-axis or an axis parallel to the X-axis, the plane is considered to be the Xp-Yp plane, and I and J are used.

$\left. \begin{array}{l} \text{G02.2} \\ \text{G03.2} \end{array} \right\} \text{ X_C_I_J_R_F_ ;}$

If the linear axis is the Y-axis or an axis parallel to the Y-axis, the plane is considered to be the Yp-Zp plane, and J and K are used.

$\left. \begin{array}{l} \text{G02.2} \\ \text{G03.2} \end{array} \right\} \text{ Y_C_J_K_R_F_ ;}$

If the linear axis is the Z-axis or an axis parallel to the Z-axis, the plane is considered to be the Zp-Xp plane, and K and I are used.

$\left. \begin{array}{l} \text{G02.2} \\ \text{G03.2} \end{array} \right\} \text{ Z_C_K_I_R_F_ ;}$

G02.2 : Clockwise involute interpolation

G03.2 : Counterclockwise involute interpolation

Example) If the linear axis is the X-axis

X, C : End point of the involute curve

I, J : Center of the basic circle of the involute curve, viewed from the start point

R : Radius of basic circle

F : Cutting feedrate

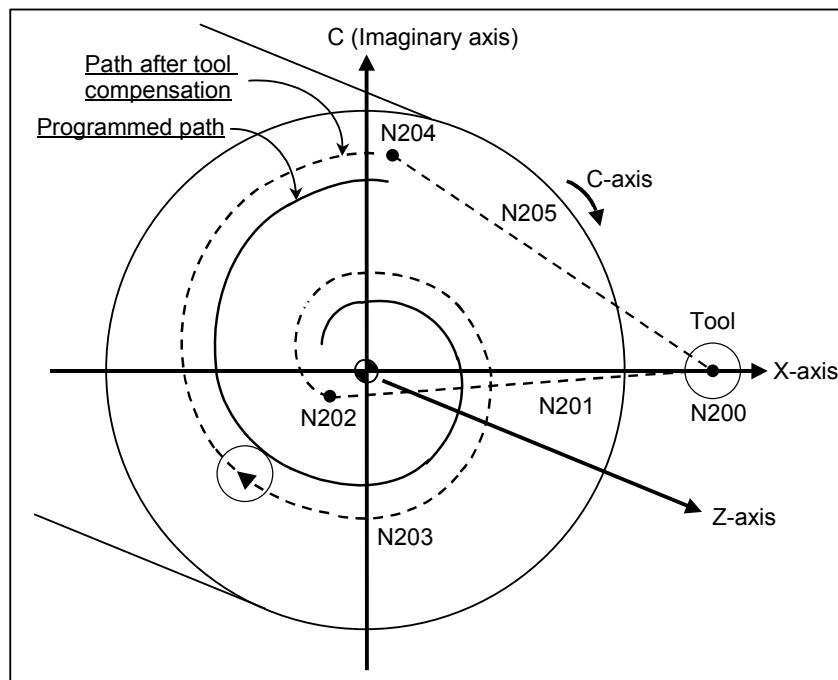
Example

Fig. 3.1 (e) Involute interpolation in the polar coordinate interpolation mode

```

O0001 ;
:
:
N010 T0101 ;
:
:
N100 G90 G00 X15.0 C0 Z0 ;           Positioning to the start point
N200 G12.1 ;                          Polar coordinate interpolation started
N201 G41 G00 X-1.0 ;
N202 G01 Z-2.0 F__ ;
N203 G02.2 X1.0 C9.425 I1.0 J0 R1.0 ;  Involute interpolation during polar coordinate interpolation
N204 G01 Z0 ;
N205 G40 G00 X15.0 C0 ;
N206 G13.1 ;                          Polar coordinate interpolation cancelled
N300 Z_ ;
N400 X_ C_ ;
:
:
M30 ;

```

Limitation**- Number of involute curve turns**

Both the start point and end point must be within 100 turns from the point where the involute curve starts. An involute curve can be specified to make one or more turns in a single block. If the specified start point or end point is beyond 100 turns from the point where the involute curve starts, alarm PS0242 is issued.

- Unspecifiable functions

In involute interpolation mode, optional chamfering and corner R cannot be specified.

- Mode that does not allow involute interpolation specification

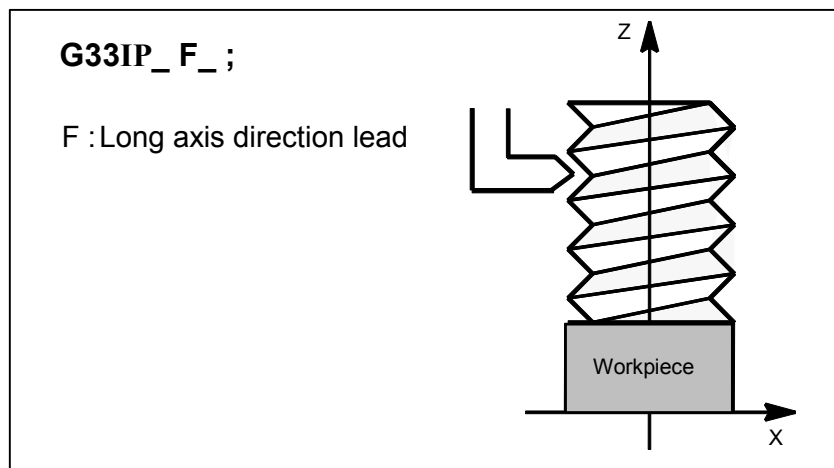
Involute interpolation cannot be used in the following mode:

G07.1: Cylindrical interpolation

3.2 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



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

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

	Least command increment	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 spindle speed is limited as follows :

$$1 \leq \text{spindle speed} \leq (\text{Maximum feedrate}) / (\text{Thread lead})$$

Spindle speed : min^{-1}

Thread lead : mm or inch

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

NOTE

- 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)

Example

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

3.3 CONTINUOUS THREADING

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

Explanation

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



Fig. 3.3 (a) Continuous threading

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

3.4 MULTIPLE THREADING

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

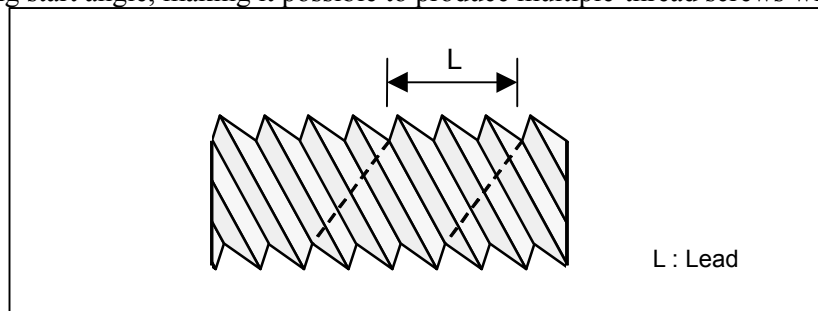


Fig. 3.4 (a) Multiple thread screws

Format**(Constant lead threading)****G33 IP _ F_ Q_ ;**

IP : End point

F_ : Lead in longitudinal direction

G33 IP _ Q_ ;

Q_ : Threading start angle

Explanation**- Available threading commands**

G33: Constant lead threading

G34: Variable lead threading

G76.7: Multiple threading cycle

Limitation**- Start angle**

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

- Start angle increment

The start angle (Q) increment is 0.001 degrees. Note that no decimal point can be specified.

Example:

For a shift angle of 180 degrees, specify Q180000.

Q180.000 cannot be specified, because it contains a decimal point.

- Specifiable start angle range

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

- Multiple threading cycle (G76.7)

For the G76 multiple threading cycle command, always use the FS15 tape format.

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

```
G00 X40.0 ;
G33 W-38.0 F4.0 Q0 ;
G00 X72.0 ;
      W38.0 ;
      X40.0 ;
G33 W-38.0 F4.0 Q180000 ;
G00 X72.0 ;
      W38.0 ;
```

3.5 CIRCULAR THREAD CUTTING B (G2.1,G3.1)

Overview

Circular thread cutting B can perform circular interpolation on two axes and, at the same time, can perform linear interpolation between the major axis of the two axes subject to circular interpolation, which has a longer traveling distance, and up to two other, arbitrary axes. This circular thread cutting function does not move the tool in synchronization with the rotation of the spindle (workpiece) using the spindle motor, but controls the rotation of the workpiece using a servo motor (rotation axis) to perform threading at equal pitches along cylindrical material, grooving, tool grinding, and other machining.

Application example

For example, the grooving shown in Fig. 3.5 (a) can be performed by executing circular interpolation on the ZpXp plane in synchronization with linear interpolation on the Z-axis and the C-axis.

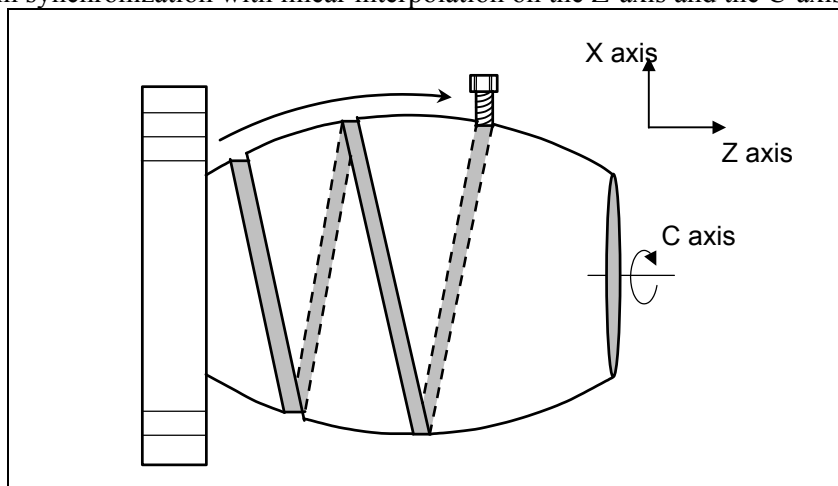


Fig. 3.5 (a) Example of grooving

Format

Xp-Yp plane

$$G17 \begin{Bmatrix} G02.1 \\ G03.1 \end{Bmatrix} X_Y_ \alpha_ \beta_ \begin{Bmatrix} I_ J_ \\ R_ \end{Bmatrix} F_ ;$$

Zp-Xp plane

$$G18 \begin{Bmatrix} G02.1 \\ G03.1 \end{Bmatrix} Z_X_ \alpha_ \beta_ \begin{Bmatrix} K_ I_ \\ R_ \end{Bmatrix} F_ ;$$

Yp-Zp plane

$$G19 \begin{Bmatrix} G02.1 \\ G03.1 \end{Bmatrix} Y_Z_ \alpha_ \beta_ \begin{Bmatrix} J_ K_ \\ R_ \end{Bmatrix} F_ ;$$

G02.1: Clockwise circular thread cutting B command
 G03.1: Counterclockwise circular thread cutting B command

X,Y,Z: Coordinates of the end point for circular interpolation
 α,β: Coordinates of the end point for linear interpolation
 I,J,K: Signed distance from the start point to the center of an arc
 R: Arc radius
 F: Feedrate in the major axis direction

α and β are arbitrary axes other than the circular interpolation axis. Up to two such axes can be specified.

X, Y, Z, I, J, K, and R are the same as those for G02 and G03.

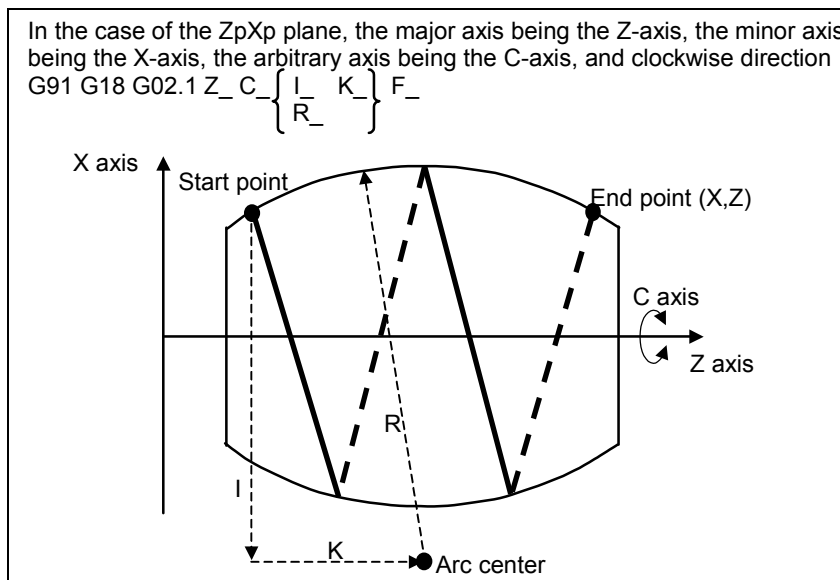


Fig. 3.5 (b)

Explanation

Circular thread cutting B can perform circular interpolation on two axes and, at the same time, can perform linear interpolation between the major axis of the two axes subject to circular interpolation, which has a longer traveling distance, and up to two other, arbitrary axes. This circular thread cutting function does not move the tool in synchronization with the rotation of the spindle (workpiece) using the spindle motor, but controls the rotation of the workpiece using a servo motor (rotation axis) to perform threading at equal pitches along cylindrical material, grooving, tool grinding, and other machining.

- Relationship between major axis and minor axis

The relationship between the major axis and minor axis is as shown in Fig. 3.5 (c).

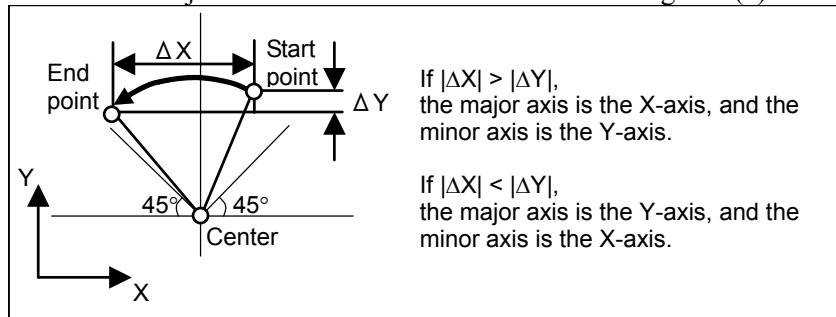


Fig. 3.5 (c)

When diameter programming is used, the relationship between the major axis and minor axis is judged with the radius value.

- Permissible arc range

If the arc goes beyond the range shown in Fig. 3.5 (d) or Fig. 3.5 (e), alarm PS2070 is issued.

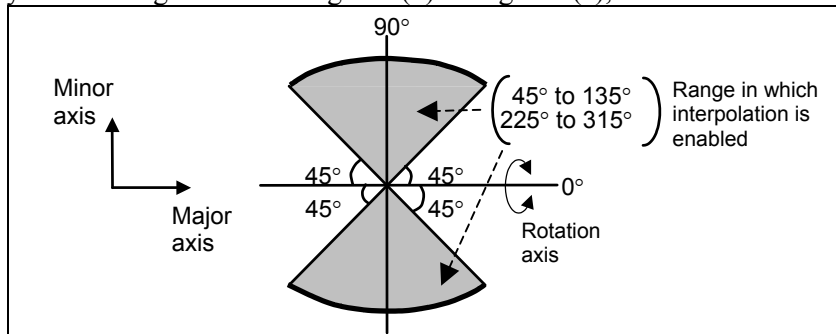


Fig. 3.5 (d)

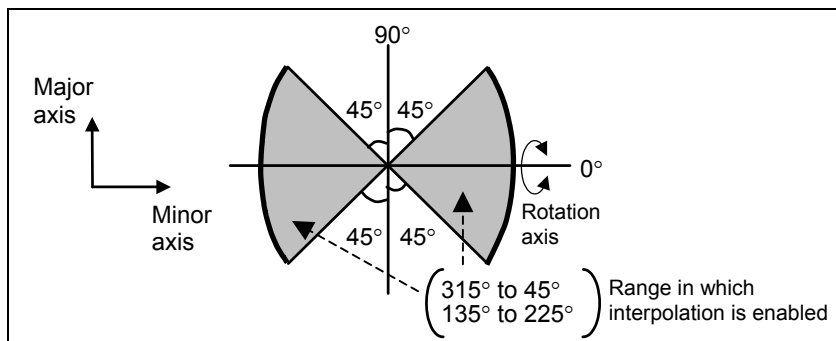


Fig. 3.5 (e)

- Feedrate

If the specified feedrate for the major axis is F , the feedrate for the minor axis F_s and feedrate for the α axis F_α are expressed as follows.

$$F_\alpha = F \times \frac{\text{Length of } \alpha \text{ axis}}{\text{Length of major axis}} \quad (\text{The maximum value is the maximum cutting feedrate for each axis.})$$

$$F_s = F \times \text{TAN } \theta$$

θ : Angle of a tangent to the major axis

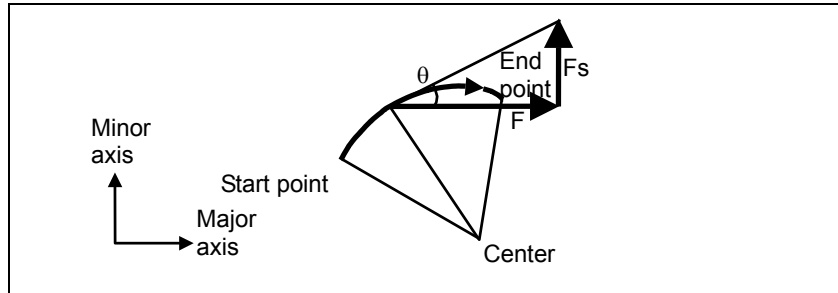


Fig. 3.5 (f)

- Tool radius compensation

Tool radius compensation applies only to two axes of the plane on which circular interpolation is performed.

Limitation

- Tool offset and tool length compensation

In a block in which circular thread cutting B is specified, tool offset or tool length compensation cannot be specified.

- I,J,K and the R command

Either I,J,K or the R command cannot be omitted.

- Unavailable functions

Circular thread cutting B cannot be used together with the following functions:

- 3-dimensional cutter compensation
- Tool center point control

3.6 GROOVE CUTTING BY CONTINUOUS CIRCLE MOTION (G12.4, G13.4)

Overview

Groove cutting with a width greater than the tool diameter can be performed by causing the tool to make continuous circle motion independently of axis movement by the groove cutting path program and superposing the continuous circle motion on the axis movement by the groove cutting path program.

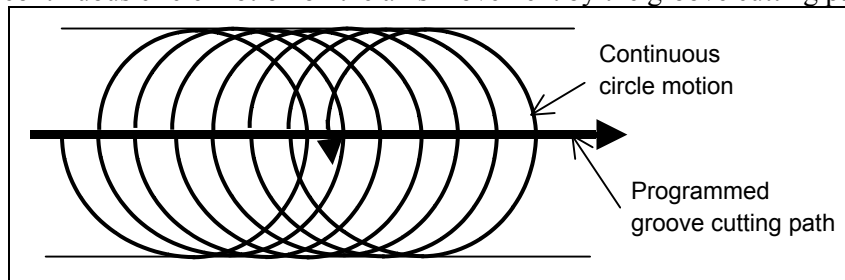


Fig. 3.6 (a)

Format

G12.4	}	P1<i>l</i> <i>K</i> <i>Q</i> <i>F</i>; (Mode on)
G13.4		
:	}	(Groove cutting path program)
:		
:		
G12.4	}	P0; (Mode cancel)
G13.4		

G12.4 : Clockwise continuous circle motion

G13.4 : Counterclockwise continuous circle motion

i : Groove width

k : Tool diameter

q : Travel distance in the groove cutting direction per continuous circle motion (pitch)

f : Feedrate (speed of the center of a tool that performs continuous circle motion)

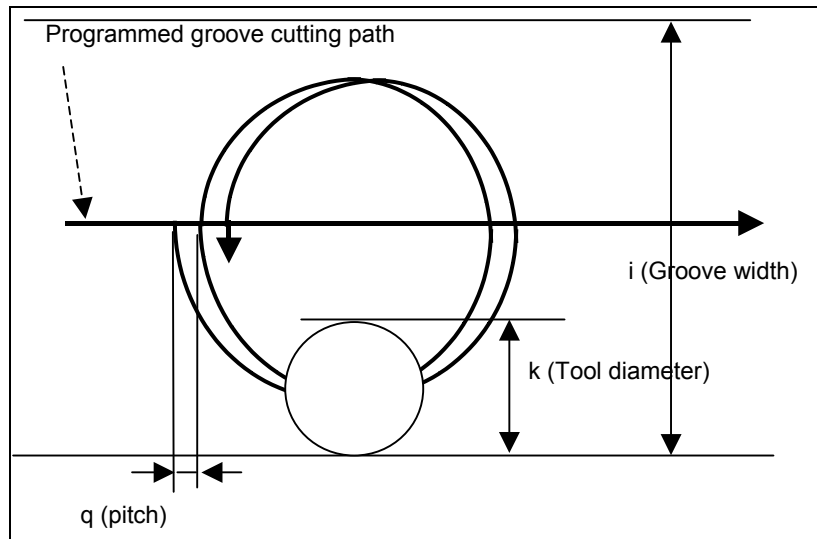


Fig. 3.6 (b)

NOTE

- 1 In the G12.4/G13.4 blocks, addresses other than the commands mentioned above cannot be used.
- 2 If bit 4 (GCC) of parameter No. 3452 is 0, continuous circle motion stops due to the stoppage of axis movement by the groove cutting path program, but because axis movement by the groove cutting path program is independent of continuous circle motion, the stoppage will not exactly be at the position of continuous circle motion converted from the groove cutting path program and the pitch.

Explanation**- Mode on**

The continuous circle motion-based groove cutting mode on command sets continuous circle motion-based groove cutting mode to the on state. The continuous circle motion-based groove cutting enable signal must be "1".

If the mode on command is specified with the continuous circle motion-based groove cutting enable signal being "0", alarm PS0010 is issued.

No axis movement is performed with the continuous circle motion-based groove cutting mode on command.

- Mode cancel

The continuous circle motion-based groove cutting mode cancel command causes the tool to move with cutting feed (continuous circle motion speed) from the present position on the continuous circle to the end-specified position (groove center) of the groove cutting path program. After the end of movement, the command sets continuous circle motion-based groove cutting mode to the cancel state.

- Startup

In the first move command block of the groove cutting path program, the tool moves to a point on the continuous circle with cutting feed (continuous circle motion speed). After the end of the movement to a point on the continuous circle, continuous circle motion is started in synchronization with the move command of the groove cutting path program.

Depending on the first move command block of the groove cutting path program, the direction of startup varies.

- In the case of an axis command perpendicular to a plane or if there is no movement along an axis used to form the currently selected plane

Assuming that $R = (I-K)/2$, the following holds true: $(X,Y) = (-R,0)$

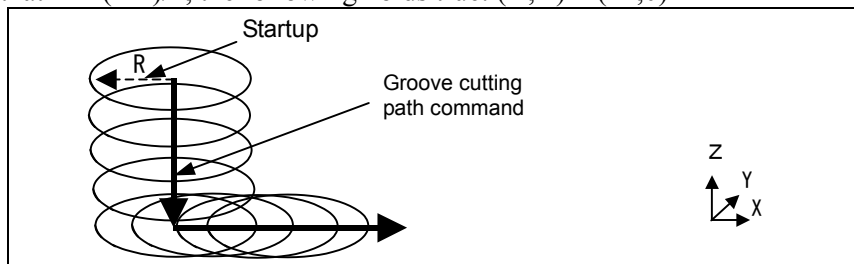


Fig. 3.6 (c)

- If there is movement along an axis used to form the selected plane
Direction opposite to the direction of movement projected onto the currently selected plane

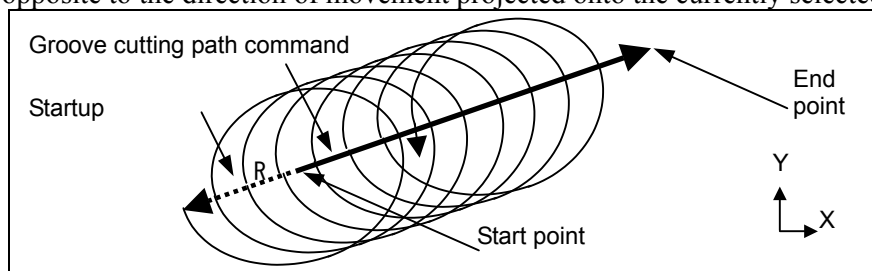


Fig. 3.6 (d)

- Increments systems for horizontal width, tool diameter, and pitch

The increment systems for I (horizontal width), K (tool diameter), and Q (pitch) follow the incremental system of the reference axis (parameter No. 1031).

- Feedrate

For the feedrate F, specify the central speed of the tool that performs continuous circle motion. The speed on the groove cutting path is:

$$\text{Speed on the groove cutting path} = F \times Q / \pi (I-K)$$

- Pitch

If the pitch is large, there may be portions left uncut. The tool diameter must be greater than the pitch.

- Groove cutting path program

The groove cutting path program specifies the path of the center of continuous circle motion.

(1) Effective commands

The groove cutting path program can execute only the G01, G02, G03, G04, G90, G91, and auxiliary functions. The G00 command causes alarm PS5256. By setting bit 0 (GG0) of parameter No. 3452, however, the G00 command can be moved as G01 movement. The modal code changes from G00 to G01.

(2) Specifying a controlled axis

The groove cutting path program can specify the control axes below.

- Commands for the axes forming a plane
- Commands for axes perpendicular to a plane (independent command for the Z-axis)
- Commands for linear axes other than the axes forming a plane
- Commands for rotation axes

(3) Specification of a pitch in a move command block

In addition to the specification of a pitch with the continuous circle motion-based groove cutting mode on command, a pitch can be specified in each move command block.

The pitch specification is modal; the pitch last specified is effective from the time the mode is entered with G12.4P1/G13.4P1 until the mode is exited.

(4) Command example

(Example 1) Example of a command for an axis perpendicular to a plane
(independent command for the Z-axis)

```

:
G12.4 P1 li Kk Ff ;
G01 Z--- Q--- ;   Specify the pitch for the Z-axis (pitch of this block).
X--- Y--- Q--- ;   Specify the pitch for the X- and Y-axes (pitch of the
                    subsequent blocks)
:

```

For a command for the Z-axis only, the pitch is regarded as the travel distance in the Z-axis direction.

(Example 2) If a move command for other than a plane is included

```

:
G12.4P1 li Kk Ff ;
G01 X--- Y--- Z--- Q--- ; This is a pitch on a slanted path.
:

```

The pitch is the travel distance in the synthetic direction of the X-, Y-, and Z-axes.

(Example 3) Pitch switching

```

:
G12.4P1 li Kk Ff ;
G01 X--- Y--- Q--- ;
X--- Y--- Q--- ;   Specify Q in the block in which the pitch is to be changed.
:

```

The pitch is the travel distance in the synthetic direction of the specified axes.

(Example 4) Command containing a rotation axis

```

:
G12.4P1 li Kk Ff ;
G01 A--- Q--- ;
X--- Y--- ;   Specify Q in the block in which the pitch is to be changed.
:

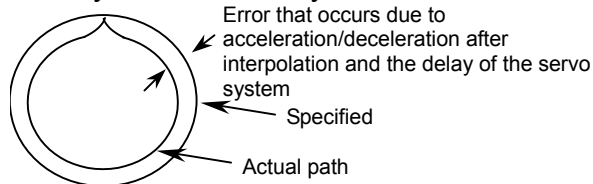
```

The increment system for Q follows that of the reference axis. Thus, if the increment system for the reference axis is IS-B, the pitch Q100 will be 0.1°.

So that a cut along the Z-axis will start after the stabilization of the radius of continuous circle motion, the continuous circle motion-based groove cutting mode on command must be specified in the air, apart from the cut position by at least the pitch per rotation. Cutting with a constant groove width is possible by letting the tool escape in the Z-axis direction at the end of cutting while keeping continuous circle motion and then after the tool comes out into the air, executing the continuous circle motion-based groove cutting mode cancel command to stop continuous circle motion.

NOTE

The radius of continuous circle motion is smaller than that specified at the start of continuous circle motion, and is larger when continuous circle motion comes to a deceleration stop. In the steady state, the radius is smaller than that specified. This is an error that occurs due to acceleration/deceleration after interpolation and the delay of the servo system.



- Mode in progress signal

This signal notifies the PMC that continuous circle motion-based groove cutting mode is in progress. The signal is set to "1" if continuous circle motion-based groove cutting mode is turned on. The signal is set to "0" if continuous circle motion-based groove cutting mode is canceled.

- Clamping the feedrate with the acceleration of continuous circle motion

Using the I and K commands in G12.4/13.4 and the acceleration clamp value for continuous circle motion (parameter No. 3490), the feedrate command F for continuous circle motion can be clamped.

$$\text{Clamp feedrate } F = \text{SQR}(\text{parameter No.3490} \times (I-K) / 2) \times 60$$

Example If parameter No. 3490 = 100

If G13.4 P1 I10.0 K5.0 Q1.0 F1000 ;, clamp feedrate = 948. Thus, execution with an F value of 948

If G13.4 P1 I10.0 K8.0 Q1.0 F1000 ;, clamp feedrate = 600. Thus, execution with an F value of 600

If G13.4 P1 I10.0 K9.0 Q1.0 F1000 ;, clamp feedrate = 424. Thus, execution with an F value of 424

A continuous circle motion feedrate override is applied to the clamped feedrate.

- Acceleration deceleration after interpolation

In continuous circle motion-based groove cutting mode, acceleration/deceleration after interpolation is enabled.

- Selecting stoppage conditions

(1) Stoppage of continuous circle motion due to a feed hold, single block, etc.

It can be selected whether to continue or stop continuous circle motion when a specified operation is stopped due to one of the various stoppage conditions such as a feed hold and a single block, using a parameter:

Bit 4 (GCC) of parameter No. 3452 = 0 : Stops continuous circle motion.

Bit 4 (GCC) of parameter No. 3452 = 1 : Continues continuous circle motion.

(2) Stoppage condition/mode switching

The conditions for groove cutting path operation, continuous circle motion stoppage/continuation, and switching to operation mode in connection with stoppage conditions are as given in Table 3.6 (a).

Table 3.6 (a)

Stoppage condition	Groove cutting path operation	Continuous circle motion stoppage/continuation		Switching to operation mode
		Stoppage (GCC = 0)	Continuation (GCC = 1)	
Feed hold	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
Single block	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
Switching to manual mode	Deceleration stop	Deceleration stop	Continuation	To be performed after the stoppage of path operation
Switching between auto modes	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
MDI operation	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
Pitch override of 0%	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
M/S/T code FIN awaiting	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
Programming error	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
Overheat alarm	Deceleration stop	Deceleration stop	Continuation	Possible after the stoppage of path operation
BG edit alarm	Continuation	Continuation	Continuation	Automatic operation does not stop.
Mode end	Deceleration stop	Withdrawal by the radius/ deceleration stop	Withdrawal by the radius/ deceleration stop	Follows other stoppage conditions.
Reset	Deceleration stop	Deceleration stop	Deceleration stop	Possible after the end of all operations
Machine lock *1	Deceleration stop	Deceleration stop	Deceleration stop	————
Servo off *1	Deceleration stop	Deceleration stop	Deceleration stop	————
Interlock *2	Deceleration stop	Deceleration stop	Deceleration stop	————
OT alarm	Deceleration stop	Deceleration stop	Deceleration stop	Possible after the end of all operations
DS alarm	Deceleration stop	Deceleration stop	Deceleration stop	Possible after the end of all operations
Emergency stop	Immediate stop	Immediate stop	Immediate stop	Possible after an emergency stop is canceled
PC-related alarm	Immediate stop	Immediate stop	Immediate stop	Possible after an alarm is canceled
Servo alarm	Immediate stop	Immediate stop	Immediate stop	Possible after an alarm is canceled
Spindle alarm	Immediate stop	Immediate stop	Immediate stop	Possible after an alarm is canceled
System alarm	Immediate stop	Immediate stop	Immediate stop	The power must be turned off and then back on.

*1: Stoppage occurs only on the axis on which the function is enabled.

*2: Stoppage occurs on all axes if the function is enabled on at least one axis.

- If bit 4 (GCC) of parameter No. 3452 is 0, continuous circle motion stops due to the stoppage of axis movement by the groove cutting path program, but because axis movement by the groove cutting path program is independent of continuous circle motion, the stoppage will not exactly be at the position of continuous circle motion converted from the groove cutting path program and the pitch.

- After the mode is switched to manual mode, the axes on which manual movement is possible do not include the axes on which to perform continuous circle motion.
- If continuous circle motion is to continue (bit 4 (GCC) of parameter No. 3452 = 1), there will be no fluctuations in radius because continuous circle motion continues.
- Movement to the continuous circle motion start position and movement to the groove cutting path program end position after the end of continuous circle motion will stop under the stoppage conditions below.

Reset	Deceleration stop
Machine lock	Deceleration stop
Servo off	Deceleration stop
Interlock	Deceleration stop
OT alarm	Deceleration stop
DS alarm	Deceleration stop
Emergency stop	Immediate stop
PC-related alarm	Immediate stop
Servo alarm	Immediate stop
Spindle alarm	Immediate stop
System alarm	Immediate stop

- Feedrate specification and pitch override

(1) Feedrate specification

For the feedrate during cutting, specify the feedrate for continuous circle motion.

This feedrate is displayed as a specified feedrate.

(2) Continuous circle motion feedrate override

The feedrate override signals (*FV0 to *FV7) are not effective to the specified feedrate for continuous circle motion. Use the continuous circle motion feedrate override signals (*CGROV7 to *CBROV0). If the continuous circle motion feedrate override is at 0%, continuous circle motion can be stopped independently of the movement on the groove cutting path.

(3) Feedrate override

The feedrate on the groove cutting path is determined with the pitch command. The feedrate override signals (*FV0 to *FV7) are effective to the feedrate on the groove cutting path. This makes it possible to change the feedrate for continuous circle motion independently of the feedrate on the groove cutting path.

Also, the override cancel signal (OVC) and the second feedrate override signal (optional function) are effective to the feedrate on the groove cutting path.

(4) Dry run

The feedrate during a dry run is as given in Table 3.6 (b).

A dry run is not effective to the feedrate on the groove cutting path.

Table 3.6 (b)

	Feedrate	Override	
Feedrate on the groove cutting path	Feedrate F for continuous circle motion × pitch / 2πR	Feedrate override	
Feedrate for continuous circle motion	Dry run rate (Parameter No.1410)	Manual rapid traverse selection signal (RT)	
		0	1
		JV	JVmax

JV : Manual feedrate override

Jvmax : Maximum manual feedrate override

(5) Feedrate display

- Specified feedrate display shows the specified speed for continuous circle motion.
- Actual cutting feedrate display show the synthetic one from the feedrate for continuous circle motion and the feedrate on the groove cutting path.

(6) Clamping with the maximum cutting feedrate

If the specification of the feedrate for continuous circle motion exceeds the maximum cutting feedrate (parameter No. 1430), the specification of the feedrate for continuous circle motion is clamped to calculate the feedrate on the groove cutting path. In addition, the feedrate for continuous circle motion and the feedrate on the groove cutting path, to which their respective overrides have been applied, are clamped with the maximum cutting feedrate.

Limitation

- Mirror image

A mirror image is effective to groove cutting path commands only.

No mirror image is applied to continuous circle motion, movement to the continuous circle motion start position, and movement to the groove cutting path program end position after the end of continuous circle motion.

- Distance to go

Movement to the continuous circle motion start position, movement of continuous circle motion, and movement to the groove cutting path program end position after the end of continuous circle motion are not reflected in the distance to go (on the position screen, etc.).

- Workpiece coordinate system and machine coordinate system

Movement to the continuous circle motion start position, movement of continuous circle motion, and movement to the groove cutting path program end position after the end of continuous circle motion are not reflected in the workpiece coordinate system. The workpiece coordinate system is the coordinate system of the groove cutting path program.

Note, however, that these movements are reflected in the machine coordinate system.

- Axis moving signal

The axis moving signal does not change due to axis moving due to continuous circle motion.

Use the continuous circle motion-based groove cutting mode in progress signal.

- Graphic display

The tool path of the program during cutting is in the workpiece coordinate system and, therefore, the groove cutting command path is drawn.

- Restart functions

The restart functions, such as program restart and tool retract and recover, cannot be used to start continuous circle motion-based groove cutting in the middle of the groove cutting path program.

- Retrace

Retrace cannot be used in continuous circle motion-based groove cutting mode.

- AI contour control

In continuous circle motion-based groove cutting mode, AI contour control is disabled. If continuous circle motion-based groove cutting is specified during AI contour control, AI contour control is temporarily canceled. If continuous circle motion-based groove cutting mode is canceled, AI contour control returns to on. Note that if AI contour control is disabled, acceleration/deceleration before look-ahead interpolation, acceleration control, and optimum torque acceleration/deceleration are also disabled.

- One-digit F code

The one-digit F code cannot be used in continuous circle motion-based groove cutting mode.

- Interruption type custom macro

The interruption type custom macro cannot be used in continuous circle motion-based groove cutting mode.

- Commands that can be executed in continuous circle motion-based groove cutting mode

The groove cutting path program can be executed with the following commands only:

- G01
- G02, G03 (except helical interpolation and helical interpolation B)
- G04
- G90, G91
- M/S/T (auxiliary function), Second auxiliary function
- G00 (An operation can be selected with bit 0 (GG0) of parameter No. 3452.
GG0 = 0: Alarm PS5256 is issued.
GG0 = 1: The G00 command is moved as G01 operation.)
- G94 (Feed per minute)

- Modes in which the continuous circle motion-based groove cutting mode command cannot be specified

The continuous circle motion-based groove cutting command cannot be executed in the function modes below.

Interpolation function

- Helical interpolation
- Helical interpolation B
- Conical interpolation / spiral interpolation
- Polar coordinate interpolation
- Cylindrical interpolation / Cutting point interpolation for cylindrical interpolation
- Exponential interpolation
- Smooth interpolation
- Nano smoothing
- NURBS interpolation
- Hypothetical axis interpolation
- Variable lead thread cutting
- Circular thread cutting
- 3-dimensional circular interpolation
- Involute interpolation
- Thread cutting

Feed function

- One-digit F code feed / feed per revolution / inverse time feed
- Exact stop / Tapping mode / automatic corner override

Coordinate value and dimension

- Polar coordinate command

Function to simplify programming

- Figure copying
- 3-dimensional coordinate system conversion
- Canned cycle for drilling
- Rigid tapping
- Index table indexing

Compensation function

- Scaling
- Programmable mirror image
- Tool offset
- Tool radius compensation
- Tool nose radius compensation / vector retention / corner circular interpolation
- 3-dimensional tool compensation
- Coordinate system rotation

Axis control function

- Polygon turning
- Arbitrary angular axis control

5-axis machining function

- Tool center point control for 5-axis machining
- Tilted working plane indexing
- Inclined Rotary Axis Control
- 3-dimensional cutter compensation

Multi-path control function

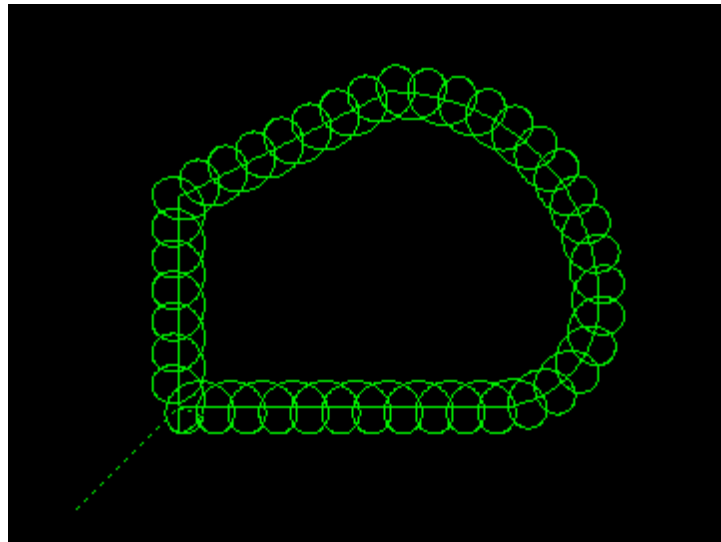
- Inter-path waiting / Path spindle control
- Synchronous/Composite/Superimposed control

Example

If the following program is executed, the center of the tool moves as shown in the figure below.

(This program is merely a sample. The Q and F commands must be determined according to the cutting conditions.)

```
O0002 ;  
N01 G90 G0 X0 Y0 Z0 ;  
N02 G91 G00 X20.0 Y20.0 ;  
N03 G01 Z-25.0 F5000 ;  
N04 G13.4 P1 I20.0 K10.0 Q5.0 F3000 ;  
N05 Y40.0 ;  
N06 X40.0 Y20.0 ;  
N07 G02 X40.0 Y-40.0 R40.0 ;  
N08 X-20.0 Y-20.0 R20.0 ;  
N09 G01 X-60. ;  
N10 G13.4 P0 ;  
N11 G00 Z25.0 ;  
N12 X-20.0 Y-20.0 M02 ;  
%
```



4 COORDINATE VALUE AND DIMENSION

Chapter 4, "COORDINATE VALUE AND DIMENSION", consists of the following sections:

4.1 POLAR COORDINATE COMMAND (G15, G16)41

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

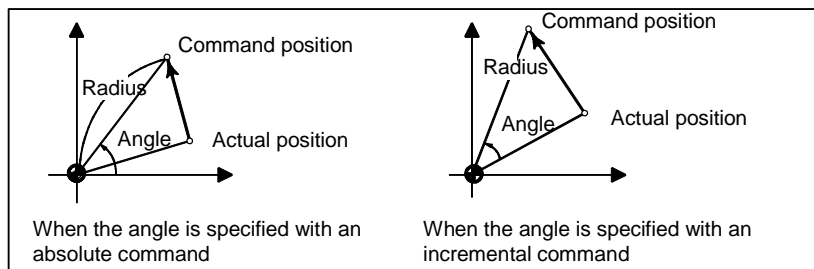
G16 : Polar coordinate command
 G15 : Polar coordinate command cancel
 Gxx : Plane selection of the polar coordinate command (G17, G18 or G19)
 Gyy : Center selection of the polar coordinate command (G90 or G91)
 G90 specifies the origin of the workpiece coordinate system as the origin of the polar coordinate 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 workpiece coordinate system as the origin of the polar coordinate system

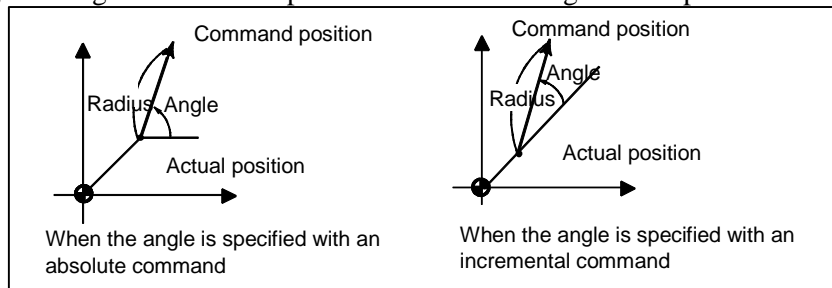
Specify the radius (the distance between the origin and the point) to be programmed with an absolute programming. The origin of the workpiece coordinate system is set as the origin of the polar coordinate system.

When a local coordinate system (G52) is used, the origin of the local coordinate system becomes the center of the polar coordinates.



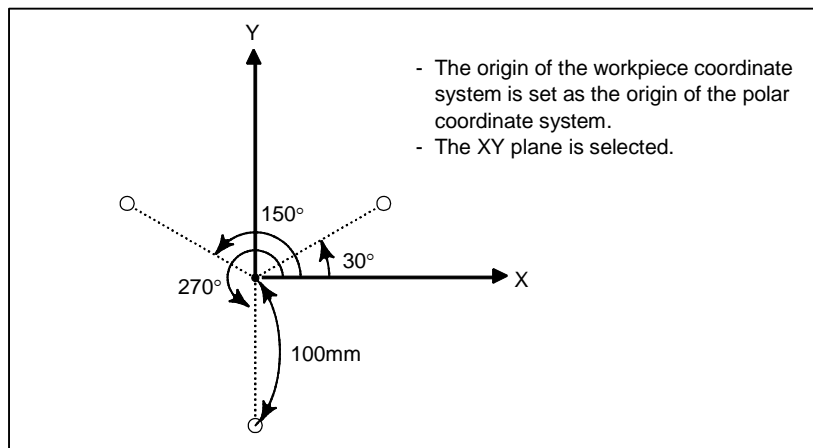
- Setting the current position as the origin of the polar coordinate system

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.



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 workpiece coordinate system as the origin of the polar coordinate system
- N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;
Specifying a distance of 100 mm and an angle of 30 degrees
- N3 Y150.0 ; Specifying a distance of 100 mm and an angle of 150 degrees
- N4 Y270.0 ; Specifying a distance of 100 mm and an angle of 270 degrees
- N5 G15 G80 ; Canceling the polar coordinate command

- Specifying angles with incremental 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 workpiece coordinate system as the origin of the polar coordinate system
- N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;
Specifying a distance of 100 mm and an angle of 30 degrees
- N3 G91 Y120.0 ; Specifying a distance of 100 mm and an angle of +120 degrees
- N4 Y120.0 ; Specifying a distance of 100 mm and an angle of +120 degrees
- N5 G15 G80 ; Canceling the polar coordinate command

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:

- 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)

- **Optional chamfering and corner R**

Optional chamfering and corner R cannot be specified in polar coordinate mode.

5 FUNCTIONS TO SIMPLIFY PROGRAMMING

Chapter 5, "FUNCTIONS TO SIMPLIFY PROGRAMMING", consists of the following sections:

5.1	CANNED CYCLE FOR DRILLING	44
5.2	IN-POSITION CHECK SWITCHING FOR DRILLING CANNED CYCLE	74
5.3	RIGID TAPPING.....	88
5.4	OPTIONAL CHAMFERING AND CORNER R.....	102
5.5	INDEX TABLE INDEXING FUNCTION.....	105
5.6	IN-FEED CONTROL (FOR GRINDING MACHINE).....	107
5.7	CANNED GRINDING CYCLE (FOR GRINDING MACHINE).....	109
5.8	MULTIPLE REPETITIVE CYCLE (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7,G76.7).....	123

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

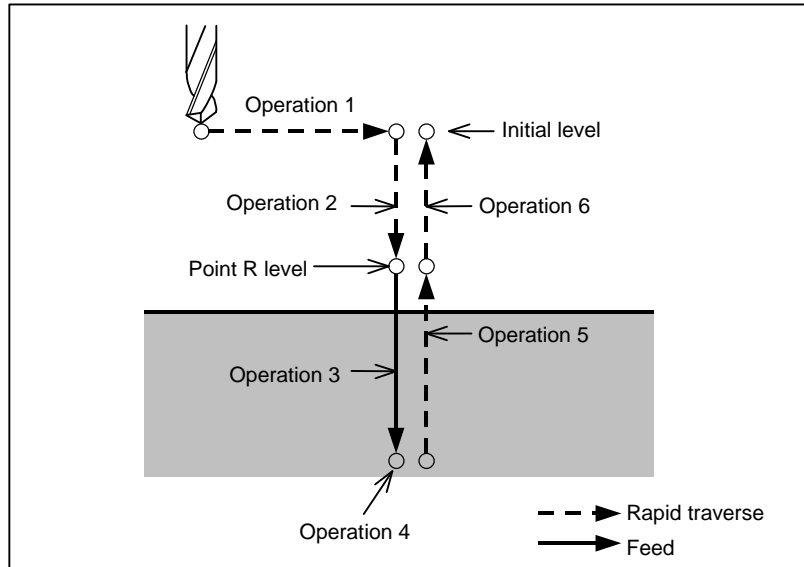


Fig. 5.1 (a) Operation sequence of canned cycle for drilling

- 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):

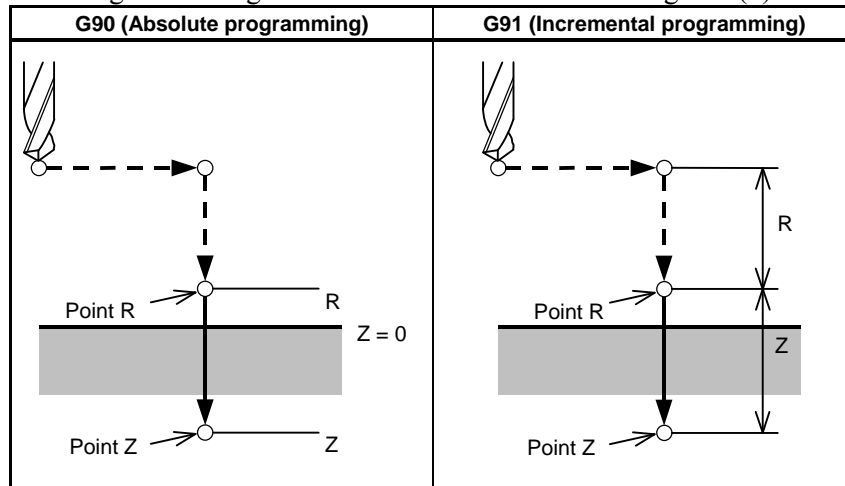


Fig. 5.1 (b) Absolute programming and incremental programming

- 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.

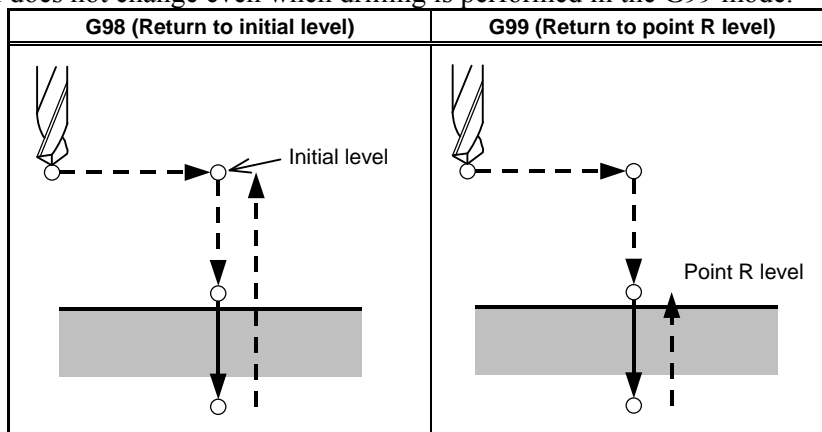


Fig. 5.1 (c) Initial level and point R level

- 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
	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_ F_ K_ ;

X_ Y_ : Hole position data

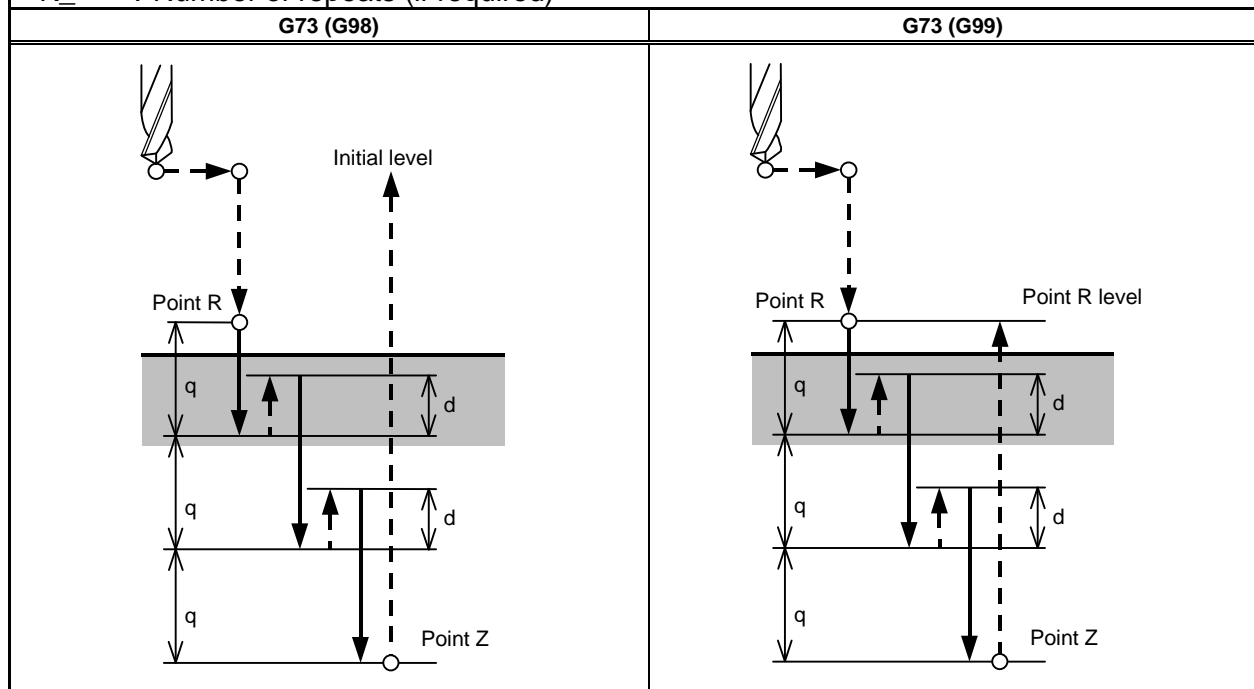
Z_ : The distance from point R to the bottom of the hole

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

Q_ : Depth of cut for each cutting feed

F_ : Cutting feedrate

K_ : Number of repeats (if required)



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 parameter 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. Y-250. Z- 150. R-100. Q15. F120. ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the 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

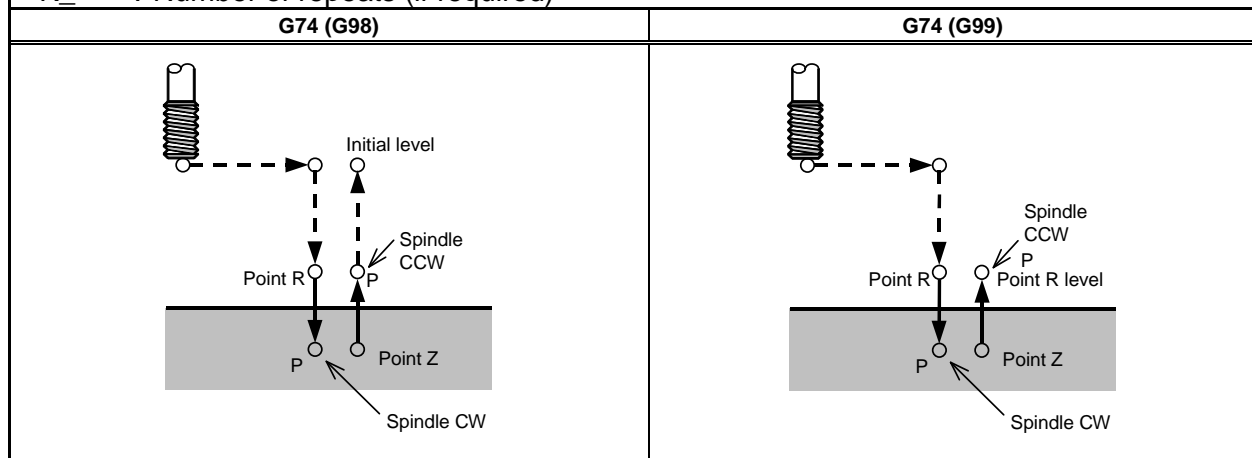
Z_ : The distance from point R to the bottom of the hole

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

P_ : Dwell time

F_ : Cutting feedrate

K_ : Number of repeats (if required)



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. Y-250. Z-150. R-120. F120. ;	Position, tapping hole 1, then return to point R.
Y-550. ;	Position, tapping hole 2, then return to point R.
Y-750. ;	Position, tapping hole 3, then return to point R.
X1000. ;	Position, tapping hole 4, then return to point R.
Y-550. ;	Position, tapping hole 5, then return to point R.
G98 Y-750. ;	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

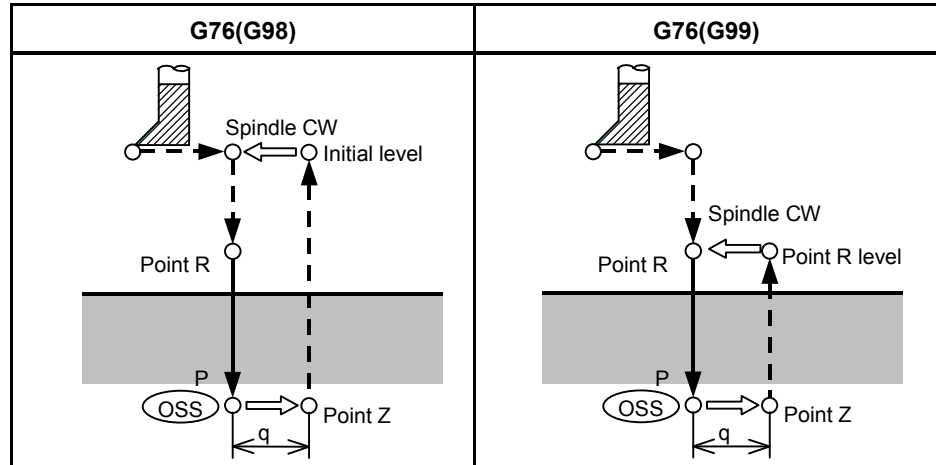
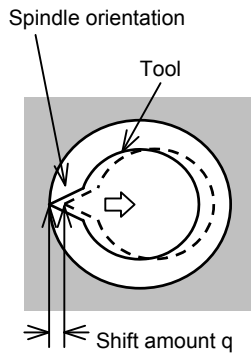
R_ : The distance from the initial level to point R level

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

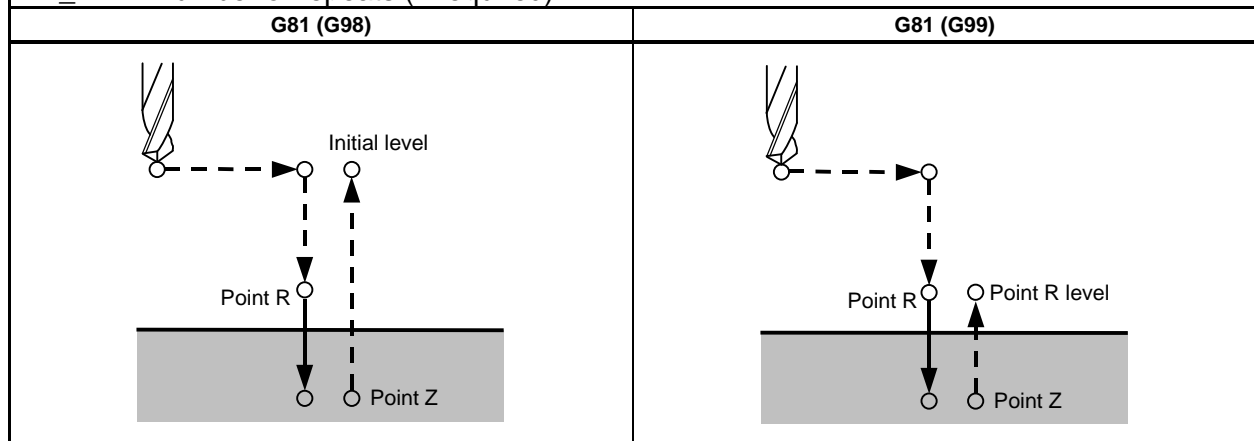
X_ Y_ : Hole position data

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)



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. Y-250. Z-150. R-100. F120. ;	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
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

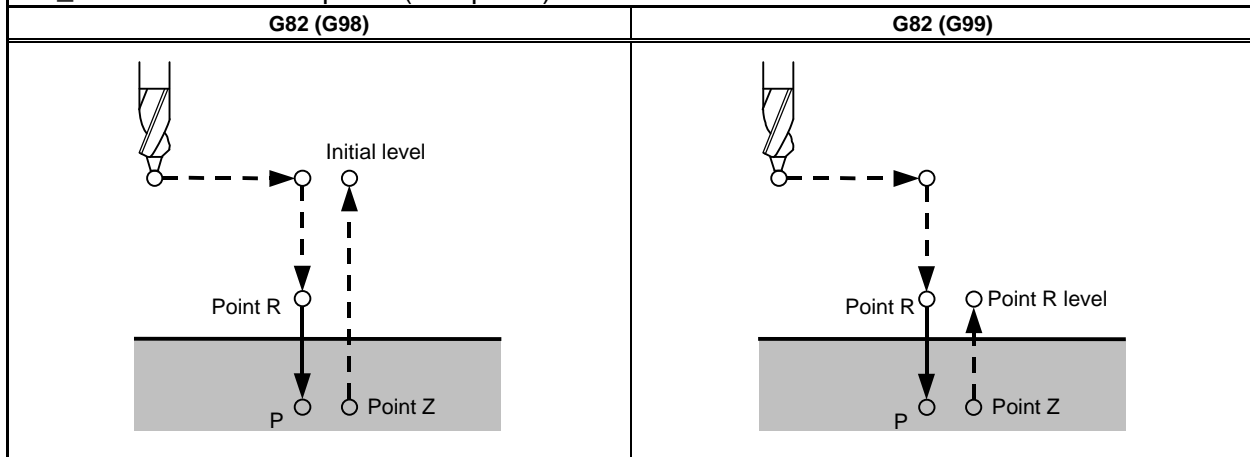
Z_ : The distance from point R to the bottom of the hole

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)

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

5.1.6 Peck Drilling Cycle (G83)

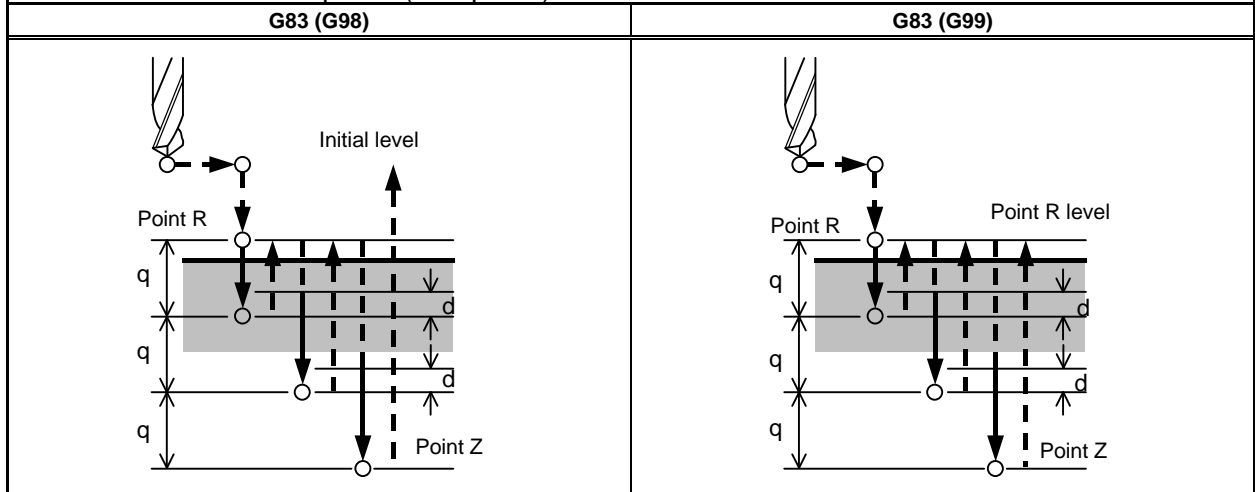
This cycle performs peck drilling.

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

Format

G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

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



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 d point just before where the last drilling ended, and cutting feed is performed again. d is set in 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. Y-250. Z-150. R-100. Q100000000000000005. F120. ;	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
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 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_ 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

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)
<p>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 (parameter 5174)</p> <p>q: Depth of each cut</p> <p>— → Path along which the tool travels at the rapid traverse rate</p> <p>→ Path along which the tool travels at the programmed cutting feedrate</p> <p>→ Path along which the tool travels at the forward or backward rate during the cycle specified with parameters (.....→)</p>	

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)
- Retraction (bottom of hole + Δ → to point R, absolute)
- Forwarding (point R → to point with hole bottom + 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 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-diameter 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.

$\text{Cutting feedrate} = F \times \alpha$

<First drilling> $\alpha=1.0$

<Second or subsequent drilling>

$\alpha=\alpha \times \beta \div 100$, where β is the rate of change for each drilling operation

When the skip signal is detected during the previous drilling operation: $\beta=b1\%$ (parameter No.5166)

When the skip signal is not detected during the previous drilling operation: $\beta=b2\%$ (parameter No.5167)

If the rate of change in cutting feedrate becomes smaller than the rate specified in parameter 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 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.

$\text{Spindle speed} = S \times \gamma$
--

<First drilling> $\gamma=1.0$

<Second or subsequent drilling>

$\gamma=\gamma \times \delta \div 100$, where δ is the rate of change for each drilling operation

When the skip signal is detected during the previous drilling operation: $\delta=d1\%$ (parameter No.5164)

When the skip signal is not detected during the previous drilling operation: $\delta=d2\%$ (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 parameter SBC (No.5105 bit 0)

- **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.

Limitation

- **Subprogram call**

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

Example

M03 S_ ;	Cause the spindle to start rotating.
M□□ ;	Specifies the small-hole peck drilling cycle mode.
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

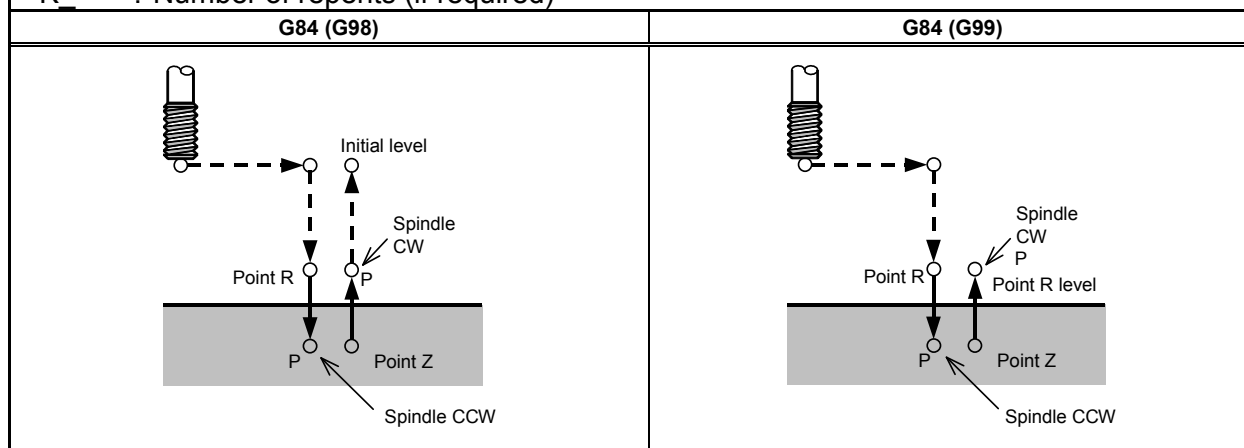
Z_ : The distance from point R to the bottom of the hole

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

P_ : Dwell time

F_ : Cutting feedrate

K_ : Number of repeats (if required)



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. Y-250. Z-150. R-120. P300 F120. ;	
Y-550. ;	Position, drill hole 1, then return to point R.
Y-750. ;	Position, drill hole 2, then return to point R.
X1000. ;	Position, drill hole 3, then return to point R.
Y-550. ;	Position, drill hole 4, then return to point R.
G98 Y-750. ;	Position, drill hole 5, then return to point R.
G80 G28 G91 X0 Y0 Z0 ;	Position, drill hole 6, then return to the initial level.
M5 ;	Return to the reference position
	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_ ;

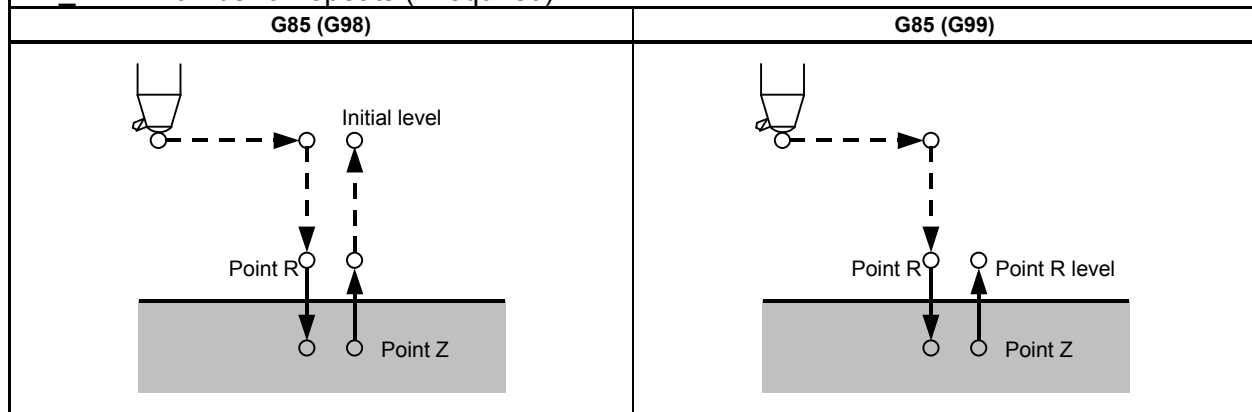
X_ Y_ : Hole position data

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)



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. Y-250. Z-150. R-120. F120. ;
                    Position, drill hole 1, then return to point R.
Y-550. ;           Position, drill hole 2, then return to point R.
Y-750. ;           Position, drill hole 3, then return to point R.
X1000. ;           Position, drill hole 4, then return to point R.
Y-550. ;           Position, drill hole 5, then return to point R.
G98 Y-750. ;       Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ; Return to the reference position
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_ ;

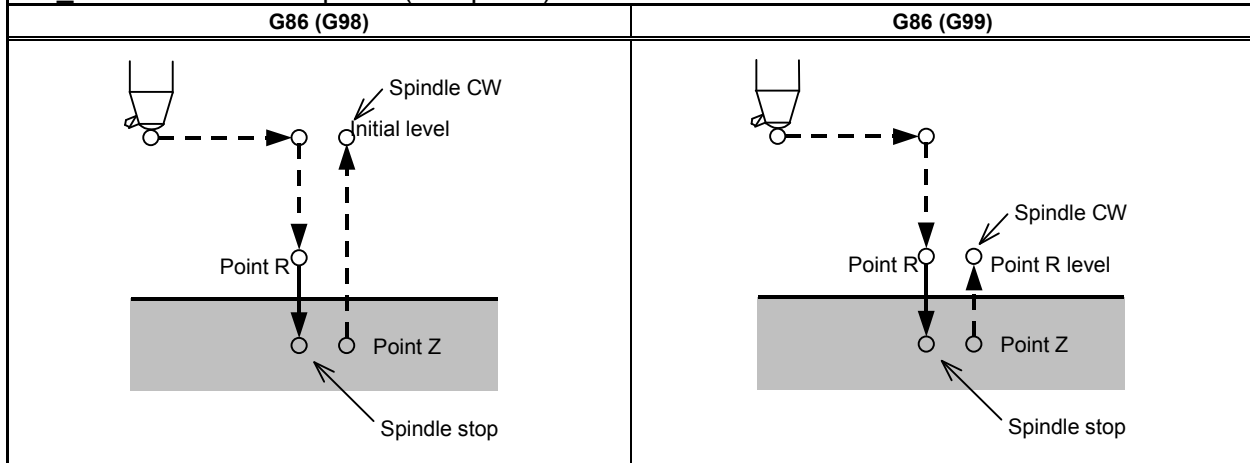
X_ Y_ : Hole position data

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)

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

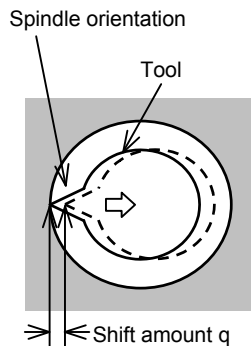
5.1.11 Back Boring Cycle (G87)

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. Y-250.	Position, bore hole 1.
Z-150. R-120. Q5.	Orient at the initial level, then shift by 5 mm.
P1000 F120. ;	Stop at point Z for 1 s.
Y-550. ;	Position, drill hole 2.
Y-750. ;	Position, drill hole 3.
X1000. ;	Position, drill hole 4.
Y-550. ;	Position, drill hole 5.
Y-750. ;	Position, drill hole 6
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
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

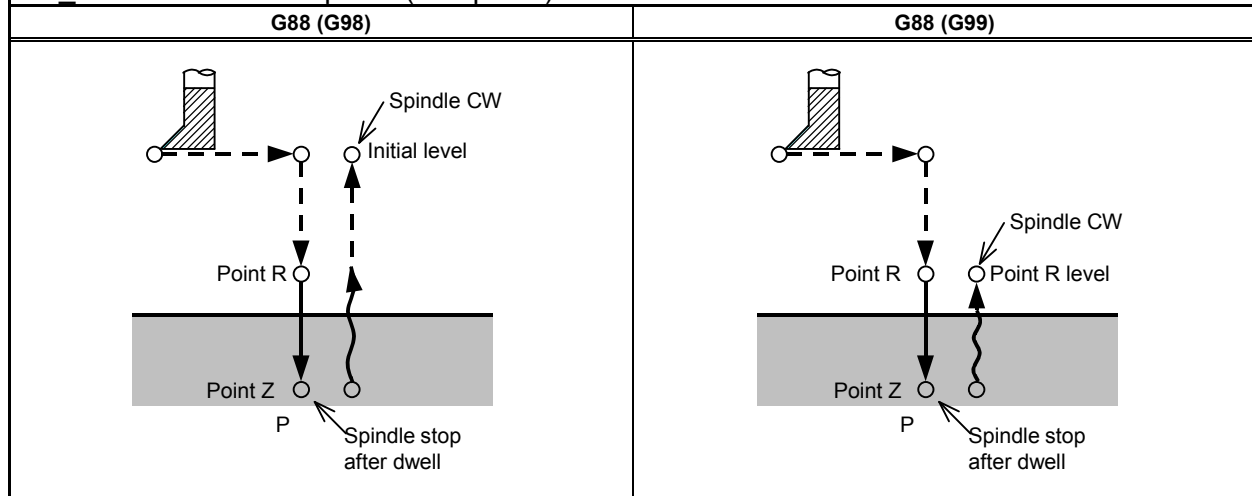
Z_ : The distance from point R to the bottom of the hole

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)



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

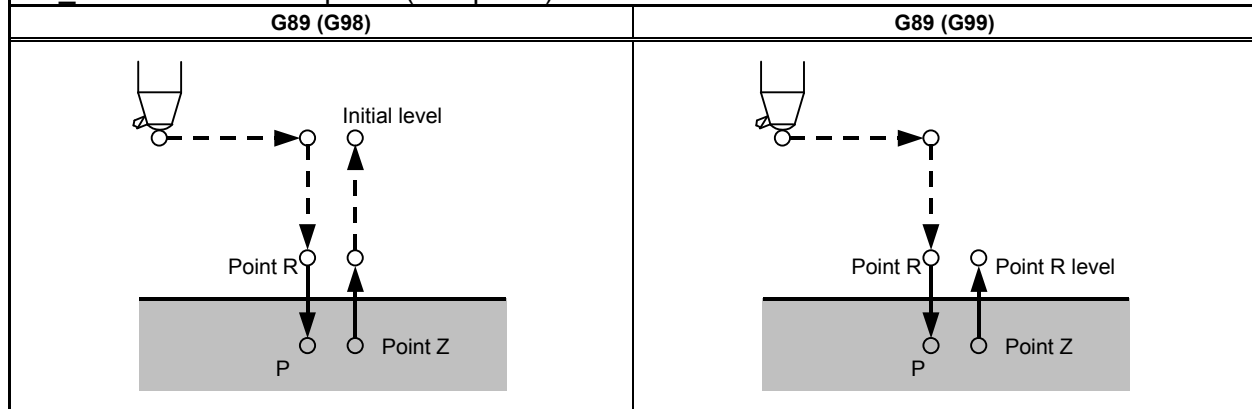
Z_ : The distance from point R to the bottom of the hole

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)



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

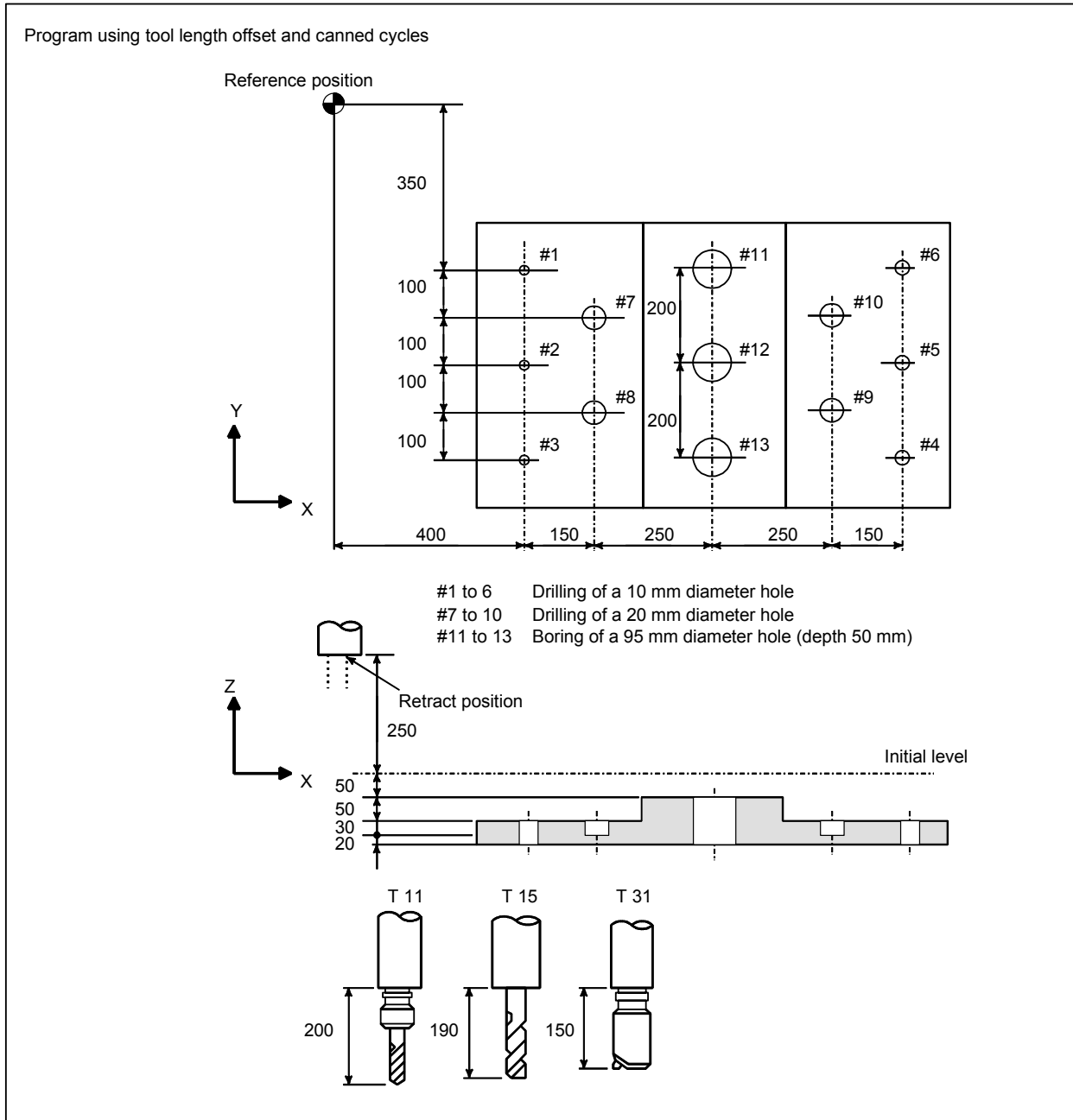


Fig. 5.1.15 (a) Example for using canned cycles for drilling

5.2 IN-POSITION CHECK SWITCHING FOR DRILLING CANNED CYCLE

Overview

This function enables dedicated in-position widths to be used for drilling canned cycle.

Up to four different dedicated in-position widths are available, one for hole bottoms and three for other than hole bottoms. Using a little large in-position width for operations at locations where no high precision is required makes drilling canned cycle faster.

Explanation

Setting bit 4 (ICS) of parameter No. 5107 to 1 enables dedicated in-position widths to be used for drilling canned cycle. Up to four different dedicated in-position widths are available, one for hole bottoms and three for other than hole bottoms.

In conventional drilling canned cycle, the same operation is performed for both in-position checks between cycles for locations where no very high precision is required (A in Fig. 5.1.15 (a)) and in-position checks between cycles for hole bottoms where a high precision is required (B in Fig. 5.1.15 (a)) because the same in-position width is used for all cycles.

Using this function makes it possible to reduce the time needed to get in an in-position state (to reduce the necessary cycle time) by setting a small in-position width for hole bottoms so as to assure a high precision while setting a little large in-position width for other than hole bottoms.

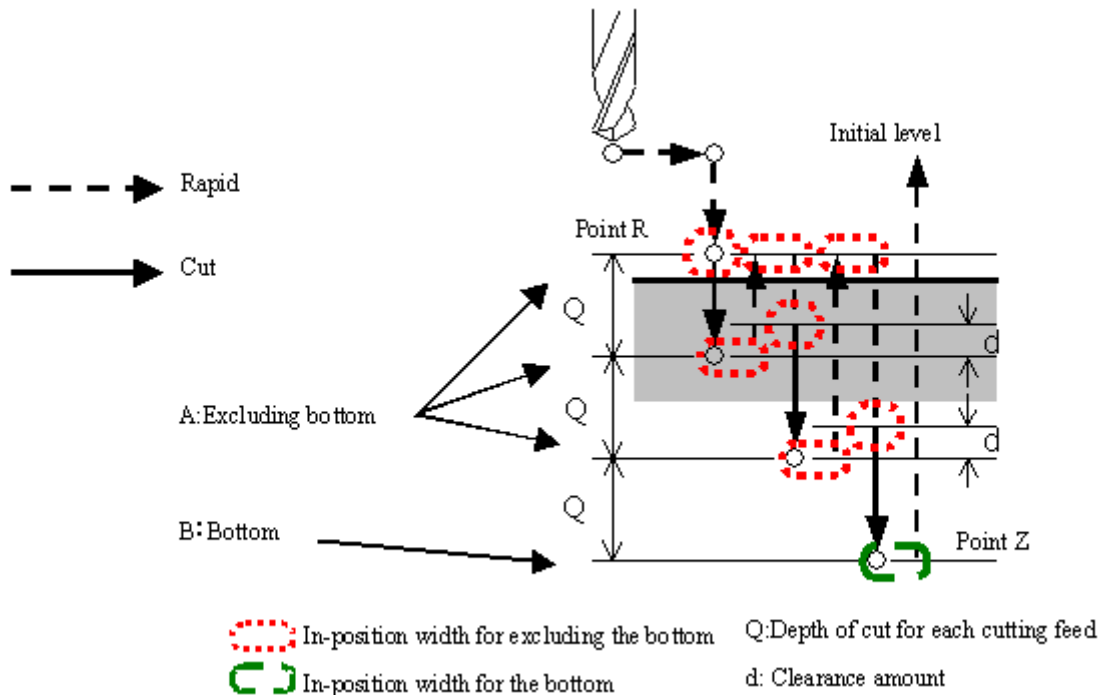






Fig. 5.1.15 (a) Example of a peck drilling cycle (G83)

- Parameters related to in-position widths

This function uses the following in-position widths.

-  In-position width for other than hole bottoms (regular) (parameter No. 5184)
-  In-position width for other than hole bottoms (for retraction in peck drilling cycle) (parameter No. 5185)
-  In-position width for other than hole bottoms (for shift in boring cycles (G76 and G87) (parameter No. 5186)
-  In-position width for hole bottoms (parameter No. 5187)

- Supported drilling canned cycle

The following table lists the drilling canned cycle for which this function is usable.

M

Table 5.1.15 (a) Drilling canned cycle for which this function is usable (machining center system)

G code	Use
G73	High-speed peck drilling cycle
G74	Left-handed tapping cycle
G76	Fine boring cycle
G81	Drilling cycle, spot drilling cycle
G82	Drilling cycle or counter boring cycle
G83	Peck drilling cycle / Small-hole peck drilling cycle
G84	Tapping cycle

G code	Use
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle

- High-speed peck drilling cycle (G73)




M

Shown below are the points where a dedicated effective area (for in-position check) is applied in high-speed peck drilling cycle.


G73 X_ Y_ Z_ R_ Q_ F_ K_ ;
 X_ Y_ : Hole position data
 Z_ : The distance from point R to the bottom of the hole
 R_ : The distance from the initial level to point R level
 Q_ : Depth of cut for each cutting feed
 F_ : Cutting feedrate
 K_ : Number of repeats (When it is needed)

G73(G98)	G73(G99)

d: Retraction distance specified in parameter No. 5114

-  In-position width for other than hole bottoms (regular)
-  In-position width for other than hole bottoms (for retraction in peck drilling cycle)
-  In-position width for hole bottoms

NOTE

When setting an effective area (for in-position check) enclosed in , pay attention to the retraction distance d (parameter No.5114). If the effective area is too large for the retraction distance, it is likely that no retraction may be performed.

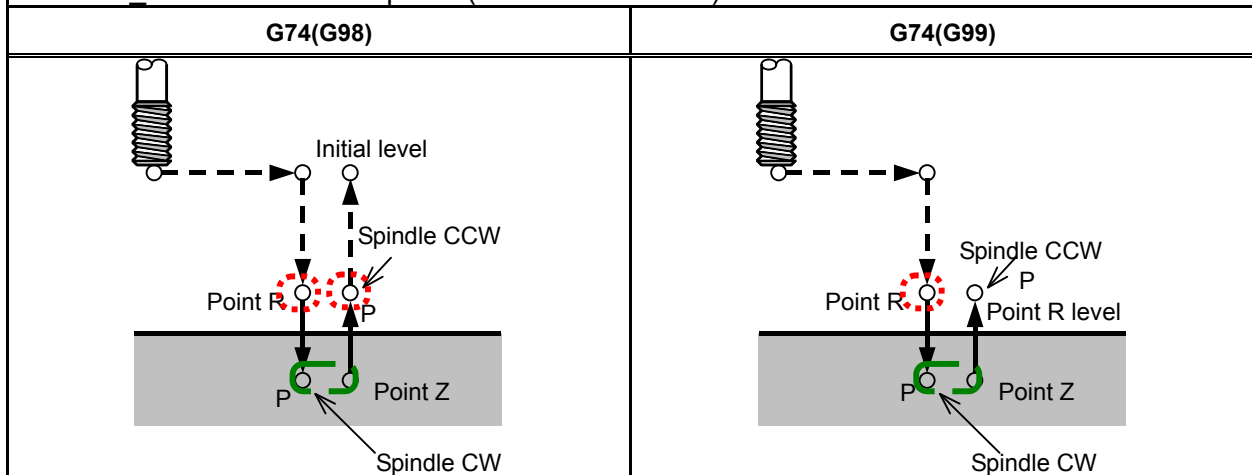
- Left-handed tapping cycle (G74)

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in left-handed tapping cycle.

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 (When it is needed)



In-position width for other than hole bottoms (regular)

In-position width for hole bottoms

CAUTION

When resuming the initial level, set an effective area (for in-position check) enclosed in , pay attention to the distance between point R and the workpiece. If the effective area is too large for the distance between point R and the workpiece, it is likely that the workpiece or tool may be damaged, because a rapid traverse may occur before the tool retracts completely from the workpiece.

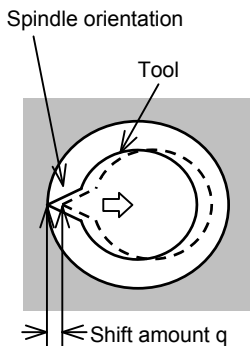
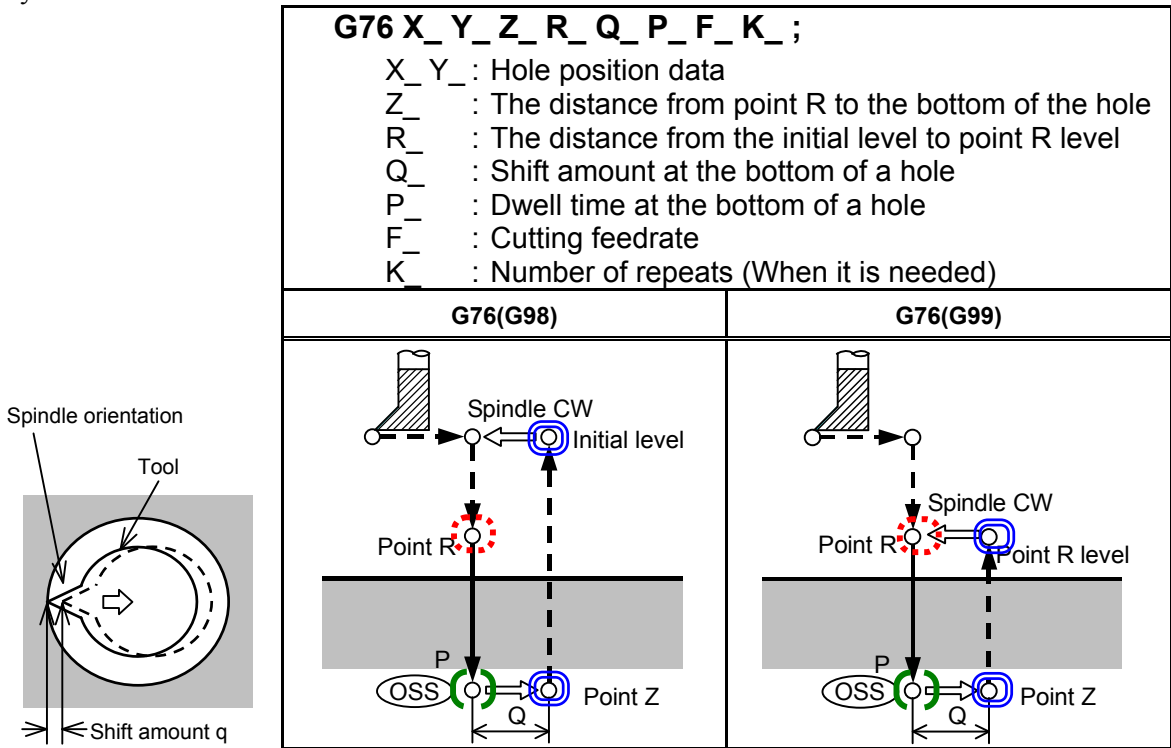
NOTE

Enabling an in-position check at hole bottoms requires setting bit 6 of parameter No. 5103 to 1.

- **Fine boring cycle (G76)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in fine boring cycle.

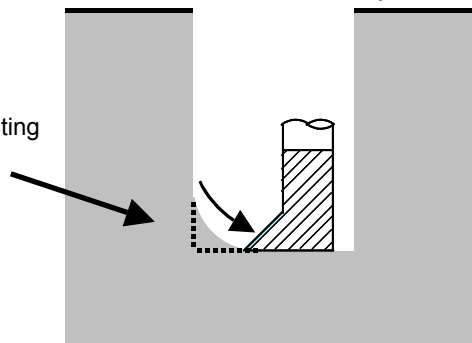


- In-position width for other than hole bottoms (regular)
- In-position width for other than hole bottoms (for shift in boring cycles (G76 and G87))
- In-position width for hole bottoms

NOTE

Exercise care in setting an effective area (for in-position check) enclosed in ○. If too large an effective area is set without dwelling, it is likely that no complete cutting is done at the hole bottom of the workpiece.

A shortcut is taken, resulting in an insufficient cutting.



- **Drilling cycle, spot drilling cycle G81)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in drilling cycle or spot drilling cycle.

<p>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 (When it is needed)</p>	
G81(G98)	G81(G99)



In-position width for other than hole bottoms (regular)

In-position width for hole bottoms

- **Drilling cycle counter boring cycle (G82)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in drilling cycle counter drilling cycle.

<p>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 feedrate K_ : Number of repeats (When it is needed)</p>	
G82(G98)	G82(G99)



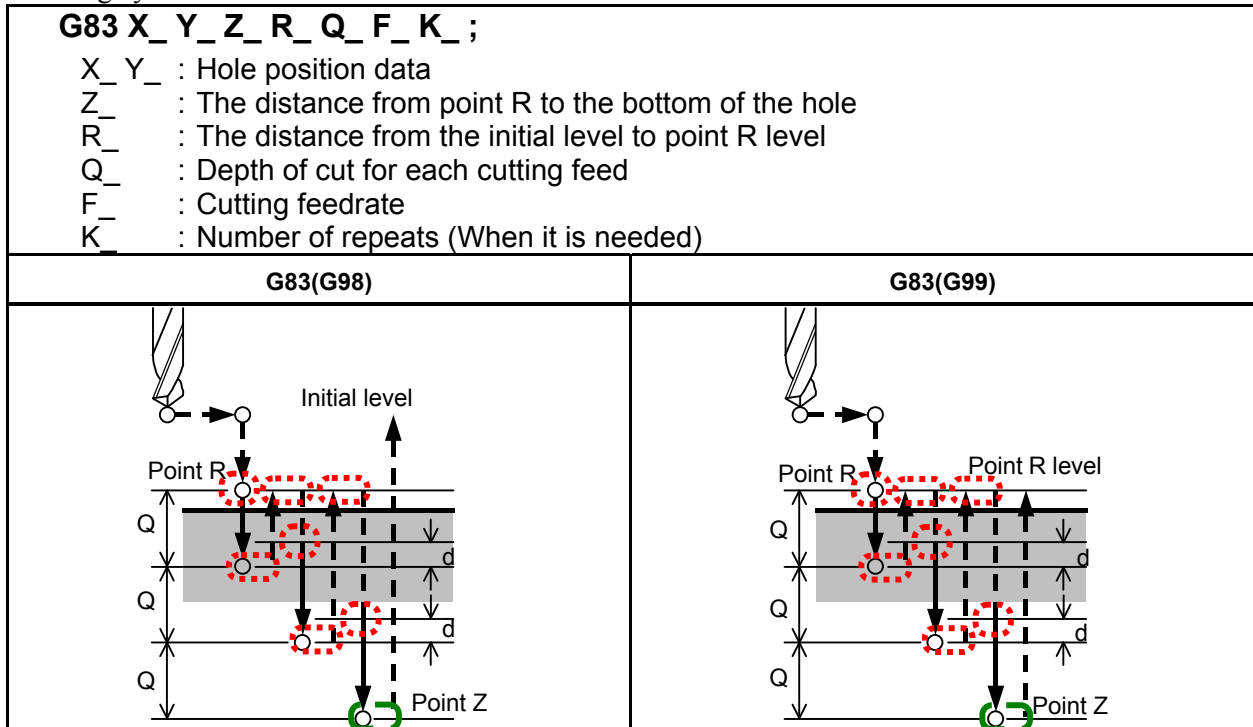
In-position width for other than hole bottoms (regular)

In-position width for hole bottoms

- **Peck drilling cycle (G83)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in peck drilling cycle.



d : Clearance specified in parameter No. 5115



In-position width for other than hole bottoms (regular)
 In-position width for hole bottoms

- **Small-hole peck drilling cycle (G83)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in small-hole peck drilling cycle.

<p>G83 X_ Y_ Z_ R_ Q_ F_ I_ K_ P_ ;</p> <p>X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level Q_ : Depth of cut for each cutting feed F_ : Cutting feedrate 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 repeats (When it is needed) P_ : Dwell time at the bottom of a hole (If this is omitted, P0 is assumed as the default.)</p>	
G83(G98)	G83(G99)
<p>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 (parameter 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 during the cycle specified with parameters (.....>)</p>	

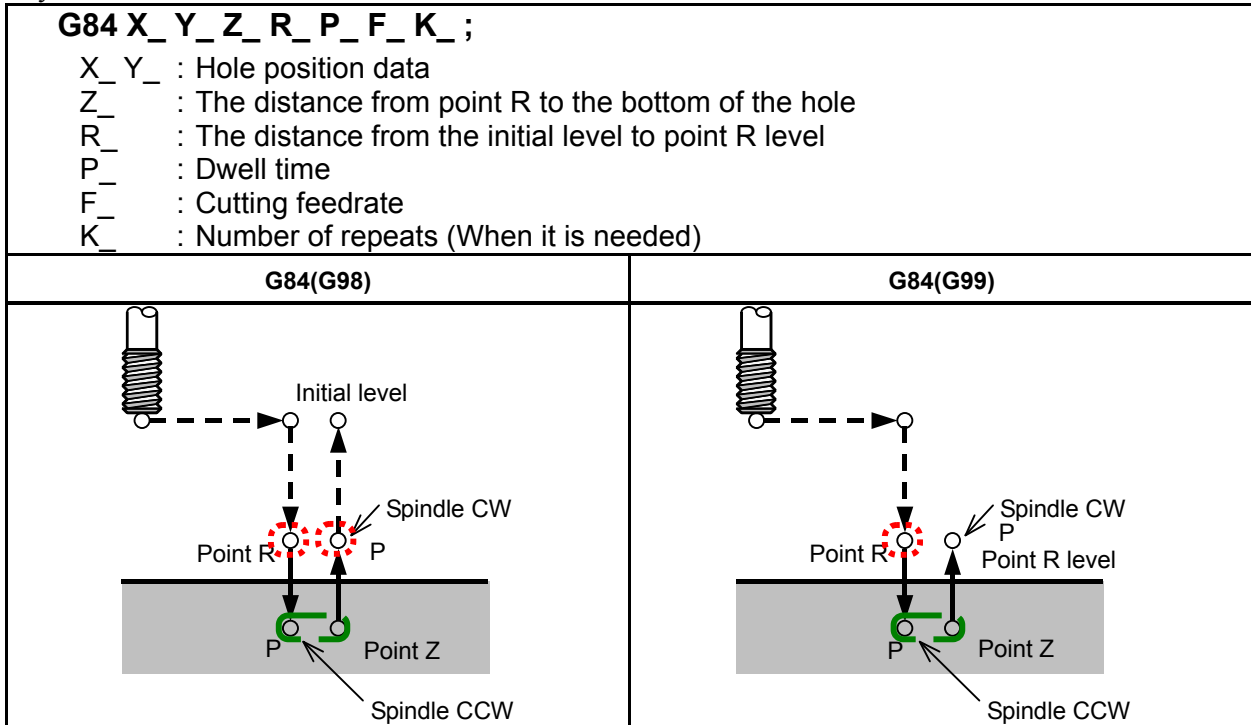
- In-position width for other than hole bottoms (regular)
- In-position width for other than hole bottoms (for retraction in peck drilling cycle)
- In-position width for hole bottoms

NOTE
 When setting an effective area (for in-position check) enclosed in , pay attention to the clearance Δ (parameter No.5174). If the effective area is too large for the retraction distance, it is likely that no retraction may be performed.

- Tapping cycle (G84)

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in tapping cycle.



In-position width for other than hole bottoms (regular)
 In-position width for hole bottoms

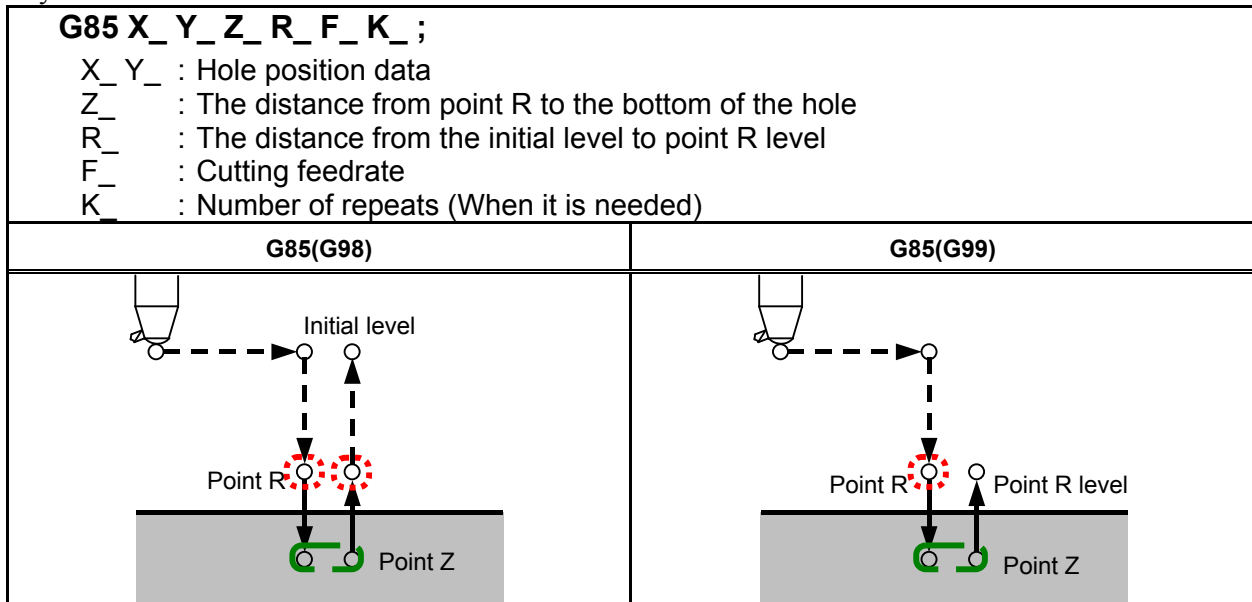
⚠ CAUTION
 When resuming the initial level, set an effective area (for in-position check) enclosed in , pay attention to the distance between point R and the workpiece. If the effective area is too large for the distance between point R and the workpiece, it is likely that the workpiece or tool may be damaged, because a rapid traverse may occur before the tool retracts completely from the workpiece.

NOTE
 Enabling an in-position check at hole bottoms requires setting bit 6 of parameter No. 5103 to 1.

- Boring cycle (G85)

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in boring cycle.



In-position width for other than hole bottoms (regular)

In-position width for hole bottoms



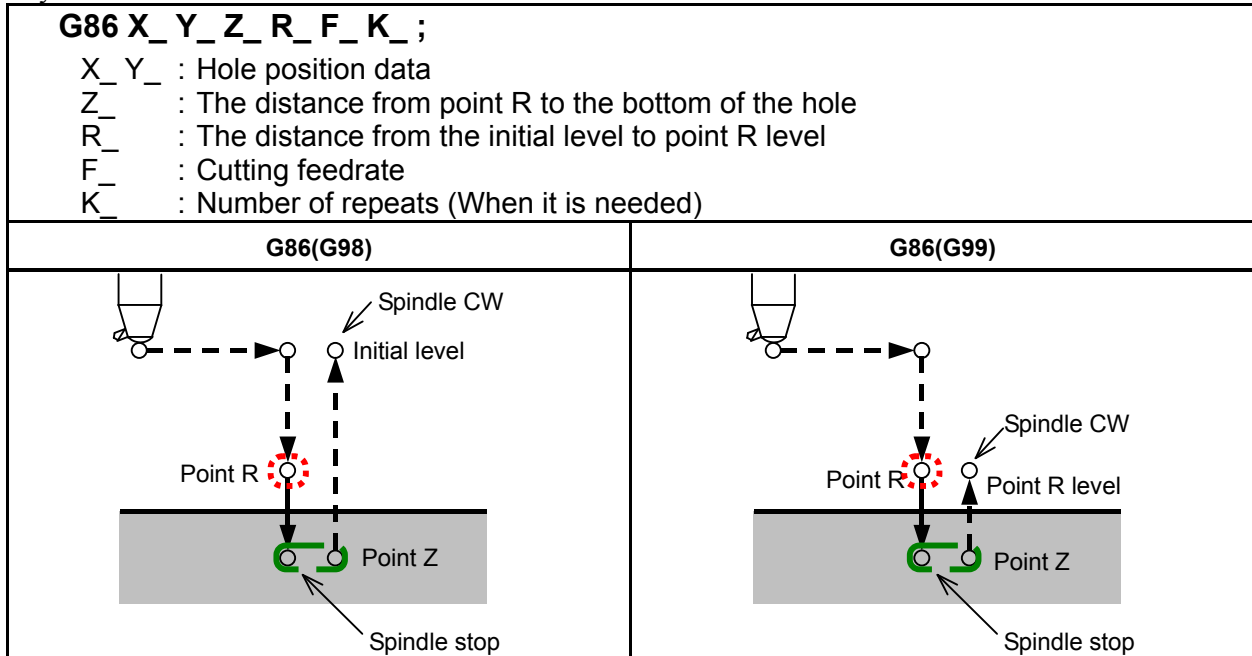
CAUTION

When resuming the initial level, set an effective area (for in-position check) enclosed in , pay attention to the distance between point R and the workpiece. If the effective area is too large for the distance between point R and the workpiece, a rapid traverse may occur before the tool retracts completely from the workpiece. Enabling an in-position check at hole bottoms requires using the exact stop mode (G61).

- Boring cycle (G86)

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in boring cycle.



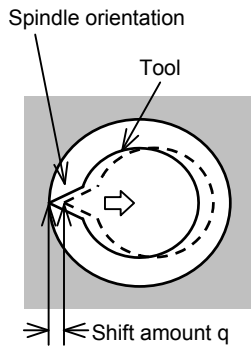
In-position width for other than hole bottoms (regular)

In-position width for hole bottoms

- **Back boring cycle (G87)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in back boring cycle.



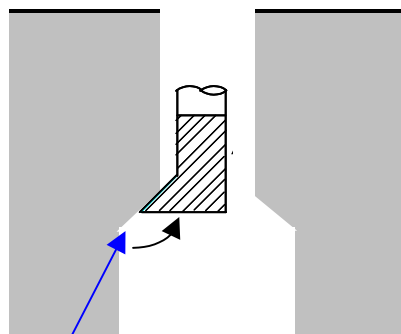
<p>G87 X_ Y_ Z_ R_ Q_ P_ F_ K_ ; X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level Q_ : Shift amount at the bottom of a hole P_ : Dwell time at the bottom of a hole F_ : Cutting feedrate K_ : Number of repeats (When it is needed)</p>	
G87(G98)	G87(G99)
	Not used



In-position width for other than hole bottoms (for shift in boring cycles (G76 and G87))
 In-position width for hole bottoms

CAUTION

When setting an effective area (for in-position check) enclosed in , pay attention to the distance between point R and the workpiece and the orientation of the spindle. If too large an effective area is set, a shortcut is taken, resulting in the tool interfering with the workpiece during rapid traverse.

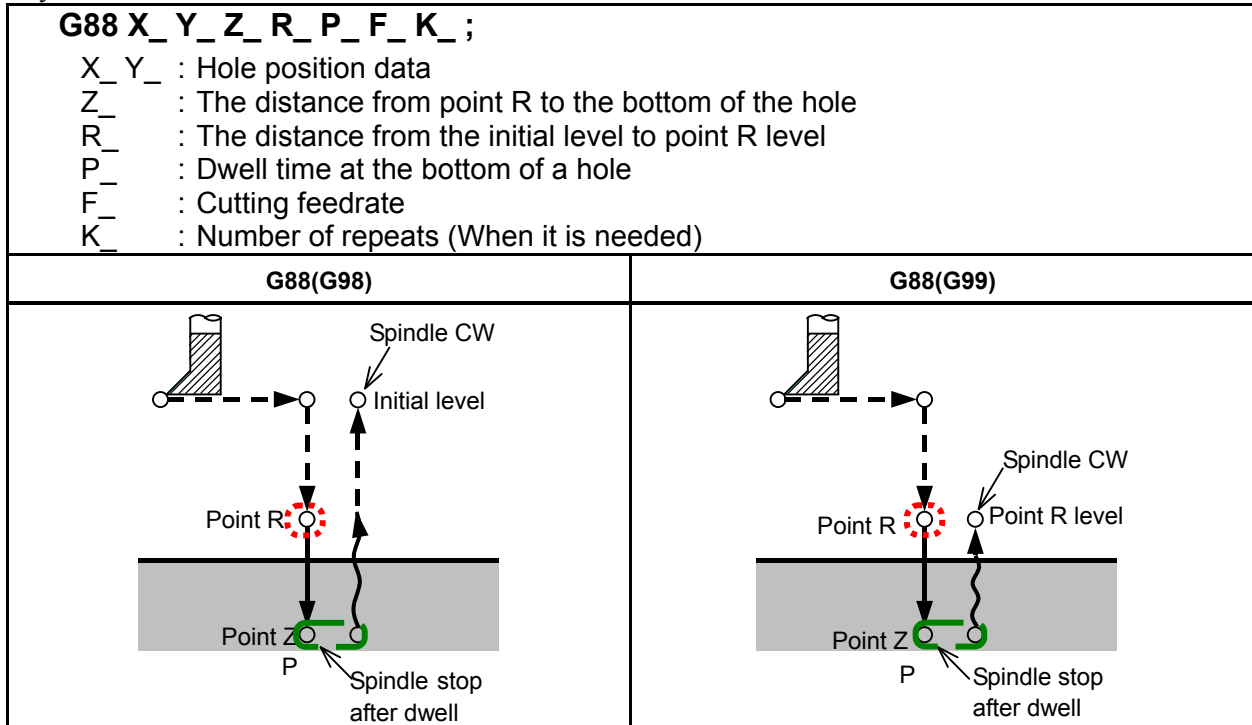


A shortcut is taken, resulting in the tool interfering with the workpiece.

- **Boring cycle (G88)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in boring cycle.



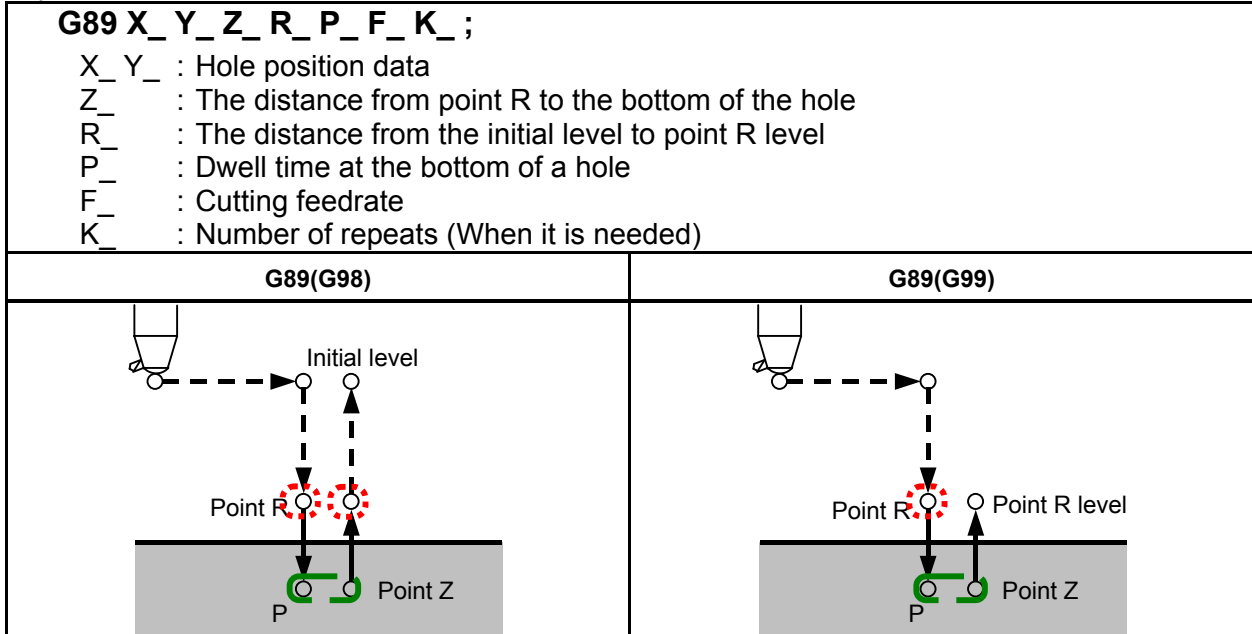
In-position width for other than hole bottoms (regular)

In-position width for hole bottoms

- **Boring cycle (G89)**

M

Shown below are the points where a dedicated effective area (for in-position check) is applied in boring cycle.



In-position width for other than hole bottoms (regular)

In-position width for hole bottoms

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

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

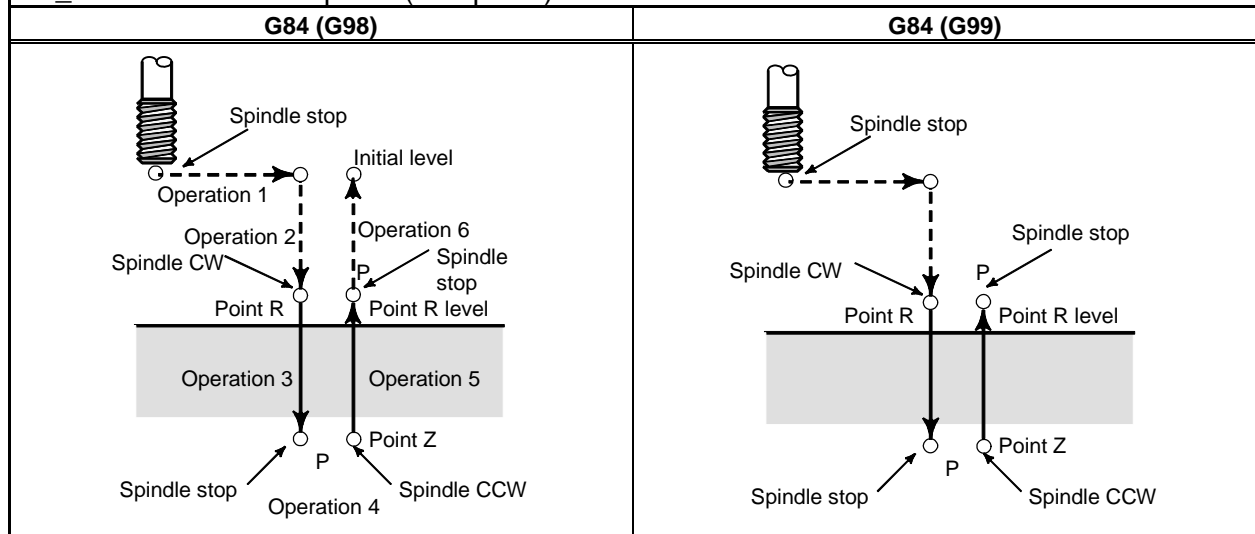
F_ : Cutting feedrate

K_ : Number of repeats (if required)

G84.2 X_ Y_ Z_ R_ P_ F_ L_ ;

(Series 15 format)

L_ : Number of repeats (if required)



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.

- Rigid mode

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

- Specify M29 S***** before a tapping command.
- Specify M29 S***** in a block which contains a tapping command.
- Specify G84 for rigid tapping (bit 0 (G84) of parameter No. 5200 set to 1).

- 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.

- Series 15 format command

Rigid tapping can be performed using 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 is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G74) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

When bit 6 (FHD) of parameter No. 5200 is set to 0, feed hold and single block are invalid in the G84 (G74) mode. When this bit is set to 1, they are valid.

- Manual feed

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

- Backlash compensation

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

Along the drilling axis, backlash compensation has been applied.

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 G74 in a single block. Otherwise, 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.

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

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.

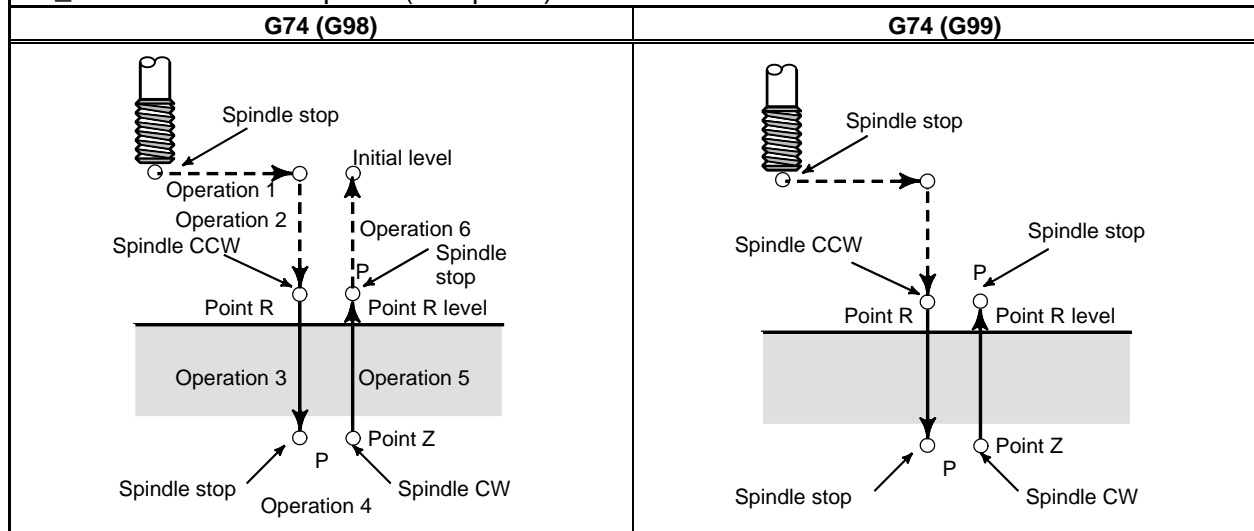
F_ : Cutting feedrate

K_ : Number of repeats (if required)

G84.3 X_ Y_ Z_ R_ P_ F_ L_ ;

(Series 15 format)

L_ : Number of repeats (if required)



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.

- Rigid mode

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

- Specify M29 S***** before a tapping command.
- Specify M29 S***** in a block which contains a tapping command.
- Specify G74 for rigid tapping. (bit 0 (G84) of parameter No. 5200 set to 1).

- 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.

- Series 15 format command

Rigid tapping can be performed using 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 is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G74) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

When bit 6 (FHD) of parameter No. 5200 is set to 0, feed hold and single block are invalid in the G84 (G74) mode. When this bit is set to 1, they are valid.

- Manual feed

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

- Backlash compensation

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

Along the drilling axis, backlash compensation has been applied.

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.

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 PCP bit (bit 5) of parameter 5200.

Format

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

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

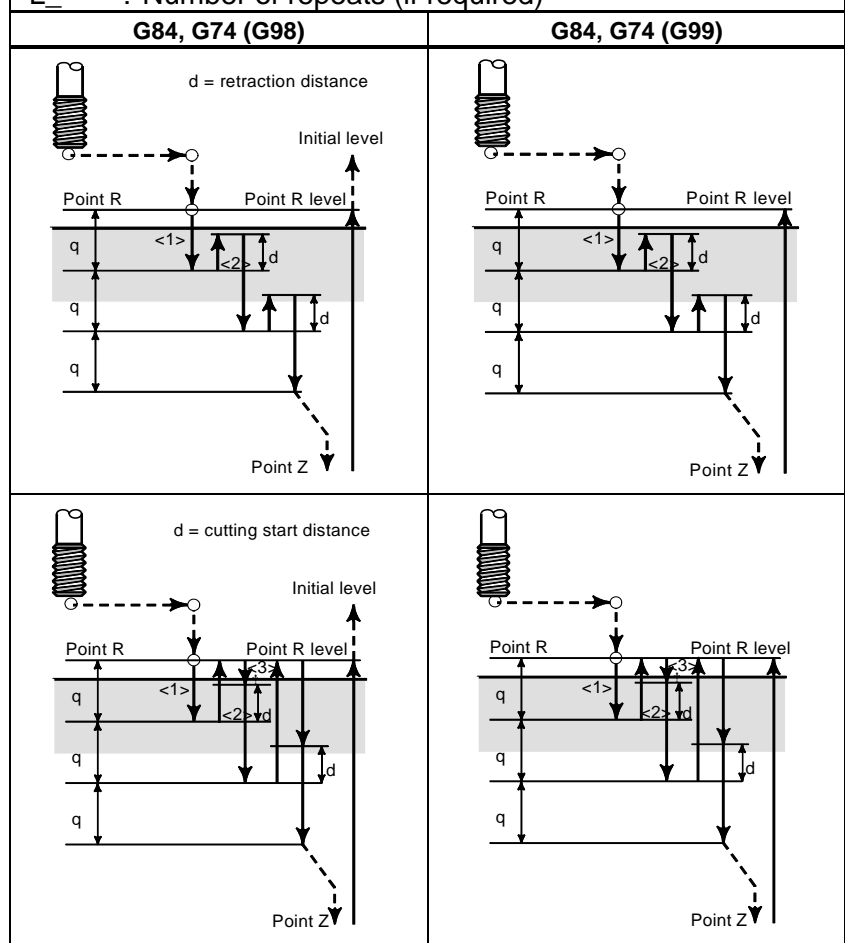
G84.2 (or G84.3) X_ Y_ Z_ R_ P_ Q_ F_ L_ ;

(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.



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 parameter 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 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 is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G74) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

When bit 6 (FHD) of parameter No. 5200 is set to 0, feed hold and single block are invalid in the G84 (G74) mode. When this bit is set to 1, they are valid.

- **Manual feed**

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

- **Backlash compensation**

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

Along the drilling axis, backlash compensation has been applied.

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 in the same block. If they are specified together, G84 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.

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%.

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

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 do not exceed the maximum used gear speed (specified in parameters Nos. 5241 to 5244). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

$$\text{Maximum override (\%)} = \frac{\text{Maximum spindle speed (specified in parameters)}}{\text{Spindle speed (specified at S)}} \times 100$$
- 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 do not exceed the maximum used gear speed (specified in parameters Nos. 5241 to 5244). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

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

- 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

Format

, C_	Chamfering
, R_	Corner R

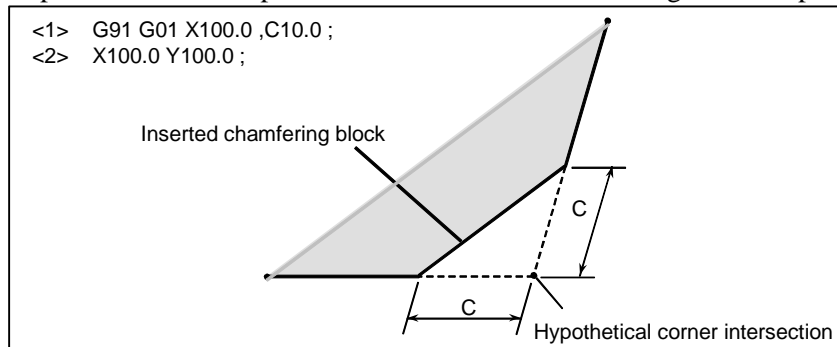
Explanation

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.

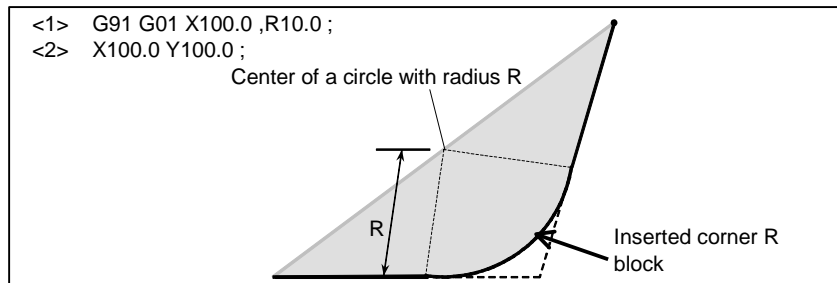
- Chamfering

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.



- Corner R

After R, specify the radius for corner R.



Example

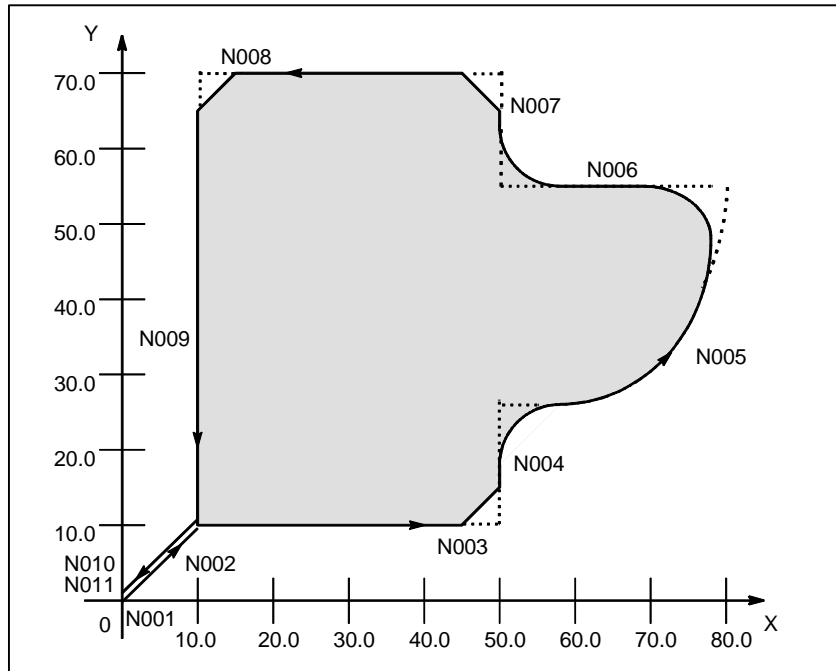
```

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;

```



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.

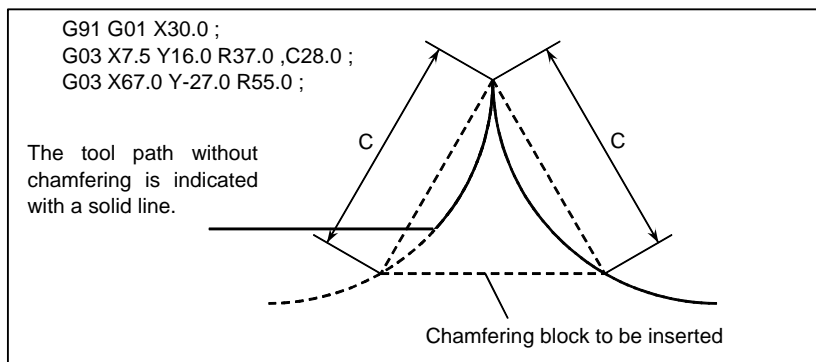


Fig 5.4 (a) Exceeding the move range

- 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 $\pm 1^\circ$.

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 $\pm 1^\circ$. 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 $\pm 1^\circ$.

- 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 the indexing axis (one rotation 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 .

Explanation

- Indexing position

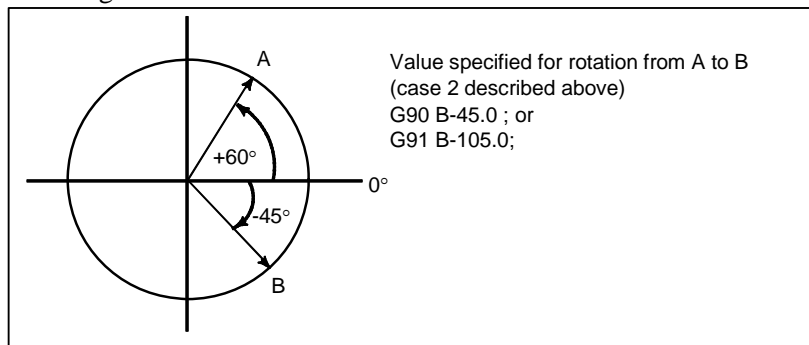
Specify an indexing position with address A, B, or C (set to bit 0 (ROT_x) of parameter No. 1006).

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.



- 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 indexing axis in the rapid traverse mode.

Dry runs cannot be executed for the indexing axis.

⚠ WARNING

- 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 waiting state which waits for completion of clamping or unclamping of the index table is indicated on diagnosis data No.12.
- 3 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.
- 4 If a reset is made while waiting for completion of clamping or unclamping, the clamp or unclamp signal is cleared and the CNC exits the completion wait state.

- **Indexing function and other functions**

Table 5.5 (a) Index 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 indexing axis.
Operations while moving the indexing axis	Unless otherwise processed by the machine, feed hold, interlock and emergency stop can be executed. Machine lock can be executed after indexing is completed.
SERVO OFF signal	Disabled The indexing axis is usually in the servo-off state.
Incremental commands for indexing the index table	The workpiece coordinate system and machine coordinate system must always agree with each other on the indexing axis (the workpiece zero point offset value is zero.).
Operations for indexing the index table	Manual operation is disabled in the JOG, INC, or HANDLE mode. A manual reference position return can be made. If the axis selection signal is set to zero during manual reference position return, movement is stopped and the clamp command is not executed.
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 the switch on the machine operator's panel 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.

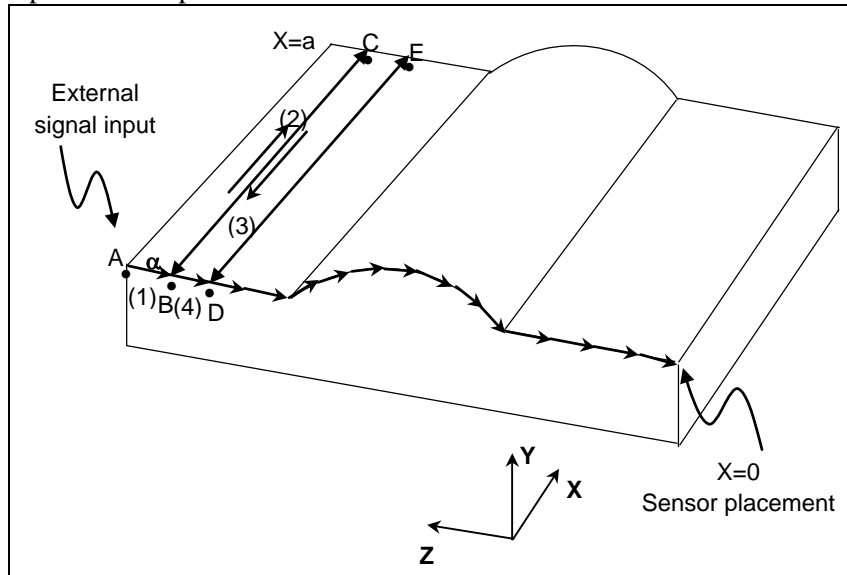


Fig. 5.6 (a)

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 a $X = 0$ position so that the switch on the machine operator's panel 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 switch on the machine operator's panel.

Then, when the sensor detects the grinding wheel, the switch on the machine operator's panel 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 switch on the machine operator's panel 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 switch on the machine operator's panel 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 switch on the machine operator's panel 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 ;

Explanation

- **G161 R_**

This specifies an operation mode and the start of a profile program.

A dept 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 switch on the machine operator's panel. When the switch on the machine operator's panel is input in this state, the machine makes a cut by the amount of cut specified with R. Later, until the end point of the program, the machine makes a cut each time the switch on the machine operator's panel 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, an unexpected movement may result.

- **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 in another way.

- **Block overlap**

In this operation mode, block overlap is disabled.

- **Switch on the machine operator's panel**

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

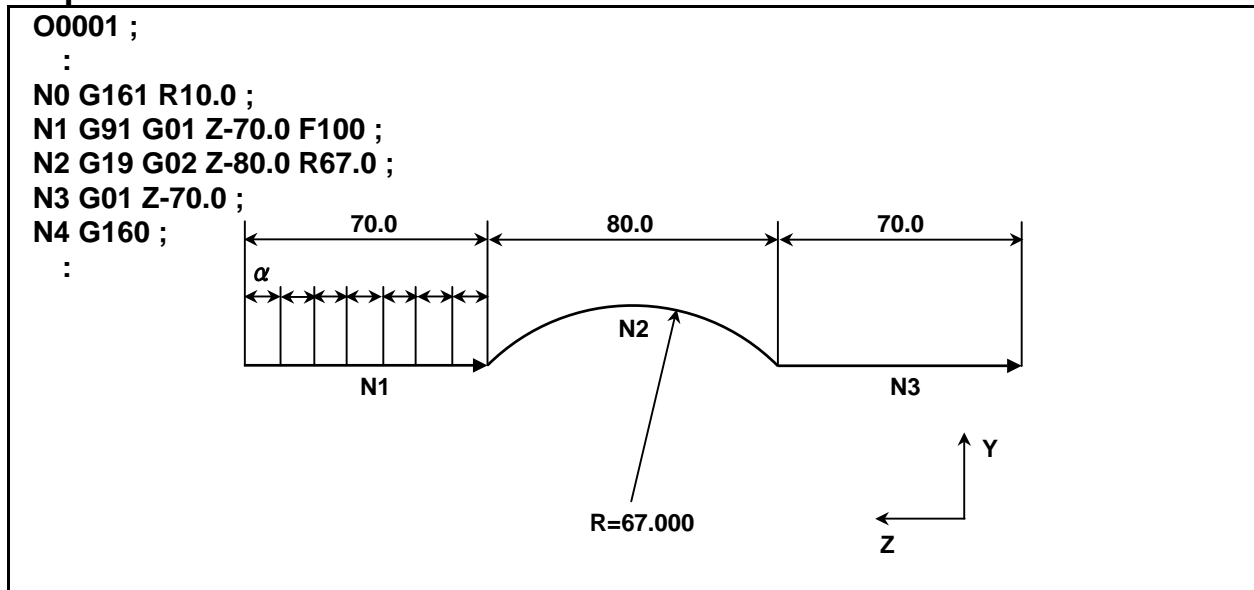
Example

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 switch on the machine operator's panel is input.

α = Travel distance at each input of the switch on the machine operator's panel.

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 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.
%
- 2 When switching from a canned cycle for drilling to a canned grinding cycle, specify G80 to cancel the canned cycle for drilling.
- 3 When switching from a canned grinding cycle to another axis move command, cancel the canned cycle according to the warning above.

5.7.1 Plunge Grinding Cycle (G75)

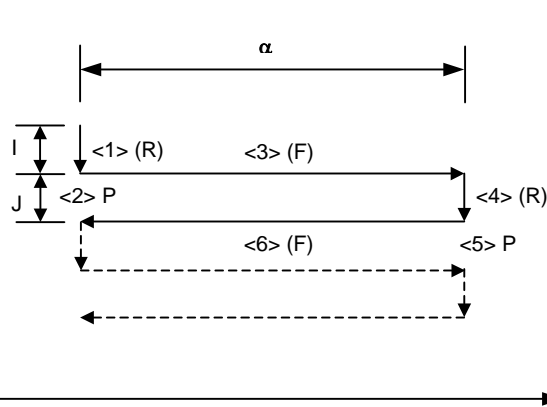
A plunge grinding cycle can be executed.

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



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.

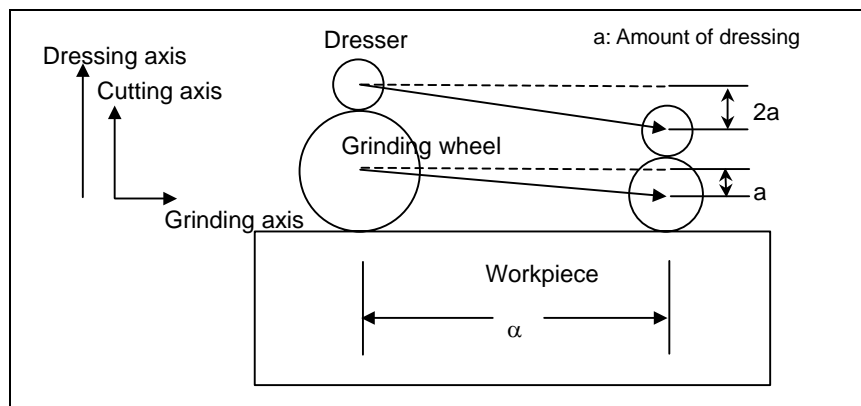
That is, continuous dressing is performed in each grinding operation in the sequence of operations in the cycle, resulting in simultaneous 3-axis interpolation with compensation in the cutting axis direction and compensation in the dressing axis direction simultaneous with movement along the grinding 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 (grinding-wheel wear compensation 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 panel.

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

$\alpha, I, J,$ 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

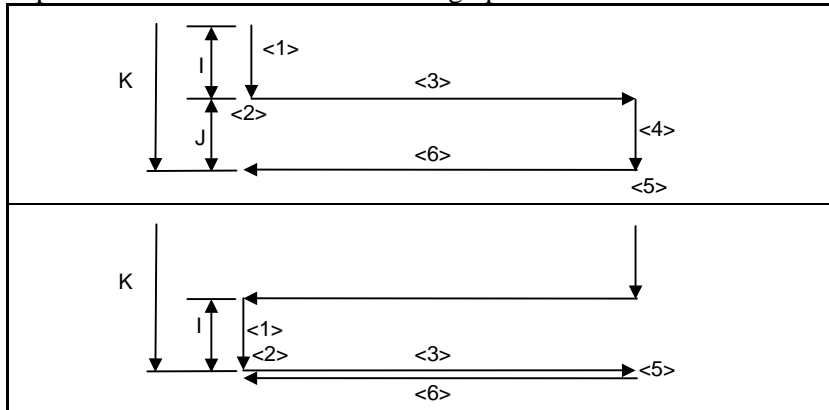
The data items I, J, K, α , R, F, and P in a canned cycle are modal information common to G75, G77, G78, and G79, so that once specified, they remain effective until specified anew. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G75, G77, G78, and G79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

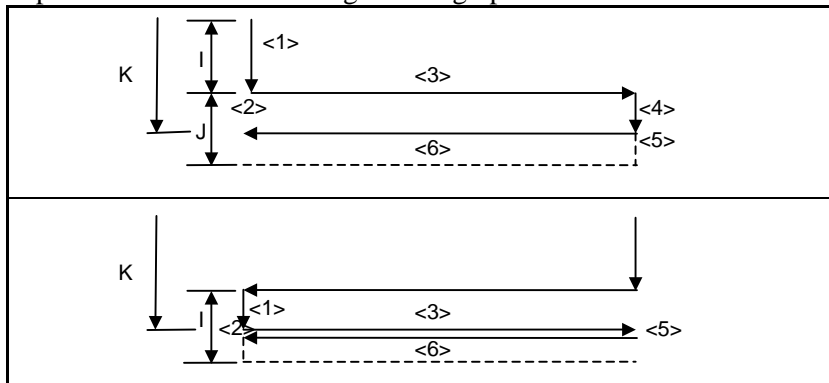
If, during cutting with I or J, the total depth of cut is reached, the cycle is ended after the subsequent operations in the sequence (up to <6>) are executed.

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)

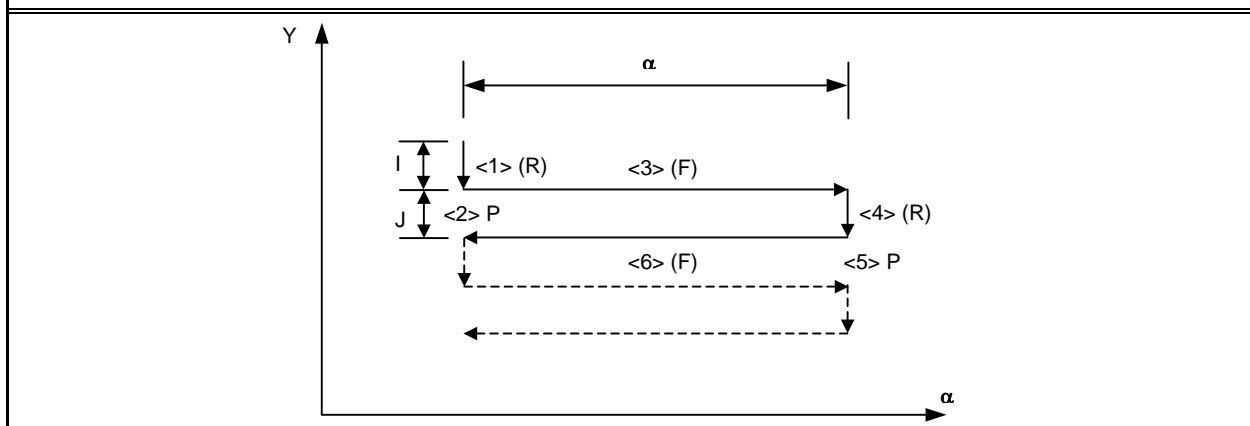
A direct constant-dimension plunge grinding cycle can be performed.

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



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

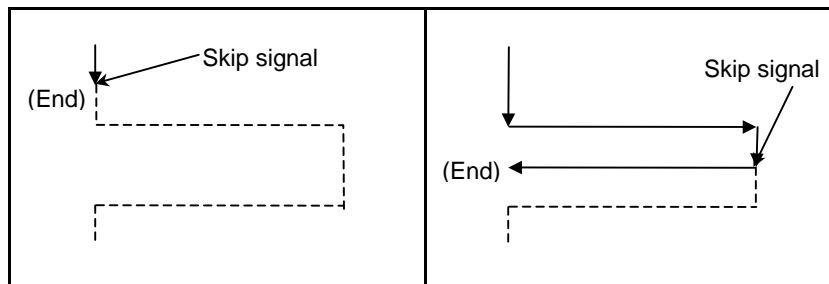
If the continuous dressing function is enabled, the grinding-wheel cut and the dresser cut are continuously compensated for according to the dressing amount specified with L during the execution of grinding. For details, see Explanation of G75.

- Operation to be performed when a skip signal is input

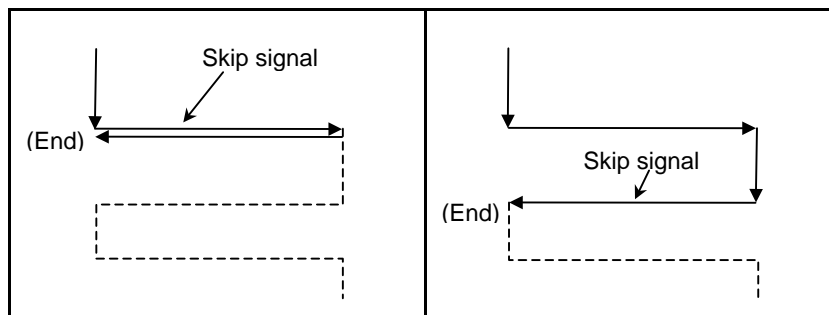
With G77, by inputting a skip signal in a cycle, it is possible to end the cycle after interrupting the current operation sequence (or after ending the current operation sequence).

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.

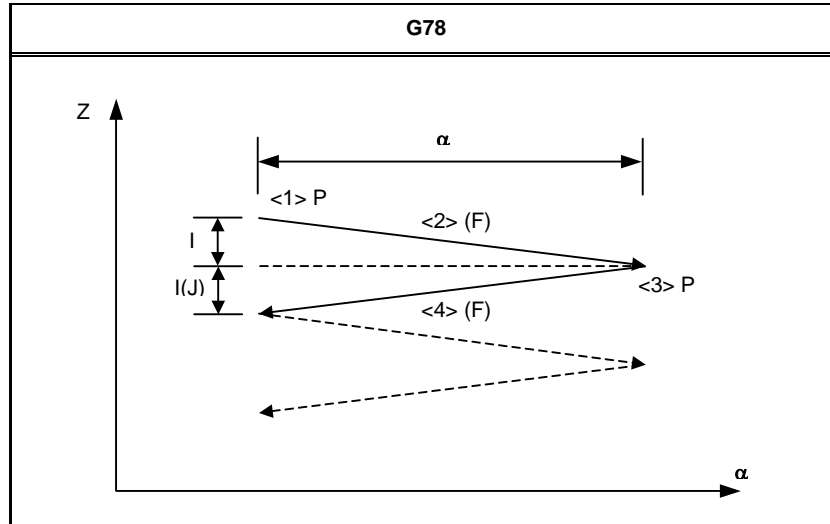
NOTE

- 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



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**

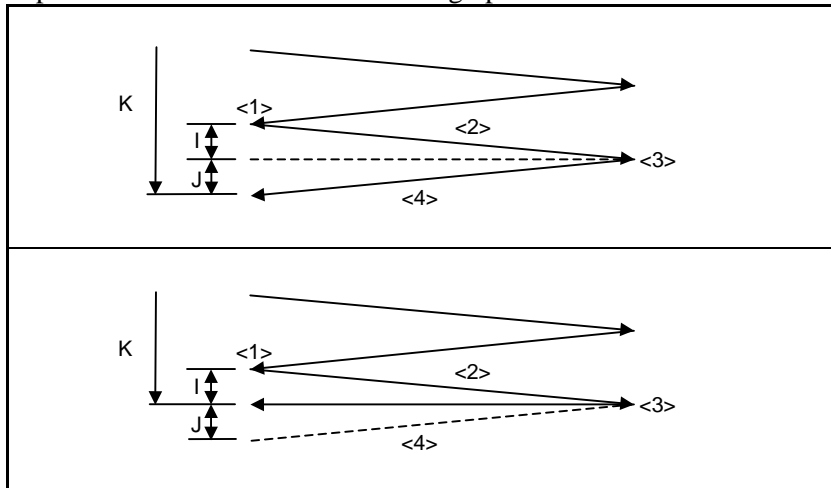
The data items I, K, α , R, F, and P in a canned cycle are modal information common to G75, G77, G78, and G79, so that once specified, they remain effective until specified anew. The data is cleared when a G code of group 00 other than G04 or a G code of group 01 other than G75, G77, G78, and G79 is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

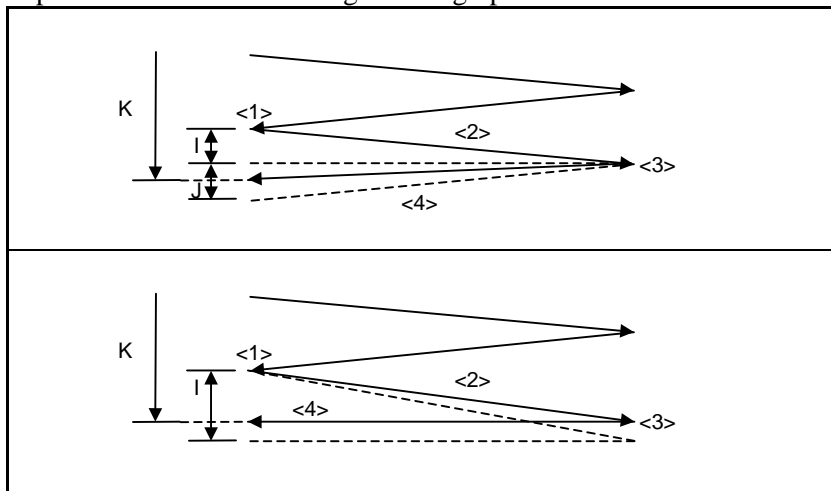
If, during cutting with I or J, the total depth of cut is reached, the cycle is ended after the subsequent operations in the sequence (up to <4>) are executed.

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)

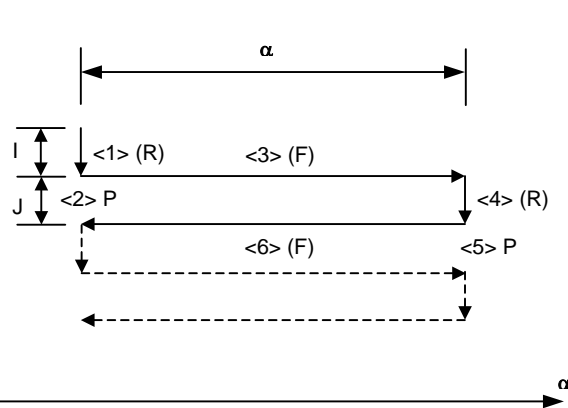
An intermittent-feed surface grinding cycle can be performed.

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



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.

NOTE

- 4 While this cycle is effective, even if G90 (absolute command) is executed, the α , I, J, and K commands are incremental ones.

5.8 MULTIPLE REPETITIVE CYCLE (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7,G76.7)

The multiple repetitive cycle is canned cycles to make CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the threading is available.

NOTE

- 1 Explanatory figures in this section use the ZX plane as the selected plane, diameter programming for the X-axis, and radius programming for the Z-axis. When radius programming is used for the X-axis, change U/2 to U and X/2 to X.
- 2 A multiple repetitive cycle can be performed on any plane.

5.8.1 Stock Removal in Turning (G71.7)

There are two types of stock removal in turning : Type I and II.
To use type II, the "multiple repetitive canned cycle 2" optional function is required.

Format

ZpXp plane

G71.7 U(Δd) R(e) ;

G71.7 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

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

YpZp plane

G71.7 W(Δd) R(e) ;

G71.7 P(ns) Q(nf) V(Δw) W(Δu) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

XpYp plane

G71.7 V(Δd) R(e) ;

G71.7 P(ns) Q(nf) U(Δw) V(Δu) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

Δd : Depth of cut

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

e : Escaping amount

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

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

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

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

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

f,s,t : Any F, S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71.7 block is effective.

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

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

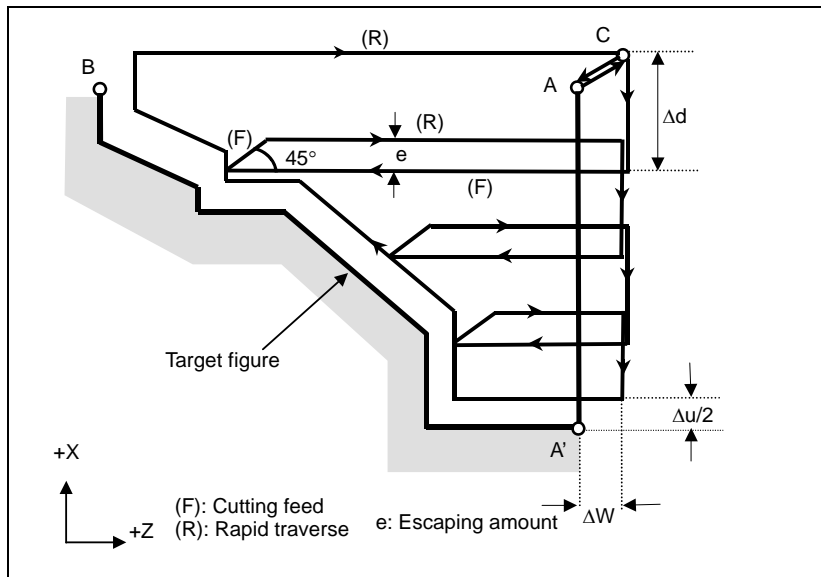


Fig. 5.8.1 (a) Cutting path in stock removal in turning (type I)

Explanation

- Operations

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

NOTE

- 1 While both Δd and Δu are specified by the same address, the meanings of them are determined by the presence of addresses P and Q.
- 2 The cycle machining is performed by G71.7 command with P and Q specification.
- 3 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71.7 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
- 4 When an option of constant surface speed control is selected, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71.7 block or the previous block is effective.

- Target figure Patterns

The four cutting patterns in the Fig. 5.8.1 (b) are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the first axis on the plane (Z-axis for the ZX plane). At this time, the signs of the finishing allowances of Δu and Δw are as follows:

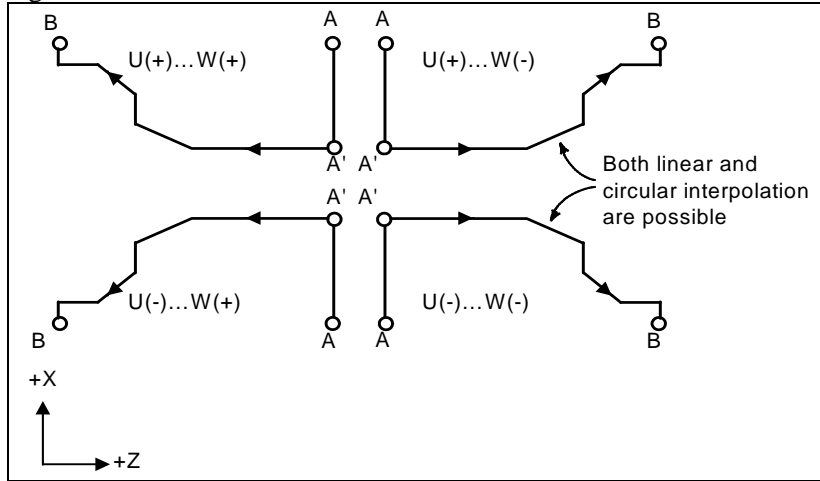


Fig. 5.8.1 (b) Four target figure patterns

Limitation

- (1) For U(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For U(-), a figure for which a position lower than the cycle start point is specified cannot be machined.
- (2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.
- (3) For type II, the figure must show monotone increase or decrease along the first axis on the plane.

- Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.

When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'. In this start block, also select type I or II.

- Check functions

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

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

The following checks can also be made.

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

- Types I and II

Selection of type I or II

For G71.7, there are types I and II.

When the target figure has pockets, be sure to use type II.

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

NOTE

To use type II, the multiple repetitive canned cycle II option is required.

Selecting type I or II

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

(1) When type I is selected

Specify the second axis on the plane (X-axis for the ZX plane). Do not specify the first axis on the plane (Z-axis for the ZX plane).

(2) When type II is selected

Specify the second axis on the plane (X-axis for the ZX plane) and first axis on the plane (Z-axis for the ZX plane).

When you want to use type II because of no movement of the tool along the first axis on the plane (Z-axis for the ZX plane), specify the function with a travel distance of 0.

- Type I

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

Example

```
ZX plane
G71.7 U10.0 R5.0 ;
G71.7 P100 Q200....;
N100 X_ ; (Specifies only the second axis on the plane.)
  : ;
  : ;
  : ;
N200.....;
```

(2) The figure along path A'-B must show monotone increase or decrease in the directions of both axes forming the plane (Z- and X-axes for the ZX plane). It must not have any pocket as shown in the Fig. 5.8.1 (c).

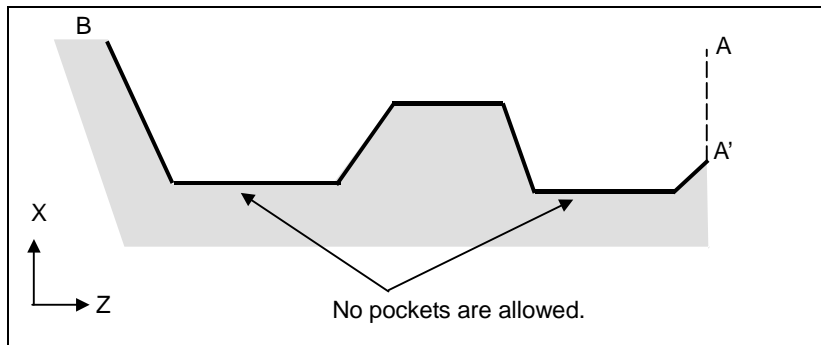


Fig. 5.8.1 (c) Figure which does not show monotone increase or decrease (type I)

⚠ CAUTION

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

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

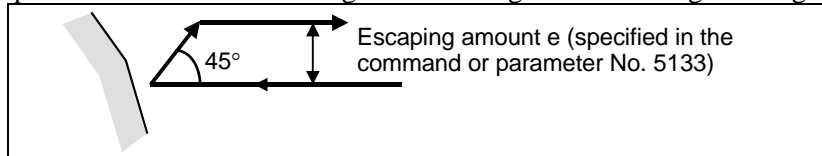


Fig. 5.8.1 (d) Cutting in the direction of 45 degrees (type I)

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

- Type II

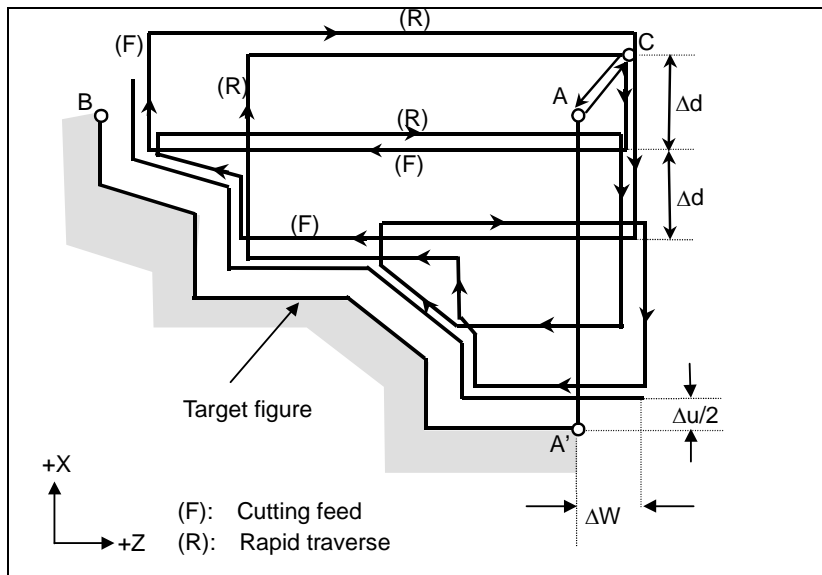


Fig. 5.8.1 (e) Cutting path in stock removal in turning (type II)

When a target figure passing through A, A', and B in this order is given by the program for a target figure as shown in the Fig. 5.8.1 (e), the specified area is removed by Δd (depth of cut), with the finishing allowance specified by $\Delta u/2$ and Δw left.

Type II differs from type I in cutting the workpiece along the figure after rough cutting in the direction of the first axis on the plane (Z-axis for the ZX plane).

After the last cutting, the tool returns to the start point specified in G71.7 and rough cutting is performed as finishing along the target figure, with the finishing allowance specified by $\Delta u/2$ and Δw left.

Type II differs from type I in the following points:

- (1) In the block with sequence number ns, the two axes forming the plane (X-axis and Z-axis for the ZX plane) must be specified. When you want to use type II because of no movement of the tool along the Z-axis on the ZX plane in the first block, specify the function with a travel distance of 0.

```

Example
ZX plane
G71.7 U10.0 R5.0;
G71.7 P100 Q200.....;
N100 X_ Z_ ; (Specifies the two axes forming the plane.)
      ;
      ;
      ;
N200.....;
    
```

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

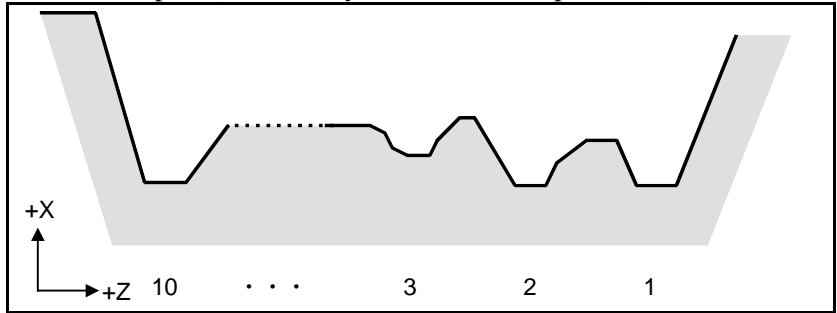


Fig. 5.8.1 (f) Figure having pockets (type II)

The figure must show monotone change in the direction of the first axis on the plane (Z-axis for the ZX plane), however. The Fig. 5.8.1 (g) cannot be machined.

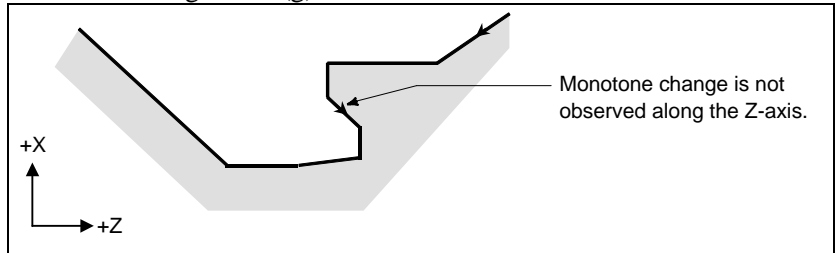


Fig. 5.8.1 (g) Figure which cannot be machined (type II)

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

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

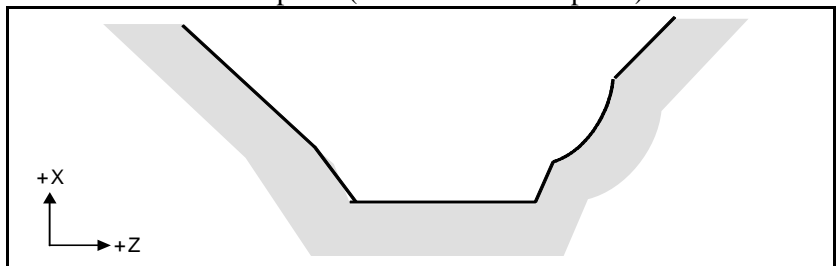


Fig. 5.8.1 (h) Figure which can be machined (type II)

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

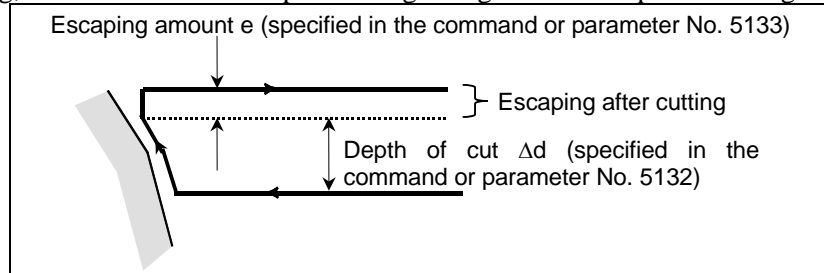


Fig. 5.8.1 (i) Cutting along the workpiece figure (type II)

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

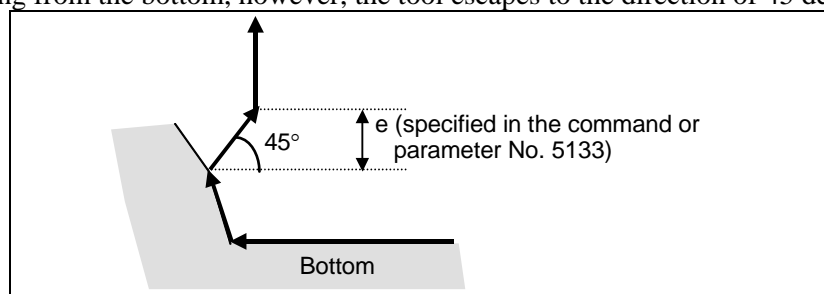


Fig. 5.8.1 (j) Escaping from the bottom to the direction of 45 degrees

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

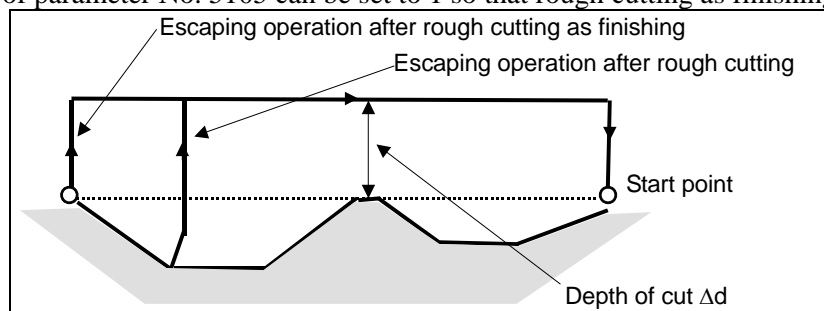


Fig. 5.8.1 (k) Escaping operation when the tool returns to the start point (type II)

(6) Order and path for rough cutting of pockets

Rough cutting is performed in the following order.

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

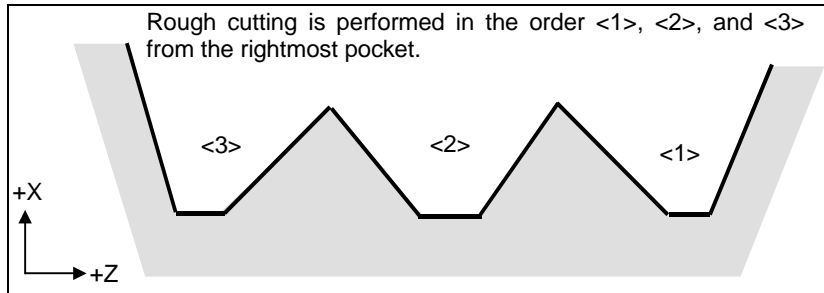


Fig. 5.8.1 (l) Rough cutting order in the case of monotone decrease (type II)

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

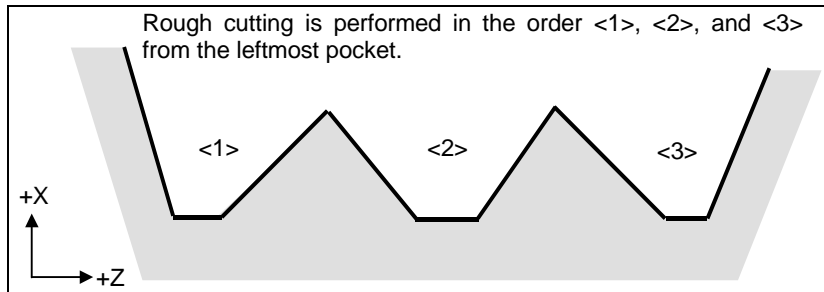


Fig. 5.8.1 (m) Rough cutting order in the case of monotone increase (type II)

The path in rough cutting is as shown Fig. 5.8.1 (n).

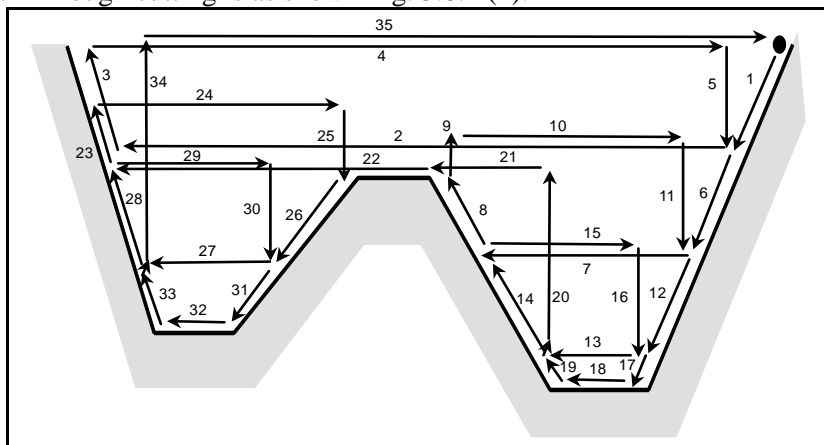


Fig. 5.8.1 (n) Cutting path for multiple pockets (type II)

The Fig. 5.8.1 (o) shows how the tool moves after rough cutting for a pocket in detail.

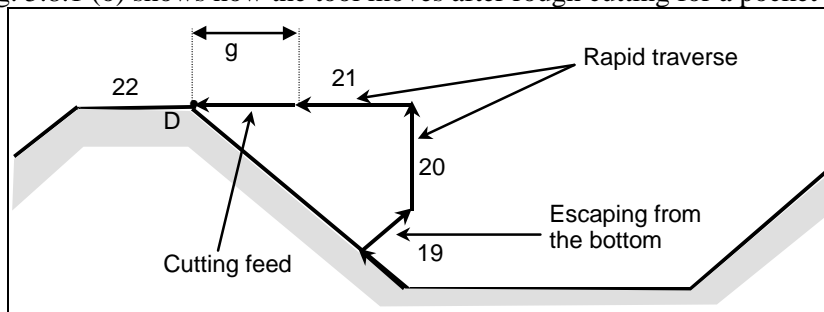


Fig. 5.8.1 (o) Details of motion after cutting for a pocket (type II)

Cuts the workpiece at the cutting feedrate and escapes to the direction of 45 degrees. (Operation 19)
 Then, moves to the height of point D in rapid traverse. (Operation 20)
 Then, moves to the position the amount of g before point D. (Operation 21)
 Finally, moves to point D in cutting feed.
 The clearance g to the cutting feed start position is set in parameter No. 5134.
 For the last pocket, after cutting the bottom, the tool escapes to the direction of 45 degrees and returns to the start point in rapid traverse. (Operations 34 and 35)

⚠ CAUTION

- 1 This CNC differs from the FANUC Series 16i/18i/21i in cutting of a pocket. The tool first cuts the nearest pocket to the start point. After cutting of the pocket terminates, the tool moves to the nearest but one pocket and starts cutting.
- 2 When the figure has a pocket, generally specify a value of 0 for Δw (finishing allowance). Otherwise, the tool may dig into the wall on one side.
- 3 This CNC differs from the FANUC Series 16i/18i/21i in the path of cutting after turning depending on the figure of the workpiece. When the tool becomes moving only along the first axis on the plane (Z-axis for the ZX plane) according to the figure of the workpiece during cutting, it starts retraction along the second axis on the plane (X-axis for the ZX plane).

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70.7, G71.7, G72.7, G73.7) and specify the cancel command (G40) outside the programs (from the block specified with P to the block specified with Q) specifying a target finishing figure. If tool nose radius compensation is specified in the program specifying a target finishing figure, alarm PS0325, “UNAVAILABLE COMMAND IS IN SHAPE PROGRAM”, is issued.

Program example

```
G42;..... Specify this command before a multiple repetitive canned cycle command.
G71.7U1.R.5; .
G71.7P10Q20;
N10G00X0;
:
N20X50.;
G40;..... Specify this command after the program specifying a target finishing figure.
```

When this cycle is specified in the tool nose radius compensation mode, offset is temporarily canceled during movement to the start point. Start-up is performed in the first block. Offset is temporarily canceled again at the return to the cycle start point after termination of cycle operation. Start-up is performed again according to the next move command. This operation is shown in the Fig. 5.8.1 (p).

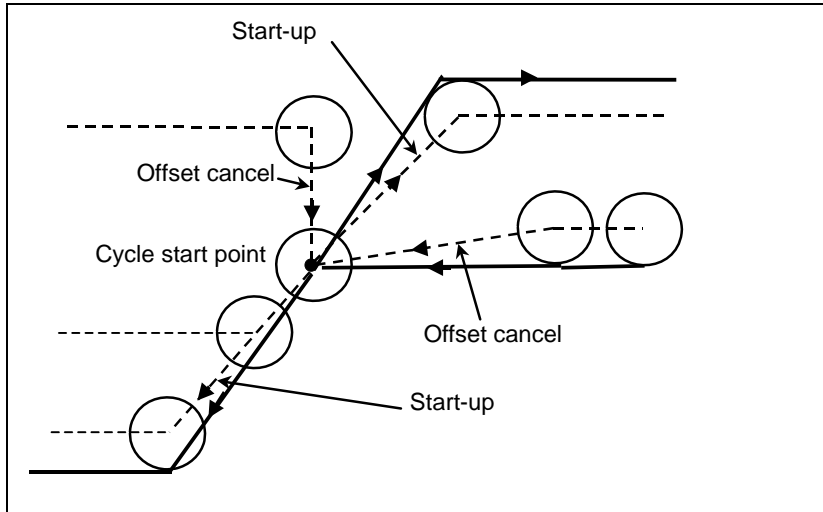


Fig. 5.8.1 (p)

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

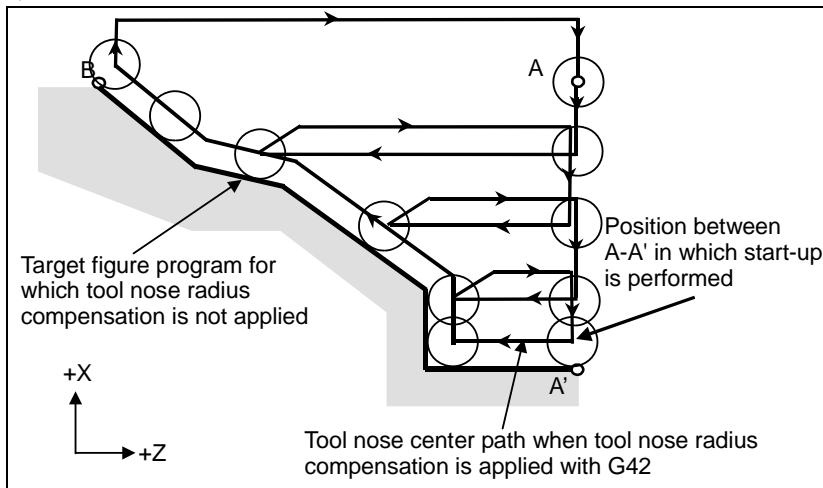


Fig. 5.8.1 (q) Path when tool nose radius compensation is applied

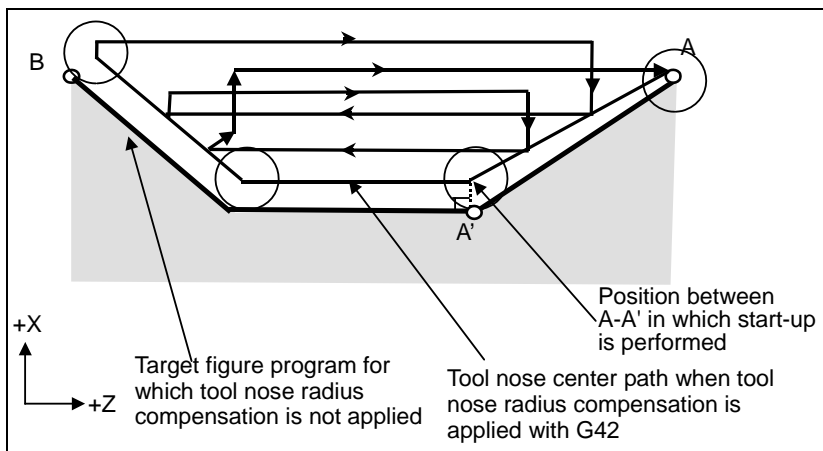


Fig. 5.8.1 (r)

NOTE

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

- Reducing the cycle time

In G71.7 and G72.7, the tool can be moved to the previous turning start point (operation 1) in rapid traverse by setting bit 0 (ASU) of parameter No. 5107 to 1.

Bit 0 (ASU) of parameter No. 5107 is valid for both type I and II commands.

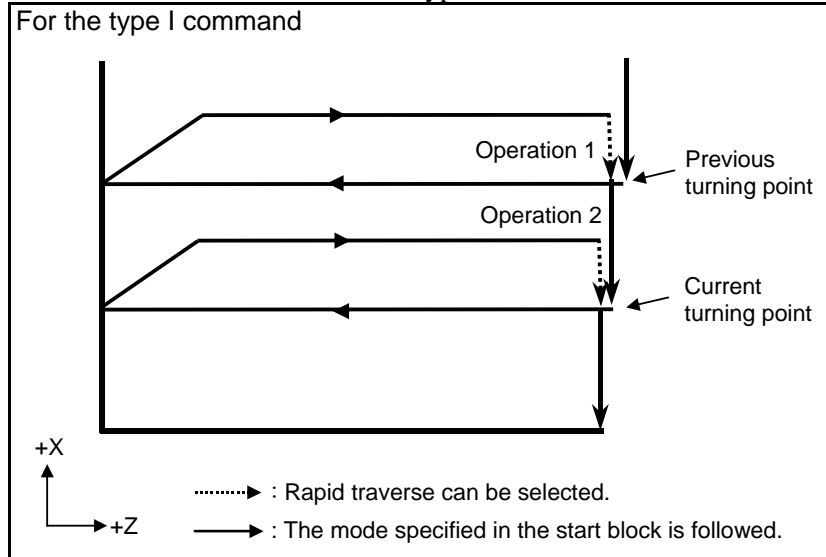


Fig. 5.8.1 (s)

For the type I G71.7 and G72.7 commands, operations 1 and 2 to the current turning start point that are usually performed in 2 cycles can be performed in 1 cycle by setting bit 1 (ASC) of parameter No. 5107 to 1. The feed mode specified in the start block in the program for a target figure (G00 or G01) is used.

Bit 1 (ASC) of parameter No. 5107 is valid only for the type I command.

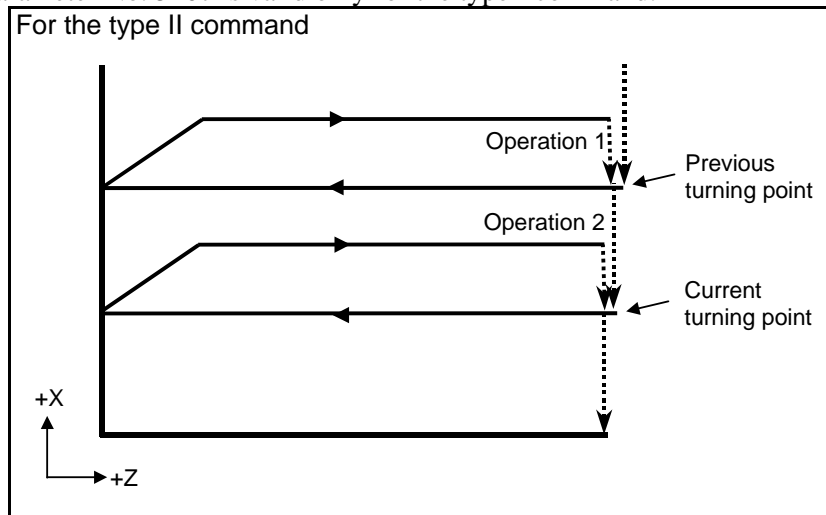


Fig. 5.8.1 (t)

5.8.2 Stock Removal in Facing (G72.7)

This cycle is the same as G71.7 except that cutting is performed by an operation parallel to the second axis on the plane (X-axis for the ZX plane).

Format

ZpXp plane

G72.7 W(Δd) R(e) ;

G72.7 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

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

YpZp plane

G72.7 V(Δd) R(e) ;

G72.7 P(ns) Q(nf) V(Δv) W(Δu) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

XpYp plane

G72.7 U(Δd) R(e) ;

G72.7 P(ns) Q(nf) U(Δu) W(Δv) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

Δd : Depth of cut

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

e : Escaping amount

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

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

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

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

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

f,s,t : Any F, S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G72.7 block is effective.

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

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

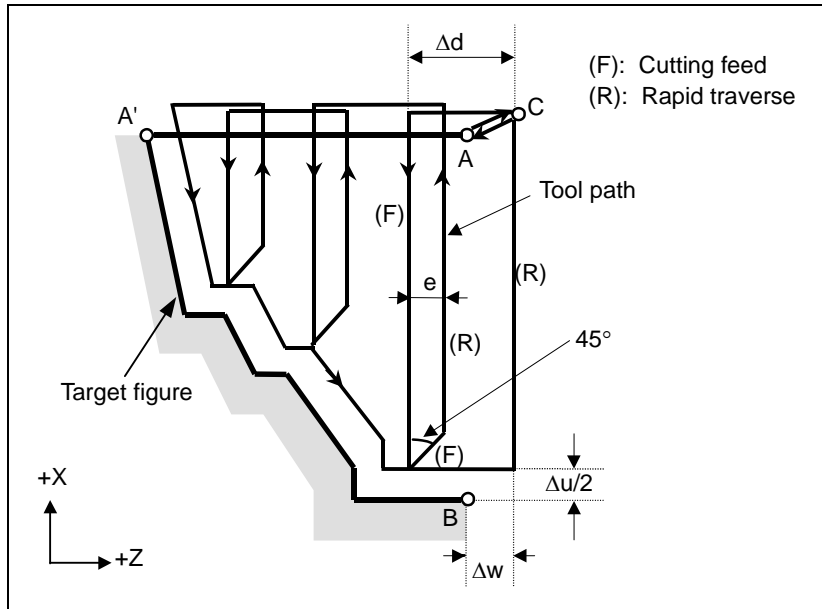


Fig. 5.8.2 (a) Cutting path in stock removal in facing (type I)

Explanation

- Operations

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

NOTE

- 1 While both Δd and Δu are specified by the same address, the meanings of them are determined by the presence of addresses P and Q.
- 2 The cycle machining is performed by G72.7 command with P and Q specification.
- 3 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G72.7 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
- 4 When an option of constant surface speed control is selected, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G72.7 block or the previous block is effective.

- Target figure Patterns

The four cutting patterns in the Fig. 5.8.2 (b) are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the second axis on the plane (X-axis for the ZX plane). At this time, the signs of the finishing allowances of Δu and Δw are as follows:

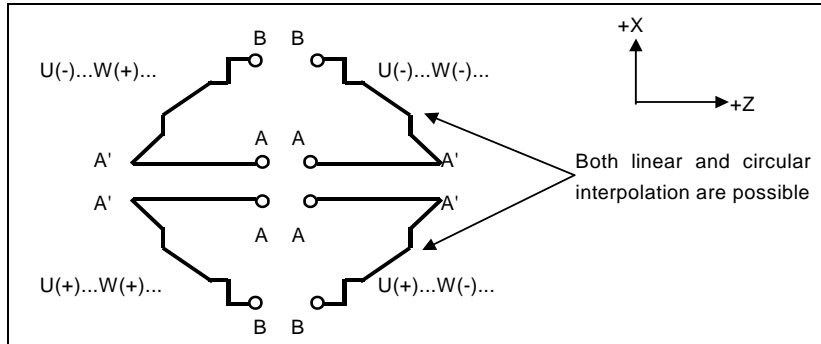


Fig. 5.8.2 (b) Signs of the values specified at U and W in stock removal in facing

Limitation

- (1) For W(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For W(-), a figure for which a position lower than the cycle start point is specified cannot be machined.
- (2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.
- (3) For type II, the figure must show monotone increase or decrease along the second axis on the plane.

- Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.

When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.

In this start block, also select type I or II.

- Check functions

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

NOTE

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

The following checks can also be made.

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

- Types I and II

Selection of type I or II

For G72.7, there are types I and II.

When the target figure has pockets, be sure to use type II.

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

Selecting type I or II

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

- (1) When type I is selected
Specify the first axis on the plane (Z-axis for the ZX plane). Do not specify the second axis on the plane (X-axis for the ZX plane).
- (2) When type II is selected
Specify the second axis on the plane (X-axis for the ZX plane) and first axis on the plane (Z-axis for the ZX plane).
When you want to use type II because of no movement of the tool along the second axis on the plane (X-axis for the ZX plane), specify the function with a travel distance of 0.

- Type I

G72.7 differs from G71.7 in the following points:

- (1) G72.7 cuts the workpiece with moving the tool in parallel with the second axis on the plane (X-axis on the ZX plane).
- (2) In the start block in the program for a target figure (block with sequence number ns), only the first axis on the plane (Z-axis for the ZX plane) must be specified.

- Type II

G72.7 differs from G71.7 in the following points:

- (1) G72.7 cuts the workpiece with moving the tool in parallel with the second axis on the plane (X-axis on the ZX plane).
- (2) The figure need not show monotone increase or decrease in the direction of the first axis on the plane (Z-axis for the ZX plane) and it may have concaves (pockets). The figure must show monotone change in the direction of the second axis on the plane (X-axis for the ZX plane), however.
- (3) When a position parallel to the second axis on the plane (X-axis for the ZX plane) is specified in a block in the program for the target figure, it is assumed to be at the bottom of a pocket.
- (4) After all rough cutting terminates along the second axis on the plane (X-axis for the ZX plane), the tool temporarily returns to the start point. Then, rough cutting as finishing is performed.

- Tool nose radius compensation

See the pages on which G71.7 is explained.

- Reducing the cycle time

See the pages on which G71.7 is explained.

5.8.3 Pattern Repeating (G73.7)

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

Format

ZpXp plane

G73.7 W(Δk) U(Δi) R(d) ;

G73.7 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

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

YpZp plane

G73.7 V(Δk) W(Δi) R(d) ;

G73.7 P(ns) Q(nf) V(Δw) W(Δu) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

XpYp plane

G73.7 U(Δk) V(Δi) R(d) ;

G73.7 P(ns) Q(nf) U(Δw) V(Δu) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

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

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5135, and the parameter is changed by the program command.

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

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5136, and the parameter is changed by the program command.

d : The number of division

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

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

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

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

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

f,s,t : Any F, S, and T function contained in the blocks between sequence number "ns" and "nf" are ignored, and the F, S, and T functions in this G73.7 block are effective.

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

NOTE

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

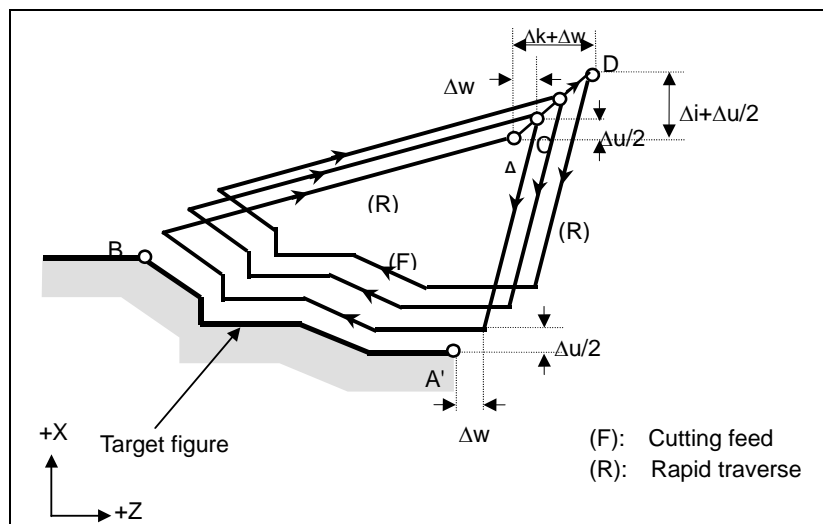


Fig. 5.8.3 (a) Cutting path in pattern repeating

Explanation

- Operations

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

NOTE

- 1 While the values Δi and Δk , or Δu and Δw are specified by the same address respectively, the meanings of them are determined by the presence of addresses P and Q.
- 2 The cycle machining is performed by G73.7 command with P and Q specification.
- 3 After cycle operation terminates, the tool returns to point A.
- 4 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G73.7 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.

- Target figure Patterns

As in the case of G71.7, there are four target figure patterns. Be careful about signs of Δu , Δw , Δi , and Δk when programming this cycle.

- Start block

In the start block in the program for a target figure (block with sequence number n_s in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065 is issued.

When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.

- Check functions

The following check can be made.

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

- Tool nose radius compensation

Like G71.7, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

5.8.4 Finishing Cycle (G70.7)

After rough cutting by G71.7, G72.7 or G73.7, the following command permits finishing.

Format

G70.7 P(ns) Q(nf) ;

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

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

Explanation

- Operations

The blocks with sequence numbers ns to nf in the program for a target figure are executed for finishing. The F, S, T, M, and second auxiliary functions specified in the G71.7, G72.7, or G73.7 block are ignored and the F, S, T, M, and second auxiliary functions specified in the blocks with sequence numbers ns to nf are effective.

When cycle operation terminates, the tool is returned to the start point in rapid traverse and the next G70.7 cycle block is read.

- Target figure

Check function

Following check can be made.

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

- Storing P and Q blocks

When rough cutting is executed by G71.7, G72.7, or G73.7, up to three memory addresses of P and Q blocks are stored. By this, the blocks indicated by P and Q are immediately found at execution of G70.7 without searching memory from the beginning for them. After some G71.7, G72.7, and G73.7 rough cutting cycles are executed, finishing cycles can be performed by G70.7 at a time. At this time, for the fourth and subsequent rough cutting cycles, the cycle time is longer because memory is searched for P and Q blocks.

Example

```
G71.7 P100 Q200 - - - ;
```

```
N100 - - - ;
```

```
- - - ;
```

```
- - - ;
```

```
N200 - - - ;
```

```
G71.7 P300 Q400 - - - ;
```

```
N300 - - - ;
```

```
- - - ;
```

```
- - - ;
```

```
N400 - - - ;
```

```
- - - ;
```

```
- - - ;
```

```
G70.7 P100 Q200 ; (Executed without a search for the first to third cycles)
```

```
G70.7 P300 Q400 ; (Executed after a search for the fourth and subsequent cycles)
```

NOTE

The memory addresses of P and Q blocks stored during rough cutting cycles by G71.7, G72.7, and G73.7 are erased after execution of G70.7.

All stored memory addresses of P and Q blocks are also erased by a reset.

- Return to the cycle start point

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

NOTE

The tool returns to the cycle start point always in the nonlinear positioning mode regardless of the setting of bit 1 (LRP) of parameter No. 1401.

Before executing a finishing cycle for a target figure with a pocket cut by G71.7 or G72.7, check that the tool does not interfere with the workpiece when returning from the end point of the target figure to the cycle start point.

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41 or G42) before a multiple repetitive canned cycle command (G70.7) and specify the cancel command (G40) after the multiple repetitive canned cycle command (G70.7).

Program example

G42;.....Specify this command before a multiple repetitive canned cycle command.

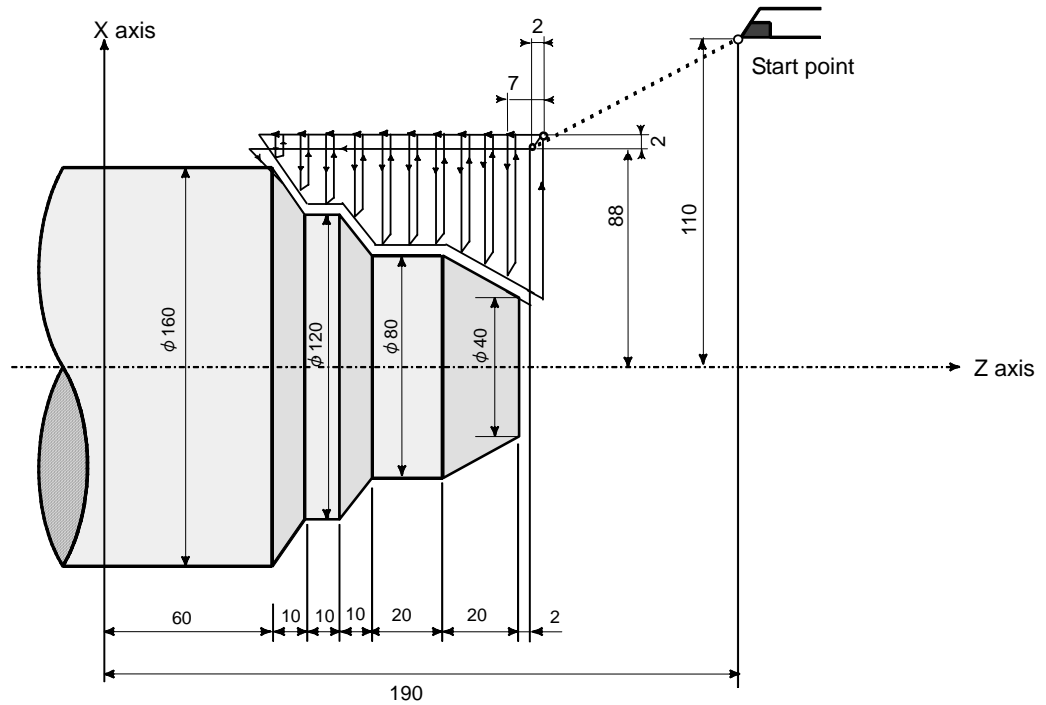
G70.7P10Q20;

G40;.....Specify this command after a multiple repetitive canned cycle command.

Like G71.7, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

Example

Stock removal in facing (G72.7)



(Diameter designation for X axis, metric input)

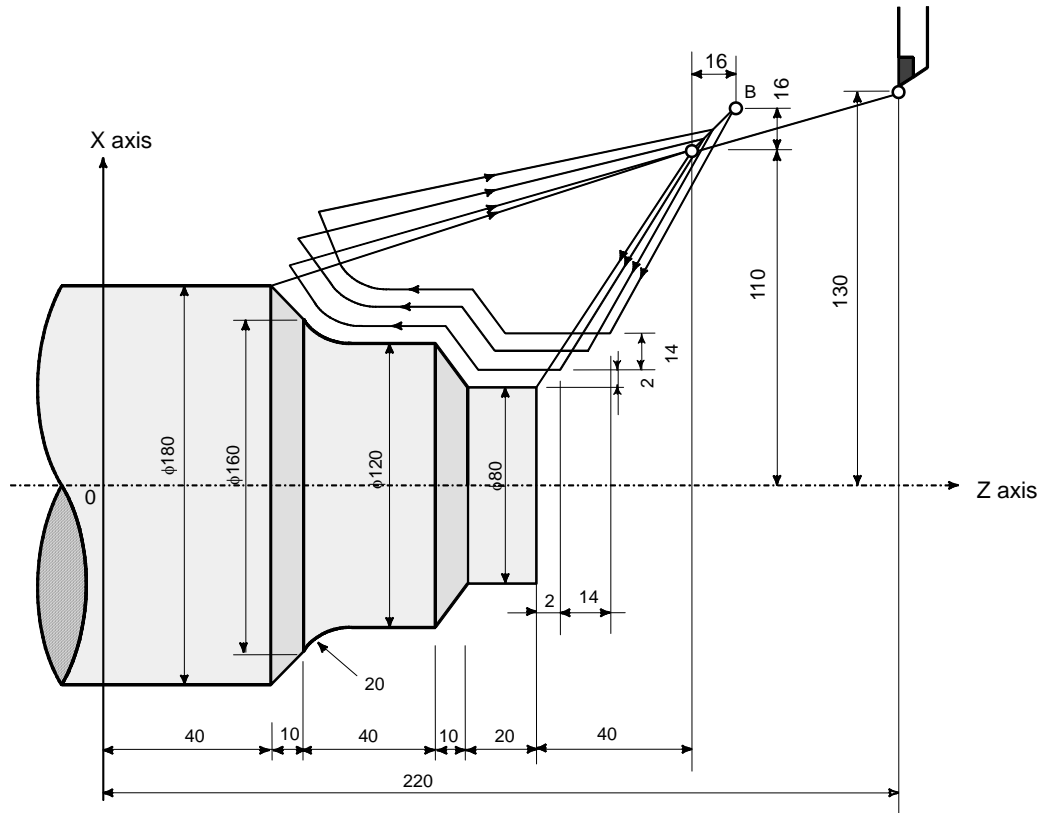
```

N010 G90G92 X220.0 Z190.0 ;
N011 G00 X176.0 Z132.0 ;
N012 G72.7 W7.0 R1.0 ;
N013 G72.7 P014 Q019 U4.0 W2.0 F0.3 S550 ;
N014 G00 Z56.0 S700 ;
N015 G01 X120.0 Z70.0 F0.15 ;
N016 Z80.0 ;
N017 X80.0 Z90.0 ;
N018 Z110.0 ;
N019 X36.0 Z132.0 ;
N020 G70.7 P014 Q019 ;
    
```

Escaping amount: 1.0

Finishing allowance (4.0 in diameter in the X direction, 2.0 in the Z direction)

Pattern repeating (G73.7)



(Diameter designation, metric input)

```

N010 G90G92 X260.0 Z220.0 ;
N011 G00 X220.0 Z160.0 ;
N012 G73.7 U14.0 W14.0 R3 ;
N013 G73.7 P014 Q019 U4.0 W2.0 F0.3 S0180 ;
N014 G00 X80.0 Z120.0 ;
N015 G01 Z100.0 F0.15 S0600 ;
N016 X120.0 Z90.0 ;
N017 Z70.0 S0400 ;
N018 G02 X160.0 Z50.0 R20.0 ;
N019 G01 X180.0 Z40.0 S0280 ;
N020 G70.7 P014 Q019 ;
    
```

5.8.5 End Face Peck Drilling Cycle (G74.7)

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

Format

G74.7R (e) ;

G74.7X_ Z_ P(Δ i) Q(Δ k) R(Δ d) F (f) ;

e : Return amount

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

X_,Z_ : Coordinate of the second axis on the plane (X-axis for the ZX plane) at point B and
Coordinate of the first axis on the plane (Z-axis for the ZX plane) at point C
For an absolute command, coordinates (X,Z)
For an incremental command, travel distance (Δ x, Δ z)

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

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

Δ d : Relief amount of the tool at the cutting bottom

f : Feedrate

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

NOTE

Normally, specify a positive value for Δ d. When X and Δ i are omitted, specify a value with the sign indicating the direction in which the tool is to escape.

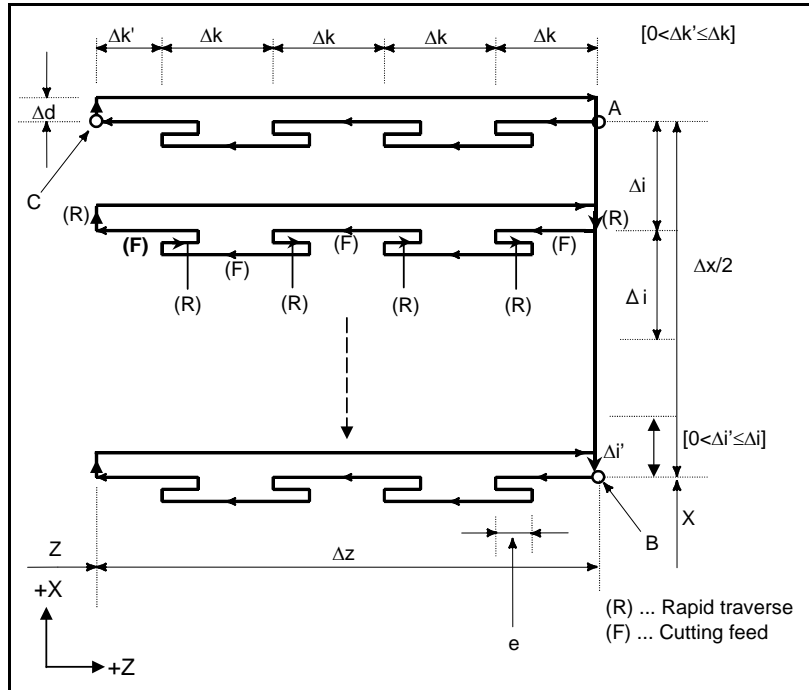


Fig. 5.8.5 (a) Cutting path in end face peak drilling cycle

Explanation

- Operations

A cycle operation of cutting by Δk and return by e is repeated.

When cutting reaches point C, the tool escapes by Δd . Then, the tool returns in rapid traverse, moves to the direction of point B by Δi , and performs cutting again.

NOTE

- 1 For e and Δd , the same address must be specified. They are distinguished from each other according to whether an axis, such as X, Y, and Z is specified. That is, if an axis is specified, the address is regarded as being that for Δd .
- 2 A cycle operation is performed with a G74.7 command with an axis specified.

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

5.8.6 Outer Diameter / Internal Diameter Drilling Cycle (G75.7)

This cycle is equivalent to G74.7 except that the second axis on the plane (X-axis for the ZX plane) changes places with the first axis on the plane (Z-axis for the ZX plane). This cycle enables chip breaking in end facing. It also enables grooving during outer diameter cutting and cutting off (when the Z-axis and Q are omitted for the first axis on the plane).

Format

G75.7R (e) ;
G75.7X_ Z_ P(Δ i) Q(Δ k) R(Δ d) F (f) ;

- e** : Return amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5139, and the parameter is changed by the program command.
- X_,Z_** : Coordinate of the second axis on the plane (X-axis for the ZX plane) at point B and Coordinate of the first axis on the plane (Z-axis for the ZX plane) at point C
For an absolute command, coordinates (X,Z)
For an incremental command, travel distance (Δ x, Δ z)
- Δ i** : Depth of cut in the direction of the second axis on the plane (X-axis for the ZX plane)
- Δ k** : Travel distance in the direction of the first axis on the plane (Z-axis for the ZX plane)
- Δ d** : Relief amount of the tool at the cutting bottom
- f** : Feedrate

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

NOTE

Normally, specify a positive value for Δ d. When Z and Δ k are omitted, specify a value with the sign indicating the direction in which the tool is to escape.

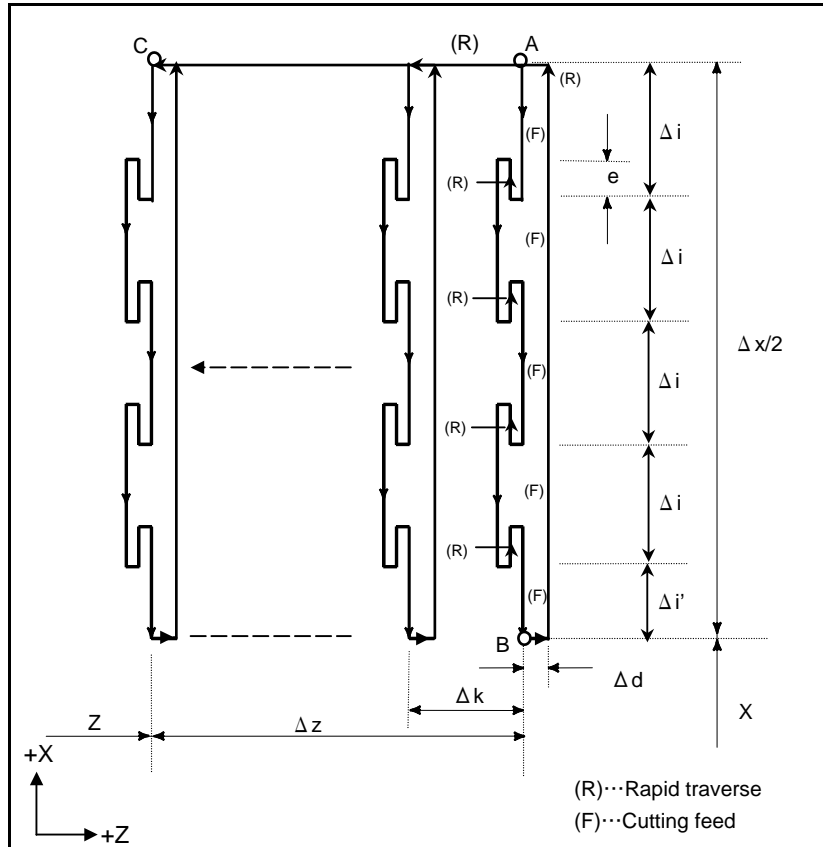


Fig. 5.8.6 (a) Outer diameter/internal diameter drilling cycle

Explanation

- Operations

A cycle operation of cutting by Δi and return by e is repeated.

When cutting reaches point B, the tool escapes by Δd . Then, the tool returns in rapid traverse, moves to the direction of point C by Δk , and performs cutting again.

Both G74.7 and G75.7 are used for grooving and drilling, and permit the tool to relief automatically. Four symmetrical patterns are considered, respectively.

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

5.8.7 Multiple Threading Cycle (G76.7)

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

Format

G76.7 P(m) (r) (a) Q(Δ min) R(d) ;

G76.7 X_ Z_ R(i) P(k) Q(Δ d) F (L) ;

m : Repetitive count in finishing (1 to 99)

This value can be specified by the parameter No. 5142, and the parameter is changed by the program command.

r : Chamfering amount (0 to 99)

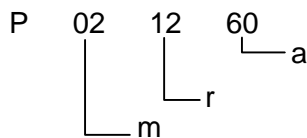
When the thread lead is expressed by L, the value of L can be set from 0.0L to 9.9L in 0.1L increment (2-digit number). This value can be specified by the parameter No. 5130, and the parameter is changed by the program command.

a : Angle of tool nose (thread angle)

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

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

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



Δ min : Minimum cutting depth

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

d : Finishing allowance

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

X_,Z_ : Coordinates of the cutting end point (point D in the Fig. 5.8.7 (a)) in the direction of the length

For an absolute command, coordinates (X,Z)

For an incremental command, travel distance (Δ x, Δ z)

i : Taper amount

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

k : Height of thread

Δ d : Depth of cut in 1st cut

L : Lead of thread

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

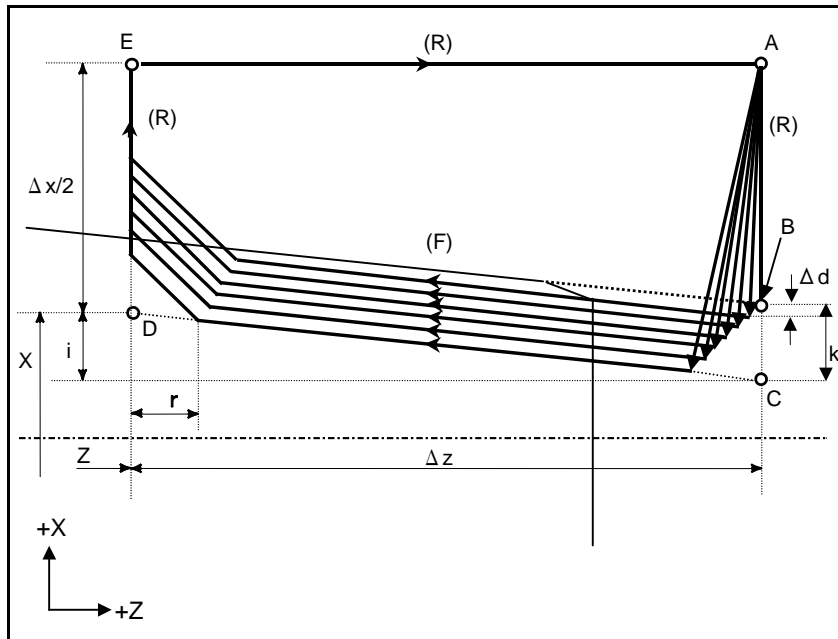


Fig. 5.8.7 (a) Cutting path in multiple threading cycle

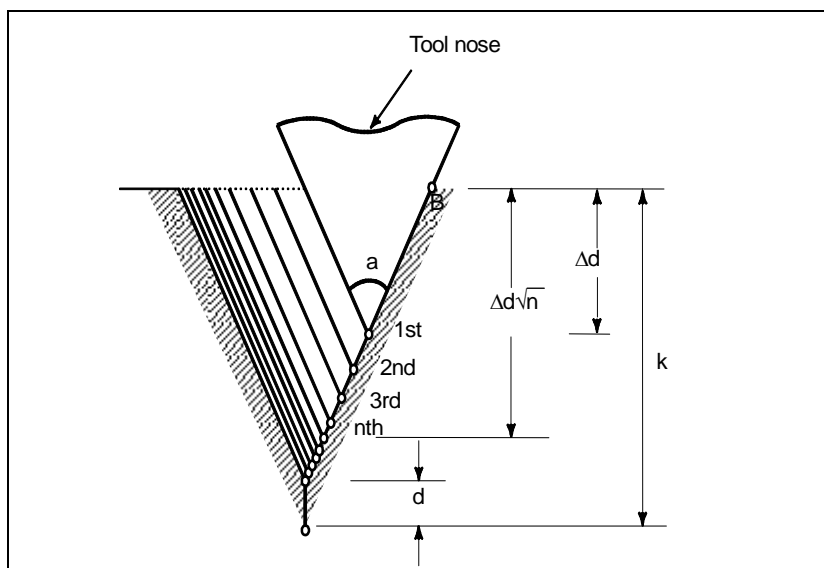


Fig. 5.8.7 (b) Detail of cutting

- Repetitive count in finishing

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

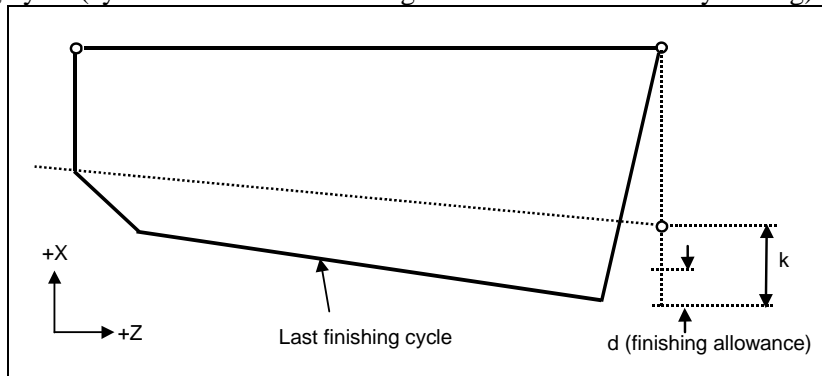


Fig. 5.8.7 (c)

Explanation

- Operations

This cycle performs threading so that the length of the lead only between C and D is made as specified in the F code. In other sections, the tool moves in rapid traverse.

NOTE

- 1 The meanings of the data specified by address P, Q, and R determined by the presence of X (U) and Z (W).
- 2 The cycle machining is performed by G76.7 command with X and Z specification.
- 3 The values specified at addresses P, Q, and R are modal and are not changed until another value is specified.
- 4 Specify a value smaller than the height of thread as the finishing allowance. ($d < k$)

CAUTION

Notes on threading are the same as those on G33 threading. For feed hold in a threading cycle, however, see "Feed hold in a threading cycle" described below.

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

The signs of incremental dimensions for the cycle shown in Fig. 5.8.7 (a) are as follows:
Cutting end point in the direction of the length for X and Z:

- Taper amount (i) : Minus (determined according to the directions of paths A-C and C-D)
- Height of thread (k) : Plus (always specified with a plus sign)
- Depth of cut in the first cut (Δd) : Plus (always specified with a plus sign)

The four patterns shown in the Table 5.8.7 (a) are considered corresponding to the sign of each address. A female thread can also be machined.

Table 5.8.7 (a)

Outer diameter machining	Internal diameter machining
1. $\Delta x < 0, \Delta z < 0, i < 0$	2. $\Delta x > 0, \Delta z < 0, i > 0$
3. $\Delta x < 0, \Delta z < 0, i > 0$ at $ i \leq \Delta x / 2 $	4. $\Delta x > 0, \Delta z < 0, i < 0$ at $ i \leq \Delta x / 2 $

- Acceleration/deceleration after interpolation for threading

Acceleration/deceleration after interpolation for threading is acceleration/deceleration of exponential interpolation type. By setting bit 5 (THLx) of parameter No. 1610, the same acceleration/deceleration as for cutting feed can be selected. (The settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610 are followed.) However, as a time constant and FL feedrate, the settings of parameter No. 1626 and No. 1627 for the threading cycle are used.

- Time constant and FL feedrate for threading

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

- Thread chamfering

Thread chamfering can be performed in this threading cycle. A signal from the machine tool initiates thread chamfering.

The maximum amount of thread chamfering (r) that can be specified in the command is 99 (9.9L). The amount can be specified in a range from 0.1L to 12.7L in 0.1L increments in parameter No. 5130.

A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.

For thread chamfering, the same type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate as for threading are used.

- Retraction after chamfering

The Table 5.8.7 (b) lists the feedrate, type of acceleration/deceleration after interpolation, and time constant of retraction after chamfering.

Table 5.8.7 (b)

Bit 0 (CFR) of parameter No. 1611	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.

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

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

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

- Shifting the start angle

The threading start angle cannot be shifted.

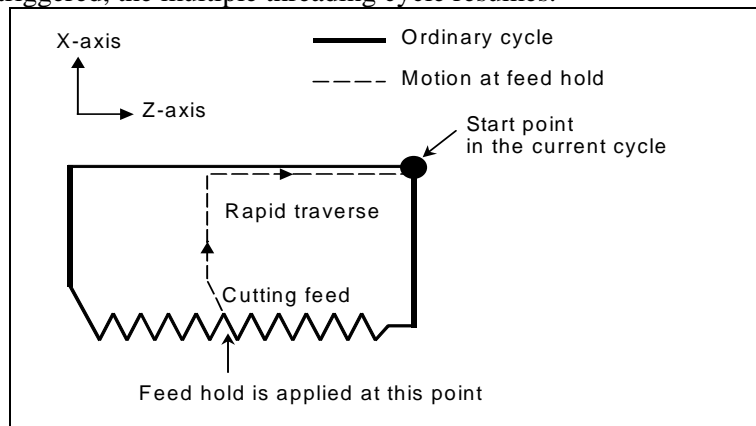
- Feed hold in a threading cycle

When the threading cycle retract function is not used, the machine stops at the end point of retraction after chamfering (point E on the cutting path for a multiple threading cycle) by feed hold applied during threading.

- Feed hold when the threading cycle retract function is used

When the "threading cycle retract" optional function is used, feed hold may be applied during threading in a multiple threading cycle (G76.7). In this case, the tool quickly retracts in the same way as for the last chamfering in a threading cycle and returns to the start point in the current cycle.

When cycle start is triggered, the multiple threading cycle resumes.



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

CAUTION
 Another feed hold cannot be performed during retraction.

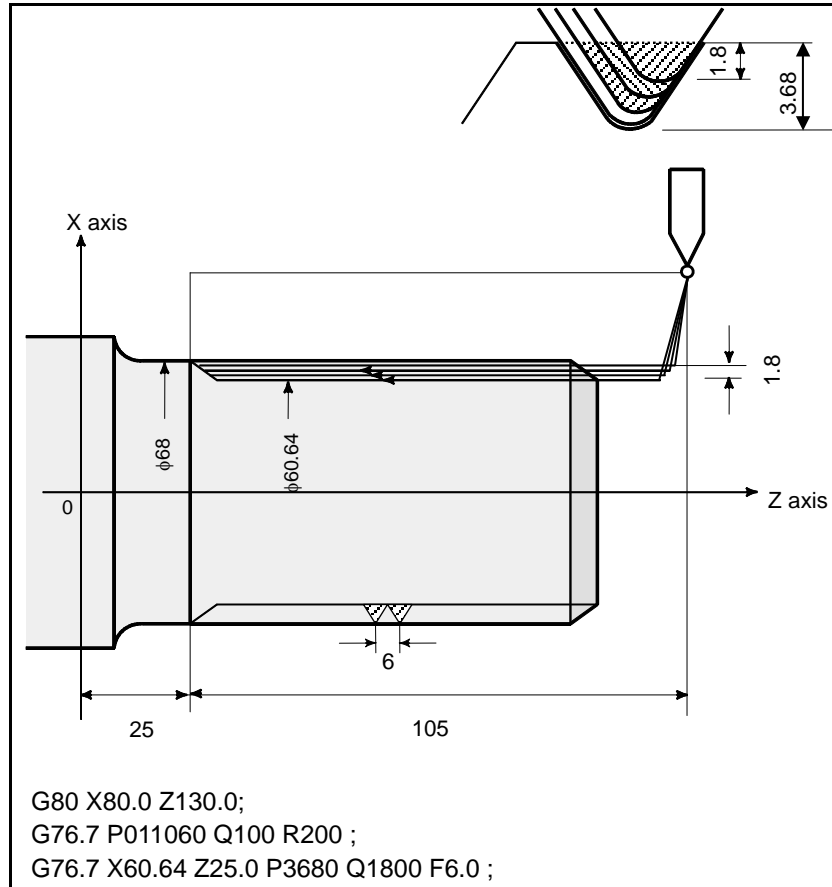
- Inch threading

Inch threading specified with address E is not allowed.

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

Example



5.8.8 Restrictions on Multiple Repetitive Cycle (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7, and G76.7)

Programmed commands

- Program memory

Programs using G70.7, G71.7, G72.7, or G73.7 must be stored in the program memory. The use of the mode in which programs stored in the program memory are called for operation enables these programs to be executed in other than the MEM mode. Programs using G74.7, G75.7, or G76.7 need not be stored in the program memory.

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

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

In a block in which G70.7, G71.7, G72.7, or G73.7 is specified, the following functions cannot be specified:

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

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

In the block which is specified by address P of a G71.7, G72.7 or G73.7, G00 or G01 code in group 01 should be commanded. If it is not commanded, alarm PS0065 is generated.

In blocks with sequence numbers between those specified at P and Q in G70.7, G71.7, G72.7, and G73.7, the following commands can be specified:

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

When G70.7, G71.7, G72.7, or G73.7 is executed, the sequence number specified by address P and Q should not be specified twice or more in the same program.

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

Relation with other functions

- Manual intervention

While a multiple repetitive cycle (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7, and G76.7) is being executed, it is possible to stop the cycle and to perform manual intervention.

The setting of manual absolute on or off is effective for manual operation.

- Interruption type macro

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

- Program restart and tool retract and recover

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

- Axis name and second auxiliary functions

Even if address U, V, or W is used as an axis name or second auxiliary function, data specified at address U, V, or W in a G71.7, G72.7, or G73.7 block is assumed to be that for the multiple repetitive cycle.

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70.7, G71.7, G72.7, G73.7) and specify the cancel command (G40) outside the programs (from the block specified with P to the block specified with Q) specifying a target finishing figure. If tool nose radius compensation is specified in the program specifying a target finishing figure, alarm PS0325, "UNAVAILABLE COMMAND IS IN SHAPE PROGRAM", is issued.

- Multi-spindle control

When a spindle selection by address P of multi-spindle control is used, S code cannot be specified at the block of multiple repetitive canned cycle command (G71.7-G73.7). (The alarm PS5305 "ILLEGAL SPINDLE NUMBER" is issued.)

In this case, instead of specifying S code at the block of multiple repetitive canned cycle command (G71.7-G73.7) is specified, specify S code before the multiple repetitive canned cycle command (G71.7-G73.7) block.

6 COMPENSATION FUNCTION

Chapter 6, "COMPENSATION FUNCTION", consists of the following sections:

6.1	TOOL LENGTH COMPENSATION SHIFT TYPES.....	157
6.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)	164
6.3	TOOL OFFSET (G45 TO G48).....	167
6.4	OVERVIEW OF CUTTER COMPENSATION (G40-G42).....	172
6.5	OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42).....	177
6.6	DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION	187
6.7	VECTOR RETENTION (G38).....	237
6.8	CORNER CIRCULAR INTERPOLATION (G39)	238
6.9	3-DIMENSIONAL TOOL COMPENSATION (G40, G41)	240
6.10	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10).....	243
6.11	COORDINATE SYSTEM ROTATION (G68, G69)	246
6.12	GRINDING WHEEL WEAR COMPENSATION	253
6.13	ACTIVE OFFSET VALUE CHANGE FUNCTION BASED ON MANUAL FEED	258
6.14	ROTARY TABLE DYNAMIC FIXTURE OFFSET	261
6.15	TOOL AXIS DIRECTION TOOL LENGTH COMPENSATION.....	267
6.16	SPINDLE UNIT COMPENSATION, NUTATING ROTARY HEAD TOOL LENGTH COMPENSATION	275

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 and
3-dimensional cutter compensation (G41.2/G42.2/
G41.3/G41.4/G42.4/G41.5/G42.5/G41.6/G42.6) mode)
- In smooth interpolation (G05.1Q2) mode

*2 The commands below are included:

- Tool length compensation (G43/G44)
- Tool length compensation in tool axis direction (G43.1)
- Tool center point control (G43.4/G43.5)

Example in which overcutting occurs in cutter compensation

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

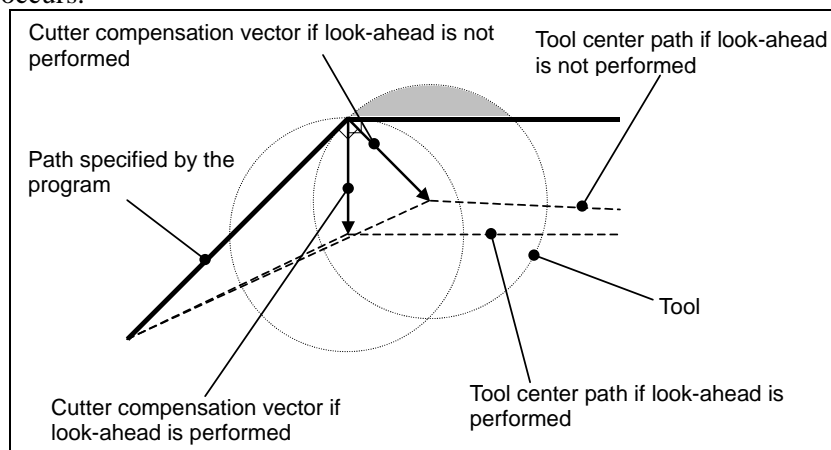
:

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 and 3-dimensional cutter compensation (G41.2/G42.2 / G41.3 / G41.4/G42.4 / G41.5/G42.5 / G41.6/G42.6) mode)

In smooth interpolation (G05.1Q2) mode

*2 The commands below are included:

Tool length compensation (G43/G44)

Tool length compensation in tool axis direction (G43.1)

Tool center point control (G43.4/G43.5)

*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, G30, or G30.1 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, G30, or G30.1) 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, G30, or G30.1 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.

⚠ CAUTION

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.

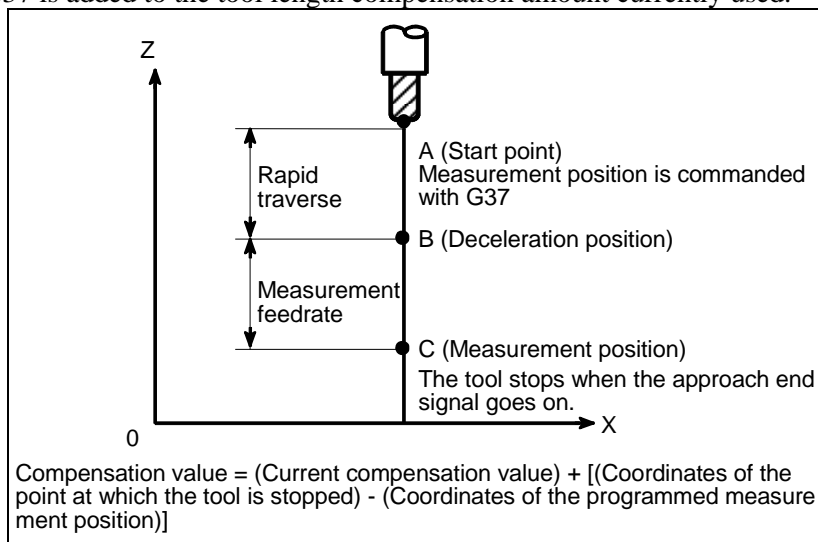


Fig. 6.2 (a) Automatic tool length measurement

Format

G92 IP_ ;	Sets the workpiece coordinate system. (It can be set with G54 to G59. See Chapter "Coordinate System" in OPERATOR'S MANUAL (Common to T/M series.))
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-, Z-, or fourth 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 feedrate halfway, then continues 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 =

$$(\text{Current compensation value}) + [(\text{Coordinates of the position at which the tool reaches for measurement}) - (\text{Coordinates specified by G37})]$$

These offset values can be manually changed from MDI.

- Alarm

When automatic tool length measurement is executed, the tool moves as shown in Fig. 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.

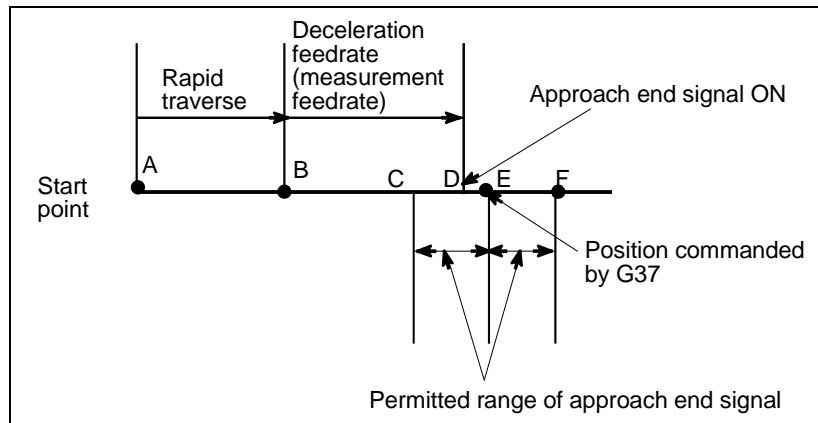


Fig. 6.2 (b) Tool movement to the measurement position

⚠ WARNING

When a manual movement is inserted into a movement at a measurement feedrate, return the tool to the position before the inserted manual movement for restart.

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 ε 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 B is used, the tool wear compensation value is changed. When tool offset memory C is used, the tool wear compensation value for the H code is changed.

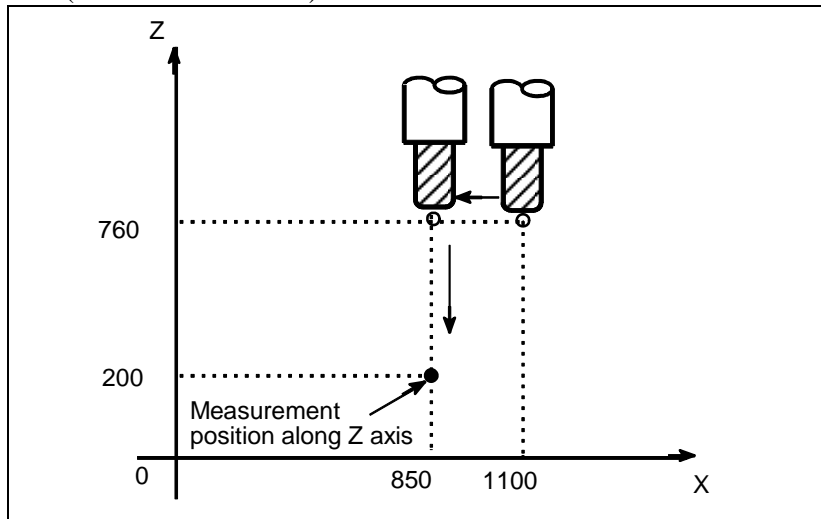
NOTE

- 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 (0.1 msec or less for high-speed measurement position arrival signal input (optional)). 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.

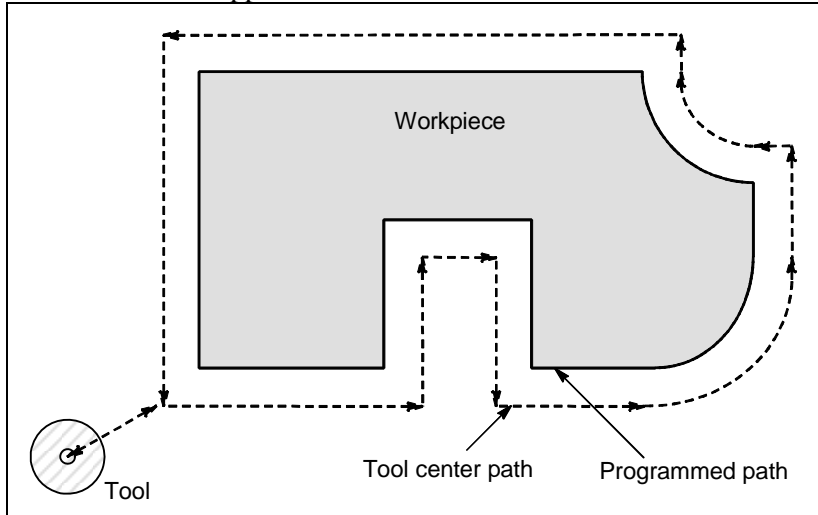
For example, if the tool reaches the measurement position with Z198.0;, the compensation value must be corrected. Because the correct measurement position is at a distance of 200 mm, the compensation value is lessened by 2.0 mm ($198.0 - 200.0 = -2.0$).



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

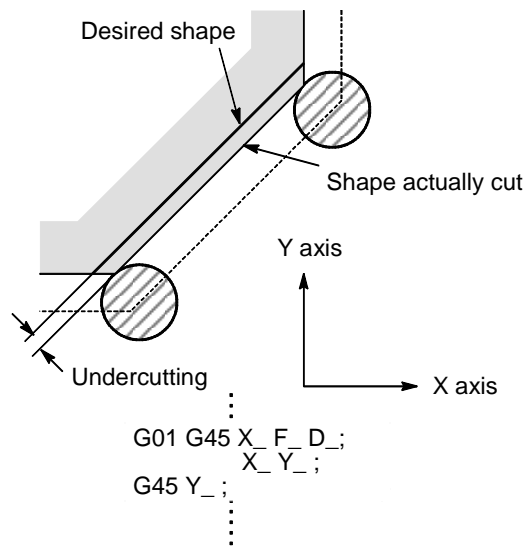
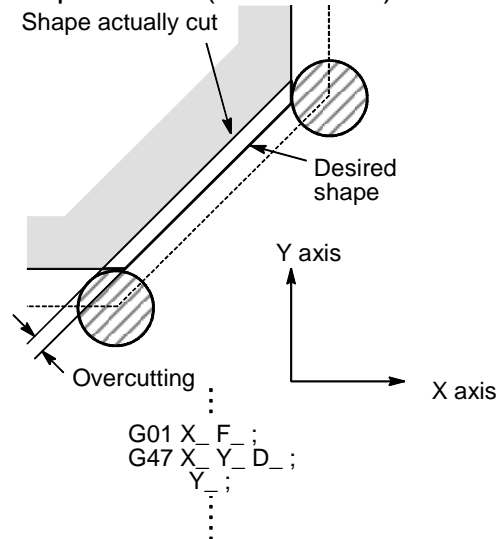
Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.

Tool offset values can be set within the following range:

D0 always indicates a tool offset value of zero.

⚠ CAUTION

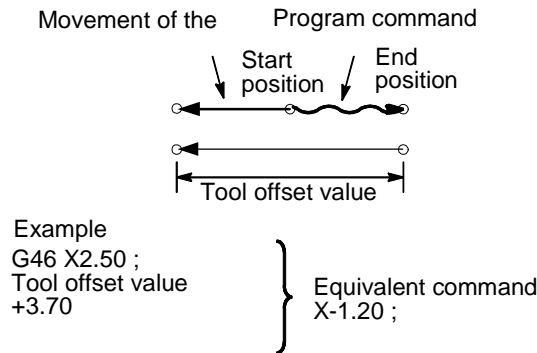
- 1 When G45 to G48 is specified to n axes (n=1-6) simultaneously in a motion block, offset is applied to all n axes. When the cutter is offset only for cutter radius or diameter in taper cutting, overcutting or undercutting occurs. Therefore, use cutter compensation (G40 or G42) shown in II-6.4 or 6.6.



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

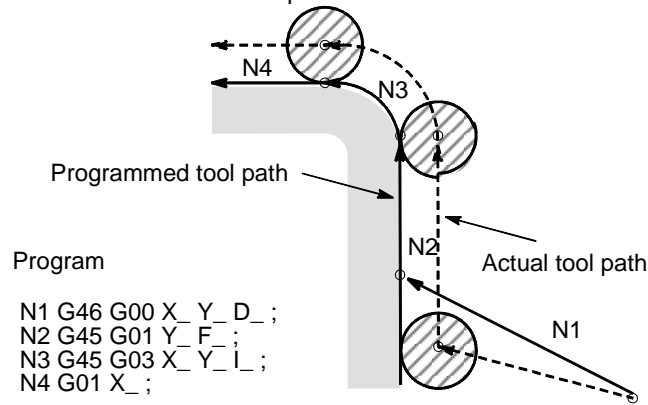
NOTE

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



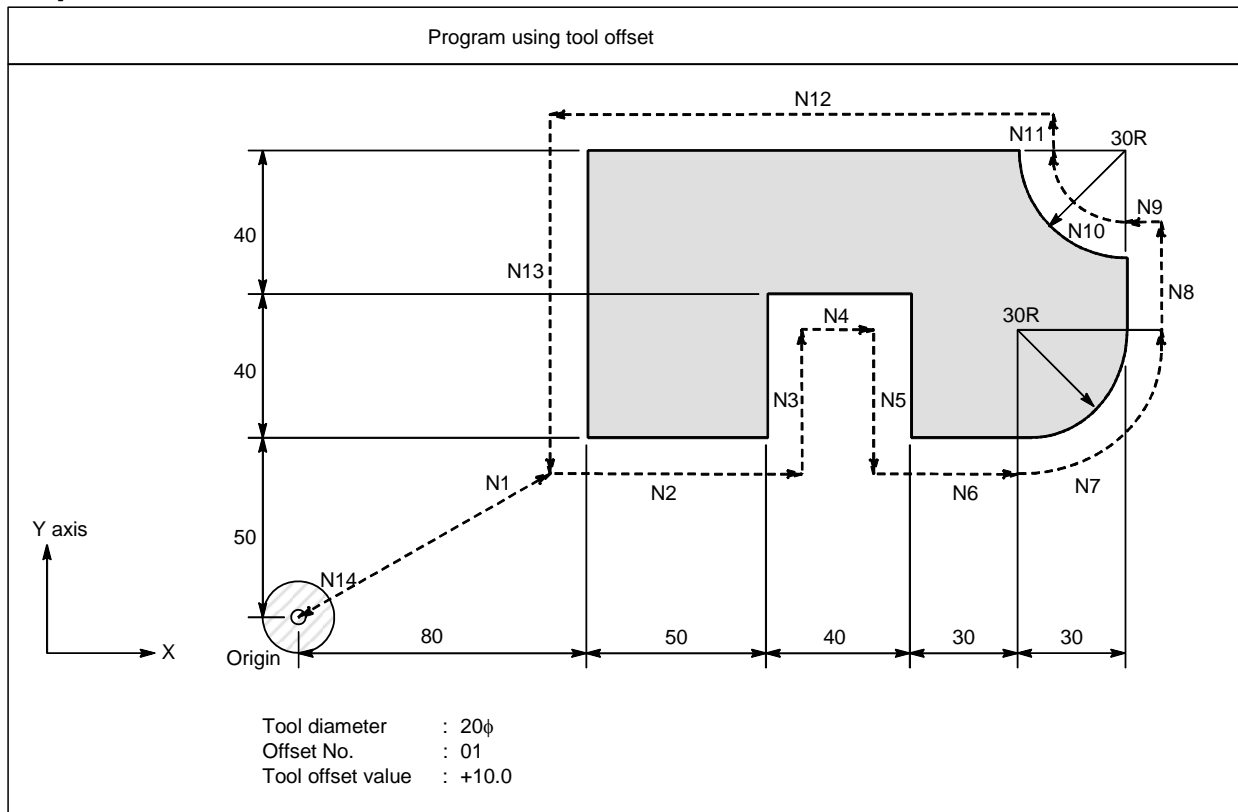
- 2 Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I, J and K by the parameter setting, providing that the coordinate system rotation be not specified at the same time. This function is provided for compatibility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

Tool offset for circular interpolation



- 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

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.

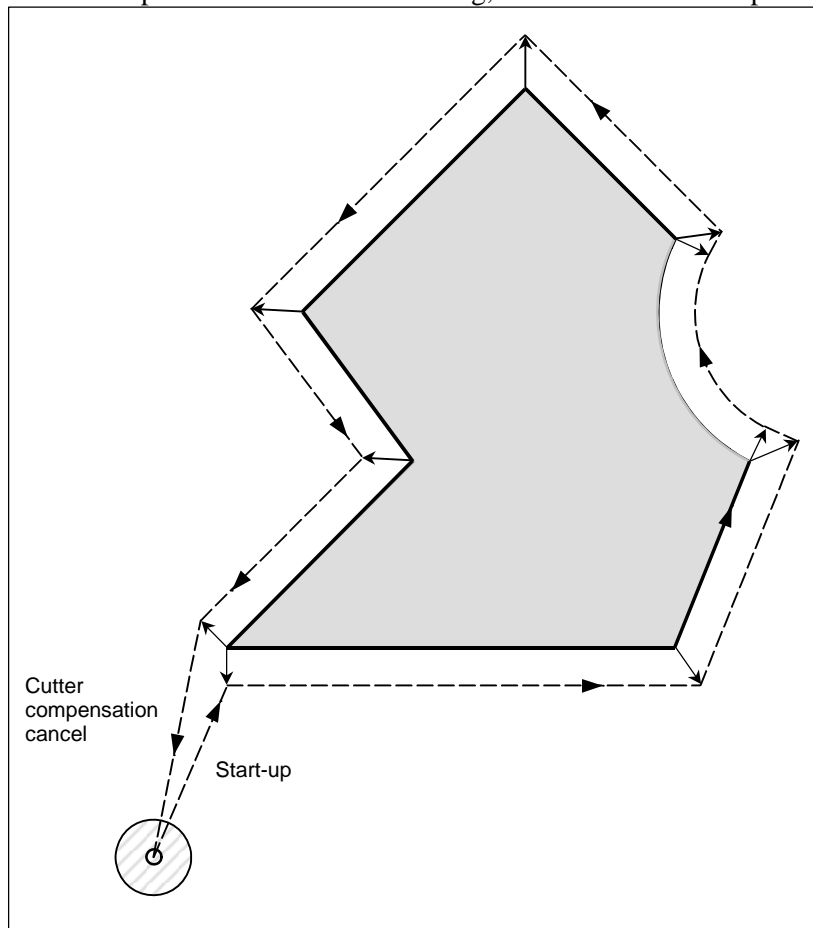


Fig. 6.4 (a) Outline of cutter compensation

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) or involute interpolation (G02.2, G03.2) is specified, alarm PS0034 occurs.

For the start-up and subsequent blocks, the CNC prereads as many blocks as the number of preread blocks set in the parameter No. 19625.

- Offset mode

In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03).

If three or more blocks that move the tool cannot be read in offset mode, the tool may make either an excessive or insufficient cut.

If the offset plane is switched in the offset mode, alarm PS0037 occurs and the tool is stopped.

- Offset mode cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the CNC enters the offset cancel mode, and the action of this block is called the offset cancel.

1. G40 has been commanded.
2. 0 has been commanded as the offset number for cutter compensation (D code).

When performing offset cancel, circular arc commands (G02 and G03) and involute commands (G02.2 and G03.2) are not available. If these commands are specified, an PS0034 is generated and the tool stops. In the offset cancel, the control executes the instructions in that block and the block in the 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.

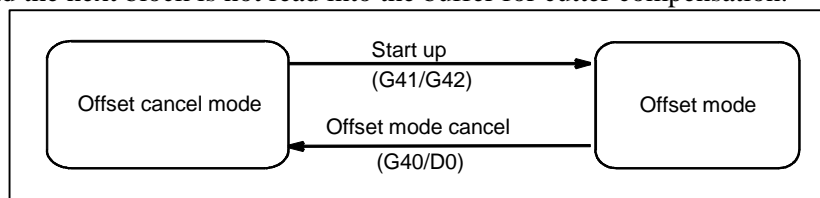


Fig. 6.4 (b) Changing the offset mode

- 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.

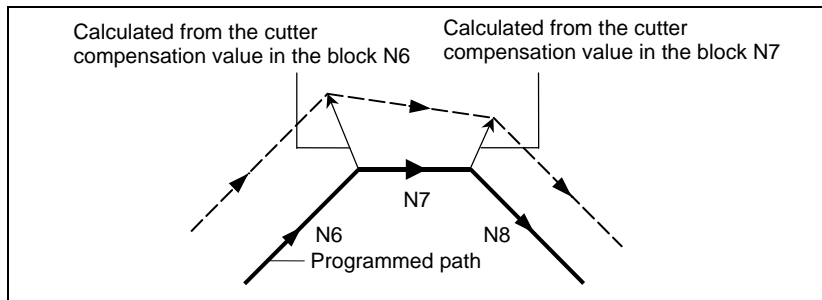


Fig. 6.4 (c) Changing the cutter compensation value

- 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.)

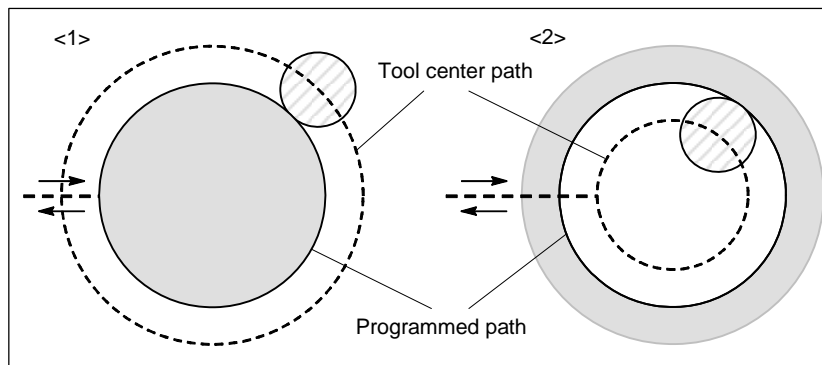


Fig. 6.4 (d) Tool center paths when positive and negative cutter compensation values are specified

- Cutter compensation value setting

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

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

OFE	OFD	OFC	OFA	Range
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

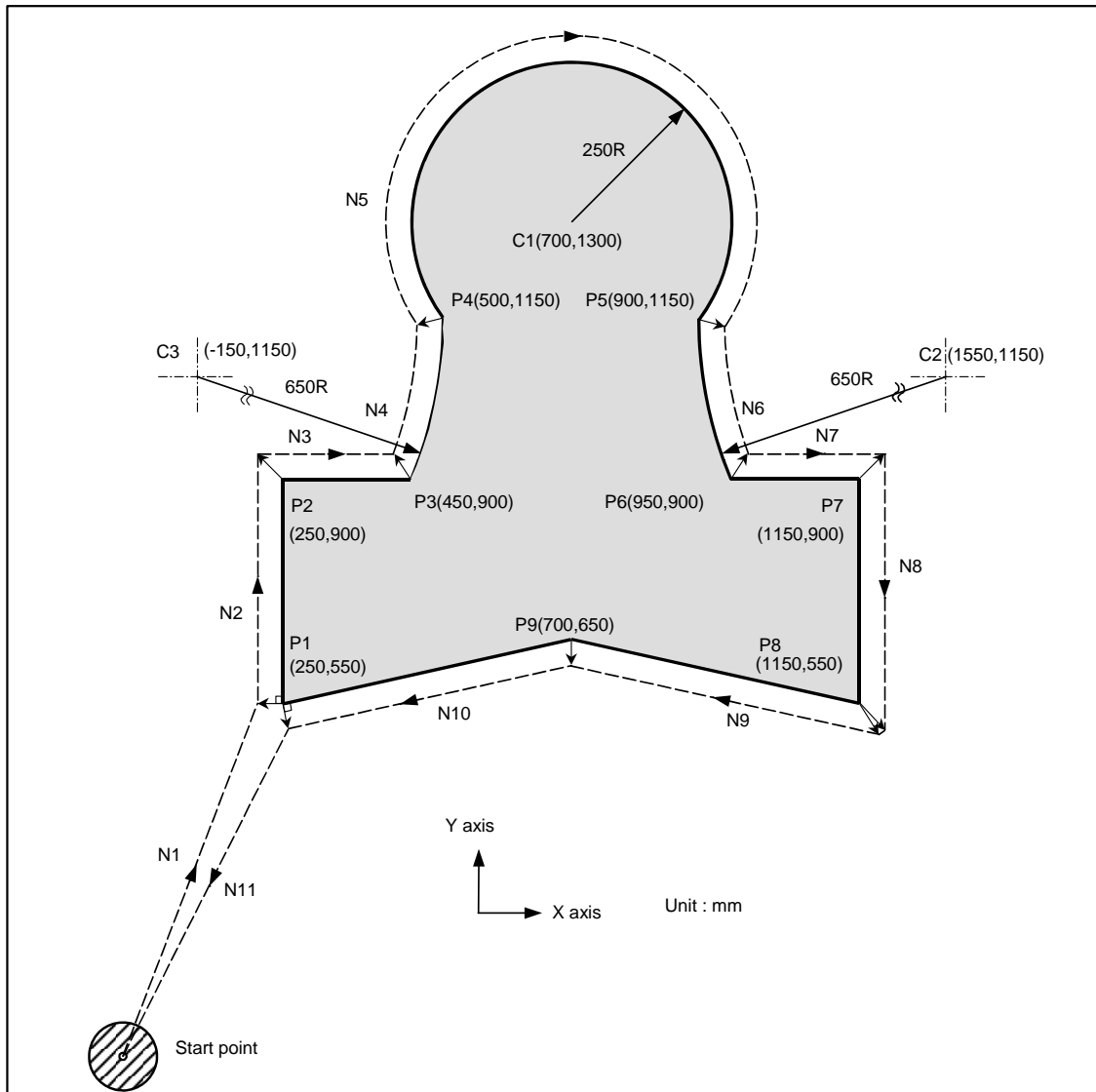
Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane.

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, an PS0037 is displayed and the machine is stopped.

Example



G17 G92 X0 Y0 Z0 ;..... Specifies absolute coordinates.

The tool is positioned at the start point (X0, Y0, Z0).

N1 G90 G00 G41 D07 X250.0 Y550.0 ;.... Starts cutter compensation (start-up).

The tool is shifted to the left of the programmed path by the distance specified in D07.

In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15 mm).

N2 G01 Y900.0 F150 ;..... Specifies machining from P1 to P2.

N3 X450.0 ;..... Specifies machining from P2 to P3.

N4 G03 X500.0 Y1150.0 R650.0 ; Specifies machining from P3 to P4.

N5 G02 X900.0 R-250.0 ; Specifies machining from P4 to P5.

N6 G03 X950.0 Y900.0 R650.0 ; Specifies machining from P5 to P6.

N7 G01 X1150.0 ;..... Specifies machining from P6 to P7.

N8 Y550.0 ;..... Specifies machining from P7 to P8.

N9 X700.0 Y650.0 ;..... Specifies machining from P8 to P9.

N10 X250.0 Y550.0 ;..... Specifies machining from P9 to P1.

N11 G00 G40 X0 Y0 ;..... Cancels the offset mode.

The tool is returned to the start point (X0, Y0, Z0).

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.

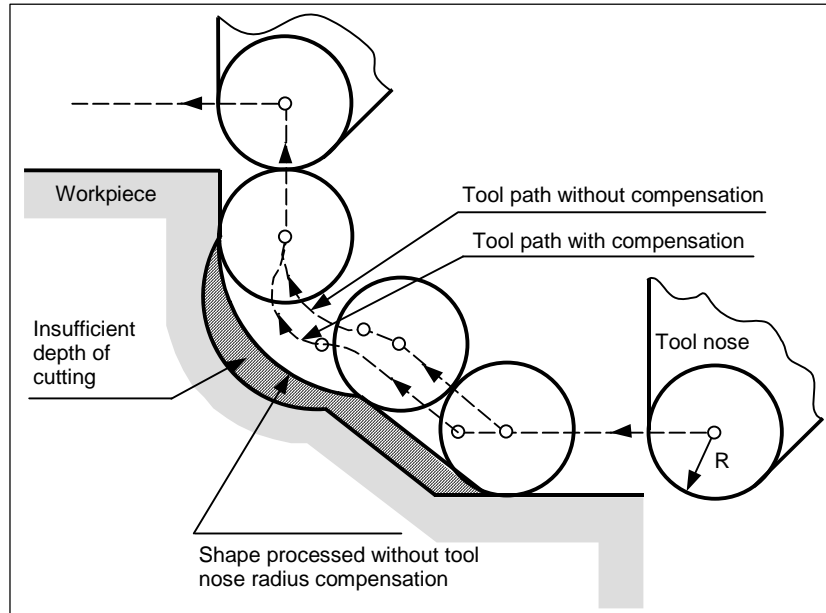


Fig. 6.5 (a) Tool path of tool nose radius compensation

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).

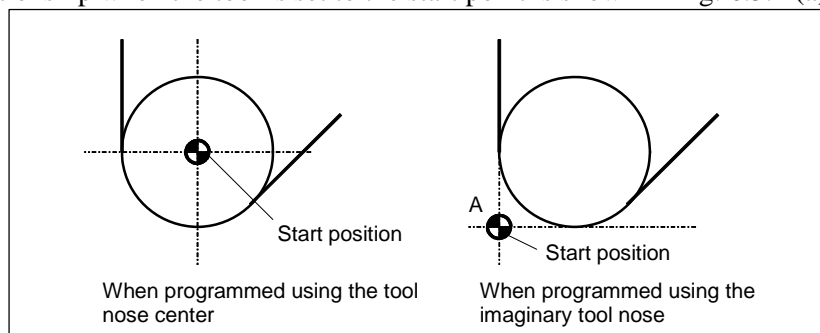


Fig. 6.5.1 (a) Tool nose radius center and imaginary tool nose

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.

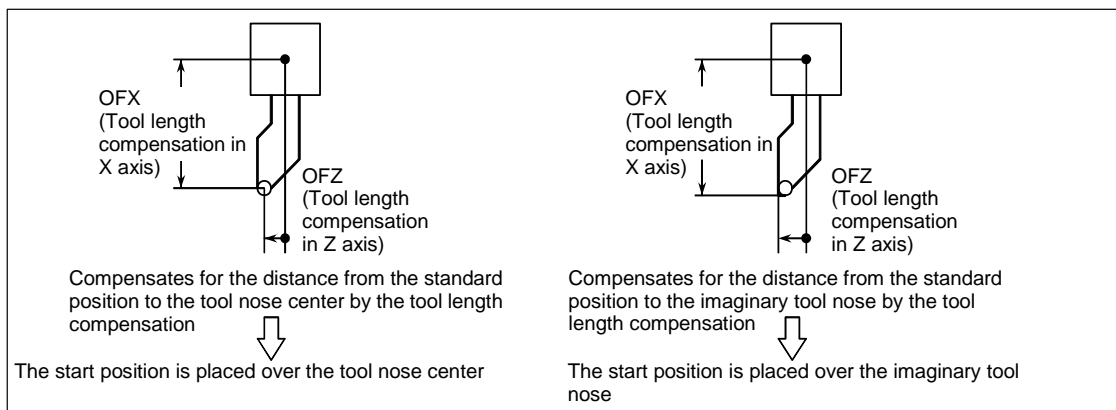


Fig. 6.5.1 (b) Tool length compensation when the turret center is placed over the start point

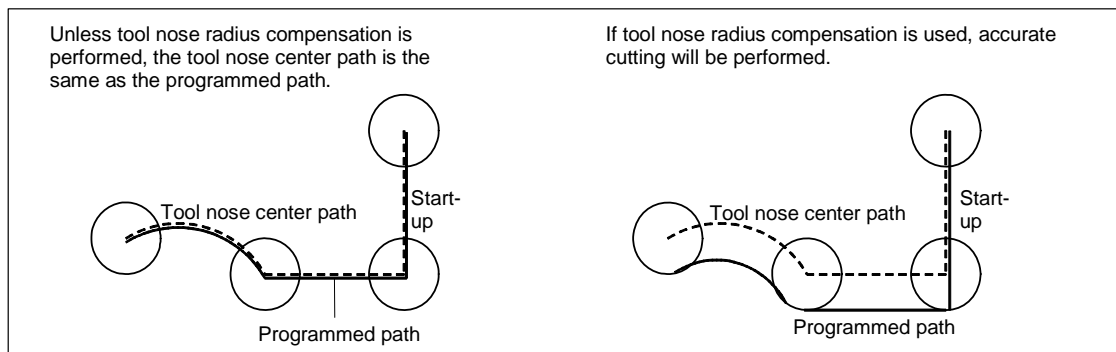


Fig. 6.5.1 (c) Tool path when programming using the tool nose center

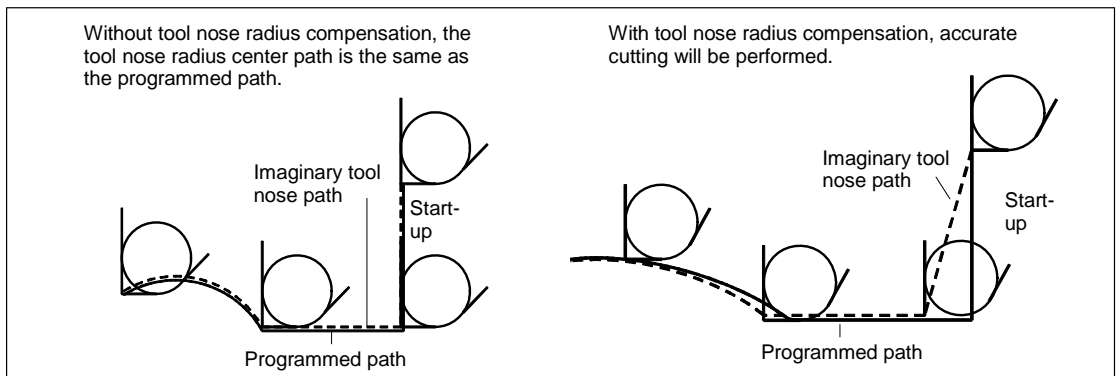


Fig. 6.5.1 (d) Tool path when programming using the imaginary tool nose

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.

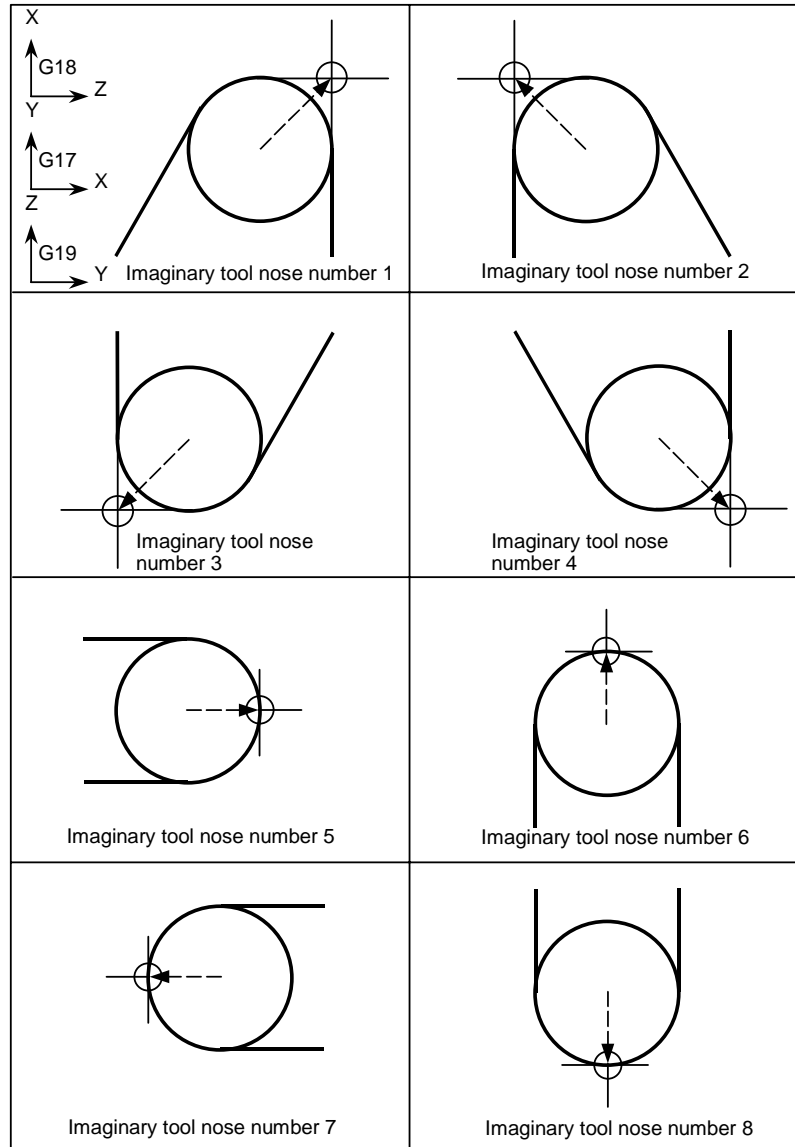
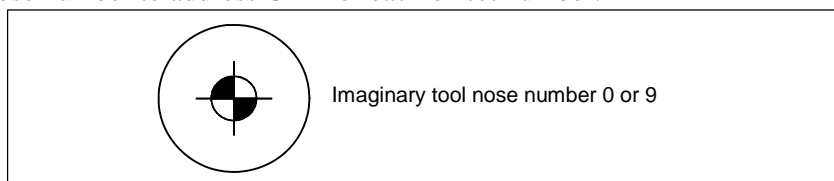


Fig. 6.5.2 (a) Direction of imaginary tool nose

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

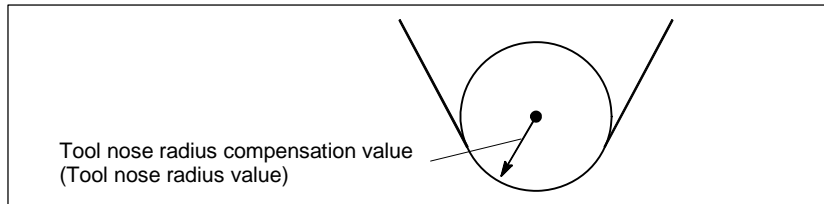


Table 6.5.3 (a) Offset number and offset value (example)

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

G code	Workpiece position	Tool 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.

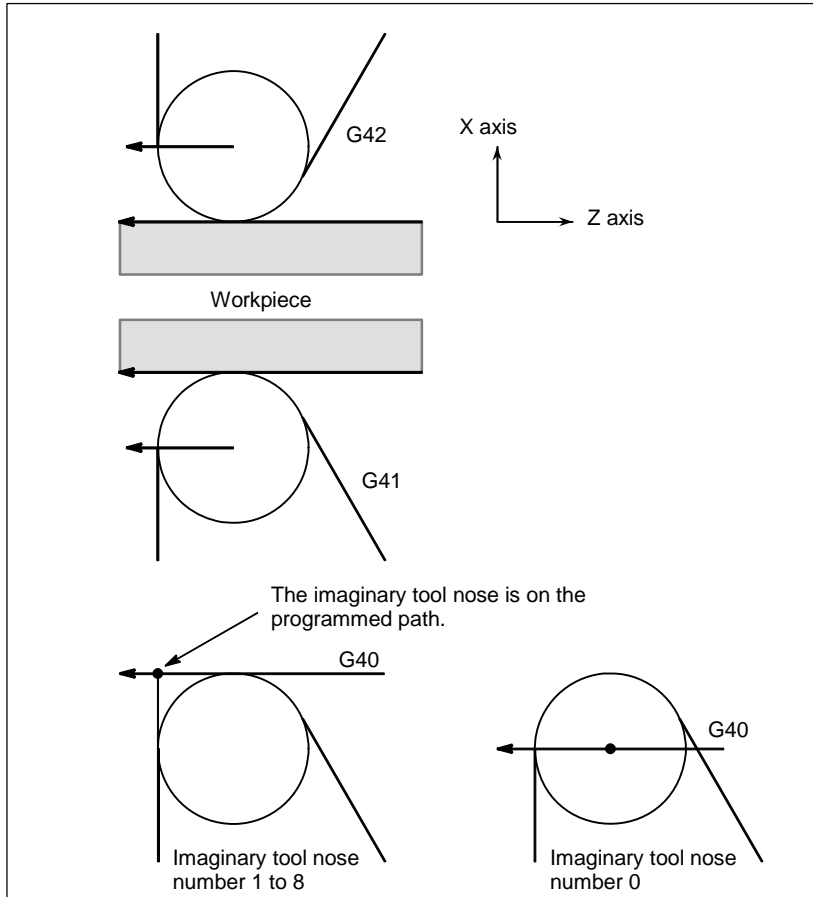


Fig. 6.5.4 (a) Workpiece position

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

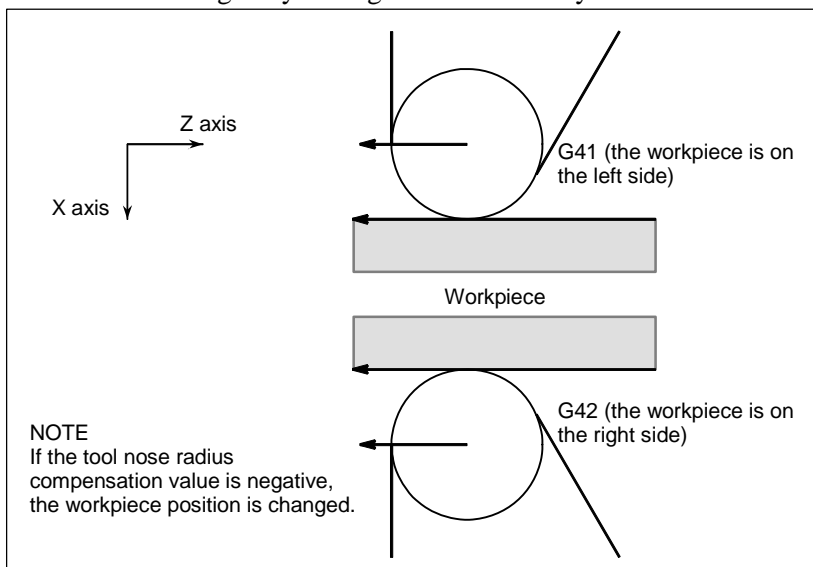


Fig. 6.5.4 (b) When the workpiece position is changed

G40, G41, and, G42 are modal.

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

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

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

⚠ CAUTION

If the sign of the compensation value is changed from plus to minus and vice versa, the offset vector of tool nose radius compensation is reversed, but the direction of the imaginary tool 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.

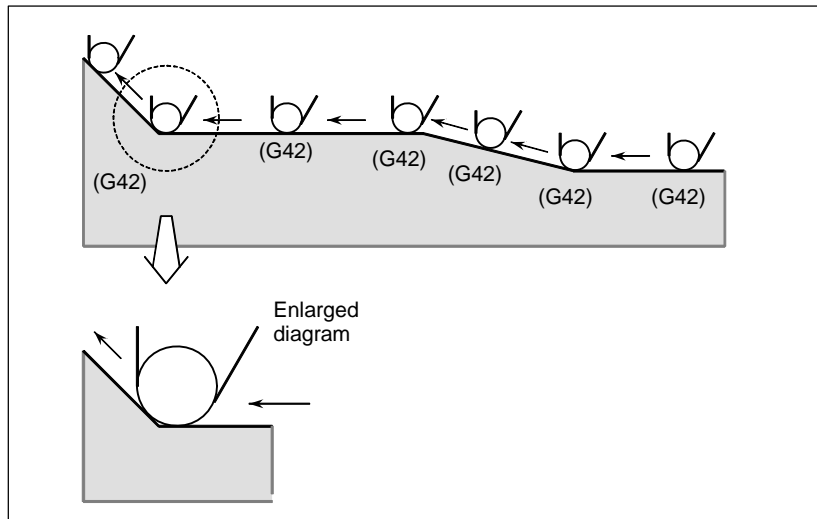


Fig. 6.5.4 (c) Tool movement when the workpiece position does not change

- 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.

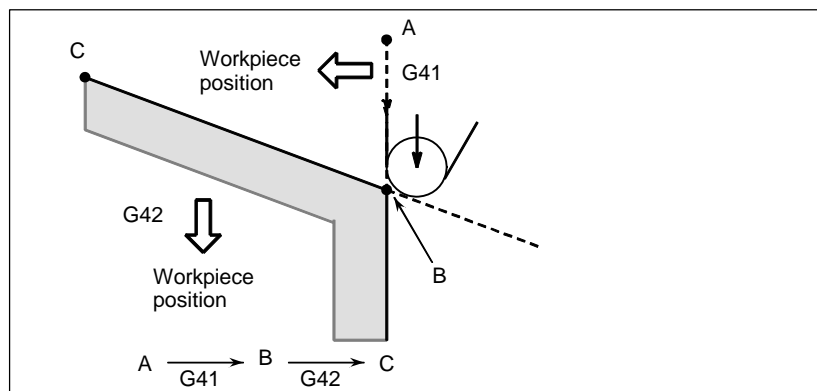


Fig. 6.5.4 (d) Tool movement when the workpiece position changes

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 (b), 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.

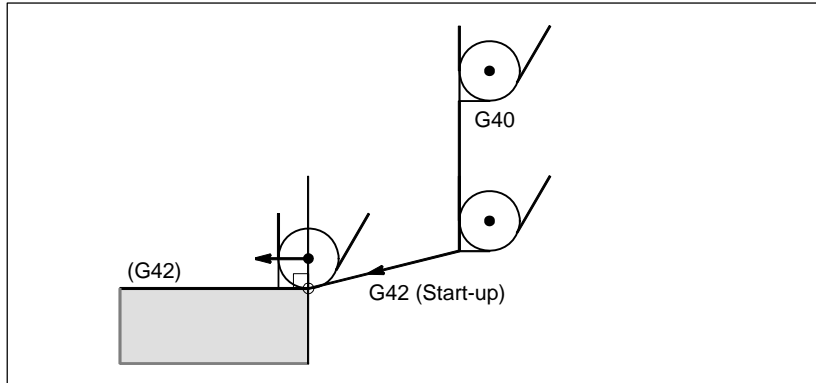


Fig. 6.5.4 (e) Start-up

- 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.

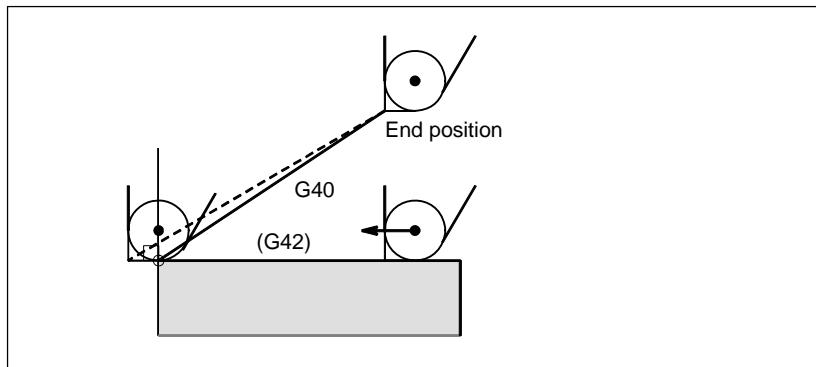


Fig. 6.5.4 (f) Offset cancel

- 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.

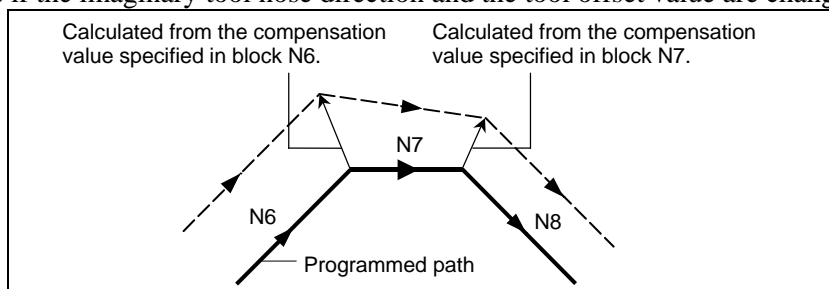


Fig. 6.5.4 (g) Changing the compensation value

- 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.

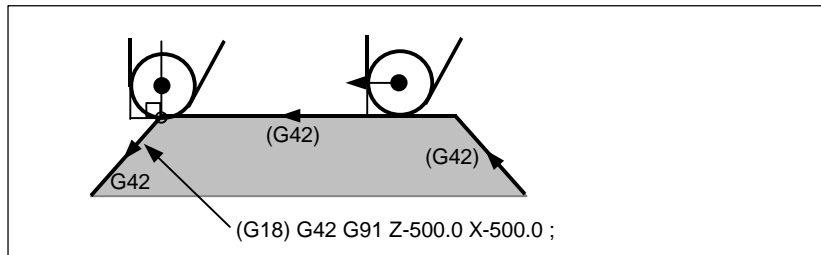


Fig. 6.5.4 (h) Specification of G41/G42 in G41/G42 mode

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.

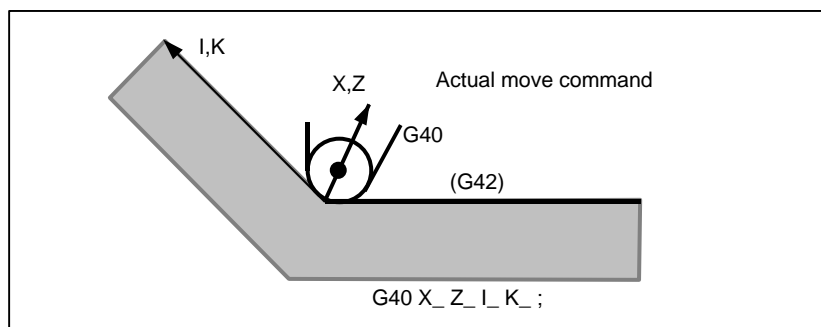


Fig. 6.5.4 (i) If I and K are specified in the same block as G40

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

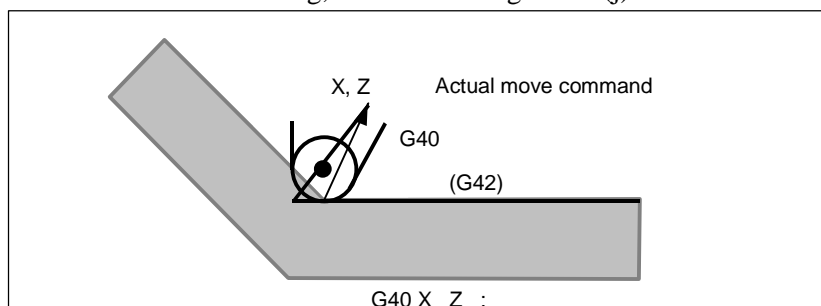


Fig. 6.5.4 (j) Case in which overcutting occurs in the same block as G40

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.

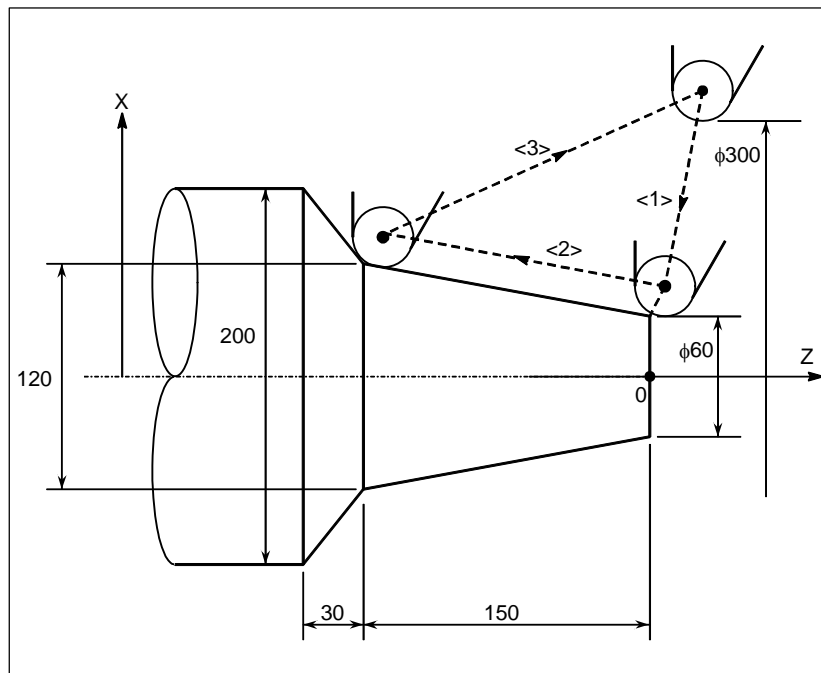
G40 X_ Z_ I_ K_ ;	Tool nose radius compensation
G02 X_ Z_ I_ K_ ;	Circular interpolation

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 ;	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.

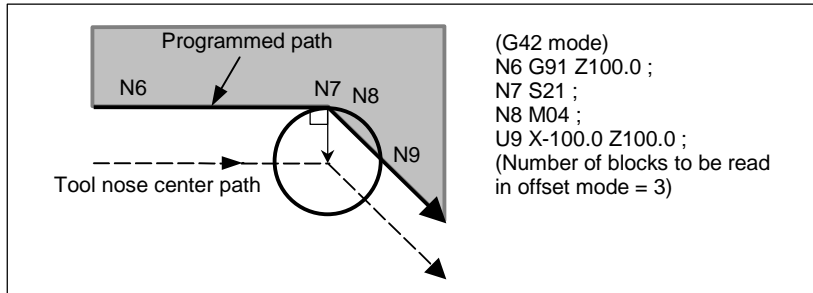


Fig. 6.5.5 (a)

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).

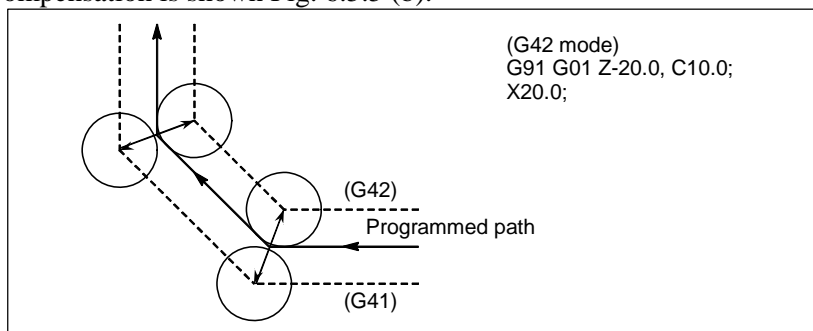


Fig. 6.5.5 (b)

- **Tool nose radius compensation when a corner R is performed**

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

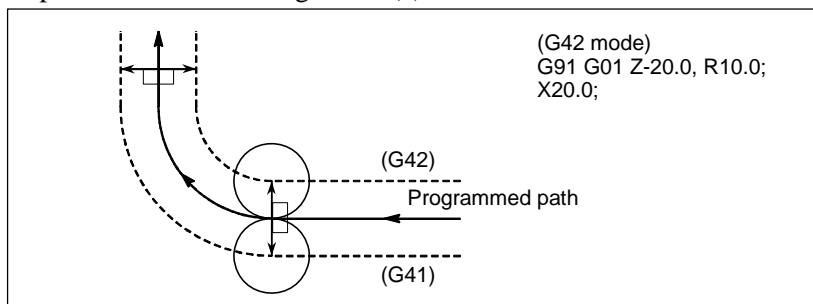


Fig. 6.5.5 (c)

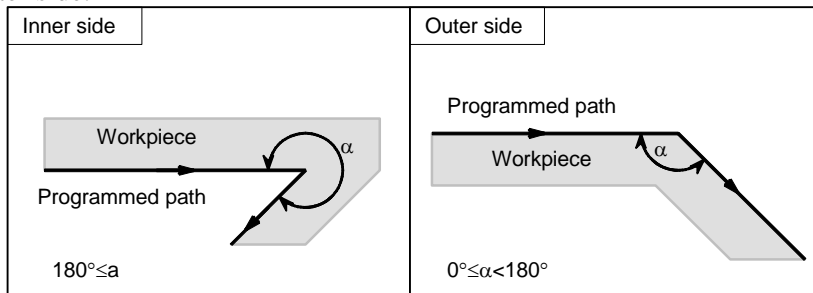
6.6 DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION

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

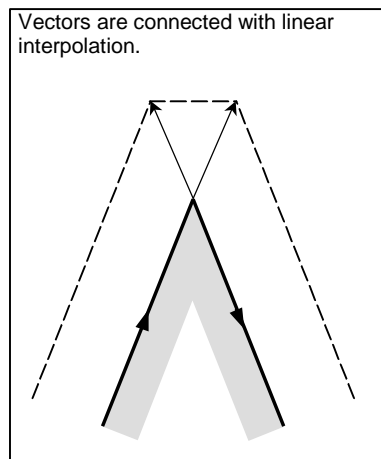
When an angle of intersection of the tool paths specified with move commands for two blocks on the workpiece side is over 180° , it is referred to as "inner side." When the angle is between 0° and 180° , it is referred to as "outer side."



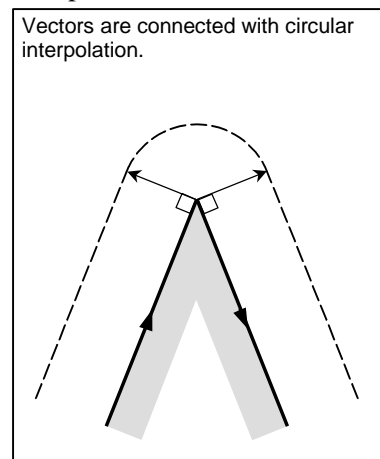
- Outer corner connection method

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  key on the MDI panel 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> $0 < \text{compensation number of cutter compensation} \leq \text{maximum compensation number}$
- <3> Positioning (G00) or linear interpolation (G01) mode
- <4> A compensation plane axis command with a travel distance of 0 (except start-up type C) is specified.

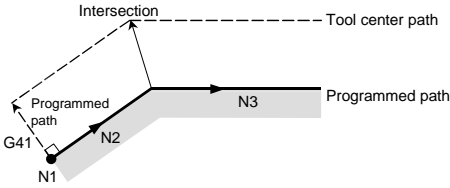
If start-up is specified in circular interpolation (G02, G03) mode, PS0034 will occur.

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

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

Table 6.6.1 (a) Start-up/cancel operation

SUV	SUP	Type	Operation
0	0	Type A	<p>A compensation vector is output, which is vertical to the block subsequent to the start-up block and the block preceding the cancel block.</p>
0	1	Type B	<p>A compensation vector is output, which is vertical to the start-up block and the cancel block. An intersection vector is also output.</p>

SUV	SUP	Type	Operation
1	0 1	Type C	<p>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.</p>  <p>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.</p>

- 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.

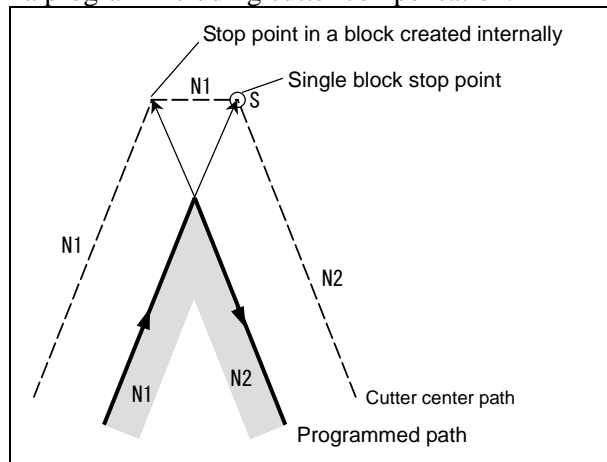
If cutter compensation cancel is to be performed, it must not be by a circular command (G02, G03). Otherwise, an alarm will occur.

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:

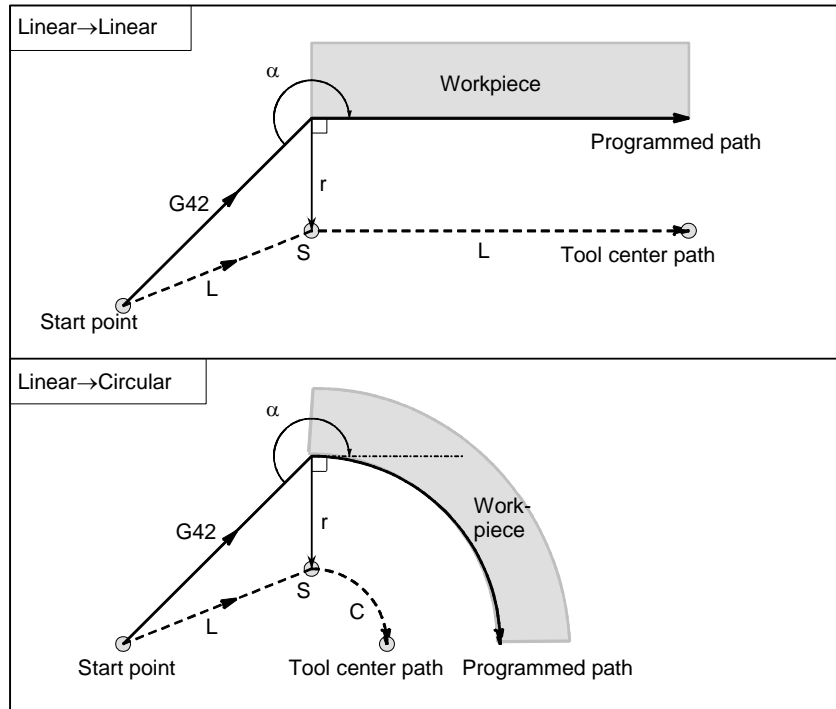
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves 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):

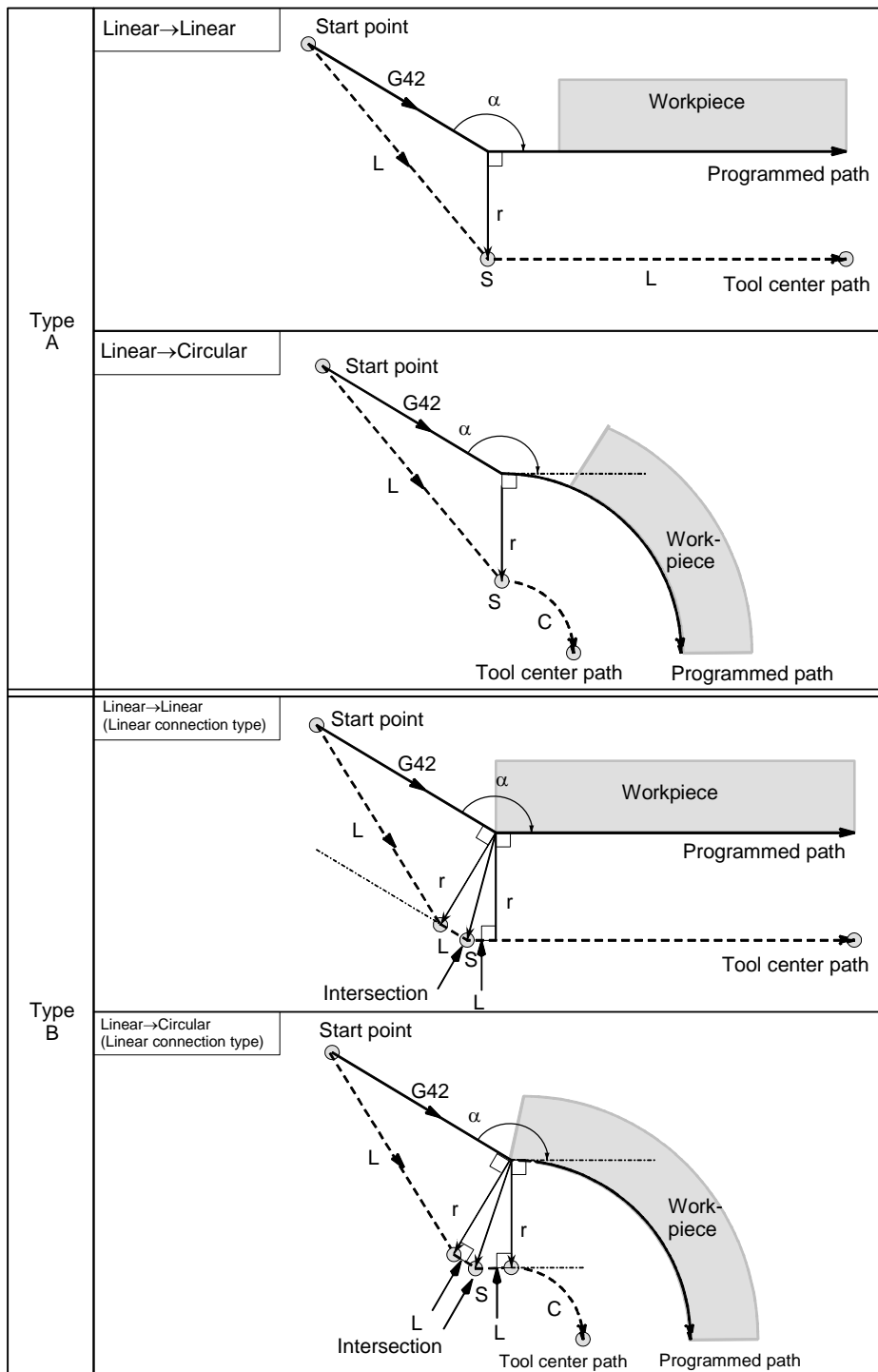
Explanation

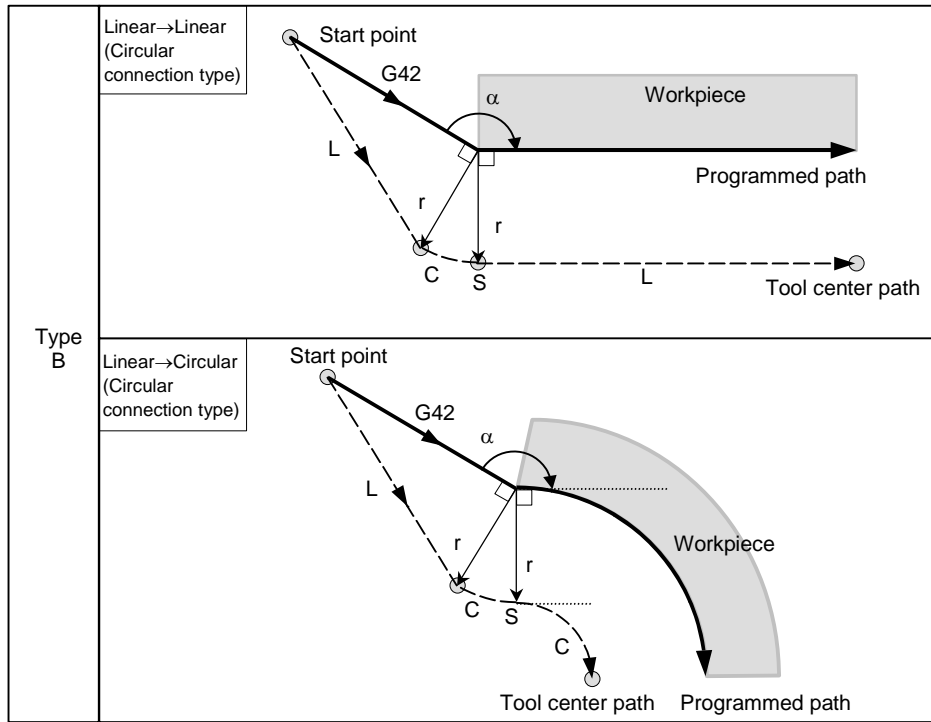
- Tool movement around an inner side of a corner ($180^\circ \leq \alpha$)



- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)

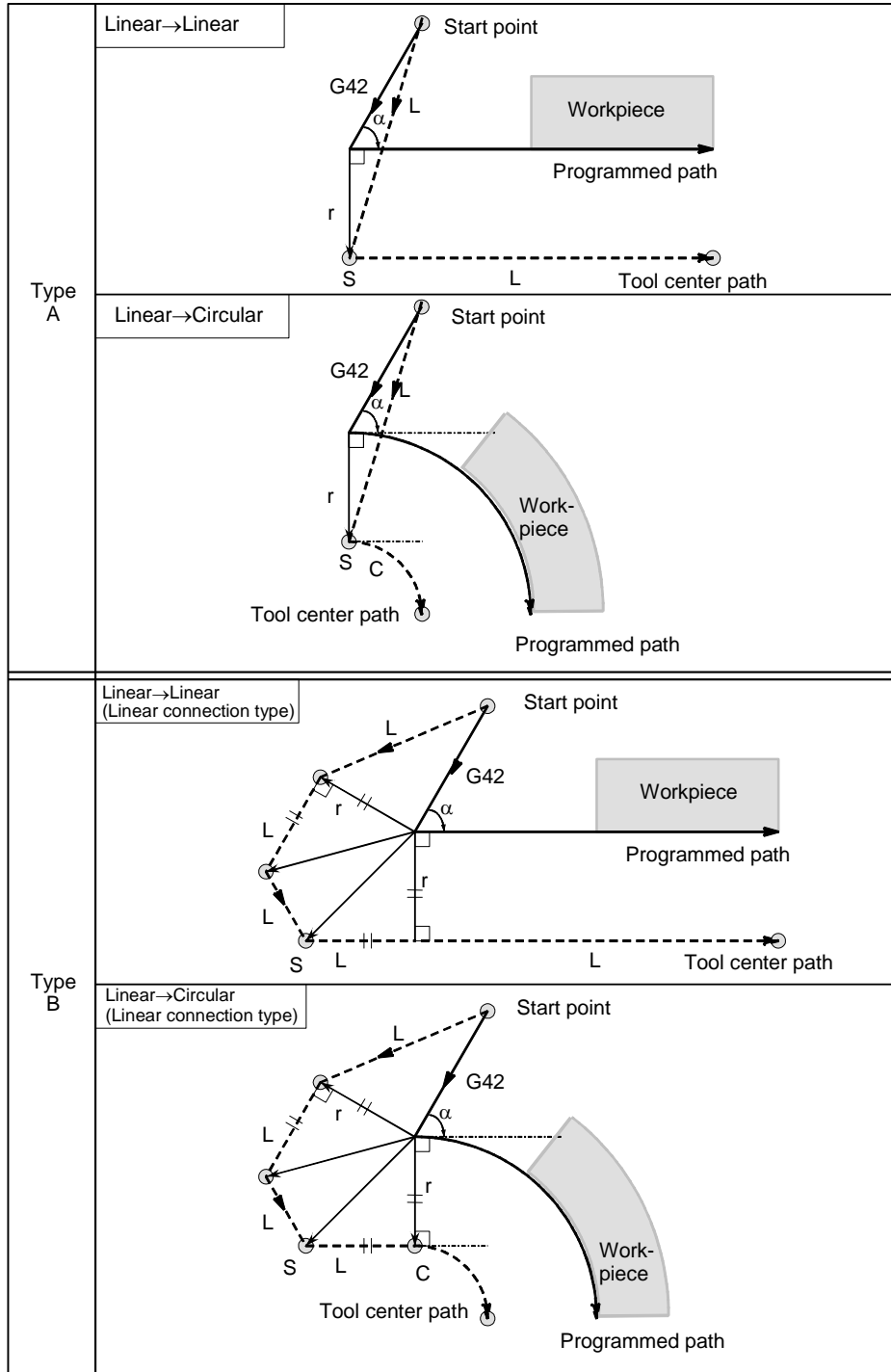
Tool path in start-up has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

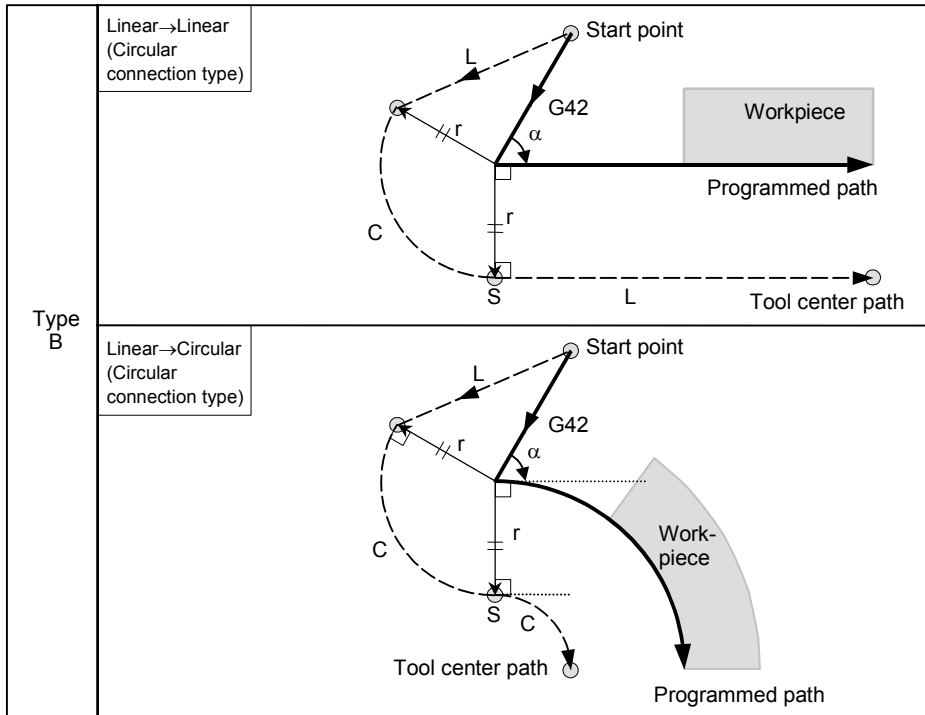




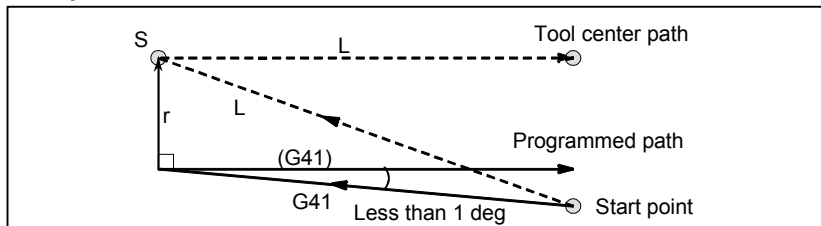
- **Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle ($\alpha < 90^\circ$)**

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





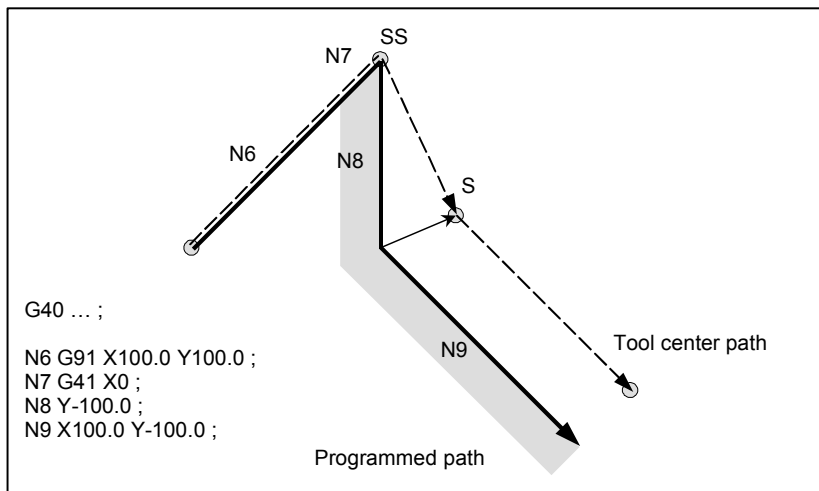
- **Tool movement around the outside linear → linear at an acute angle less than 1 degree ($\alpha < 1^\circ$)**



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

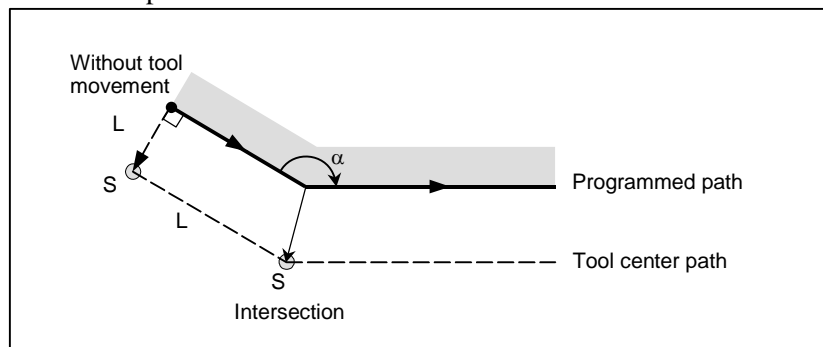
For type A and B

If the command is specified at start-up, the offset vector is not created. The tool does not operate in a start-up block.



For type C

The tool shifts by the compensation value in the direction vertical to the block with tool movement subsequent to the start-up block.



6.6.3 Tool Movement in Offset Mode

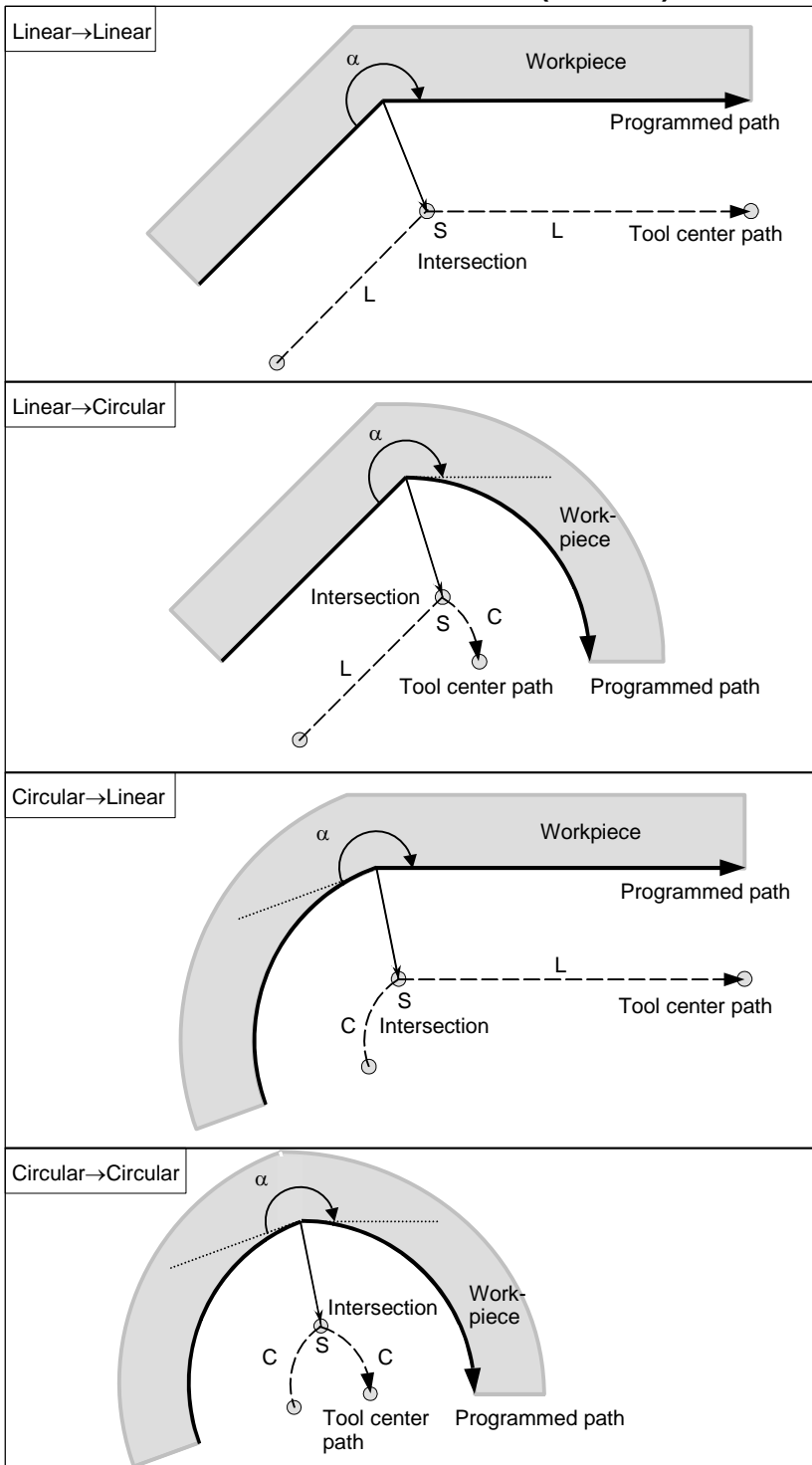
In offset mode, compensation is performed even for positioning commands, not to speak of linear and circular interpolations. To perform intersection calculation, it is necessary to read at least two blocks with tool movement. If, therefore, two or more blocks with tool movement cannot be read in offset 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 $(N - 2) \geq M$. For example, if the maximum number of blocks to read in offset mode is 5, intersection calculation is possible even if up to three blocks without tool movement are specified.

NOTE

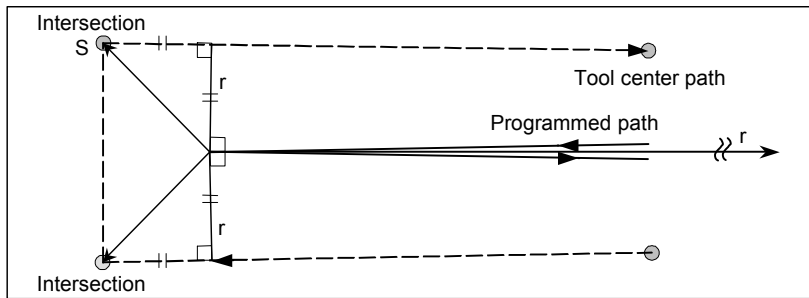
The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the interference check.

If a G or M code in which buffering is suppressed is specified, no subsequent commands can be read before that block is executed, regardless of the setting of parameter No. 19625. Excessive or insufficient cutting may, therefore, occur because of an intersection calculation failure.

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

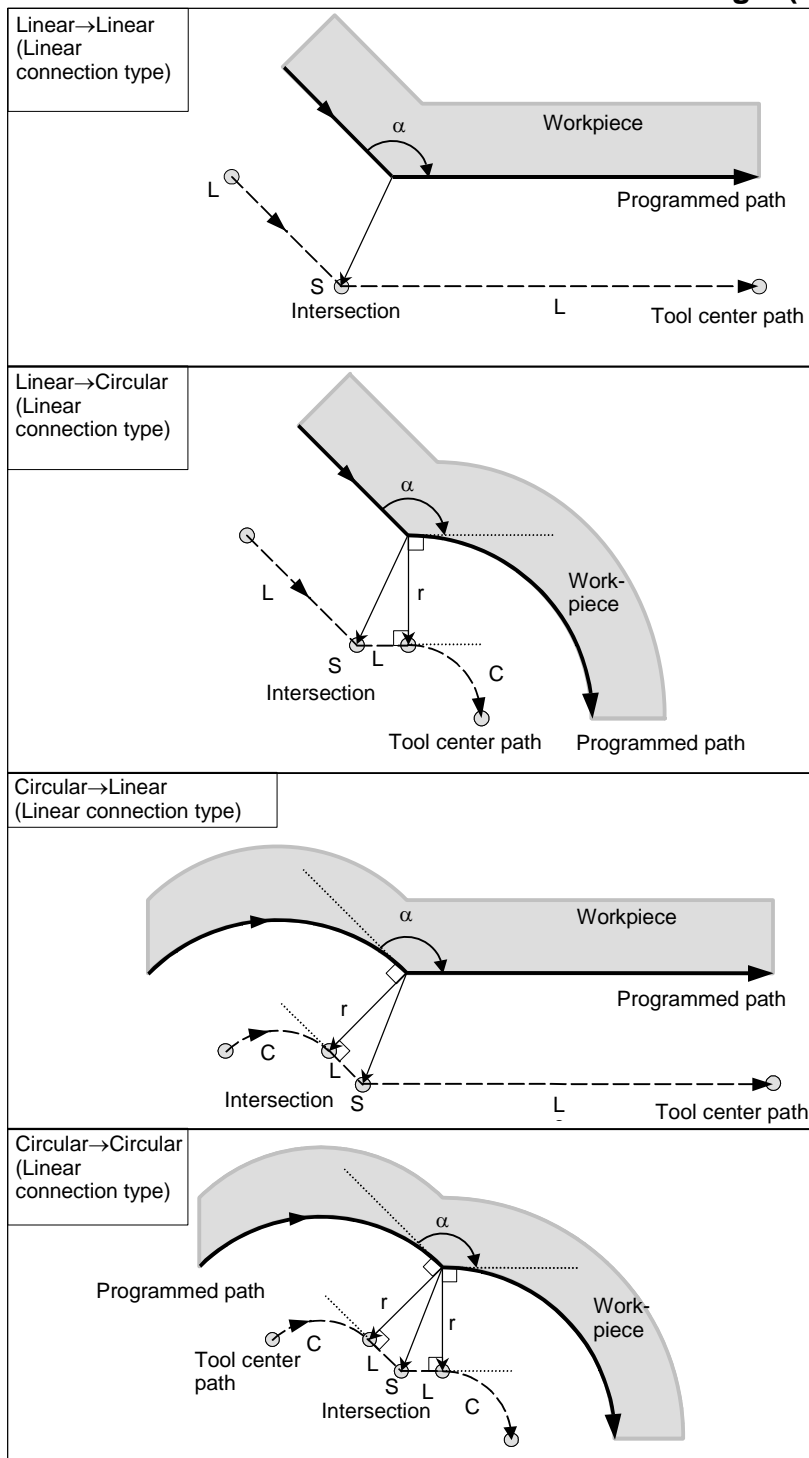


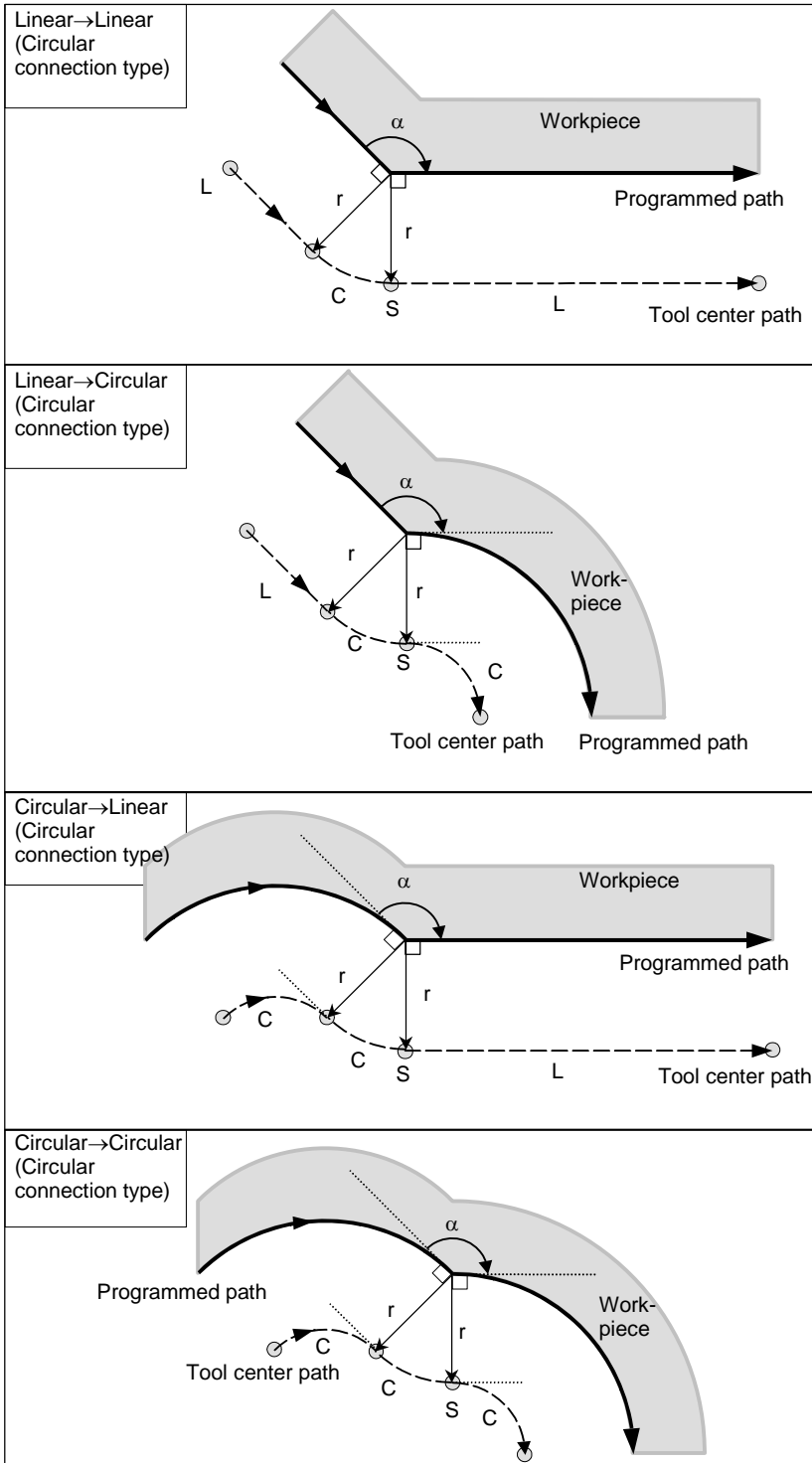
- **Tool movement around the inside ($\alpha < 1^\circ$) with an abnormally long vector, linear \rightarrow linear**



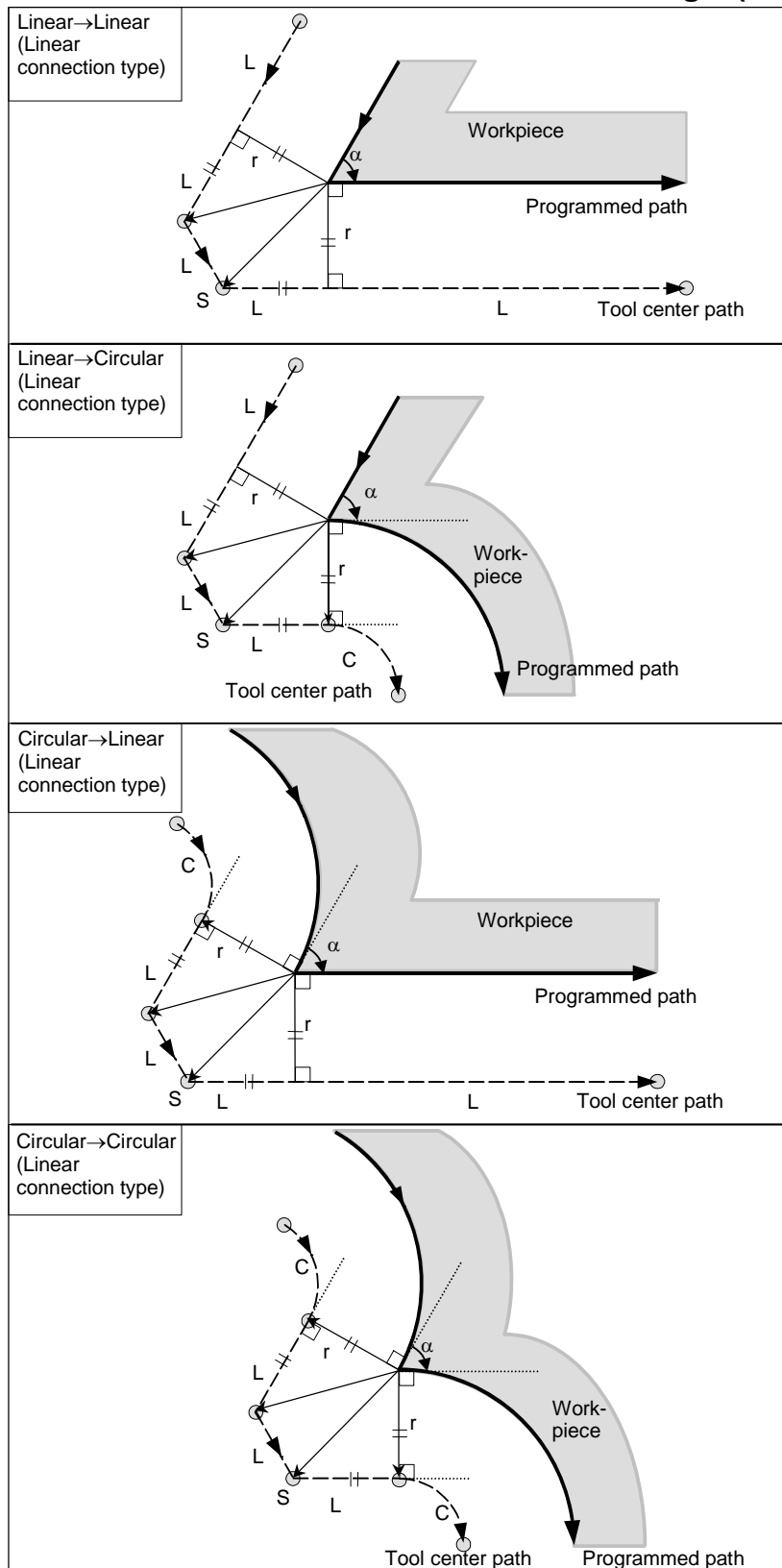
Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

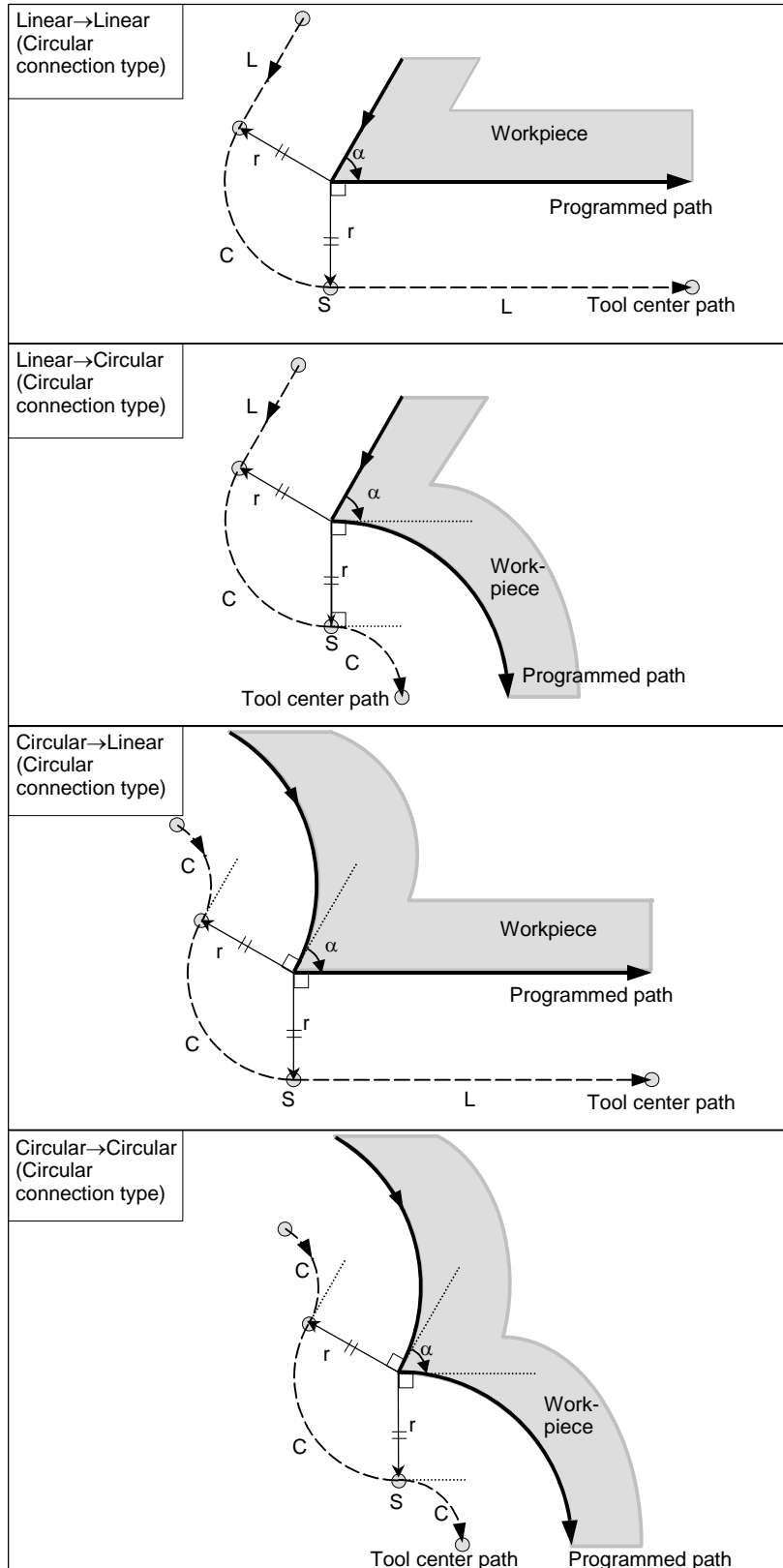
- Tool movement around the outside corner at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)





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

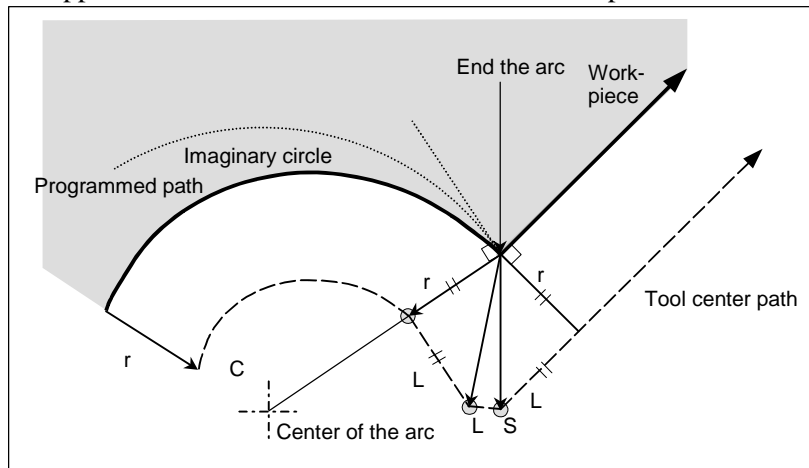




- When it is exceptional

End point for the arc is not on the arc

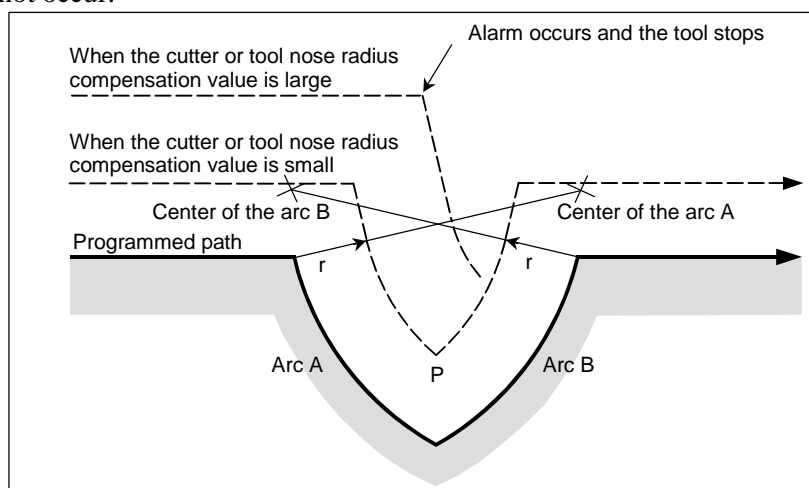
If the end of a line leading to an arc is not on the arc, the system assumes that the cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end point. Based on this assumption, the system creates a vector and carries out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



There is no inner intersection

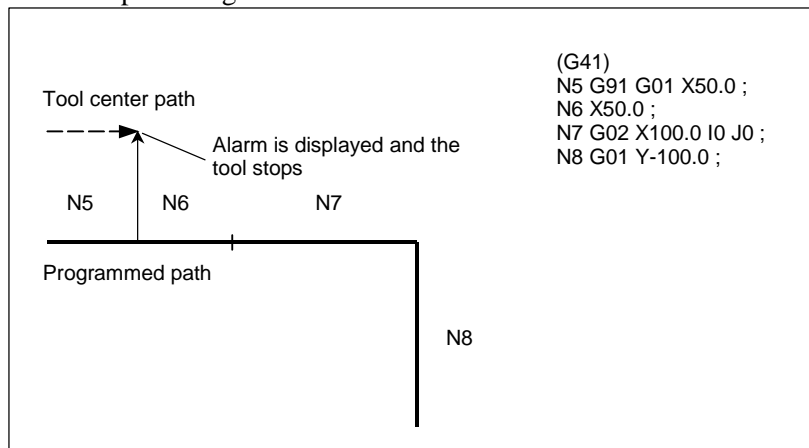
If the tool radius / tool nose radius compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool radius · tool nose radius compensation. When this is predicted, PS0033 occurs at the end of the previous block and the tool is stopped.

For example, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool radius · 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, PS0041 is displayed, and the tool will stop at the start point of the preceding block of the arc.



- Change in the offset direction in the offset mode

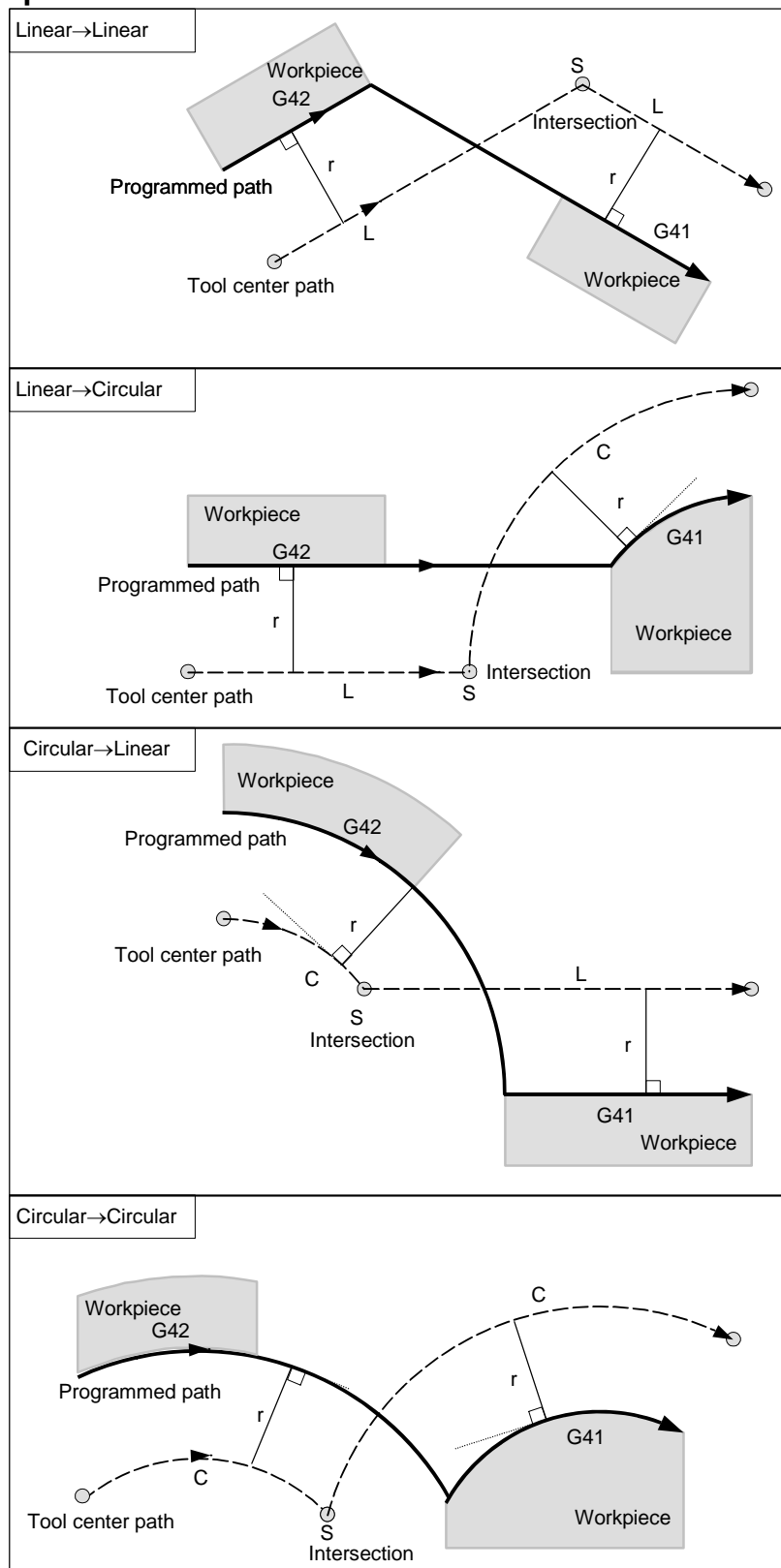
The offset direction is decided by G codes (G41 and G42) for tool radius · tool nose radius compensation and the sign of the compensation value as follows.

G code	Sign of compensation	+	-
	G41		Left 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.

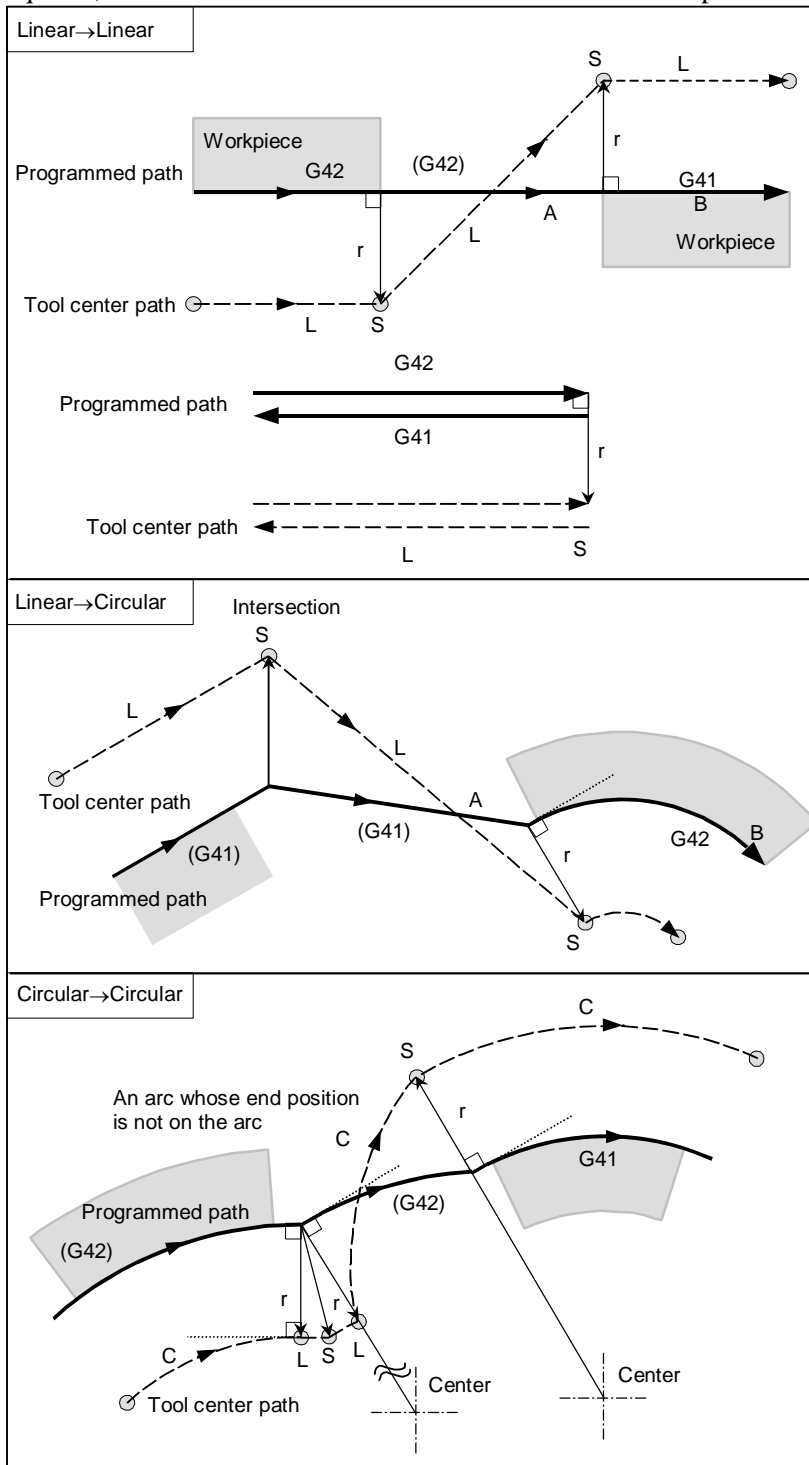
However, the change is not available in the start-up block and the block following it.

- Tool center path with an intersection



- Tool center path without an intersection

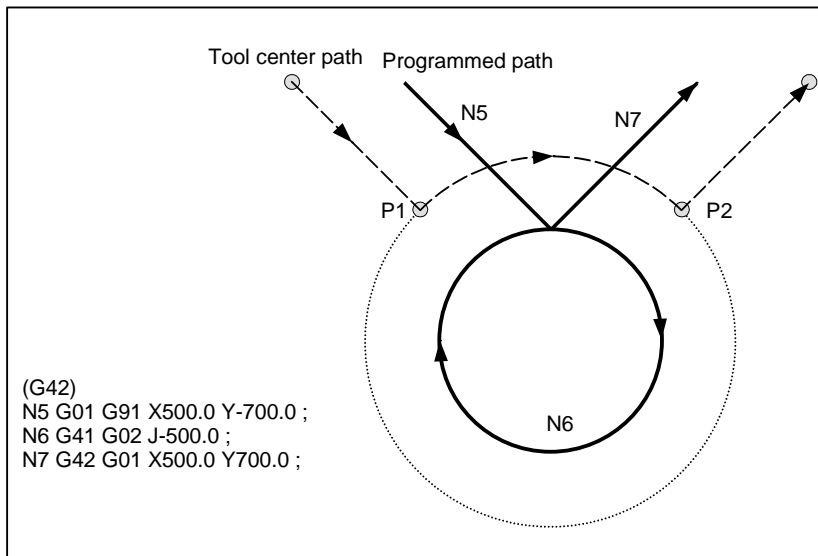
When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.



- 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.

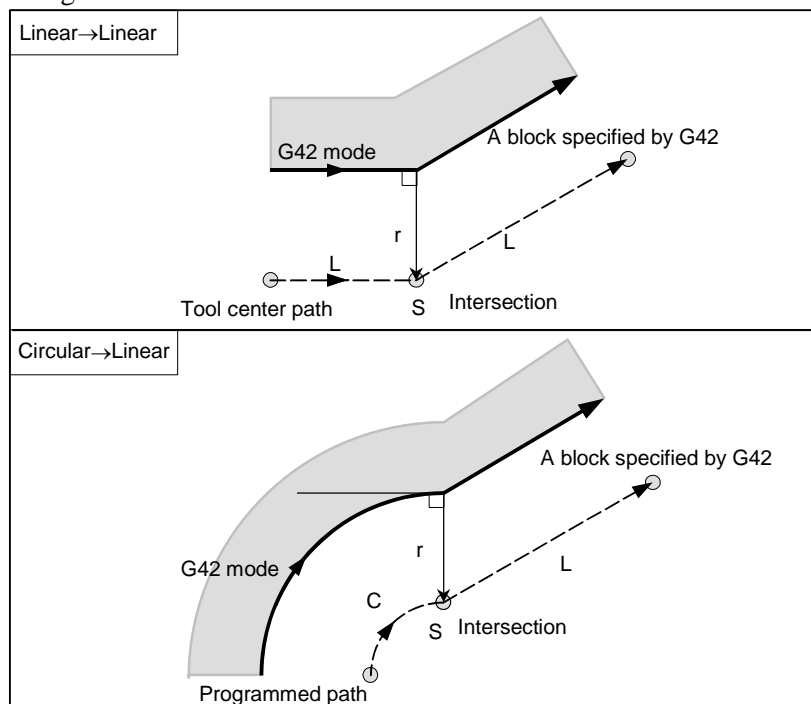
The cutter compensation is not performed with more than one circle circumference: an arc is formed from P₁ to P₂ as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.



- 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".

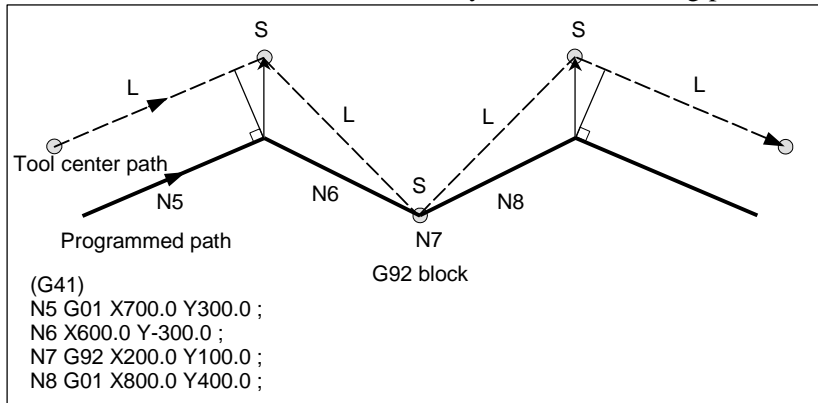


- Command canceling the offset vector temporarily

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), G30.1 (floating 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

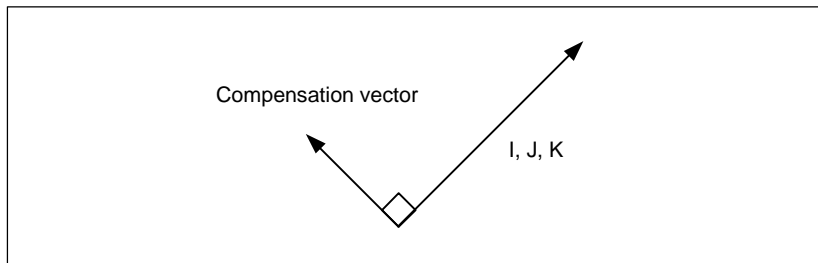
At the start of cutter compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This makes it possible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

IJ type vector (XY plane)

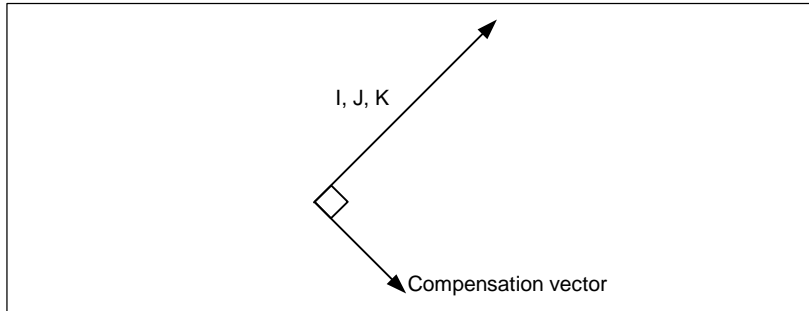
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) It is assumed that the compensation vector (IJ type vector) is the vector 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. ;
    
```

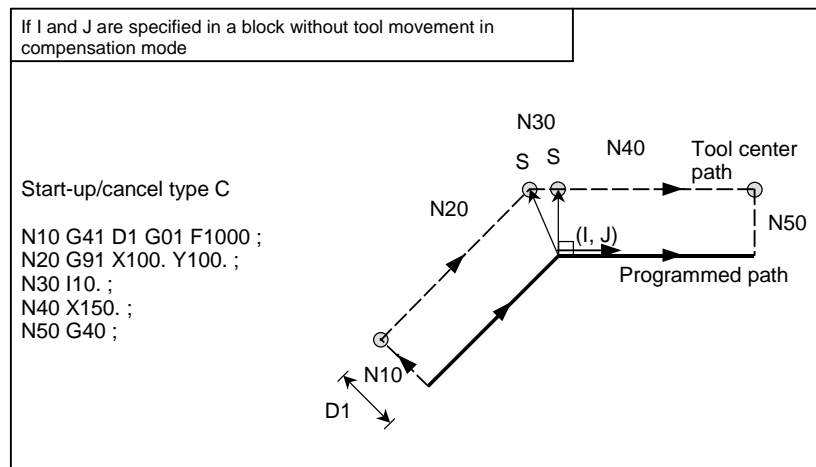
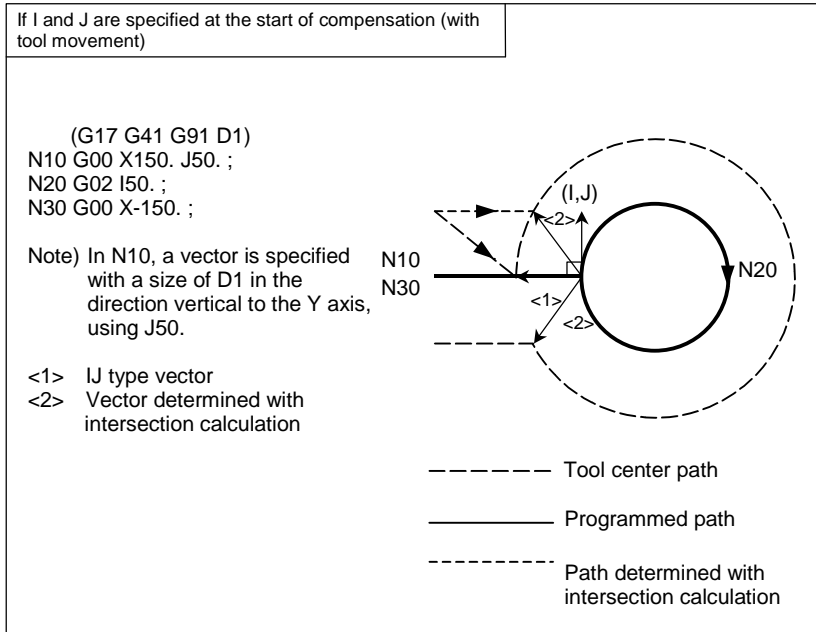
Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.

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

```

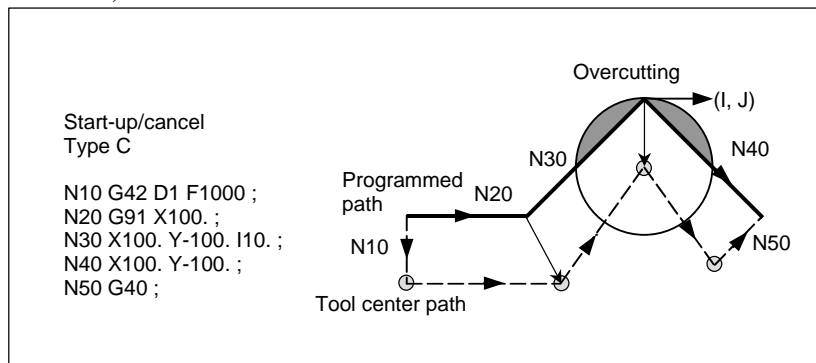
(G40)
N10 G41 I1 D1 ;
N20 G91 X100. Y100. ;
N30 X150. ;
    
```

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



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.



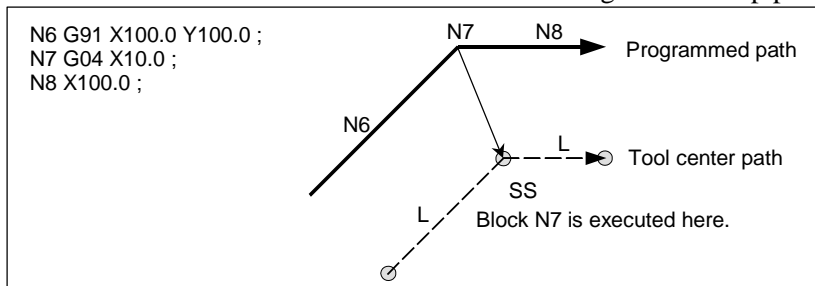
- A block without tool movement

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

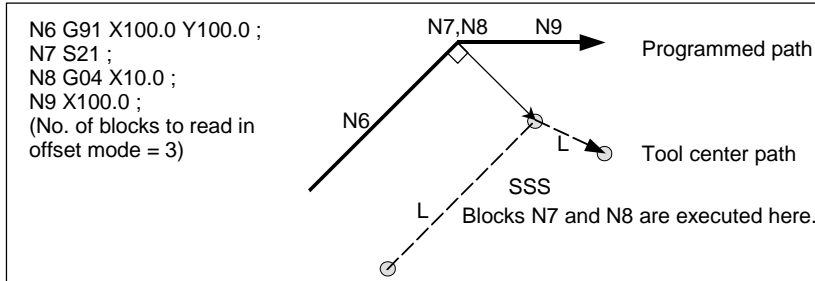
M05 ;	:	M code output
S21 ;	:	S code output
G04 X10.0 ;	:	Dwell
G22 X100000 ;	:	Machining area setting
G10 L11 P01 R10.0 ;	:	Cutter compensation value setting/changing
(G17) Z200.0 ;	:	Move command not included in the offset plane.
G90 ;, O10 ;, N20 ;	:	G, O, and N codes only
G91 X0 ;	:	Move distance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

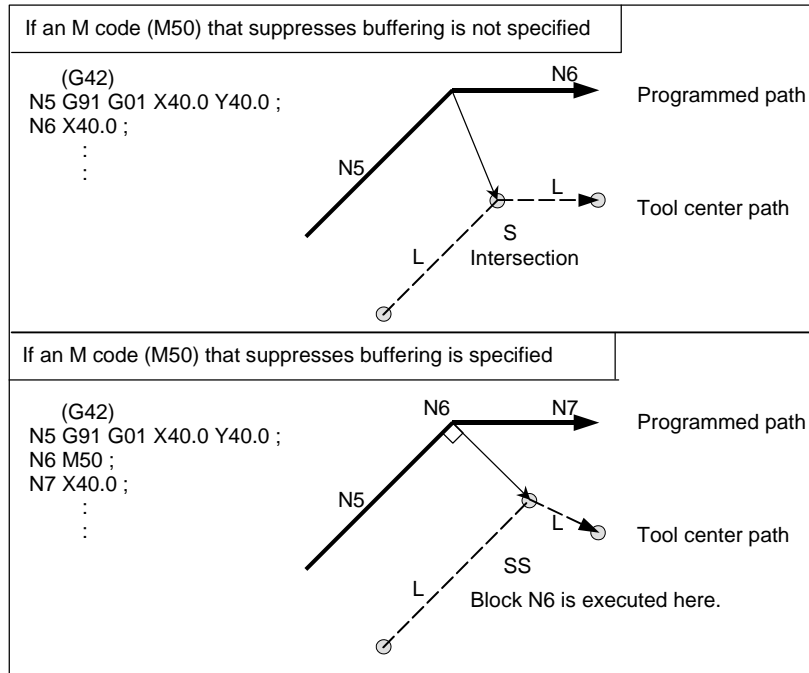


In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter No. 19625)). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier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



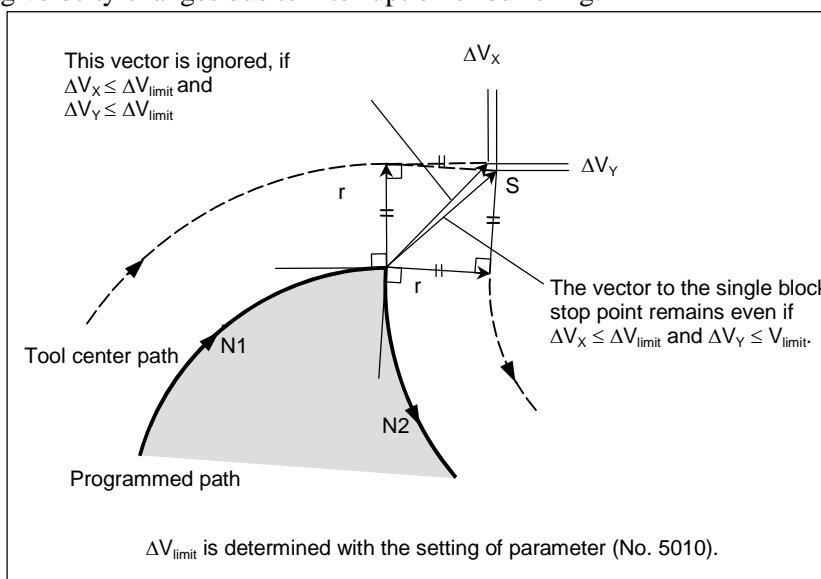
- Workpiece coordinate system or local coordinate system command in the offset mode

If the local coordinate system (G52) or workpiece coordinate system (G92) is specified in the cutter compensation (G41 or G42) or 3-dimensional cutter compensation (G41.2 to G41.6 or G42.2 to G42.6) 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.

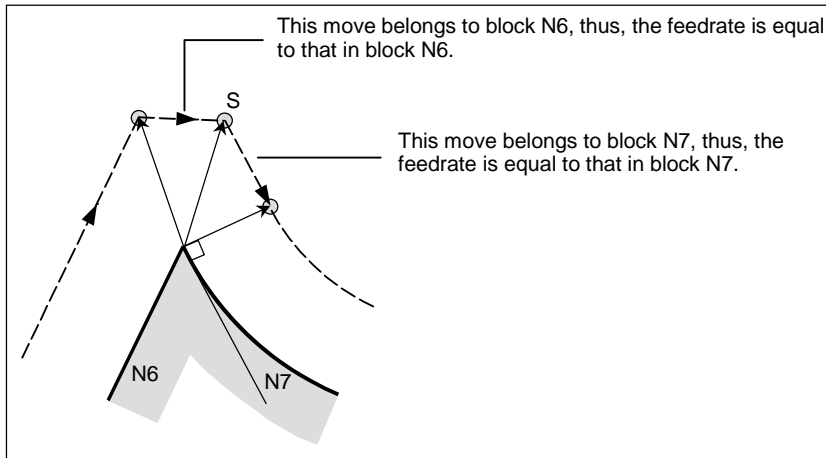
- Corner movement

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.

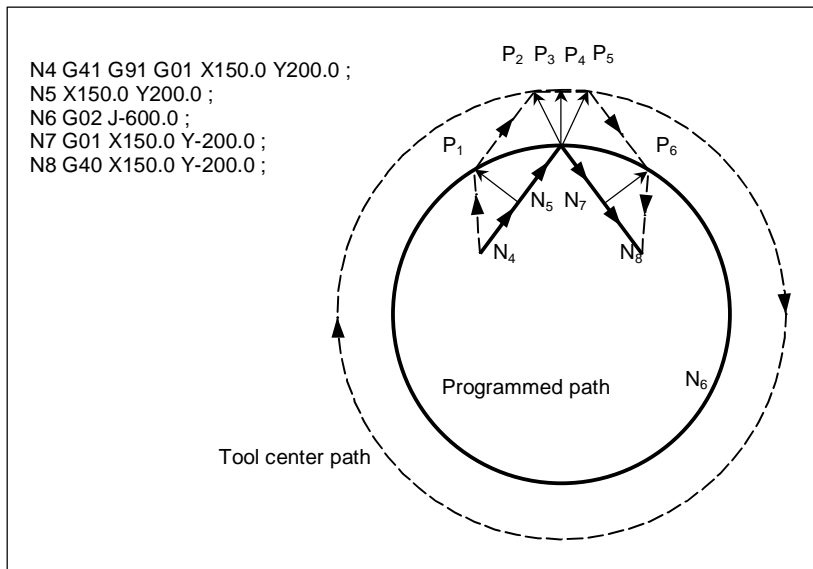
If these vectors almost coincide with each other (the distance of corner movement between the vectors is judged short due to the setting of parameter No. 5010), corner movement is not performed. In this case, the vector to the single block stop point takes precedence and remains, while other vectors 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:

$P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow (\text{Circle}) \rightarrow P_4 \rightarrow P_5 \rightarrow P_6$

But if the distance between P_2 and P_3 is negligible, the point P_3 is ignored. Therefore, the tool path is as follows:

$P_2 \rightarrow P_4$

Namely, circle cutting by the block N6 is ignored.

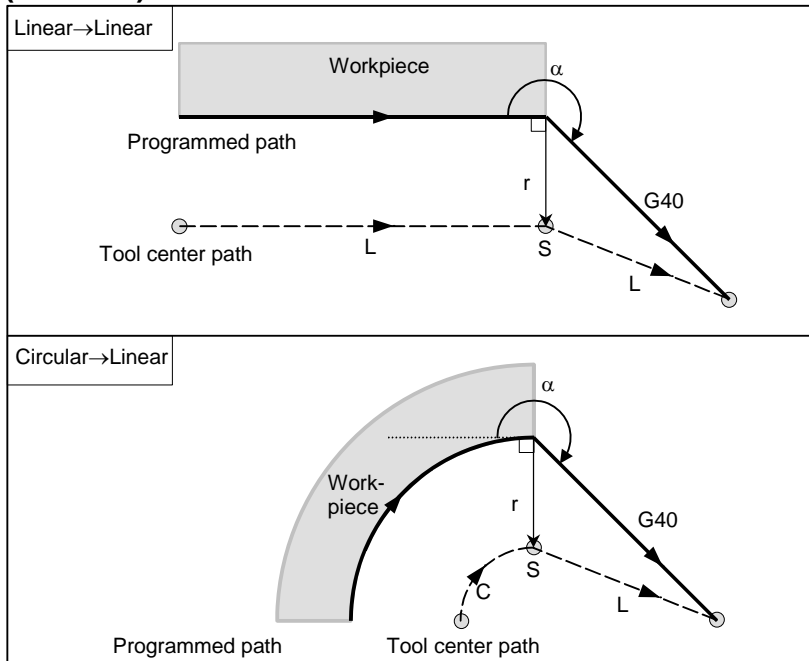
- Interruption of manual operation

For manual operation during the offset mode, see "Manual Absolute ON and OFF."

6.6.4 Tool Movement in Offset Mode Cancel

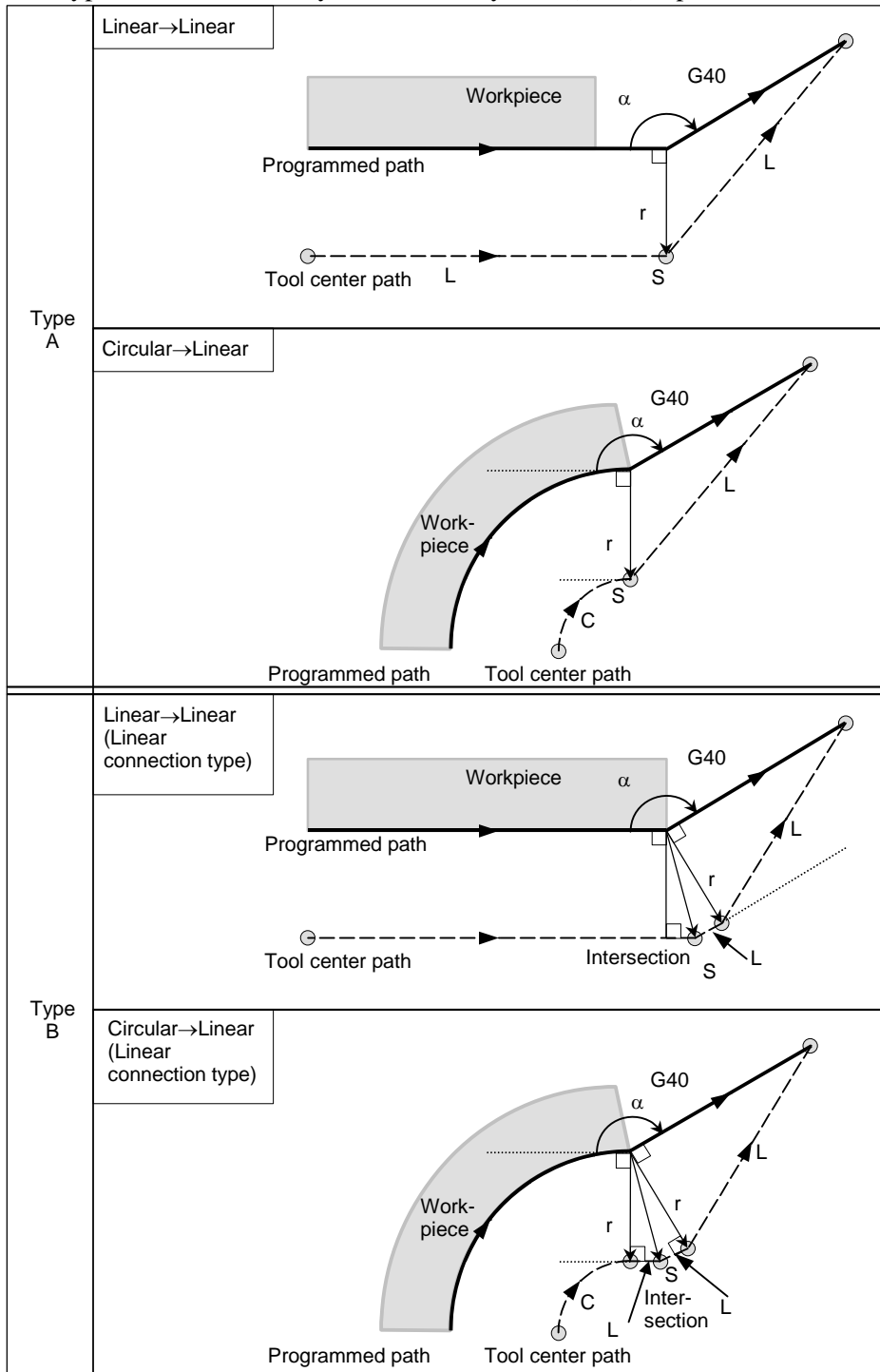
Explanation

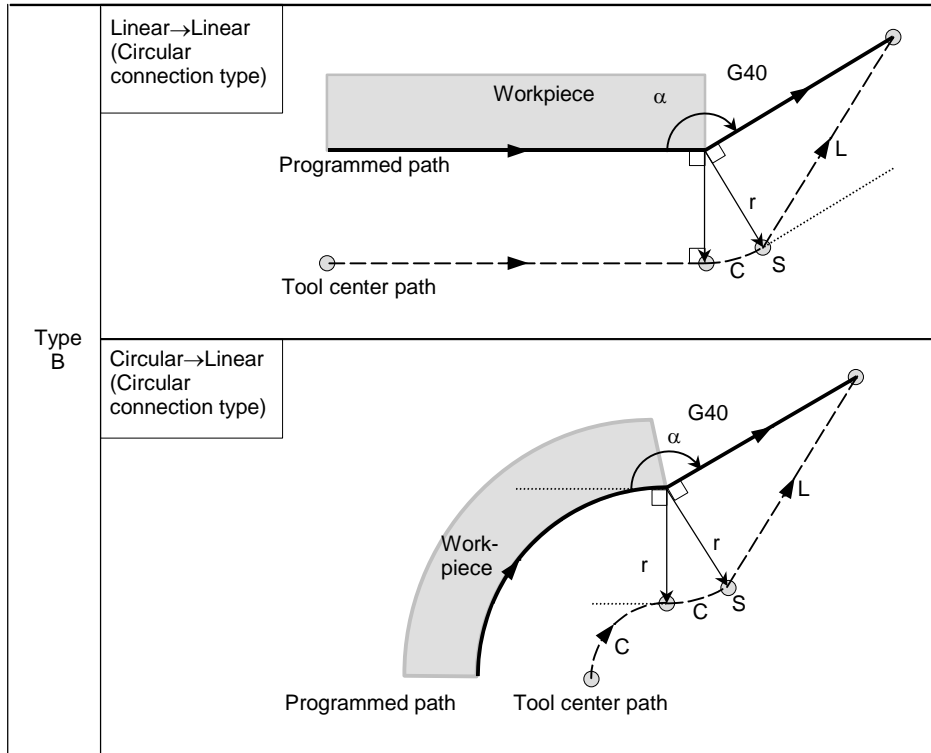
- If the cancel block is a block with tool movement, and the tool moves around the inside ($180^\circ \leq \alpha$)



- If the cancel block is a block with tool movement, and the tool moves around the outside at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)

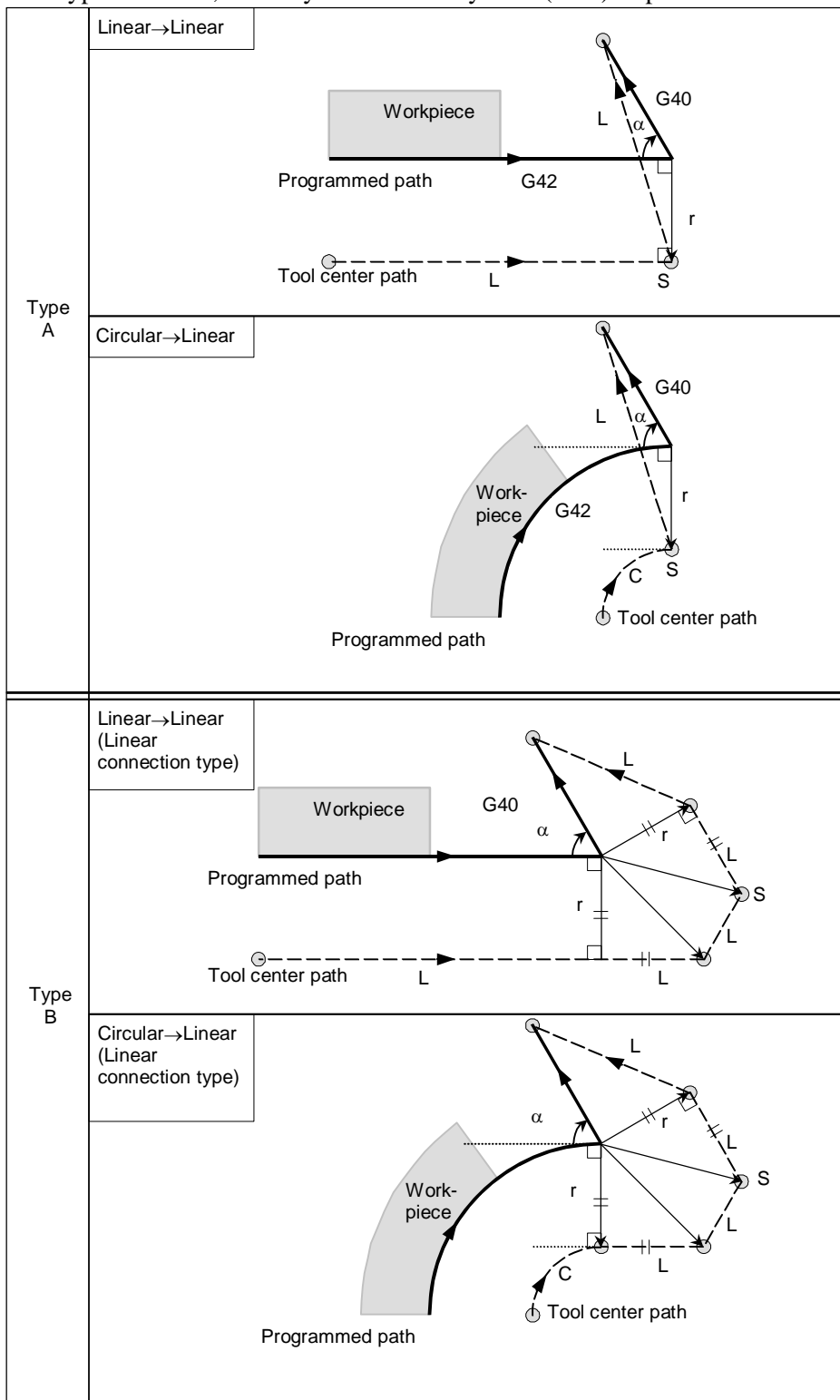
Tool path has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

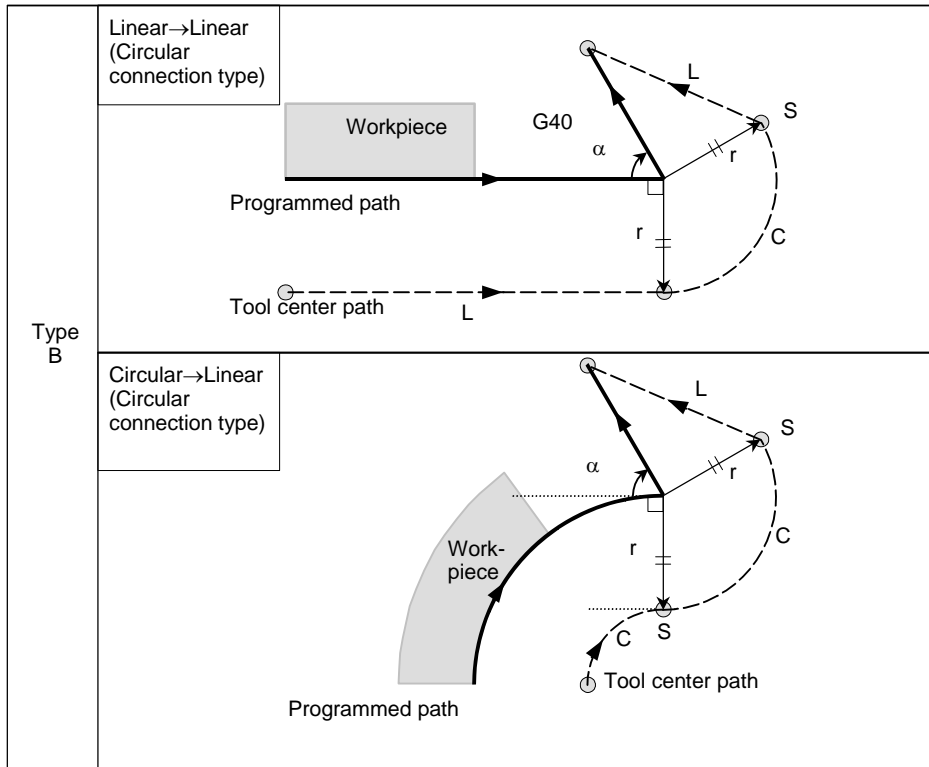




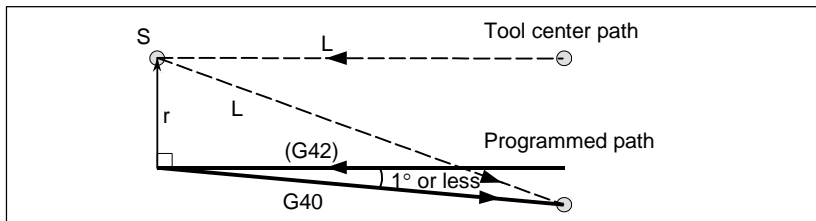
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle ($\alpha < 90^\circ$)

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





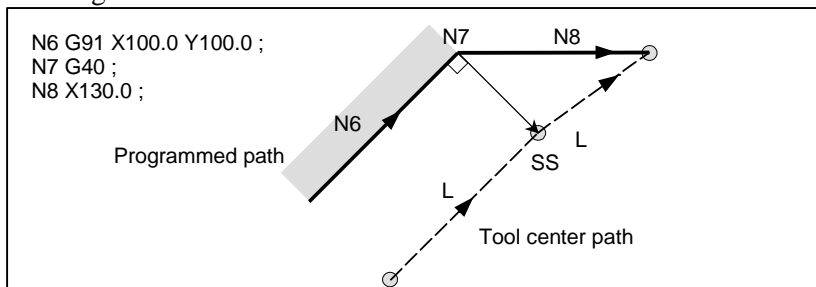
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle of 1 degree or less in a linear → linear manner ($\alpha \leq 1^\circ$)



- A block without tool movement specified together with offset cancel

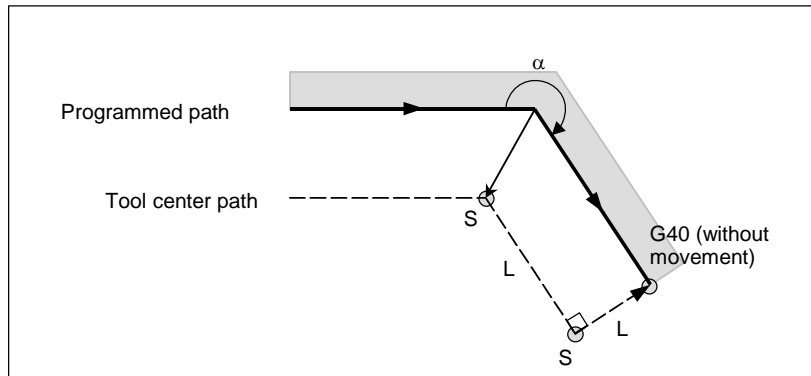
For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the tool radius · 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.



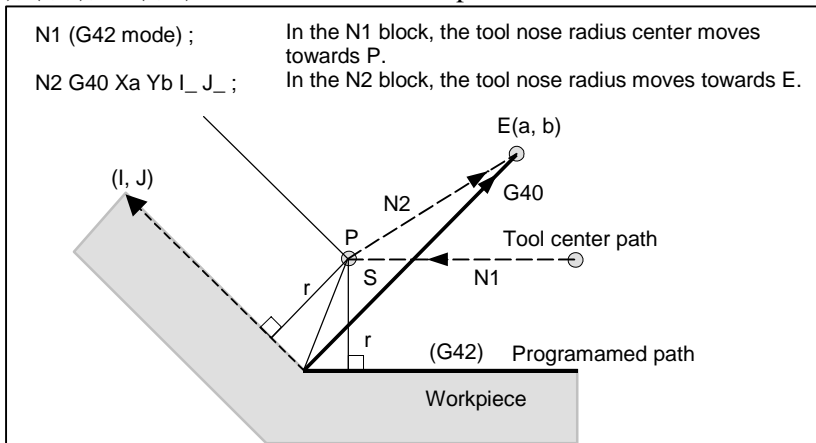
For type C

The tool shifts by the compensation value in the direction vertical to the block preceding the cancel block.

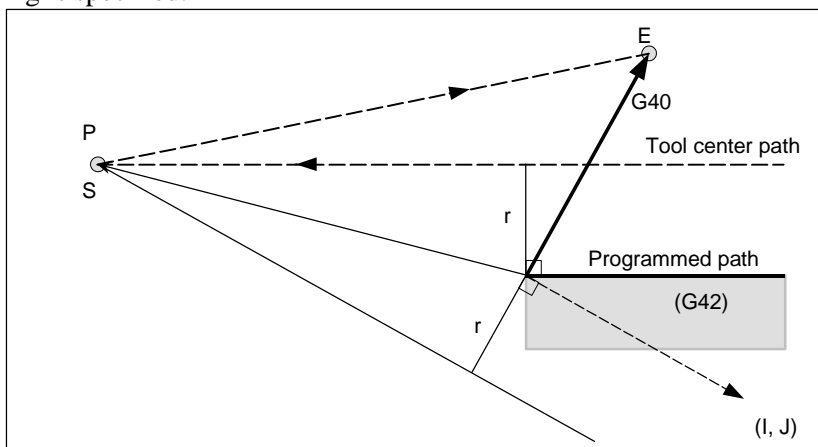


- Block containing G40 and I_J_K_
The previous block contains G41 or G42

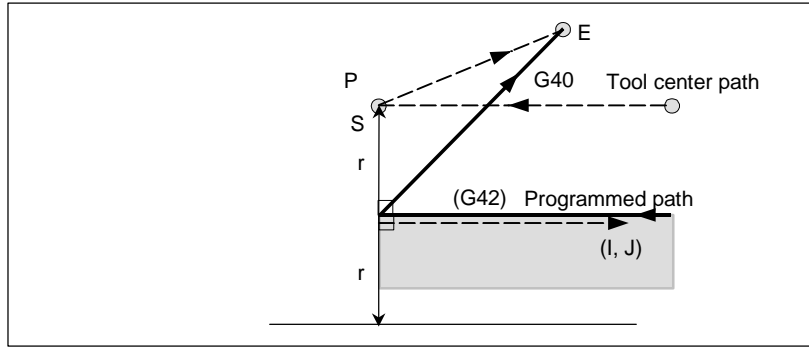
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end point determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified.



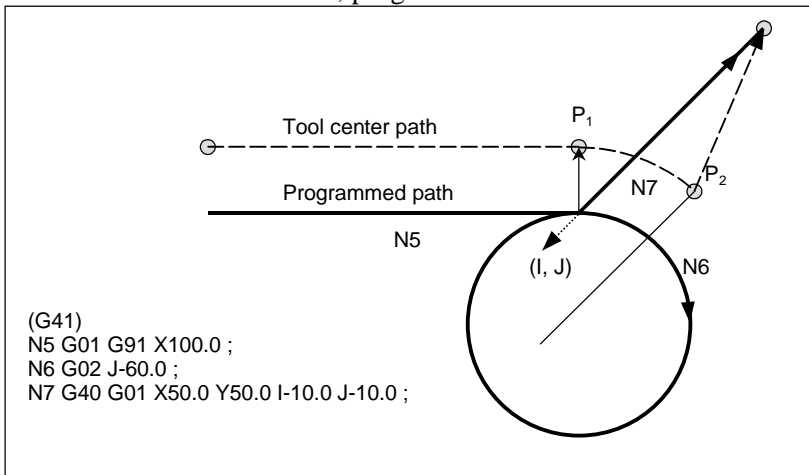
When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



- Length of the tool center path larger than the circumference of a circle

In the Example shown below, the tool does not trace the circle more than once. It moves along the arc from P₁ to P₂. The interference check function described 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.

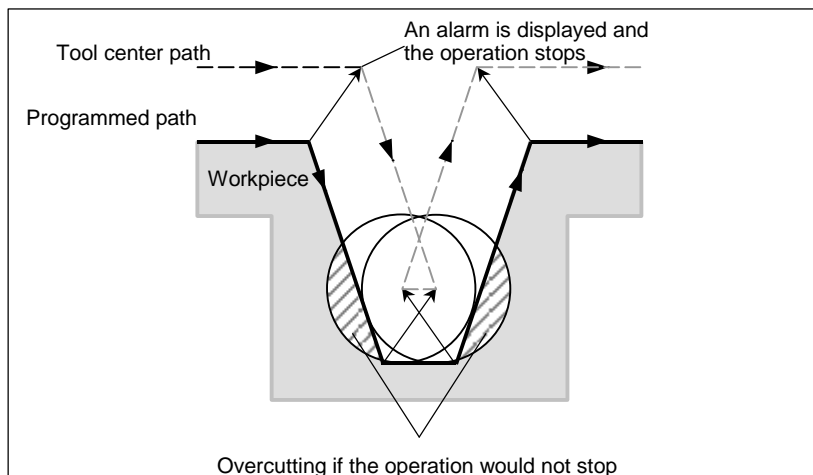


Fig. 6.6.5 (a) Machining a groove smaller than the diameter of the tool

- 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.

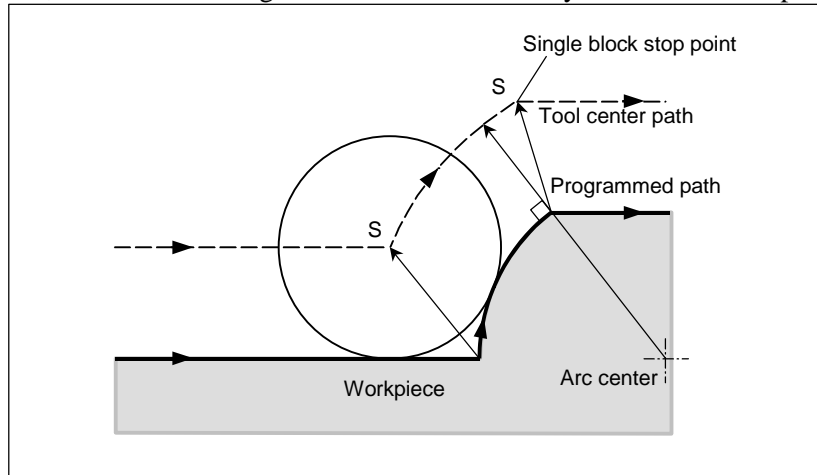


Fig. 6.6.5 (b) Machining a step larger than the tool radius

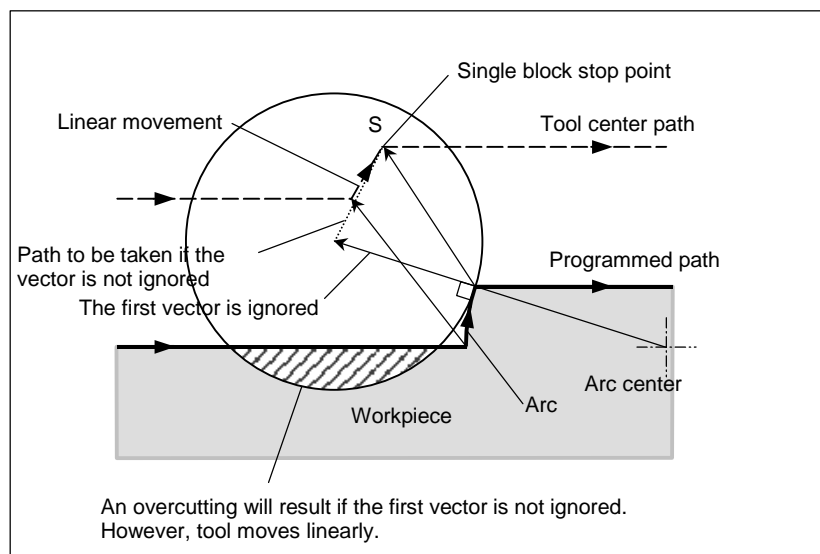


Fig. 6.6.5 (c) Machining a step smaller than the tool radius

- 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.

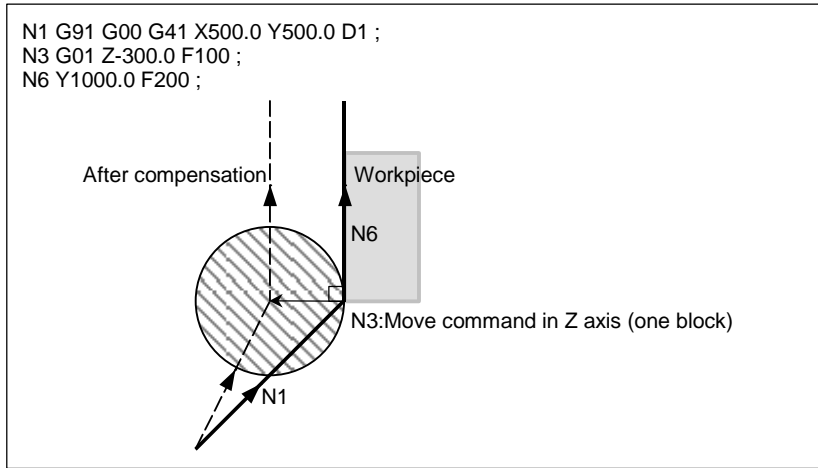


Fig. 6.6.5 (d)

In the program example in the Fig. 6.6.5 (d), when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the Fig. 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).

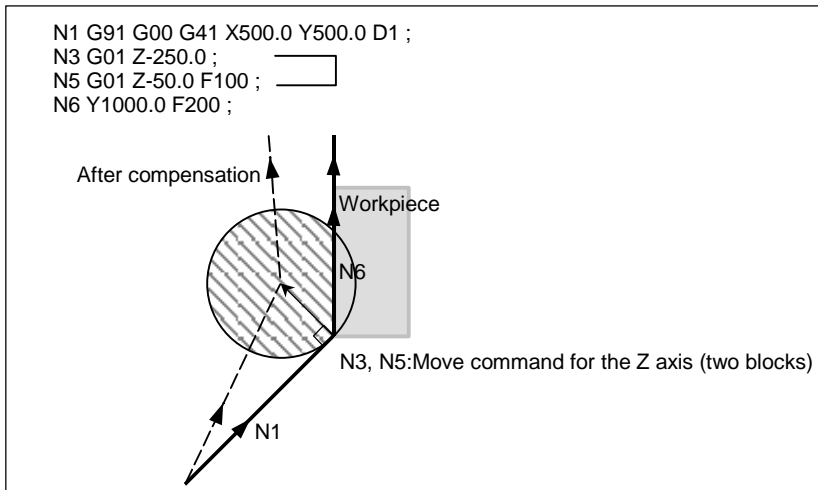


Fig. 6.6.5 (e)

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.

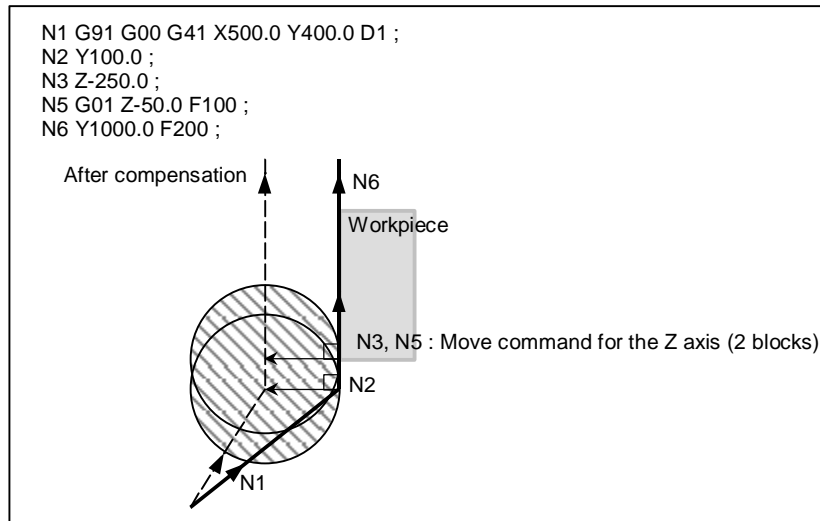


Fig. 6.6.5 (f)

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.

Parameter CNV	Parameter CNC	Operation
0	0	An interference check is enabled, and a direction check and a circular angle check can be performed.
0	1	An interference check is enabled, and only a circular angle check is performed.
1	-	An interference check is disabled.

NOTE
 There are no settings for performing a direction check only.

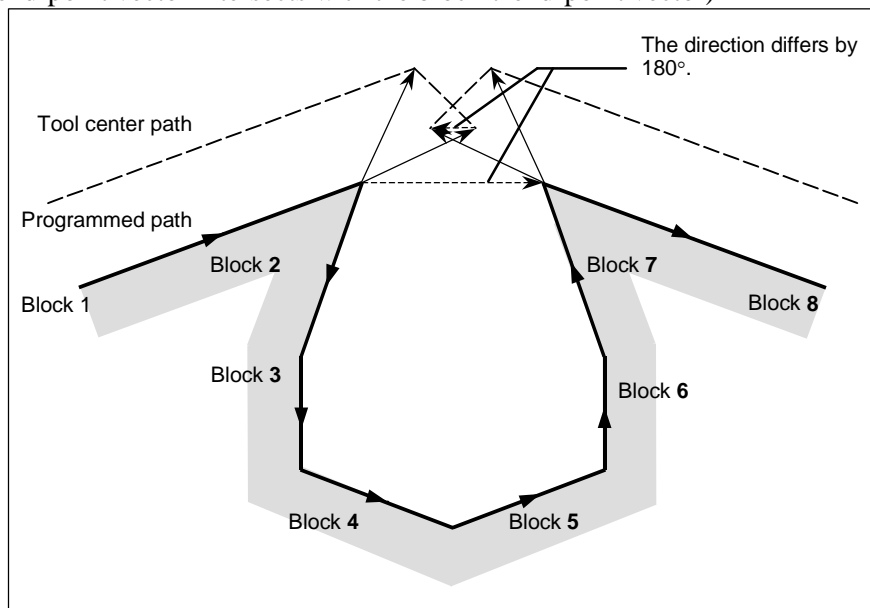
- Interference reference <1> (direction check)

Assuming the number of blocks to read during 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)

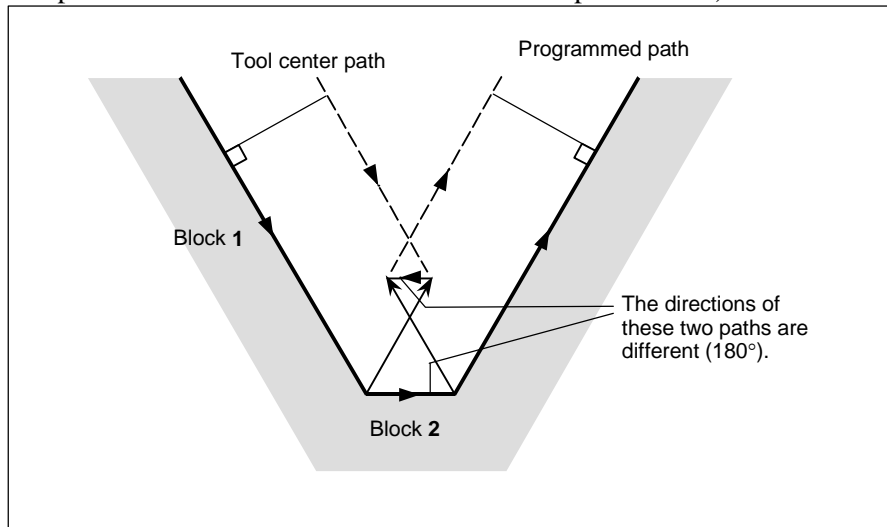
Even if multiple number of compensation vector groups are generated, a check is performed on all pairs. The judgment method is as follows: For a check on the compensation vector group in (block 1 - block 2) and those in (block N-1 - block N), the direction vector from the specified (end point of block 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>

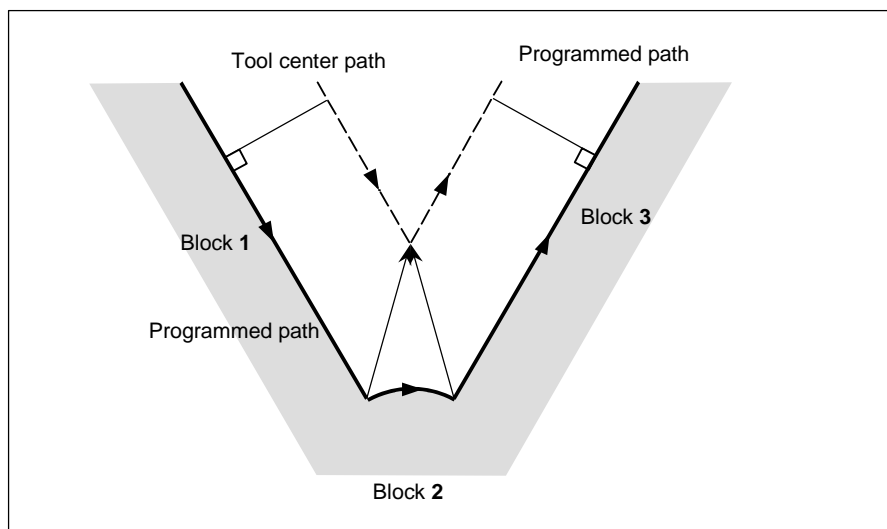
(If the block 1 end-point vector intersects with the block 2 end-point vector)



- Interference reference <2> (circular angle check)

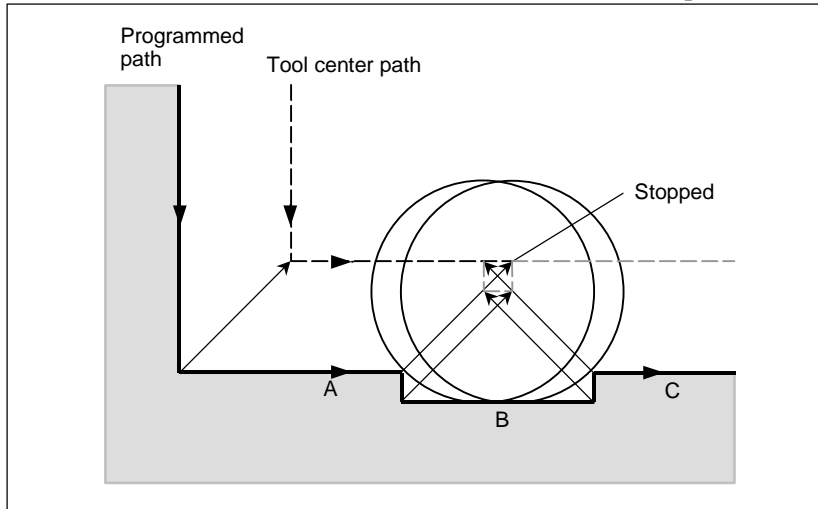
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed 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)



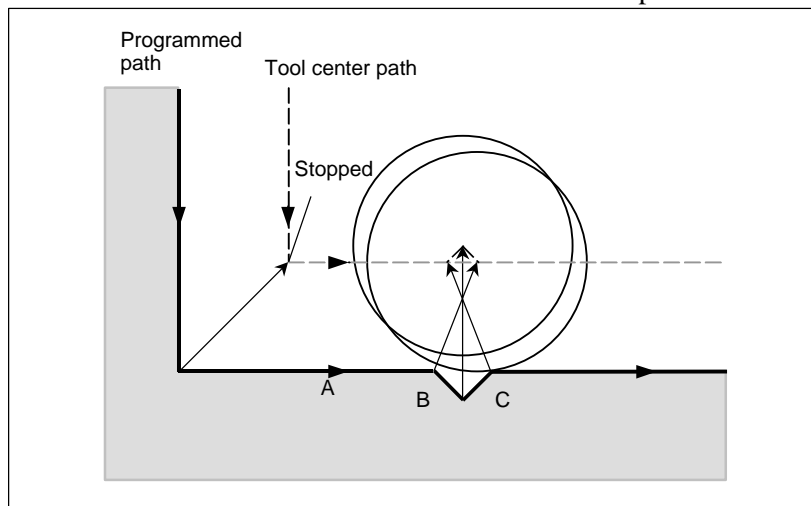
- **When interference is assumed although actual interference does not occur**

<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.

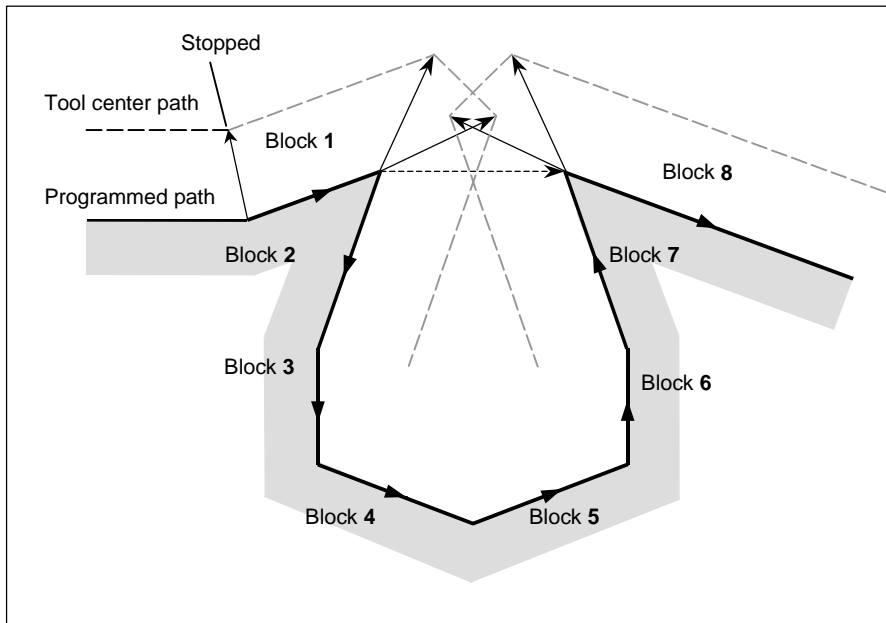


Fig. 6.6.6.2 (a)

- 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.

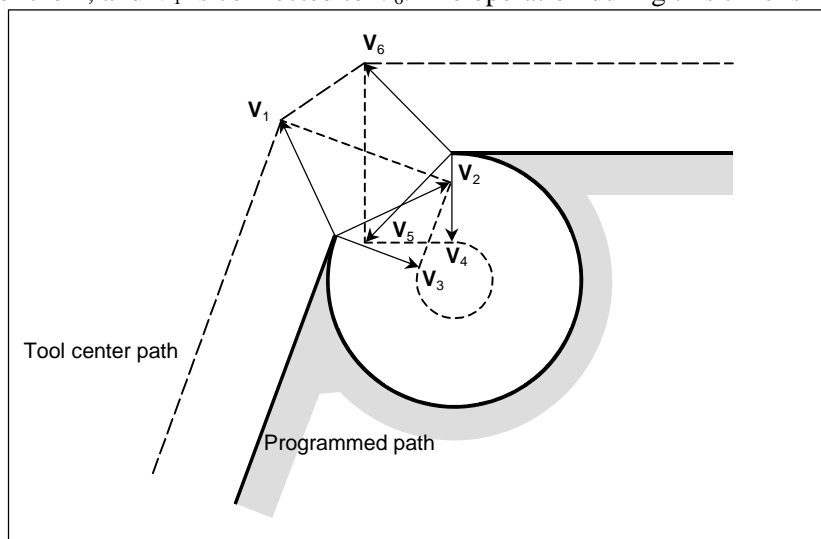


Fig. 6.6.6.2 (b)

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.

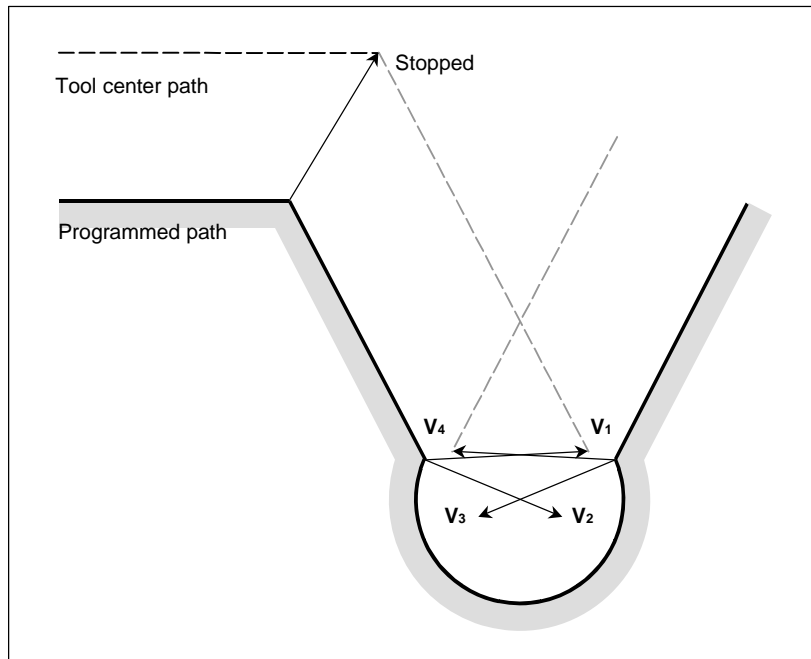


Fig. 6.6.6.2 (c)

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.

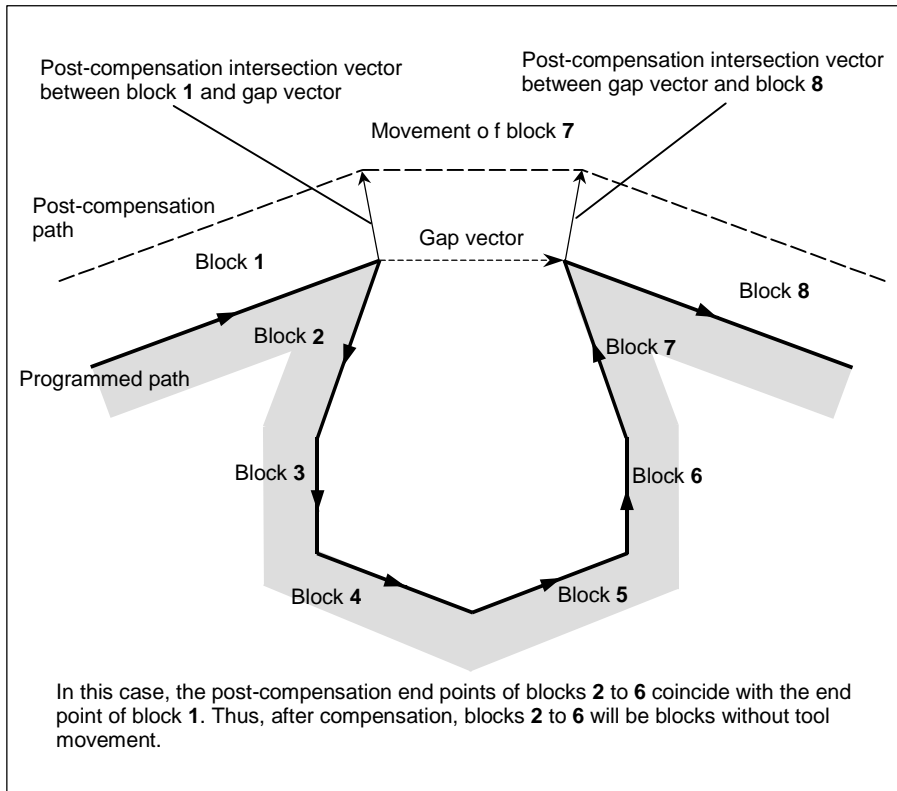


Fig. 6.6.6.3 (a)

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.

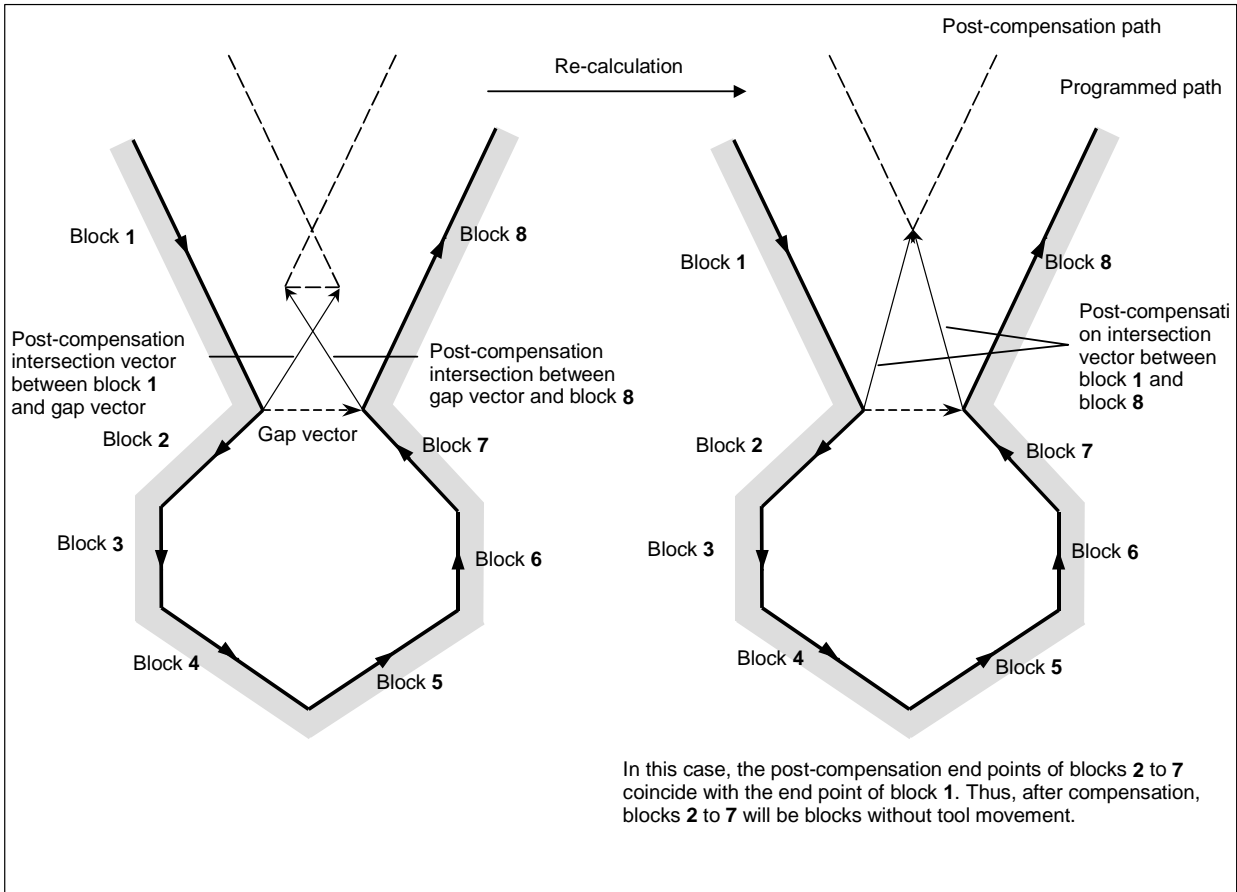


Fig. 6.6.6.3 (b)

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.

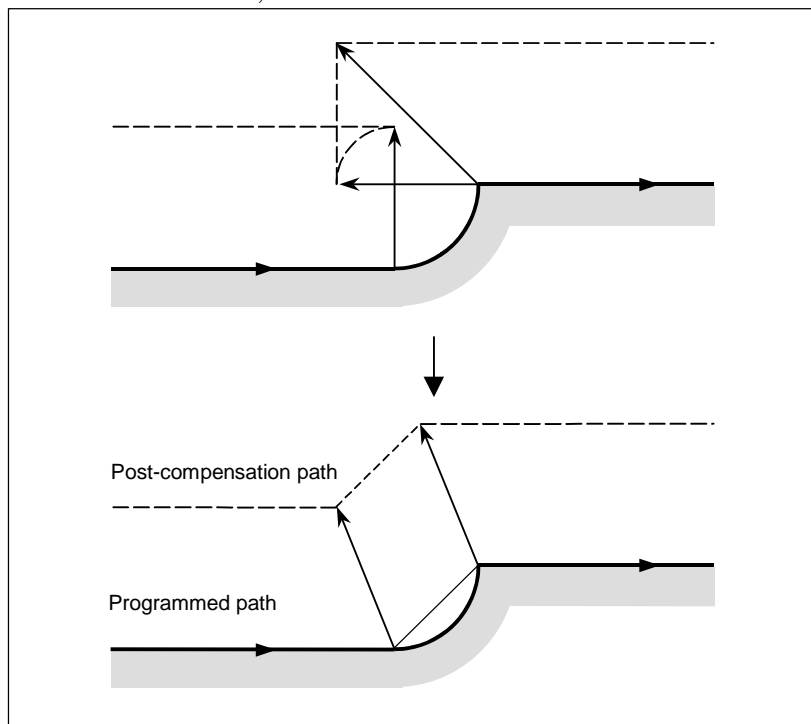
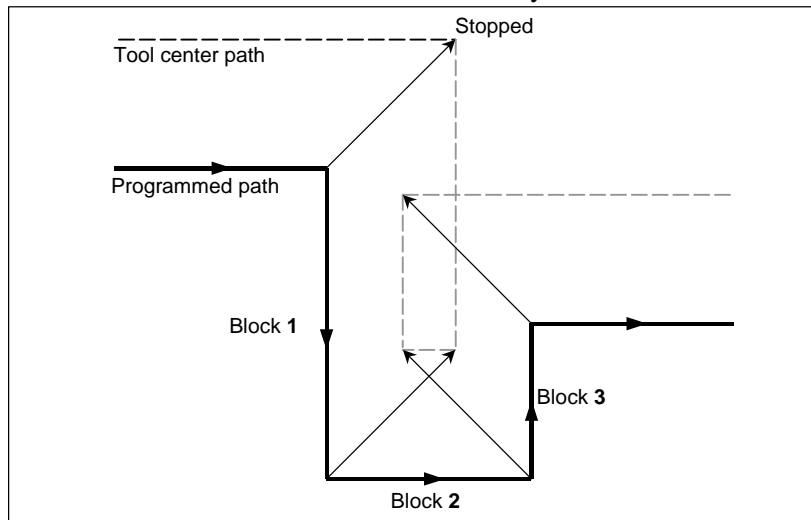


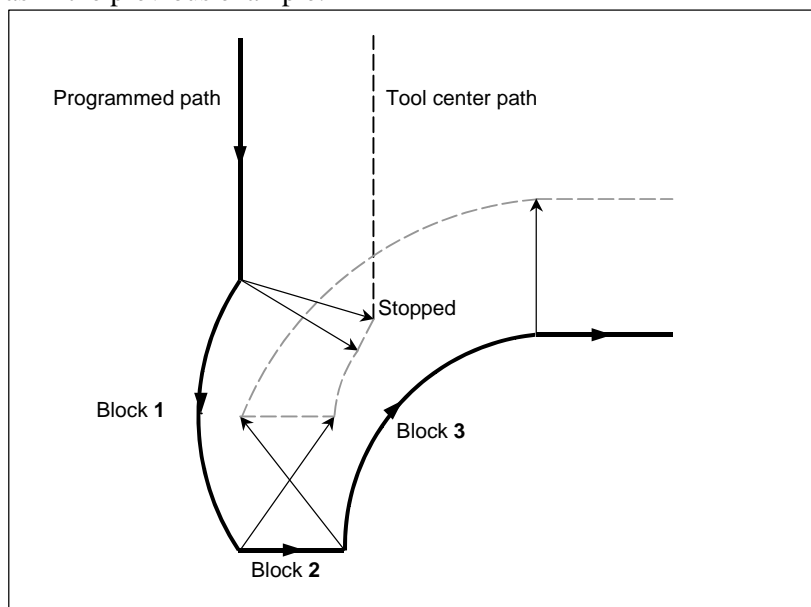
Fig. 6.6.6.3 (c)

- 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 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 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 will occur immediately before block 1 and the tool will stop.

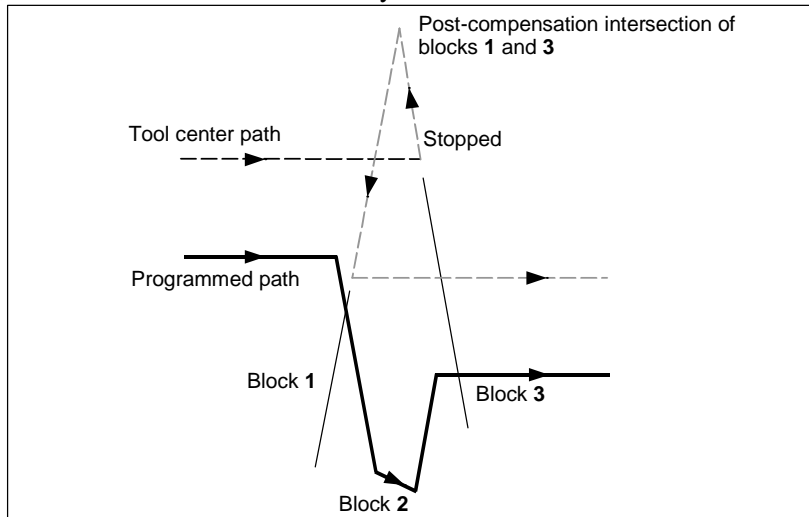


Fig. 6.6.6.3 (f)

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 judged dangerous; an alarm will occur immediately before block 1 and the tool will stop.

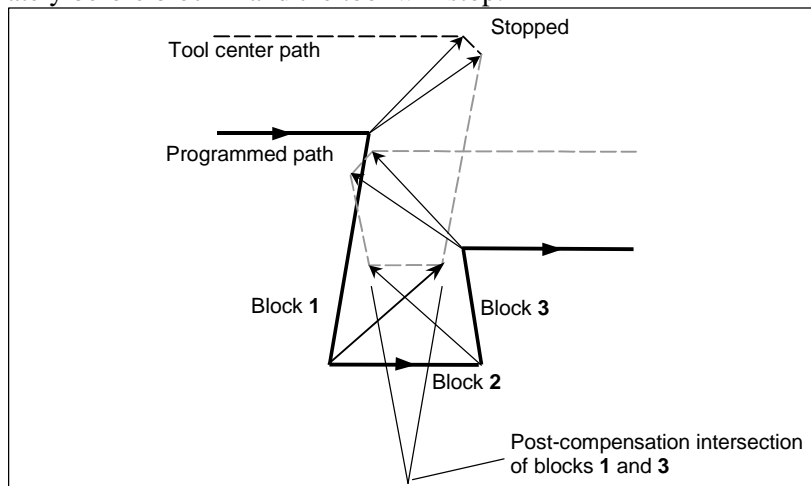


Fig. 6.6.6.3 (g)

- 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 will occur immediately before the block and the tool will stop.

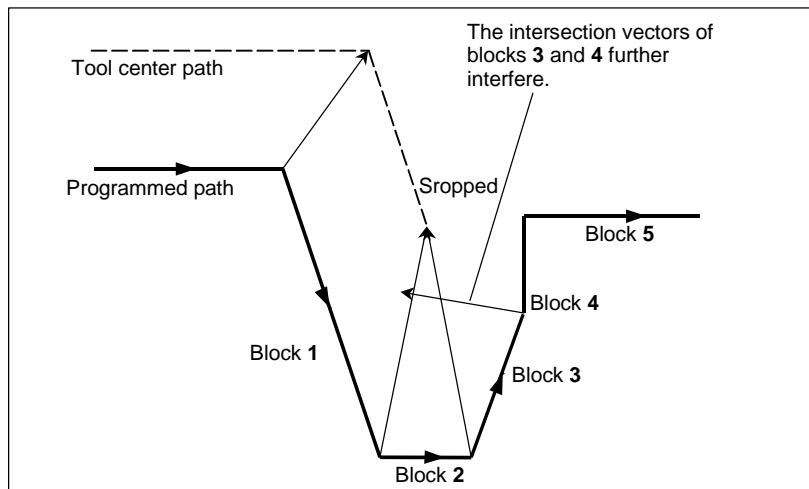


Fig. 6.6.6.3 (h)

NOTE

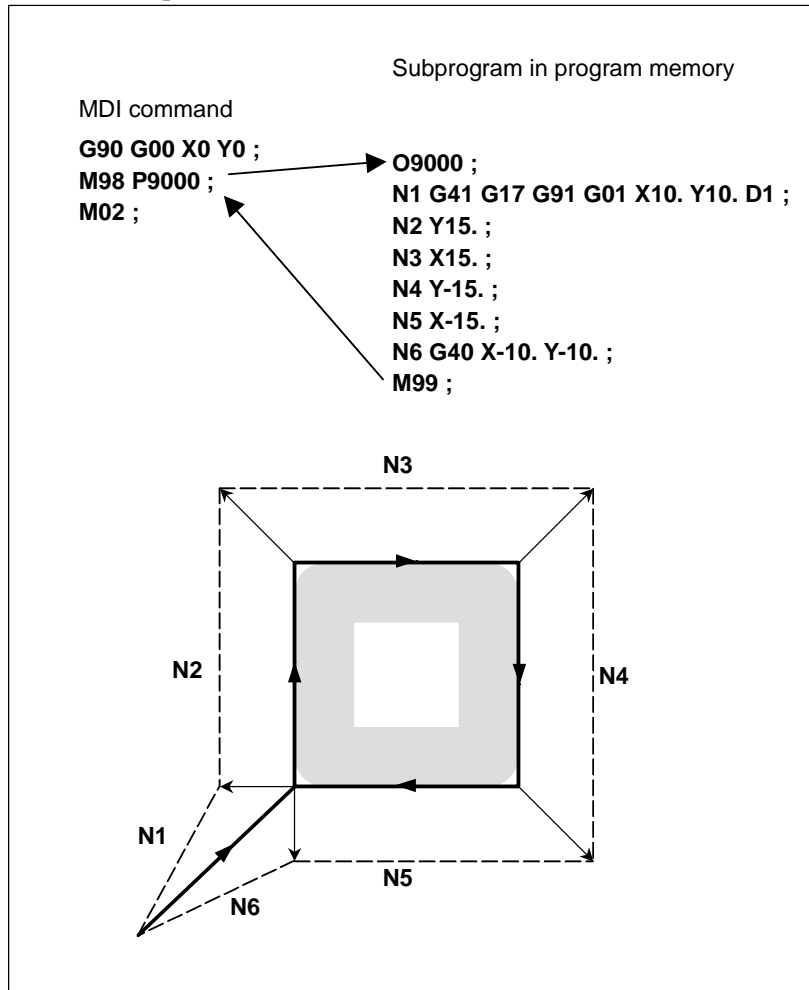
- 1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting bit 6 (NAA) of parameter No. 19607 appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", 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

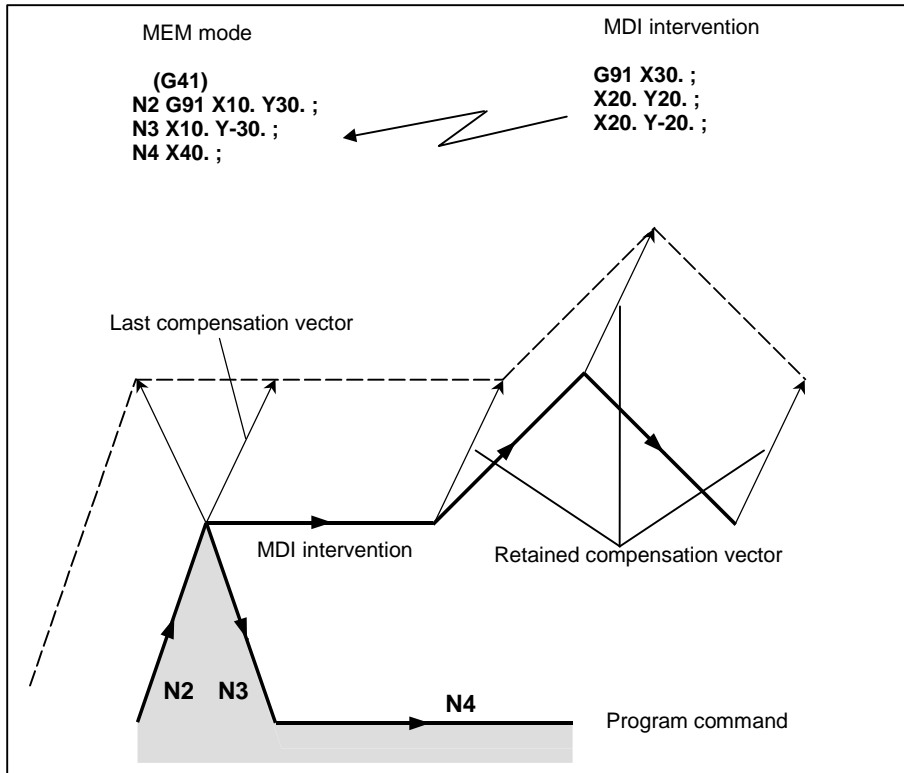
- MDI operation

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.



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

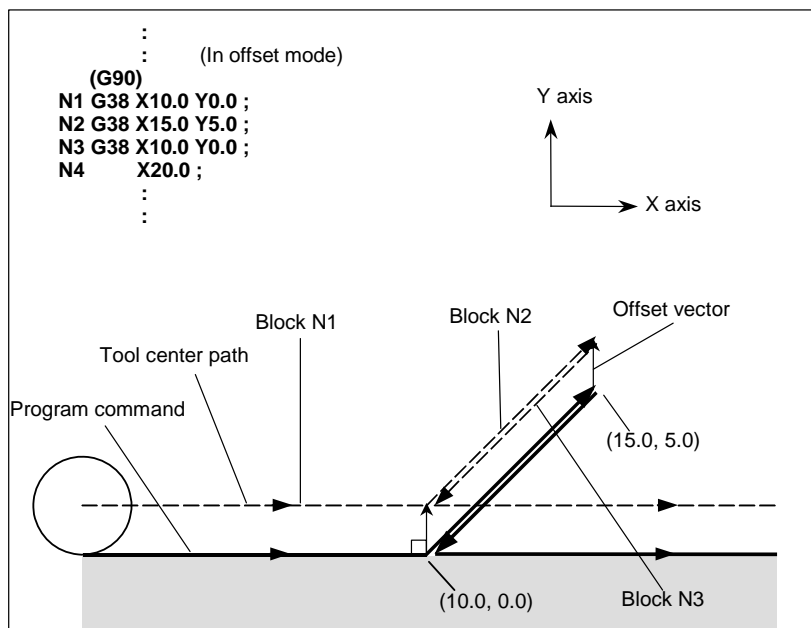
Specify G38 in either G00 or G01 mode. If it is specified in G02 or G03 (circular interpolation) mode, a radial error may occur at the start and end points.

- 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 ;

G39 $\left\{ \begin{array}{l} \underline{I} \ \underline{J} \\ \underline{I} \ \underline{K} \\ \underline{J} \ \underline{K} \end{array} \right\} ;$

Explanation

- Corner circular interpolation

When the command indicated above is specified, corner circular interpolation in which the radius equals compensation value can be performed. G41 or G42 preceding the command determines whether the arc is clockwise or counterclockwise. G39 is a one-shot G code.

- G39 without I, J, or K

When G39 is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block.

- G39 with I, J, and K

When G39 is specified with I, J, and K, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the vector defined by the I, J, and K values.

Limitation

- Move command

In a block containing G39, no move command can be specified. Otherwise, an alarm will occur.

- Inner corner

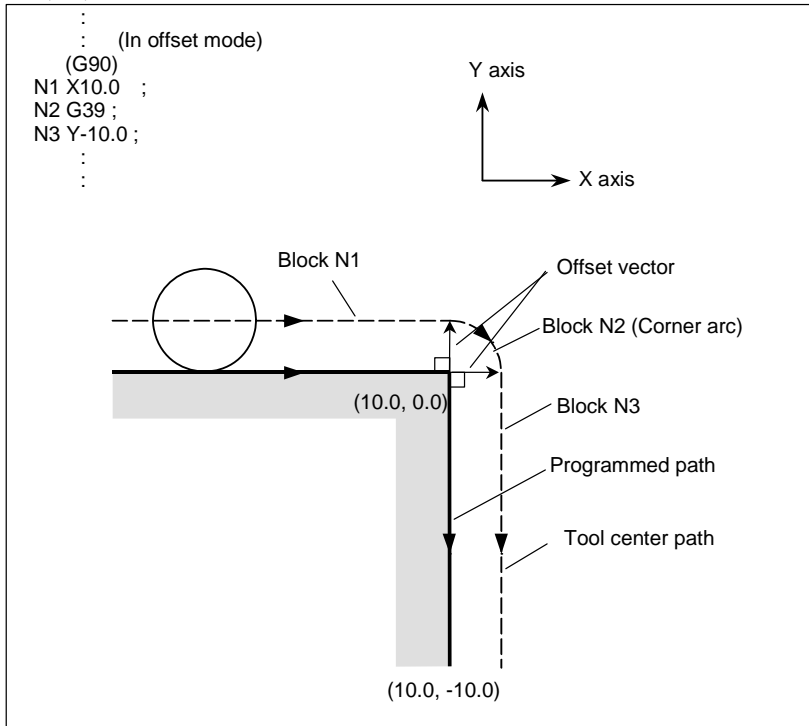
In an inner corner block, G39 cannot be specified. Otherwise, overcutting will occur.

- Corner arc velocity

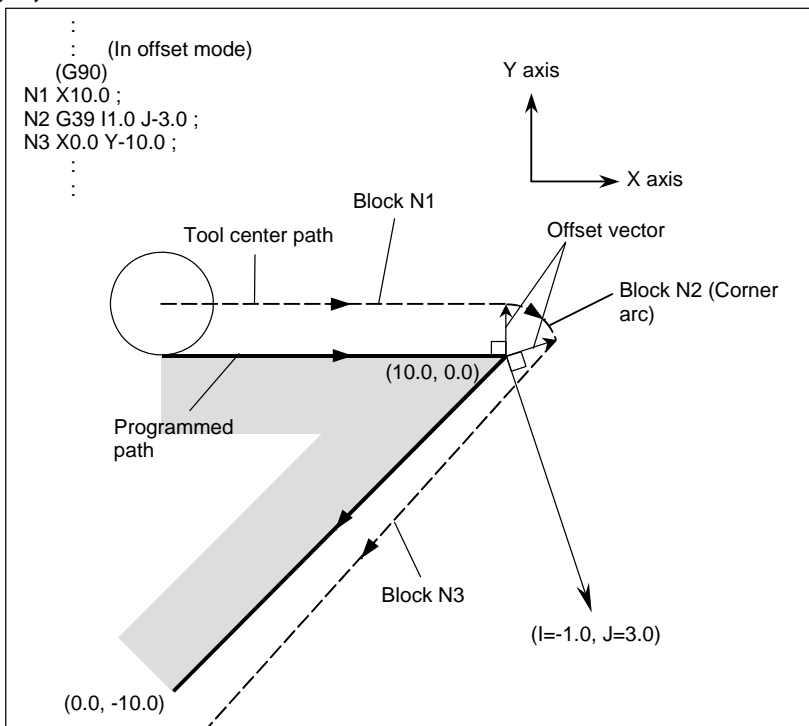
If a corner arc is specified with G39 in G00 mode, the corner arc block velocity will be that of the F command previously specified. If G39 is specified in a state in which no F command has never been specified, the velocity of the corner arc block will be that specified with parameter No. 1411.

Example

- G39 without I, J, or K



- G39 with I, J, and K



6.9 3-DIMENSIONAL TOOL COMPENSATION (G40, G41)

In cutter compensation C, two-dimensional offsetting is performed for a selected plane. In 3-dimensional tool compensation, the tool can be shifted 3-dimensionally when a 3-dimensional offset direction is programmed.

Format

- Start up (Starting 3-dimensional tool compensation)

When the following command is executed in the cutter compensation cancel mode, the 3-dimensional tool compensation mode is set:

G41 Xp_Yp_Zp_I_J_K_D_;

Xp : X-axis or a parallel axis

Yp : Y-axis or a parallel axis

Zp : Z-axis or a parallel axis

I :

J : } See "Explanation".

K :

D : Code for specifying as the cutter compensation value (1-3 digits) (D code)

- Canceling 3-dimensional tool compensation

When the following command is executed in the 3-dimensional tool compensation mode, the cutter compensation cancel mode is set:

- When canceling the 3-dimensional tool compensation mode and tool movement at the same time

G40 Xp_Yp_Zp_;

or

Xp_Yp_Zp_D00;

- When only canceling the vector

G40;

or

D00;

- Selecting offset space

The 3-dimensional space where 3-dimensional tool compensation is to be executed is determined by the axis addresses specified in the startup block containing the G41 command. If Xp, Yp, or Zp is omitted, the corresponding axis, X-, Y-, or Z-axis (the basic three axis), is assumed.

(Example)

When the U-axis is parallel to the X-axis, the V-axis is parallel to the Y-axis, and the W-axis is parallel to the Z-axis

G41 X_I_J_K_D_; XYZ space

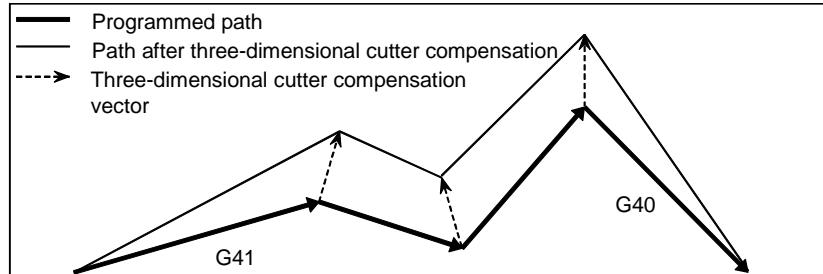
G41 U_V_Z_I_J_K_D_; UVZ space

G41 W_I_J_K_D_; XYW space

Explanation

- 3-dimensional tool compensation vector

In 3-dimensional tool compensation mode, the following 3-dimensional tool compensation vector is generated at the end of each block:



The 3-dimensional tool compensation vector is obtained from the following expressions:

$$V_x = \frac{i \times r}{p} \quad (\text{Vector component along the } X_p\text{-axis})$$

$$V_y = \frac{j \times r}{p} \quad (\text{Vector component along the } Y_p\text{-axis})$$

$$V_z = \frac{k \times r}{p} \quad (\text{Vector component along the } Z_p\text{-axis})$$

In the above expressions, i , j , and k are the values specified in addresses I, J, and K in the block. r is the offset value corresponding to the specified offset number. p is the value obtained from the following expression:

$$p = \sqrt{i^2 + j^2 + k^2}$$

When the user wants to program the magnitude of a 3-dimensional tool compensation vector as well as its direction, the value of p in the expressions of V_x , V_y , and V_z can be set as a constant in parameter No. 5011.

If the parameter is set to 0, however, p is determined as follows:

$$p = \sqrt{i^2 + j^2 + k^2}$$

- Relationship between 3-dimensional tool compensation and other compensation functions

Tool length compensation	The specified path is shifted by 3-dimensional tool compensation and the subsequent path is shifted by tool length compensation.
Tool offset	When tool offset is specified in the 3-dimensional tool compensation mode, an alarm is issued (alarm PS0042).
Cutter compensation	When addresses I, J, and K are all specified at startup, 3-dimensional tool compensation mode is set. When not all of the addresses are specified, cutter compensation mode is set. Therefore, cutter compensation cannot be specified in 3-dimensional tool compensation mode and 3-dimensional tool compensation cannot be specified in cutter compensation mode.

- Specifying I, J, and K

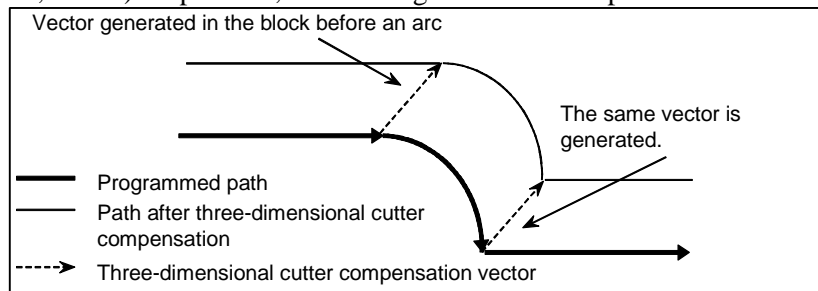
Addresses I, J, and K must all be specified to start 3-dimensional tool compensation. When even one of the three addresses is omitted, two-dimensional cutter compensation is activated. When a block specified in 3-dimensional tool compensation mode contains none of addresses I, J, and K, the same vector as the vector generated in the previous block is generated at the end of the block.

- G42

Generally, G41 is specified to start 3-dimensional tool compensation. Instead of G41, G42 can be specified for startup. With G42, 3-dimensional tool compensation is performed in the opposite direction.

- Offset vector in interpolation

When circular interpolation, helical interpolation (both specified with G02, G03), or involute interpolation (G02.2, G03.2) is specified, the vector generated in the previous block is maintained.



- Reference position return check (G27)

Before specifying reference position return check (G27), cancel 3-dimensional tool compensation. In the compensation mode, G27 brings the tool to a position shifted by the offset value. If the position the tool reached is not the reference position, the reference position return LED does not go on (the alarm PS0092 alarm is issued).

- Return to a reference position (G28, G30, G30.1)

When return to the reference position (G28), to the second, third, or fourth reference position (G30), or to the floating reference position (G30.1) is specified, the vector is cleared at a middle point.

- Alarm issued at startup

If one of the following conditions is present at the startup of 3-dimensional tool compensation, an alarm is issued:

- Two or more axes are specified in the same direction. (alarm PS0047)
- Although Xp, Yp, or Zp is omitted, the basic three axes are not set. (alarm PS0048)

- Alarm during 3-dimensional tool compensation

If one of the following G codes is specified in the 3-dimensional tool compensation mode, an alarm is issued:

G05 High-speed cycle machining (alarm PS0178)

G31 Skip function (alarm PS0036)

G51 Scaling (alarm PS0141)

- Commands that clear the vector

When one of the following G codes is specified in 3-dimensional tool compensation mode, the vector is cleared:

G73 Peck drilling cycle

G74 Reverse tapping cycle

G76 Fine boring

G80 Canned cycle cancel

G81 Drilling cycle, spot boring

G82 Drilling cycle, counter boring

G83 Peck drilling cycle

G84 Tapping cycle

G85 Boring cycle

G86 Boring cycle

G87 Back boring cycle

G88 Boring cycle

G89 Boring cycle

G53 Machine coordinate system selection

- Commands that generate the same vector as the vector in the previous block

When one of the following G codes is specified in 3-dimensional tool compensation mode, the same vector as the vector generated in the previous block is generated at the end point of the next movement:

- G02 Circular or helical interpolation (CW)
- G03 Circular or helical interpolation (CCW)
- G02.2 Involute interpolation (CW)
- G03.2 Involute interpolation (CCW)
- G04 Dwell
- G10 Data setting
- G22 Stored stroke check function enabled

6.10 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. 6.10 (a)).

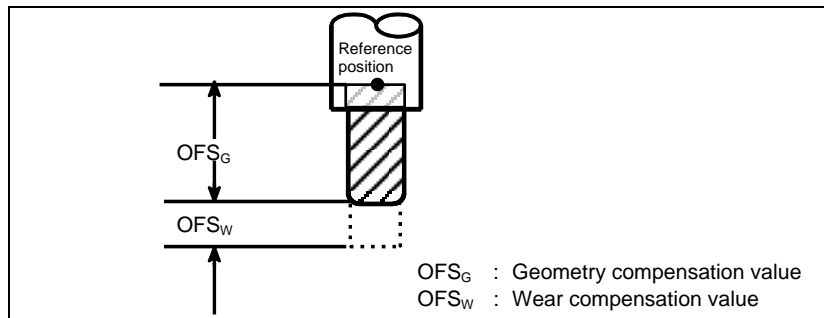


Fig. 6.10 (a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the MDI panel (see section III-11.1.1) 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.

Three types of tool compensation memories are available according to the compensation value configuration: tool compensation memory A, B, 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 B

In tool compensation memory B, memory for geometry compensation and memory for wear compensation are prepared separately. So, geometry compensation values and wear compensation values can be set separately. However, 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	For geometry compensation	For wear compensation	Common to D code/H code
001	10.100	0.100	For D code
002	20.200	0.200	For D code
003	100.000	0.100	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
:	:	:	:	:

In addition, if the cutting point command option is enabled, additional memory for corner R offset (for D code) is prepared separately.

Setting example

Compensation number	D code (For tool compensation)		D code (For corner R offset)		H code	
	For geometry compensation	For wear compensation	For geometry compensation	For wear compensation	For geometry compensation	For wear compensation
001	10.000	0.100	1.000	0.100	100.000	0.100
002	20.000	0.200	2.000	0.400	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

OFE	OFD	OFC	OFA	Unit	Valid range
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 B

```
G10 L_ P_ R_ Q_ ;
L_   : Type of compensation memory
      L10 : Geometry compensation value
      L11 : Wear compensation value
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.

6.11 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.

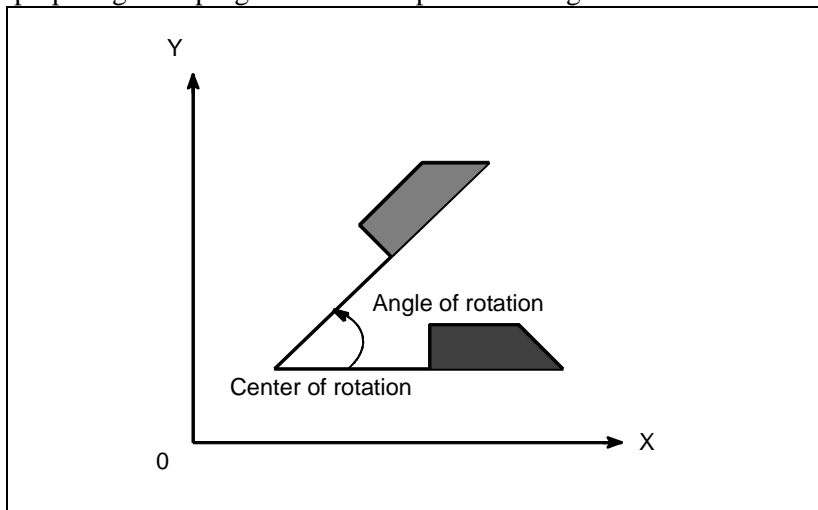


Fig. 6.11 (a) Coordinate system rotation

Format

$\left. \begin{array}{l} \text{G17} \\ \text{G18} \\ \text{G19} \end{array} \right\}$	$\text{G68 } \alpha_ \beta_ R_ ;$	Start rotation of a coordinate system.
$\left. \begin{array}{l} : \\ : \end{array} \right\}$		Coordinate system rotation mode (The coordinate system is rotated.)
G69 ;		Coordinate system rotation cancel command

G17 (G18 or G19) : Select the plane in which contains the figure to be rotated.

$\alpha_ \beta_ :$ Absolute programming for two of the X_, Y_, and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68

R_ : Angular displacement with a positive value indicates counter clockwise rotation. Bit 0 (RIN) of parameter No. 5400 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).

Least input increment : 0.001 deg

Valid data range : -360,000 to 360,000

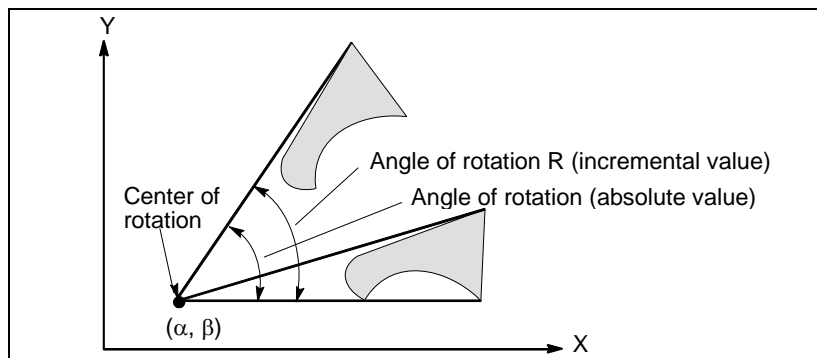


Fig. 6.11 (b) Coordinate system rotation

NOTE

When a decimal fraction is used to specify angular displacement ($R_{.}$), the 1's digit corresponds to degree units.

Explanation

- **G code for selecting a plane: G17,G18 or G19**

The G code for selecting a plane (G17,G18,or G19) can be specified before the block containing the G code for coordinate system rotation (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. 6.11 (c)).

- **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.

- 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.

- Note on the specification of one axis in coordinate system rotation

With the parameter below, a move position in the case where one axis is specified in the absolute mode can be selected. If two axes are specified, a movement is made to the same position, regardless of the setting of the parameter.

Bit 5 (AX1) of parameter No. 11600

If one axis is specified in the absolute mode when the coordinate system rotation mode is set:

0: The specified position is first calculated in the coordinate system before rotation then the coordinate system is rotated.

1: The coordinate system is first rotated then a movement is made to the specified position in the rotated coordinate system. (FS16i/18i/21i-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

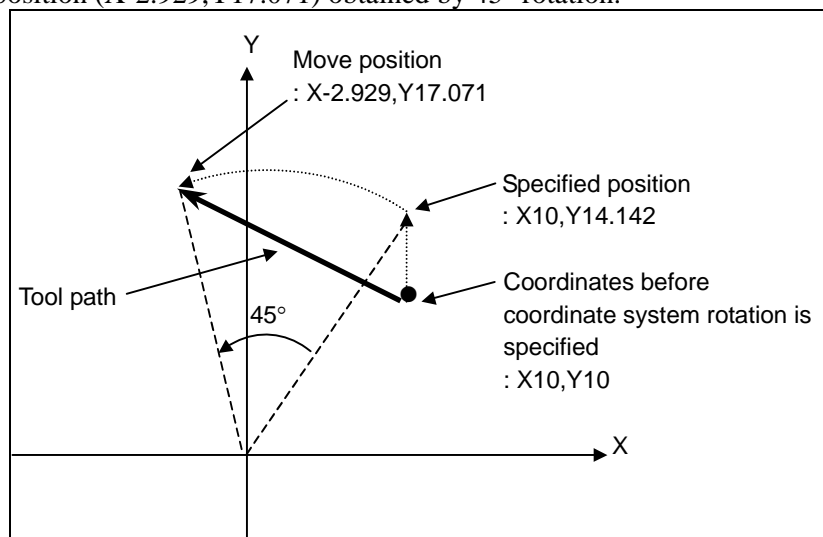
G68 X0 Y0 R45..... Specifies coordinate system rotation.

Y14.142..... Specifies one axis(1)

G69

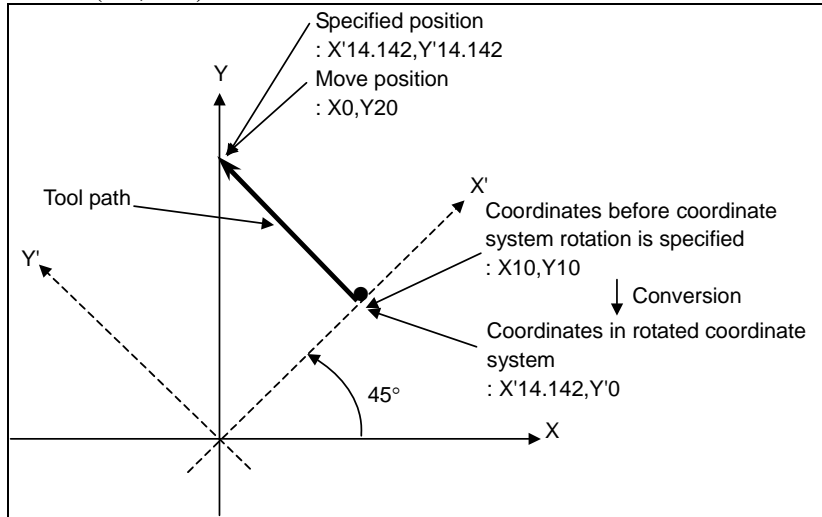
When bit 5 (AX1) of parameter No. 11600= 0:

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.



When bit 5 (AX1) of parameter No. 11600= 1:

With the specification of (1), coordinates (X10,Y10) before coordinate system rotation are converted to coordinates (X'14.142,Y'0) in the coordinate system (X'Y') obtained by 45° rotation. Next, a movement is made to the specified position (X'14.142,Y'14.142), that is, the move position (X0,Y20).



Explanation

- Absolute/Incremental position commands

```

N1 G92 X-500.0 Y-500.0 G69 G17 ;
N2 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 ;
    
```

Tool path when the incremental command is designated in the N3 block (in parenthesis)

Originally programmed tool path

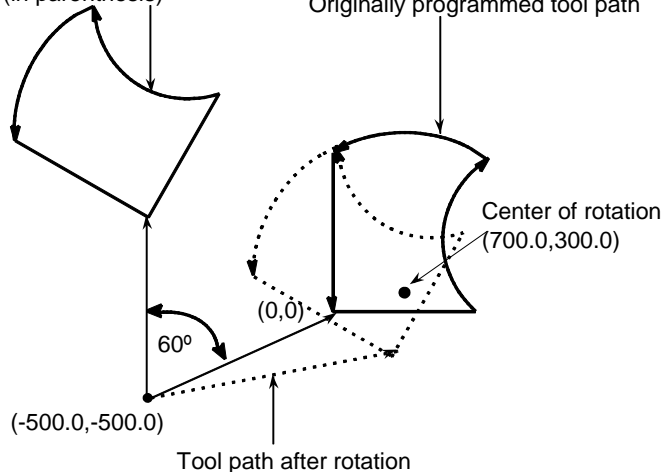


Fig. 6.11 (c) Absolute/incremental programming during coordinate system rotation

- Cutter compensation and coordinate system rotation

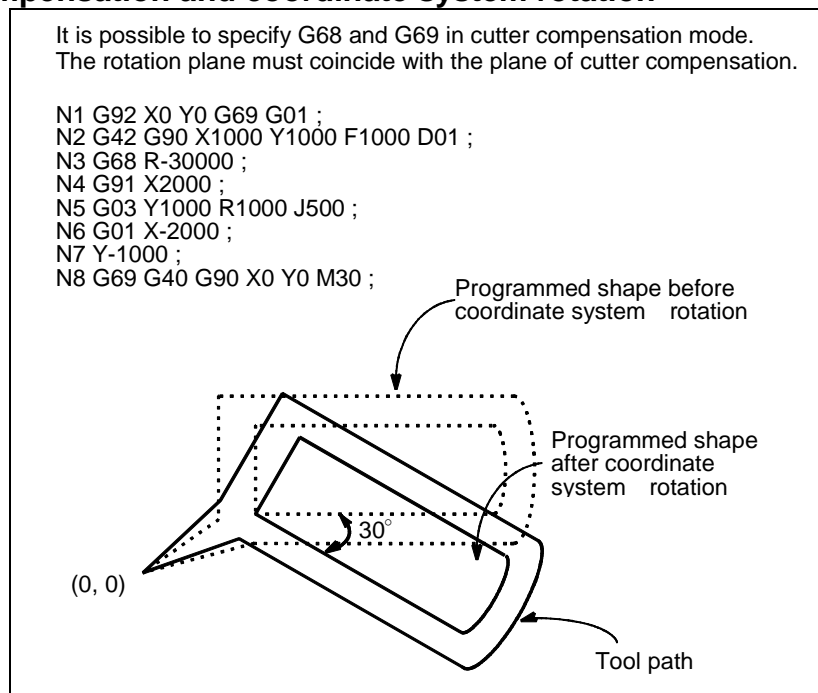


Fig. 6.11 (d) Cutter compensation and coordinate system rotation

- 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.

- 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
- When the system is in cutter compensation, specify the commands in the following order (Fig. 6.11 (e)) :
 - (cutter compensation cancel)
 - G51 ; Scaling mode start
 - G68 ; Coordinate system rotation start
 - :
 - G41 ; Cutter compensation mode start
 - :

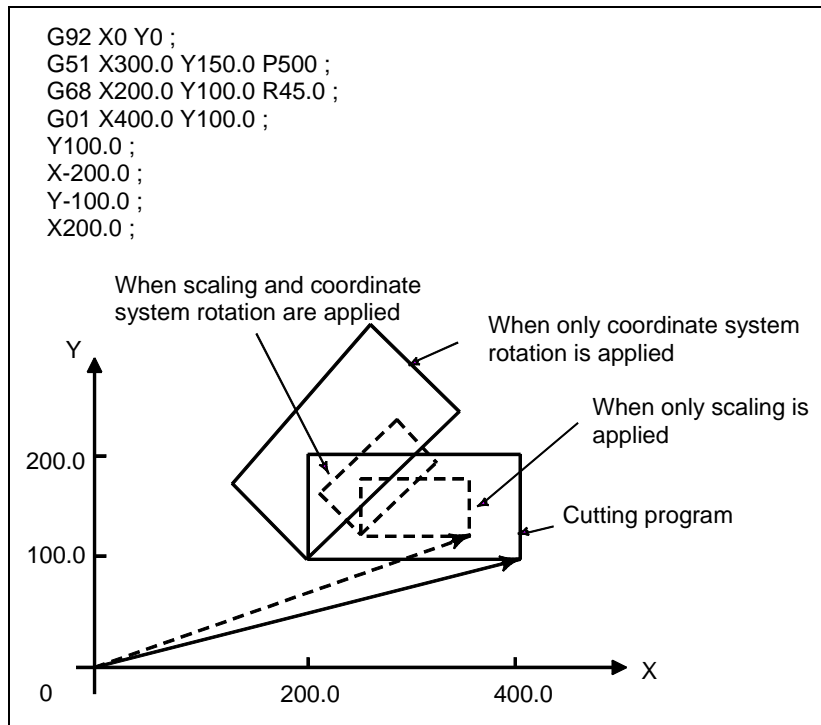


Fig. 6.11 (e) Scaling and coordinate system rotation in cutter compensation mode

- 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 ;
```

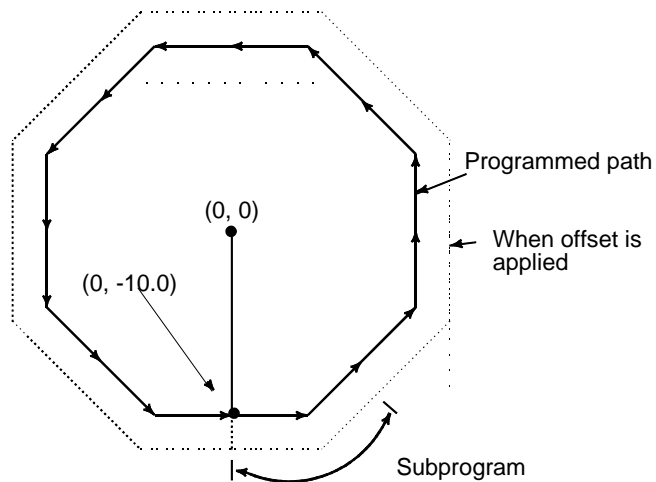


Fig. 6.11 (f) Coordinate system rotation command

6.12 GRINDING WHEEL WEAR COMPENSATION

A compensation vector is created on an extension of the line from a specified point (compensation center) to a specified end point position on a specified compensation plane.

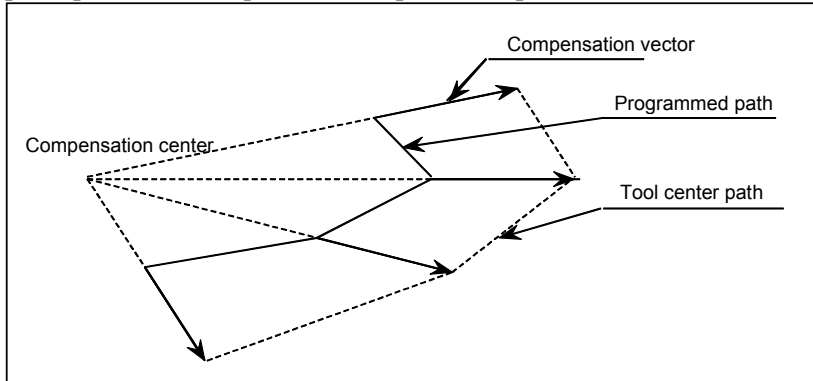


Fig. 6.12 (a)

Format

Compensation center selection

G41 P_(n=1,2,3);

G41P1: First compensation center selection Coordinates
(parameters Nos. 5081 and 5082))

G41P2: Second compensation center selection Coordinates
(parameters Nos. 5083 and 5084))

G41P3: Third compensation center selection Coordinates
(parameters Nos. 5085 and 5086))

Start-up

D_ ; non-zero D code

Compensation mode cancel

D0;

Compensation vector retention

G40;

Explanation

- Compensation center setting and selection

There can be three compensation centers, and the coordinates of the centers can be set with parameters Nos. 5081 to 5086.

Which of these three centers to use can be determined with G41Pn (where n = 1, 2, or 3).

G41 P1 ; First compensation center selection

G41 P2 ; Second compensation center selection

G41 P3 ; Third compensation center selection

When selecting a compensation center, be sure to specify P1, P2, or P3 at the same time.

If no P is specified or if a value other than 1 to 3 is specified, alarm PS 1618, "ILLEGAL P-DATA(WHEEL WEAR COMPENSATION)" is issued.

The coordinates of the compensation centers (parameters Nos. 5081 to 5086) must be set as those in the workpiece coordinate system.

Start-up

By specifying a compensation center and specifying a non-zero D code, the system enters compensation mode.

Even if the block in which the D code is specified does not contain a move command, a compensation vector is created and movement is performed.

- Compensation vector

A compensation vector is created on an extension of the line from the compensation center to a specified end point position, the length of the vector being equal to the offset value specified with its offset number with a D code. (See Fig. 6.12 (a).)

If the offset value is positive, the compensation vector is added to the specified end point position, and if the offset value is negative, the compensation vector is subtracted from the specified end point position.

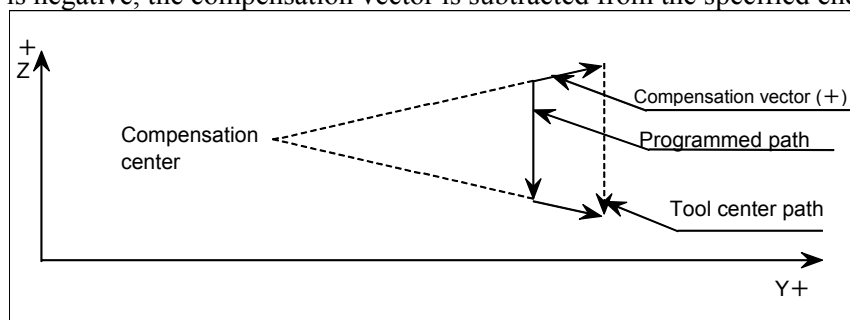


Fig. 6.12 (b) Offset value (+)

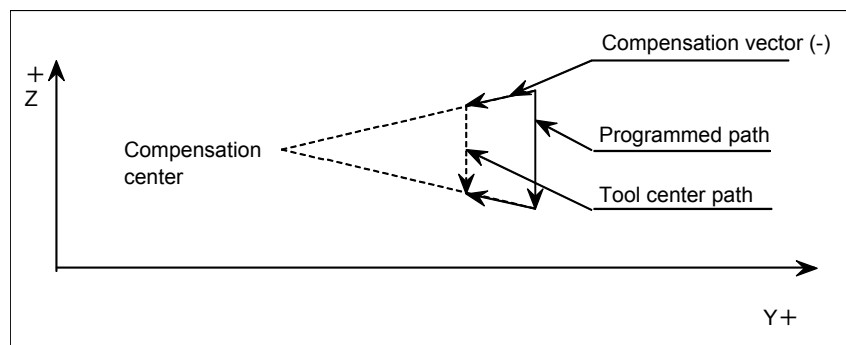


Fig. 6.12 (c) Offset value (-)

Compensation mode cancel

Specifying D0 causes the compensation mode to be canceled, so that the compensation vector is not created.

- Compensation vector retention

By specifying G40, it is possible to place the system in a mode in which the created compensation vector is retained. Until the system enters the next new compensation mode, the specified end point position is shifted by the vector.

If D0 is specified in the compensation vector retention mode, the retained vector is cleared and the compensation mode is canceled.

- Circular and helical interpolation

This compensation is also effective to circular interpolation. If the radius at the start point of the arc differs from the radius at the end point, the arc cannot be a correct one. It will be a spiral one.

This is also true of helical interpolation.

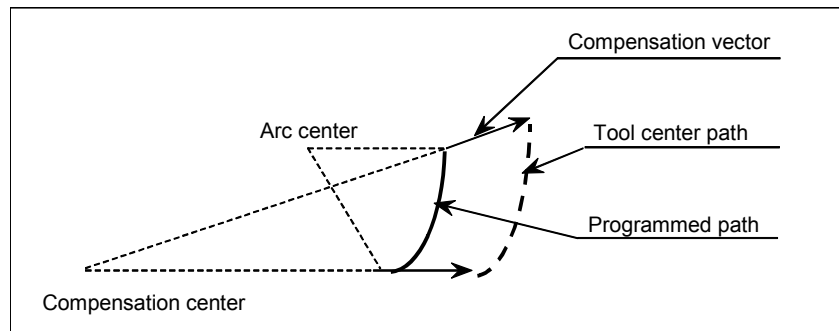


Fig. 6.12 (d)

A check of the arc radius error limit (parameter No. 3410) is performed even on the value after compensation.

- Circular interpolation in G40 mode

Executing a G40 mode arc command causes the arc center to be shifted by the vector. This makes an arc, not a spiral.

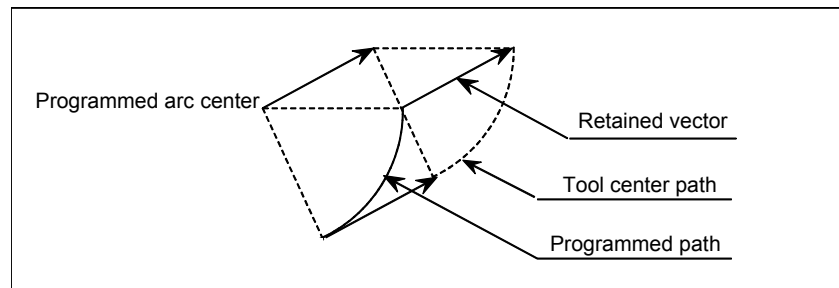


Fig. 6.12 (e)

Exponential interpolation

This compensation is also effective to exponential interpolation. Exponential interpolation is performed on the position after compensation is applied.

- Compensation plane and plane selection with G17/G18/G19

In the compensation mode (and in the compensation vector retention mode), a compensation vector is constantly created for the axes on the compensation plane determined with a parameter. Creation of a compensation vector is not related to the plane selection with G17/G18/G19.

It is possible to apply compensation on a compensation plane (for example, the YZ plane) while performing circular interpolation on the XY (G17) plane.

If, in the compensation mode, a move command is issued for one compensation axis, and the compensation vector components on the other axis are changed due to the creation of a compensation vector, movement along this axis is performed.

(Example 1)

The compensation axes are set as the Y- and Z-axes, and a linear interpolation command is executed on the X- and Y-axes.

Programmed path $a \rightarrow b$

Path after compensation $a' \rightarrow b'$

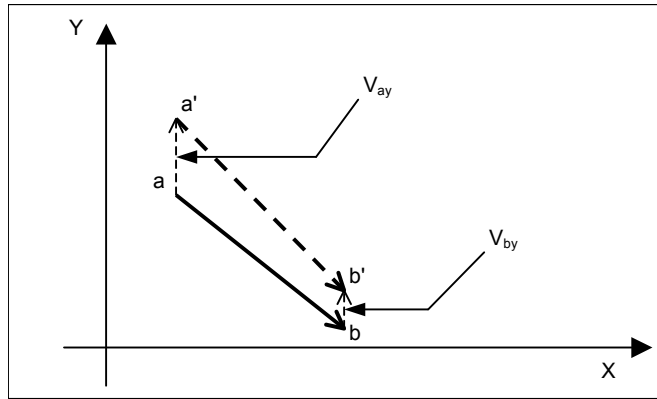


Fig. 6.12 (f) Path on the X-Y plane

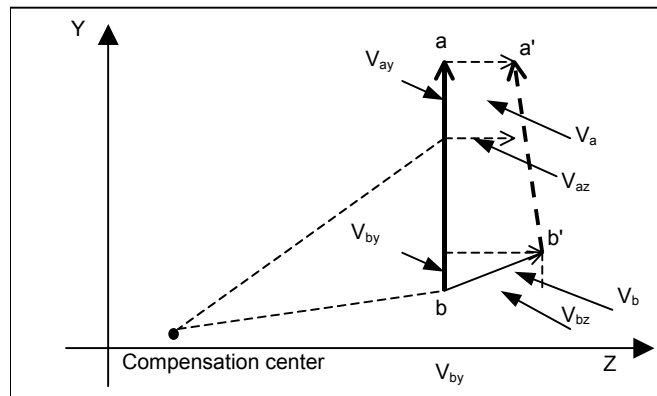


Fig. 6.12 (g) Path on the Y-Z plane

(Example 2)

The compensation axes are set as the Y- and Z-axes, and a circular interpolation command is executed on the X- and Y-axes.

Programmed path $a \rightarrow b$

Path after compensation $a' \rightarrow b'$

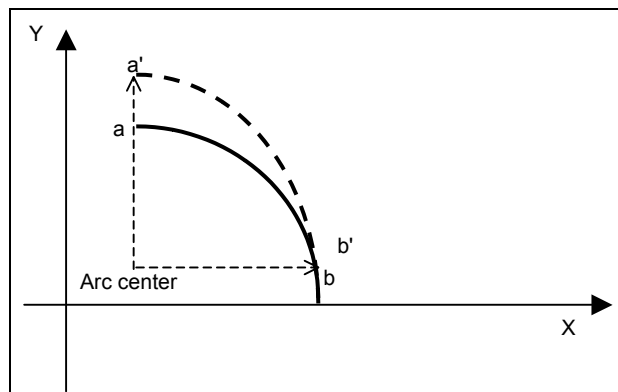


Fig. 6.12 (h) Path on the X-Y plane

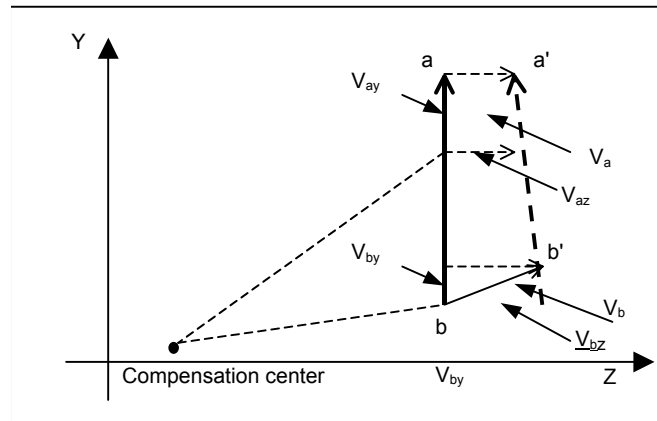


Fig. 6.12 (i) Path on the Y-Z plane

- **Compensation cancel mode**

Immediately after the power is turned on and after a reset, the system is in the compensation cancel mode.

- **Changing the coordinate system**

To change the coordinate system, cancel the compensation mode first.

- **Return to the reference position (G28, G30)**

To specify a return to the reference position (G28, G30), cancel the compensation mode.

Limitation

- **Relations with the coordinate conversion function**

Functions such as programmable mirror image, scaling, and coordinate system rotation cannot be applied to the coordinates of the compensation center.

- **Relations with other offset functions**

In a system with this compensation attached, cutter compensation and 3-dimensional tool compensation cannot be used. This compensation can be used simultaneously as tool length offset and tool offset.

- **Changing the compensation axis**

To change the compensation axis, enter the compensation cancel mode.

6.13 ACTIVE OFFSET VALUE CHANGE FUNCTION BASED ON MANUAL FEED

Overview

When rough machining/semifinish machining is to be performed using a single tool, you may make a fine adjustment of a tool length compensation value or cutter compensation value. Moreover, at setup time, you may want to make a fine adjustment of a workpiece origin offset once set. With this function, a travel distance moved on an axis by manual feed is automatically added to the workpiece coordinate system or the currently valid offset number among the specified offset values (tool length compensation value/cutter compensation value/workpiece origin offset) to make a offset value change.

Explanation

- Active offset value change mode

The active offset value change mode is set using the active offset value change mode signal. In this mode, a travel distance moved on an axis by manual feed is automatically added to the workpiece coordinate system or the currently valid offset number among the specified offset values (tool length compensation value/cutter compensation value/workpiece origin offset). The types of manual feed usable to make an offset value change in this mode are manual handle feed, incremental feed, and jog feed.

CAUTION

- 1 When a movement is being made on an axis for which an offset value is to be changed, do not set the active offset value change mode.
- 2 In the active offset value change mode, do not reset the relative coordinate to 0 or preset the relative coordinate to a specified value.

- Specifying an offset value to be changed

The active offset selection signal is used to specify one of three types of offset values: tool length compensation value, cutter compensation value, and workpiece origin offset. In the active offset value change mode, an offset value selected is indicated by blinking display in the state display area on the screen as follows:

Offset value selected	State display
Tool length compensation value	LEN
Cutter compensation value	RAD
Workpiece origin offset	WZR

CAUTION

When a movement is being made on an axis for which an offset value is to be changed in the active offset value change mode, do not change the specification of the offset value to be changed.

- Changing a tool length compensation value

The tool length compensation value with the offset number corresponding to an H code specified in automatic operation is changed. If there is no currently valid tool length compensation value as in a case where no H code is specified after a cycle start, no tool length compensation value change is made even when a movement is made on an axis by manual feed.

With a movement on a linear axis, a tool length compensation value change can be made. With a movement on a rotation axis, no tool length compensation value change can be made. While a tool length compensation value is being changed, a movement by manual feed can be made on one axis only.

Example

- Specified H code: H10
- Value set with offset number 10: 54.700 mm
- Travel distance on the Z-axis by manual feed: -2.583 mm

In this example, the value of offset number 10 becomes:

$$54.700 + (-2.583) = 52.117 \text{ mm}$$

⚠ CAUTION

A tool length compensation value can be changed by a movement on any linear axis. When an offset value change for an axis is undesirable, interlock the axis.

NOTE

A changed tool length compensation value is handled according to bit 6 (EVO) of parameter No. 5001 and bit 6 (AON) of parameter No. 5041.

- Changing a cutter compensation value

The cutter compensation value with the offset number corresponding to a D code specified in automatic operation is changed. If there is no currently valid cutter compensation value as in a case where no D code is specified after a cycle start, no cutter compensation value change is made even when a movement is made on an axis by manual feed.

With a movement on a linear axis, a cutter compensation value change can be made. With a movement on a rotation axis, no cutter compensation value change can be made. While a cutter compensation value is being changed, a movement by manual feed can be made on one axis only.

When operation is stopped in the cutter compensation mode to make a cutter compensation value change, a movable travel distance on one axis is added, regardless of the direction of the compensation vector at stop time.

Example

- Specified D code: H15
- Value set with offset number 15: 6.500mm
- Travel distance on the X-axis by manual feed: 2.379mm
- Travel distance on the Y-axis by manual feed: -0.572mm

In this example, the value of offset number 15 becomes:

$$6.500 + 2.379 + (-0.572) = 8.307 \text{ mm}$$

⚠ CAUTION

A cutter compensation value can be changed by a movement on any linear axis. When an offset value change for an axis is undesirable, interlock the axis.

NOTE

A changed cutter compensation value is handled according to bit 4 (EVR) of parameter No. 5001.

- Changing a workpiece origin offset value

The workpiece origin offset of the workpiece coordinate system corresponding to a G code from G54 to G59 or from G54.1 P1 to P48 (300) specified during automatic operation is changed on an axis-by-axis basis. A valid workpiece coordinate system exists at all times. So, when a movement is made on an axis by manual feed, the workpiece origin offset of the workpiece coordinate system is changed without fail. This change can be made by a movement on an arbitrary axis, which may be a linear axis or a rotation axis. While a workpiece origin offset change is being made, movements can be made on multiple axes by manual feed.

Example

- Specified workpiece coordinate system : G56
- Workpiece origin offset of G56 (X axis) : 50.000
- Workpiece origin offset of G56 (Y axis) : -60.000
- Workpiece origin offset of G56 (Z axis) : 5.000
- Workpiece origin offset of G56 (A axis) : 5.000
- Workpiece origin offset of G56 (B axis) : 15.000
- Travel distance on the X axis by manual feed : -10.000mm
- Travel distance on the Y axis by manual feed : -5.000mm
- Travel distance on the Z axis by manual feed : 10.000mm
- Travel distance on the A axis by manual feed : 8.000mm
- Travel distance on the B axis by manual feed : -2.000mm

In this example, the workpiece origin offsets of G56 are as follows:

- X axis : $50.000 + (-10.000) = 40.000$
- Y axis : $-60.000 + (-5.000) = -65.000$
- Z axis : $5.000 + 10.000 = 15.000$
- A axis : $5.000 + 8.000 = 13.000$
- B axis : $15.000 + (-2.000) = 13.000$

- Operation depending on each tool offset memory

Offset value change operation varies according to tool offset memory A/B/C as follows:

Tool offset memory	Changed offset value
A	No distinction is made between a tool length compensation value and cutter compensation value. The value specified with the offset number corresponding to the currently valid H code or D code is changed.
B	No distinction is made between a tool length compensation value and cutter compensation value. The value specified with the offset number corresponding to the currently valid H code or D code is changed. Depending on the setting of bit 4 (ASG) of parameter No. 5000, a geometry compensation value or wear compensation value is changed.
C	The tool length compensation value and cutter compensation value specified with the offset numbers corresponding to the currently valid H code and D code are changed. Depending on the setting of bit 4 (ASG) of parameter No. 5000, a geometry compensation value or wear compensation value is changed.

- Presetting the relative position indication

By setting bit 5 (APL) of parameter No. 3115 to 1, the relative position indication (counter) can be automatically preset to 0 when the active offset value change mode is selected. In this case, the changed offset value can be restored to the original value by returning the relative position indication (counter) to 0 by manual feed.

- Emergency stop, servo alarm

If an emergency stop occurs, a servo alarm is issued, or servo excitation is turned off, an offset value change is made also for a travel distance on an axis moved by follow-up in the active offset value change mode.

NOTE

If a tool length compensation value or cutter compensation value is selected as an offset value to be changed, no offset value change is made for a travel distance on a rotation axis moved by follow-up.

Limitation

- Manual operation that cannot change an active offset value

In a mode other than the manual handle feed mode/incremental feed mode/jog feed mode, no active offset value can be changed.

Moreover, no active offset value can be changed in the manual reference position return mode.

Even in the modes mentioned above, do not change an active offset value in the following operations:

- 3-dimensional manual feed
- Manual numerical command
- PMC axis control

- Axis that disables an active offset value from being changed

With a rotation axis, no tool length compensation/cutter compensation value can be changed using this function.

6.14 ROTARY TABLE DYNAMIC FIXTURE OFFSET

Overview

The rotary table dynamic fixture offset function saves the operator the trouble of resetting the workpiece coordinate system when the rotary table rotates before cutting is started. With this function the operator simply sets the position of a workpiece placed at a certain position on the rotary table as a reference fixture offset. If the rotary table rotates, the system automatically obtains a current fixture offset from the angular displacement of the rotary table and creates a suitable workpiece coordinate system. After the reference fixture offset is set, the workpiece coordinate system is prepared dynamically, wherever the rotary table is located.

The zero point of the workpiece coordinate system is obtained by adding the fixture offset to the offset from the workpiece reference point.

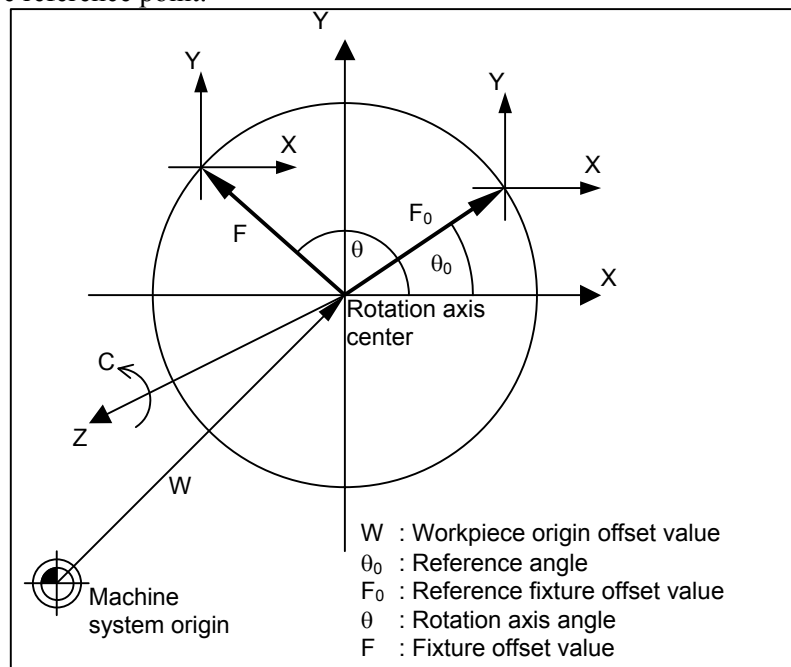


Fig. 6.14 (a) Fixture offset

Format**- Fixture offset command****G54.2 Pn ;**

n : Reference fixture offset value number (1 to 8)

- Fixture offset cancel command**G54.2 P0 ;****NOTE**

- 1 In the G54.2 mode, a change made to the setting of parameter or to the reference fixture offset becomes effective when the next block is buffered.
- 2 It depends on the current continuous-state code of the 01 group whether a change in the fixture offset vector causes a movement. If the system is in a mode other than the G00 or G01 mode (G02, G03, etc.), the movement is made temporarily in the G01 mode.
- 3 When a rotation axis that is related to fixture offset, command is specified in the G54.2 mode, the vector is calculated with the coordinate value of the end of the block and the movement is performed to the command position on the coordinate system pointed by the vector .
- 4 In calculation of the fixture offset, the coordinate of the rotation axis on the workpiece coordinate system is used. If a tool offset or another offset is applied, the coordinate before the offset is used.
If mirror image or scaling is performed, the coordinate before the operation is used.

Explanation**- Fixture offset command**

When a command G54.2Pn is specified, a fixture offset value is calculated from the current rotation angle and the data specified with n, and enable the fixture offset value.

If n = 0, the fixture offset value is disabled.

- When a move command is specified for a rotation axis in G54.2 mode

When a command to move the tool about a rotation axis involved with a fixture offset is specified in the G54.2 mode, the coordinates about the rotation axis at the end of the block are used to calculate a vector. The tools moved to the specified position on the workpiece coordinate system that is indicated by the vector.

- Operation at reset

Whether to cancel the fixture offset at a reset depends on the settings of bit 6 (CLR) of parameter No. 3402 and of bit 7 (C23) of parameter No. 3408.

When CLR is set to 0 or CLR and C23 are set to 1, the vector before the reset is saved.

When CLR is set to 1 and C23 is set to 0, the vector is cleared. The machine does not move by the cleared vector regardless of the setting of bit 0 (FTP) of parameter No. 7570, however.

- Data setting

- (1) Setting a group of three parameters which specify one rotation axis and two linear axes constituting the plane of rotation (Parameters Nos. 7580 to 7588).

In each group, specify the number of the rotation axis as the first parameter and the numbers of the linear axes as the second and third parameters. The rotation in the normal direction about the rotation axis must agree with the rotation from the positive side of the linear axis set as the second parameter to the positive side of the linear axis set as the third parameter.

Example)

Suppose that a machine has four axes, X, Y, Z, and C. The X-, Y-, and Z-axes form a right-handed coordinate system. The C-axis is a rotation axis. When viewed from the positive side of the Z-axis, a rotation in the normal direction about the C-axis is treated as the counterclockwise rotation around the Z-axis.

For this machine, specify the parameters as follows

First parameter : 4 (C-axis)

Second parameter : 1 (X-axis)

Third parameter : 2 (Y-axis)

Up to three groups of parameters can be set. In calculation of the fixture offset, the data of the rotation axis specified in the first group is calculated first. Then, the data of the second and third groups are calculated.

If a machine has two or more rotation axes and the plane of rotation depends on the rotation about another rotation axis, the plane of rotation is set when the angular displacement about the rotation axis is 0.

- (2) Setting the reference angle of the rotation axis and the corresponding reference fixture offset
Set the reference angle of the rotation axis and the fixture offset that corresponds to the reference angle.
Set the data on the fixture offset screen. Eight groups of data items can be specified.
- (3) Setting a parameter for enabling or disabling the fixture offset of each axis
(bit 0 (FAX) of parameter No. 7575)
For the axis for which the fixture offset is enabled, set the parameter to 1. This need not be specified for a rotation axis.
- (4) Setting the type of fixture offset (bit 0 (FTP) of parameter No. 7570)
Specify whether to cause a movement according to the increment or decrement of the fixture offset vector when the vector changes (when G54.2 is specified or when a rotation axis movement occurs in the G54.2 mode).
When 0 is set, the movement is made. (The current position on the workpiece coordinate system does not change. The position on the machine coordinate system changes.)
When 1 is set, the movement is not made. (The current position on the workpiece coordinate system changes. The position on the machine coordinate system does not change.)

- Input/output of fixture offset

The data can be programmed and can be input from and output to external equipment, as described below:

- (1) Setting the reference fixture offset by G10
G10 L21 Pn P ;
n: Reference fixture offset number
P: Reference fixture offset or reference angle of each axis
With this command, a reference fixture offset or reference angle can be programmed.
If the command is executed in the G90 mode, the specified value is set directly. If the command is executed in the G91 mode, the sum of the specified value and the previous value is set.
- (2) Reading/writing based on a custom macro system variable
The following system variable number can be used to read and write a reference fixture offset value or a reference angle. However, it is impossible to write to a system variable area (5500 to 5508) if n = 0.

System variable number = 5500 + 20 * n + m

n: Fixture offset number (1 to 8)

(The current offset is used if n = 0.)

m: Axis number (1 to number of controlled axes)

NOTE

The custom macro function is needed.

(3) Output to external units

Selecting [OUTPUT] on the fixture offset screen enables outputting to external units such as a floppy cassette and memory card via RS-232-C.

Output data is in the G10 format with no program number.

NOTE

The reader/punch interface function is needed.

(4) Input from an external units.

Selecting [NPUT] on the program edit screen, the data can be input from a Floppy Cassette and memory card via RS-232-C.

(Amount of the Fixture offset is set by inputting as NC program, and executing it.)

NOTE

The reader/punch interface function is needed.

- Calculating a Fixture Offset values

(1) Relationship between the rotation axes and linear axes

First group : 4 (B-axis) , 3 (Z-axis) , 1 (X-axis)

Second group : 5 (C-axis) , 1 (X-axis) , 2 (Y-axis)

Third group : 0 , 0 , 0

(2) Reference angle and reference fixture offset

X : F_{0X}

Y : F_{0Y}

Z : F_{0Z}

B : θ_0

C : ϕ_0

If the above data is set up, the method of calculating fixture offset value is as follows:

O : Rotary table center

W : Workpiece origin offset value

F_0 : Fixture offset value when $B=\theta_0$, $C=\phi_0$

F_A : Fixture offset value (F_{AX}, F_{AY}, F_{AZ}) when $B=0$, $C=0$

F : Fixture offset value (F_X, F_Y, F_Z) when $B=\theta$, $C=\phi$

Then, the following expression is used for fixture offset calculation.

$$\begin{bmatrix} F_{AX} \\ F_{AY} \\ F_{AZ} \end{bmatrix} = \begin{bmatrix} \cos(-\theta_0) & 0 & \sin(-\theta_0) \\ 0 & 1 & 0 \\ -\sin(-\theta_0) & 0 & \cos(-\theta_0) \end{bmatrix} \begin{bmatrix} \cos(-\phi_0) & -\sin(-\phi_0) & 0 \\ \sin(-\phi_0) & \cos(-\phi_0) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} F_{0X} \\ F_{0Y} \\ F_{0Z} \end{bmatrix}$$

$$\begin{bmatrix} F_X \\ F_Y \\ F_Z \end{bmatrix} = \begin{bmatrix} \cos(\phi) & -\sin(\phi) & 0 \\ \sin(\phi) & \cos(\phi) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \cos(\theta) & 0 & \sin(\theta) \\ 0 & 1 & 0 \\ -\sin(\theta) & 0 & \cos(\theta) \end{bmatrix} \begin{bmatrix} F_{AX} \\ F_{AY} \\ F_{AZ} \end{bmatrix}$$

- If manual intervention is made on the rotation axis

When the automatic operation is stopped by the SBK stop or similar in the G54.2 mode, and a manual movement is made about the rotation axis, the vector of the fixture offset does not change. When a rotation axis command is specified in automatic operation or in MDI operation or When G54.2 is specified, the vector is calculated.

When manual intervention is performed with bit 3 (CFA) of parameter No. 7570 =0 and in the manual absolute switch is set on and then a rotation axis command is specified in the incremental (G91) mode, the vector is calculated using the coordinates which do not reflect the amount of manual intervention.

Example)

N1 G90 G00 C10.0 ;

N2 G54.2 P1 ;

After executing the program, perform manual intervention with the manual absolute switch is set to on. Then, movement of +20.0 about the C-axis.

After restart

N3 G91 C30.0 ;

is specified, the coordinate value of C-axis is 60.0 in the workpiece coordinate system.

In the fixture offset calculation, however, the coordinate value of C-axis is considered as 40.0.

If, in N3, the bit 3 (CFA) of parameter No. 7570 is 1 and bit 1 (ABS) of parameter No. 7001 is 1, the coordinate value of C-axis is considered as 10.0 (30.0 - 20.0) as specified when the calculation is performed.

Limitation

- Multi-path system

This function cannot be used with the multi-path system.

- Command for suppressing fixture offset calculation

If the following commands are specified for the rotation axis in the G54.2 mode, the fixture offset vector is not calculated:

Command related to the machine coordinate system: G53

Command specifying a change of the workpiece coordinate system: G54 to G59, G54.1, G92, and G52

Command specifying a return to the reference position: G27, G28, G29, G30, G30.1

- Rotation axis used for fixture offset

The rotation axis used for polar coordinate interpolation (G12.1) cannot be set as the rotation axis for the fixture offset.

- Rotation axis roll over

When using the rotary axis roll over function, always specify 360 degrees for the amount of travel per revolution of the rotation axis.

- Functions that cannot be specified

In the G54.2 mode, the functions listed below cannot be specified.

Coordinate system rotation function

Figure copying function

Example

Parameter

No. 7580=4 (C-axis)

No. 7581=1 (X-axis)

No. 7582=2 (Y-axis)

Nos. 7583 to 7588=0

No. 7575#0(X)=1 (The offset is valid for the X-axis.)

No. 7575#0(Y)=1 (The offset is valid for the Y-axis.)

No. 7570#0=0 (When No. 7570#0=1, the values in square brackets ([]) are calculated.)

Data of fixture offset 1 (n = 1)

C= 180.0 (reference angle)

X= -10.0

Y= 0.0

When these parameters and data are set, the machine operates as shown below :

Table 6.14 (a) Example of fixture offset

Coordinates Program	Position on the workpiece coordinate system (ABSOLUTE)			Position on the machine coordinate system (MACHINE)			Fixture offset		
	X	Y	C	X	Y	C	X	Y	C
N1 G90 G00 X0 Y0 C90. ;	0.0	0.0	90.0	0.0	0.0	90.0	0.0	0.0	0.0
N2 G54.2 P1 ;	0.0	0.0	90.0	0.0	10.0	90.0	0.0	10.0	0.0
	[0.0	-10.0	90.0]	[0.0	0.0	90.0]	[0.0	10.0	0.0]
N3 G01 X10. Y2. F100. ;	10.0	2.0	90.0	10.0	12.0	90.0	0.0	10.0	0.0
N4 G02 X2. Y10. R10. ;	2.0	10.0	90.0	2.0	20.0	90.0	0.0	10.0	0.0
N5 G01 X0 Y0 ;	0.0	0.0	90.0	0.0	10.0	90.0	0.0	10.0	0.0
...									

The values enclosed in brackets ([]) apply when bit 0 (FTP) of parameter No. 7570 is set to 1.

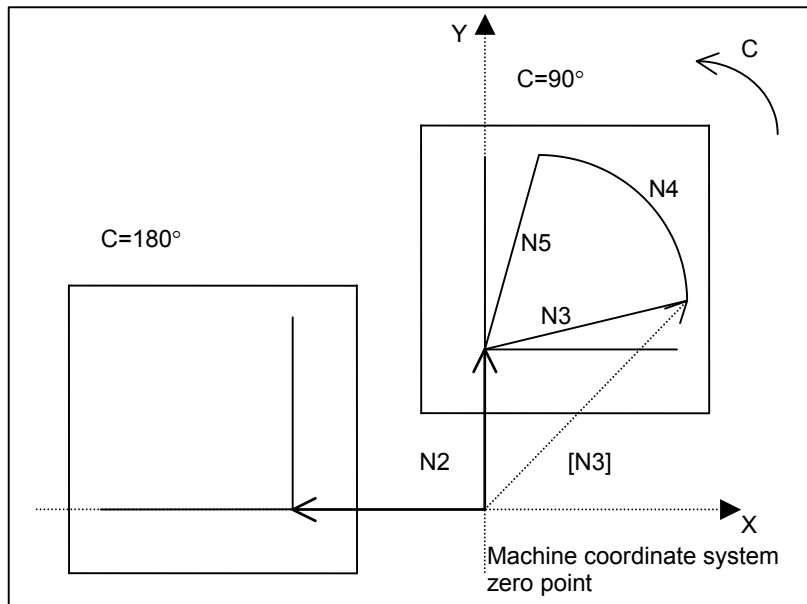


Fig. 6.14 (b) Example of fixture offset

When G54.2 P1 is specified in the N2 block, the fixture offset vector (X=0, Y=10.0) is calculated. The vector is handled in the same way as the offset from the workpiece reference point. The current position on the workpiece coordinate system is (X=0, Y=-10.0). If bit 0 (FTP) of parameter 7570 is set to 0, the tool is moved according to the vector. The resultant position on the workpiece coordinate system is (X=0, Y=0), the position before the command is specified.

6.15 TOOL AXIS DIRECTION TOOL LENGTH COMPENSATION

Overview

When a five-axis machine that has two axes for rotating the tool is used, tool length compensation can be performed in a specified tool axis direction on a rotation axis. When a rotation axis is specified in tool axis direction tool length compensation mode, tool length compensation is applied in a specified tool axis direction on the rotation axis by the compensation value specified in the H code. That is, movement is made along the three linear axes (X_p , Y_p , Z_p). Unless otherwise noted in the explanation of this function, the two rotation axes are assumed to be the B-axis and C-axis.

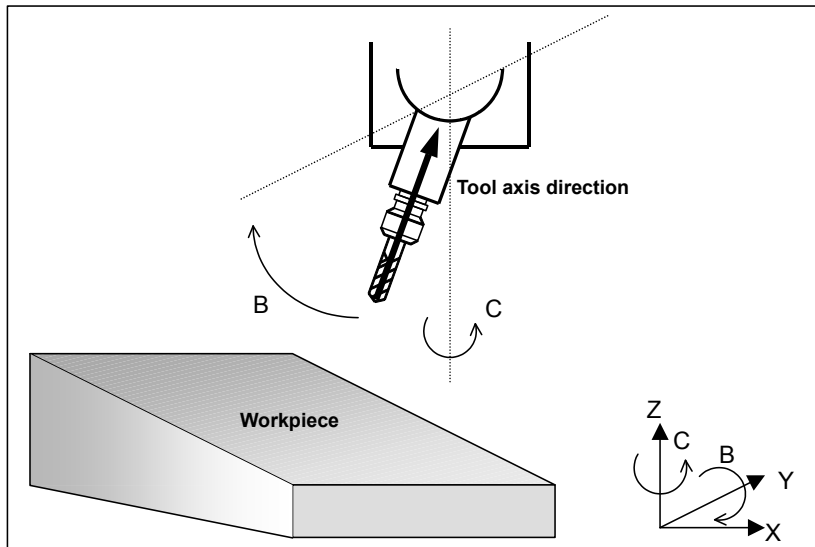


Fig. 6.15 (a) Tool Axis Direction Tool Length Compensation

Format

- Tool axis direction tool length compensation command

G43.1 Hn ;

n: Compensation number

- Tool axis direction tool length compensation cancel command

G49 ;

Explanation

- Command for tool axis direction tool length compensation

The tool compensation vector changes as the offset value changes or movement is made on a rotation axis. When the tool compensation vector changes, movement is made according to the change value along the X-axis, Y-axis, and Z-axis.

When the command specifies movement on a rotation axis only, the position of the tool tip is the same both before and after execution of the command. (During rotation axis movement, however, the tool tip moves.)

- Examples of machine configuration and rotation axis calculation formats

Let V_x , V_y , V_z , L_c , a , b , and c be as follows :

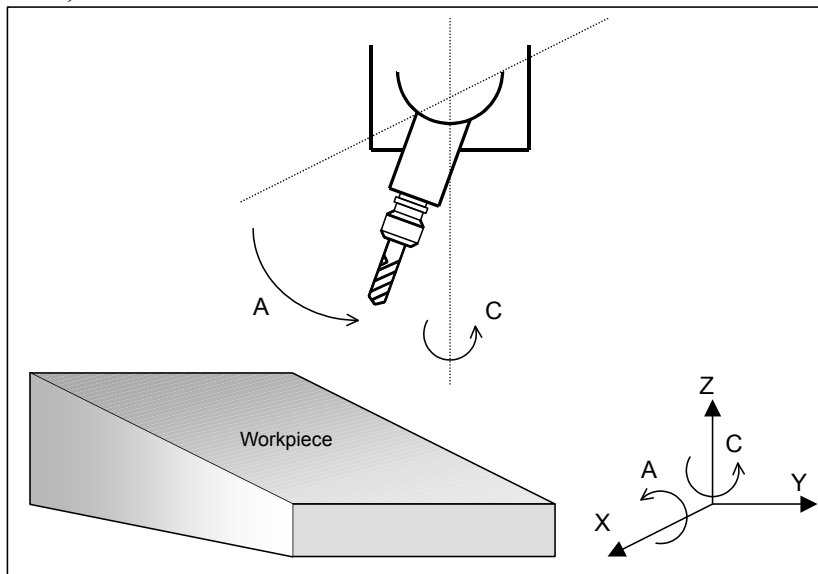
V_x, V_y, V_z : Tool compensation vectors along the X-axis, Y-axis, and Z-axis

L_c : Offset value

a, b, c : Absolute coordinates on the A-axis, B-axis and C-axis

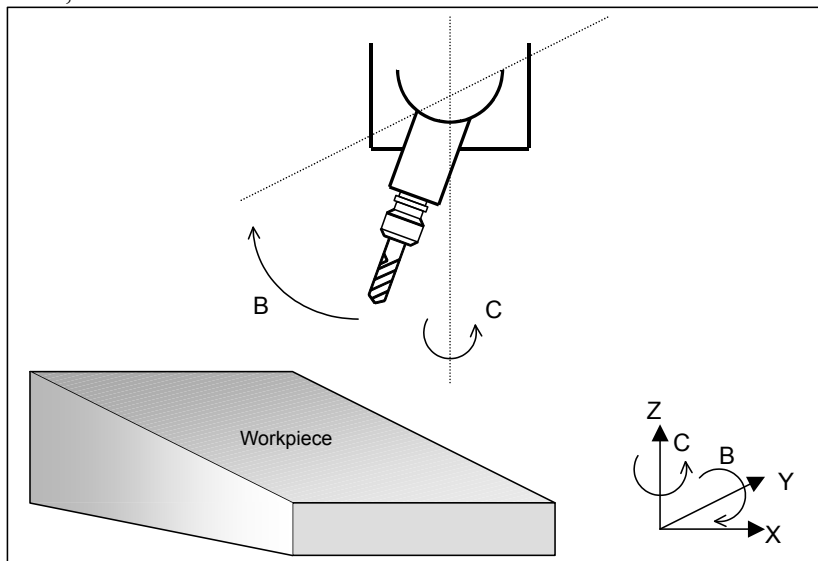
Then, the tool compensation vector on each axis in each machine configuration is indicated below.

- (1) A-axis and C-axis, with the tool axis on the Z-axis



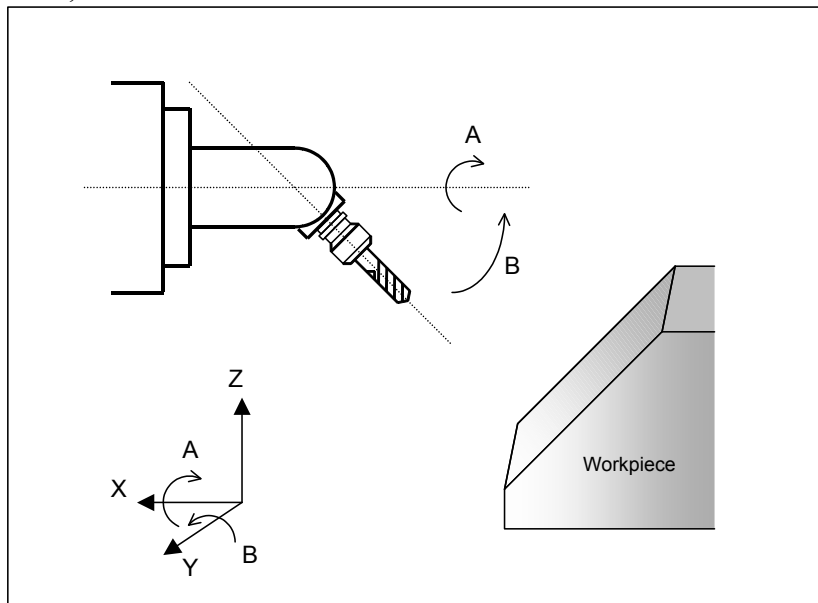
$$\begin{aligned} V_x &= L_c * \sin(a) * \sin(c) \\ V_y &= -L_c * \sin(a) * \cos(c) \\ V_z &= L_c * \cos(a) \end{aligned}$$

- (2) B-axis and C-axis, with the tool axis on the Z-axis



$$\begin{aligned} V_x &= L_c * \sin(b) * \cos(c) \\ V_y &= L_c * \sin(b) * \sin(c) \\ V_z &= L_c * \cos(b) \end{aligned}$$

(3) A-axis and B-axis, with the tool axis on the X-axis

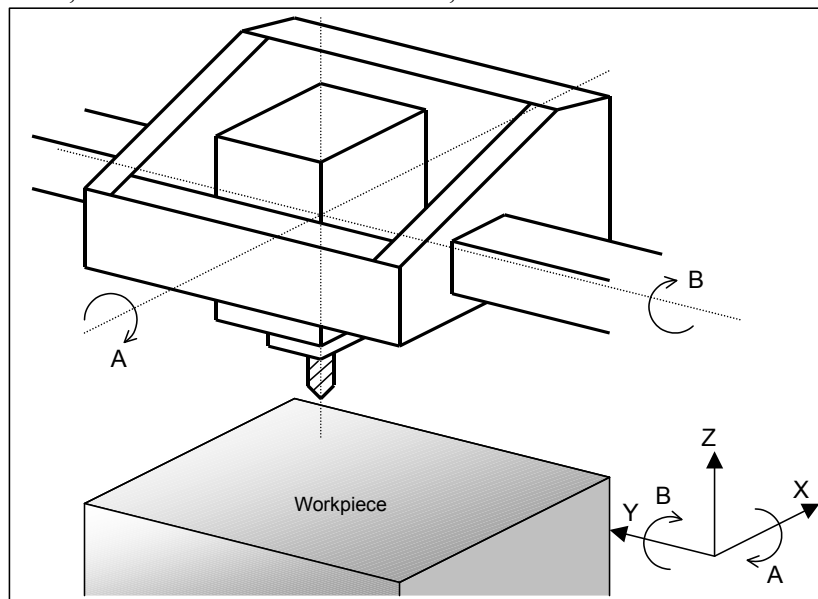


$$V_x = L_c * \cos(b)$$

$$V_y = L_c * \sin(b) * \sin(a)$$

$$V_z = -L_c * \sin(b) * \cos(a)$$

(4) A-axis and B-axis, with the tool axis on the Z-axis, and the B-axis used as the master

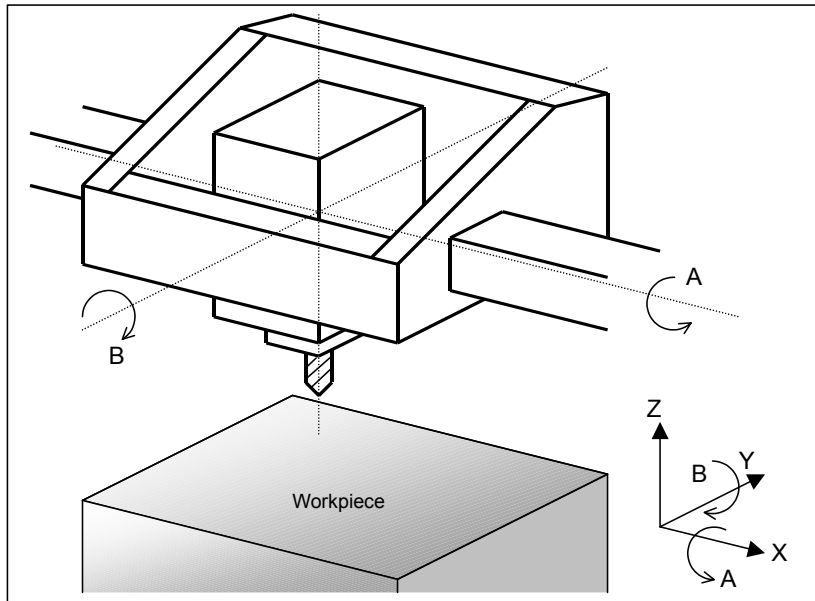


$$V_x = L_c * \cos(a) * \sin(b)$$

$$V_y = -L_c * \sin(a)$$

$$V_z = L_c * \cos(a) * \cos(b)$$

- (5) A-axis and B-axis, with the tool axis on the Z-axis, and the A-axis used as the master



$$\begin{aligned} V_x &= L_c * \sin(b) \\ V_y &= -L_c * \sin(a) * \cos(b) \\ V_z &= L_c * \cos(a) * \cos(b) \end{aligned}$$

- Tool holder offset

The machine-specific length from the rotation center of the tool rotation axes (A- and B-axes, A- and C-axes, and B- and C-axes) to the tool mounting position is referred to as the tool holder offset. Unlike a tool length offset value, a tool holder offset value is set in parameter No.19666. When tool axis direction tool length compensation is applied, the sum of the tool holder offset and tool length offset is handled as a tool length for compensation calculation.

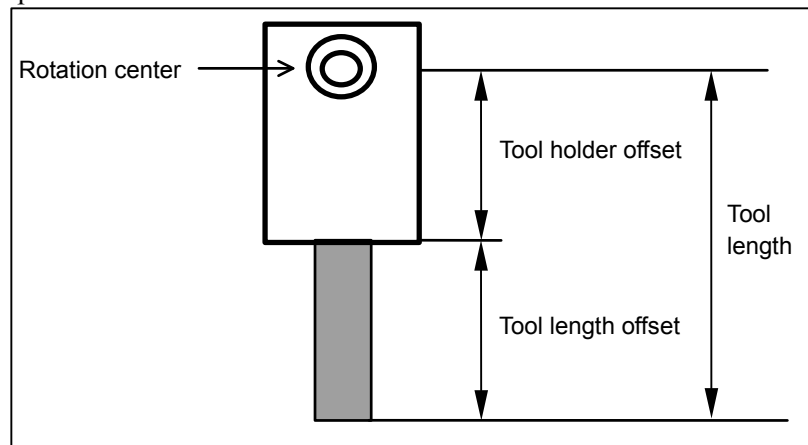


Fig. 6.15 (b) Tool Holder Offset

- Parameter-based rotation angle specification

A tool compensation vector is found from the coordinates on the rotation axes for controlling the tool axis direction. However, the configuration of some machines is such that the tool axis is inclined using a fixed attachment. In such a case, the rotation angles of the rotation axes can be set using parameters. Set bit 1 (RAP) of parameter No. 19650 to 1, and set the coordinates in parameter No. 19658.

- Rotation axis origin compensation

This function compensates for a slight shift of the rotation axis origin caused, for example, by thermal displacement. Specify a compensation value in parameter No. 19660.

When the tool axis is on the Z-axis, and the rotation axes are the B-axis and C-axis, a compensation vector is calculated as follows :

$$X_p = L_c * \sin(B-B_z) * \cos(C-C_z)$$

$$Y_p = L_c * \sin(B-B_z) * \sin(C-C_z)$$

$$Z_p = L_c * \cos(B-B_z)$$

X_p, Y_p, Z_p : Compensation pulse on each axis after origin shift compensation

L_c : Offset value

B, C : Machine position on B-axis and C-axis

B_z, C_z : Origin compensation value on B-axis and C-axis

- Rotation axis offset

Set offsets relative to the rotation angles of the rotation axes in parameter No. 19659. The compensation vector calculation formula is the same as that used for rotation axis origin compensation, except that B_p and C_p are changed to rotation axis offsets.

When rotation axis origin compensation and rotation offsetting are set at the same time, both compensations are performed.

When the tool axis is on the Z-axis, and the rotation axes are the B-axis and C-axis, compensation vector calculation is performed as follows :

$$X_p = L_c * \sin(B-(B_z+B_o)) * \cos(C-(C_z+C_o))$$

$$Y_p = L_c * \sin(B-(B_z+B_o)) * \sin(C-(C_z+C_o))$$

$$Z_p = L_c * \cos(B-(B_z+B_o))$$

B_z, C_z : B-axis and C-axis origin compensation values

B_o, C_o : B-axis and C-axis rotation axis offset values

Limitation

- Automatic reference position return command (G28, G29, G30)

Never specify an automatic reference position return command (G28, G29, or G30) in tool axis direction tool length compensation mode.

- Machine coordinate system positioning (G53)

Machine coordinate system positioning (G53) cannot be performed in tool length compensation mode in tool axis direction.

6.15.1 Control Point Compensation of Tool Length Compensation Along Tool Axis

Normally, the control point of tool length compensation along the tool axis is the point of intersection of the centers of two rotation axes. The machine coordinates also indicate this control point.

This section explains the compensation performed when the centers of the two rotation axes do not intersect and also explains how to place the control point at a convenient position on the machine.

Explanation

- Compensation of the rotation centers of two rotation axes

Compensation when the rotation centers of two rotation axes do not match is performed.

The length from the tool mounting position to the first rotation axis center is set as the tool holder offset value in parameter No. 19666.

The vector from the first rotation axis center to the second rotation axis center is set as the rotation center compensation vector in parameter No. 19661. Since parameter No. 19661 is an axis type parameter, the compensation amount for three axes (X, Y, and Z) can be set in this parameter.

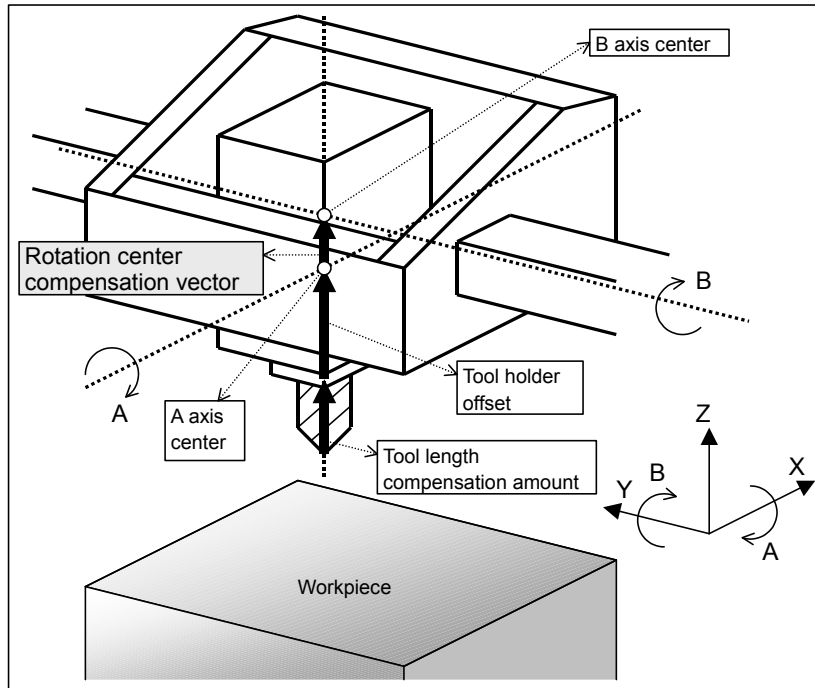


Fig. 6.15.1 (a) Compensation of Rotation Centers of Two Rotation Axes

According to the machine type, set the values listed in the Table 6.15 (a):

Table 6.15 (a) Setting the Tool Holder Offset and Rotation Center Compensation Vector

Machine type	Tool holder offset Parameter No. 19666	Rotation center compensation vector Parameter No. 19661
(1) A- and C-axes. Tool axis is Z-axis.	Length from tool mounting position to A-axis center	Vector from A-axis center to C-axis center
(2) B- and C-axes. Tool axis is Z-axis.	Length from tool mounting position to B-axis center	Vector from B-axis center to C-axis center
(3) A- and B-axes. Tool axis is X-axis.	Length from tool mounting position to B-axis center	Vector from B-axis center to A-axis center
(4) A- and B-axes. Tool axis is Z-axis. B-axis is master.	Length from tool mounting position to A-axis center	Vector from A-axis center to B-axis center
(5) A- and B-axes. Tool axis is Z-axis. A-axis is master	Length from tool mounting position to B-axis center	Vector from B-axis center to A-axis center

NOTE

When using the spindle center compensation described below, set the length from the tool mounting position to the spindle center as the tool holder offset.

- Spindle center compensation

Compensation of the spindle center is performed.

The amount of spindle center compensation is set in parameter No. 19662. Since parameter No. 19662 is an axis type parameter, the compensation amount for three axes (X, Y, and Z) can be set in this parameter.

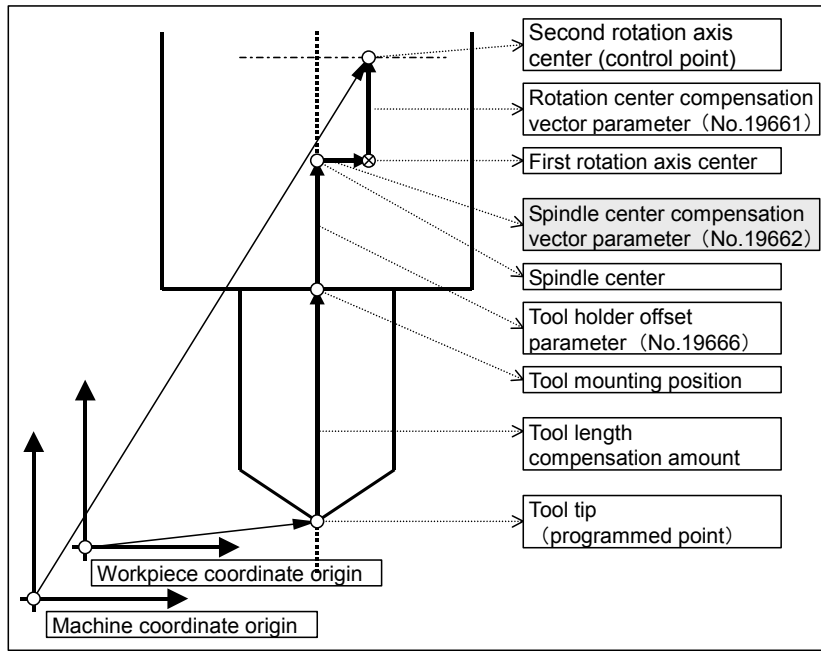


Fig. 6.15.1 (b) Spindle Center Compensation

- Shifting the control point

Conventionally, the center of a rotation axis was used as the control point. The control point can now be shifted as shown in the figure below.

Then, when the rotation axis is at the 0-degree position also in tool length compensation along the tool axis (G43.1), the control point can be set to the same position as that for ordinary tool length compensation (G43).

The control point here is indicated with machine coordinates.

When linear interpolation is specified, for example, this control point moves linearly.

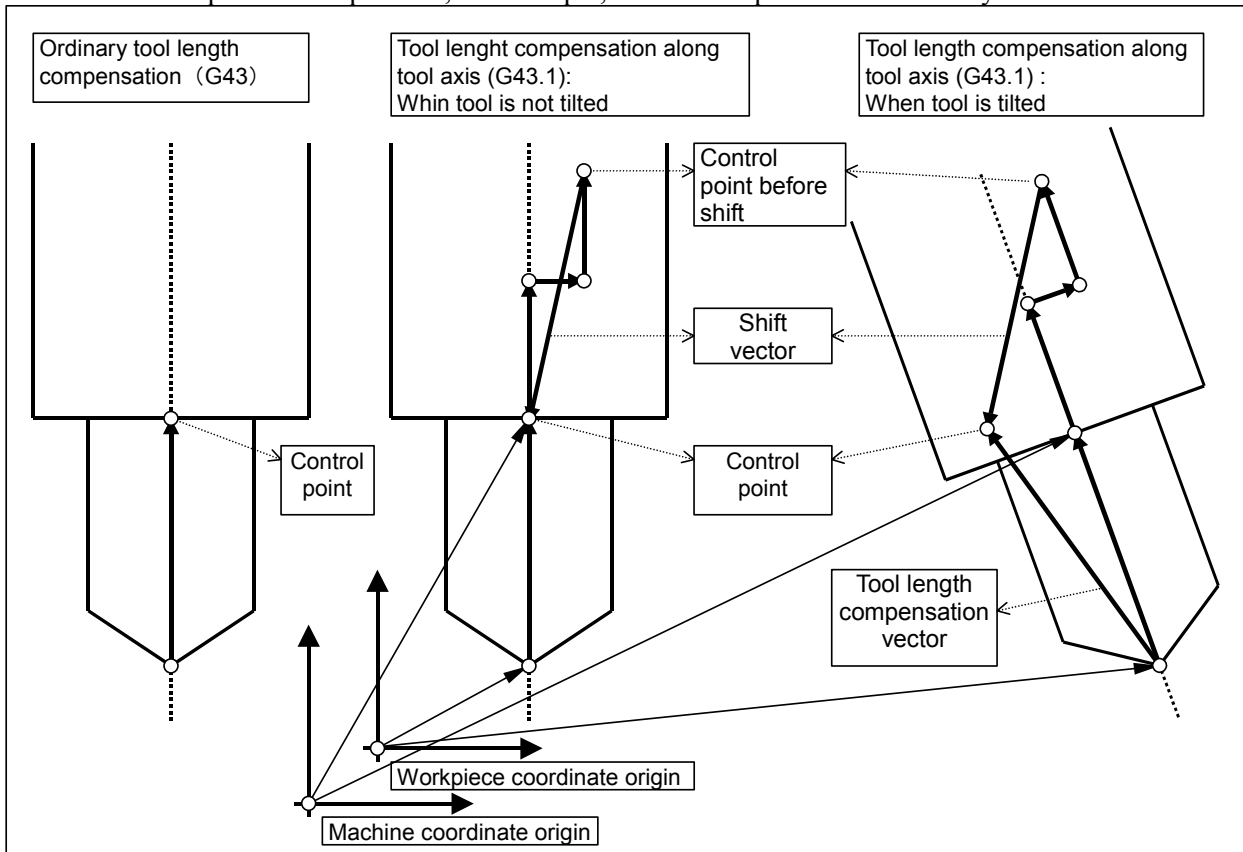


Fig. 6.15.1 (c) Shift of the Control Point

The method of shifting the control point can be selected using the following parameters:

Table 6.15 (b) Methods of Shifting the Control Point

Bit 5 (SVC) of parameter No. 19665	Bit 4 (SPR) of parameter No. 19665	Shift of controlled point
0	-	Shift is not performed and not done conventionally.
1	0	The controlled point is shifted according to the result of the following automatic calculation: -(rotation center compensation vector (parameter No. 19661) + spindle center compensation vector (parameter No. 19662) + tool holder offset (parameter No. 19666))
1	1	The controlled point is shifted. As the shift vector, the vector set in parameter No. 19667 is used.

- Equation for each machine type

- V_x, V_y, V_z : Tool length compensation vector
 A, B, C : Absolute coordinates of A-, B-, and C-axes
 To : Tool offset value
 Ho : Tool holder offset value
 J_x, J_y, J_z : Rotation center compensation vector
 C_x, C_y, C_z : Spindle center compensation vector
 S_x, S_y, S_z : Shift vector

Suppose the above. Then, the tool length compensation vector for each axis is calculated depending on the machine type, as follows:

- (1) A-axis and C-axis. The tool axis is the Z-axis.

$$\begin{bmatrix} V_x \\ V_y \\ V_z \end{bmatrix} = \begin{bmatrix} \cos C & -\sin C & 0 \\ \sin C & \cos C & 0 \\ 0 & 0 & 1 \end{bmatrix} \left(\begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \begin{bmatrix} C_x \\ C_y \\ To + Ho + C_z \end{bmatrix} + \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} \right) + \begin{bmatrix} S_x \\ S_y \\ S_z \end{bmatrix}$$

- (2) B-axis and C-axis. The tool axis is the Z-axis.

$$\begin{bmatrix} V_x \\ V_y \\ V_z \end{bmatrix} = \begin{bmatrix} \cos C & -\sin C & 0 \\ \sin C & \cos C & 0 \\ 0 & 0 & 1 \end{bmatrix} \left(\begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \begin{bmatrix} C_x \\ C_y \\ To + Ho + C_z \end{bmatrix} + \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} \right) + \begin{bmatrix} S_x \\ S_y \\ S_z \end{bmatrix}$$

- (3) A-axis and B-axis. The tool axis is the X-axis.

$$\begin{bmatrix} V_x \\ V_y \\ V_z \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \left(\begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \begin{bmatrix} To + Ho + C_x \\ C_y \\ C_z \end{bmatrix} + \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} \right) + \begin{bmatrix} S_x \\ S_y \\ S_z \end{bmatrix}$$

- (4) A-axis and B-axis. The tool axis is the Z-axis, and the B-axis is the master.

$$\begin{bmatrix} V_x \\ V_y \\ V_z \end{bmatrix} = \begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \left(\begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \begin{bmatrix} C_x \\ C_y \\ To + Ho + C_z \end{bmatrix} + \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} \right) + \begin{bmatrix} S_x \\ S_y \\ S_z \end{bmatrix}$$

- (5) A-axis and B-axis. The tool axis is the Z-axis and the A-axis is the master.

$$\begin{bmatrix} V_x \\ V_y \\ V_z \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \left(\begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \begin{bmatrix} C_x \\ C_y \\ To + Ho + C_z \end{bmatrix} + \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} \right) + \begin{bmatrix} S_x \\ S_y \\ S_z \end{bmatrix}$$

The shift vector (S_x, S_y, S_z) is calculated as follows:

- (A) When bit 5 (SVC) of parameter No. 19665 = 0, the vector is set to 0.
 (B) When bit 5 (SVC) of parameter No. 19665 = 1, and bit 4 (SBP) of parameter No. 19665 = 0:
 When the machine type is other than (3)

$$\begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix} = - \begin{bmatrix} Cx + Jx \\ Cy + Jy \\ Cz + Jz + Ho \end{bmatrix}$$

When the machine type is (3)

$$\begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix} = - \begin{bmatrix} Cx + Jx + Ho \\ Cy + Jy \\ Cz + Jz \end{bmatrix}$$

(C) When bit 5 (SVC) of parameter No. 19665 = 1, and bit 4 (SBP) of parameter No. 19665 = 1, the vector specified in parameter No. 19667 is set.

6.16 SPINDLE UNIT COMPENSATION, NUTATING ROTARY HEAD TOOL LENGTH COMPENSATION

Overview

For a machine with multiple spindle units, characteristic parameters, compensation amounts, tool axis directions, etc. can be set for each unit. By compensating a selected unit during machining, it is possible to create programs not influenced by spindle unit compensation amounts/tool length compensation amounts.

If a spindle unit is controlled with a rotation axis, the compensation amount is automatically calculated from the rotation angle of the rotation axis to perform compensation.

Application example

As shown in Fig. 6.16 (a), let us consider a spindle unit controlled with the C-axis. When the rotation axis (C-axis) is rotated, the spindle unit compensation vector/nutating rotary head tool length compensation vector are calculated from that angle to perform compensation.

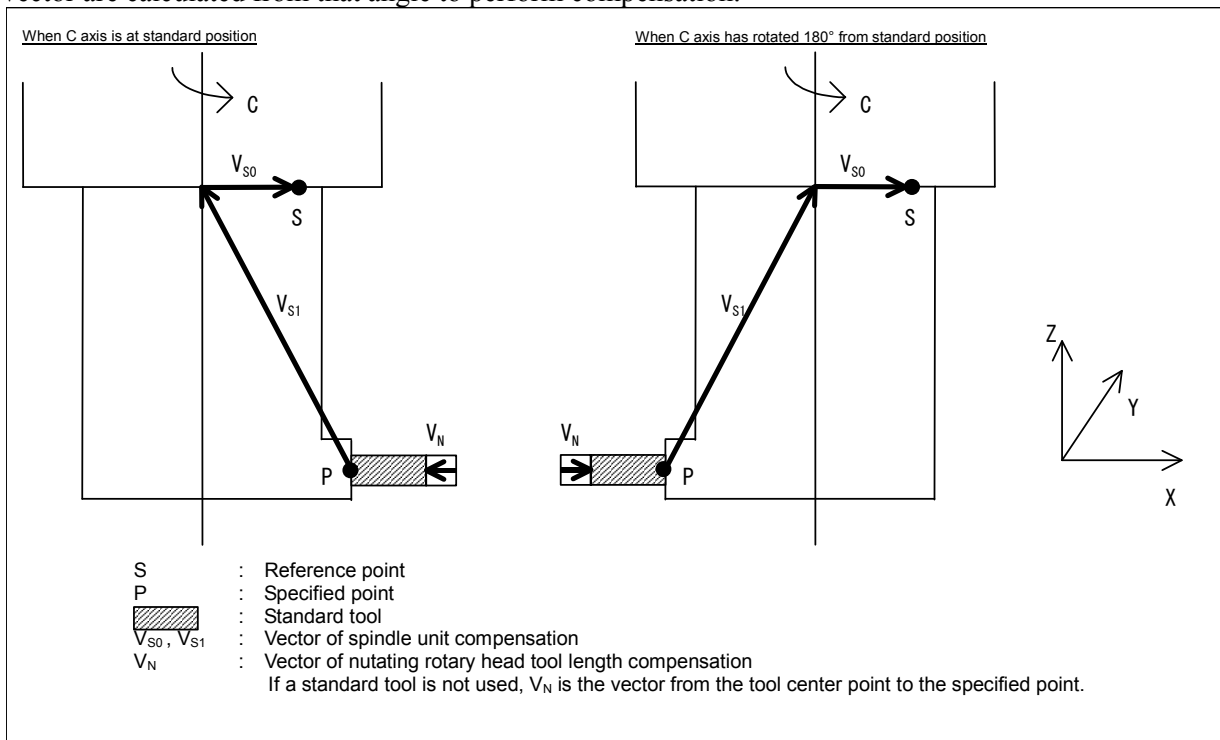


Fig. 6.16 (a) Example of vector of spindle unit compensation and nutating rotary head tool length compensation

Format**- Spindle unit compensation****G44.9 ; Enables spindle unit compensation****G49.9 ; Disables spindle unit compensation**

The G44.9 command calculates the spindle unit compensation amount from the preset data and the angle of the rotation axis, and enables compensation.

The G49.9 command cancels spindle unit compensation.

If, in G44.9 mode, the rotation axis related to spindle unit compensation is rotated, the vector is recalculated by using the end point value of that rotation axis.

- Nutating rotary head tool length compensation**G43.3 Hn ; Enables nutating rotary head tool length compensation****G49 ; Disables nutating rotary head tool length compensation**

n : Tool compensation amount number

The G43.3 command calculates the tool length compensation amount from the tool compensation amount data specified for n and the angle of the rotation axis, and enables compensation.

The G49 command cancels tool length compensation.

If, in G43.3 mode, the rotation axis related to tool length compensation is rotated, the tool length compensation vector is recalculated by using the end point value of that rotation axis.

- Mode retention function

When the power is turned on or the system is reset in spindle unit compensation (G44.9) and nutating rotary head tool length compensation (G43.3), G44.9 and G43.3 modes, as well as the tool compensation amount and the vector, are retained.

(1) Retention of G44.9 and G43.3 modes when the power is turned on or the system is reset

By making either of the following parameter settings, G44.9 mode is retained when the power is turned on or the system is reset.

- Bit 6 (CLR) of parameter No. 3402 is set to 0.
- Bit 6 (CLR) of parameter No. 3402 is set to 1 and bit 3 (C27) of parameter No. 3409 is set to 1.

By making either of the following parameter settings, G43.3 mode and H code are retained when the power is turned on or the system is reset.

- Bit 6 (CLR) of parameter No. 3402 is set to 0.
- Bit 6 (CLR) of parameter No. 3402 is set to 1, bit 0 (C08) of parameter No. 3407 is set to 1, and bit 7 (CFH) of parameter No. 3409 is set to 1.

(2) Calculation of G44.9 and G43.3 vectors when the power is turned on

By setting bit 2 (SCV) of parameter No. 25860 to 1, the spindle unit compensation vector is calculated when the power is turned on.

By setting bit 3 (NCV) of parameter No. 25860 to 1, the nutating rotary head tool length compensation vector is calculated when the power is turned on.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR						

[Input type] Parameter input

[Data type] Bit path

#6 CLR Reset button on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal

0: Cause reset state.


1: Cause clear state.

For the reset and clear states, refer to Appendix in the OPERATOR'S MANUAL.

	#7	#6	#5	#4	#3	#2	#1	#0
3406	C07	C06	C05	C04	C03	C02	C01	
	#7	#6	#5	#4	#3	#2	#1	#0
3407	C15	C14	C13	C12	C11	C10	C09	C08
	#7	#6	#5	#4	#3	#2	#1	#0
3408	C23	C22	C21	C20	C19	C18	C17	C16
	#7	#6	#5	#4	#3	#2	#1	#0
3409	CFH	C30	C29	C28	C27	C26	C25	C24

[Input type] Parameter input

[Data type] Bit

C01 to C30 If bit 6 (CLR) of parameter No.3402 is set to 1, set a group of G codes to be placed in the cleared state when the CNC is reset by the  key of the MDI panel, the external reset signal, the reset & rewind signal, or the emergency stop signal.

The Table 6.16 (a) indicates the correspondence between bits and G code groups


The setting of a bit has the following meaning:

0: Places the G code group in the cleared state.

1: Does not place G code group in the cleared state.

Table 6.16 (a)

Parameter	G code group
C01	01
C02	02
C03	03
:	:
C30	30

#7 CFH When bit 6 (CLR) of parameter No.3402 is 1, the  key on the MDI panel, the external reset signal, the reset and rewind signal, or emergency stop will,

0: Clear F codes, H codes (for the M series), D codes (for the M series), and T codes (for the T series).

1: Not clear F codes, H codes (for the M series), D codes (for the M series), and T codes (for the T series).

	#7	#6	#5	#4	#3	#2	#1	#0
25860					NCV	SCV		

[Input type] Parameter input

[Data type] Bit path

#2 SCV At power-on, a spindle unit compensation vector is:

0: Not calculated.

1: Calculated.

NOTE

This parameter is effective in the case of either of the following settings:

- Bit 6 (CLR) of parameter No. 3402 = 0
- Bit 6 (CLR) of parameter No. 3402 = 1 and bit 3 (C27) of parameter No. 3409 = 1

- #3 **NCV** At power-on, an nutating rotary head tool length compensation vector is:
0: Not calculated.
1: Calculated.

NOTE

This parameter is effective in the case of either of the following settings:

- Bit 6 (CLR) of parameter No. 3402 = 0
- Bit 6 (CLR) of parameter No. 3402 = 1, bit 0 (C08) of parameter No. 3407 = 1, and bit 7 (CFH) of parameter No. 3409 = 1

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 30*i* and Series 15. The Series 15 program formats can be processed for memory operation.

Other data formats must comply with the Series 30*i*. When a value out of the specified range for the Series 30*i* is registered, an alarm occurs. Functions not available in the Series 30*i* cannot be registered or used for memory operation.

- 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 30	M98 P○○○ □□□□ ; <div style="display: flex; justify-content: space-around; margin-top: 5px;"> Repetition count Subprogram number (1 to 9999) </div>

If, however, the custom macro option is enabled, both formats can be used.

- Address for the canned cycle repetition count for drilling

The Series 15 and this CNC 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 30	K

7.1 MULTIPLE REPETITIVE CYCLE

The multiple repetitive cycle is canned cycles to make CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the threading is available.

NOTE

- 1 Explanatory figures in this section use the ZX plane as the selected plane, diameter programming for the X-axis, and radius programming for the Z-axis. When radius programming is used for the X-axis, change U/2 to U and X/2 to X.
- 2 A multiple repetitive cycle can be performed on any plane (including parallel axes for plane definition).

7.1.1 Stock Removal in Turning (G71.7)

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

To use type II, the "multiple repetitive canned cycle 2" option function is required.

Format

ZpXp plane

G71.7 P(ns) Q(nf) U(Δu) W(Δw) I(Δi) K(Δk) D(Δd) F(f) S(s) T(t);

N (ns) ;

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

YpZp plane

G71.7 P(ns) Q(nf) V(Δw) W(Δu) J(Δk) K(Δi) D(Δd) F(f) S(s) T(t);

N (ns) ;

N (nf) ;

XpYp plane

G71.7 P(ns) Q(nf) U(Δw) V(Δu) I(Δk) J(Δi) D(Δd) F(f) S(s) T(t);

N (ns) ;

N (nf) ;

Δd : Depth of cut

The cutting direction depends on the direction AA'.

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

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

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

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

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

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

f,s,t : Any F, S, or T function contained in blocks ns to nf in the cycle is ignored, and the F,
S, or T function in this G71.7 block is effective.

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

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

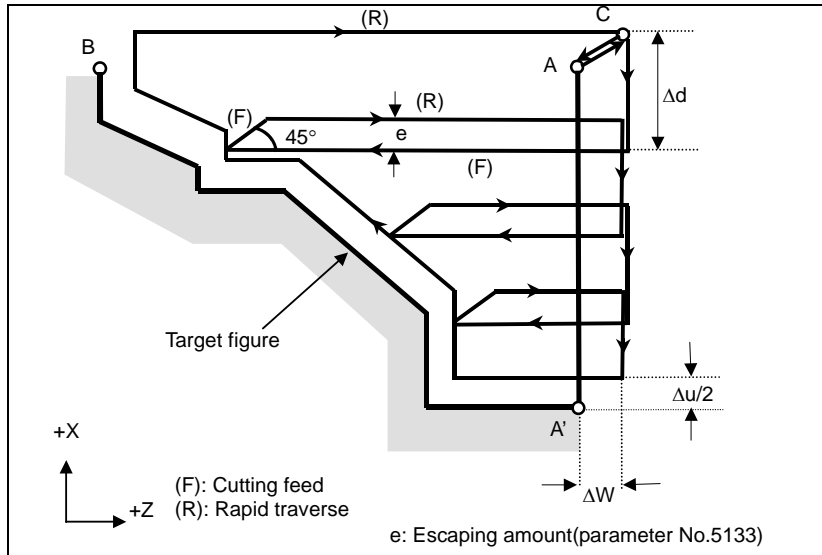


Fig. 7.1.1 (a) Cutting path (type I) of the stock removal in turning without the finishing allowance for rough cutting

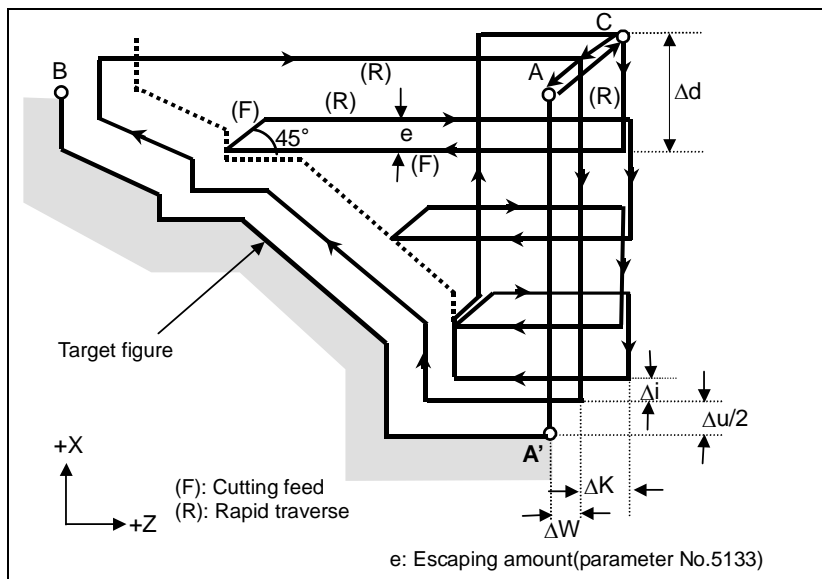


Fig. 7.1.1 (b) Cutting path (type I) of the stock removal in turning with the finishing allowance for rough cutting

Explanation

- Operations

If the target figure from A to A' to B is provided by a program, the tool grinds by the depth of cut Δd . The path differs as described below, depending on whether the finishing allowance for rough cutting is specified or not.

(1) If the finishing allowance for rough cutting is not specified

The tool grinds by the depth of cut Δd , leaving the finishing allowances of $\Delta u/2$ and Δw , and after the last cut, performs rough cutting as finishing in accordance with the target figure program.

- (2) If the finishing allowance for rough cutting is specified
The tool grinds by the depth of cut Δd , leaving the finishing allowances of $\Delta u/2+\Delta i$ and $\Delta w+\Delta k$, and after the last cut, returns to the start point (A) and then performs rough cutting as finishing along the target figure, to remove the allowances of Δi and Δk .
After rough cutting as finishing, the block next to the sequence block specified at Q is executed.

NOTE

- 1 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71.7 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
- 2 When an option of constant surface speed control is selected, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71.7 block or the previous block is effective.

- Escaping amount (e)

Set an escaping amount (e) in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5133	Depends on the increment system for the reference axis.	Radius programming	Not required

- Target figure Patterns

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

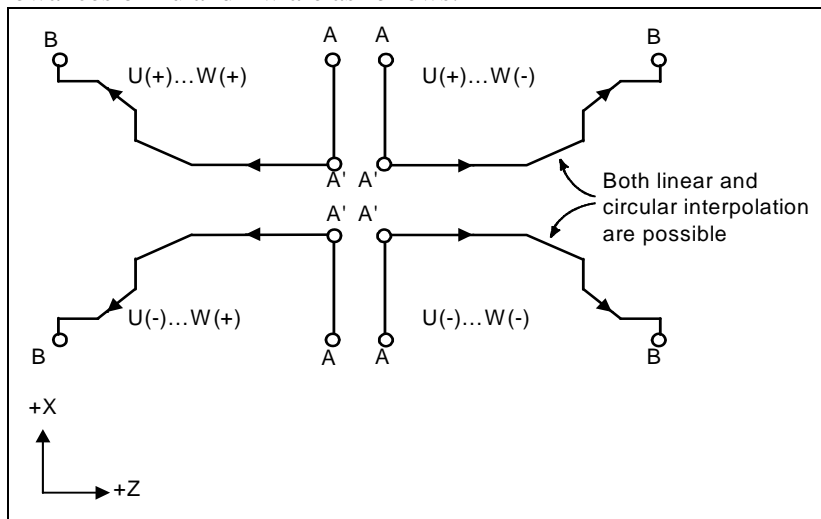


Fig. 7.1.1 (c) Four target figure patterns

Limitation

- (1) For U(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For U(-), a figure for which a position lower than the cycle start point is specified cannot be machined.
- (2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.
- (3) For type II, the figure must show monotone increase or decrease along the first axis on the plane.

- Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065, "G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM" is issued.

When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'. In this start block, also select type I or II.

- Check functions

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

NOTE

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

The following checks can also be made.

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

- Types I and II

Selection of type I or II

For G71.7, there are types I and II.

When the target figure has pockets, be sure to use type II.

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

NOTE

To use type II, the multiple repetitive canned cycle II option is required.

Selecting type I or II

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

(1) When type I is selected

Specify the second axis on the plane (X-axis for the ZX plane). Do not specify the first axis on the plane (Z-axis for the ZX plane).

(2) When type II is selected

Specify the second axis on the plane (X-axis for the ZX plane) and first axis on the plane (Z-axis for the ZX plane).

To use type II without movement along the first axis on the plane (Z-axis for the ZX plane) in the first block, specify the first axis with a travel distance of 0.

- Type I

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

```

Example
ZX plane
G71.7 P100 Q200....;
N100 X_ ;(Specifies only the second axis on the plane.)
    :;
    :;
    :;
N200.....;

```

(2) The figure along path A'-B must show monotone increase or decrease in the directions of both axes forming the plane (Z- and X-axes for the ZX plane). It must not have any pocket as shown in the Fig. 7.1.1 (d).

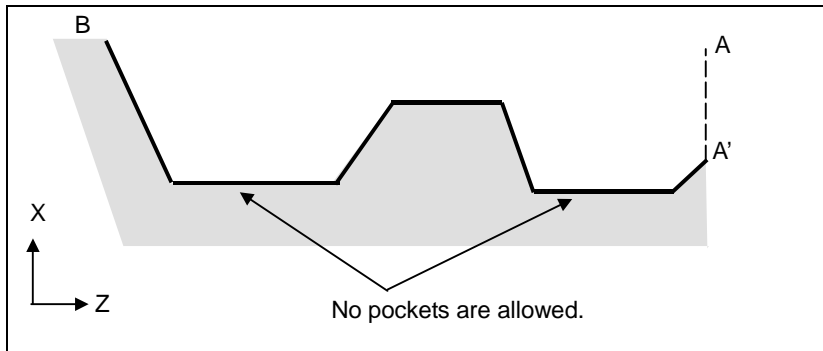


Fig. 7.1.1 (d) Figure which does not show monotone increase or decrease (type I)

⚠ CAUTION
If a figure does not show monotone change along the first or second axis on the plane, alarm PS0064, "THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)" or PS0329, "THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)" is issued. If the movement does not show monotone change, but is very small, and it can be determined that the movement is not dangerous, however, the permissible amount can be specified in parameters Nos. 5145 and 5146 to specify that the alarm is not issued in this case.

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

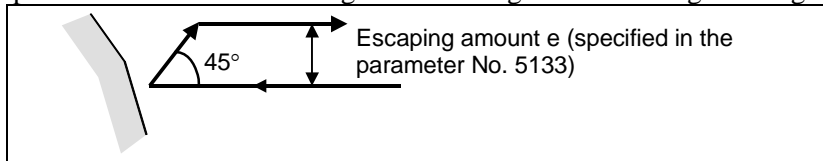


Fig. 7.1.1 (e) Cutting in the direction of 45 degrees (type I)

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

- Type II

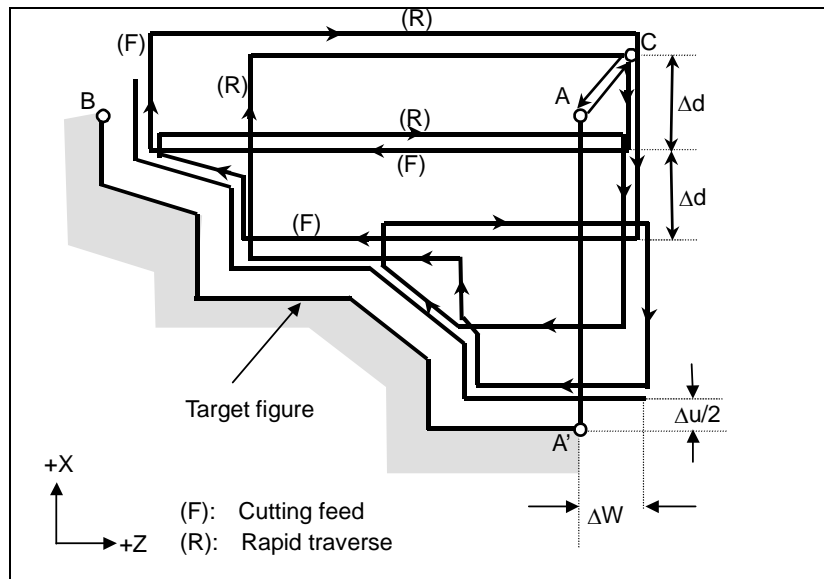


Fig. 7.1.1 (f) Cutting path in stock removal in turning (type II)

When the target figure program from A to A' to B is specified, the tool grinds by the depth of cut Δd . The difference from type I is that the tool cuts up along the figure after rough cutting in the first axis on the plane (Z-axis for the ZX plane).

The path differs as described below, depending on whether the finishing allowance for rough cutting is specified or not.

- (1) If the finishing allowance for rough cutting is not specified

The tool grinds by the depth of cut Δd , leaving the finishing allowances of $\Delta u/2$ and Δw , and after grinding with the last cut (and after grinding the area of Pn to Pm in the Fig. 7.1.1 (f) because the area is parallel to the Z-axis and is, therefore, regarded as a single pocket), returns to the start point (A) and performs rough cutting as finishing in accordance with the target figure program, leaving the allowances of $\Delta u/2$ and Δw .

- (2) If the finishing allowance for rough cutting is specified

The tool grinds by the depth of cut Δd , leaving the finishing allowances of $\Delta u/2 + \Delta i$ and $\Delta w + \Delta k$, and after the last cut, returns to the start point (A) and then performs rough cutting as finishing in accordance with the target figure program, to remove the allowances of Δi and Δk .

After rough cutting as finishing, the block next to the sequence block specified at Q is executed.

Type II differs from type I in the following points:

- (1) In the block with sequence number ns, the two axes forming the plane (X-axis and Z-axis for the ZX plane) must be specified. To use type II without movement along the Z-axis on the ZX plane in the first block, specify the Z axis with a travel distance of 0.

Example

```
ZX plane
G71.7 P100 Q200.....;
N100 X_ Z_ ; (Specifies the two axes forming the plane.)
    ;;
    ;;
N200.....;
```

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

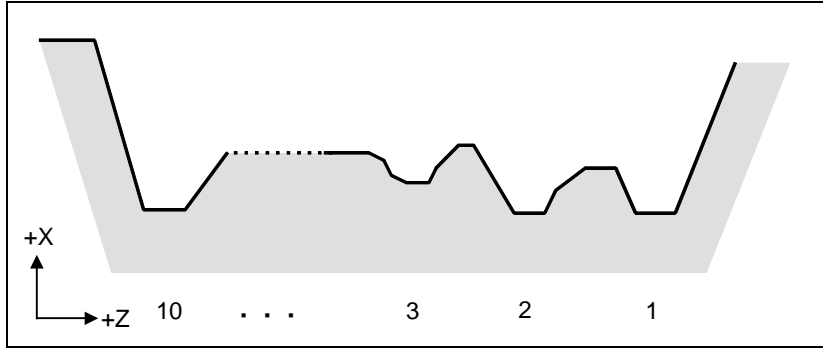


Fig. 7.1.1 (g) Figure having pockets (type II)

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

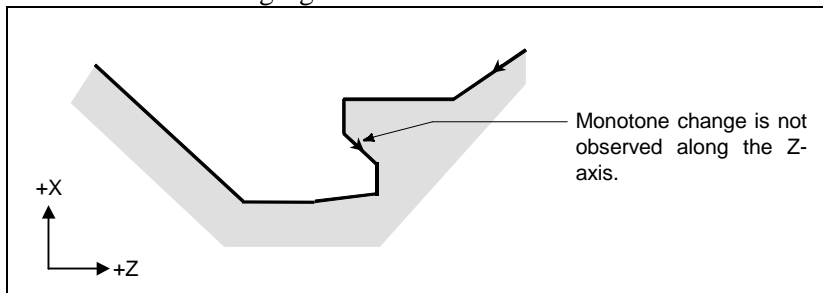


Fig. 7.1.1 (h) Figure which cannot be machined (type II)

⚠ CAUTION

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

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

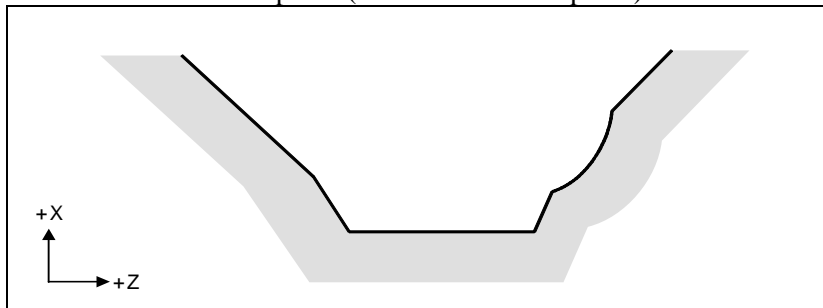


Fig. 7.1.1 (i) Figure which can be machined (type II)

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

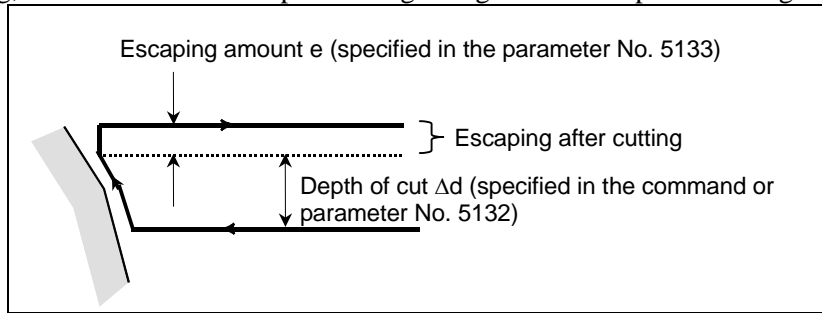


Fig. 7.1.1 (j) Cutting along the workpiece figure (type II)

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

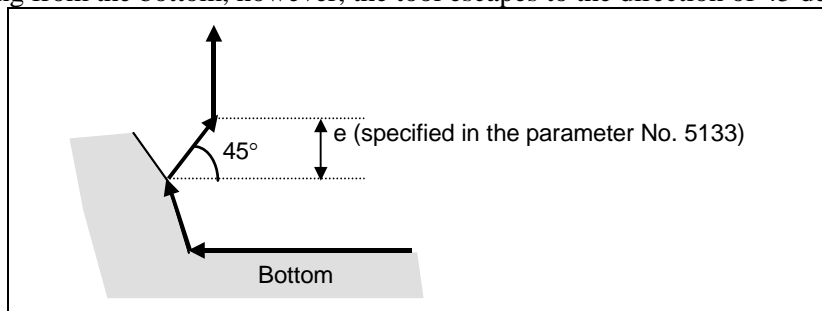


Fig. 7.1.1 (k) Escaping from the bottom to the direction of 45 degrees

- (4) When a position parallel to the first axis on the plane (Z-axis for the ZX plane) is specified in a block in the program for the target figure, it is assumed to be at the bottom of a pocket.
- (5) After all rough cutting terminates along the first axis on the plane (Z-axis for the ZX plane), the tool temporarily returns to the cycle start point. At this time, when there is a position whose height equals to that at the start point, the tool passes through the point in the position obtained by adding depth of cut Δd to the position of the figure and returns to the start point. Then, rough cutting is performed as finishing along the target figure. At this time, the tool passes through the point in the obtained position (to which depth of cut Δd is added) when returning to the start point.

Bit 2 (RF2) of parameter No. 5105 can be set to 1 so that rough cutting as finishing is not performed.

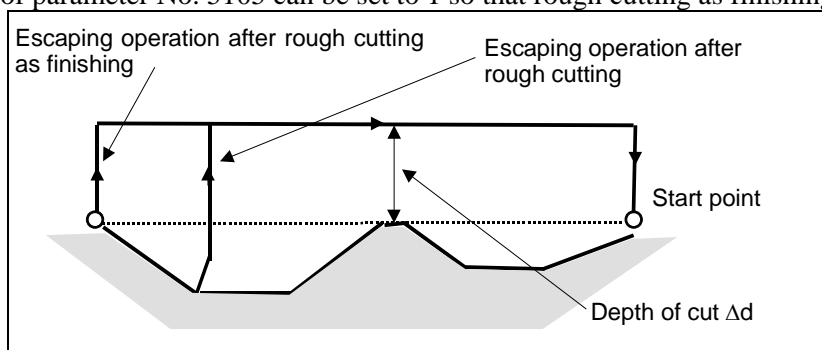


Fig. 7.1.1 (l) Escaping operation when the tool returns to the start point (type II)

(6) Order and path for rough cutting of pockets

Rough cutting is performed in the following order.

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

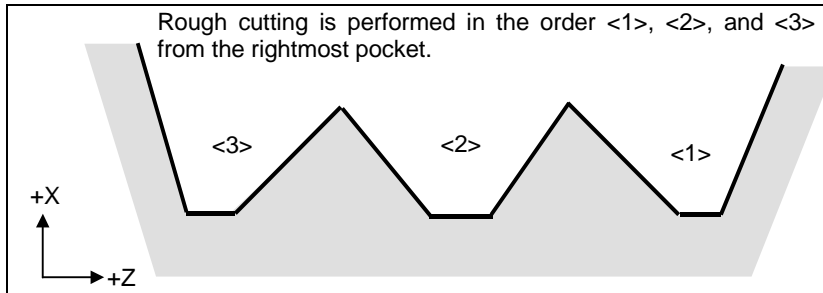


Fig. 7.1.1 (m) Rough cutting order in the case of monotone decrease (type II)

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

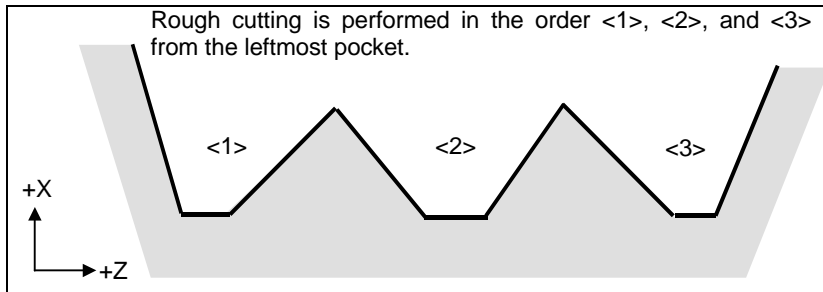


Fig. 7.1.1 (n) Rough cutting order in the case of monotone increase (type II)

The path in rough cutting is as shown Fig. 7.1.1 (o).

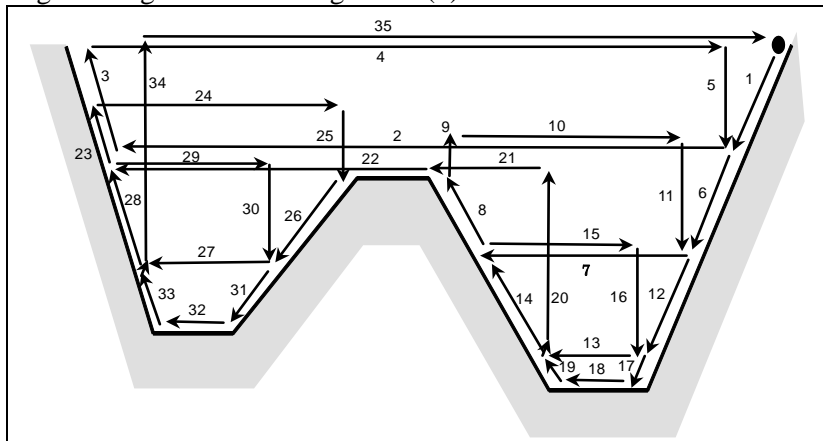


Fig. 7.1.1 (o) Cutting path for multiple pockets (type II)

The Fig. 7.1.1 (p) shows how the tool moves after rough cutting for a pocket in detail.

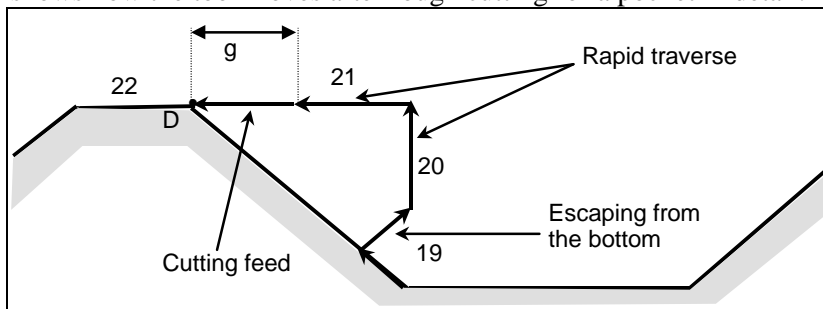


Fig. 7.1.1 (p)

Cuts the workpiece at the cutting feedrate and escapes to the direction of 45 degrees. (Operation 19)

Then, moves to the height of point D in rapid traverse. (Operation 20)

Then, moves to the position the amount of g before point D. (Operation 21)

Finally, moves to point D in cutting feed.

The clearance g to the cutting feed start position is set in parameter No. 5134.

For the last pocket, after cutting the bottom, the tool escapes to the direction of 45 degrees and returns to the start point in rapid traverse. (Operations 34 and 35)

⚠ CAUTION

- 1 This CNC differs from the FANUC Series 16i/18i/21i in cutting of a pocket. The tool first cuts the nearest pocket to the start point. After cutting of the pocket terminates, the tool moves to the nearest but one pocket and starts cutting.
- 2 When the figure has a pocket, generally specify a value of 0 for Δw (finishing allowance). Otherwise, the tool may dig into the wall on one side.
- 3 This CNC differs from the FANUC Series 16i/18i/21i in the path of cutting after turning depending on the figure of the workpiece. When the tool becomes moving only along the first axis on the plane (Z-axis for the ZX plane) according to the figure of the workpiece during cutting, it starts retraction along the second axis on the plane (X-axis for the ZX plane).

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70.7, G71.7, G72.7, G73.7) and specify the cancel command (G40) outside the programs (from the block specified with P to the block specified with Q) specifying a target finishing figure. If tool nose radius compensation is specified in the program specifying a target finishing figure, alarm PS0325, "UNAVAILABLE COMMAND IS IN SHAPE PROGRAM", is issued.

Program example

G42;.....Specify this command before a multiple repetitive canned cycle command.

G71.7P10Q20D10000;

N10G00X0;

:

N20X50.;

G40;.....Specify this command after the program specifying a target finishing figure.

When this cycle is specified in the tool nose radius compensation mode, offset is temporarily canceled during movement to the start point. Start-up is performed in the first block. Offset is temporarily canceled again at the return to the cycle start point after termination of cycle operation. Start-up is performed again according to the next move command. This operation is shown in the Fig. 7.1.1 (q).

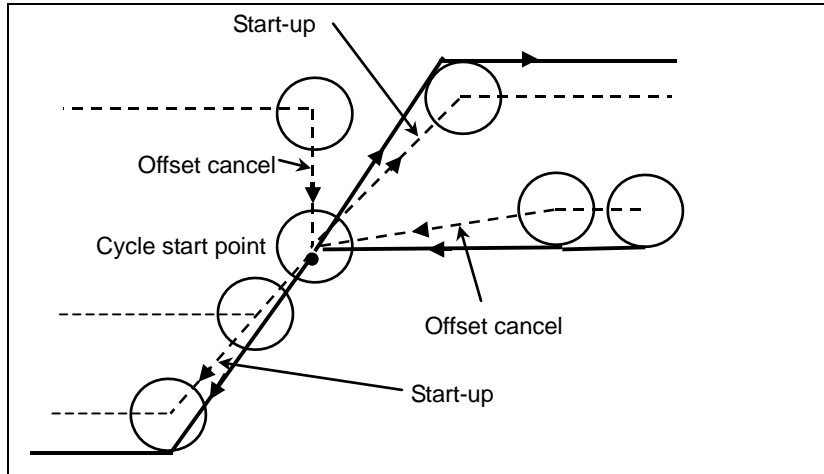


Fig. 7.1.1 (q)

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

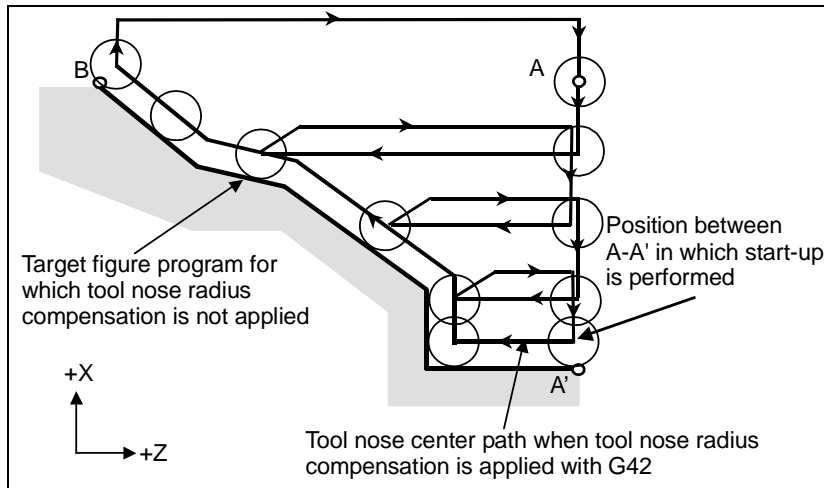


Fig. 7.1.1 (r) Path when tool nose radius compensation is applied

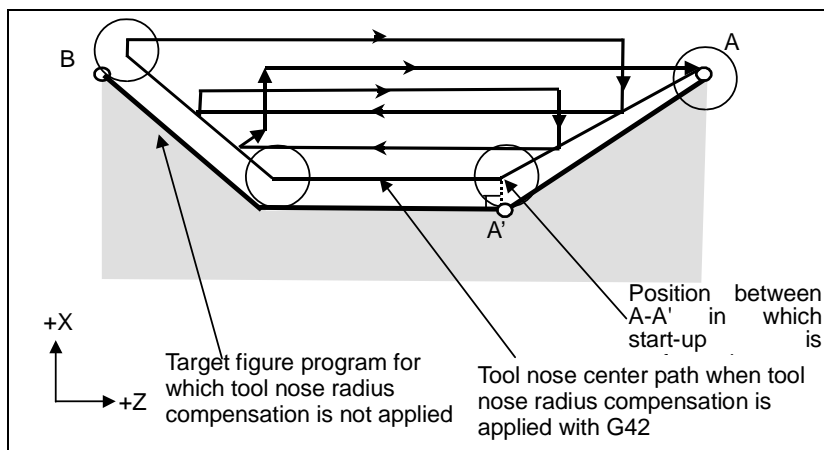


Fig. 7.1.1 (s)

NOTE

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

- Reducing the cycle time

In G71.7 and G72.7, the tool can be moved to the previous turning start point (operation 1) in rapid traverse by setting bit 0 (ASU) of parameter No. 5107 to 1.

Bit 0 (ASU) of parameter No. 5107 is valid for both type I and II commands.

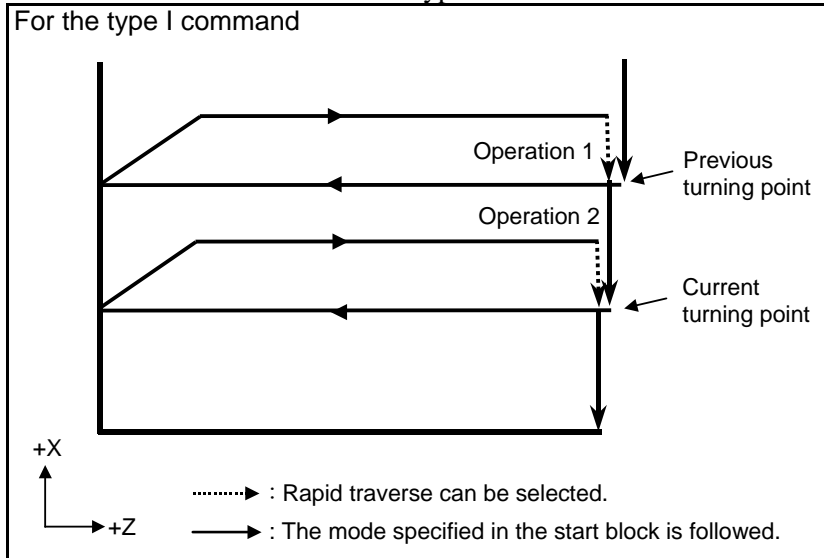


Fig. 7.1.1 (t)

For the type I G71.7 and G72.7 commands, operations 1 and 2 to the current turning start point that are usually performed in 2 cycles can be performed in 1 cycle by setting bit 1 (ASC) of parameter No. 5107 to 1. The feed mode specified in the start block in the program for a target figure (G00 or G01) is used.

Bit 1 (ASC) of parameter No. 5107 is valid only for the type I command.

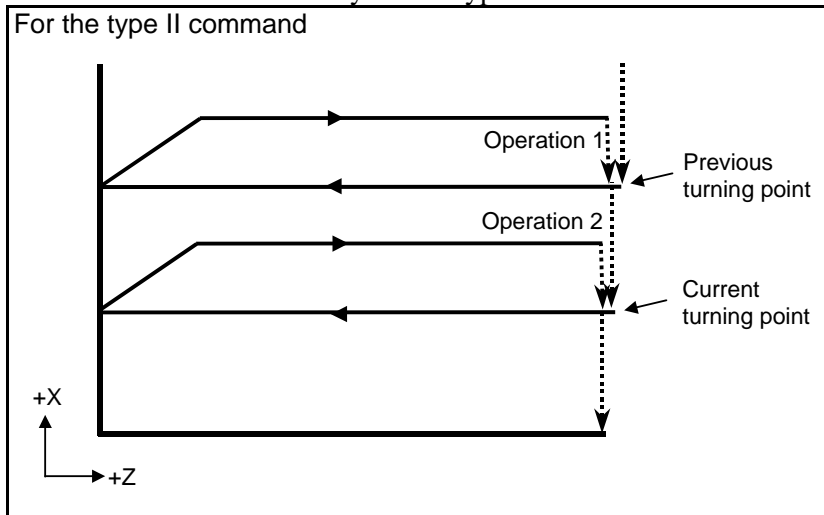


Fig. 7.1.1 (u)

7.1.2 Stock Removal in Facing (G72.7)

This cycle is the same as G71.7 except that cutting is performed by an operation parallel to the second axis on the plane (X-axis for the ZX plane).

Format

ZpXp plane

G72.7 P(ns) Q(nf) U(Δ u) W(Δ w) I(Δ i) K(Δ k) D(Δ d) F(f) S(s) T(t) ;

N (ns) ;

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

YpZp plane

G72.7 P(ns) Q(nf) V(Δ w) W(Δ u) J(Δ k) K(Δ i) D(Δ d) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

XpYp plane

G72.7 P(ns) Q(nf) U(Δ w) V(Δ u) I(Δ k) J(Δ i) D(Δ d) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

Δ d : Depth of cut

The cutting direction depends on the direction AA'.

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

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

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

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

Δ i : Distance of the finishing allowance for rough cutting in the direction of the second
axis on the plane (X-axis for the ZX plane)

Δ k : Distance of the finishing allowance for rough cutting in the direction of the first axis
on the plane (Z-axis for the ZX plane)

f,s,t : Any F, S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G72.7 block is effective.

	Unit	Diameter/radius programming	Sign	Decimal point input
Δ d	Depends on the increment system for the reference axis.	Radius programming	Not required	Not allowed
Δ u	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δ w	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed
Δ i	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed
Δ k	Depends on the increment system for the reference axis.	Radius programming	Not required	Allowed

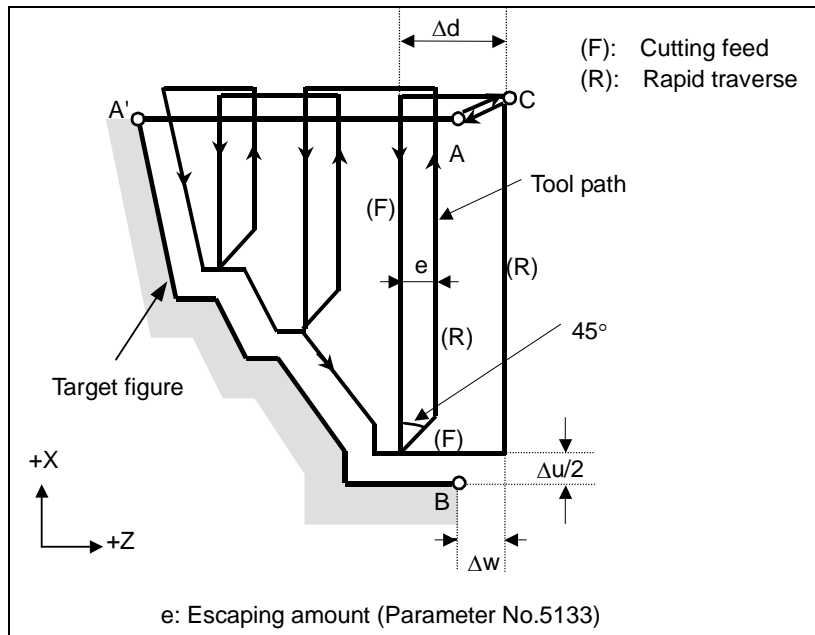


Fig. 7.1.2 (a) Cutting path (type I) of the stock removal in facing without the finishing allowance for rough cutting

Explanation

- Operations

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

NOTE

- 1 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G72.7 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
- 2 When an option of constant surface speed control is selected, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G72.7 block or the previous block is effective.

- Escaping amount (e)

The escaping amount (e) is set in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5133	Depends on the increment system for the reference axis.	Radius programming	Not required

- Target figure Patterns

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

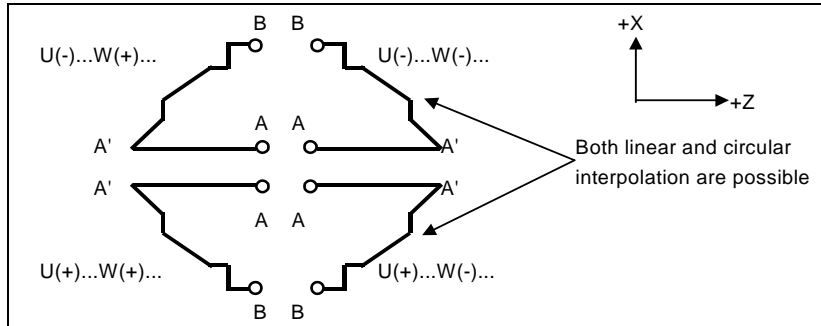


Fig. 7.1.2 (b) Signs of the values specified at U and W in stock removal in facing

Limitation

- (1) For W(+), a figure for which a position higher than the cycle start point is specified cannot be machined.
For W(-), a figure for which a position lower than the cycle start point is specified cannot be machined.
- (2) For type I, the figure must show monotone increase or decrease along the first and second axes on the plane.
- (3) For type II, the figure must show monotone increase or decrease along the second axis on the plane.

- Start block

In the start block in the program for a target figure (block with sequence number ns in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065, "G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM" is issued.

When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.

In this start block, also select type I or II.

- Check functions

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

NOTE

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

The following checks can also be made.

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

- Types I and II

Selection of type I or II

For G71.7 and G72.7, there are types I and II.

When the target figure has pockets, be sure to use type II.

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

Selecting type I or II

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

(1) When type I is selected

Specify the first axis on the plane (Z-axis for the ZX plane).

Do not specify the second axis on the plane (X-axis for the ZX plane).

(2) When type II is selected

Specify the second axis on the plane (X-axis for the ZX plane) and first axis on the plane (Z-axis for the ZX plane).

To use type II without movement along the second axis on the plane (X-axis for the ZX plane), specify the second axis with a travel distance of 0.

- Type I

G72.7 differs from G71.7 in the following points:

- (1) G72.7 cuts the workpiece with moving the tool in parallel with the second axis on the plane (X-axis on the ZX plane).
- (2) In the start block in the program for a target figure (block with sequence number ns), only the first axis on the plane (Z-axis for the ZX plane) must be specified.

- Type II

G72.7 differs from G71.7 in the following points:

- (1) G72.7 cuts the workpiece with moving the tool in parallel with the second axis on the plane (X-axis on the ZX plane).
- (2) The figure need not show monotone increase or decrease in the direction of the first axis on the plane (Z-axis for the ZX plane) and it may have concaves (pockets). The figure must show monotone change in the direction of the second axis on the plane (X-axis for the ZX plane), however.
- (3) When a position parallel to the second axis on the plane (X-axis for the ZX plane) is specified in a block in the program for the target figure, it is assumed to be at the bottom of a pocket.
- (4) After all rough cutting terminates along the second axis on the plane (X-axis for the ZX plane), the tool temporarily returns to the start point. Then, rough cutting as finishing is performed.

- Tool nose radius compensation

See the pages on which G71.7 is explained.

- Reducing the cycle time

See the pages on which G71.7 is explained.

7.1.3 Pattern Repeating (G73.7)

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

Format

ZpXp plane

G73.7 P(ns) Q(nf) U(Δ u) W(Δ w) I(Δ i) K(Δ k) D(d) F(f) S(s) T(t) ;

N (ns) ;

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

YpZp plane

G73.7 P(ns) Q(nf) V(Δ w) W(Δ u) J(Δ k) K(Δ i) D(d) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

XpYp plane

G73.7 P(ns) Q(nf) U(Δ w) V(Δ u) I(Δ k) J(Δ i) D(d) F(f) S(s) T(t) ;

N (ns) ;

N (nf) ;

Δ i : Distance of escape in the direction of the second axis on the plane (X-axis for the ZX plane)

Δ k : Distance of escape in the direction of the first axis on the plane (Z-axis for the ZX plane)

d : The number of division

This value is the same as the repetitive count for rough cutting.

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

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

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

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

f,s,t : Any F, S, and T function contained in the blocks between sequence number "ns" and "nf" are ignored, and the F, S, and T functions in this G73.7 block are effective.

	Unit	Diameter/radius programming	Sign	Decimal point input
Δ i	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
Δ K	Depends on the increment system for the reference axis.	Radius programming	Required	Allowed
Δ u	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required	Allowed
Δ w	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required	Allowed

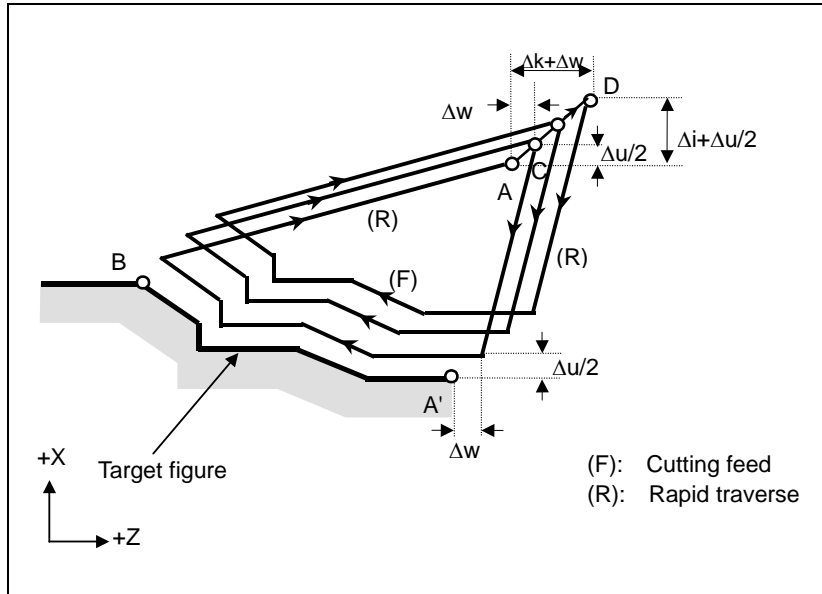


Fig. 7.1.3 (a) Cutting path in pattern repeating

Explanation

- Operations

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

NOTE

- 1 After cycle operation terminates, the tool returns to point A.
- 2 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G73.7 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.

- Target figure Patterns

As in the case of G71.7, there are four target figure patterns. Be careful about signs of Δu , Δw , Δi , and Δk when programming this cycle.

- Start block

In the start block in the program for a target figure (block with sequence number n_s in which the path between A and A' is specified), G00 or G01 must be specified. If it is not specified, alarm PS0065, "G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM" is issued.

When G00 is specified, positioning is performed along A-A'. When G01 is specified, linear interpolation is performed with cutting feed along A-A'.

- Check functions

The following check can be made.

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

- Tool nose radius compensation

Like G71.7, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

7.1.4 Finishing Cycle (G70.7)

After rough cutting by G71.7, G72.7 or G73.7, the following command permits finishing.

Format

G70.7 P(ns) Q(nf) ;

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

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

Explanation

- Operations

The blocks with sequence numbers ns to nf in the program for a target figure are executed for finishing. The F, S, T, M, and second auxiliary functions specified in the G71.7, G72.7, or G73.7 block are ignored and the F, S, T, M, and second auxiliary functions specified in the blocks with sequence numbers ns to nf are effective.

When cycle operation terminates, the tool is returned to the start point in rapid traverse and the next G70.7 cycle block is read.

- Target figure

Check function

Following check can be made.

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

- Storing P and Q blocks

When rough cutting is executed by G71.7, G72.7, or G73.7, up to three memory addresses of P and Q blocks are stored. By this, the blocks indicated by P and Q are immediately found at execution of G70.7 without searching memory from the beginning for them. After some G71.7, G72.7, and G73.7 rough cutting cycles are executed, finishing cycles can be performed by G70.7 at a time. At this time, for the fourth and subsequent rough cutting cycles, the cycle time is longer because memory is searched for P and Q blocks.

Example

G71.7 P100 Q200 --- ;

N100 --- ;

--- ;

--- ;

N200 --- ;

G71.7 P300 Q400 --- ;

N300 --- ;

--- ;

--- ;

N400 --- ;

--- ;

--- ;

G70.7 P100 Q200 ; (Executed without a search for the first to third cycles)

G70.7 P300 Q400 ; (Executed after a search for the fourth and subsequent cycles)

NOTE

The memory addresses of P and Q blocks stored during rough cutting cycles by G71.7, G72.7, and G73.7 are erased after execution of G70.7.

All stored memory addresses of P and Q blocks are also erased by a reset.

- Return to the cycle start point

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

NOTE

The tool returns to the cycle start point always in the nonlinear positioning mode regardless of the setting of bit 1 (LRP) of parameter No. 1401.

Before executing a finishing cycle for a target figure with a pocket cut by G71.7 or G72.7, check that the tool does not interfere with the workpiece when returning from the end point of the target figure to the cycle start point.

- Tool nose radius compensation

When using tool nose radius compensation, specify a tool nose radius compensation command (G41 or G42) before a multiple repetitive canned cycle command (G70.7) and specify the cancel command (G40) after the multiple repetitive canned cycle command (G70.7).

Program example

G42;.....Specify this command before a multiple repetitive canned cycle command.

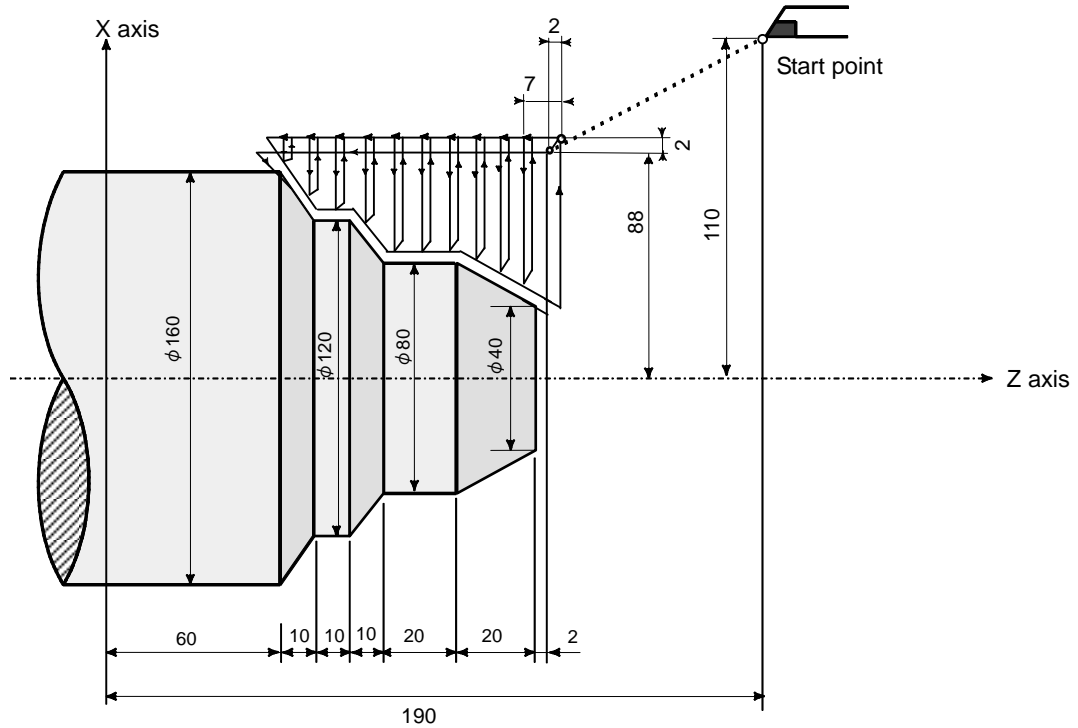
G70.7P10Q20;.....

G40;.....Specify this command after a multiple repetitive canned cycle command.

Like G71.7, this cycle operation is performed according to the figure determined by the tool nose radius compensation path when the offset vector is 0 at start point A and start-up is performed in a block between path A-A'.

Example

Stock removal in facing (G72.7)



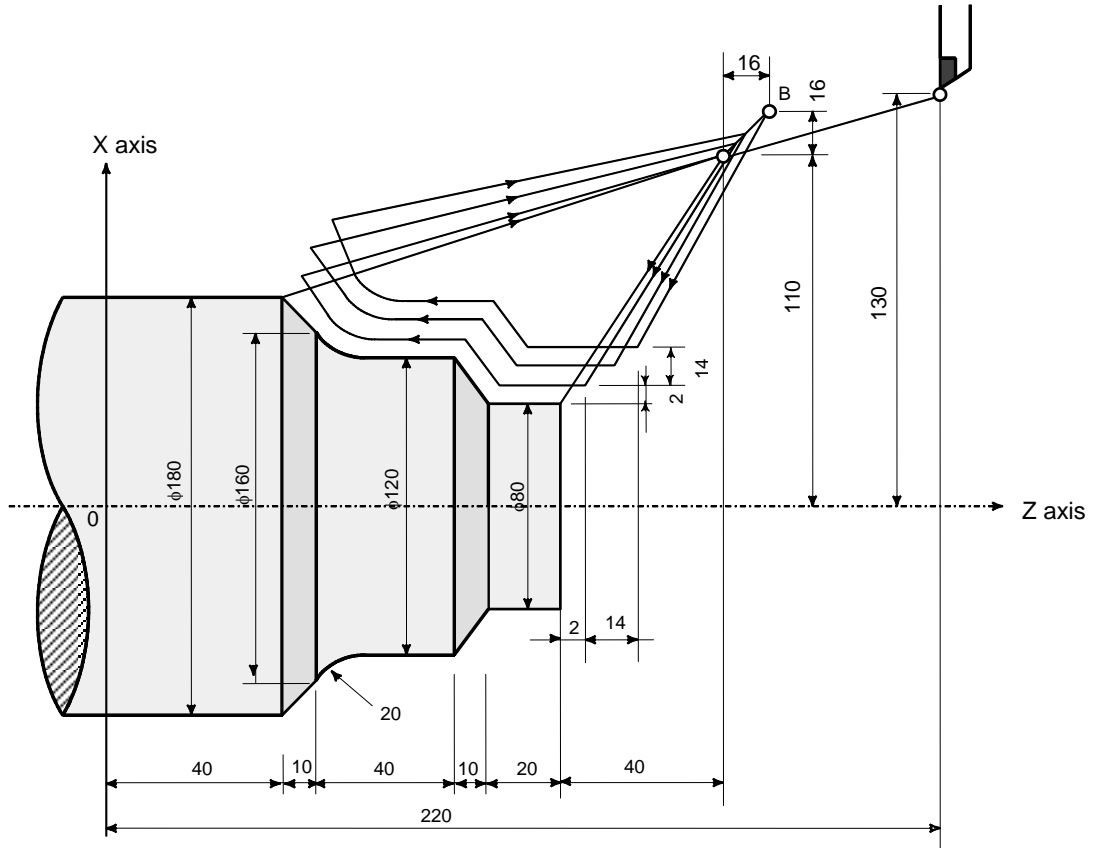
(Diameter designation for X axis, metric input)

```

N011 G90 G92 X220.0 Z190.0 ;
N012 G00 X176.0 Z132.0 ;
N013 G72.7 P014 Q019 U4.0 W2.0 D7000 F0.3 S550 ;
N014 G00 Z56.0 S700 ;
N015 G90 G01 X120.0 Z70.0 F0.15 ;
N016 Z80.0 ;
N017 X80.0 Z90.0 ;
N018 Z110.0 ;
N019 X36.0 Z132.0 ;
N020 G70.7 P014 Q019 ;
    
```

Parameter No.5133=1.0 (Escaping amount)
 Finishing allowance (4.0 in diameter in the X direction, 2.0 in the Z direction)

Pattern repeating (G73.7)



(Diameter designation, metric input)

```

N011 G90 G92 X260.0 Z220.0 ;
N012 G00 X220.0 Z160.0 ;
N013 G73.7 P014 Q019 U4.0 W2.0 I14.0 K14.0 D3 F0.3 S0180 ;
N014 G00 X80.0 Z120.0 ;
N015 G01 Z100.0 F0.15 S0600 ;
N016 X120.0 Z90.0 ;
N017 Z70.0 S0400 ;
N018 G02 X160.0 Z50.0 R20.0 ;
N019 G01 X180.0 Z40.0 S0280 ;
N020 G70.7 P014 Q019 ;
    
```

7.1.5 End Face Peck Drilling Cycle (G74.7)

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

Format

G74.7X_ Z_ I(Δi) K(Δk) D(Δd) F(f) ;

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

For an absolute command, coordinates (X,Z)

For an incremental command, travel distance (Δx , Δz)

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

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

Δd : Relief amount of the tool at the cutting bottom

f : Feedrate

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

NOTE

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

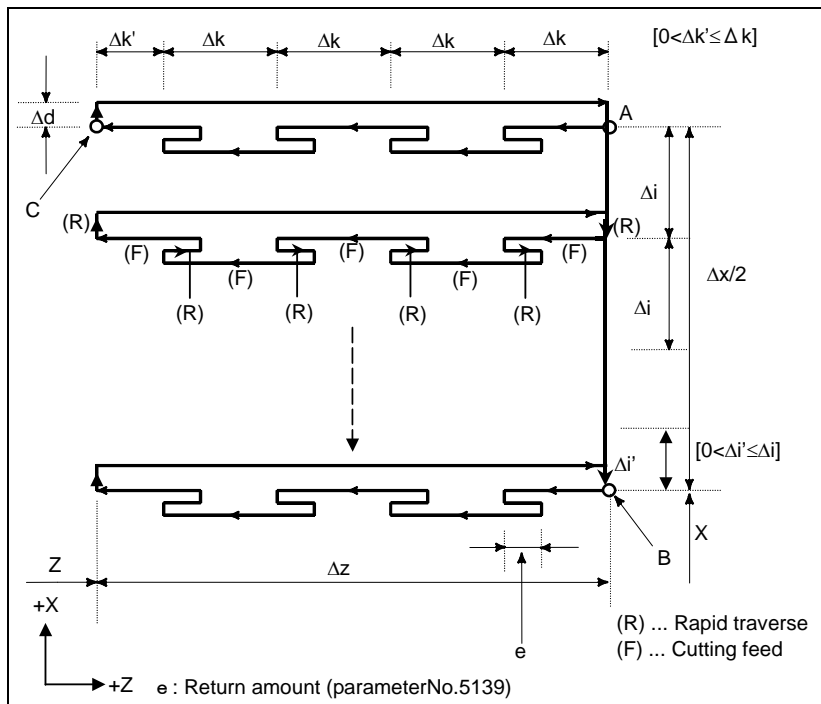


Fig. 7.1.5 (a) Cutting path in end face peck drilling cycle

Explanation**- Operations**

A cycle operation of cutting by Δk and return by e is repeated.

When cutting reaches point C, the tool escapes by Δd . Then, the tool returns in rapid traverse, moves to the direction of point B by Δi , and performs cutting again.

- Return amount (e)

Set a return amount in parameter No. 5139.

No.	Unit	Diameter/radius programming	Sign
5139	Depends on the increment system for the reference axis.	Radius programming	Not required

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

7.1.6 Outer Diameter / Internal Diameter Drilling Cycle (G75.7)

This cycle is equivalent to G74.7 except that the second axis on the plane (X-axis for the ZX plane) changes places with the first axis on the plane (Z-axis for the ZX plane). This cycle enables chip breaking in end facing. It also enables grooving during outer diameter cutting and cutting off (when the Z-axis and Q are omitted for the first axis on the plane).

Format

G75.7 X_ Z_ I(Δi) K(Δk) D(Δd) F (f) ;

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

For an absolute command, coordinates (X,Z)

For an incremental command, travel distance (Δx , Δz)

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

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

Δd : Relief amount of the tool at the cutting bottom

f : Feedrate

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

NOTE

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

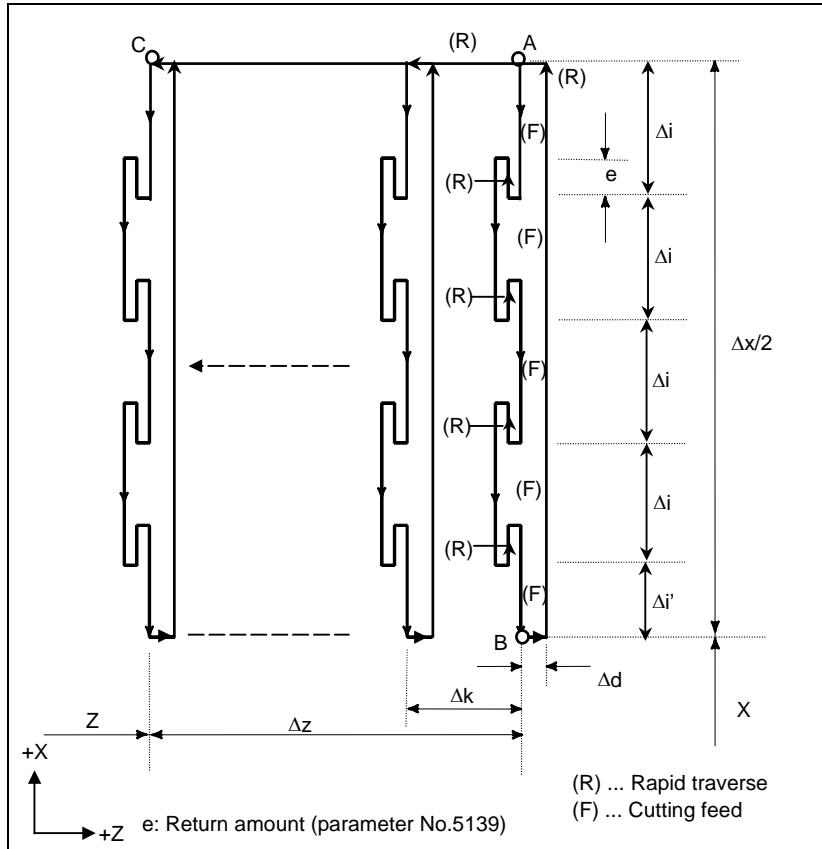


Fig. 7.1.6 (a) Outer diameter/internal diameter drilling cycle

Explanation

- Operations

A cycle operation of cutting by Δi and return by e is repeated.

When cutting reaches point B, the tool escapes by Δd . Then, the tool returns in rapid traverse, moves to the direction of point C by Δk , and performs cutting again.

Both G74.7 and G75.7 are used for grooving and drilling, and permit the tool to relief automatically. Four symmetrical patterns are considered, respectively.

- Return amount (e)

Set a return amount (e) in parameter No. 5139.

No.	Unit	Diameter/radius programming	Sign
5139	Depends on the increment system for the reference axis.	Radius programming	Not required

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

7.1.7 Multiple Threading Cycle (G76.7)

This threading cycle allows selection of four cut methods.

Format

G76.7 X_ Z_ I(i) K(k) D(Δ d) A(a) F(L) P(p) Q(q) ;

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

For an absolute command, coordinates (X,Z)

For an incremental command, travel distance (Δ x, Δ z)

a : Angle of tool nose (thread angle) in degree units, the range being 0 to 120.
(If omitted, the angle is assumed 0 degrees.)

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

k : Height of thread

Δ d : Depth of cut in 1st cut

L : Lead of thread

p : Cut method (if omitted and if P0 is specified, the method is single-edged cutting with a constant cutting amount.)

P1: Single-edged cutting with a constant cutting amount

P2: Both-edge zigzag thread cutting with a constant cutting amount

P3: Single-edged cutting with a constant depth of cut

P4: Both-edge zigzag thread cutting with a constant depth of cut

q : Threading start angle shift
(0.001-degree units, range being 0 to 360 degrees)

NOTE

- 1 Address A is invalid even if a decimal point is added. That is, if 120 degrees is to be specified, A120. is regarded as equivalent to A120.
- 2 Using cut methods P2 to P4 requires the multiple repetitive canned cycle II option.

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

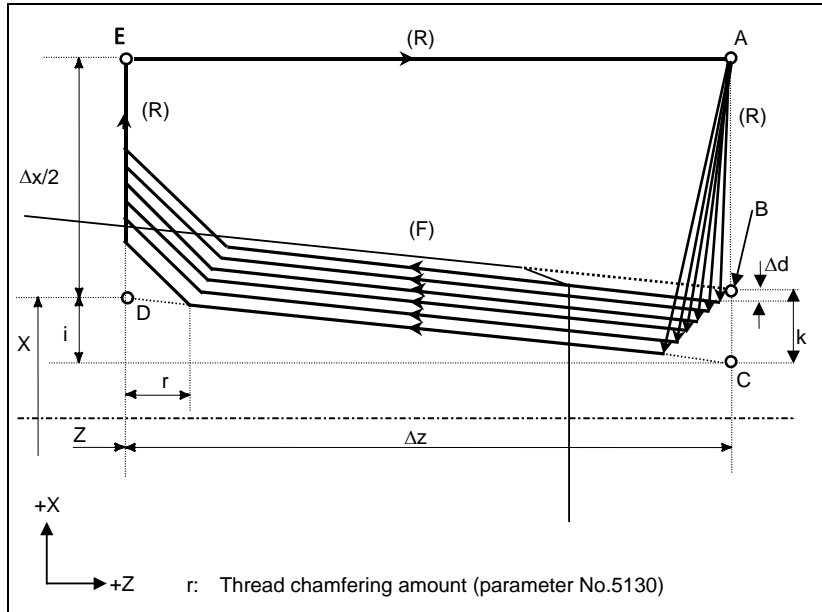


Fig. 7.1.7 (a) Cutting path in multiple threading cycle

Explanation

This cycle performs threading so that the length of the lead only between C and D is made as specified in the F code. In other sections, the tool moves in rapid traverse.

CAUTION

Notes on threading are the same as those on G33 threading. For feed hold in a threading cycle, however, see "Feed hold in a threading cycle" described below.

- Cut methods

Four cut methods are available.

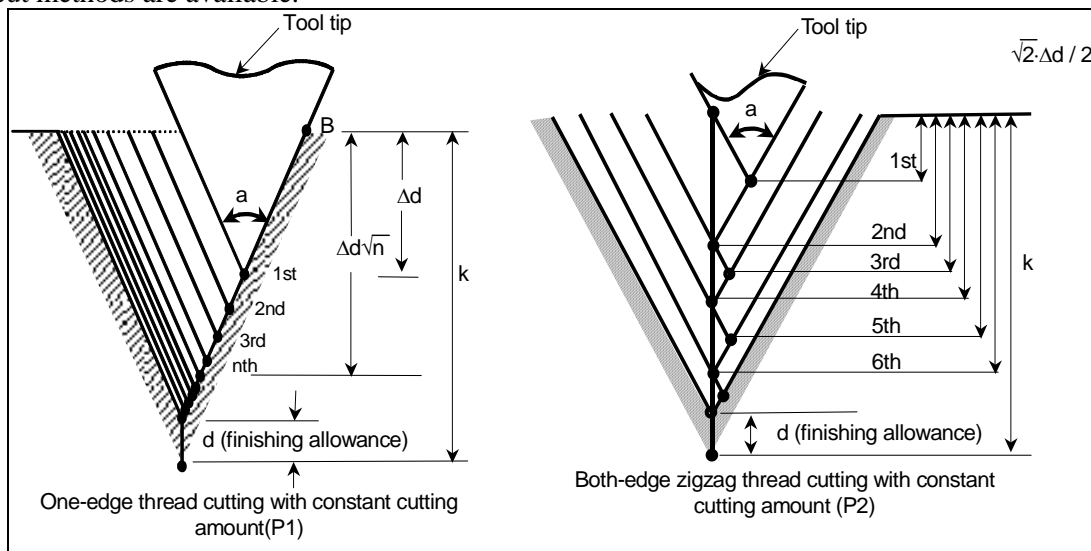


Fig. 7.1.7 (b) Single-edge cutting/both-edge zigzag thread cutting with a constant cutting amount (P1/2)

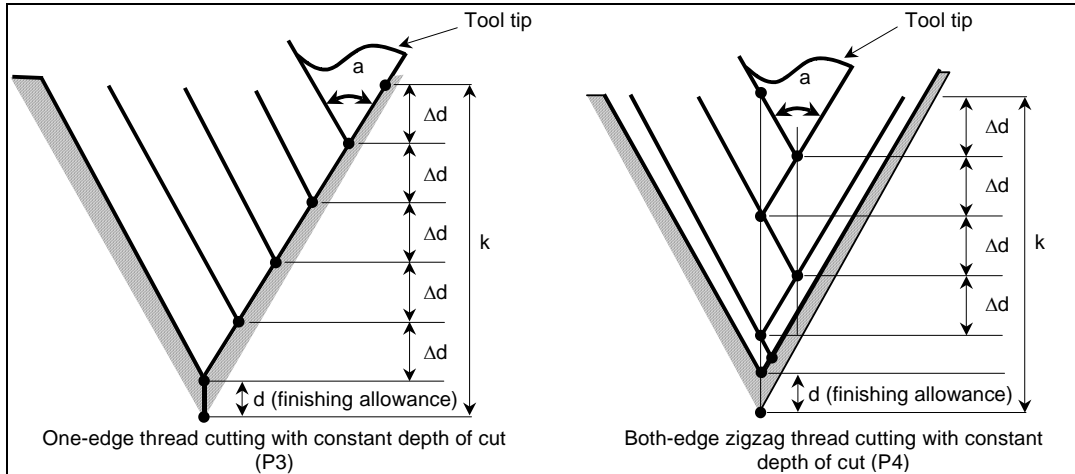


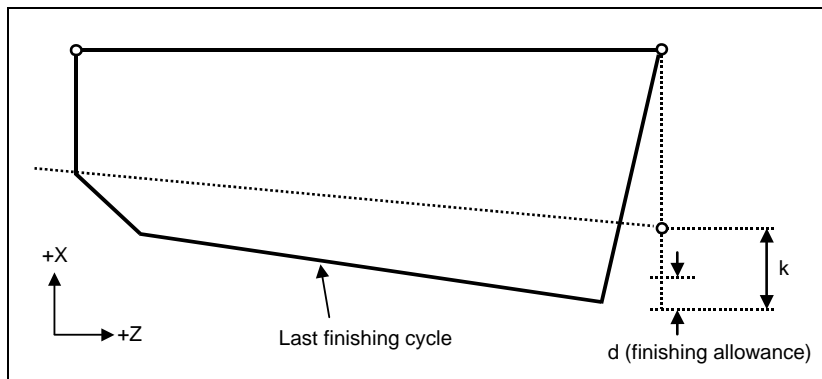
Fig. 7.1.7 (c) Single-edge cutting/both-edge zigzag thread cutting with a constant cut-in value (P3/4)

- Repetitive count in finishing

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

Set a repetitive count in parameter No. 5142.

If the setting is 0, it is assumed to be 1.



- Minimum cut-in value

If cut-in with a constant cutting amount is selected (P1, P2), it can be clamped to a minimum cut-in value so that the cut-in value does not become too small.

Set a minimum cut-in value in parameter No. 5140.

No.	Unit	Diameter/radius programming	Sign
5140	Depends on the increment system for the reference axis.	Radius programming	Not required

- Finishing allowance

Set a finishing allowance in parameter No. 5141.

No.	Unit	Diameter/radius programming	Sign
5141	Depends on the increment system for the reference axis.	Radius programming	Not required

NOTE

Specify a value smaller than the height of thread as the finishing allowance.
(dNo.5141 < k)

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

The signs of incremental dimensions for the cycle shown in Fig. 7.1.7 (a) are as follows:

Cutting end point in the direction of the length for X and Z:

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

Taper amount (i) : Minus (determined according to the direction of path A-C)

Height of thread (k) : Plus (always specified with a plus sign)

Depth of cut in the first cut (Δd) : Plus (always specified with a plus sign)

The four patterns shown in the Table 7.1.7 (a) are considered corresponding to the sign of each address. A female thread can also be machined.

Table 7.1.7 (a)

Outer diameter machining	Internal diameter machining
<p>1. $\Delta x < 0, \Delta z < 0, i < 0$</p>	<p>2. $\Delta x > 0, \Delta z < 0, i > 0$</p>
<p>3. $\Delta x < 0, \Delta z < 0, i > 0$ at $i \leq \Delta x / 2$</p>	<p>4. $\Delta x > 0, \Delta z < 0, i < 0$ at $i \leq \Delta x / 2$</p>

- Acceleration/deceleration after interpolation for threading

Acceleration/deceleration after interpolation for threading is acceleration/deceleration of exponential interpolation type. By setting bit 5 (THLx) of parameter No. 1610, the same acceleration/deceleration as for cutting feed can be selected. (The settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610 are followed.) However, as a time constant and FL feedrate, the settings of parameters Nos. 1626 and 1627 for the threading cycle are used.

- Time constant and FL feedrate for threading

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

- Thread chamfering

Thread chamfering can be performed in this threading cycle. A signal from the machine tool initiates thread chamfering.

Assuming that the lead is set to L with parameter No. 5130, the thread chamfering value may be set to any value in the range of 0.1L to 12.7L, in 0.1L steps.

A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.

For thread chamfering, the same type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate as for threading are used.

- Retraction after chamfering

The Table 7.1.7 (b) lists the feedrate, type of acceleration/deceleration after interpolation, and time constant of retraction after chamfering.

Table 7.1.7 (b)

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

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

NOTE

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

- Shifting the start angle

Address Q can be used to shift the threading start angle.

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

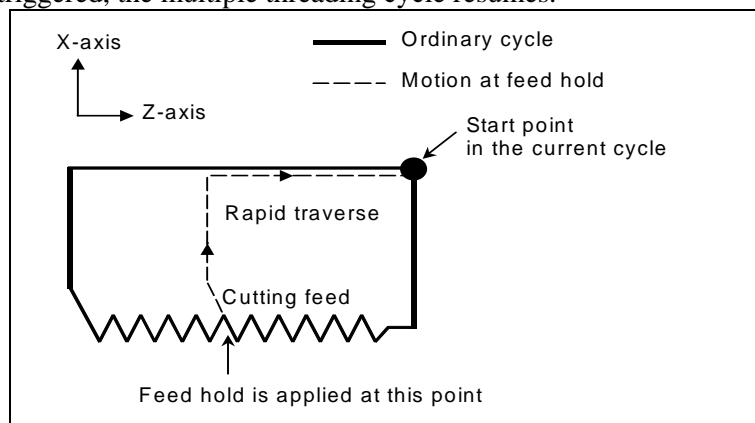
- Feed hold in a threading cycle

When the threading cycle retract function is not used, the machine stops at the end point of retraction after chamfering (point E on the cutting path for a multiple threading cycle) by feed hold applied during threading.

- Feed hold when the threading cycle retract function is used

When the "threading cycle retract" option function is used, feed hold may be applied during threading in a multiple threading cycle (G76). In this case, the tool quickly retracts in the same way as for the last chamfering in a threading cycle and returns to the start point in the current cycle.

When cycle start is triggered, the multiple threading cycle resumes.



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



CAUTION

Another feed hold cannot be performed during retraction.

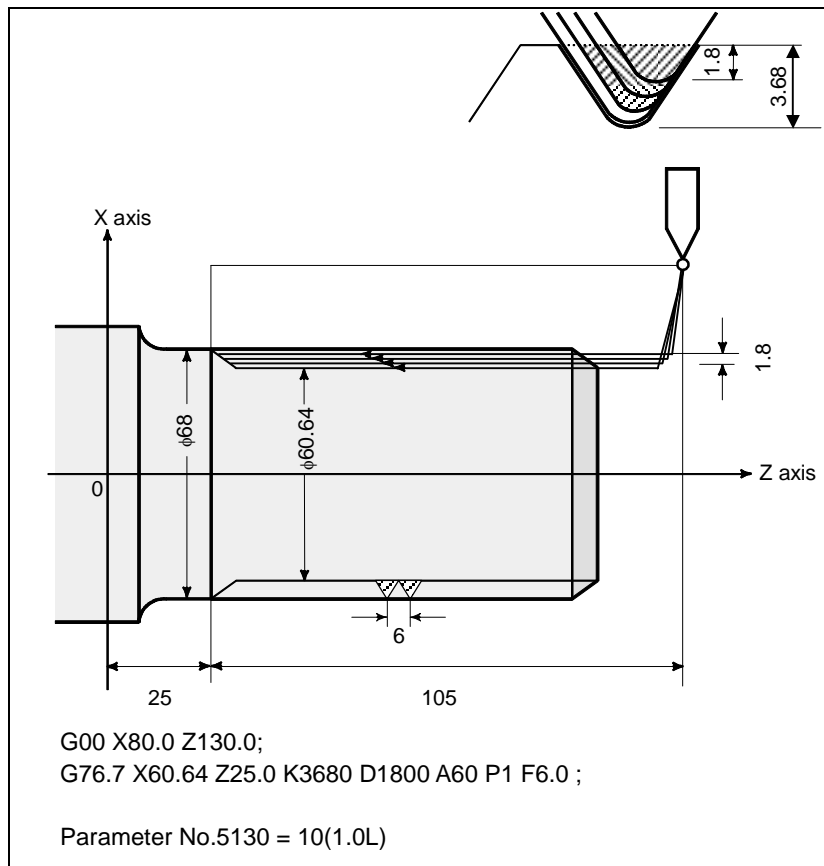
- Inch threading

Inch threading specified with address E is allowed.

- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

Example



7.1.8 Restrictions on Multiple Repetitive Cycle

Programmed commands

- Program memory

Programs using G70.7, G71.7, G72.7, or G73.7 must be stored in the program memory. The use of the mode in which programs stored in the program memory are called for operation enables these programs to be executed in other than the MEM mode. Programs using G74.7, G75.7, or G76.7 need not be stored in the program memory.

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

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

In a block in which G70.7, G71.7, G72.7, or G73.7 is specified, the following functions cannot be specified:

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

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

In the block which is specified by address P of a G71.7, G72.7 or G73.7, G00 or G01 code in group 01 should be commanded. If it is not commanded, alarm PS0065, “G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM” is generated.

In blocks with sequence numbers between those specified at P and Q in G70.7, G71.7, G72.7, and G73.7, the following commands can be specified:

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

When G70.7, G71.7, G72.7, or G73.7 is executed, the sequence number specified by address P and Q should not be specified twice or more in the same program.

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

Relation with other functions

- **Manual intervention**

While a multiple repetitive cycle (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7, and G76.7) is being executed, it is possible to stop the cycle and to perform manual intervention.

The setting of manual absolute on or off is effective for manual operation.

- **Interruption type macro**

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

- **Program restart and tool retract and recover**

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

- **Axis name and second auxiliary functions**

Even if address U, V, W, or A is used as an axis name or second auxiliary function, data specified at address U, V, W, or A in a G71.7, G72.7, or G73.7 block is assumed to be that for the multiple repetitive cycle.

- **Tool nose radius compensation**

When using tool nose radius compensation, specify a tool nose radius compensation command (G41, G42) before a multiple repetitive canned cycle command (G70.7, G71.7, G72.7, G73.7) and specify the cancel command (G40) outside the programs (from the block specified with P to the block specified with Q) specifying a target finishing figure. If tool nose radius compensation is specified in the program specifying a target finishing figure, alarm PS0325, “UNAVAILABLE COMMAND IS IN SHAPE PROGRAM”, is issued.

8

AXIS CONTROL FUNCTIONS

Chapter 8, "AXIS CONTROL FUNCTIONS", consists of the following sections:

8.1 PARALLEL AXIS CONTROL314

8.1 PARALLEL AXIS CONTROL

Overview

If a machine contains multiple heads and tables to machine multiple workpieces simultaneously, it is possible to move multiple controlled axes assigned the same axis name, by using a move command for a single program axis. This is called parallel operation.

Two or more axes that are moved simultaneously when a command for a single program axis, i.e., a command with a single address, is issued are called parallel axes.

This function is effective to automatic operation, MDI operation, and manual numerical commands on 1-path machining centers. In manual operation, parallel operation is not possible, each controlled axis is moved independently.

In parallel operation, the controlled axes belonging to a single program axis are moved in the same way in principle. It is, however, possible to select specified ones from among multiple parallel axes and move them (park the other axes), using appropriate external signals.

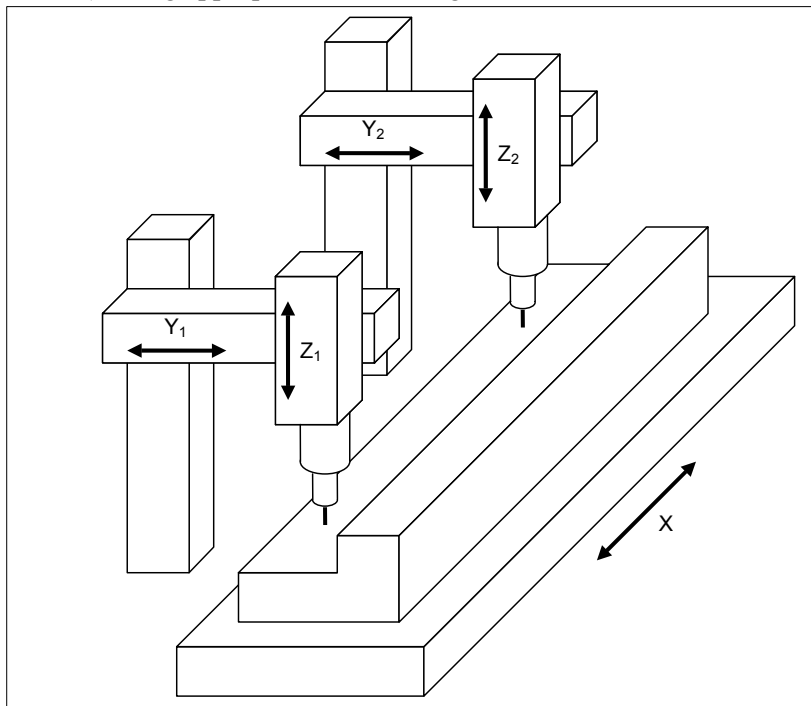


Fig. 8.1 (a)

In the Fig. 8.1 (a), Y_1 and Y_2 are parallel axes, and are operated in parallel with a command for a single address Y .

Z_1 and Z_2 are also parallel axes, and are operated in parallel with a command for a single address Z .

There are two movements of each parallel axis, which are selected with appropriate external input signals.

- Normal (parking off)
The axis moves as commanded.
- Parking (parking on)
The axis ignores the command, and does not move.

NOTE

- 1 The parallel axis control function is effective to 1-path machining centers only.
- 2 The parallel axis control function does not support the functions below.
 - (1) Smooth interpolation
 - (2) Nano smoothing
 - (3) 3-dimensional coordinate system conversion
 - (4) 3-dimensional tool compensation
 - (5) Tool length compensation in tool axis direction
 - (6) Tool center point control
 - (7) Tool posture control
 - (8) 3-dimensional cutter compensation
 - (9) Tilted working plane indexing
 - (10) Nano smoothing 2
 - (11) Workpiece setting error compensation
 - (12) Tandem control
 - (13) Twin table control
 - (14) Synchronous/Composite control
 - (15) Superimposed Control
- 3 If coordinate system rotation (G68) is to be specified, the program coordinate systems of the parallel axes must be identical. In the immediately preceding block, therefore, it is necessary to execute a move command on all parallel axes, using an absolute command (G90). (As long as the command is executed, there may be any axes in parking.)
If the program coordinate systems of the parallel axes are not identical, alarm PS0508, "G CODE TO NEED G90(PAC)" is issued.

Axis configuration in parallel axis control

The following limitations are imposed on the axis configuration in parallel axis control:

- (1) All parallel axes must have the same axis name.
If different axis names are set, the axes cannot be recognized as parallel axes.
- (2) To distinguish parallel axes from one another, specify the subscript for each of the axes.
- (3) Parallel axes are regarded as controlled axes, and their respective speeds are added to the actual cutting feedrate display. It is possible not to add the speed of a slave axis to the actual cutting feedrate display (by setting bit 3 (NDFx) of parameter No. 3115 to 1).
- (4) To perform circular interpolation or cutter compensation, it is necessary to specify to which basic coordinate system each axis belongs, using parameter No. 1022. At this time, make the same settings for parallel axes.
- (5) If parallel axis control is used together with feed axis synchronization control, make sure that the axes subject to feed axis synchronization control have different axis names.

The following shows an example in which parallel axis control is performed on a machine with five axes X, Y1, Y2, Z1, and Z2.

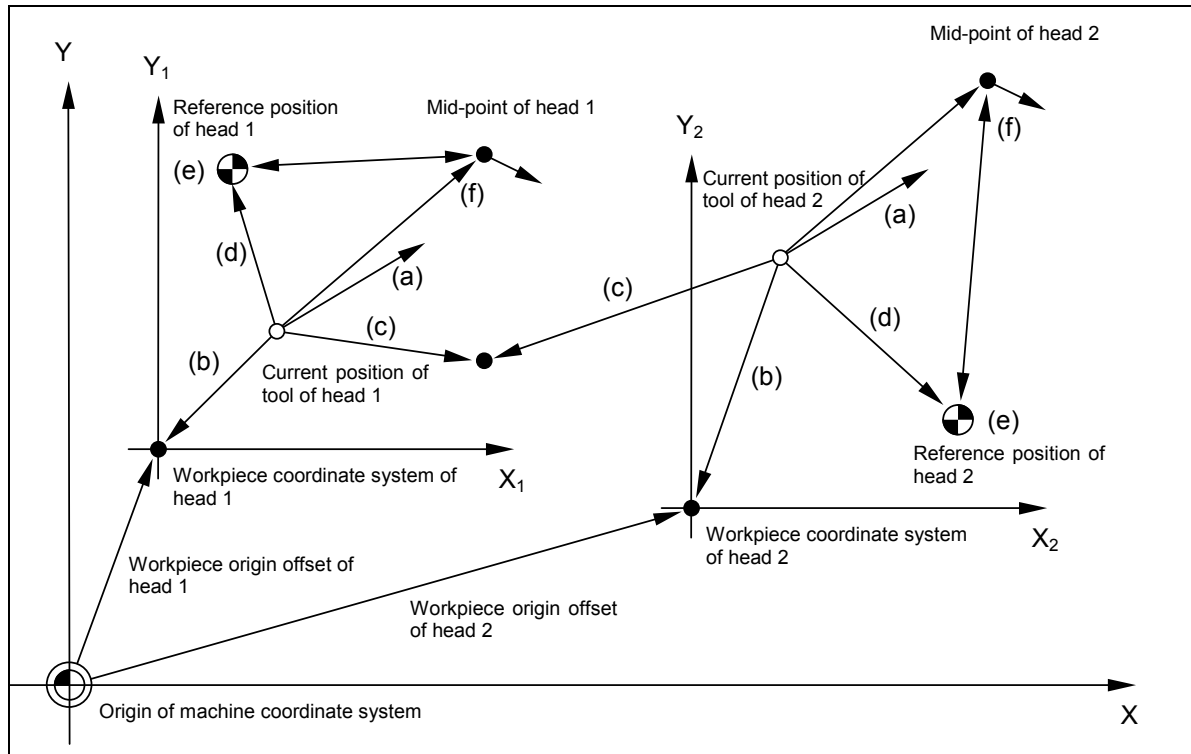
Number of controlled axes = 5

Axis number	Displayed axis name	Axis name (No. 1020)	Subscript (No. 3131)	Basic coordinate axis setting (No. 1022)	Servo axis number (No. 1023)
1	X	88	0	1	1
2	Y1	89	1	2	2
3	Y2	89	2	2	3
4	Z1	90	1	3	4
5	Z2	90	2	3	5

Selecting coordinate systems on parallel axes

It is possible to set individual workpiece coordinate system offset values for the controlled axes belonging to the same program axis. This makes it possible to create a program by considering independent coordinate systems for the individual controlled axes.

For example, if there are two heads, each of which have two controlled axes (X and Y), the tool movement is as shown below.



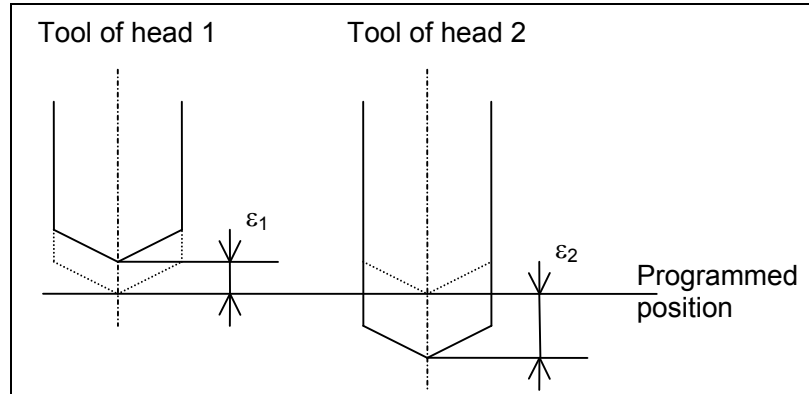
- (a) Movement with an incremental command
(Example) G91 X_ Y_;
- (b) Movement with an absolute command
(Example) G90 X0 Y0;
- (c) Move command with the machine coordinate system
Each axis moves to a single point on the machine coordinate system.
(Example) G90 G53 X_ Y_;
- (d) Automatic reference position return (G28, G30)
Each axis returns to the reference position of its controlled axis individually set with a parameter.
(Example) G91 G28 X0 Y0;
- (e) Reference position return check (G27)
A check is performed to see if each axis has returned to its reference position.
(Example) G91 G27 X0 Y0;
- (f) Automatic return to the reference position (G29)
Each axis is positioned to the position specified via its mid-point.
(Example) G91 G29 X30. Y50.;

Tool length compensation and tool offset on parallel axes

It is possible to set the difference between an H code number and the offset data number selected in accordance with this number, i.e., bias value, in a parameter for each axis, and perform tool length compensation on a different tool for each axis. It is also possible to apply an offset in the case of the tool offset function.

Set the bias value of tool length compensation in parameter No. 10361.

Set the bias amount of tool offset in parameter No. 10360.



Head	Offset number	Bias value	Offset data number	Offset value
Head 1	07	10	17	ε_1
Head 2	07	20	27	ε_2

If parallel operation is to be performed with the third and fourth axes as the Z1- and Z2-axes, let us assume, for example, that 10 is set in parameter No. 10361 as the bias value for the Z1-axis, and 20 for the Z2-axis. If a tool length compensation command G43 H07 ; is specified, tool length compensation is performed by using the offset data number 17 (= 07 + 10) as the offset value for the tool of head 1 and the offset data number 27 (= 07 + 20) as the offset value for the tool of head 2.

NOTE

The offset value for the offset number 00, i.e., H00, is always 0 regardless of the bias value.

Travel distances on parallel axes

The travel distances on parallel axes differ as described below depending on whether the command is an incremental or absolute one.

- (1) For an incremental command
 - Rapid traverse and linear interpolation
The travel distances on all parallel axes are the same.
 - Circular interpolation and helical interpolation
The travel distances on all parallel axes are the same.
This means that the same arc is interpolated simultaneously for a multiple number of times.
- (2) For an absolute command
 - Rapid traverse and linear interpolation
The end points of all parallel axes are the same.
This means that if the start positions differ, the travel distances on the parallel axes differ.
 - Circular interpolation and helical interpolation
Interpolation is performed using the data for the first un-parked one in the list of controlled axes, and for the other parallel axes, the same data as that used for interpolation is output.
This means that even if the command is an absolute one, if the start position of a parallel axis differs from that of the axis the data for which is used for interpolation, the end point differs from the specified one.

Feedrate calculation

For linear interpolation, the feedrate during parallel operation is calculated by selecting the one of the parallel axes that has the longest travel distance. The data for the other parallel axes is not reflected in the calculation of the feedrate.

(Example)

If the X-axis has parallel axes (X_1 , X_2)

Start point position

X_1 : 0.0

X_2 : 5.0

Y : 0.0

Command

G01 G90 X10. Y20. F500

The travel distance on X_1 is 10.0 and that on X_2 is 5.0. Thus, the feedrate is calculated using the distance of X_1 , which is longer, as follows:

Assume that the feedrate on each axis is

$$L = \sqrt{10^2 + 20^2}$$

$$X_1: 500 * 10 / L$$

$$X_2: 500 * 5 / L$$

$$Y : 500 * 20 / L$$

9 GAS CUTTING MACHINE

Chapter 9, "GAS CUTTING MACHINE", consists of the following sections:

9.1	TOOL OFFSET B	319
9.2	CONER CONTROL BY FEED RATE	322
9.3	AUTOMATIC EXACT STOP CHECK	324
9.4	AXIS SWITCHING	327
9.5	GENTLE CURVE CUTTING	330
9.6	GENTLE NORMAL DIRECTION CONTROL	332

9.1 TOOL OFFSET B

Overview

With the G43H_ or G44H_ command, tool offset not affected by mirror image can be applied.
 With the G49 command, tool offset can be canceled without being affected by mirror image.
 The offset axes are the first axis and second axis for plane selection.

Format

G43 H_ ; Shifts the coordinate system by an offset value in the + direction.
G44 H_ ; Shifts the coordinate system by an offset value in the - direction.
G49 ; or G43 (or G44) H0 ; Tool offset B cancel
 G43 (or G44) : Tool offset in the + (or -) direction
 G49 (or H0) : Tool offset cancel
 H_ : Tool offset number
 (G43,G44,G49: Modal G code)

Explanation

By setting bit 0 (GOB) of parameter No. 5033 to 1 and specifying G43H_ ; or G44H_ ;, tool offset not affected by mirror image can be applied. The difference between the programmed tool position and the tool position actually used for machining can be corrected using this function without modifying the program, by setting the difference in the offset memory.

By using the H code, specify the offset memory number where a tool offset value is set.

As shown in Fig. 9.1 (a), eight offset directions are present for each of G43 and G44. Select a direction with parameter No. 5032.

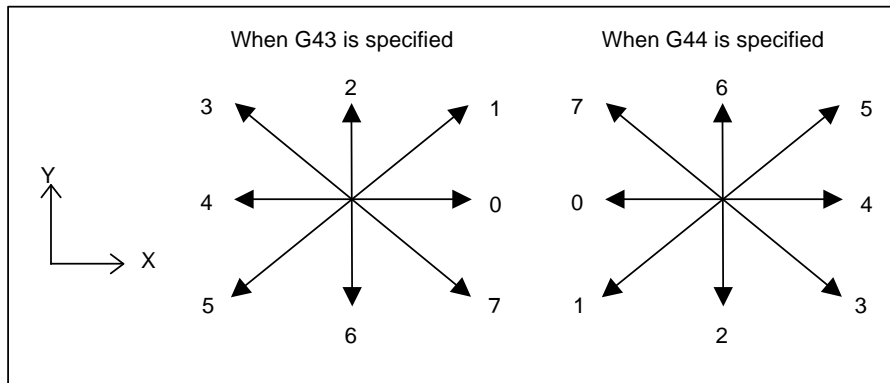


Fig. 9.1 (a)

Value set in parameter No. 5032	Offset direction	
	G43	G44
0	X+a	X-a
1	X+a Y+a	X-a Y-a
2	Y+a	Y-a
3	X-a Y+a	X+a Y-a
4	X-a	X+a
5	X-a Y-a	X+a Y+a
6	Y-a	Y+a
7	X+a Y-a	X-a Y+a

a: Offset value set in the offset memory number specified with the H code

By specifying
G49 ;

tool offset can be canceled without being affected by mirror image.

For example, this function can correct the difference between the position of the nozzle of the gas cutting machine actually used for machining and the position of the marking pen for drawing used to check the program, without modifying the program.

As in the sample program indicated below, the program need not be modified if the difference between the position of the nozzle of the gas cutting machine and the position of the marking pen is set as an offset value in offset number 02, and the optional block skip signal BDTn is set to 1 to perform gas cutting or is set to 0 to perform marking.

```
O0001;
:
/G43 H02;
:
/G49;
:
%
```

NOTE
In a G43, G44, or G49 block, no move command must be specified.
Otherwise, an alarm is issued.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5033								GOB

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 GOB The tool offset B function (for a gas cutting machine) is:
0: Disabled.
1: Enabled.

5032	
	Direction of tool offset B

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to 7

Specify the offset direction of tool offset B (G43, G44).

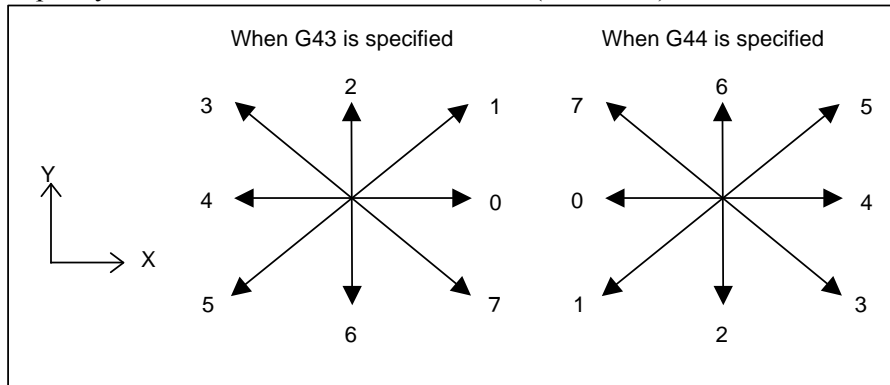


Fig. 9.1 (b)

Value set in parameter No. 5032	Offset direction	
	G43	G44
0	X+a	X-a
1	X+a Y+a	X-a Y-a
2	Y+a	Y-a
3	X-a Y+a	X+a Y-a
4	X-a	X+a
5	X-a Y-a	X+a Y+a
6	Y-a	Y+a
7	X+a Y-a	X-a Y+a

a: Offset value set in the offset memory number specified with the H code

Note

- This function is optional. The tool offset B function needs to be specified.
- This function can be used only with the M series.

9.2 CONER CONTROL BY FEED RATE

Overview

If a block for cutting feed is followed by another block for cutting feed, the number of accumulated pulses in the automatic acceleration/deceleration circuit of each axis in the block being executed is assumed to be 0 and machining proceeds to the next block when the feedrate being decelerated on each axis with the automatic acceleration/deceleration circuit has decreased to or below the feedrate set in parameter No. 1474.

Explanation

If a block (block A) for cutting feed is followed by another block (block B) for cutting feed, the number of accumulated pulses in the automatic acceleration/deceleration circuit of each axis in block A is assumed to be 0 and machining proceeds to block B when the feedrate being decelerated on each axis with the automatic acceleration/deceleration circuit has decreased to or below the feedrate set in parameter No. 1474.

This function is enabled when block A is followed by block B and one of 1) through 4) is applicable. In Fig. 9.2 (a), for example, there are accumulated pulses in the shaded area at time t, but machining proceeds to the next block because the feedrate decelerated by the automatic acceleration/deceleration circuit has decreased to or below the parameter-set value.

- 1) Block A specifies G09 (exact stop check).
- 2) Block A is in the G61 mode (exact stop check mode).
- 3) In the automatic exact stop check mode, the angle formed by block A and block B is equal to or less than the automatic exact stop check critical angle (parameter No. 1496).
- 4) The in-position check signal SMZ <Gn053.6> is set to 1.

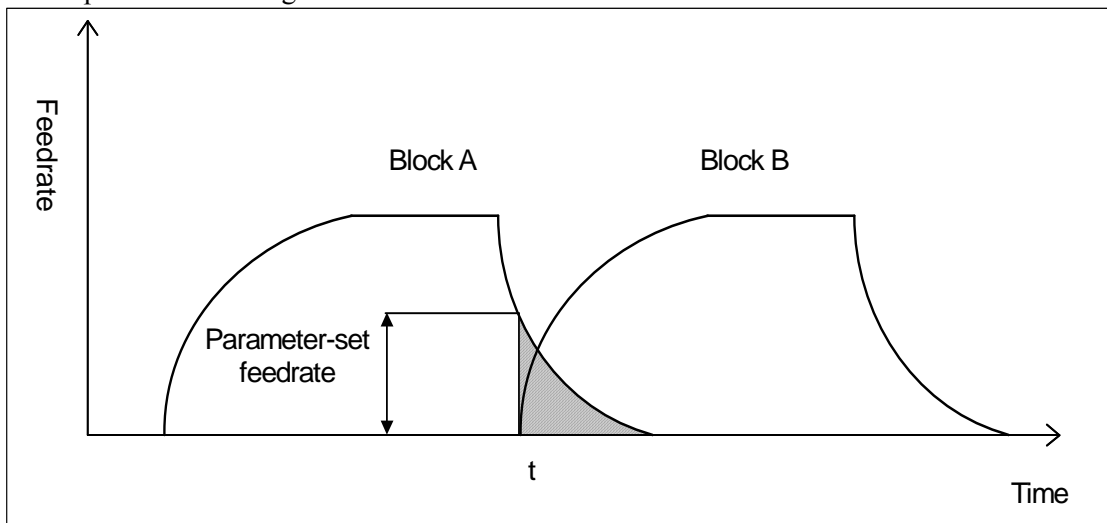


Fig. 9.2 (a)

Please refer to Subsection, “In-position check” in Connection Manual (FUNCTION) (B-64483EN-1) about the in-position check signal.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403					EDT			

[Input type] Parameter input

[Data type] Bit path

#3 EDT The function for corner control by feedrate (for a gas cutting machine) is:

0: Disabled.

1: Enabled.

When the feedrate has reduced to the feedrate set in parameter No. 1474, from which the system regards the number of accumulated pulses as being 0, the next block is executed.

1474	Feedrate regarded as accumulated pulse 0. (corner control by feedrate (for gas cutting machine))
------	---

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/min,inch/min,deg/min (machine unit)

[Valid data range] 0 to 32767

When a cutting feed block (block A) is followed by another cutting feed block (block B), execution proceeds to block B if the feedrate in the automatic acceleration/deceleration circuit for each axis is reduced to the setting of this parameter, and the number of accumulated pulses in the automatic acceleration/deceleration circuit is assumed to be 0. This setting is used for corner control by feedrate (for a gas cutting machine).

Note

- 1) This function is optional. The function for corner control by feedrate needs to be specified.
- 2) This function can be used only with the M series.
- 3) If a block for cutting feed is followed by a block specifying a miscellaneous function only or a block for rapid traverse, machining proceeds to the next block when the number of accumulated pulses has actually decreased to 0.
- 4) When enabling this function by setting bit 3 (EDT) of parameter No. 1403 to 1, set bit 5 (NCI) of parameter No. 1601 to 1 and check only that the specified feedrate is 0 at deceleration time (delay in acceleration/deceleration is 0).
- 5) If a value other than 0 is specified in parameter No. 1474, the number of accumulated pulses in the automatic acceleration/deceleration circuit of each axis is assumed to be 0 and machining proceeds to the next block when the feedrate being decelerated on each axis with the automatic acceleration/deceleration circuit has decreased to or below the feedrate set in parameter No. 1474. So, the path at a corner may be round as shown below (Fig. 9.2 (b)).

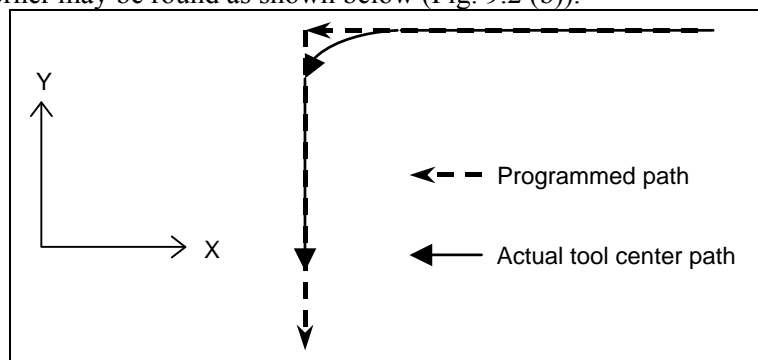


Fig. 9.2 (b)

So, when a sharp corner is needed particularly, insert a block such as for dwell (G04) between successive blocks for cutting feed.

<Example>

G09 G01 X Y ;

G04 X0;

G09 G01 X Y ;

- 6) When acceleration/deceleration before interpolation is performed, this function is disabled.
- 7) If feed per revolution is specified in one of successive blocks, this function is disabled.

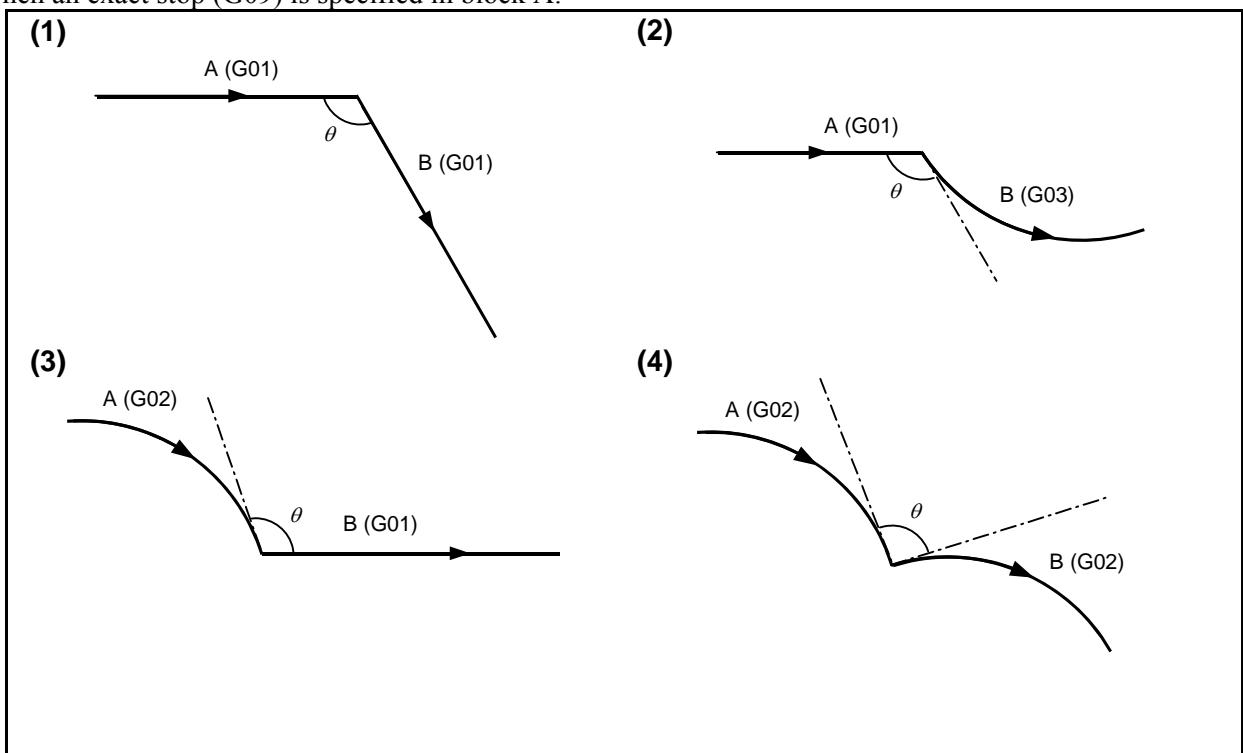
9.3 AUTOMATIC EXACT STOP CHECK

Overview

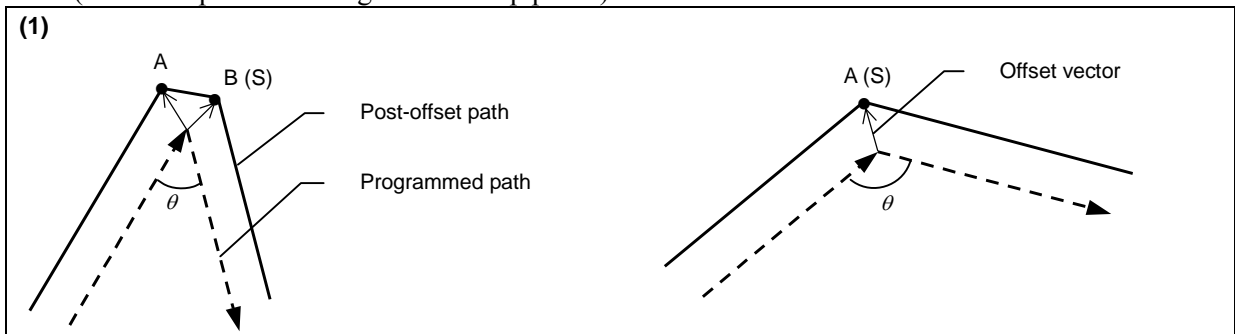
This function checks the corner inner angle between successive blocks specifying linear interpolation (G01) or circular interpolation (G02, G03), and performs an exact stop automatically between the blocks if the angle is equal to or less than the parameter-set critical angle.

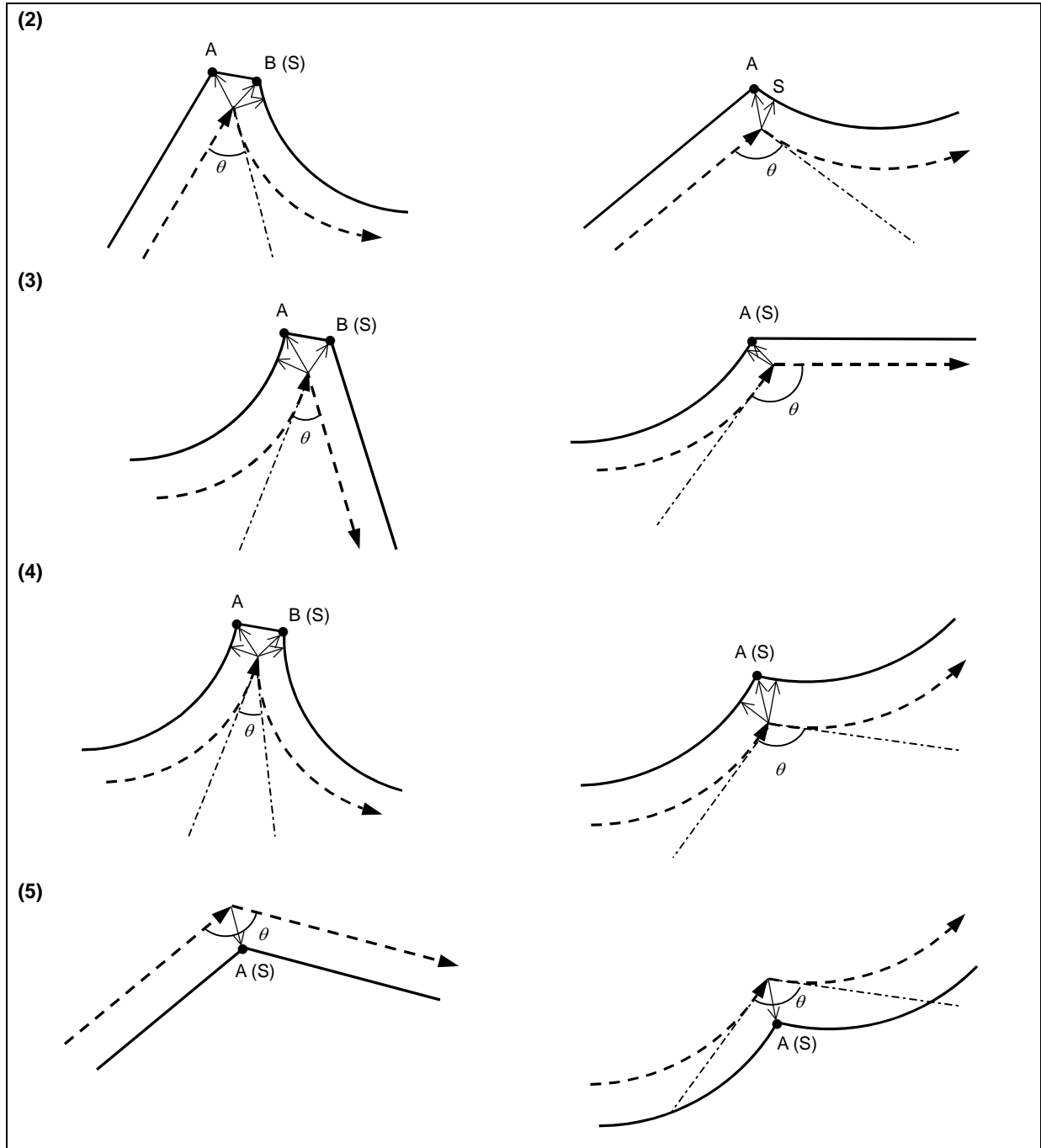
Explanation

When successive blocks specify linear interpolation (G01) or circular interpolation (G02, G03) in the cutting mode (G64), this function checks the corner inner angle between the blocks on the currently selected plane and determine whether to automatically perform an exact stop. In the figure below, an exact stop is automatically performed at the end of block A if the corner inner angle θ is equal to or less than the critical angle set in parameter No. 1496. This operation is equivalent to operation performed when an exact stop (G09) is specified in block A.



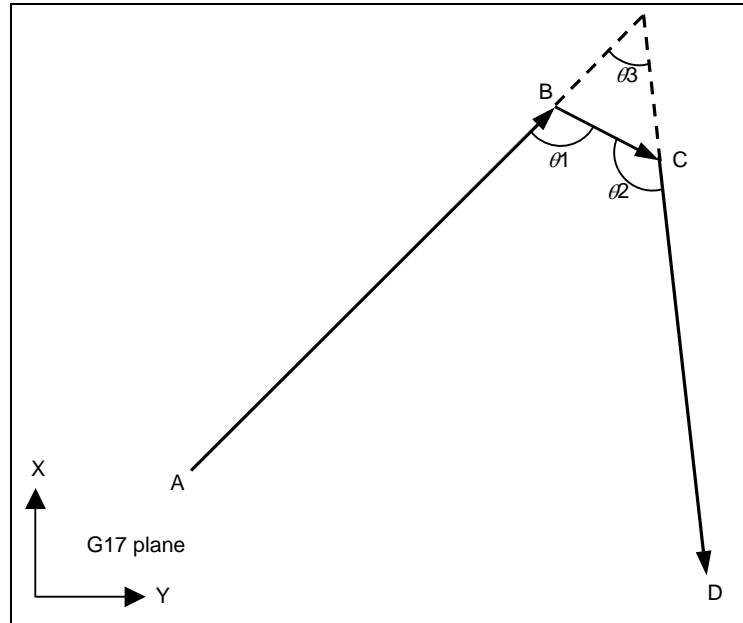
In the tool radius compensation mode as well, an automatic exact stop check is made for the corner inner angles of the programmed path. At this time, an exact stop is performed at points A and B in the figure below. (Point S represents a single-block stop point.)





If a small block specifies linear interpolation (G01) and also specifies the amounts of movement on the two axes on the currently selected plane that are less than the value set in parameter No. 1497, the exact stop check function can be disabled in the block.

When linear interpolation (G01) is performed for $A \rightarrow B \rightarrow C \rightarrow D$, an automatic exact stop check is made at point B based on angle θ_1 formed by lines AB and BC and at point C based on angle θ_2 formed by lines BC and CD. If the amount of movement on each of the two axes (X-axis and Y-axis in the figure below) in the block for BC is less than the value set in parameter No. 1497, an automatic exact stop check is made at point C based on angle θ_3 formed by lines AB and CD.

**NOTE**

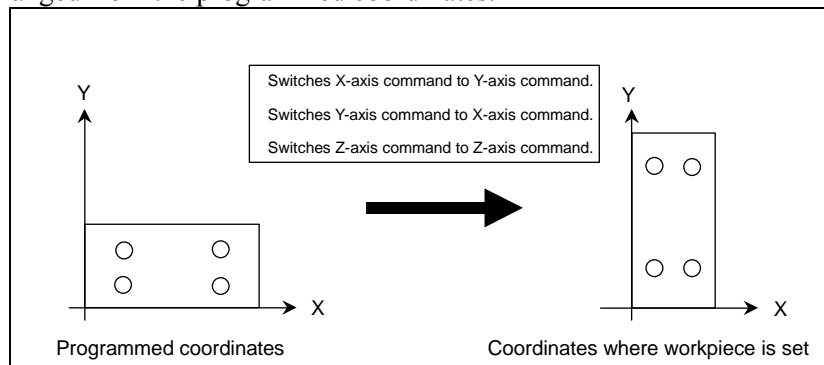
- 1 If successive small blocks specify a value less than the value set in parameter No. 1497, the automatic exact stop check function is disabled in the first small block and enabled in the next block, and this disable/enable cycle is repeated for the rest.
- 2 In the cutter compensation mode, the automatic exact stop check function cannot be disabled by parameter No. 1497.

9.4 AXIS SWITCHING

Overview

The machine axis to be actually used for movement by specifying X, Y, or Z in memory operation, DNC operation, or MDI operation can be switched by using the setting data (No. 10371) or the switch on the machine operator's panel.

As shown in the example below, this function is useful when the coordinates where a workpiece is actually set are changed from the programmed coordinates.



Explanation

▪ Axis switching No.

Six types of axis switching are possible as indicated below. The program addresses X, Y, and Z in each type of axis switching correspond to the actually specified axes x, y, and z.

Axis switching No.	Programmed address		
	X	Y	Z
0	x	y	z
1	x	z	y
2	y	x	z
3	y	z	x
4	z	x	y
5	z	y	x

The axis switching number 0 means that axis switching is not performed.

▪ Axis switching specification method

- (1) Specification based on the setting data
In the setting data (No. 10371), set an axis switching number (0 to 5).
- (2) Specification based on programming
Specify programmable parameter input (G10L52) and set an axis switching number (0 to 5) in the setting data (No. 10371).
- (3) Specification using the switch on the machine operator's panel
For the method of specification, refer to the manual issued by the machine tool builder.

The table below indicates the relationship between specification (including programming) based on setting data and specification using the switch on the machine operator's panel.

	Setting of setting data (No. 10371)	Setting of switch on machine operator's panel	Valid setting
1	0	0	Axis switching is disabled.
2	1 to 5	0	Setting data
3	0	1 to 5	Switch on machine operator's panel

	Setting of setting data (No. 10371)	Setting of switch on machine operator's panel	Valid setting
4	1 to 5	1 to 5	Switch on machine operator's panel

When axis switching is not performed, set both of the setting data and the switch on the machine operator's panel to 0.

▪ Cases where axis switching is disabled

Axis switching is disabled when the commands below are specified.

- (1) Manual operation
- (2) Move command for moving to a particular machine position, command related to machine coordinates, and command for coordinate system setting
 - (a) Automatic reference position return
G28, G30
 - (b) Floating reference position return
G30.1
 - (c) Return from reference position
G29
When bit 0 (RPC) of parameter No. 10370 is set to 1, axis switching is applied to the G29 command.
 - (d) Stored stroke check
G22, G23
 - (e) Coordinate system setting
G92, G68.2, G52
 - (f) Offset value setting
G10
 - (g) Dwell
G04
 - (h) Machine coordinate positioning
G53
- (3) Real time custom macro

Example

When a command for axis switching is specified, the command is executed by performing program interpretation in such a way that an address change is made according to the axis switching number.

Example) When axis switching No. = 4 is specified:

Specified command : G00 X100.0 Y200.0 Z300.0;

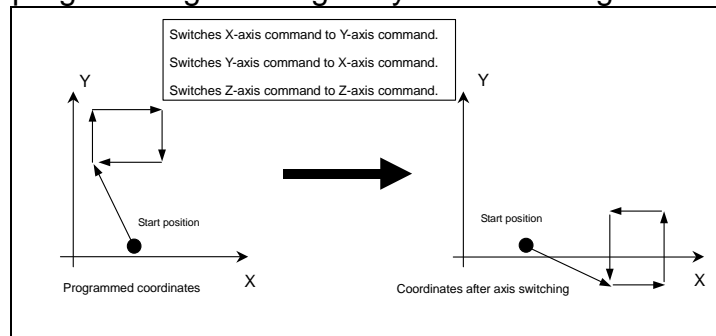
Axis switching command interpretation : G00 X200.0 Y300.0 Z100.0;

NOTE

- 1 When the same program is used by axis switching, for example, the amount and direction of movement are changed by axis switching, depending on the start position and whether incremental or absolute programming is used.

(Example)

When the same start position is used but the direction of movement in incremental programming is changed by axis switching



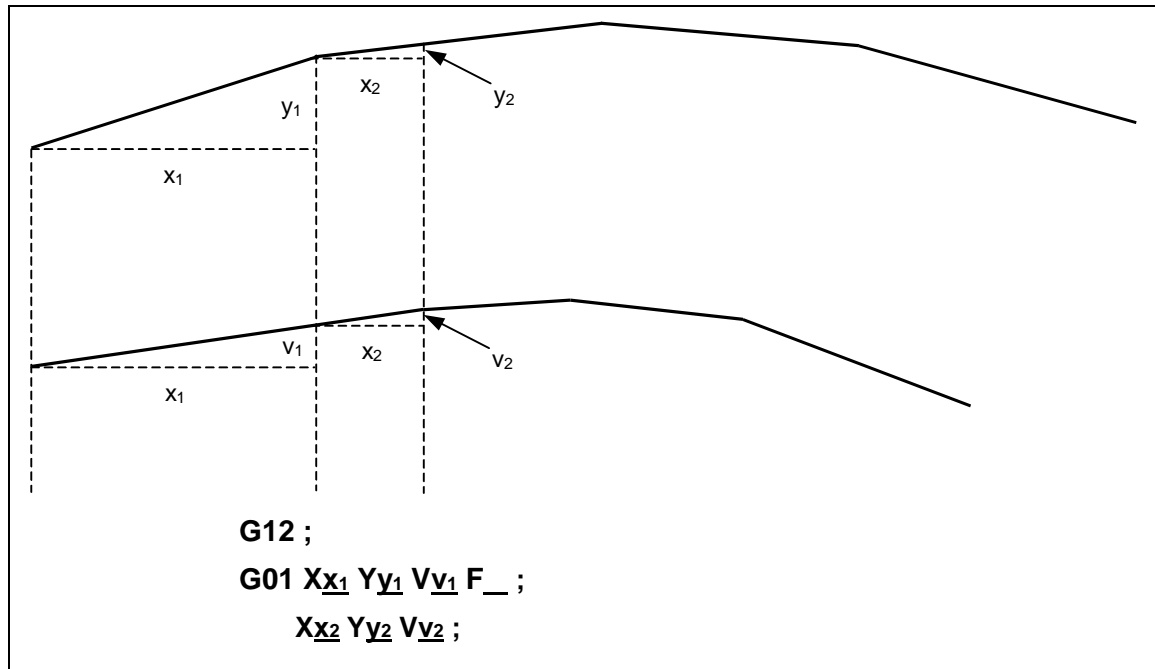
- 2 The hole machining axis in a canned cycle and tool length compensation and the axis to which tool length compensation is applied can be fixed at the Z-axis by setting bits 1 (TLB) and 0 (TLC) of parameter No. 5001 and bit 0 (FXY) of parameter No. 5101. Even in this case, however, if program address Z is changed to the machine axis x or y by axis switching, the hole machining axis and the axis to which tool length compensation is applied are also changed to x or y.

9.5 GENTLE CURVE CUTTING

Overview

If the V-axis is added as an axis parallel with the Y-axis, the V-axis is treated as an axis parallel with the Y-axis in the gentle curve cutting cancel mode (G13), and a command for the Y-axis alone is assumed to be specified for the V-axis as well and enables movement also on the V-axis.

In the gentle curve cutting mode (G12), a command for the Y-axis is valid for the Y-axis alone, and a command for the V-axis is valid for the V-axis alone.



NOTE

Be sure to specify G12 or G13 in a block specifying no other commands.

Explanation

If the V-axis is added as an axis parallel with the Y-axis, the V-axis is treated as an axis parallel with the Y-axis in the gentle curve cutting cancel mode (G13), and a command for the Y-axis alone is assumed to be specified for the V-axis as well and enables movement also on the V-axis. This means that the setting of an axis parallel with the Y-axis is equivalent to the specification of the same axis name as the Y-axis. However, the V-axis is treated as a slave of the Y-axis, so that no command may be specified for the V-axis. If a command is specified for the V-axis, the alarm PS009, "IMPROPER NC-ADDRESS" is issued. For details of a parallel axis, see the item of parallel axis control.

In the gentle curve cutting mode (G12), a command for the Y-axis is valid for the Y-axis alone, and a command for the V-axis is valid for the V-axis alone.

G13;	Gentle curve cutting mode cancel		
G12;	Gentle curve cutting mode		
<Sample program>		Y move amount	V move amount
G13;			
G91 Y1000.0;	—————>	1000	1000
Y500.0;	—————>	500	500
G12;			
Y35.0V50.0;	—————>	35	50
Y40.0V120.0;	—————>	40	120

NOTE

- 1 Be sure to specify G13 or G12 in a block specifying no other commands.
- 2 Ensure that the axis number of the basic axis is smaller than the axis number of a parallel axis. Otherwise, no operation is enabled on the V-axis in the G13 mode.
- 3 If a command is specified for the V-axis in the G13 mode, the alarm PS0009, "IMPROPER NC-ADDRESS" is issued.
- 4 In the G12 mode, do not specify circular interpolation (G02/G03).
- 5 In the G12 mode, be sure to set the G40 (cutter compensation cancel) mode.

Parameter

1022

Setting of each axis in the basic coordinate system

[Input type] Parameter input

[Data type] Byte axis

[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) and a 3-dimensional tool compensation space (XpYpZp), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.

A basic axis (X, Y, or Z) can be specified only for one control axis.

Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotation axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

9.6 GENTLE NORMAL DIRECTION CONTROL

Overview

Gentle normal direction control enables movement on a rotary axis (C-axis) under normal direction control simultaneously with movement on a linear axis (X/Y-axis). This makes the function applicable to a gas cutting machine.

Explanation

Gentle normal direction control executes C-axis rotation and X/Y-axis movement at the same time when the C-axis rotation angle generated by normal direction control is less than the value of parameter No. 5485.

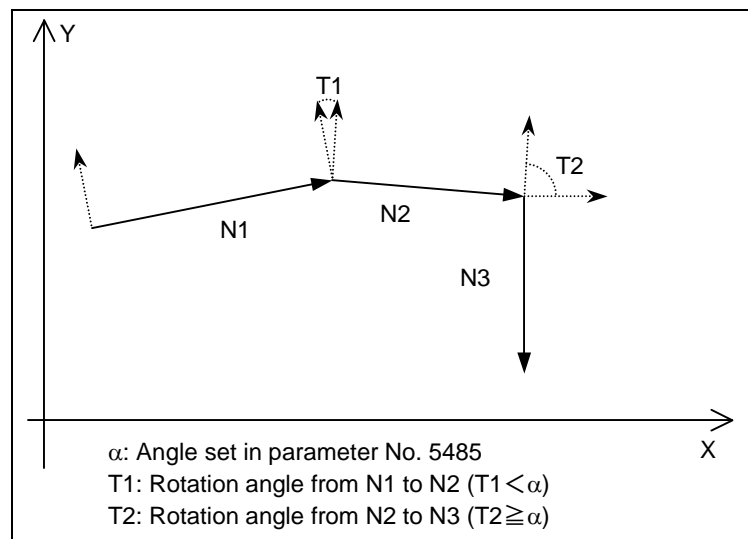


Fig. 9.6 (a)

As shown in the Fig. 9.6 (a), when movement in the order of N1, N2, and N3 is specified:

- 1) From N1 to N2, because T1 is less than α , the C-axis rotates angle T1 together with the movement of the X- and Y-axes indicated by N2.
In this case, the movement distance of the C-axis is not included in calculation and the speed on the XY plane is specified by F. (normal direction control)
- 2) From N2 to N3, because T2 is equal to or greater than α , the C-axis rotates angle T2 before movement of the X- and Y-axes in N3.(gentle normal direction control)

When the angle formed by a line and an arc or the angle formed between arcs is less than the angle set in parameter No. 5485, the C-axis rotates with the movement of the X- and Y-axes as in the angle formed between lines.

NOTE

- 1 When C-axis rotation included in an X/Y-axis move block is performed at the same time, actual cutting feedrate display in linear interpolation (cutting) mode shows the feedrate obtained from the following equation:

$$F' = F \times \sqrt{(X^2+Y^2+C^2)} / \sqrt{(X^2+Y^2)}$$
- 2 The C-axis moves in response to X/Y-axis movement. If the C-axis feedrate exceeds the maximum cutting speed or rapid traverse rate, however, the C-axis feedrate is clamped at the maximum cutting speed or rapid traverse rate. The X/Y-axis feedrate may therefore be different from the specified feedrate.

Limitation

- (1) The normal direction control function is required for this function to be used.
- (2) If the number of controlled axes including the C-axis exceeds the maximum number of simultaneously controlled axes, the option for "simultaneously controlled axes expansion" is required.
- (3) F1 digit code cannot be specified.
- (4) Restrictions on normal direction control applies to this function as well.

9.6.1 Linear Distance Setting

Explanation

By setting parameter No. 5486 to a travel distance for a block to move until the rotation of a normal direction axis is completed, the normal direction axis is controlled to start rotation simultaneously with the command block and terminates it upon completion of movement over the parameter-specified distance.

This function is valid when the conditions for executing gentle normal direction control for rotating the normal direction axis simultaneously with the command block are satisfied.

Assuming parameter No. 5486 specifying "a", the normal direction axis rotates an angle of T1 when the N2 block having started execution reaches distance "a".

When the travel distance of the N2 block is less than or equal to the setting of parameter No. 5486, the normal direction axis moves that travel distance.

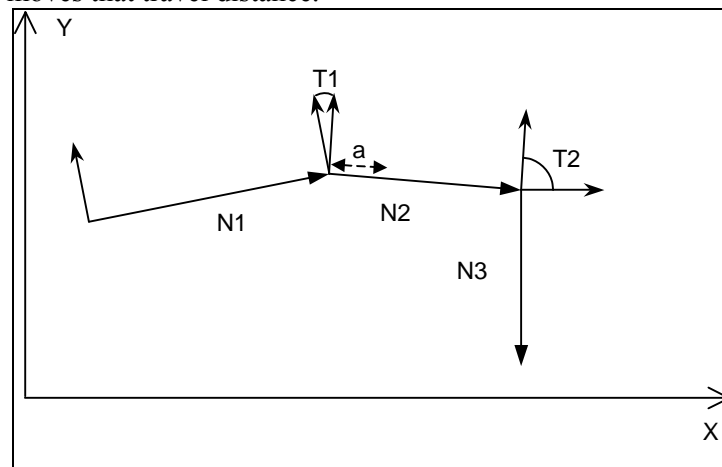


Fig. 9.6.1 (a)

• Conditions

All of the following conditions must be satisfied for execution of gentle normal direction control:

- (1) Gentle normal direction control is valid. (Bit 0 of parameter No. 5484 is set to 1.)
- (2) The travel distance of the command block is equal to or more than the setting of parameter No. 5483 maintaining the normal direction angle of the previous block.
- (3) The rotation angle calculated under normal direction control is more than the setting of parameter No. 5482 for rotation of the normal direction control axis.
- (4) The rotation angle calculated under normal direction control is less than the setting of parameter No. 5485 for rotation under gentle normal direction control.

This function rotates the normal direction axis according to the travel distance specified by parameter No. 5486 when the travel distance of the command block is more than the setting of parameter No. 5486 while the conditions for execution of gentle normal direction control are satisfied.

• Limitation

- This function is invalid when the command block contains an arc command, in which case a regular type of gentle normal direction control is exercised.
- This function cannot be used in inverse time feed mode.


NOTE

- 1 The normal direction axis moves in response to X/Y-axis movement. If the feedrate of the normal direction axis exceeds the maximum cutting speed or rapid traverse rate, however, the feedrate of the normal direction axis is clamped at the maximum cutting speed or rapid traverse rate. The X/Y-axis feedrate may therefore be lower than the specified feedrate.
- 2 When this function is used in a positioning command block, in-position check is performed before the command block moves the remaining distance after the rotation of the normal direction axis is completed.


III. OPERATION

1 SETTING AND DISPLAYING DATA

Chapter 1, "SETTING AND DISPLAYING DATA", consists of the following sections:

1.1	SCREENS DISPLAYED BY FUNCTION KEY 	337
1.1.1	Setting and Displaying the Tool Compensation Value	337
1.1.2	Tool Length Measurement	342
1.1.3	Tool Length/Workpiece Origin Measurement	345
1.1.4	Setting and Displaying the Rotary Table Dynamic Fixture Offset	364
1.1.5	Input of Tool Offset Value Measured B.....	367
1.1.6	Spindle Unit Compensation, Nutating Rotary Head Tool Length Compensation.....	367

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 length/workpiece origin measurement
4. Rotary table dynamic fixture offset
5. Tool offset value
6. Spindle unit compensation, nutating rotary head tool length compensation

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.

OFFSET		00123 N00000			
NO.	DATA	NO.	DATA	NO.	DATA
001	0.000	018	0.000		
002	0.000	019	0.000		
003	0.000	020	0.000		
004	0.000	021	0.000		
005	0.000	022	0.000		
006	0.000	023	0.000		
007	0.000	024	0.000		
008	0.000	025	0.000		
009	0.000	026	0.000		
010	0.000	027	0.000		
011	0.000	028	0.000		
012	0.000	029	0.000		
013	0.000	030	0.000		
014	0.000	031	0.000		
015	0.000	032	0.000		
016	0.000				
017	0.000				

RELATIVE	
X	0.000
Y	0.000
Z	0.000
B	0.000
C	0.000

ABSOLUTE	
X	0.000
Y	0.000
Z	0.000
B	0.000
C	0.000

MACHINE	
X	0.000
Y	0.000
Z	0.000
B	0.000
C	0.000

A>_

<	NO. SRH	INP. C.	+INPUT	INPUT	MEM *****	12:00:00			
			ERASE	READ	PUNCH				

Fig. 1.1.1 (a) Tool compensation memory A (10.4-inch display unit)

OFFSET		00123 N00000			
NO.	GEOM	WEAR			
001	0.000	0.000			
002	0.000	0.000			
003	0.000	0.000			
004	0.000	0.000			
005	0.000	0.000			
006	0.000	0.000			
007	0.000	0.000			
008	0.000	0.000			
009	0.000	0.000			
010	0.000	0.000			
011	0.000	0.000			
012	0.000	0.000			
013	0.000	0.000			
014	0.000	0.000			
015	0.000	0.000			
016	0.000	0.000			
017	0.000	0.000			

RELATIVE	
X	0.000
Y	0.000
Z	0.000
B	0.000
C	0.000

ABSOLUTE	
X	0.000
Y	0.000
Z	0.000
B	0.000
C	0.000

MACHINE	
X	0.000
Y	0.000
Z	0.000
B	0.000
C	0.000

A>_

<	NO. SRH	INP. C.	+INPUT	INPUT	MEM *****	12:00:00			
			ERASE	READ	PUNCH				

Fig. 1.1.1 (b) Tool compensation memory B (10.4-inch display unit)


OFFSET					00123 N00000		
NO.	<LENGTH>		<RADIUS>		RELATIVE		
	GEOM	WEAR	GEOM	WEAR	X	Y	Z
001	0.000	0.000	0.000	0.000	X	0.000	0.000
002	0.000	0.000	0.000	0.000	Y	0.000	0.000
003	0.000	0.000	0.000	0.000	Z	0.000	0.000
004	0.000	0.000	0.000	0.000	B	0.000	0.000
005	0.000	0.000	0.000	0.000	C	0.000	0.000
006	0.000	0.000	0.000	0.000	ABSOLUTE		
007	0.000	0.000	0.000	0.000	X	0.000	0.000
008	0.000	0.000	0.000	0.000	Y	0.000	0.000
009	0.000	0.000	0.000	0.000	Z	0.000	0.000
010	0.000	0.000	0.000	0.000	B	0.000	0.000
011	0.000	0.000	0.000	0.000	C	0.000	0.000
012	0.000	0.000	0.000	0.000	MACHINE		
013	0.000	0.000	0.000	0.000	X	0.000	0.000
014	0.000	0.000	0.000	0.000	Y	0.000	0.000
015	0.000	0.000	0.000	0.000	Z	0.000	0.000
016	0.000	0.000	0.000	0.000	B	0.000	0.000
					C	0.000	0.000

A>_

MEM ***** 12:00:00

< NO.SRH INP.C. +INPUT INPUT ERASE READ PUNCH

Fig. 1.1.1 (c) Tool compensation memory C (10.4-inch display unit)

If the cutting point command option is enabled, press the MDI key  several times on the tool compensation memory C screen, the corner R offset setting screen is displayed.

OFFSET			00123 N00000		
NO.	<CNR R>		RELATIVE		
	GEOM	WEAR	X	Y	Z
001	0.000	0.000	X	0.000	0.000
002	0.000	0.000	Y	0.000	0.000
003	0.000	0.000	Z	0.000	0.000
004	0.000	0.000	B	0.000	0.000
005	0.000	0.000	C	0.000	0.000
006	0.000	0.000	ABSOLUTE		
007	0.000	0.000	X	0.000	0.000
008	0.000	0.000	Y	0.000	0.000
009	0.000	0.000	Z	0.000	0.000
010	0.000	0.000	B	0.000	0.000
011	0.000	0.000	C	0.000	0.000
012	0.000	0.000	MACHINE		
013	0.000	0.000	X	0.000	0.000
014	0.000	0.000	Y	0.000	0.000
015	0.000	0.000	Z	0.000	0.000
			B	0.000	0.000
			C	0.000	0.000

A>_

MEM ***** 12:00:00


< NO.SRH INP.C. +INPUT INPUT ERASE READ PUNCH

Fig. 1.1.1 (d) Screen used for the cutting point command (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 the chapter selection key, in Fig.1.1.1 (e) Tool compensation memory A (15-inch display unit), vertical 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.

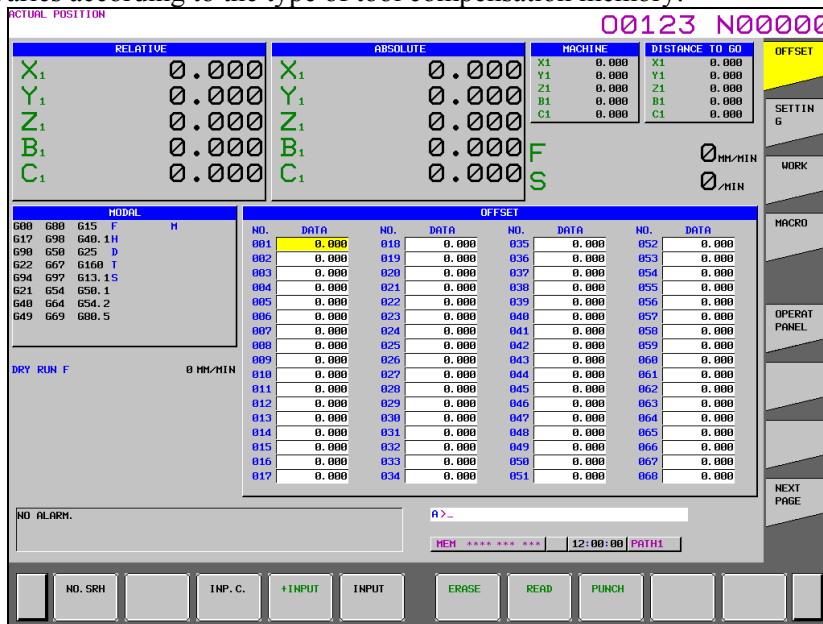


Fig.1.1.1 (e) Tool compensation memory A (15-inch display unit)

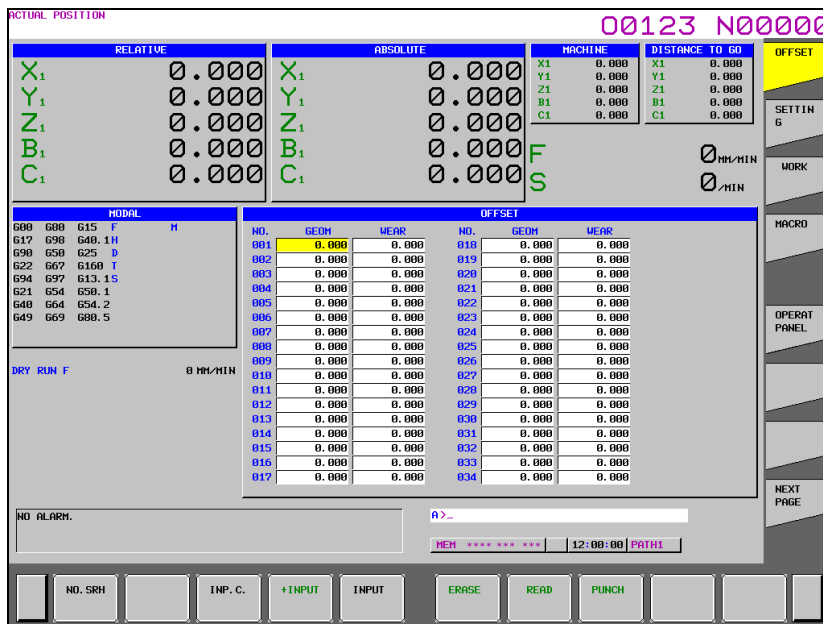


Fig. 1.1.1 (f) Tool compensation memory B (15-inch display unit)

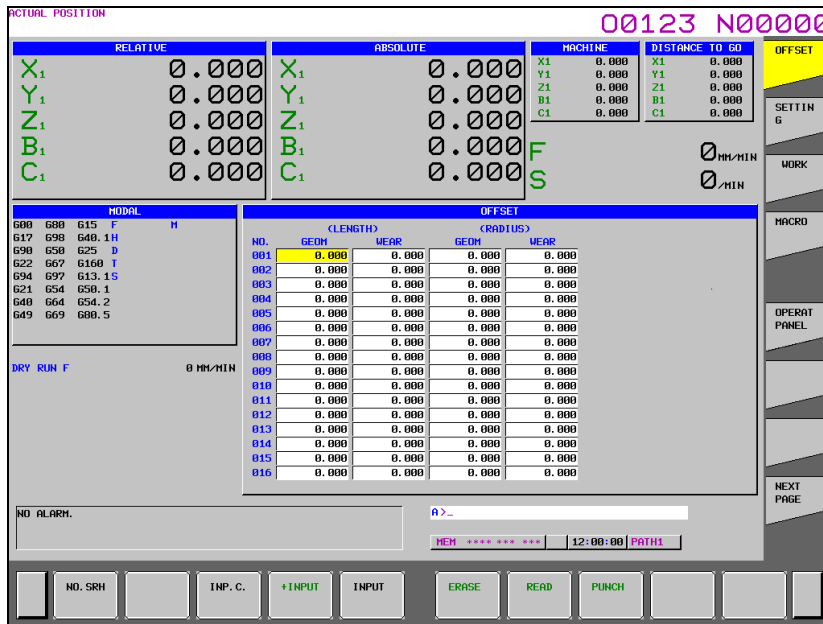


Fig. 1.1.1 (g) Tool compensation memory C (15-inch display unit)

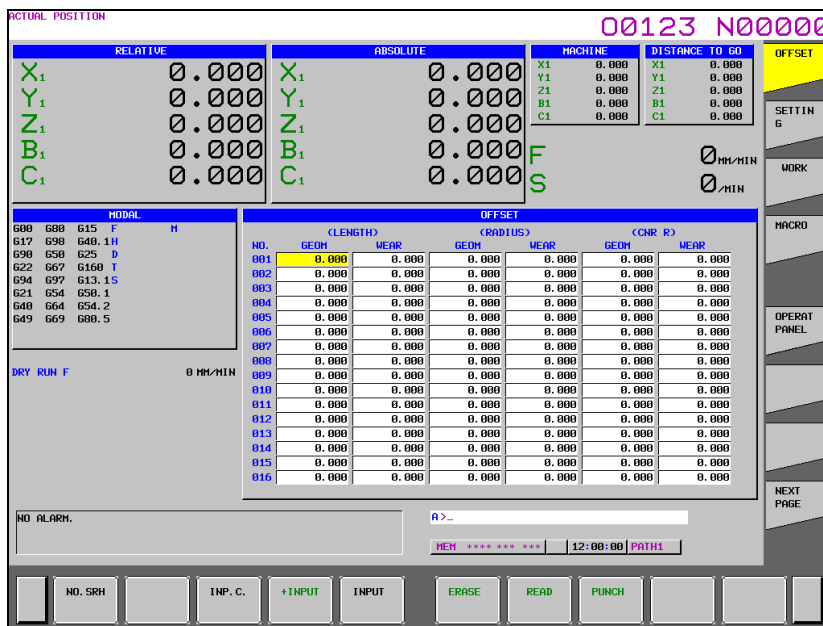


Fig. 1.1.1 (h) Screen used for the cutting point command (15-inch display unit)

- 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 III-8 in OPERATOR'S MANUAL (Common to T/M). 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, B, 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 B

D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated differently.

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.

Procedure for tool length measurement (for 8.4/10.4-inch display unit)

Procedure

- 1 Use manual operation to move the reference tool until it touches the specified position on the machine (or workpiece.)
- 2 Press function key  several times until the current position display screen with relative coordinates is displayed.

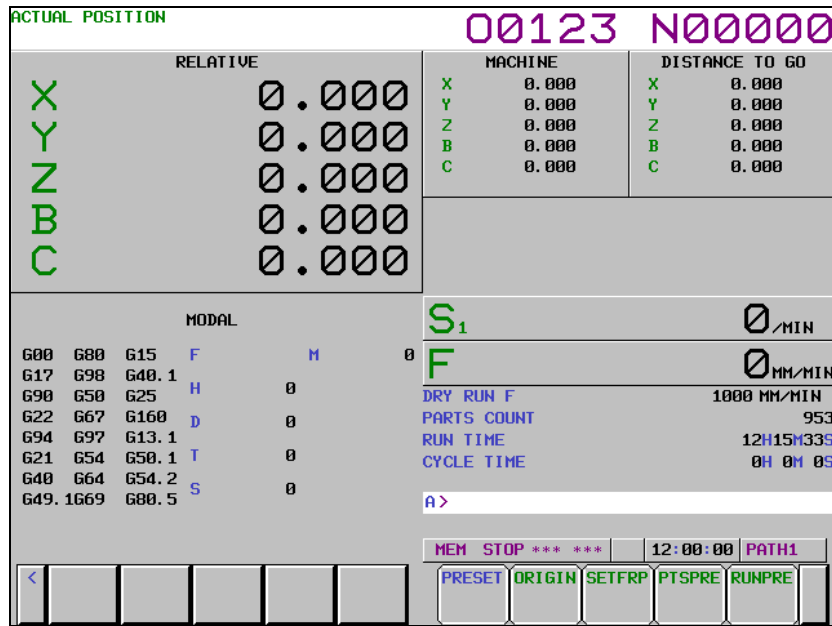





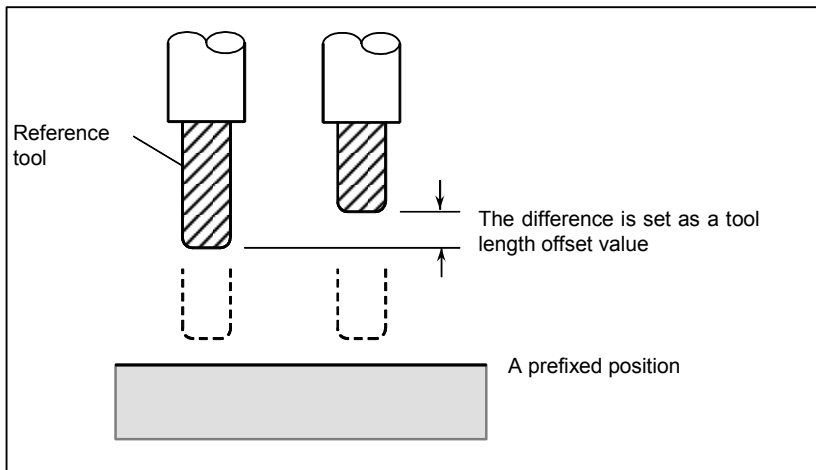



Fig. 1.1.2 (a) Current position display screen (10.4-inch display unit)

- 3 Reset the relative coordinate for the Z-axis to 0.
- 4 Press function key  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 . If either  or  key is depressed instead of  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 (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  to display the overall position display screen.

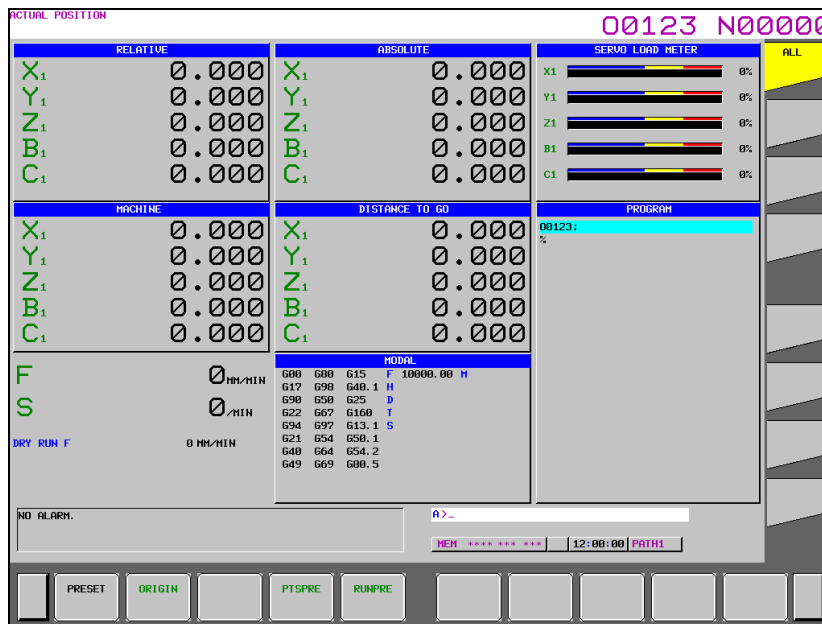





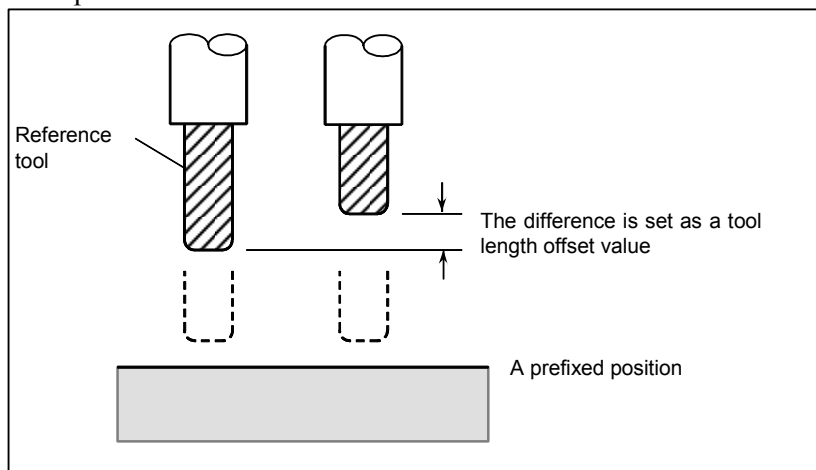


Fig. 1.1.2 (b) Current position display screen (15-inch display unit)

- 3 Reset the relative coordinate for the Z-axis to 0.
- 4 Press function key  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 . If either  or  key is depressed instead of  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.



1.1.3 Tool Length/Workpiece Origin Measurement

To enable measurement of the tool length, the following functions are supported: automatic measurement of the tool length by using a program command (G37) (automatic tool length measurement, described in Section II-6.2) and measurement of the tool length by manually moving the tool until it touches a reference position, such as the workpiece top surface (tool length measurement, described in Subsection III-1.1.2). In addition to these functions, tool length/workpiece origin measurement is supported to simplify the tool length measurement procedure, thus facilitating and reducing the time required for machining setup. This function also facilitates the measurement of the workpiece origin offsets. This function allows the operator to specify T/M code commands or reference position return, by means of a manual numeric command, while the tool length compensation measurement screen is displayed.

Procedure for measuring the tool length compensation value (for 8.4/10.4-inch display unit)

Procedure

The tool length compensation value can be measured by manually moving the tool until it touches the workpiece or a reference block. For details of this operation, refer to the manual supplied by the machine tool builder.

- 1 Move the tool to the tool change position by means of manual reference position return, for example.
- 2 Press mode selection switch HANDLE or JOG.
- 3 Set the tool offset value measurement mode switch on the machine operator's panel to ON. The tool length compensation measurement screen appears and "OFST" blinks in the status display at the bottom of the screen.

The tool length compensation measurement screen varies slightly depending on whether tool length compensation memory A, B (geometry compensation and wear compensation are treated differently), or C (geometry compensation and wear compensation are treated differently, and cutter compensation and tool length compensation are treated differently) is used.

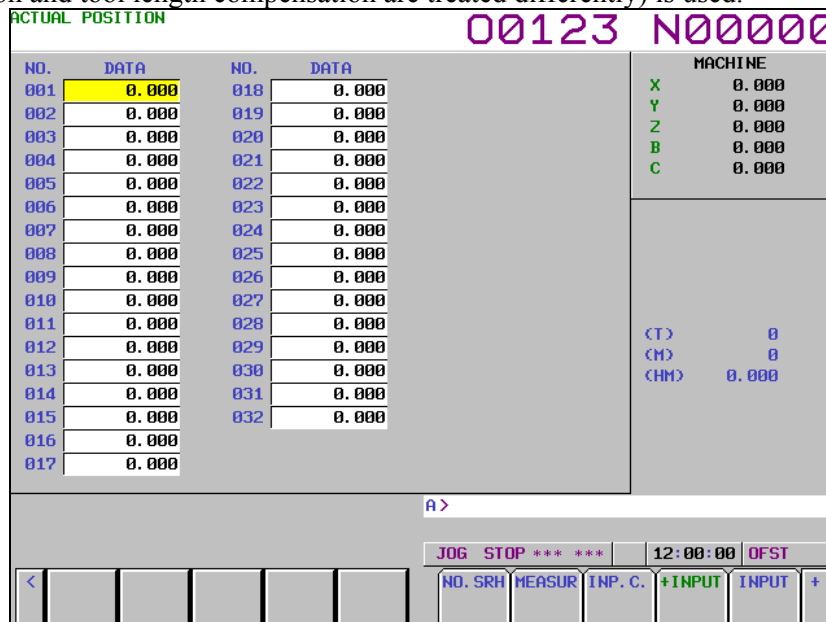


Fig. 1.1.3 (a) Tool length compensation measurement screen for tool compensation memory A (10.4-inch display unit)

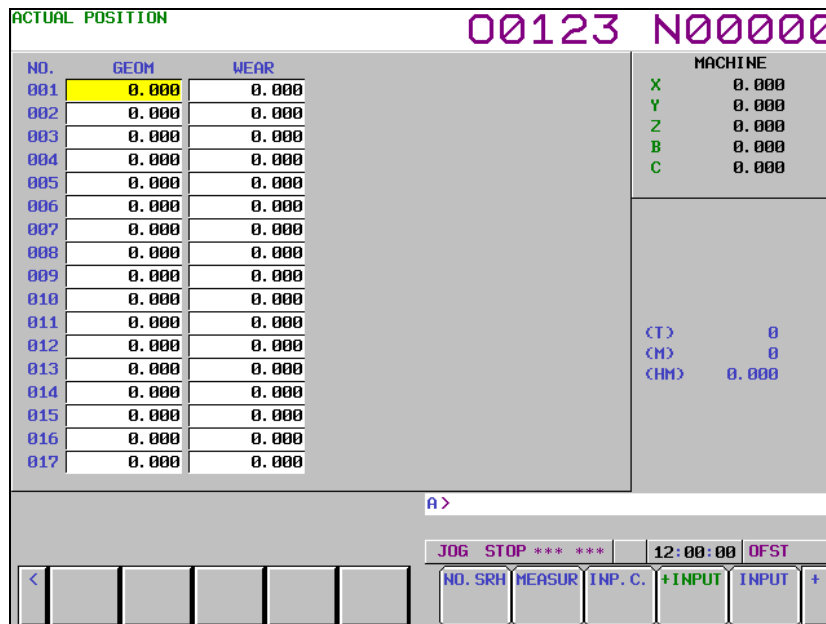


Fig. 1.1.3 (b) Tool length compensation measurement screen for tool compensation memory B (10.4-inch display unit)

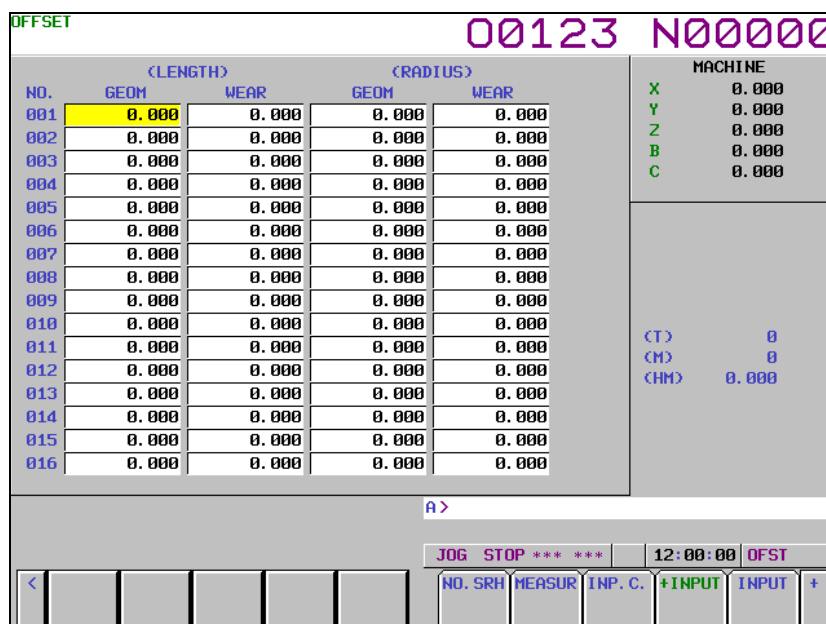



Fig. 1.1.3 (c) Tool length compensation measurement screen for tool compensation memory C (10.4-inch display unit)

NOTE

Pressing the  key resets the displayed T and M addresses to 0. Once MEM or MDI mode has been selected, however, the modal T and M codes are displayed.

- 4 Use the numeric keys to enter the distance from the base measurement surface to the measurement surface, then press soft key [HM INPUT] to set the distance. For details of the measurement surface and base measurement surface, see Explanations, below.

- 5 Select the tool for which the tool length compensation value is to be measured.
While "OFST" is blinking at the bottom of the tool length compensation measurement screen, a T code or M code can be specified in manual handle feed or jog feed mode (manual numeric command). First, enter Tttt (where tttt is a T code number), then press the cycle start button on the machine operator's panel or MDI unit.
The Tttt command is executed, thus selecting the tool to be measured. Then, usually, enter the M06 command to move the tool to the spindle position. Once the tool for which the tool length compensation is to be measured has been selected at the spindle position, position the cursor to the tool offset number with which the tool length compensation for the selected tool is to be stored. The positioning of the cursor to the offset number is usually done by the operator. Some machines, however, automatically position the cursor to an appropriate tool offset number upon the completion of tool selection, if bit 5 (QNI) of parameter No. 5005 is set to 1.
- 6 Perform manual handle feed or jog feed to move the tool until it touches the measurement surface of the workpiece or reference block.
- 7 Press soft key [MEASUR]. The tool length compensation is stored in the tool compensation memory. If tool compensation memory B or C is being used, the tool length compensation is set as the tool geometry value, while 0 is set as the tool wear offset. The cursor remains positioned to the set tool offset number. To automatically advance the cursor to the next tool offset number upon the completion of the setting an offset, press soft key [MEASUR+], instead of [MEASUR].
- 8 Once the tool length compensation has been set, the tool is automatically moved to the tool change position.
- 9 This completes tool length compensation measurement for a single tool. To measure the tool length compensations of other tools, repeat steps 5 to 8.
- 10 Once the tool length compensations of all tools have been measured, set the tool offset measurement mode switch on the machine operator's panel to OFF. The "OFST" blinking indication is cleared from the bottom of the screen.

Procedure for measuring the tool length compensation value (for 15/19-inch display unit)

Procedure

The tool length compensation value can be measured by manually moving the tool until it touches the workpiece or a reference block. For details of this operation, refer to the manual supplied by the machine tool builder.

- 1 Move the tool to the tool change position by means of manual reference position return, for example.
- 2 Press mode selection switch HANDLE or JOG.
- 3 Set the tool offset value measurement mode switch on the machine operator's panel to ON. The tool length compensation measurement screen appears and "OFST" blinks in the status display at the bottom of the screen.

The tool length compensation measurement screen varies slightly depending on whether tool length compensation memory A, B (geometry compensation and wear compensation are treated differently), or C (geometry compensation and wear compensation are treated differently, and cutter compensation and tool length compensation are treated differently) is used.

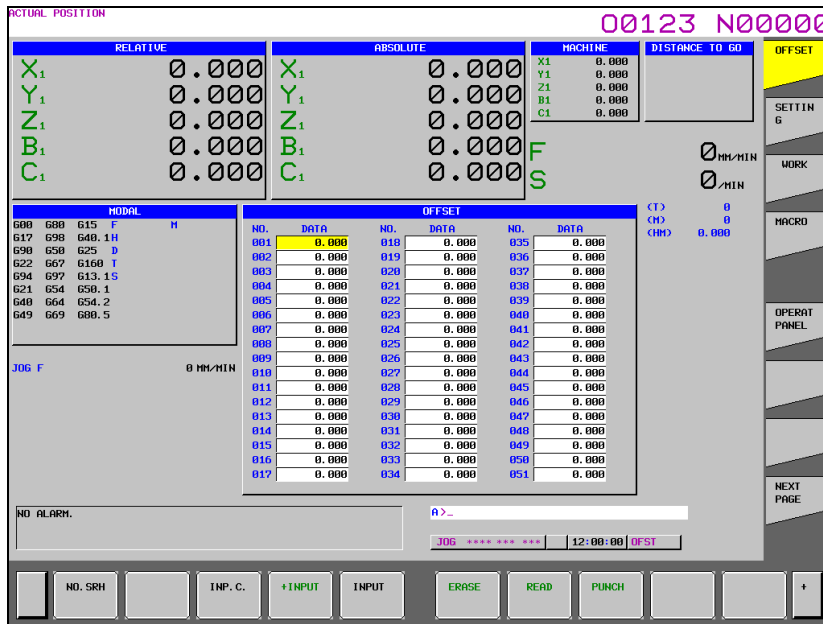


Fig. 1.1.3 (d) Tool length compensation measurement screen for tool compensation memory A (15-inch display unit)

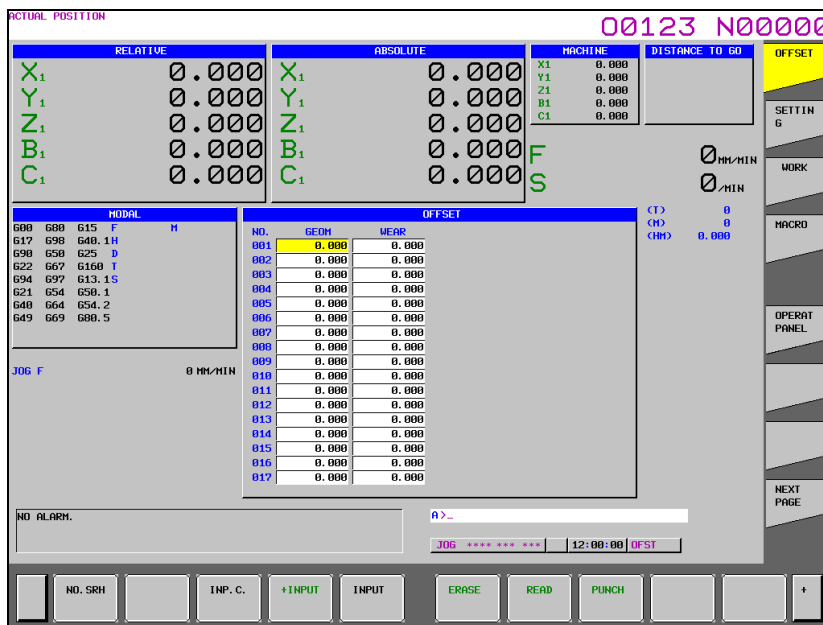


Fig. 1.1.3 (e) Tool length compensation measurement screen for tool compensation memory B (15-inch display unit)

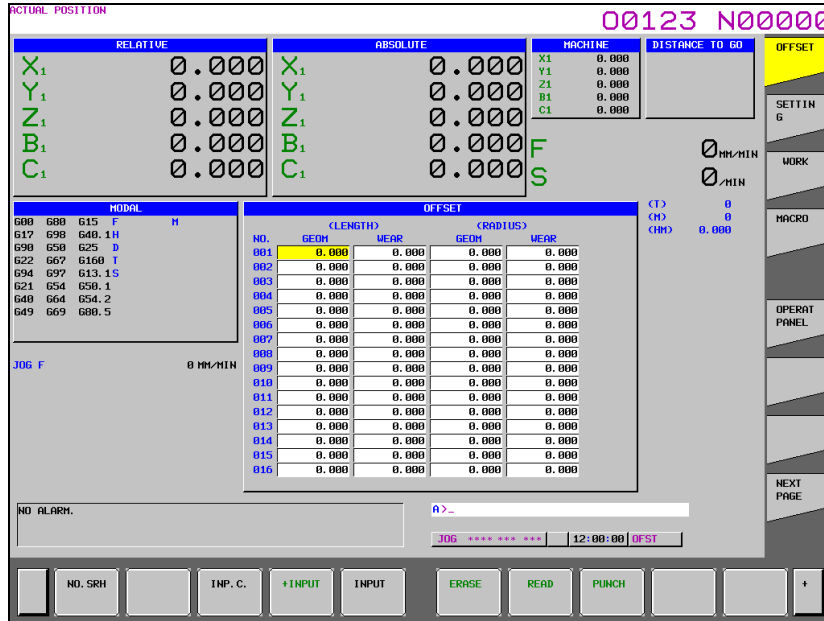



Fig. 1.1.3 (f) Tool length compensation measurement screen for tool compensation memory C (15-inch display unit)

NOTE

Pressing the  key resets the displayed T and M addresses to 0. Once MEM or MDI mode has been selected, however, the modal T and M codes are displayed.

- 4 Use the numeric keys to enter the distance from the base measurement surface to the measurement surface, then press horizontal soft key [HM INPUT] to set the distance. For details of the measurement surface and base measurement surface, see Explanations, below.
- 5 Select the tool for which the tool length compensation value is to be measured. While "OFST" is blinking at the bottom of the tool length compensation measurement screen, a T code or M code can be specified in manual handle feed or jog feed mode (manual numeric command). First, enter Tttt (where tttt is a T code number), then press the cycle start button on the machine operator's panel or MDI unit. The Tttt command is executed, thus selecting the tool to be measured. Then, usually, enter the M06 command to move the tool to the spindle position. Once the tool for which the tool length compensation is to be measured has been selected at the spindle position, position the cursor to the tool offset number with which the tool length compensation for the selected tool is to be stored. The positioning of the cursor to the offset number is usually done by the operator. Some machines, however, automatically position the cursor to an appropriate tool offset number upon the completion of tool selection, if bit 5 (QNI) of parameter No. 5005 is set to 1.
- 6 Perform manual handle feed or jog feed to move the tool until it touches the measurement surface of the workpiece or reference block.
- 7 Press horizontal soft key [MEASUR]. The tool length compensation is stored in the tool compensation memory. If tool compensation memory B or C is being used, the tool length compensation is set as the tool geometry value, while 0 is set as the tool wear offset. The cursor remains positioned to the set tool offset number. To automatically advance the cursor to the next tool offset number upon the completion of the setting an offset, press horizontal soft key [MEASUR+], instead of [MEASUR].
- 8 Once the tool length compensation has been set, the tool is automatically moved to the tool change position.
- 9 This completes tool length compensation measurement for a single tool. To measure the tool length compensations of other tools, repeat steps 5 to 8.

- 10 Once the tool length compensations of all tools have been measured, set the tool offset measurement mode switch on the machine operator's panel to OFF. The "OFST" blinking indication is cleared from the bottom of the screen.

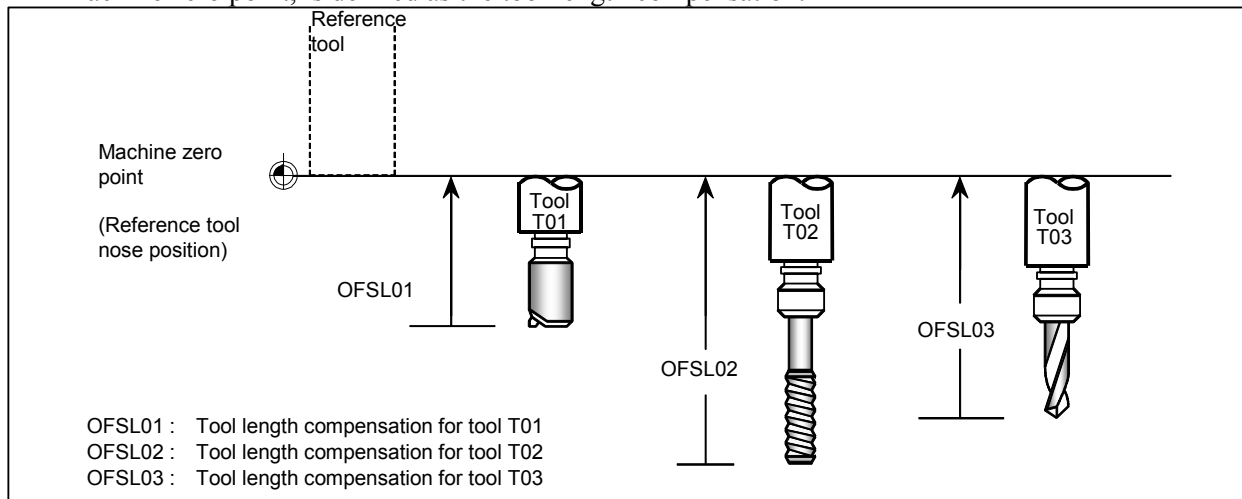
Explanation

- Definition of tool length compensation value

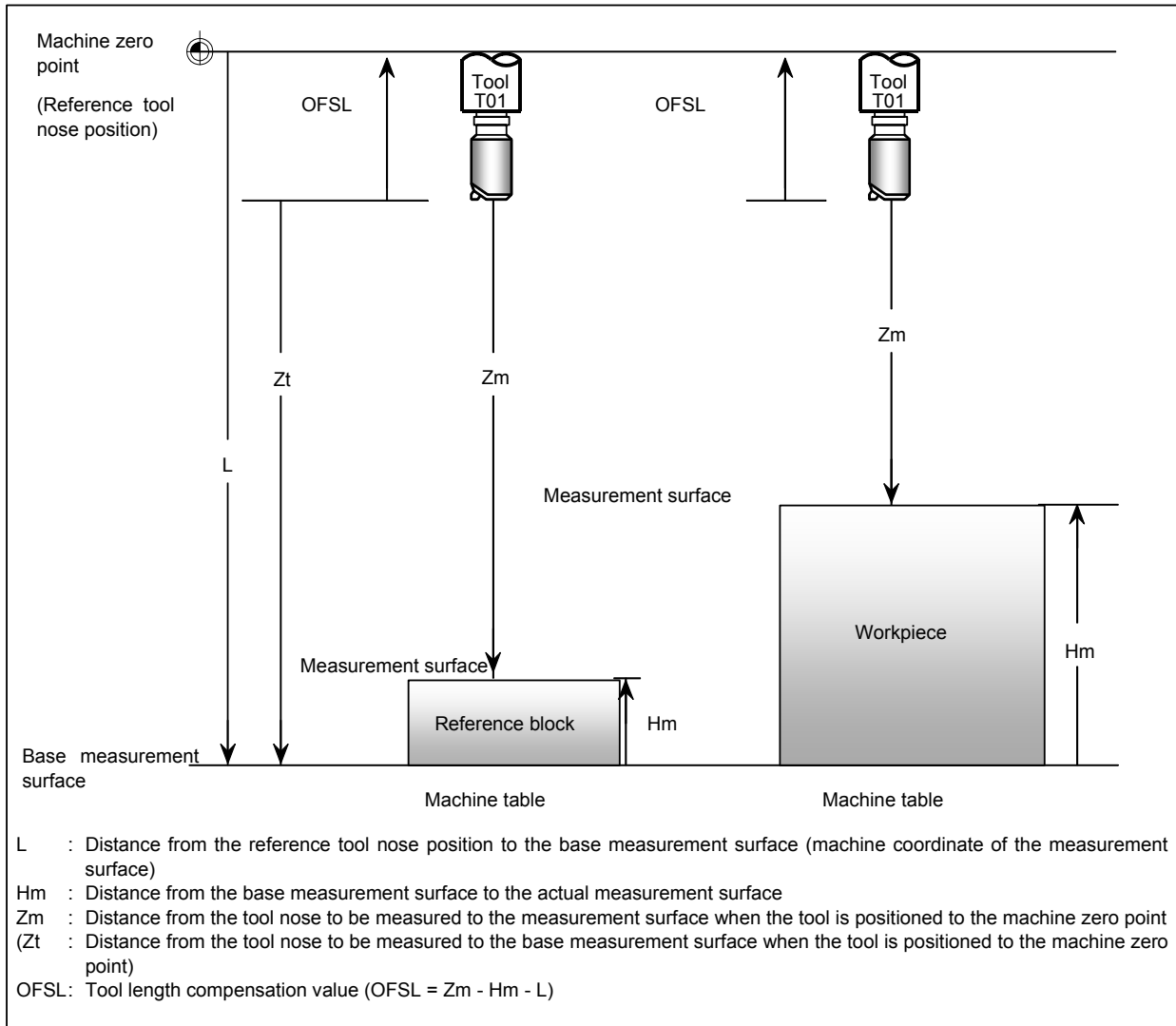
In general, the tool length compensation value can be defined in either of the following two ways. Both methods are based on the same concept: The difference between the tool nose position of the tool and that of a reference tool is used as the tool offset.

(1) Definition 1

The first method involves using the actual tool length as the tool length compensation. In this case, the reference tool is an imaginary tool which has its tool nose at the machine zero point when the machine is positioned to the Z-axis machine zero point. The difference between the tool nose position of the tool to be measured and that of the reference tool, that is, the distance along the Z-axis from the machine zero point to the tool nose when the machine is positioned to the Z-axis machine zero point, is defined as the tool length compensation.



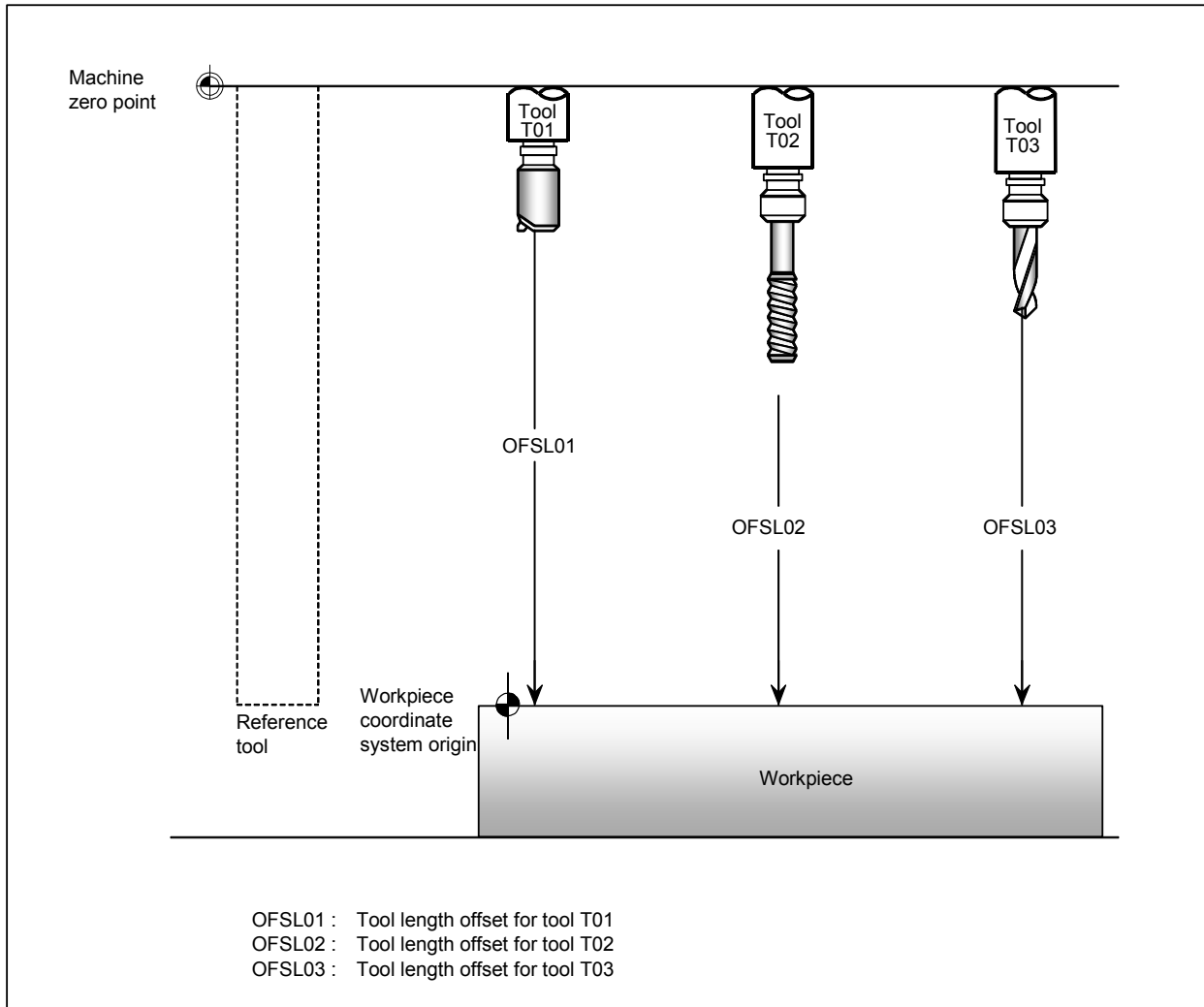
Also, with this function, the tool is manually moved by means of jog feed until its tool nose touches the top surface of the workpiece or reference block. This surface is called the measurement surface. Assume that the top surface of the machine's table is set as the measurement surface, although this is actually not allowed because the machine would be damaged. In such a case, distance L from the machine zero point to the machine table top surface is specific to that machine. Set distance L in a parameter No. 5022. Assume Z_t to be the machine coordinate of the tool at the position where it would touch the machine table top surface if that surface were set as the measurement surface. The tool length compensation (OFSL) can then be easily calculated from L and Z_t . Because the machine table top surface cannot actually be used as the measurement surface, however, that surface is defined as the base measurement surface and the distance from the base measurement surface to the actual measurement surface, that is, the height of the workpiece or reference block (H_m) must be set. The tool length compensation value (OFSL) can thus be obtained from the formula shown below.



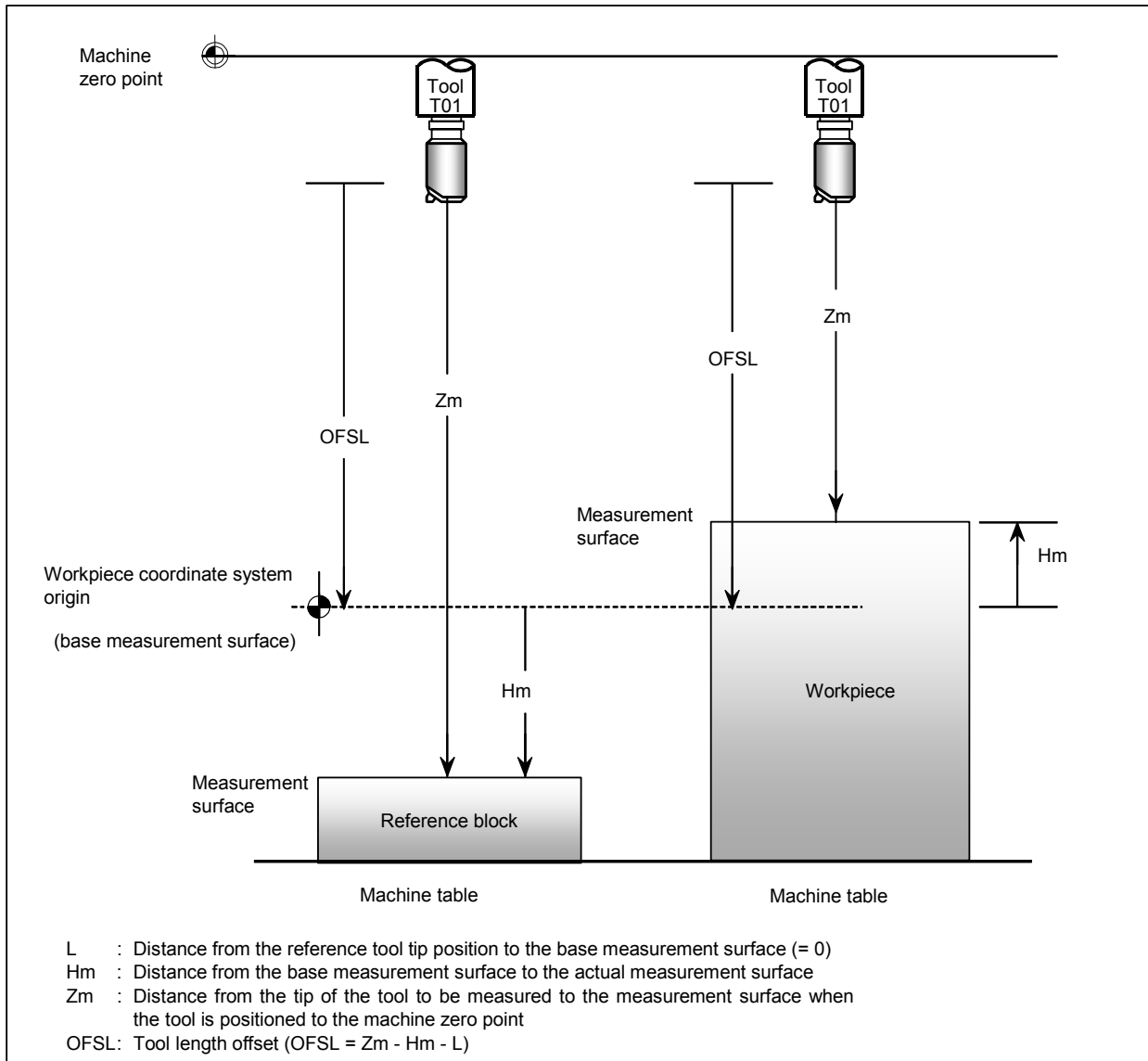
Defining the actual tool length as the tool length compensation has the advantage of eliminating the need for remeasuring, even if the workpiece is changed, provided the tool is not worn. Another advantage is that the tool length compensation need not be re-set when multiple workpieces are machined. In this case, assign a workpiece coordinate system to each workpiece, using G54 to G59, and set the workpiece origin offset for each workpiece.

(2) Definition 2

In the second definition method, the tool length compensation is the distance from the tool nose position to the workpiece coordinate system origin when the machine is positioned to the Z-axis zero point. A tool length compensation defined in this way will be equal to the difference between the length of the tool to be measured and that of the reference tool, in the same way as with definition 1. The reference tool for definition 2 is, however, an imaginary tool which has a tool nose at the workpiece coordinate system origin when the machine is positioned to the Z-axis zero point.



The base measurement surface for this definition is located at the workpiece coordinate system origin. Because the reference tool nose is also located at the workpiece coordinate system origin, distance L from the reference tool nose position to the base measurement surface is 0. Set, therefore, 0 in the parameter for distance L (No. 5022). The actual measurement surface is usually the same as the base measurement surface, located at the workpiece coordinate system origin. If, however, the measurement surface is the top surface of the reference block, or if the workpiece coordinate system origin is located on other than the top surface of the workpiece (for example, when the origin is shifted from the workpiece top surface by an amount equal to the cutting allowance), set the distance from the base measurement surface to the actual measurement surface as Hm, such that the tool length compensation (OFSL) can be calculated using the same formula as that used for definition 1.



The reference tool for definition 2 has a tool nose at the workpiece coordinate system origin when the machine is positioned to the Z-axis zero point. Whenever the workpiece is changed, therefore, the tool length compensation must be remeasured. Remeasuring is not, however, necessary if the difference between the workpiece coordinate system origin for a new workpiece and that when the tool length compensation value was measured is set as the new workpiece origin offset (any of G54 to G59). In such a case, the tool length compensation need not be modified, even when the workpiece is changed.

Taking a different point of view, definition 2 can be thought of as setting the workpiece origin offset as the tool length compensation for each tool.

- Measuring the tool length compensation along a specified axis

Because the tool is usually mounted in parallel with the Z-axis, the tool length compensation is measured by moving the tool along the Z-axis. Some machines, however, have their W-axis in parallel with the Z-axis, making it necessary to measure the tool length compensation by moving the tool along the W-axis. Moreover, some machines, when fitted with an attachment, support the mounting of the tool in parallel with an axis other than the Z-axis. For such a machine, the tool length compensation can be measured along a specified axis by setting bit 2 (TMA) of parameter No. 5007 to 1.

To measure the tool length compensation along an axis other than the Z-axis, first set distance L from the reference tool nose position to the base measurement surface, for each of the axes along which the tool length compensation may be measured, in parameter No. 5022, in addition to distance L along the Z-axis.

Next, set distance Hm from the base measurement surface to the actual measurement surface for the axis along which the tool length compensation is to be measured (see Explanations, below). Finally, move the tool along that axis until it touches the workpiece or reference block, then enter the name of that axis before pressing soft key [MEASUR] or [MEASUR+]. When the tool offset is measured along the W-axis, for example, enter W then press soft key [MEASUR] or [MEASUR+].

- Tool change position

The tool change position must be set beforehand, using bits 1 (TC3) and 0 (TC2) of parameter No. 5007.

Table 1.1.3 (a)

TC3	TC2	Meaning
0	0	The tool change position is the first reference position (G28)
0	1	The tool change position is the second reference position (G30 P2)
1	0	The tool change position is the third reference position (G30 P3)
1	1	The tool change position is the fourth reference position (G30 P4)

Procedure for measuring the workpiece origin offset (for 8.4/10.4-inch display unit)

In addition to the workpiece origin offset along the tool lengthwise axis, that is, the Z-axis, the workpiece origin offsets along the X- and Y-axes, on a plane perpendicular to the Z-axis, can also be measured easily. The workpiece origin offsets along the X- and Y-axes can be measured regardless of whether the workpiece origin is located on a surface of the workpiece or at the center of a hole to be machined. For details of this measurement, refer to the manual supplied by the machine tool builder.

- Measuring the Z-axis workpiece origin offset

- 1 Select a tool using an MDI command, then move it to the spindle position (see the explanation of the procedure for measuring the tool length compensation). The tool length compensation for the selected tool must be measured beforehand.
- 2 Press mode selection switch HANDLE or JOG.
- 3 Set the workpiece origin offset measurement mode switch on the machine operator's panel to ON. The workpiece origin offset screen appears and "WOFS" blinks in the status display at the bottom of the screen.
- 4 Enter the tool length compensation for the selected tool. Enter the offset using numeric keys then press soft key [TL-INP].

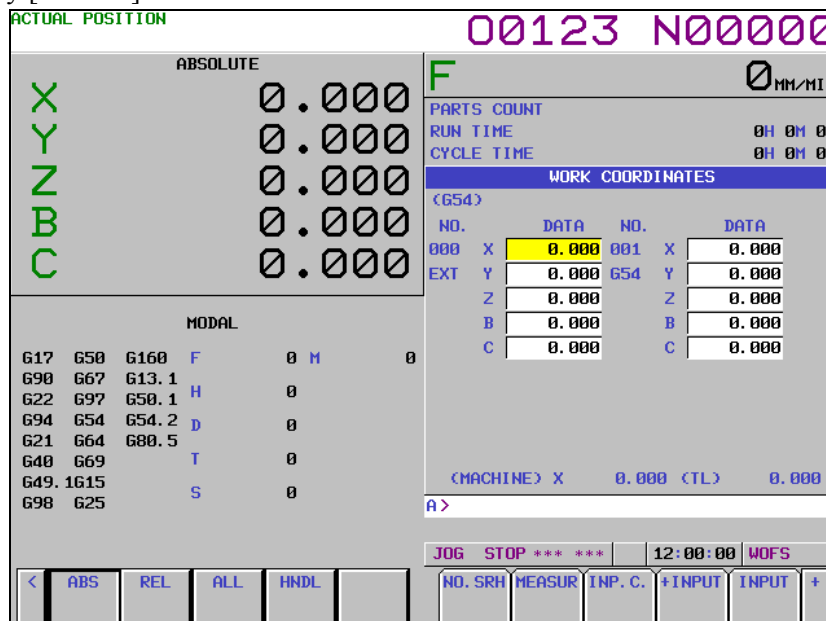


Fig. 1.1.3 (g) Workpiece origin offset setting screen (10.4-inch display unit)

- 5 Position the cursor to the workpiece origin offset number to be used to store the offset (any of G54 to G59). No problem will arise even if the cursor is positioned to the offset for other than the Z-axis.
- 6 Move the tool by means of manual handle feed or jog feed until it touches the top surface of the workpiece.
- 7 Enter the axis name, Z, press soft key [MEASUR], then press soft key [INPUT]. The Z-axis workpiece origin offset value is set and the cursor is positioned to the set Z-axis workpiece origin offset. There is no need to enter Z provided the parameter has been set so that only the Z-axis workpiece origin offset is to be measured (bit 3 (WMA) of parameter No. 5007 = 0).
To set the workpiece origin on other than the workpiece top surface (for example, when the origin is shifted from the workpiece top surface by an amount equal to the cutting allowance), enter the amount of shift (S in the Fig. 1.1.3 (h)) using the numeric keys, press soft key [MEASUR], then press soft key [INPUT].

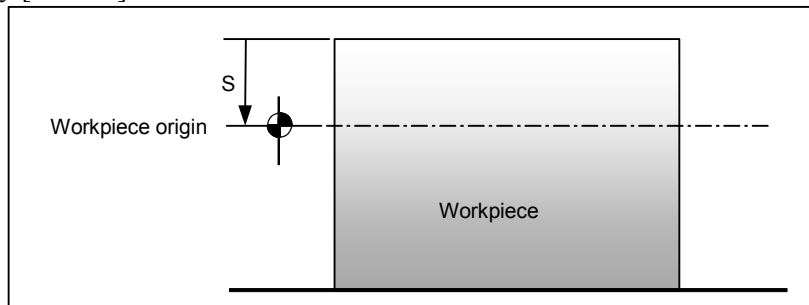


Fig. 1.1.3 (h)

- 8 To measure any subsequent workpiece origins, retract the tool from the workpiece, then repeat steps 5 to 7.

- Measuring the X-/Y-axis workpiece origin offset based on a reference surface

To set the X- or Y-axis workpiece origin offset on a specified surface of the workpiece, set bit 3 (WMA) of parameter No. 5007 to 1, then follow the same procedure as that for measuring the Z-axis workpiece origin offset. In step 4, however, enter the cutter compensation value for the selected tool, instead of the tool length compensation. After entering the cutter compensation value with the numeric keys, press soft key [TL-INP].

⚠ CAUTION

When entering the cutter compensation value, ensure that its sign is entered correctly.

- When the measurement surface is located in the positive (+) direction relative to the tool, enter a minus (-) sign.
- When the measurement surface is located in the negative (-) direction relative to the tool, enter a plus (+) sign.

- Measuring the X-/Y-axis workpiece origin offset based on a reference hole

- 1 Connect a measurement probe, fitted with a sensor, to the spindle.
- 2 Press mode selection switch HANDLE or JOG.
- 3 Set the workpiece origin offset measurement mode switch on the machine operator's panel to ON. The workpiece origin offset screen appears and "WOFS" blinks in the status display at the bottom of the screen, indicating that the preparation required prior to measuring the workpiece origin offset has been completed.
- 4 Position the cursor to the workpiece origin offset number to be used to store the offset (any of G54 to G59). No problem will arise even if the cursor is positioned to the offset for other than the X- or Y-axis.
- 5 Move the tool by means of manual handle feed or jog feed until the measurement probe touches the circumference of the hole. Do not move the tool along more than one axis at any one time.

- 6 As soon as the sensor detects contact with the circumference, input a skip signal to the machine, thus stopping the axial movement of manual handle feed or jog feed. Simultaneously, the position at which feed stopped is stored as the first measurement point. The machine coordinates of the stored measurement point are displayed at the bottom right of the screen, as Fig. 1.1.3 (i):

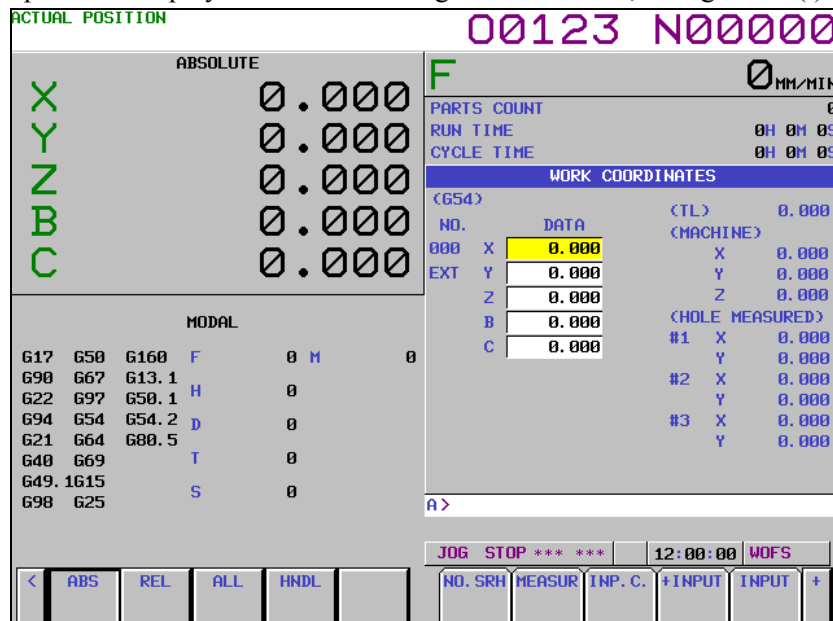




Fig. 1.1.3 (i) Workpiece origin offset setting screen (10.4-inch display unit)

- 7 Move the measurement probe to the second measurement point. At this time, the CNC interlocks the machine to prevent the probe from moving in the direction in which it was moved so as to touch the current measurement point. For example, when the probe touched the measurement point after being moved in the +X direction, movement of the probe to the next measurement point is allowed only in the -X direction. Movement in the +X, +Y, or -Y direction is interlocked until the skip signal is set to 0. Once the probe touches the second measurement point, follow the same procedure as that for storing the first measurement point.
- 8 Once the probe has touched the third measurement point, press soft key [MEASUR], then [CENTER]. This calculates the center of the hole from the coordinates of the three measured points, then sets the X- and Y-axis workpiece origin offsets. To cancel and restart measurement at any point, press the  key. Pressing the  key clears the coordinates of all stored measurement points.

Procedure for measuring the workpiece origin offset (for 15/19-inch display unit)

In addition to the workpiece origin offset along the tool lengthwise axis, that is, the Z-axis, the workpiece origin offsets along the X- and Y-axes, on a plane perpendicular to the Z-axis, can also be measured easily. The workpiece origin offsets along the X- and Y-axes can be measured regardless of whether the workpiece origin is located on a surface of the workpiece or at the center of a hole to be machined. For details of this measurement, refer to the manual supplied by the machine tool builder.

- Measuring the Z-axis workpiece origin offset

- 1 Select a tool using an MDI command, then move it to the spindle position (see the explanation of the procedure for measuring the tool length compensation). The tool length compensation for the selected tool must be measured beforehand.
- 2 Press mode selection switch HANDLE or JOG.
- 3 Set the workpiece origin offset measurement mode switch on the machine operator's panel to ON. The workpiece origin offset screen appears and "WOFS" blinks in the status display at the bottom of the screen.

- 4 Enter the tool length compensation for the selected tool. Enter the offset using numeric keys then press horizontal soft key [TL-INP].

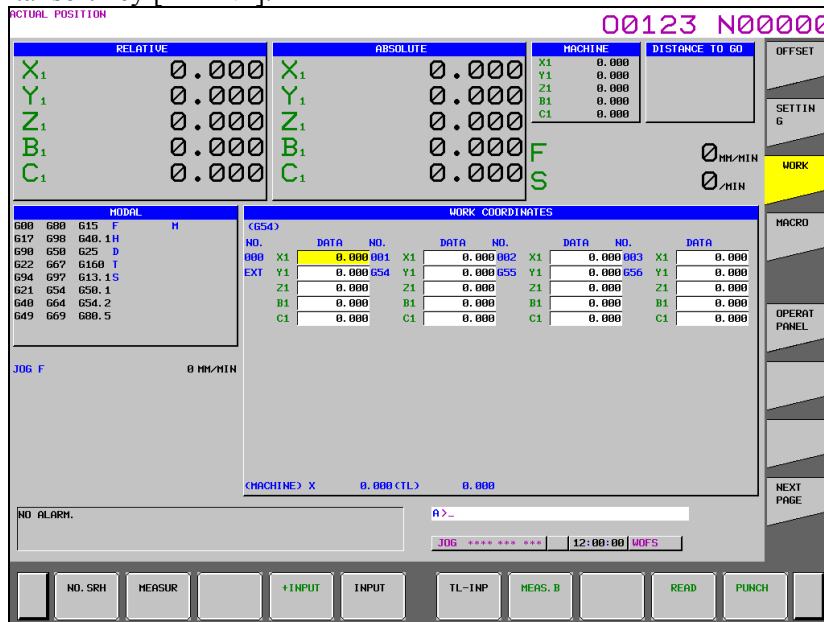


Fig. 1.1.3 (j) Workpiece origin offset setting screen (15-inch display unit)

- 5 Position the cursor to the workpiece origin offset number to be used to store the offset (any of G54 to G59). No problem will arise even if the cursor is positioned to the offset for other than the Z-axis.
- 6 Move the tool by means of manual handle feed or jog feed until it touches the top surface of the workpiece.
- 7 Enter the axis name, Z, press horizontal soft key [MEASUR], then press horizontal soft key [INPUT]. The Z-axis workpiece origin offset value is set and the cursor is positioned to the set Z-axis workpiece origin offset. There is no need to enter Z provided the parameter has been set so that only the Z-axis workpiece origin offset is to be measured (bit 3 (WMA) of No. 5007 = 0).
To set the workpiece origin on other than the workpiece top surface (for example, when the origin is shifted from the workpiece top surface by an amount equal to the cutting allowance), enter the amount of shift (S in the Fig. 1.1.3 (k)) using the numeric keys, press horizontal soft key [MEASUR], then press horizontal soft key [INPUT].

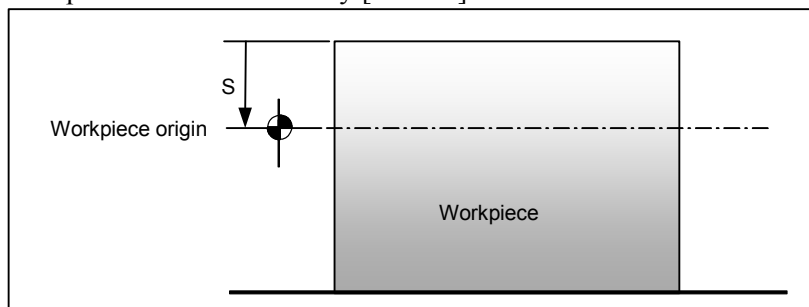


Fig. 1.1.3 (k)

- 8 To measure any subsequent workpiece origins, retract the tool from the workpiece, then repeat steps 5 to 7.

- Measuring the X-/Y-axis workpiece origin offset based on a reference surface
To set the X- or Y-axis workpiece origin offset on a specified surface of the workpiece, set bit 3 (WMA) of parameter No. 5007 to 1, then follow the same procedure as that for measuring the Z-axis workpiece origin offset. In step 4, however, enter the cutter compensation value for the selected tool, instead of the tool length compensation. After entering the cutter compensation value with the numeric keys, press soft key [TL INPUT].

CAUTION
 When entering the cutter compensation value, ensure that its sign is entered correctly.

- When the measurement surface is located in the positive (+) direction relative to the tool, enter a minus (-) sign.
- When the measurement surface is located in the negative (-) direction relative to the tool, enter a plus (+) sign.

- Measuring the X-/Y-axis workpiece origin offset based on a reference hole

- 1 Connect a measurement probe, fitted with a sensor, to the spindle.
- 2 Press mode selection switch HANDLE or JOG.
- 3 Set the workpiece origin offset measurement mode switch on the machine operator's panel to ON. The workpiece origin offset screen appears and "WOFS" blinks in the status display at the bottom of the screen, indicating that the preparation required prior to measuring the workpiece origin offset has been completed.
- 4 Position the cursor to the workpiece origin offset number to be used to store the offset (any of G54 to G59). No problem will arise even if the cursor is positioned to the offset for other than the X- or Y-axis.
- 5 Move the tool by means of manual handle feed or jog feed until the measurement probe touches the circumference of the hole. Do not move the tool along more than one axis at any one time.
- 6 As soon as the sensor detects contact with the circumference, input a skip signal to the machine, thus stopping the axial movement of manual handle feed or jog feed. Simultaneously, the position at which feed stopped is stored as the first measurement point. The machine coordinates of the stored measurement point are displayed at the bottom right of the screen, as Fig. 1.1.3 (l):

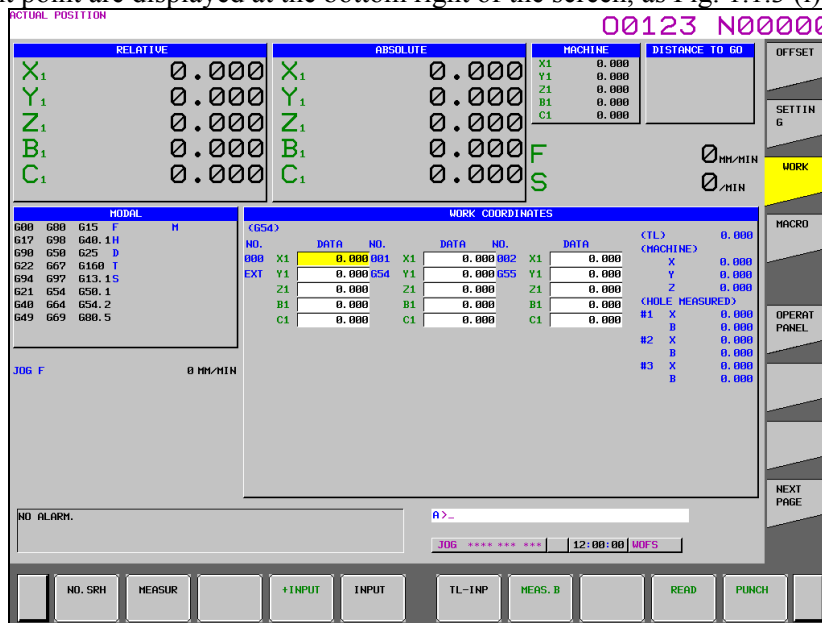




Fig. 1.1.3 (l) Workpiece origin offset setting screen (15-inch display unit)

- 7 Move the measurement probe to the second measurement point. At this time, the CNC interlocks the machine to prevent the probe from moving in the direction in which it was moved so as to touch the current measurement point. For example, when the probe touched the measurement point after being moved in the +X direction, movement of the probe to the next measurement point is allowed only in the -X direction. Movement in the +X, +Y, or -Y direction is interlocked until the skip signal is set to 0. Once the probe touches the second measurement point, follow the same procedure as that for storing the first measurement point.

- 8 Once the probe has touched the third measurement point, press horizontal soft key [MEASUR], then [CENTER]. This calculates the center of the hole from the coordinates of the three measured points, then sets the X- and Y-axis workpiece origin offsets. To cancel and restart measurement at any point, press the  key. Pressing the  key clears the coordinates of all stored measurement points.

Explanation

- Z-axis workpiece origin offset

Definitions 1 and 2, described in "Definition of tool length compensation" in Explanations for measuring the tool length compensation, also apply to the general concept of the Z-axis workpiece origin offset, as follows:

(1) Definition 1

In definition 1, the Z-axis workpiece origin offset is defined as the distance from the machine zero point to the origin of the workpiece coordinate system (Fig. 1.1.3 (m)).

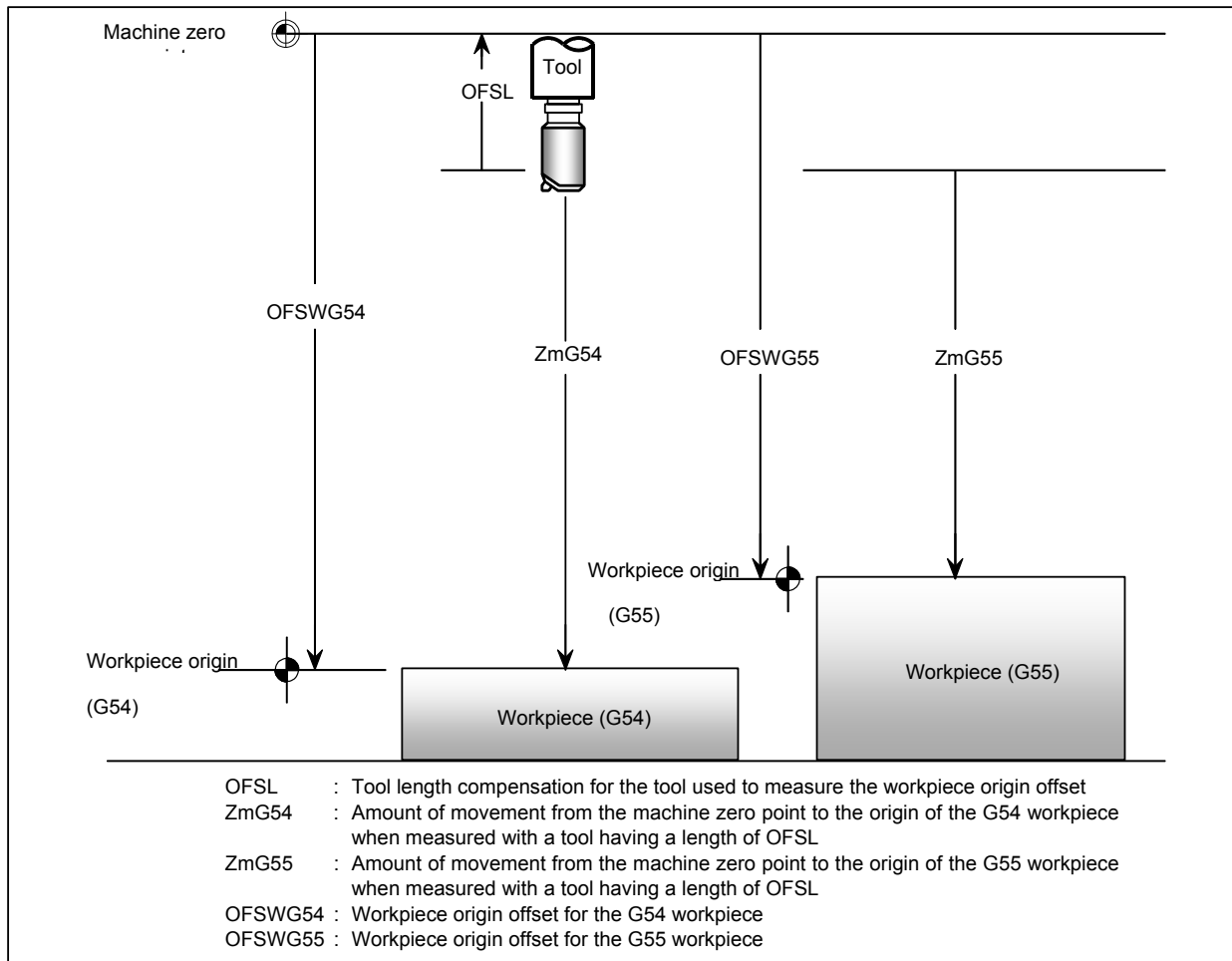


Fig. 1.1.3 (m)

As can be seen from the Fig. 1.1.3 (m), the Z-axis workpiece origin offset can be calculated from the following formula:

$$\text{OFSW} = \text{Zm} - \text{OFSL}$$

where

OFSW : Workpiece origin offset

OFSL : Tool length compensation for the tool used to measure the workpiece origin offset

Zm : Amount of movement from the machine zero point to the workpiece origin when measured with a tool having a length of OFSL

(2) Definition 2

The tool length compensation in definition 2 equals the Z-axis workpiece origin offset, as described above. Usually in this case, therefore, the workpiece origin offset need not be set. If, however, the workpiece is changed after its tool length compensation has been measured, or if multiple workpieces are machined, the workpiece origin coordinates can be set as follows when assigning workpiece coordinate systems to G54 to G59, thus eliminating the need to remeasure the tool length compensation.

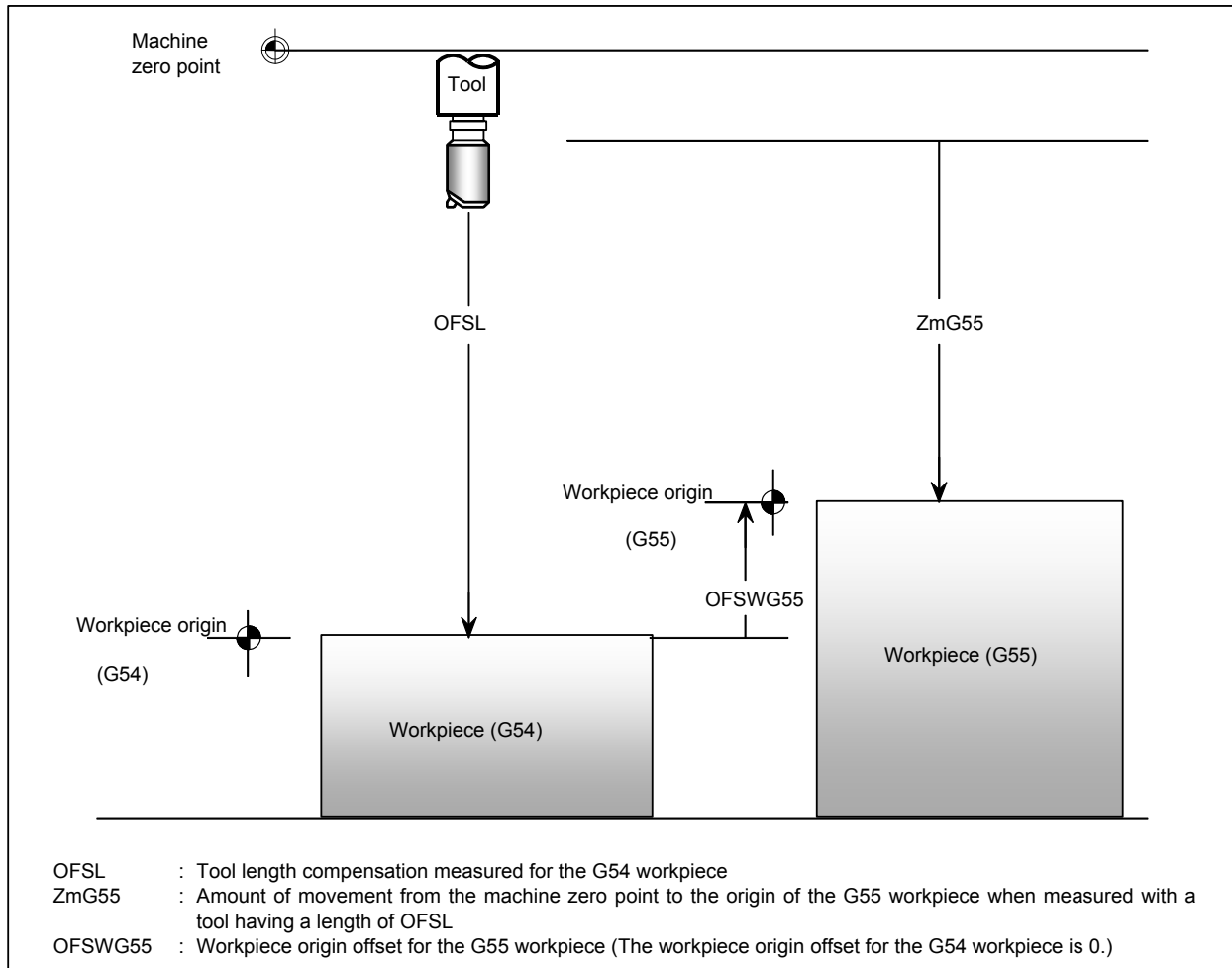


Fig. 1.1.3 (n)

For definition 2, the workpiece origin offset can be calculated using the same formula as that used for definition 1:

$$\text{OFSW} = \text{Zm} - \text{OFSL}$$

where

OFSW : Workpiece origin offset

OFSL : Tool length compensation for the tool used to measure the workpiece origin offset

Zm : Amount of movement from the machine zero point to the workpiece origin when measured with a tool having a length of OFSL

- X-/Y-axis workpiece origin offset

The X- and Y-axis workpiece origin offsets can be measured regardless of whether the workpiece origin is located on a surface of the workpiece or at the center of a hole to be machined.

(1) When the workpiece origin is located on a surface

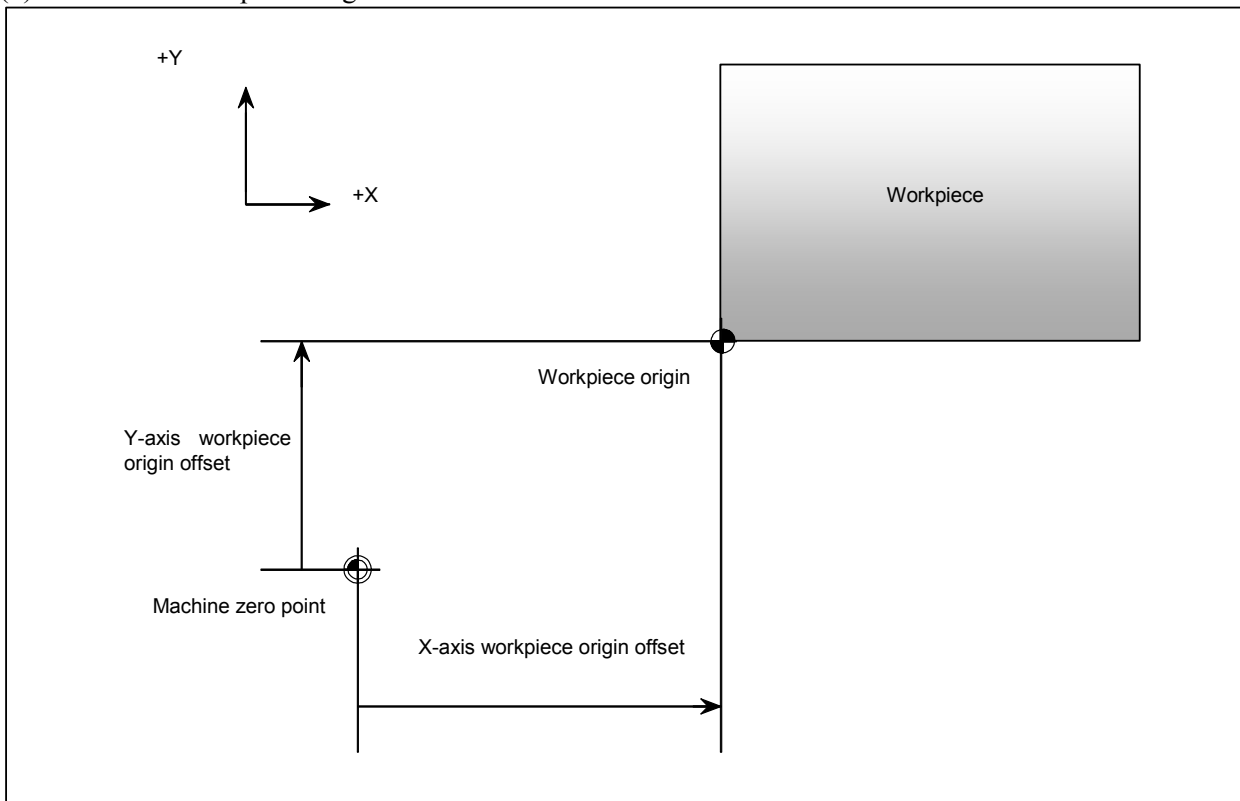


Fig. 1.1.3 (o)

In the case of the Fig. 1.1.3 (o), the workpiece origin is located on a side surface of the workpiece. The measurement of the X-/Y-axis workpiece origin offset when the origin is located on a surface of the workpiece is the same as that for the Z-axis workpiece origin offset, but with the following exception: The tool length compensation for the tool used to measure the offset is used to calculate the Z-axis workpiece origin offset, while the cutter compensation value for the tool is used to calculate the X-/Y-axis workpiece origin offset.

(2) When the workpiece origin is located at the center of a hole.

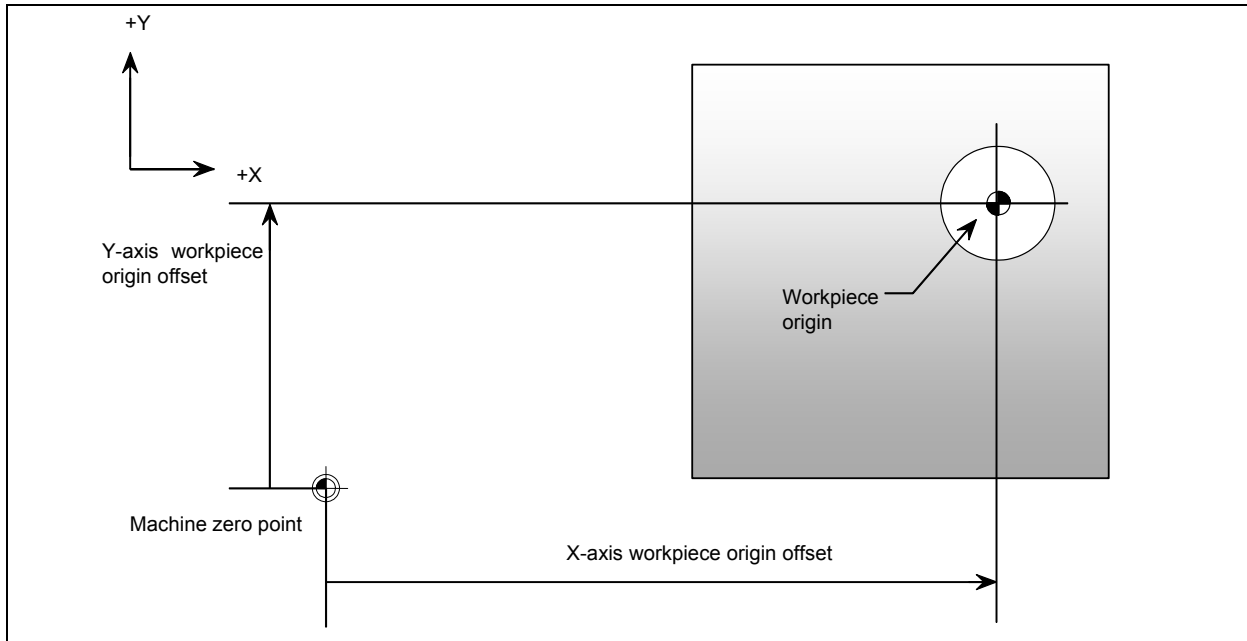
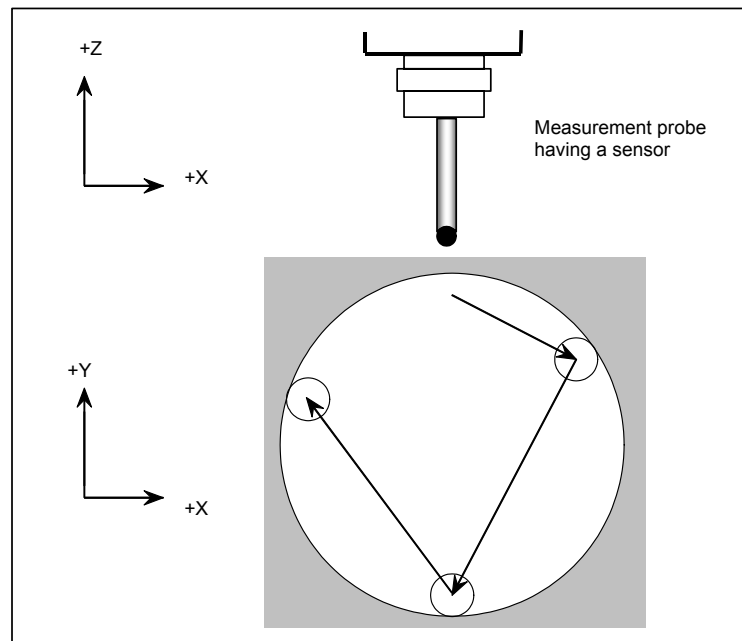


Fig. 1.1.3 (q)

In the case of the Fig. 1.1.3 (q), the workpiece origin is located at the center of a hole in the workpiece. A measurement probe having a sensor at its tool nose is used to measure the positions of three arbitrary points on the circumference of the hole. The three points prescribe a unique circle, the center of which is set as the X-/Y-axis workpiece origin. Set bit 4 (WMH) of parameter No. 5007 to 1 before starting the measurement.



- Using a skip signal



A measurement probe, fitted with a sensor, can also be used to measure the Z-axis workpiece origin offset or measure the X-/Y-axis workpiece origin offset based on a surface, in the same way as when measuring the X-/Y-axis workpiece origin offset based on a hole. By inputting a skip signal as soon as the probe touches the workpiece surface, feed is automatically stopped. Subsequently, apply the same procedure as that for each measurement.

1.1.4 Setting and Displaying the Rotary Table Dynamic Fixture Offset

The fixture offset screen is either a fixture offset (ACT) screen for verifying the currently selected fixture offset value or a fixture offset screen for setting and verifying eight fixture offset value sets.

Active Fixture offset screen (for 8.4/10.4-inch display unit)

Procedure

- 1 Press function key .
- 2 Press the continuous menu key  several times, until soft key [F-ACT] appears.
- 3 Press soft key [F-ACT].
The fixture offset (ACT) screen displays.

This screen displays the currently selected fixture offset number (P) and fixture offset vector.

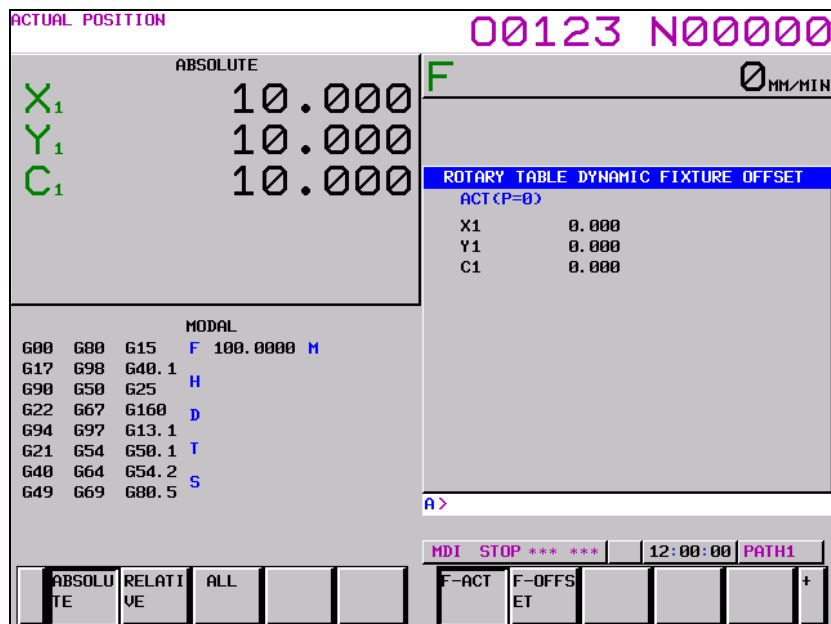



Fig. 1.1.4 (a) Active fixture offset display screen (10.4-inch display unit)

Active Fixture offset screen (for 15/19-inch display unit)

Procedure

- 1 Press function key .
- 2 Press vertical soft key [NEXT PAGE] until vertical soft key [F-ACT] appears.
- 3 Press vertical soft key [F-ACT].
The fixture offset (ACT) screen displays.

This screen displays the currently selected fixture offset number (P) and fixture offset vector.

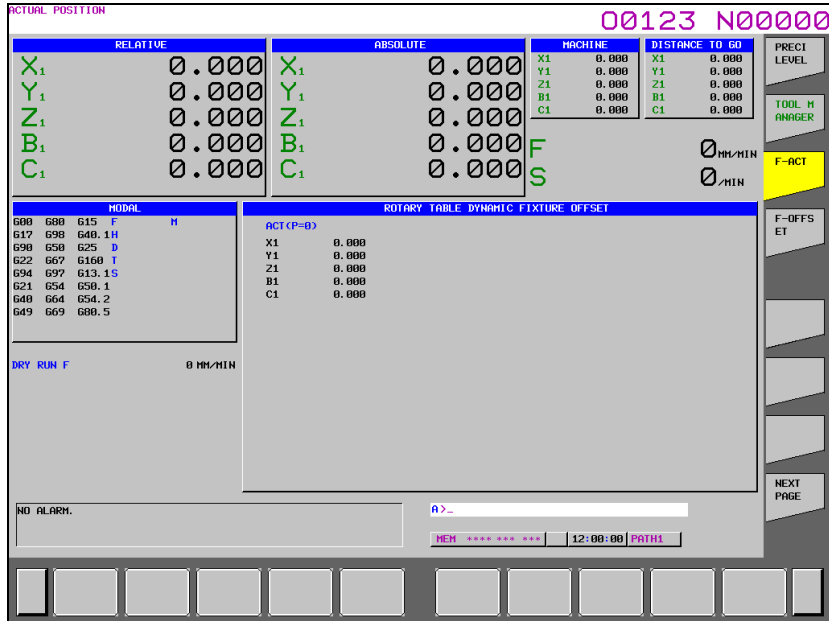


Fig. 1.1.4 (b) Active fixture offset display screen (15-inch display unit)

Fixture offset setting screen (for 8.4/10.4-inch display unit)

Procedure

- 1 Press function key .
- 2 Press the continuous menu key  several times, until soft key [F-OFFSET] appears.
- 3 Press soft key [F-OFFSET].
The fixture offset (ACT) screen displays.

The number of groups that are displayed on one screen is fixed 1 to 4 groups by number of control axis.

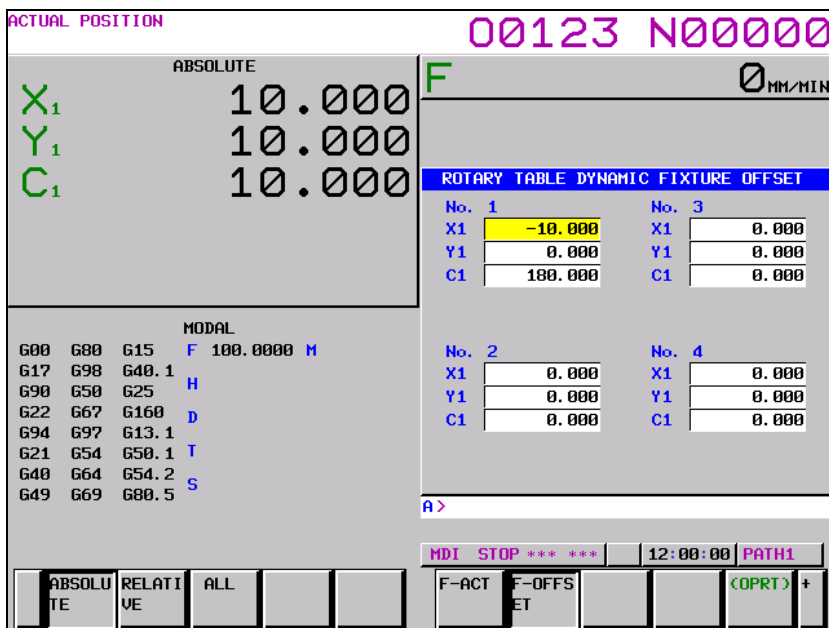


Fig. 1.1.4 (c) Fixture offset setting screen (10.4-inch display unit)

Operation

- **Entering numeric values**
- Pres soft key [(OPRT)] to display the following operation soft key.



- Use the page and cursor keys, and soft key [NO.SRH] to place cursor at a desired items to be set.
- Enter data, then press soft key [INPUT]
- To add a value to already set data, press soft key [+INPUT].

Data can be set using the MDI key.

- **Number of groups of fixture offset values**
- NO. 01 to NO. 08 indicates the number of a group of fixture offset values. There are eight groups. Soft key [NO. SRH] can be used to place cursor at a desired items to be set.

- **Inputting fixture offset values**
- Soft key [F INPUT] can be used to output fixture offset values from an external device.

- **Outputting fixture offset values**
- Soft key [F OUTPUT] can be used to output fixture offset values to an external device.

Fixture offset setting screen (for 15/19-inch display unit)

Procedure

- 1 Press function key .
- 2 Press vertical soft key [NEXT PAGE] until vertical soft key [F-OFFSET] appears.
- 3 Press vertical soft key [F-OFFSET].
The fixture offset (ACT) screen displays.

The number of groups that are displayed on one screen is fixed 1 to 8 groups by number of control axis.

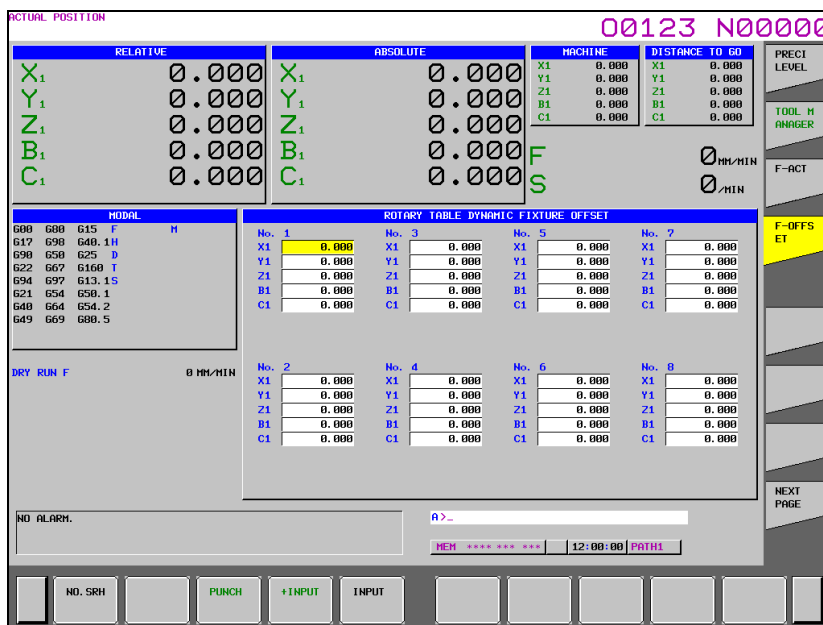



Fig. 1.1.4 (d) Fixture offset setting screen (15-inch display unit)

Operation

- Entering numeric values

- Use the page and cursor keys, and horizontal soft key [NO.SRH] to place cursor at a desired items to be set.
- Enter data, then press horizontal soft key [INPUT]
- To add a value to already set data, press horizontal soft key [+INPUT].

Data can be set using the  MDI key.

- Number of groups of fixture offset values

NO. 01 to NO. 08 indicates the number of a group of fixture offset values.

There are eight groups. Horizontal soft key [NO. SRH] can be used to place cursor at a desired items to be set.

- Inputting fixture offset values

Horizontal soft key [F INPUT] can be used to output fixture offset values from an external device.

- Outputting fixture offset values

Soft key [F OUTPUT] can be used to output fixture offset values to an external device.

1.1.5 Input of Tool Offset Value Measured B

Explanation

- Basic procedure for setting the tool offset value

Shown below is the basic procedure for setting the tool offset value:

- (1) Execute manual reference position return.
- (2) Select manual handle mode or manual continuous feed mode and set the tool compensation value write mode select signal GQSMC to 1.
- (3) Select a tool to be measured.
- (4) Use tool offset number select signal OFNC0 to OFNC9 to select the tool offset number to be set.
- (5) Near the tool to the touch sensor by manual operation.
- (6) When the tool nose touches the contact surface of the touch sensor and the tool offset value write signal +MIT1 is output, the following operations are performed:
 - i) The tool is subject to axis interlock in the axis direction in which the tool is moving and automatically stops manual operation.
 - ii) The calculated tool offset value is set with the tool offset number set by tool offset number select signal OFNC0 to OFNC9.
- (7) For both X-axis and Z-axis, their offset values are set by operations (5) and (6).
- (8) Repeat operations (3) to (7) for all necessary tools.
- (9) Set the tool compensation value writing mode signal GQSMC to 0.

1.1.6 Spindle Unit Compensation, Nutating Rotary Head Tool Length Compensation

On the spindle unit compensation/nutating rotary head tool length compensation screen, the following items are displayed:

- Parameters Nos. 25861 to 25888
- Spindle unit compensation vector
- Nutating rotary head tool length compensation vector

If bit 7 (SU3) of parameter No. 25860 is set to 1, parameter input is possible.



The following table shows the correspondence between the screen names and parameter numbers.

Table 1.1.6 Parameter list

Parameter number	Screen name		Description
25861	(SET1)	R-AX	Axis number of the rotation axis (1st group)
25862	(SET1)	L-AX1	Axis number of the linear axis 1 (1st group)
25863	(SET1)	L-AX2	Axis number of the linear axis 2 (1st group)
25864	(SET1)	L-AX3	Axis number of the linear axis 3 (1st group)
25865	(SET1)	Inclination	Inclination (1st group)
25866	(SET2)	R-AX	Axis number of the rotation axis (2nd group)
25867	(SET2)	L-AX1	Axis number of the linear axis 1 (2nd group)
25868	(SET2)	L-AX2	Axis number of the linear axis 2 (2nd group)
25869	(SET2)	L-AX3	Axis number of the linear axis 3 (2nd group)
25870	(SET2)	Inclination	Inclination (2nd group)
25871	(V2/SU)	1st line	Component of linear axis 1 of V_2 vector
25872	(V2/SU)	2nd line	Component of linear axis 2 of V_2 vector
25873	(V2/SU)	3rd line	Component of linear axis 3 of V_2 vector
25874	(V1/SU)	1st line	Component of linear axis 1 of V_1 vector
25875	(V1/SU)	2nd line	Component of linear axis 2 of V_1 vector
25876	(V1/SU)	3rd line	Component of linear axis 3 of V_1 vector
25877	(V0/SU)	1st line	Component of linear axis 1 of V_0 vector
25878	(V0/SU)	2nd line	Component of linear axis 2 of V_0 vector
25879	(V0/SU)	3rd line	Component of linear axis 3 of V_0 vector
25880	(S0/SU)	1st line	Rotation axis standard angle of the 1st group
25881	(S0/SU)	2nd line	Rotation axis standard angle of the 2nd group
25882	(S1/SU)	1st line	Rotation axis compensation amount of the 1st group
25883	(S1/SU)	2nd line	Rotation axis compensation amount of the 2nd group
25884	(S0/TOOL)	1st line	Rotation axis standard angle of the 1st group
25885	(S0/TOOL)	2nd line	Rotation axis standard angle of the 2nd group
25886	(ANGLE/TOOL)	RA	Tool axis standard angle (in the plane by the linear axes 2-3)
25887	(ANGLE/TOOL)	RB	Tool axis standard angle (in the plane by the linear axes 3-1)
25888	(SU ANGLE OFFSET)		Inclination compensation amount

Spindle unit compensation/nutating rotary head tool length compensation display screen

Procedure

- 1 Press function key .
- 2 Press the continuous menu key  several times, until soft key [SU OFFSET] appears.
- 3 Press soft key [SU OFFSET].
The SU&NUTATOR OFFSET screen displays.

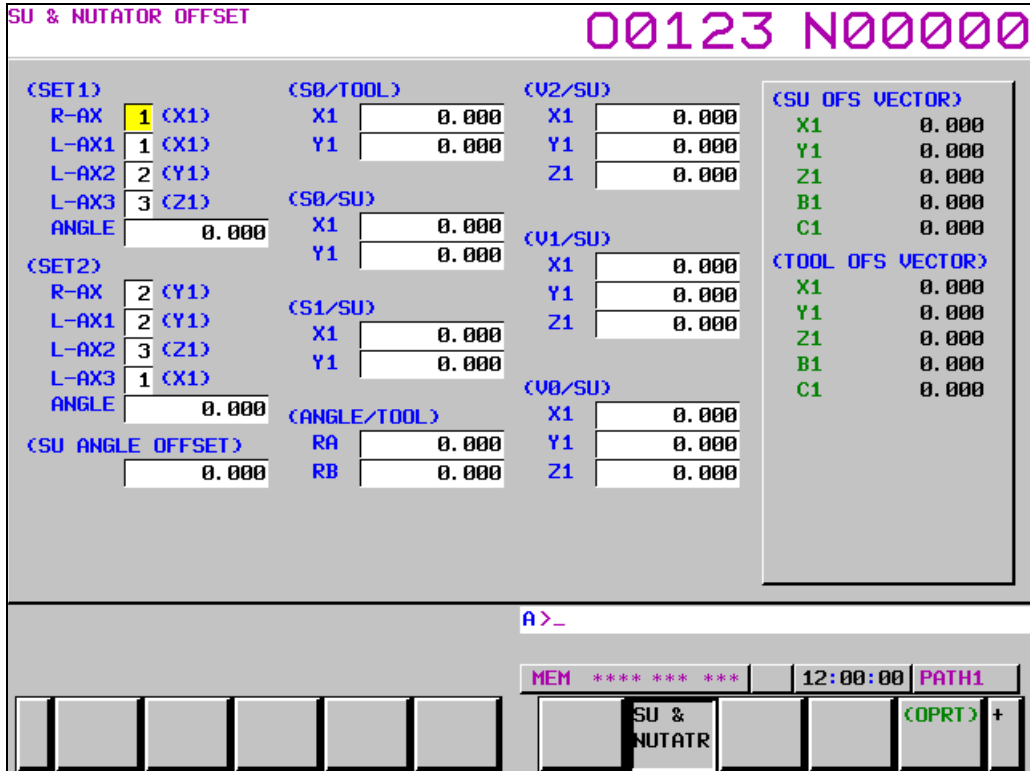


Fig. 1.1.6 SU&NUTATOR OFFSET screen

APPENDIX

A PARAMETERS

This manual describes all parameters indicated in this manual.

For those parameters that are not indicated in this manual and other parameters, refer to the parameter manual.

Appendix A, "PARAMETERS", consists of the following sections:

A.1 DESCRIPTION OF PARAMETERS373
 A.2 DATA TYPE.....420
 A.3 STANDARD PARAMETER SETTING TABLES421

A.1 DESCRIPTION OF PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
0001							FCV	

[Input type] Setting input

[Data type] Bit path

#1 FCV Program format

0: Series 16 standard format

1: Series 15 format

NOTE

1 Programs created in the Series 15 program format can be used for operation on the following functions:

- 1 Subprogram call M98
- 2 Multiple repetitive canned cycle G71.7 to G71.6
- 3 Drilling canned cycle G73, G74, G76, G80 to G89

2 When the program format used in the Series 15 is used for this CNC, some limits may add. Refer to the Operator's Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
1013					ISEx	ISDx	ISCx	ISAx

[Input type] Parameter input

[Data type] Bit axis

NOTE

When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ISAx

#1 ISCx

#2 ISDx

#3 ISEx Increment system of each axis

Increment system	Bit 3 (ISE)	Bit 2 (ISD)	Bit 1 (ISC)	Bit 0 (ISA)
IS-A	0	0	0	1
IS-B	0	0	0	0

Increment system	Bit 3 (ISE)	Bit 2 (ISD)	Bit 1 (ISC)	Bit 0 (ISA)
IS-C	0	0	1	0
IS-D	0	1	0	0
IS-E	1	0	0	0

1020	Program axis name for each axis
-------------	--

- [Input type] Parameter input
- [Data type] Byte axis
- [Valid data range] 65 to 67, 85 to 90

An axis name (axis name 1: parameter No. 1020) can be arbitrarily selected from A, B, C, U, V, W, X, Y, and Z. (When G code system A is used with the lathe system, however, U, V, and W are not selectable.) When bit 0 (EEA) of parameter No. 1000 is set to 1, the length of an axis name can be extended to three characters by setting axis name 2 (parameter No. 1025) and axis name 3 (parameter No. 1026) (extended axis name).

For axis names 2 and 3, a character from 0 to 9 and A to Z of ASCII code can be arbitrarily selected. However, the setting of axis name 3 for each axis is invalid if axis name 2 is not set. Moreover, if a character from 0 to 9 is set as axis name 2, do not use a character from A to Z as axis name 3.

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

When G code system A is used with the lathe system, and the character X, Y, Z, or C is used as axis name 1 of an axis, a command with U, V, W, or H specified for axis name 1 represents an incremental programming for the axis.

NOTE

- 1 When a multiple repetitive canned cycle for turning is used, no character other than X, Y, and Z can be used as the address of the axis.
- 2 An address other than addresses A, B, and C cannot be used as the address of a rotary axis used for the function for tool length compensation in a specified direction or the tool center point control function.
- 3 When the custom macro function is enabled, the same extended axis name as a reserved word cannot be used. Such an extended axis name is regarded as a reserved word.
Because of reserved words of custom macros, extended axis names that start with the following two characters cannot be used:
AB, AC, AD, AN, AS, AT, AX, BC, BI, BP, CA, CL, CO, US, WH, WR, XO, ZD, ZE, ZO, ZW
- 4 In a macro call, no extended axis name can be used as an argument.

1022	Setting of each axis in the basic coordinate system
-------------	--

- [Input type] Parameter input
- [Data type] Byte axis
- [Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) and a 3-dimensional tool compensation space (XpYpZp), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.

A basic axis (X, Y, or Z) can be specified only for one control axis.

Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1023

Number of the servo axis for each axis

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Byte axis

[Valid data range] 0 to Number of controlled axes

Set the servo axis for each control axis.

Usually set to same number as the control axis number.

The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

- With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.

Example)

When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.

- For tandem controlled axes or electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.

Tandem axis: For a master axis, set an odd (1, 3, 5, 7, ...) servo axis number. For a slave axis to be paired, set a value obtained by adding 1 to the value set for the master axis.

EGB axis: For a slave axis, set an odd (1, 3, 5, 7, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

1031

Reference axis

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 1 to Number of controlled axes

The unit of some parameters common to all axes such as those for dry run feedrate and single-digit F1 feedrate may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.

Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0			LRP	

[Input type] Parameter input

[Data type] Bit path

#1 LRP Positioning (G00)

0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.

1: Positioning is performed with linear interpolation so that the tool moves in a straight line.

When using 3-dimensional coordinate system conversion, set this parameter to 1.

#4 RF0 When cutting feedrate override is 0% during rapid traverse,

0: The machine tool does not stop moving.

1: The machine tool stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0
1403				ROC	EDT			

[Input type] Parameter input

[Data type] Bit path

#3 EDT The function for corner control by feedrate (for a gas cutting machine) is:

0: Disabled.

1: Enabled.

When the feedrate has reduced to the feedrate set in parameter No. 1474, from which the system regards the number of accumulated pulses as being 0, the next block is executed.

#4 ROC In the threading cycle G76.7, rapid traverse override for retraction after threading is finished is:

0: Effective

1: Not effective (Override of 100%)

1410	Dry run rate
------	--------------

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm/min, inch/min, degree/min (machine unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the dry run rate at the 100% position on the jog feedrate specification dial. The unit of data depends on the increment system of the reference axis.

1411

Cutting feedrate

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Setting input

[Data type] Real path

[Unit of data] mm/min, inch/min, degree/min (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

A cutting feedrate can be specified with this parameter for a machine which does not have to change the cutting feedrate frequently during machining. This eliminates the need to specify a cutting feedrate (F code) in the NC program.

1420

Rapid traverse rate for each axis

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/min, inch/min, degree/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1430

Maximum cutting feedrate for each axis

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/min, inch/min, degree/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Specify the maximum cutting feedrate for each axis.

1466

Feedrate for retraction in threading cycle G76.7

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm/min, inch/min (machine unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

When threading cycle G76.7 is specified, retraction is performed after threading. Set a feedrate for this retraction.

NOTE

When this parameter is set to 0 or bit 1 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

1474	Feedrate regarded as accumulated pulse 0. (corner control by feedrate (for gas cutting machine))
-------------	---

[Input type] Parameter input
 [Data type] Real axis
 [Unit of data] mm/min, inch/min, deg/min (machine unit)
 [Valid data range] 0 to 32767
 When a cutting feed block (block A) is followed by another cutting feed block (block B), execution proceeds to block B if the feedrate in the automatic acceleration/deceleration circuit for each axis is reduced to the setting of this parameter, and the number of accumulated pulses in the automatic acceleration/deceleration circuit is assumed to be 0. This setting is used for corner control by feedrate (for a gas cutting machine).

1496	Critical angle of automatic exact stop check
-------------	---

[Input type] Parameter input
 [Data type] Word path
 [Unit of data] degree
 [Valid data range] 0 to 179
 Set the critical internal angle of a corner in 1-degree steps to cause an exact stop in the automatic exact stop check. If 0 is set, the automatic exact stop check is disabled.

1497	The amount of Minute block movement of automatic exact stop check
-------------	--

[Input type] Parameter input
 [Data type] Real path
 [Unit of data] mm, inch (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 0 or positive 9 digit of minimum unit of data (Refer to the standard parameter setting table (B))
 (When the increment system is IS-B, 0.0 to +999999.999)
 Set the amount of Minute block movement of automatic exact stop check. If the amounts of movements along the two axes of the currently selected plane in a specified block are both smaller than the setting of this parameter, the automatic exact stop check for that block is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Input type] Parameter input
 [Data type] Bit path

#5 NCI An in-position check:
 0: Confirms that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time and that the machine position has reached a specified position (the servo positional deviation is within the in-position width set by parameter No. 1826).
 1: Confirms only that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time.

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx	JGLx			CTBx	CTLx

[Input type] Parameter input
 [Data type] Bit axis

- #0 **CTLx** Acceleration/deceleration in cutting feed or dry run during cutting feed
 0: Exponential acceleration/deceleration is applied.
 1: Linear acceleration/deceleration after interpolation is applied.

- #1 **CTBx** Acceleration/deceleration in cutting feed or dry run during cutting feed
 0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied.
 (depending on the setting in bit 0 (CTLx) of parameter No. 1610)
 1: Bell-shaped acceleration/deceleration is applied.

- #4 **JGLx** Acceleration/deceleration in jog feed
 0: Exponential acceleration/deceleration is applied.
 1: The same acceleration/deceleration as for cutting feedrate is applied.
 (Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)

- #5 **THLx** Acceleration/deceleration in threading cycles
 0: Exponential acceleration/deceleration is applied.
 1: The same acceleration/deceleration as for cutting feedrate is applied.
 (Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)
 As the time constant and FL rate, however, the settings of parameters Nos. 1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
 [Data type] Bit path

- #0 **CFR** For retraction after threading in the threading cycle G76.7:
 0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL rate (parameter No. 1627).
 1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

NOTE

If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse before look-ahead interpolation and optimum torque acceleration/deceleration are disabled.

1626	Acceleration/deceleration time constant in threading cycles for each axis
------	---

[Input type] Parameter input
 [Data type] Word axis
 [Unit of data] msec
 [Valid data range] 0 to 4000

Set a time constant for acceleration/deceleration after interpolation in the threading cycle G76.7 for each axis.

1627

FL rate for acceleration/deceleration in threading cycles for each axis

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/min, inch/min, degree/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set an FL rate for acceleration/deceleration after interpolation in the threading cycle G76.7 for each axis. Set 0 at all times except in a special case.

1732

Minimum allowable feedrate for the deceleration function based on acceleration in circular interpolation

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm/min, inch/min, degree/min (machine unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

With the deceleration function based on acceleration in circular interpolation, an optimum feedrate is automatically calculated so that acceleration produced by changing the move direction in circular interpolation does not exceed the maximum allowable acceleration rate specified in parameter No. 1735.

If the radius of an arc is very small, a calculated feedrate may become too low.

In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

NOTE

During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1735

Maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation for each axis

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/sec², inch/sec², degree/sec² (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation.

Feedrate is controlled so that acceleration produced by changing the move direction in circular interpolation does not exceed the value specified in this parameter.

For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled.

If a different value is set in this parameter for each axis, a feedrate is determined from the smaller of the acceleration rates specified for the two circular axes.

NOTE
 During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1826	In-position width for each axis
-------------	--

[Input type] Parameter input
 [Data type] 2-word axis
 [Unit of data] Detection unit
 [Valid data range] 0 to 99999999

The in-position width is set for each axis.
 When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

	#7	#6	#5	#4	#3	#2	#1	#0
3115			APLx		NDFx			

[Input type] Parameter input
 [Data type] Bit axis

#3 NDFx In calculation for actual cutting feedrate display, the feedrate of a selected axis is:
 0: Considered.
 1: Not considered.

#5 APLx When the active offset value modification mode based on manual feed is selected, the relative position display is automatically:
 0: Not preset.
 1: Preset.
 Use this parameter when returning a modified offset value to the original value before modification in the active offset value modification mode based on manual feed. The offset value can be returned to the original value by making a movement on the axis by manual feed so that the relative position display (counter) indicates the position 0.

3131	Subscript of axis name
-------------	-------------------------------

[Input type] Parameter input
 [Data type] Byte axis
 [Valid data range] 0 to 9, 65 to 90

In order to distinguish axes under parallel operation, synchronization control, and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than a parallel axis, synchronization control axis, and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a subscript.

[Example] When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

If a multi-path system is used, no extended axis name is used within a path, and no subscript is set for the axis names, then the path number is automatically used as the subscript for the axis names. To disable the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

NOTE
 If even one axis in a path uses an extended axis name when bit 2 (EAS) of parameter No. 11308 is set to 0, subscripts cannot be used for axis names in the path.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Input type] Parameter input
 [Data type] Bit path

- #0 WOF** Setting the tool offset value (tool wear offset) by MDI key input is:
 0: Not disabled.
 1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

NOTE
 When tool offset memory A is selected, the tool offset set in the bit 0 (WOF) of parameter No.3290 is followed.

- #1 GOF** Setting the tool geometry offset value by MDI key input is:
 0: Not disabled.
 1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled
-------------	--

3295	Number of tool offset values (from the start number) whose input by MDI is disabled
-------------	--

[Input type] Parameter input
 [Data type] Word path
 [Valid data range] 0 to 999

When the modification of tool offset values by MDI key input is to be disabled using bits 0 (WOF) and 1 (GOF) of parameter No. 3290, parameters Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No. 3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No. 3295, set the number of such values. In the following cases, however, none of the tool offset values may be modified:

- When 0 or a negative value is set in parameter No. 3294
- When 0 or a negative value is set in parameter No. 3295
- When a value greater than the maximum tool offset number is set in parameter No. 3294

In the following case, a modification to the values ranging from the value set in parameter No. 3294 to the maximum tool offset number is disabled:

- When the value of parameter No. 3294 added to the value of parameter No. 3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI panel, the warning "WRITE PROTECT" is issued.

[Example] When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:

- Bit 1 (GOF) of parameter No. 3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No. 3290 = 1 (to disable tool wear offset value modification)
- Parameter No. 3294 = 51
- Parameter No. 3295 = 10

If the setting of bit 0 (WOF) of parameter No. 3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Input type] Parameter input

[Data type] Bit path

#0 DPI When a decimal point is omitted in an address that can include a decimal point

0: The least input increment is assumed. (Normal decimal point input)

1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point input)

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR			G91	G19	G18	G01

[Input type] Parameter input

[Data type] Bit path

#0 G01 G01 Mode entered when the power is turned on or when the control is cleared

0: G00 mode (positioning)

1: G01 mode (linear interpolation)

#1 G18 Plane selected when power is turned on or when the control is cleared

0: G17 mode (plane XY)

1: G18 mode (plane ZX)

#2 G19 Plane selected when power is turned on or when the control is cleared

0: The setting of bit 1 (G18) of parameter No. 3402 is followed.

1: G19 mode (plane YZ)

When this bit is set to 1, set bit 1 (G18) of parameter No. 3402 to 0.

#3 G91 When the power is turned on or when the control is cleared

0: G90 mode (absolute programming)

1: G91 mode (incremental programming)

#6 CLR Reset button on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal

0: Cause reset state.


1: Cause clear state.

For the reset and clear states, refer to Appendix in the OPERATOR'S MANUAL.

- #7 **G23** When the power is turned on
 - 0: G22 mode (stored stroke check on)
 - 1: G23 mode (stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
3408	C23							

[Input type] Parameter input
 [Data type] Bit

- #7 **C23** If bit 6 (CLR) of parameter No. 3402 is set to 1, set G codes of group number 23 to be placed in the cleared state when the CNC is reset by the  key of the MDI panel, the external reset signal, the reset & rewind signal, or the emergency stop signal. The table below indicates the correspondence between bits and G code groups. The setting of a bit has the following meaning:
 - 0: Places the G code group in the cleared state.
 - 1: Does not place G code group in the cleared state.

3410	Tolerance of arc radius
------	-------------------------

[Input type] Setting input
 [Data type] Real path
 [Unit of data] mm, inch (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
 (When the increment system is IS-B, 0.0 to +999999.999)
 When a circular interpolation command is executed, the tolerance for the radius between the start point and the end point is set.

	#7	#6	#5	#4	#3	#2	#1	#0
3452				GCC				GC0

[Input type] Parameter input
 [Data type] Bit path

- #0 **GC0** When G00 is specified in the mode of groove cutting by continuous circle motion:
 - 0: A P/S alarm is issued.
 - 1: G01 is assumed to have been specified and is executed.
- #4 **GCC** When groove cutting along a path is stopped, continuous circle motion is:
 - 0: Stopped.
 - 1: Continued.

3490	Clamp value of acceleration in continuous circle motion
------	---

[Input type] Parameter input
 [Data type] Real path
 [Unit of data] mm/sec², inch/sec², degree/sec² (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] Refer to the standard parameter setting table (D)
 Feedrate command F for continuous circle movement can be clamped by specifying I and K of G12.4/G13.4 and this parameter.

Clamp feedrate $F = \text{SQR}(\text{parameter No. } 3490 \times (I-K)/2) \times 60$

Continuous circle motion feedrate override is applied to the clamped feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
5000				ASG			MOF	

[Input type] Setting input

[Data type] Bit path

- #1 MOF** When the tool length compensation shift type (bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400 is set to 1) is used, if the tool length compensation amount is changed^(NOTE 3) in the tool length compensation mode^(NOTE 1) when look-ahead blocks are present^(NOTE 2):
- 0: Compensation is performed for the change in compensation amount as the movement type.
- 1: Compensation is not performed for the change until a tool length compensation command (offset number) and an absolute programming for the compensation axis are specified.

NOTE

- 1 The tool length compensation mode refers to the following state:
 - Tool length offset (G43/G44)
 - Tool length compensation in tool axis direction (G43.1)
 - Tool center point control (G43.4/G43.5)
- 2 "When look-ahead blocks are present" means as follows:
 - The modal G code of the G codes (such as tool radius - tool nose radius compensation) of group 07 is other than G40.
 - In the smooth interpolation (G05.1Q2) mode
 One look-ahead block during automatic operation and multiple look-ahead blocks in the AI contour control mode are not included in the state "when look-ahead blocks are present".
- 3 Changes in tool length compensation amount are as follows:
 - When the tool length compensation number is changed by H code (or D code for the extended tool selection function for lathe systems)
 - When G43 or G44 is specified to change the direction of tool length compensation
 - When the tool length compensation amount is changed using the offset screen, G10 command, system variable, PMC window, and so forth during automatic operation if bit 1 (EVO) of parameter No. 5001 is set to 1.

- #4 ASG** When tool compensation memory B/C is valid, the compensation amount to be modified by the active offset value change mode based on manual feed is:
- 0: Geometry compensation value
- 1: Wear compensation value

NOTE

This parameter is valid when the option for tool compensation memory B/C is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO		EVR	TAL		TLB	TLC

[Input type] Parameter input
 [Data type] Bit path

#0 TLC

#1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.

Tool length compensation A : Z-axis at all times

Tool length compensation B : Axis perpendicular to a specified plane (G17/G18/G19)

Tool length compensation C : Axis specified in a block that specifies G43/G44

#3 TAL Tool length compensation C

0: Generates an alarm when two or more axes are offset

1: Not generate an alarm even if two or more axes are offset

#4 EVR When a tool compensation value is changed in tool radius - tool nose radius compensation mode:

0: Enables the change, starting from that block where the next D or H code is specified.

1: Enables the change, starting from that block where buffering is next performed.

#6 EVO If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):

0: The new value becomes valid in a block where G43, G44, or an H code is specified next.

1: The new value becomes valid in a block where buffering is performed next.

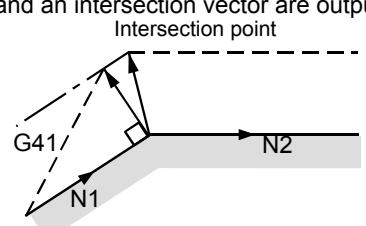
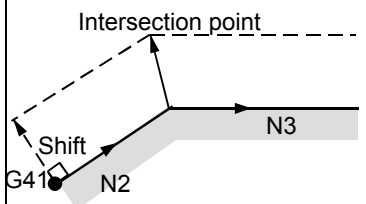
	#7	#6	#5	#4	#3	#2	#1	#0
5003							SUV	SUP

[Input type] Parameter input
 [Data type] Bit path

#0 SUP

#1 SUV These bits are used to specify the type of startup/cancellation of tool radius - tool nose radius compensation.

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output.

SUV	SUP	Type	Operation
0	1	Type B	<p>A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output.</p>  <p>Tool nose radius center path / Tool center path</p> <p>Programmed path</p>
1	0 1	Type C	<p>When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.</p>  <p>Tool nose radius center path / Tool center path</p> <p>Programmed path</p> <p>When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.</p>

NOTE
 When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS16i-T is performed.

5005	#7	#6	#5	#4	#3	#2	#1	#0
			QNI					

[Input type] Parameter input
 [Data type] Bit path

- #5 QNI** With the tool length measurement function or the function for direct input of offset value measured B, a tool compensation number is selected by:
- 0: Operation through the MDI panel by the operator (selection based on cursor operation).
 - 1: Signal input from the PMC.

5006	#7	#6	#5	#4	#3	#2	#1	#0
		TOS						

[Input type] Parameter input
 [Data type] Bit

- #6 TOS** Set a tool length compensation or tool offset operation.
- 0: Tool length compensation or tool offset operation is performed by an axis movement.
 - 1: Tool length compensation or tool offset operation is performed by shifting the coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
5007				WMH	WMA	TMA	TC3	TC2

[Input type] Parameter input

[Data type] Bit path

#0 TC2

#1 TC3 If a tool length compensation value is set by pressing the [MEASURE] or [+MEASURE] soft key in tool length measurement, the tool automatically moves to the tool change position. Specify at which reference position the tool change position is located.

TC3	TC2	Meaning
0	0	The tool change position is at the first reference position.
0	1	The tool change position is at the second reference position.
1	0	The tool change position is at the third reference position.
1	1	The tool change position is at the fourth reference position.

#2 TMA 0: Tool length measurement is enabled along the Z-axis only.

1: Tool length measurement is enabled along each axis.

#3 WMA 0: Surface-based measurement of a workpiece zero point offset value is enabled along the Z-axis only.

1: Surface-based measurement of a workpiece zero point offset value is enabled along each axis.

#4 WMH 0: Hole-based measurement of a workpiece zero point offset value is disabled.

1: Hole-based measurement of a workpiece zero point offset value is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5008					CNV		CNC	

[Input type] Parameter input

[Data type] Bit path

#1 CNC

#3 CNV These bits are used to select an interference check method in the tool radius - tool nose radius compensation mode.

CNV	CNC	Operation
0	0	Interference check is enabled. The direction and the angle of an arc are checked.
0	1	Interference check is enabled. Only the angle of an arc is checked.
1	-	Interference check is disabled.

For the operation taken when the interference check shows the occurrence of an reference (overcutting) , see the description of bit 5 (CAV) of parameter No. 19607.

NOTE

Checking of only the direction cannot be set.

5010	Limit for ignoring the small movement resulting from tool radius - tool nose radius compensation
------	--

[Input type] Setting input

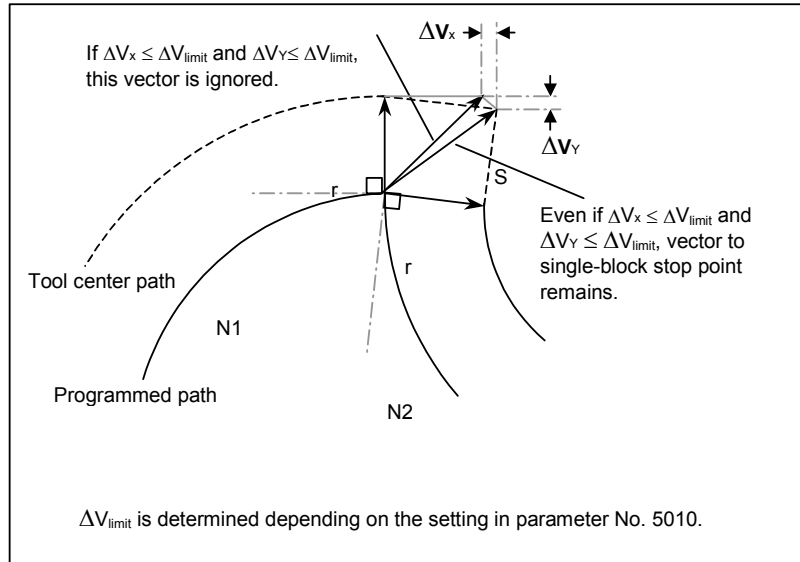
[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

When the tool moves around a corner in cutter compensation or tool nose radius compensation mode, the limit for ignoring the small travel amount resulting from compensation is set. This limit eliminates the interruption of buffering caused by the small travel amount generated at the corner and any change in feedrate due to the interruption.



5011

Constant denominator for 3-dimensional tool compensation or tool length compensation in a specified direction

[Input type] Setting input
 [Data type] Real path
 [Unit of data] mm, inch (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
 (When the increment system is IS-B, -999999.999 to +999999.999)
 This parameter sets the value of p in the expressions used for finding a 3-dimensional tool compensation vector:

$$Vx = i \times r / p$$

$$Vy = j \times r / p$$

$$Vz = k \times r / p$$

where,

- Vx, Vy, Vz : Components of a 3-dimensional tool compensation vector along the X-axis, Y-axis, and Z-axis, or their parallel axes
- i, j, k : Values specified in addresses I, J, and K in the program
- r : Compensation value
- p : Value set in this parameter

When 0 is set in this parameter, the following is assumed:

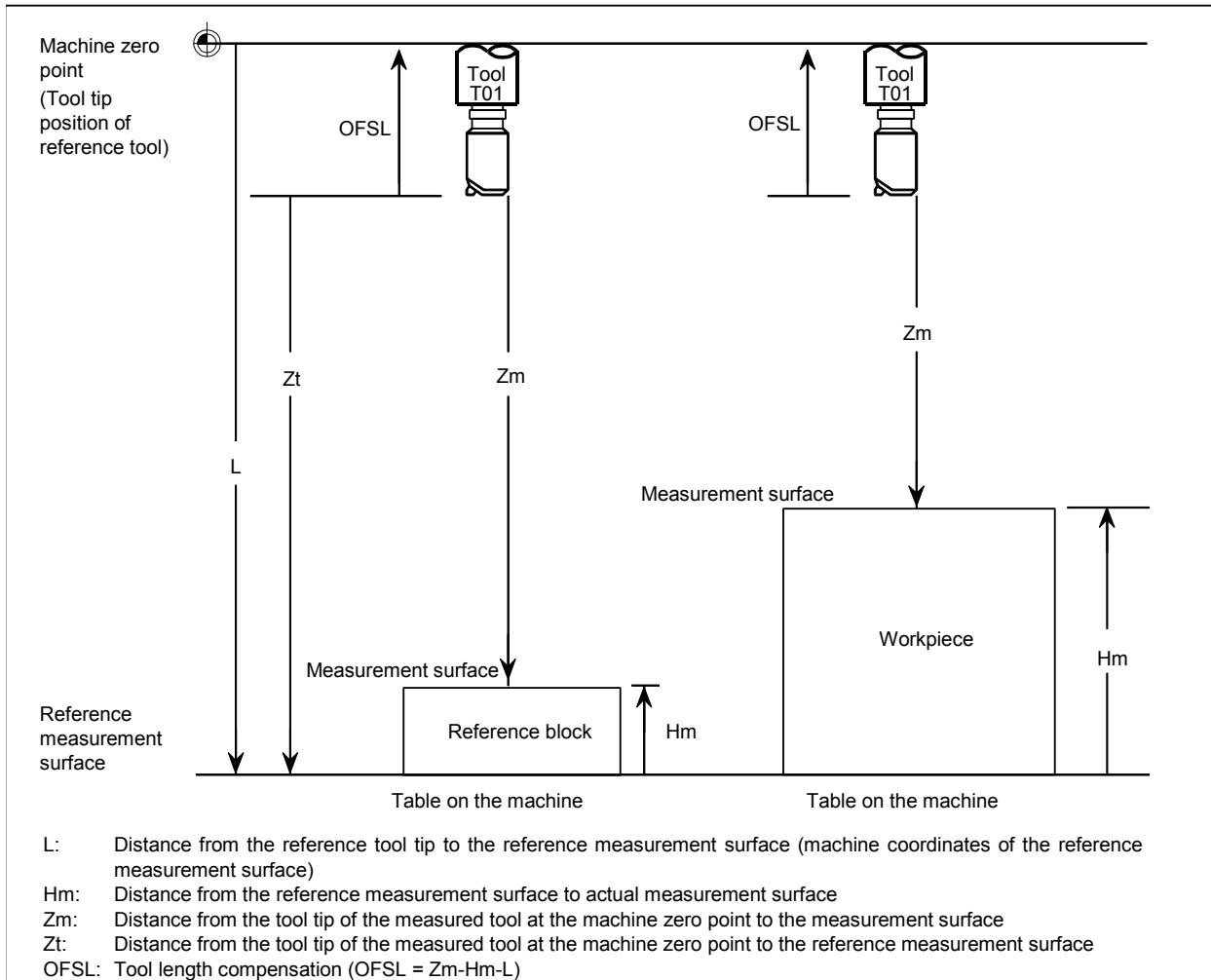
$$p = \sqrt{I^2 + J^2 + K^2}$$

5022

Distance (L) from reference tool tip position to the reference measurement surface

[Input type] Parameter input
 [Data type] Real axis
 [Unit of data] mm, inch (machine unit)
 [Min. unit of data] Depend on the increment system of the applied axis
 [Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
 (When the increment system is IS-B, -999999.999 to +999999.999)

For each axis, this parameter sets the distance from the reference tool tip position to the reference measurement surface when the machine is at the machine zero point.



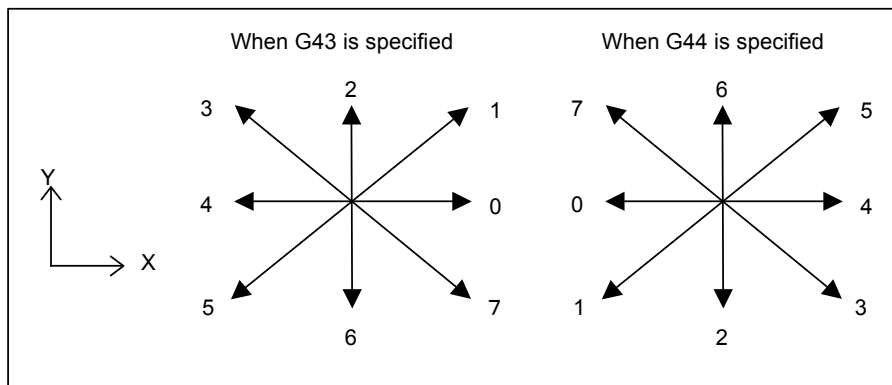
5032	Direction of tool offset B
-------------	-----------------------------------

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to 7

Specify the offset direction of tool offset B (G43, G44).



Setting value of parameter No. 5032	Offset direction	
	G43	G44
0	X+a	X-a
1	X+a Y+a	X-a Y-a
2	Y+a	Y-a
3	X-a Y+a	X+a Y-a
4	X-a	X+a
5	X-a Y-a	X+a Y+a
6	Y-a	Y+a
7	X+a Y-a	X-a Y+a

a : Offset value set to offset memory number specified by H code

	#7	#6	#5	#4	#3	#2	#1	#0
5033								GOB

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Bit path

#0 GOB The tool offset B function (for a gas cutting machine) is:

0: Disabled.

1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5041		AON						

[Input type] Parameter input

[Data type] Bit path

#6 AON When the tool length compensation value used with tool length compensation A/B is changed in the active offset value modification mode:

0: The change becomes effective starting with the next block specifying G43, G44, or an H code.

1: The change becomes effective starting with the next block to be buffered.

NOTE

1 This parameter is valid when bit 6 (EVO) of parameter No. 5001 is set to 0.

2 The operation of this parameter set to 1 is valid even if a new compensation value is further changed by MDI input or a G10 command before the new compensation value becomes effective.

3 The operation of this parameter set to 1 is invalid if a reset operation is performed before a new compensation value becomes effective.

	#7	#6	#5	#4	#3	#2	#1	#0
5042					OFE	OFD	OFC	OFA

[Input type] Parameter input

[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

- #0 **OFA**
- #1 **OFC**
- #2 **OFD**
- #3 **OFE** These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFE	OFD	OFC	OFA	Unit	Valid data 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

For inch input

OFE	OFD	OFC	OFA	Unit	Valid data 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

5081	1st-axis coordinate value of compensation center 1 in grinding-wheel wear compensation
5082	2nd-axis coordinate value of compensation center 1 in grinding-wheel wear compensation
5083	1st-axis coordinate value of compensation center 2 in grinding-wheel wear compensation
5084	2nd-axis coordinate value of compensation center 2 in grinding-wheel wear compensation
5085	1st-axis coordinate value of compensation center 3 in grinding-wheel wear compensation
5086	2nd-axis coordinate value of compensation center 3 in grinding-wheel wear compensation

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value (in the workpiece coordinate system) of the center of compensation in grinding-wheel wear compensation.

	#7	#6	#5	#4	#3	#2	#1	#0
5101								FX Y

[Input type] Parameter input

[Data type] Bit path

#0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:

- 0: In case of the Drilling canned cycle:
Z-axis at all times.
- In case of the Grinding canned cycle:
G75,G77 command :Y-axis
G78,G79 command :Z-axis
- 1: Axis selected by the program

	#7	#6	#5	#4	#3	#2	#1	#0
5102						QSR		

[Input type] Parameter input
[Data type] Bit path

#2 QSR Before a multiple repetitive canned cycle (G70.7 to G73.7) is started, a check to see if the program contains a block that has the sequence number specified in address Q is:

- 0: Not made.
- 1: Made.

When 1 is set in this parameter and the sequence number specified in address Q is not found, the alarm PS0063, “THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND” is issued and the canned cycle is not executed.

	#7	#6	#5	#4	#3	#2	#1	#0
5104						FCK		

[Input type] Parameter input
[Data type] Bit path

#2 FCK In a multiple repetitive canned cycle (G71.7, G72.7), the machining profile is:

- 0: Not checked.
- 1: Checked.

The target figure specified by G71.7 or G72.7 is checked for the following before machining operation:

- If the start point of the canned cycle is less than the maximum value of the machining profile even when the plus sign is specified for a finishing allowance, the alarm PS0322, “FINISHING SHAPE WHICH OVER OF STARTING POINT” is issued.
- If the start point of the canned cycle is greater than the minimum value of the machining profile even when the minus sign is specified for a finishing allowance, the alarm PS0322 is issued.
- If an unmonotonous command of type I is specified for the axis in the cutting direction, the alarm PS0064, “THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)” or alarm PS0329, “THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)” is issued.
- If an unmonotonous command is specified for the axis in the roughing direction, the alarm PS0064 or alarm PS0329 is issued.
- If the program does not include a block that has a sequence number specified by address Q, the alarm PS0063, “THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND” is issued. This check is made, regardless of bit 2 (QSR) of parameter No. 5102.

- If a command (G41/G42) on the blank side in tool nose radius compensation is inadequate, the alarm PS0328, “ILLEGAL WORK POSITION IS IN THE TOOL NOSE RADIUS COMPENSATION” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5105						RF2	RF1	SBC

[Input type] Parameter input

[Data type] Bit path

#0 SBC In a drilling canned cycle, chamfer cycle, or corner rounding cycle:

0: A single block stop is not performed.

1: A single block stop is performed.

#1 RF1 In a multiple repetitive canned cycle (G71.7, G72.7) of type I, roughing is:

0: Performed.

1: Not performed.

NOTE

When a roughing allowance ($\Delta i/\Delta k$) is specified using the Series 15 program format, roughing is performed, regardless of the setting of this parameter.

#2 RF2 In a multiple repetitive canned cycle (G71.7, G72.7) of type II, roughing is:

0: Performed.

1: Not performed.

NOTE

When a roughing allowance ($\Delta i/\Delta k$) is specified using the Series 15 program format, roughing is performed, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5107							ASC	ASU

[Input type] Parameter input

[Data type] Bit path

#0 ASU For G71.7 or G72.7, movement to the last turning start position is performed by:

0: Cutting feed.

1: Rapid traverse.

For two-cycle operation to move toward the current turning start position, this parameter selects the feed in the first cycle (movement to the last turning start position). The feed in the second cycle (movement from the last turning start position to the current turning start position) follows the feed in the first block of the shape program.

This parameter is valid to both of type-I and type-II commands.

#1 ASC The G71.7/G72.7 TYPE1 commands execute the movement toward the current turning start position in:

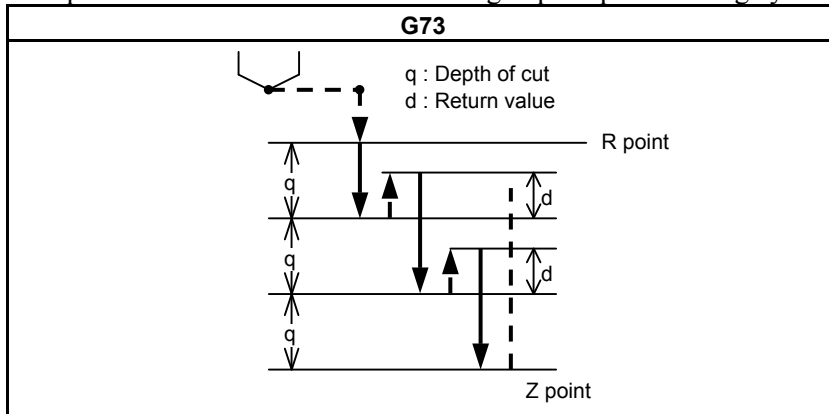
0: Two cycles.

1: One cycle.

You can change the two-cycle operation to move to the current turning start position from two cycles to one cycle. The feed mode follows the mode (G00, G01) in the first block of the shape program. This parameter is valid only to type-I commands.

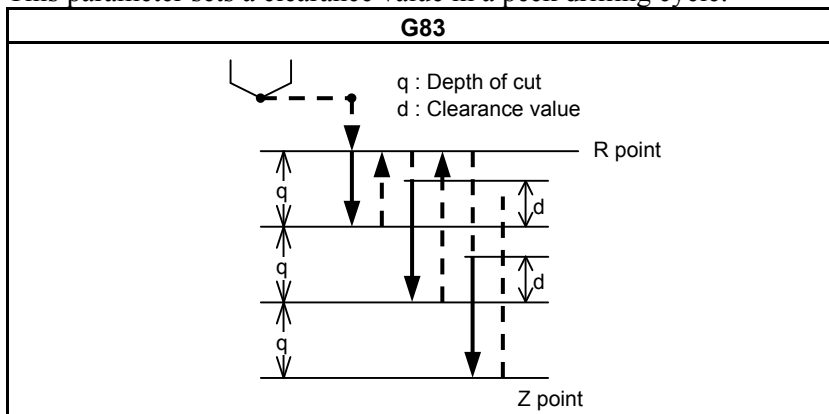
5114 **Return value of high-speed peck drilling cycle**

- [Input type] Parameter input
 - [Data type] Real path
 - [Unit of data] mm, inch (input unit)
 - [Min. unit of data] Depend on the increment system of the reference axis
 - [Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
- This parameter sets the return value in high-speed peck drilling cycle.



5115 **Clearance value in a peck drilling cycle**

- [Input type] Parameter input
 - [Data type] Real path
 - [Unit of data] mm, inch (input unit)
 - [Min. unit of data] Depend on the increment system of the reference axis
 - [Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
- This parameter sets a clearance value in a peck drilling cycle.



5130 **Cutting value (chamfering value) in thread cutting cycle G76.7**

- [Input type] Parameter input
- [Data type] Byte path
- [Unit of data] 0.1
- [Valid data range] 0 to 127

This parameter sets a cutting value (chamfering value) in the thread cutting cycle (G76.7) of a multiple repetitive canned cycle.

Let L be a lead. Then, a cutting value range from 0.1L to 12.7L is allowed.

To specify a cutting value of 10.0L, for example, specify 100 in this parameter.

5131

Cutting angle in thread cutting cycle G76.7

[Input type] Parameter input

[Data type] Byte path

[Unit of data] Degree

[Valid data range] 1 to 89

This parameter sets a thread cutting angle in a thread cutting cycle (G76.7).

When 0 is set, an angle of 45 degrees is specified.

5132

Depth of cut in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the depth of cut in multiple repetitive canned cycles G71.7 and G72.7.

This parameter is not used with the Series 15 program format.

NOTE

Specify a radius value at all times.

5133

Escape in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the escape in multiple repetitive canned cycles G71.7 and G72.7.

NOTE

Specify a radius value at all times.

5134

Clearance value in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets a clearance value up to the cutting feed start point in multiple repetitive canned cycles (G71.7/G72.7).

NOTE

Specify a radius value at all times.

5135**Retraction distance in the multiple repetitive canned cycle G73.7 (second axis on the plane)**

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a retraction distance along the second axis on the plane in the multiple repetitive canned cycle G73.7. This parameter is not used with the Series 15 program format.

NOTE

Specify a radius value at all times.

5136**Retraction distance in the multiple repetitive canned cycle G73.7 (first axis on the plane)**

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a retraction distance along the first axis on the plane in the multiple repetitive canned cycle G73.7. This parameter is not used with the Series 15 program format.

NOTE

Specify a radius value at all times.

5137**Number of divisions in the multiple repetitive canned cycle G73.7**

[Input type] Parameter input

[Data type] 2-word path

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the number of divisions in the multiple repetitive canned cycle G73.7. This parameter is not used with the Series 15 program format.

5139**Return in multiple repetitive canned cycles G74.7 and G75.7**

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
 (When the increment system is IS-B, 0.0 to +999999.999)
 This parameter sets the return in multiple repetitive canned cycles G74.7 and G75.7.

NOTE

Specify a radius value at all times.

5140**Minimum depth of cut in the multiple repetitive canned cycle G76.7**

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
 (When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets a minimum depth of cut in the multiple repetitive canned cycle G76.7 so that the depth of cut does not become too small when the depth of cut is constant.

NOTE

Specify a radius value at all times.

5141**Finishing allowance in the multiple repetitive canned cycle G76.7**

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
 (When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the finishing allowance in multiple repetitive canned cycle G76.7.

NOTE

Specify a radius value at all times.

5142**Repetition count of final finishing in multiple repetitive canned cycle G76.7**

[Input type] Parameter input

[Data type] 2-word path

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the number of final finishing cycle repeats in the multiple repetitive canned cycle G76.7.

When 0 is set, only one final finishing cycle is executed.

5143**Tool nose angle in multiple repetitive canned cycle G76.7**

[Input type] Parameter input

[Data type] Byte path

[Unit of data] Degree

[Valid data range] 0, 29, 30, 55, 60, 80

This parameter sets the tool nose angle in multiple repetitive canned cycle G76.7.
This parameter is not used with the Series 15 program format.

5145

Allowable value 1 in multiple repetitive canned cycles G71.7 and G72.7

- [Input type] Parameter input
 [Data type] Real path
 [Unit of data] mm, inch (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
 (When the increment system is IS-B, 0.0 to +999999.999)
 If a monotonous command of type I or II is not specified for the axis in the roughing direction, the alarm PS0064, "THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)" or alarm PS0329, "THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)" is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71.7 and G72.7 cycles can be executed even in a program including an unmonotonous figure.
- [Example] Suppose that a G71 or G71.7 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the plus direction along the Z-axis is specified in a target figure program, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

NOTE

A check for a monotonous figure is made at all times during G71.7 and G72.7 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71.7 or G72.7 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked.
 Note that no alarm is issued when an allowable value is set.
 Use a radius value to set this parameter at all times.

5146

Allowable value 2 in multiple repetitive canned cycles G71.7 and G72.7

- [Input type] Parameter input
 [Data type] Real path
 [Unit of data] mm, inch (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 0 to cut of depth
 If a monotonous command of type I is not specified for the axis in the cutting direction, the alarm PS0064, "THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)" or PS0329, "THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)" is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71.7 and G72.7 cycles can be executed even in a program including an unmonotonous figure.
 The allowable value is clamped to the depth of cut specified by a multiple repetitive canned cycle.

[Example] Suppose that a G71.7 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the minus direction along the X-axis is specified in a target figure program for moving from the bottom of cutting to the end point, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

NOTE
 A check for a monotonous figure is made at all times during G71.7 and G72.7 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71.7 or G72.7 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked.
 Note that no alarm is issued when an allowable value is set.
 Use a radius value to set this parameter at all times.

5148	Tool retraction direction after orientation in a fine boring cycle or back boring cycle
-------------	--

[Input type] Parameter input
 [Data type] Byte axis
 [Valid data range] -24 to 24
 This parameter sets an axis and direction for tool retraction after spindle orientation in a fine boring cycle or back boring cycle. For each boring axis, an axis and direction for tool retraction after orientation can be set. Set a signed axis number.

[Example] Suppose that:
 When the boring axis is the X-axis, the tool retraction direction after orientation is -Y.
 When the boring axis is the Y-axis, the tool retraction direction after orientation is +Z.
 When the boring axis is the Z-axis, the tool retraction direction after orientation is -X.
 Then, set the following (assuming that the first, second, and third axes are the X-axis, Y-axis, and Z-axis, respectively):
 Set -2 in the parameter for the first axis. (The tool retraction direction is -Y.)
 Set 3 in the parameter for the second axis. (The tool retraction direction is +Z.)
 Set -1 in the parameter for the third axis. (The tool retraction direction is -X.)
 Set 0 for other axes.

	#7	#6	#5	#4	#3	#2	#1	#0
5160						NOL	OLS	

[Input type] Parameter input
 [Data type] Bit path

- #1 OLS** When an overload torque detection signal is received in a peck drilling cycle of a small diameter, the feedrate and spindle speed are:
 0: Not changed.
 1: Changed.

- #2 NOL** When the depth of cut per action is satisfied although no overload torque detection signal is received in a peck drilling cycle of a small diameter, the feedrate and spindle speed are:
 0: Not changed.
 1: Changed.

5163**M code that specifies the peck drilling cycle mode of a small diameter**

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 1 to 99999999

This parameter sets an M code that specifies the peck drilling cycle mode of a small diameter.

5164**Percentage of the spindle speed to be changed at the start of the next advancing after an overload torque detection signal is received**

[Input type] Parameter input

[Data type] Word path

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted because the overload torque detection signal is received.

$$S2 = S1 \times d1 \div 100$$

S1: Spindle speed to be changed

S2: Spindle speed changed

Set d1 as a percentage.

NOTE

When 0 is set, the spindle speed is not changed.

5165**Percentage of the spindle speed to be changed at the start of the next advancing when no overload torque detection signal is received**

[Input type] Parameter input

[Data type] Word path

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted without the overload torque detection signal received.

$$S2 = S1 \times d2 \div 100$$

S1: Spindle speed to be changed

S2: Spindle speed changed

Set d2 as a percentage.

NOTE

When 0 is set, the spindle speed is not changed.

5166**Percentage of the cutting feedrate to be changed at the start of the next cutting after an overload torque detection signal is received**

[Input type] Parameter input

[Data type] Word path

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances because the overload torque detection signal is received.

$$F2 = F1 \times b1 \div 100$$

F1: Cutting feedrate to be changed

F2: Cutting feedrate changed

Set b1 as a percentage.

NOTE

When 0 is set, the cutting feedrate is not changed.

5167

Percentage of the cutting feedrate to be changed at the start of the next cutting when no overload torque detection signal is received

[Input type] Parameter input

[Data type] Word path

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances without the overload torque detection signal received.

$$F2 = F1 \times b2 \div 100$$

F1: Cutting feedrate to be changed

F2: Cutting feedrate changed

Set b2 as a percentage.

NOTE

When 0 is set, the cutting feedrate is not changed.

5168

Lower limit of the percentage of the cutting feedrate in a peck drilling cycle of a small diameter

[Input type] Parameter input

[Data type] Byte path

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the lower limit of the percentage of the cutting feedrate changed repeatedly to the specified cutting feedrate.

$$FL = F \times b3 \div 100$$

F: Specified cutting feedrate

FL: Changed cutting feedrate

Set b3 as a percentage.

5170

Number of the macro variable to which to output the total number of retractions during cutting

[Input type] Parameter input

[Data type] Word path

[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted during cutting. The total number cannot be output to common variables #500 to #599.

5171

Number of the macro variable to which to output the total number of retractions because of the reception of an overload torque detection signal

[Input type] Parameter input

[Data type] Word path

[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted after the overload torque detection signal is received during cutting. The total number cannot be output to common variables #500 to #599.

5172

Feedrate of retraction to point R when no address I is specified

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm/min, inch/min (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets the feedrate of retraction to point R when no address I is specified.

5173

Feedrate of advancing to the position just before the bottom of a hole when no address I is specified

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm/min, inch/min (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets the feedrate of advancing to the position just before the bottom of a previously machined hole when no address I is specified.

5174

Clearance in a peck drilling cycle of a small diameter

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the clearance in a peck drilling cycle of a small diameter.

5176

Grinding axis number in Plunge Grinding Cycle(G75)

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes

Set the Grinding axis number of Plunge Grinding Cycle(G75).

NOTE

The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5177

Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle(G77)

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle (G77).

NOTE

The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5178

Grinding axis number of Continuous feed surface grinding cycle(G78)

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Continuous feed surface grinding cycle(G78).

NOTE

The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5179

Grinding axis number of Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Intermittent feed surface grinding cycle(G79).

NOTE

The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5180

Axis number of dressing axis in Plunge grinding cycle(G75)

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Plunge grinding cycle(G75).

NOTE

The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5181**Axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77)**

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77).

NOTE

The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5182**Axis number of dressing axis in Continuous feed surface grinding cycle(G78)**

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Continuous feed surface grinding cycle(G78).

NOTE

The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5183**Axis number of dressing axis in Intermittent feed surface grinding cycle(G79)**

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Intermittent feed surface grinding cycle(G79).

NOTE
 The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5200		FHD	PCP	DOV				G84

[Input type] Parameter input
 [Data type] Bit path

- #0 G84** Method for specifying rigid tapping:
 0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No. 5210).
 1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

- #4 DOV** Override during extraction in rigid tapping:
 0: Invalidated
 1: Validated (The override value is set in parameter No. 5211. However, set an override value for rigid tapping return in parameter No. 5381.)

- #5 PCP** Rigid tapping:
 0: Used as a high-speed peck tapping cycle
 1: Not used as a high-speed peck tapping cycle

- #6 FHD** Feed hold and single block in rigid tapping:
 0: Invalidated
 1: Validated

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU			

[Input type] Parameter input
 [Data type] Bit path

- #3 OVU** The increment unit of the override parameter No. 5211 for tool rigid tapping extraction is:
 0: 1%
 1: 10%

- #4 OV3** A spindle speed for extraction is programmed, so override for extraction operation is:
 0: Disabled.
 1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

[Input type] Parameter input
 [Data type] Bit path

#4 OVS In rigid tapping, override by the feedrate override select signal and cancellation of override by the override cancel signal is:

0: Disabled.

1: Enabled.

When feedrate override is enabled, extraction override is disabled.

The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

5211	Override value during rigid tapping extraction
-------------	---

[Input type] Parameter input

[Data type] Word path

[Unit of data] 1% or 10%

[Valid data range] 0 to 200

The parameter sets the override value during rigid tapping extraction.

NOTE
 The override value is valid when bit 4 (DOV) of parameter No. 5200 is set to 1. When bit 3 (OVU) of parameter No. 5201 is set to 1, the unit of set data is 10%. An override of up to 200% can be applied to extraction.

5213	Return in peck rigid tapping cycle
-------------	---

[Input type] Setting input

[Data type] Real path

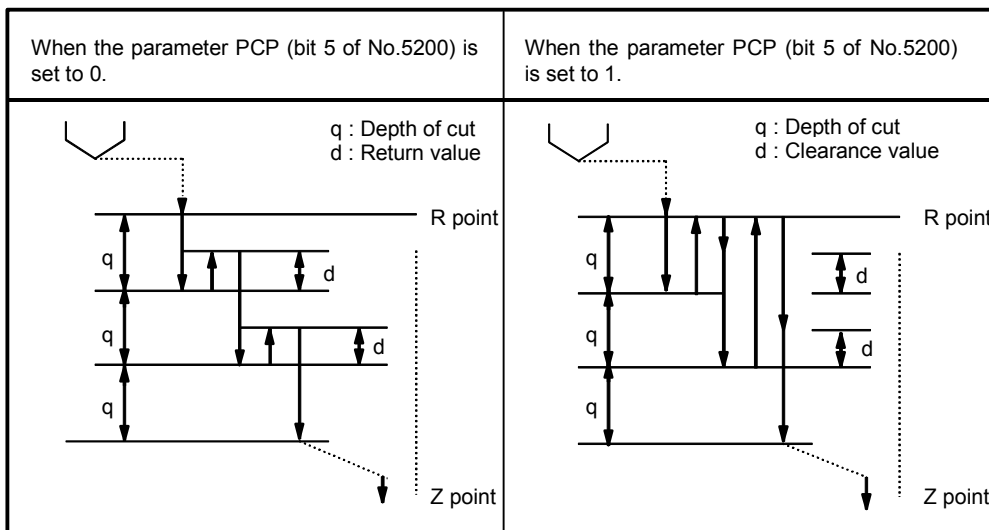
[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the drilling axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the return or clearance in the peck tapping cycle.



5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)

[Input type] Parameter input
 [Data type] 2-word spindle
 [Unit of data] min⁻¹
 [Valid data range] 0 to 9999

Spindle position coder gear ratio
 1 : 1 0 to 7400
 1 : 2 0 to 9999
 1 : 4 0 to 9999
 1 : 8 0 to 9999

Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.
 Set the same value for both parameter No. 5241 and parameter No. 5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200, "ILLEGAL S CODE COMMAND" will be issued. This applies to the M series.

5321	Spindle backlash in rigid tapping (first-stage gear)
5322	Spindle backlash in rigid tapping (second-stage gear)
5323	Spindle backlash in rigid tapping (third-stage gear)

[Input type] Parameter input
 [Data type] Word spindle
 [Unit of data] Detection unit
 [Valid data range] -9999 to 9999

Each of these parameters is used to set a spindle backlash.

	#7	#6	#5	#4	#3	#2	#1	#0
5400								RIN

[Input type] Parameter input
 [Data type] Bit path

#0 RIN Coordinate rotation angle command (R) :
 0: Specified by an absolute method
 1: Specified by an absolute method (G90) or incremental method (G91)

5410	Angular displacement used when no angular displacement is specified for coordinate system rotation
------	--

[Input type] Setting input
 [Data type] 2-word path
 [Unit of data] 0.001 degree
 [Valid data range] -360000 to 360000

This parameter sets the angular displacement for coordinate system rotation. When the angular displacement for coordinate system rotation is not specified with address R in the block where G68 is specified, the setting of this parameter is used as the angular displacement for coordinate system rotation.

	#7	#6	#5	#4	#3	#2	#1	#0
5431								MDL

[Input type] Parameter input

[Data type] Bit path

NOTE

When this parameter is set, the power must be turned off before operation is continued.

#0 MDL The G60 code (single direction positioning) is:

0: One-shot G code (group 00).

1: Modal G code (group 01).

5480	Number of the axis for controlling the normal direction
------	---

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 1 to the maximum controlled axis number

This parameter sets the controlled axis number of the axis for controlling the normal direction.

5481	Feedrate of rotation of the normal direction controlled axis
------	--

[Input type] Parameter input

[Data type] Real axis

[Unit of data] deg/min

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (C)

This parameter sets the feedrate of the movement along the normal direction controlled axis that is inserted at the start point of a block during normal direction control.

5482	Limit value used to determine whether to ignore the rotation insertion of the normal direction controlled axis
------	--

[Input type] Parameter input

[Data type] Real path

[Unit of data] Degree

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

The rotation block of the normal direction controlled axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting.

The ignored rotation angle is added to the next rotation insertion angle, and the block insertion is then judged.

NOTE

1 No rotation block is inserted when 360 or more degrees are set.

2 If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation setting is 180 or more degrees.

	#7	#6	#5	#4	#3	#2	#1	#0
5484								SDC

[Input type] Parameter input
 [Data type] Bit path

#0 SDC Gentle normal direction control function is:
 0: Disabled.
 1: Enabled.

5485	Limit for single-block rotation by the gentle normal direction control function
------	---

[Input type] Parameter input
 [Data type] Real path
 [Unit of data] degree
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 0 to 360

This parameter is valid when bit 0 (SDC) of parameter No. 5484 is set to 1.
 Gentle normal direction control rotates a normal direction control axis simultaneously with an X/Y-axis move block when the calculated rotation insertion angle is smaller than this parameter setting. If it is greater than the setting, the normal direction control axis is rotated with a single block.

NOTE

- 1 This parameter is valid only when the normal direction control axis is rotated according to the settings of parameters Nos. 5482 and 5483.
- 2 If this parameter is set to 360 or greater, the setting is assumed to be 360, and the normal direction control axis is rotated simultaneously with the X/Y-axis move block.
- 3 If this parameter is set to 180 or greater, the normal direction control axis is rotated simultaneously with the X/Y-axis move block unless circular interpolation is used.
- 4 If this parameter is set to a negative value, the setting is assumed to be 0, and the normal direction control axis is rotated with the single block.

5486	Block distance to move until the end of rotation of the normal direction control axis
------	---

[Input type] Parameter input
 [Data type] Real path
 [Unit of data] mm, inch (input unit)
 [Min. unit of data] Depend on the increment system of the reference axis
 [Valid data range] 0 or positive 9 digit of minimum unit of data

This parameter sets the command block distance to move until the rotation of the normal direction control axis is completed. When 0 is specified, this function is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5500		SIM		G90	INC	ABS	REL	

[Input type] Parameter input
 [Data type] Bit path

- #1 REL** The position display of the index table indexing axis in the relative coordinate system is:
 0: Not rounded by one rotation.
 1: Rounded by one rotation.
- #2 ABS** The position display of the index table indexing axis in the absolute coordinate system is:
 0: Not rounded by one rotation.
 1: Rounded by one rotation.
- #3 INC** When the M code that specifies rotation in the negative direction (parameter No. 5511) is not set, rotation in the G90 mode is:
 0: Not set to the shorter way around the circumference.
 1: Set to the shorter way around the circumference.
- #4 G90** A command for the index table indexing axis is:
 0: Assumed to be an absolute or incremental programming according to the mode.
 1: Always assumed to be an absolute programming.
- #6 SIM** When the same block includes a command for the index table indexing axis and a command for another controlled axis:
 0: The setting of bit 0 (IXSx) of parameter No. 5502 is followed.
 1: The commands are executed.

NOTE

Even when this parameter is set to 1, an alarm PS1564, "INDEX TABLE AXIS - OTHER AXIS SAME TIME" is issued if the block is neither G00, G28, nor G30 (or the G00 mode).

	#7	#6	#5	#4	#3	#2	#1	#0
5501								ITI

[Input type] Parameter input

[Data type] Bit path

- #0 ITI** The index table indexing function is:
 0: Enabled.
 1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5502								IXSx

[Input type] Parameter input

[Data type] Bit axis

- #0 IXSx** When a command is specified in a block that contains a command for the index table indexing axis:
 0: An alarm PS1564, "INDEX TABLE AXIS - OTHER AXIS SAME TIME" is issued.
 1: The command is executed.

If bit 6 (SIM) of parameter No. 5500 is set to 1, a simultaneous operation with all axes except the index table indexing axis can be performed regardless of the setting of this parameter.

To set an axis that allows simultaneous operation for each axis, set SIM to 0, and set this parameter.

NOTE

Even when this parameter is set to 1, an alarm PS1564, "INDEX TABLE AXIS - OTHER AXIS SAME TIME" is issued if the block is neither G00, G28, nor G30 (or the G00 mode).

5511**M code that specifies rotation in the negative direction for index table indexing**

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 99999999

0: The rotation direction for the index table indexing axis is determined according to the setting of bit 3 (INC) of parameter No. 5500 and a command.

1 to 99999999:

The rotation for the index table indexing axis is always performed in the positive direction. It is performed in the negative direction only when a move command is specified together with the M code set in this parameter.

NOTE

Be sure to set bit 2 (ABS) of parameter No. 5500 to 1.

5512**Minimum positioning angle for the index table indexing axis**

[Input type] Parameter input

[Data type] Real path

[Unit of data] deg

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the minimum positioning angle (travel distance) for the index table indexing axis. The travel distance specified in the positioning command must always be an integer multiple of this setting. When 0 is set, the travel distance is not checked.

The minimum positioning angle is checked not only for the command, but also for the coordinate system setting and workpiece origin offset.

5610**Limit of initial permissible error during involute interpolation**

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the allowable limit of deviation between an involute curve passing through a start point and an involute curve passing through an end point for an involute interpolation command.

5620**Lower override limit in automatic feedrate control during involute interpolation**

[Input type] Parameter input

[Data type] Byte path

[Unit of data] %

[Valid data range] 0 to 100

In "override in the cutter compensation mode" under involute interpolation automatic feedrate control, the feedrate of the tool center near a basic circle may become very low in the case of an inner offset. To prevent this, set a lower override limit in this parameter. Thus, the feedrate is clamped so that the feedrate is not lower than a specified feedrate multiplied by the lower override limit set in this parameter.

NOTE
 When 0 or a value not within the valid data range is set, involute interpolation automatic feedrate control ("override in the cutter compensation mode" and "acceleration clamping near a basic circle") is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6000				HGO			MGO	

[Input type] Parameter input
 [Data type] Bit path

#1 MGO When a GOTO statement for specifying custom macro control is executed, a high-speed branch to 20 sequence numbers executed from the start of the program is:
 0: A high-speed branch is not caused to n sequence numbers from the start of the executed program.
 1: A high-speed branch is caused to n sequence numbers from the start of the program.

#4 HGO When a GOTO statement in a custom macro control command is executed, a high-speed branch to the 30 sequence numbers immediately before the executed statement is:
 0: Not made.
 1: Made.

	#7	#6	#5	#4	#3	#2	#1	#0
6210		MDC						

[Input type] Parameter input
 [Data type] Bit path

#6 MDC The measurement result of automatic tool length measurement is:
 0: Added to the current offset.
 1: Subtracted from the current offset.

6241	Feedrate during measurement of automatic tool length measurement (for the XAE1 and GAE1 signals)
-------------	---

6242	Feedrate during measurement of automatic tool length measurement (for the XAE2 and GAE2 signals)
-------------	---

6243	Feedrate during measurement of automatic tool length measurement (for the XAE3 and GAE3 signals)
-------------	---

[Input type] Parameter input
 [Data type] Real path
 [Unit of data] mm/min, inch/min, deg/min (machine unit)
 [Min. unit of data] Depend on the increment system of the applied axis
 [Valid data range] Refer to the standard parameter setting table (C)
 (When the increment system is IS-B, 0.0 to +999000.0)
 These parameters set the relevant feedrate during measurement of automatic tool length measurement.

NOTE
 When the setting of parameter No. 6242 or 6243 is 0, the setting of parameter No. 6241 is used.

6251	γ value during automatic tool length measurement (for the XAE1 and GAE1 signals)
6252	γ value during automatic tool length measurement (for the XAE2 and GAE2 signals)
6253	γ value during automatic tool length measurement (for the XAE3 and GAE3 signals)

[Input type] Parameter input
 [Data type] 2-word path
 [Unit of data] mm, inch, deg (machine unit)
 [Min. unit of data] Depend on the increment system of the applied axis
 [Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
 (When the increment system is IS-B, -999999.999 to +999999.999)
 These parameters set the relevant γ value during automatic tool length measurement.

NOTE
 1 When the setting of parameter No. 6252 or 6253 is 0, the setting of parameter No. 6251 is used.
 2 Set a radius value regardless of whether diameter or radius programming is specified.

6254	ϵ value during automatic tool length measurement (for the XAE1 and GAE1 signals)
6255	ϵ value during automatic tool length measurement (for the XAE2 and GAE2 signals)
6256	ϵ value during automatic tool length measurement (for the XAE3 and GAE3 signals)

[Input type] Parameter input
 [Data type] 2-word path
 [Unit of data] mm, inch, deg (machine unit)
 [Min. unit of data] Depend on the increment system of the applied axis
 [Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
 (When the increment system is IS-B, -999999.999 to +999999.999)
 These parameters set the relevant ϵ value during automatic tool length measurement.

NOTE
 1 When the setting of parameter No. 6252 or 6253 is 0, the setting of parameter No. 6251 is used.
 2 Set a radius value regardless of whether diameter or radius programming is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
7001							ABS	

[Input type] Parameter input
 [Data type] Bit path

- #1 **ABS** For the move command after manual intervention in the manual absolute on state:
 - 0: Different paths are used in the absolute (G90) and incremental (G91) modes.
 - 1: The same path (path in the absolute mode) is used in the absolute (G90) and incremental (G91) modes.

	#7	#6	#5	#4	#3	#2	#1	#0
7570					CFA			FTP

[Input type] Parameter input
 [Data type] Bit path

- #0 **FTP** Fixture offset type setting
 - 0: Movement type (The tool moves when the fixture offset changes.)
 - 1: Shift type (The tool does not move when the fixture offset changes.)

- #3 **CFA** When the fixture offset function is used, and a rotation axis is specified in the increment mode (G91 mode) after manual intervention in the state where the manual absolute switch is on:
 - 0: A vector calculation is made using coordinates not reflecting a manual intervention amount.
 - 1: A vector calculation is made using coordinates reflecting a manual intervention amount.

	#7	#6	#5	#4	#3	#2	#1	#0
7575								FAXx

[Input type] Parameter input
 [Data type] Bit axis

- #0 **FAXx** Fixture offset on each axis is:
 - 0: Disabled.
 - 1: Enabled.

7580	Rotation axis for fixture offset (first group)
7581	Linear axis 1 for fixture offset (first group)
7582	Linear axis 2 for fixture offset (first group)
7583	Rotation axis for fixture offset (second group)
7584	Linear axis 1 for fixture offset (second group)
7585	Linear axis 2 for fixture offset (second group)
7586	Rotation axis for fixture offset (third group)
7587	Linear axis 1 for fixture offset (third group)
7588	Linear axis 2 for fixture offset (third group)

[Input type] Parameter input
 [Data type] Byte path
 [Valid data range] 0 to Number of controlled axes

These parameters specify rotation axes for fixture offset and pairs of linear axes for selecting a rotation plane. Specify a pair of linear axes so that rotation from the positive direction of linear axis 1 to the positive direction is in the normal direction of the rotation axis.

Up to three groups of a rotation axis setting and two linear axis settings can be specified. The fixture offset value is calculated first for the rotation axis in the first group. Then, for the second and third groups, the fixture value is sequentially calculated using the previous calculation result. When you do not need the third group, set 0 for the rotation axis.

10360	Bias value for the offset number of a tool offset for each axis
--------------	--

[Input type] Setting input
 [Data type] Word axis
 [Valid data range] 0 to the number of tool offsets
 When parallel operation is performed, this parameter specifies a bias value for the offset number of a tool offset for each axis. The offset data to be used as a tool offset for an axis has a number obtained by adding a value set in this parameter for the axis to a specified offset number.

10361	Bias for the offset number of tool length compensation for each axis
--------------	---

[Input type] Setting input
 [Data type] Word axis
 [Valid data range] 0 to the number of tool offsets
 When parallel operation is performed, this parameter specifies a bias value for the offset number of tool length compensation for each axis. The offset data to be used as the tool length compensation amount for an axis has a number obtained by adding a value set in this parameter for the axis to a specified offset number.

10370	#7	#6	#5	#4	#3	#2	#1	#0
								RPC

[Input type] Setting input
 [Data type] Bit path
#0 RPC When a return from the reference position (G29) is made, axis switching is:
 0: Disabled.
 1: Enabled.

10371	Axis switching number
--------------	------------------------------

[Input type] Setting input
 [Data type] Byte path
 [Valid data range] 0 to 5
 One of six types of axis switching can be selected by setting its axis switching number in this parameter. Programmed addresses X, Y, and Z correspond to machine axes x, y, and z as follows:

Axis switching No.	Programmed address		
	X	Y	Z
0	x	y	z
1	x	z	y
2	y	x	z
3	y	z	x

Axis switching No.	Programmed address		
	X	Y	Z
4	z	x	y
5	z	y	x

Axis switching number 0 indicates that axis switching is not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
11400						TOP		

[Input type] Parameter input

[Data type] Bit path

#2 TOP Set a tool length compensation or tool offset operation.

0: Tool length compensation or tool offset operation is performed by an axis movement.

1: Tool length compensation or tool offset operation is performed by shifting the coordinate system.

NOTE

This parameter is an-individual path parameter having the same function as bit 6 (TOS) of parameter No. 5006.

To use different compensation types for individual paths, set the bit 6 (TOS) of parameter No.5006 with 0 and specify a compensation type for each path separately, using the parameter TOP. If the bit 6 (TOS) of parameter No.5006 is 1, the bit 2 (TOP) of parameter No.11400 is assumed to be 1 even if it is 0.

	#7	#6	#5	#4	#3	#2	#1	#0
11600			AX1					

[Input type] Parameter input

[Data type] Bit path

#5 AX1 If, in coordinate system rotation mode, a 1-axis command is issued in absolute mode,

0: First, the specified position is calculated in the coordinate system before rotation, and then the coordinate system is rotated.

1: First, the coordinate system is rotated, and then the tool moves to the specified position in the coordinate system.

(FS16i/18i/21i compatible specification)

	#7	#6	#5	#4	#3	#2	#1	#0
11630								FRD

[Input type] Parameter input

[Data type] Bit path

#0 FRD The minimum command unit of the rotation angles of coordinate rotation and 3-dimensional coordinate system conversion is:

0: 0.001 degree.

1: 0.00001 degree. (1/100,000)

	#7	#6	#5	#4	#3	#2	#1	#0
19607		NAA	CAV			CCC		

[Input type] Parameter input

[Data type] Bit path

#2 CCC In the cutter compensation/tool nose radius compensation mode, the outer corner connection method is based on:
 0: Linear connection type.
 1: Circular connection type.

#5 CAV When an interference check finds that interference (overcutting) occurred:
 0: Machining stops with the alarm PS0041, "INTERFERENCE IN CUTTER COMPENSATION". (Interference check alarm function)
 1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)
 For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

#6 NAA When the interference check avoidance function considers that an avoidance operation is dangerous or that a further interference to the interference avoidance vector occurs:
 0: An alarm is issued.
 When an avoidance operation is considered to be dangerous, the alarm PS5447, "DANGEROUS AVOIDANCE AT G41/G42" is issued.
 When a further interference to the interference avoidance vector is considered to occur, the alarm PS5448, "INTERFERENCE TO AVD. AT G41/G42" is issued.
 1: No alarm is issued, and the avoidance operation is continued.



CAUTION

Usually set this parameter to 0.

19625	Number of blocks to be read in the cutter compensation/tool nose radius compensation mode							
-------	---	--	--	--	--	--	--	--

[Input type] Setting input

[Data type] Byte path

[Valid data range] 3 to 8

This parameter sets the number of blocks to be read in the cutter compensation/tool nose radius compensation mode. When a value less than 3 is set, the specification of 3 is assumed. When a value greater than 8 is set, the specification of 8 is assumed. As a greater number of blocks are read, an overcutting (interference) forecast can be made with a command farther ahead. However, the number of blocks read and analyzed increases, so that a longer block processing time becomes necessary.

Even if the setting of this parameter is modified in the MDI mode by stopping in the cutter compensation/tool nose radius compensation mode, the setting does not become valid immediately. Before the new setting of this parameter can become valid, the cutter compensation/tool nose radius compensation mode must be canceled, then the mode must be entered again.

	#7	#6	#5	#4	#3	#2	#1	#0
25860					NCV	SCV		

[Input type] Parameter input

[Data type] Bit path

- #2 SCV** At power-on, a spindle unit compensation vector is:
0: Not calculated.
1: Calculated.

NOTE

This parameter is effective in the case of either of the following settings:

- Bit 6 (CLR) of parameter No. 3402 = 0
- Bit 6 (CLR) of parameter No. 3402 = 1 and bit 3 (C27) of parameter No. 3409 = 1

- #3 NCV** At power-on, an nutating rotary head tool length compensation vector is:
0: Not calculated.
1: Calculated.

NOTE

This parameter is effective in the case of either of the following settings:

- Bit 6 (CLR) of parameter No. 3402 = 0
- Bit 6 (CLR) of parameter No. 3402 = 1, bit 0 (C08) of parameter No. 3407 = 1, and bit 7 (CFH) of parameter No. 3409 = 1

A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit	0 or 1	
Bit machine group		
Bit path		
Bit axis		
Bit spindle		
Byte	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Byte machine group		
Byte path		
Byte axis		
Byte spindle		
Word	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
Word machine group		
Word path		
Word axis		
Word spindle		
2-word	0 to ±999999999	Some parameters handle these types of data as unsigned data.
2-word machine group		
2-word path		
2-word axis		
2-word spindle		
Real	See the Standard Parameter Setting Tables.	
Real machine group		
Real path		
Real axis		
Real spindle		

NOTE

- Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
- For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
- For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
- For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
- For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
- The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

NOTE

- 1 Values are rounded up or down to the nearest multiples of the minimum data unit.
- 2 A valid data range means data input limits, and may differ from values representing actual performance.
- 3 For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value" in the OPERATOR'S MANUAL (Common to Lathe System / Machining Center System).

(A) Length and angle parameters (type 1)

Unit of data	Increment system	Minimum data unit	Valid data range	
mm deg.	IS-A	0.01	-999999.99	to +999999.99
	IS-B	0.001	-999999.999	to +999999.999
	IS-C	0.0001	-99999.9999	to +99999.9999
	IS-D	0.00001	-9999.99999	to +9999.99999
	IS-E	0.000001	-999.999999	to +999.999999
inch	IS-A	0.001	-99999.999	to +99999.999
	IS-B	0.0001	-99999.9999	to +99999.9999
	IS-C	0.00001	-9999.99999	to +9999.99999
	IS-D	0.000001	-999.999999	to +999.999999
	IS-E	0.0000001	-99.9999999	to +99.9999999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range	
mm deg.	IS-A	0.01	0.00	to +999999.99
	IS-B	0.001	0.000	to +999999.999
	IS-C	0.0001	0.0000	to +99999.9999
	IS-D	0.00001	0.00000	to +9999.99999
	IS-E	0.000001	0.000000	to +999.999999
inch	IS-A	0.001	0.000	to +99999.999
	IS-B	0.0001	0.0000	to +99999.9999
	IS-C	0.00001	0.00000	to +9999.99999
	IS-D	0.000001	0.000000	to +999.999999
	IS-E	0.0000001	0.0000000	to +99.9999999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range	
mm/min degree/min	IS-A	0.01	0.00	to +999000.00
	IS-B	0.001	0.000	to +999000.000
	IS-C	0.0001	0.0000	to +99999.9999
	IS-D	0.00001	0.00000	to +9999.99999
	IS-E	0.000001	0.000000	to +999.999999
inch/min	IS-A	0.001	0.000	to +96000.000
	IS-B	0.0001	0.0000	to +9600.0000
	IS-C	0.00001	0.00000	to +4000.00000
	IS-D	0.000001	0.000000	to +400.000000
	IS-E	0.0000001	0.0000000	to +40.0000000

(D) Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range	
mm/sec ² deg./sec ²	IS-A	0.01	0.00	to +999999.99
	IS-B	0.001	0.000	to +999999.999
	IS-C	0.0001	0.0000	to +99999.9999
	IS-D	0.00001	0.00000	to +9999.99999
	IS-E	0.000001	0.000000	to +999.999999
inch/sec ²	IS-A	0.001	0.000	to +99999.999
	IS-B	0.0001	0.0000	to +99999.9999
	IS-C	0.00001	0.00000	to +9999.99999
	IS-D	0.000001	0.000000	to +999.999999
	IS-E	0.0000001	0.0000000	to +99.9999999

INDEX

<Number>

3-DIMENSIONAL TOOL COMPENSATION (G40, G41).....240

<A>

ACTIVE OFFSET VALUE CHANGE FUNCTION
 BASED ON MANUAL FEED258
 AUTOMATIC EXACT STOP CHECK.....324
 Automatic Speed Control for Involute Interpolation.....19
 AUTOMATIC TOOL LENGTH MEASUREMENT
 (G37)164
 AXIS CONTROL FUNCTIONS314
 AXIS SWITCHING327

Back Boring Cycle (G87).....67
 Boring Cycle (G85).....64
 Boring Cycle (G86).....65
 Boring Cycle (G88).....69
 Boring Cycle (G89).....71

<C>

Canned Cycle Cancel (G80).....99
 Canned Cycle Cancel for Drilling (G80)72
 CANNED CYCLE FOR DRILLING.....44
 CANNED GRINDING CYCLE (FOR GRINDING
 MACHINE).....109
 CIRCULAR THREAD CUTTING B (G2.1,G3.1)26
 COMPENSATION FUNCTION.....157
 CONER CONTROL BY FEED RATE.....322
 CONTINUOUS THREADING24
 Continuous-feed Surface Grinding Cycle (G78).....117
 Control Point Compensation of Tool Length
 Compensation Along Tool Axis271
 COORDINATE SYSTEM ROTATION (G68, G69)...246
 COORDINATE VALUE AND DIMENSION.....41
 CORNER CIRCULAR INTERPOLATION (G39).....238

<D>

DATA TYPE.....420
 DEFINITION OF WARNING, CAUTION, AND
 NOTEs-1
 DESCRIPTION OF PARAMETERS.....373
 DETAILS OF CUTTER OR TOOL NOSE RADIUS
 COMPENSATION187
 Direct Constant-Dimension Plunge Grinding Cycle
 (G77)114
 Direction of Imaginary Tool Nose179
 Drilling Cycle Counter Boring Cycle (G82)54
 Drilling Cycle, Spot Drilling (G81)53

<E>

End Face Peck Drilling Cycle (G74.7).....146,304
 Example for Using Canned Cycles for Drilling73
 Extraction override.....99

<F>

Fine Boring Cycle (G76).....51
 Finishing Cycle (G70.7).....142,300
 FUNCTIONS TO SIMPLIFY PROGRAMMING44

<G>

GAS CUTTING MACHINE.....319
 GENERAL3,9
 GENERAL WARNINGS AND CAUTIONSs-2
 GENTLE CURVE CUTTING330
 GENTLE NORMAL DIRECTION CONTROL332
 GRINDING WHEEL WEAR COMPENSATION.....253
 GROOVE CUTTING BY CONTINUOUS CIRCLE
 MOTION (G12.4, G13.4).....30

<H>

Helical Involute Interpolation (G02.2, G03.2)20
 High-Speed Peck Drilling Cycle (G73)48

<I>

Imaginary Tool Nose177
 INDEX TABLE INDEXING FUNCTION105
 IN-FEED CONTROL (FOR GRINDING MACHINE)107
 IN-POSITION CHECK SWITCHING FOR
 DRILLING CANNED CYCLE.....74
 Input of Tool Offset Value Measured B.....367
 Interference Check224
 Interference check alarm function.....228
 Interference check avoidance function.....229
 Intermittent-feed Surface Grinding Cycle (G79)120
 INTERPOLATION FUNCTION15
 INVOLUTE INTERPOLATION (G02.2, G03.2).....15
 Involute Interpolation on Linear Axis and Rotary Axis
 (G02.2, G03.2)21

<L>

Left-Handed Rigid Tapping Cycle (G74)92
 Left-Handed Tapping Cycle (G74)50
 Linear Distance Setting.....333

<M>

MEMORY OPERATION USING Series 15
 PROGRAM FORMAT279
 MULTIPLE REPETITIVE CYCLE280
 MULTIPLE REPETITIVE CYCLE (G70.7, G71.7,
 G72.7, G73.7, G74.7, G75.7,G76.7)123
 MULTIPLE THREADING.....24
 Multiple Threading Cycle (G76.7).....150,307

<N>

NOTES ON READING THIS MANUAL6
 Notes on Tool Nose Radius Compensation.....185
 NOTES ON VARIOUS KINDS OF DATA6

<O>

Offset Number and Offset Value.....	180
Operation to be performed if an interference is judged to occur	227
OPTIONAL CHAMFERING AND CORNER R	102
Outer Diameter / Internal Diameter Drilling Cycle (G75.7)	148,305
Override during Rigid Tapping	99
Override signal.....	100
Overview	187
OVERVIEW OF CUTTER COMPENSATION (G40-G42)	172
OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)	177


<P>

PARALLEL AXIS CONTROL	314
PARAMETERS	373
Pattern Repeating (G73.7).....	139,297
Peck Drilling Cycle (G83)	56
Peck Rigid Tapping Cycle (G84 or G74).....	96
Plunge Grinding Cycle (G75)	111
POLAR COORDINATE COMMAND (G15, G16).....	41
PREPARATORY FUNCTION (G FUNCTION).....	10
Prevention of Overcutting Due to Tool Radius / Tool Nose Radius Compensation.....	221

<R>

Restrictions on Multiple Repetitive Cycle	312
Restrictions on Multiple Repetitive Cycle (G70.7, G71.7, G72.7, G73.7, G74.7, G75.7, and G76.7)	155
RIGID TAPPING.....	88
Rigid Tapping (G84).....	88
ROTARY TABLE DYNAMIC FIXTURE OFFSET	261

<S>

SAFETY PRECAUTIONS	S-1
SCREENS DISPLAYED BY FUNCTION KEY 	337
SETTING AND DISPLAYING DATA	337
Setting and Displaying the Rotary Table Dynamic Fixture Offset	364
Setting and Displaying the Tool Compensation Value.....	337
Small-Hole Peck Drilling Cycle (G83)	58
SPINDLE UNIT COMPENSATION, NUTATING ROTARY HEAD TOOL LENGTH COMPENSATION.....	275,367
STANDARD PARAMETER SETTING TABLES.....	421
Stock Removal in Facing (G72.7).....	135,293
Stock Removal in Turning (G71.7).....	124,281

<T>

Tapping Cycle (G84).....	62
THREADING (G33)	23
TOOL AXIS DIRECTION TOOL LENGTH COMPENSATION.....	267
TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)	243

TOOL FIGURE AND TOOL MOTION BY

PROGRAM	9
TOOL LENGTH COMPENSATION SHIFT TYPES	157
Tool Length Measurement	342
Tool Length/Workpiece Origin Measurement	345
Tool Movement in Offset Mode.....	197
Tool Movement in Offset Mode Cancel.....	215
Tool Movement in Start-up.....	191
TOOL OFFSET (G45 TO G48).....	167
TOOL OFFSET B.....	319
Tool Radius / Tool Nose Radius Compensation for Input from MDI	235

<V>

VECTOR RETENTION (G38).....	237
-----------------------------	-----

<W>

WARNINGS AND CAUTIONS RELATED TO HANDLING.....	s-5
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING.....	s-3
WARNINGS RELATED TO DAILY MAINTENANCE.....	s-7
Workpiece Position and Move Command.....	180

REVISION RECORD

Edition	Date	Contents
03	Aug., 2011	<ul style="list-style-type: none">• Deletion of following functions<ul style="list-style-type: none">- Chopping function- Chopping function by flexible synchronous control• Correction of errors
02	Oct., 2010	<ul style="list-style-type: none">• Addition of following G code<ul style="list-style-type: none">- Plane conversion function• Correction of errors
01	Jun., 2010	

