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

For Machining Center System

OPERATOR'S MANUAL

B-64304EN-2/02

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

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

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

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 panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

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

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

⚠ **CAUTION**

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

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off.
Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

⚠ **WARNING**

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

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

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

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

⚠ **WARNING**

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

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

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

8 **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.

9 **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.

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

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

WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied 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.

⚠ **WARNING**

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

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

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

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

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

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

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

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

⚠ **WARNING**

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

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

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

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

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 T/M series) for details of the battery replacement procedure.

⚠ WARNING

2 **Absolute pulse coder battery replacement**

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

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

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

NOTE

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

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

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

Refer to the Section “Method of replacing battery” in the Operator’s Manual (Common to T/M series) 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

III. OPERATION

APPENDIX

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, valid data ranges, and alarms.

NOTE

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

Applicable models

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

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

NOTE

1 For explanatory purposes, these models may be classified as shown below:
- M series: 0*i* -MD / 0*i* Mate -MD

2 Some functions described in this manual may not be applied to some products. For details, refer to the Descriptions (B-64302EN).

3 For the 0*i*-D / 0*i* Mate-D, parameters need to be set to enable or disable some basic functions.
For these parameters, refer to Section 4.51, " PARAMETERS OF 0*i*-D / 0*i* Mate-D BASIC FUNCTIONS" in the PARAMETER MANUAL (B-64310EN).

Special symbols

This manual uses the following symbols:

- IP

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

- ;

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

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

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

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64302EN	
CONNECTION MANUAL (HARDWARE)	B-64303EN	
CONNECTION MANUAL (FUNCTION)	B-64303EN-1	
OPERATOR’S MANUAL (Common to Lathe System/Machining Center System)	B-64304EN	
OPERATOR’S MANUAL (For Lathe System)	B-64304EN-1	
OPERATOR’S MANUAL (For Machining Center System)	B-64304EN-2	*
MAINTENANCE MANUAL	B-64305EN	
PARAMETER MANUAL	B-64310EN	
START-UP MANUAL	B-64304EN-3	
Programming		
Macro Compiler / Macro Executor PROGRAMMING MANUAL	B-64303EN-2	
Macro Compiler OPERATOR’S MANUAL	B-64304EN-5	
C Language PROGRAMMING MANUAL	B-64303EN-3	
PMC		
PMCPROGRAMMING MANUAL	B-64393EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-64403EN	
Fast Ethernet / Fast Data Server OPERATOR’S MANUAL	B-64414EN	
DeviceNet Board CONNECTION MANUAL	B-64443EN	
FL-net Board CONNECTION MANUAL	B-64453EN	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64303EN-4	

Manual name	Specification number	
Operation guidance function		
MANUAL GUIDE *i* (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE *i* (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN−1	
MANUAL GUIDE 0*i* OPERATOR'S MANUAL	B-64434EN	
TURN MATE *i* OPERATOR'S MANUAL	B-64254EN	

Related manuals of SERVO MOTOR α*i*/β*i* series

The following table lists the manuals related to SERVO MOTOR α*i*/β*i* series

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B-65272EN
FANUC AC SERVO MOTOR β*i* series DESCRIPTIONS	B-65302EN
FANUC AC SPINDLE MOTOR β*i* series DESCRIPTIONS	B-65312EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER β*i* series DESCRIPTIONS	B-65322EN
FANUC SERVO MOTOR α*i*s series FANUC SERVO MOTOR α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B-65285EN
FANUC SERVO MOTOR β*i*s series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR α*i*/β*i* 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 α*i*/β*i* series, BUILT-IN SPINDLE MOTOR B*i* series PARAMETER MANUAL	B-65280EN

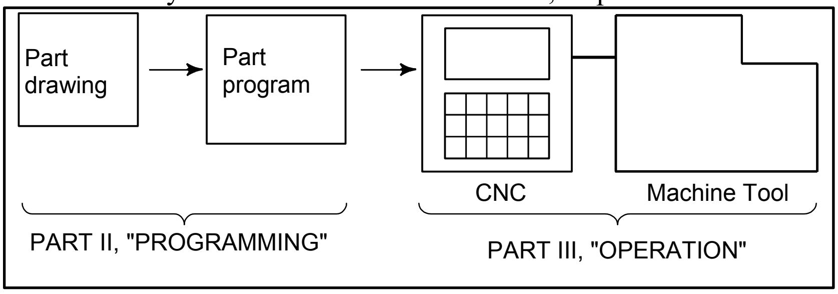
This manual mainly assumes that the FANUC SERVO MOTOR α*i* series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL

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

(1) First, prepare the program from a part drawing to operate the CNC machine tool.
How to prepare the program is described in the Part II, "Programming."

(2) The program is to be read into the CNC system. Then, mount the workpieces and tools on the machine, and operate the tools according to the programming. Finally, execute the machining actually.
How to operate the CNC system is described in the Part III, "Operation."



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

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

Decide the cutting method in every cutting process.

Cutting procedure \ Cutting process	1 End face cutting	2 Outer diameter cutting	3 Grooving
1. Cutting method : Rough Semi Finish			
2. Cutting tools			
3. Cutting conditions : Feedrate Cutting depth			
4. Tool path			

1.2 NOTES ON READING THIS MANUAL

⚠ **CAUTION**

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

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

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

1.3 NOTES ON VARIOUS KINDS OF DATA

⚠ **CAUTION**

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

II. PROGRAMMING

1 GENERAL

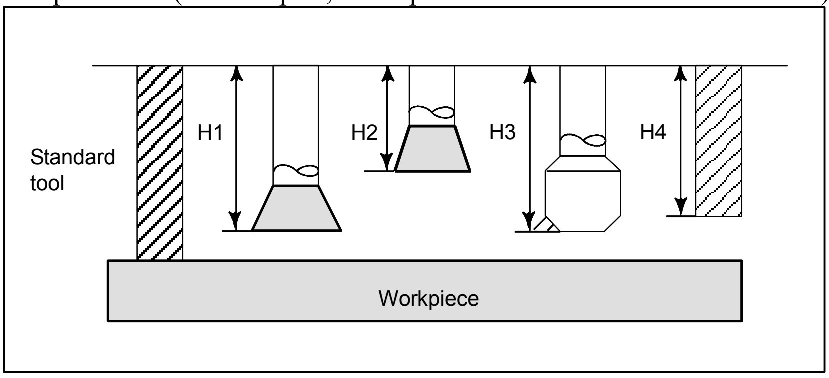
Chapter 1, "GENERAL", consists of the following sections:

1.1 TOOL FIGURE AND TOOL MOTION BY PROGRAM

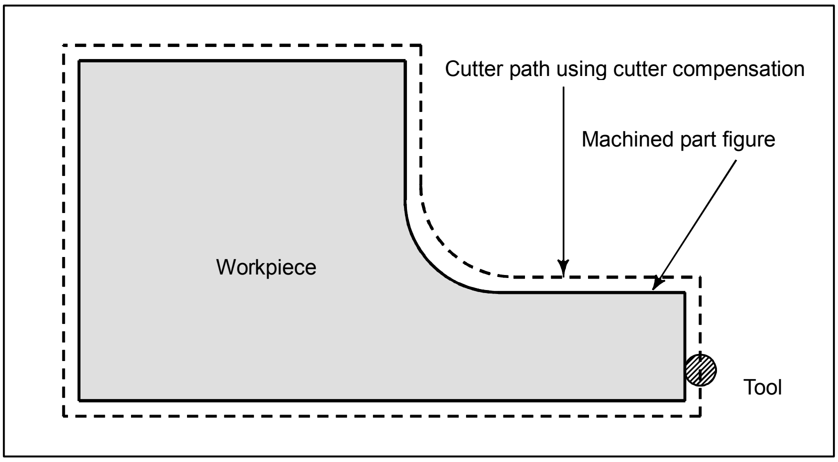
Explanation

- 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 Chapter, “Compensation Function” in this manual).



- 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” in this manual).

2 PREPARATORY FUNCTION (G FUNCTION)

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

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

(Example)

G01 and G00 are modal G codes in group 01.

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

Explanation

1. When the clear state (parameter CLR (No. 3402#6)) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◤ as indicated in Table 2.
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by parameter G23 (No. 3402#7). However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting parameter G01 (No. 3402#0).
   (5) The user can select G90 or G91 by setting parameter G91 (No. 3402#3).
   When G code system B or C is used in the lathe system, setting parameter G91 (No. 3402#3) determines which code, either G90 or G91, is effective.
   (6) In the machining center system, the user can select G17, G18, or G19 by setting parameters G18 and G19 (No. 3402#1 and #2).
2. G codes in group 00 other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010 occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a 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 parameter MDL (No. 5431#0). (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

Table 2 G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G04	00	Dwell, Exact stop	
G05.1		AI advanced preview control / AI contour control / AI contour control II	
G05.4		HRV3 on/off	
G07.1 (G107)		Cylindrical interpolation	
G09		Exact stop	
G10		Programmable data input	
G11		Programmable data input mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G31		Skip function	
G33	01	Threading	
G37	00	Automatic tool length measurement	
G39		Cutter compensation : corner circular interpolation	
G40	07	Cutter compensation : cancel	
G41		Cutter compensation : left	
G42		Cutter compensation : right	
G40.1	19	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 -	
G45	00	Tool offset : increase	
G46		Tool offset : decrease	
G47		Tool offset : double increase	
G48		Tool offset : double decrease	
G49	08	Tool length compensation cancel	
G50	11	Scaling cancel	
G51		Scaling	
G50.1	22	Programmable mirror image cancel	
G51.1		Programmable mirror image	
G52	00	Local coordinate system setting	
G53		Machine coordinate system setting	

Table 2 G code list

G code	Group	Function
G54	14	Workpiece coordinate system 1 selection
G54.1		Additional workpiece coordinate system selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call
G67		Macro modal call cancel
G68	16	Coordinate system rotation mode on
G69		Coordinate system rotation mode off
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G75	01	Plunge grinding cycle (for grinding machine)
G76	09	Fine boring cycle
G77	01	Plunge direct sizing/grinding cycle (for grinding machine)
G78		Continuous-feed surface grinding cycle (for grinding machine)
G79		Intermittent-feed surface grinding cycle (for grinding machine)
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
G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS10/11 format)
G84.3		Left-handed rigid tapping cycle (FS10/11 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

Table 2 G code list

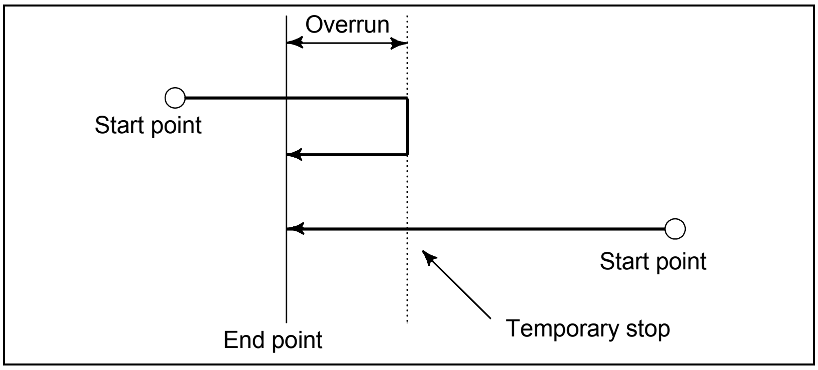
G code	Group	Function
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G160	20	In-feed control cancel (for grinding machine)
G161		In-feed control (for grinding machine)

3 INTERPOLATION FUNCTION

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

3.1 SINGLE DIRECTION POSITIONING (G60)

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



Format

G60 IP_ ;

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

Explanation

An overrun and a positioning direction are set by the parameter No. 5440. Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.
G60, which is a one-shot G-code, can be used as a modal G-code in group 01 by setting 1 to the bit 0 (MDL) of parameter No. 5431.
This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for a one-shot G60 command. When a one-shot G code is specified in the single direction positioning mode, the one-shot G command is effective like G codes in group 01.

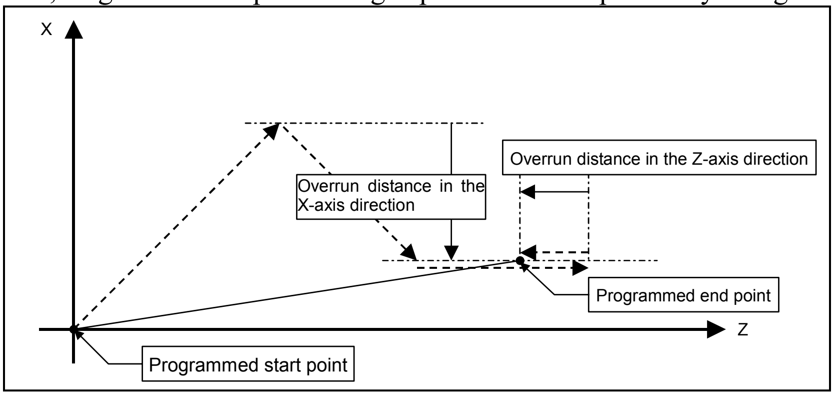
(Example)

When one-shot G60 commands are used.	
G90;	
G60 X0Y0; G60 X100; G60 Y100;	Single direction positioning
G04 X10;	
G00 X0Y0;	
When modal G60 command is used.	
G90G60;	Single direction positioning mode start
X0Y0; X100; Y100;	Single direction positioning

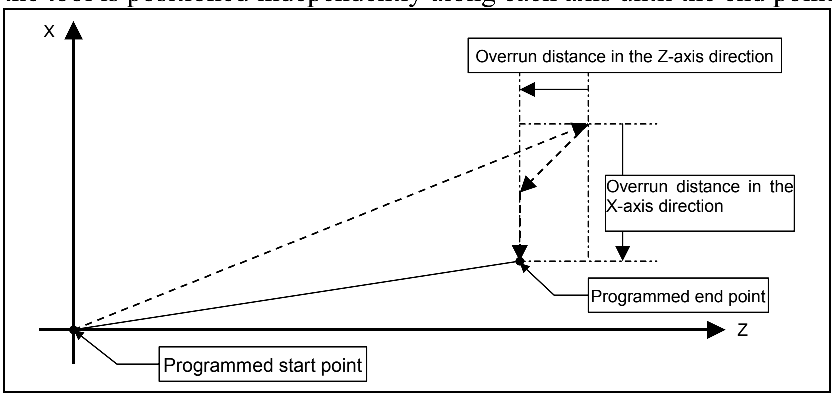
```
G04X10;
G00X0 Y0;      Single direction positioning mode cancel
```

- Overview of operation

- **In the case of positioning of non-linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 0)**
  As shown below, single direction positioning is performed independently along each axis.



- **In the case of positioning of linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 1)**
  Positioning of interpolation type is performed until the tool once stops before or after a specified end point. Then, the tool is positioned independently along each axis until the end point is reached.



Limitation

- Single direction positioning is not performed along an axis for which no overrun distance is set in parameter No. 5440.
- Single direction positioning is not performed along an axis for which travel distance 0 is specified.
- The mirror image function is not applied in a parameter-set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged. If positioning of linear interpolation type is used, and the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, an alarm is issued. When switching mirror image in the middle of a program, disable looking ahead by specifying a non-buffering M code. Then, switch mirror image when there is no look-ahead block.
- In the cylindrical interpolation mode (G07.1), single direction positioning cannot be used.
- When specifying single direction positioning on a machine that uses arbitrary angular axis control, first position the angular axis then specify the positioning of the Cartesian axis. If the reverse specification order is used, or the angular axis and Cartesian axis are specified in the same block, an incorrect positioning direction can result.
- In positioning at a restart position by program restart function, single direction positioning is not performed.
- During canned cycle for drilling, no single direction positioning is effected in drilling axis.

- The single direction positioning does not apply to the shift motion in the canned cycles of G76 and G87.

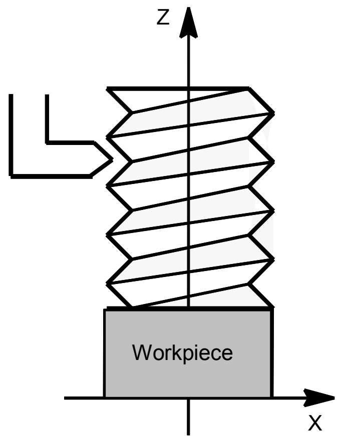
3.2 THREADING (G33)

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

G33IP_ F_ ;

F : Long axis direction lead



Explanation

In general, threading is repeated along the same tool path in rough cutting through finish cutting for a screw. Since threading starts when the position coder mounted on the spindle outputs a 1-turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated threading. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.
In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.
Table 3.2 (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 ≤ spindle speed ≤ (Maximum feedrate) / (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
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 NANO SMOOTHING

Overview

When a desired sculptured surface is approximated by minute segments, the Nano smoothing function generates a smooth curve inferred from the programmed segments and performs necessary interpolation.
The Nano smoothing function infers a curve from a programmed figure approximated with segments within tolerance. The interpolation of the curve reduces the segment approximation error, and the nano-interpolation makes the cutting surface smoother.
The interpolation of the curve reduces the segment approximation error, and the nano interpolation makes the cutting surface smoother.
For this function, the AI contour control II option is required.

Format

G5.1 Q3 Xp0 Yp0 Zp0 ; **:** Nano smoothing mode on
G5.1 Q0 ; **:** Nano smoothing mode off

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

NOTE

1. Specify G5.1 alone in a block.
   (Avoid specifying any other G code in the same block.)
2. Specify position 0 for the axis programmed in the Nano smoothing mode on block. The specified axis is subjected to Nano smoothing, but no movement is made even in the absolute programming mode.
   (Axis moving is not performed in the G05.1Q3 block.)
3. Nano smoothing mode is also turned off at a reset.

In the G5.1 Q3 block, specify the axis subject to Nano smoothing. Note that up to three axes can be subject to the Nano smoothing command at a time and that only the following axes can be specified.

- Basic three axes (X,Y,Z)
- Axes parallel to the basic three axes

If specifying the machining condition selecting function, specify G5.1 Q1 Rx first and then Nano smoothing.

Example

```
O0010                   If the following functions are required
…                       before nano smoothing, specify G5.1.
(G5.1 Q1 R1;) ————————  - AI contour control II
G5.1 Q3 X0 Y0 Z0;       - Machining condition selecting function
…
G5.1 Q0; ———————————    Nano smoothing mode off
                        AI contour control II mode off
…
M30;
```

Explanation

Generally, a program approximates a sculptured surface with minute segments with a tolerance of about 10 μm.

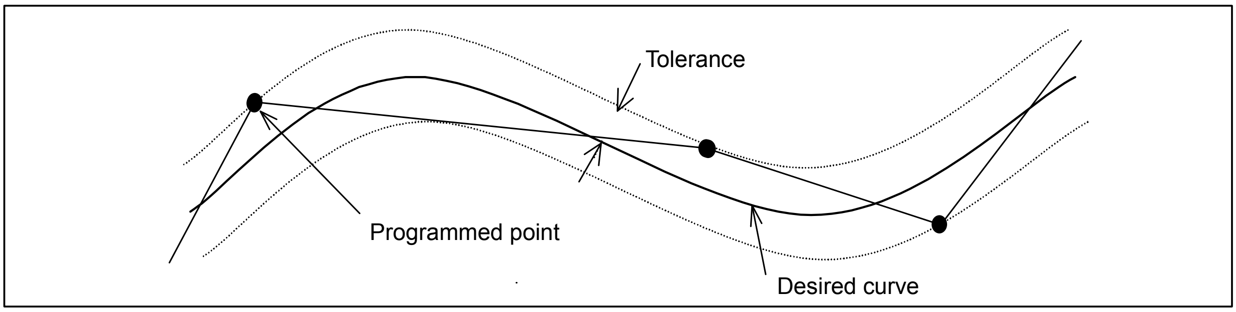


Fig. 3.3 (a)

Many programmed points are placed on the boundary of tolerance. The programmed points also have a rounding error owing to the least input increment of the CNC. The Nano smoothing function creates multiple insertion points between adjacent programmed points so that a smooth curve can be created from the approximation segments. The desired curve is inferred from the insertion points of multiple blocks including buffered blocks.

Many insertion points are closer to the desired curve than the programmed points. A stable curve can be inferred with the insertion points created from multiple blocks including buffered blocks. Because the position of each insertion point is corrected in a unit smaller than the least input increment of the CNC within tolerance, the impact of rounding error is reduced.
Nano-interpolation is performed for the curve inferred from the corrected insertion points, so the resultant cutting surface becomes smooth.

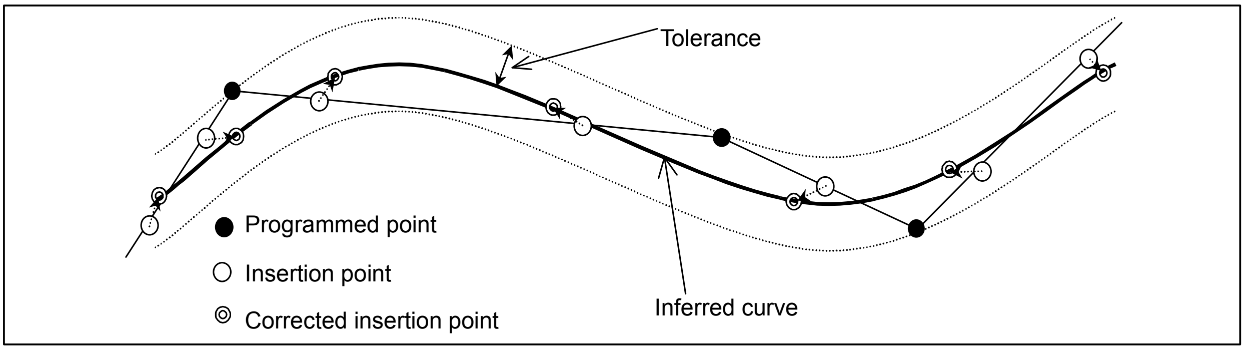


Fig. 3.3 (b)

- Specifying the tolerance

The tolerance of the program of Nano smoothing is specified in parameter No. 19581.
The insertion points are corrected within tolerance, and a curve is inferred accordingly.
If 0 is specified in parameter No. 19581, the minimum travel distance in the increment system is considered to be the tolerance.

- Making a decision on the basis of the spacing between adjacent programmed points

If the spacing between adjacent programmed points (block length) exceeds the value specified in parameter No. 8486 or falls below the value specified in parameter No. 8490 in the Nano smoothing mode, the Nano smoothing mode is cancelled at the start point of the block. Linear interpolation can be performed in the block.
When a decision is made on the basis of the spacing between adjacent programmed points, only the basic three axes (or their parallel axes) are considered, and the rotation axes are excluded. When the Nano smoothing mode is canceled in a block, Nano smoothing for the rotation axes is not performed, either.
If the values specified in the parameters are 0, no decision is made on the basis of the spacing between adjacent programmed points.

- Making a decision at a corner

If the difference in angle (see the Fig. 3.3 (c)) between adjacent programmed blocks exceeds the value specified in parameter No. 8487 in the Nano smoothing mode, the Nano smoothing mode is cancelled at the corner.
The decision at the corner is made by considering the basic three axes (or their parallel axes) only; the rotation axes are not considered. When the Nano smoothing mode is canceled in a block, Nano smoothing for the rotation axes is not performed, either.

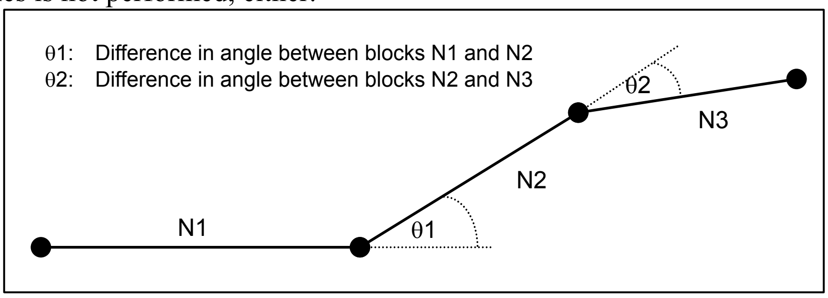


Fig. 3.3 (c)

If the value specified in the parameter is 0, no decision is made at the corner on the basis of the difference in angle.
Very minute blocks created for some reasons such as a calculation error of CAM can be ignored, and a smooth connection can be made at a corner. To do this, specify parameter No. 19582 to the minimum travel distance with which a decision is made on the basis of difference in angle. Then, the decision at a corner is disabled for a block of which distance is less than the specified minimum travel distance.

However, a decision based on the spacing between adjacent programmed points specified in parameter No. 8490 has higher priority than the decision at a corner. Therefore, the value specified in parameter No. 19582 must be greater than the value specified in parameter No. 8490.

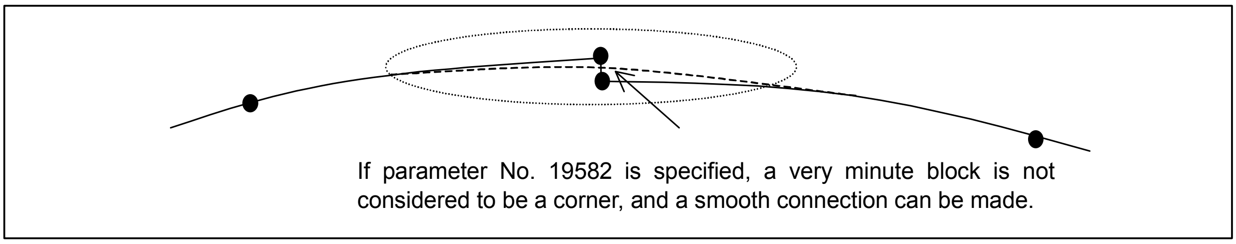


Fig. 3.3 (d)

- Automatically turning on and off AI contour control II with Nano smoothing

Specifying G5.1 Q3 also enables Nano smoothing and AI contour control II to be turned on at the same time. The automatic velocity control by AI contour control II reduces impacts on the mechanical system. Specifying G5.1 Q0 cancels the Nano smoothing and the AI contour control II mode at the same time.

- Conditions for enabling Nano smoothing

Nano smoothing is enabled if the conditions below are satisfied.
In a block that does not satisfy the conditions for enabling it, Nano smoothing is canceled, and it is judged in the next block whether to perform Nano smoothing anew.
In the following description, "block length" and "angle difference between blocks" apply to the basic three axes (or axes parallel to them) only, not rotation axes. Note, however, that in a block in which nano smoothing mode is canceled due to any of these conditions, nano smoothing on rotation axes will not be performed, either.

(1) The specified block length is less than parameter No. 8486.
(2) The specified block length is greater than parameter No. 8490.
(3) The angle difference between the specified blocks is less than parameter No. 8487.
(4) The mode is one of the following:
 - Linear interpolation
 - Feed per minute
 - Tool radius compensation cancel
 - Canned cycle cancel
 - Scaling cancel
 - Macro modal call cancel
 - Constant surface speed control cancel
 - Cutting mode
 - Coordinate system rotation cancel
 - Polar coordinate command cancel
 - Normal direction control cancel
 - Programmable mirror image cancel
(5) The block does not contain a one shot G code command.
(6) The block does not suppress look ahead (buffering).
(7) The block contains a move command for only an axis subject to nano smoothing.

- Checking the nano smoothing

Diagnostic data (No. 5000) indicates whether the nano smoothing mode is enabled in the current block.
If the nano smoothing mode is enabled, "smoothing on" bit is set to 1.

Limitation

- Modal G codes usable when Nano smoothing is specified

In a modal G code state listed below, Nano smoothing can be specified.
Do not specify smooth interpolation in modal states other than these.

G15	:	Polar coordinate command cancel
G40	:	Tool radius compensation cancel
G40.1	:	Normal direction control cancel
G49,G43,G44	:	Tool length compensation cancel or tool length compensation
G50	:	Scaling cancel
G50.1	:	Programmable mirror image cancel
G64	:	Cutting mode cancel
G67	:	Macro modal call cancel
G69	:	Coordinate system rotation/3-dimensional coordinate system conversion cancel
G80	:	Canned cycle cancel
G94	:	Feed per minute
G97	:	Constant surface speed control cancel

- Single-block operation

When single-block operation is carried out in the Nano smoothing mode, the operation stops at a corrected insertion point not at a programmed point.
Even in the Nano smoothing mode, normal single-block operation is carried out for a block that does not satisfy the conditions of Nano smoothing mode.

- Tool length compensation

To carry out tool length compensation, specify the command before specifying Nano smoothing. Avoid changing the amount of compensation in the Nano smoothing mode.
If G43, G44, or G49 is specified in a block between the block in which the command of Nano smoothing mode on (G5.1 Q3) is specified and the block in which the command of Nano smoothing mode off (G5.1 Q0) is specified, an alarm PS0343 will be issued.

- Tool radius/tool nose radius compensation

If tool radius/tool nose radius compensation is specified in the Nano smoothing mode, the Nano smoothing mode is cancelled. Then, when the command of tool radius/tool nose radius compensation cancel (G40) is specified, a decision is made whether to start Nano smoothing from the next block. The startup and cancel operations of type C are always carried out for the tool radius/tool nose radius compensation specified in the Nano smoothing mode, irrespective of the parameter setting.
A command related to tool radius/tool nose radius compensation should not be specified in the Nano smoothing mode unless it is absolutely necessary.

- Interruption type custom macro

No interruption type custom macro can be used in the Nano smoothing mode.
If the Nano smoothing mode is specified while an interruption type custom macro is enabled or if an interruption type custom macro is enabled in the Nano smoothing mode, an alarm PS0342 will be issued.

- Manual intervention

Manual intervention by specifying the manual absolute on command cannot be performed in the Nano smoothing mode. If this is attempted, an alarm PS0340 will be issued at the cycle start after manual intervention.

- Number of blocks that can be specified successively

Up to about 300,000,000 blocks can be specified successively in the Nano smoothing mode. If more blocks are specified, an alarm PS0341 will be issued.
However, when a block which does not satisfy the conditions of the Nano smoothing mode is encountered, the mode is canceled and the counted number of successive blocks is reset to 0.

- Continuity of a program

Curve interpolation is carried out for multiple programmed blocks including buffered blocks in the Nano smoothing mode.
Therefore, the programmed commands must be executed continuously in the Nano smoothing mode.
The continuity of a program may be lost, and continuous execution may not be performed, in some cases such as the following: A single-block stop is made in the Nano smoothing mode; and another program is executed in the MDI mode. If this occurs, an alarm PS0344 will be issued.

- Restrictions on resumption of automatic operation

(1) Resuming a program
Curve interpolation is performed for corrected insertion points not for programmed points in the Nano smoothing mode. Accordingly, when a sequence number is specified to resume the program, the operation cannot be resumed from a programmed point in a block.
To resume a program, specify a block number, using the block counter displayed in the program screen.

(2) Retracing (Retrace)
Retracing cannot be performed in the Nano smoothing mode.

(3) Manual handle retrace
In Nano smoothing mode, manual handle retrace cannot be performed.

- Dynamic graphic display

The dynamic graphic display function draws the path in the Nano smoothing mode by linear interpolation.

4 COORDINATE VALUE AND DIMENSION

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

4.1 POLAR COORDINATE COMMAND (G15, G16)

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

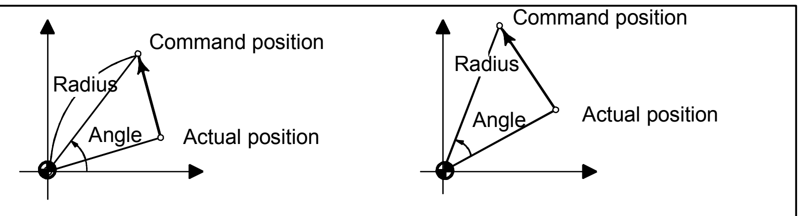
Format

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

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

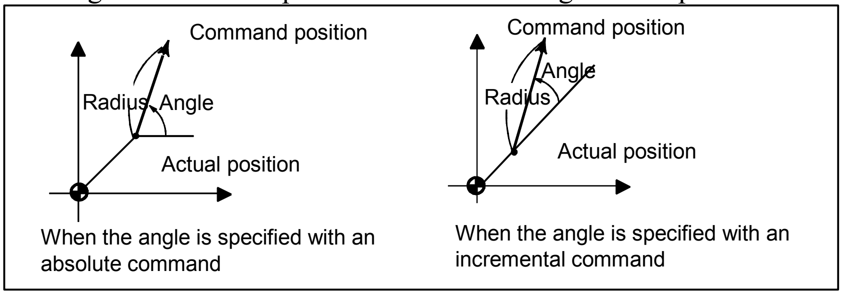


When the angle is specified with an absolute command

When the angle is specified with an incremental command

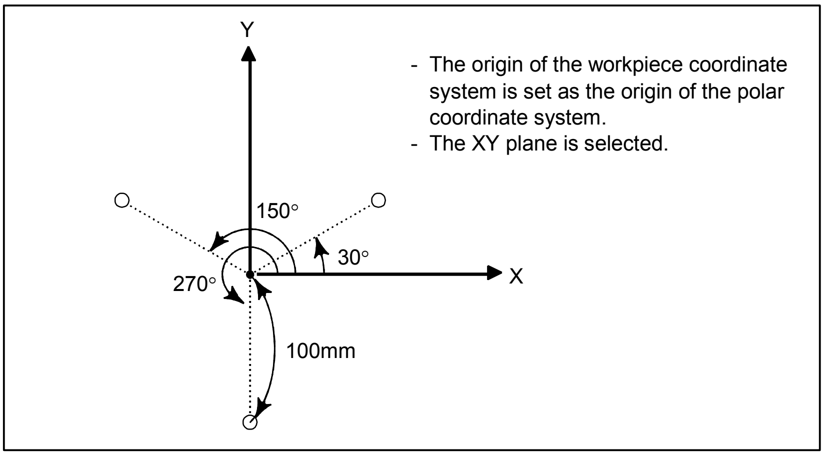
- Setting the current position as the origin of the polar coordinate system

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



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

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	Oriented spindle stop	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

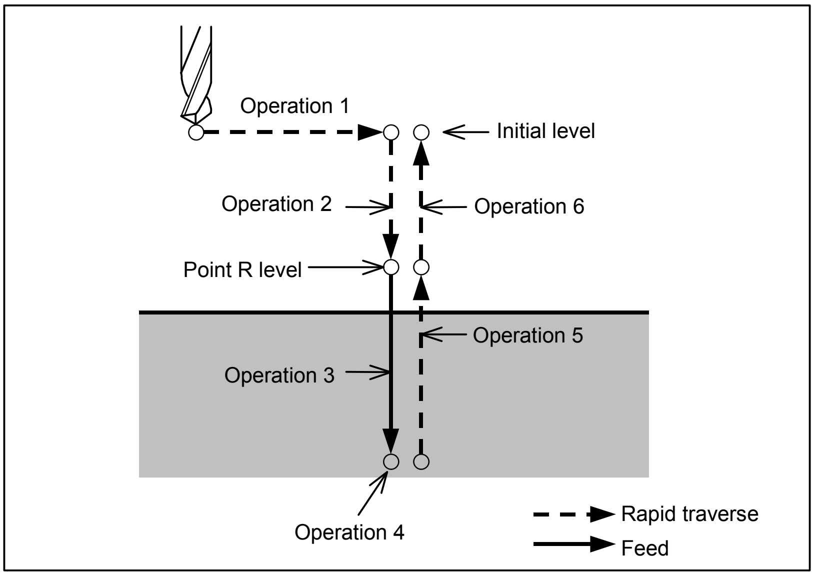


Fig. 5.1 (a) Operation sequence of canned cycle for drilling

- 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 parameter FXY (No. 5101 #0) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

- Travel distance along the drilling axis G90/G91

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

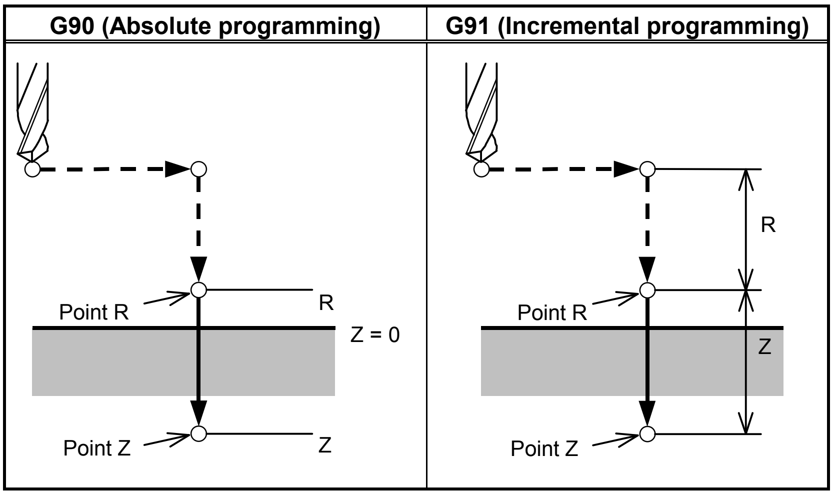


Fig. 5.1 (b) Absolute programming and incremental programming

- 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 following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.
The initial level does not change even when drilling is performed in the G99 mode.

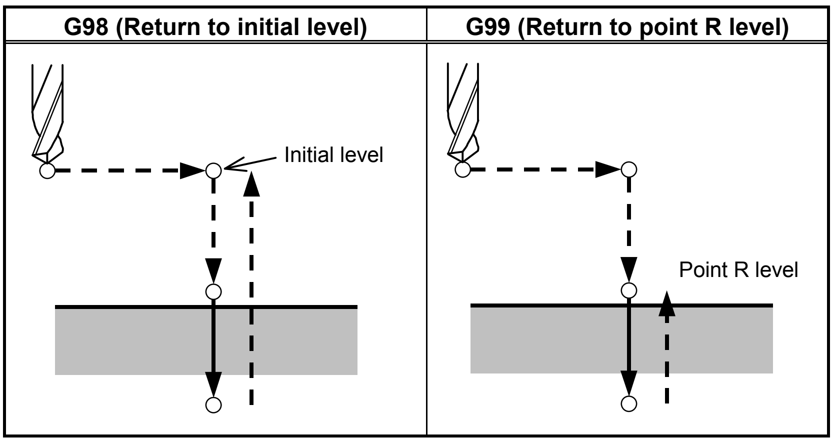


Fig. 5.1 (c) Initial level and point R level

- 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 Figure 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.

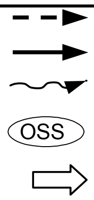
- Cancel

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)
G60 : Single directional positioning (if bit 0 (MDL) of parameter No. 5431 is "1")

- 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

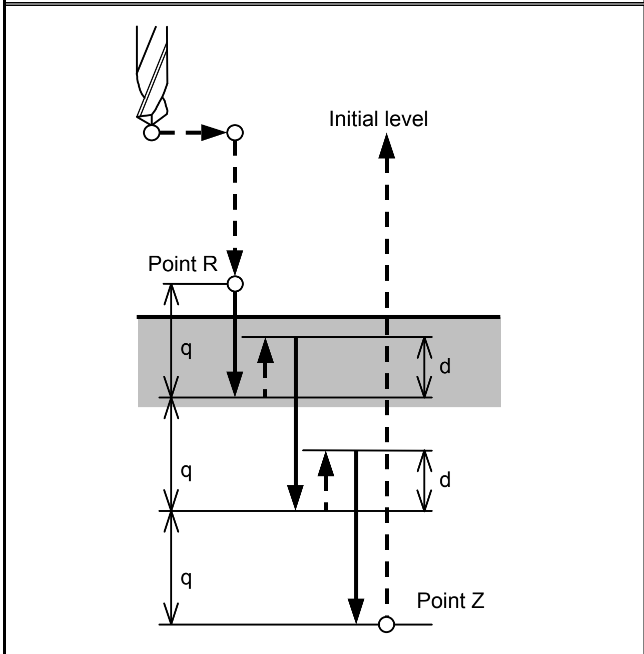
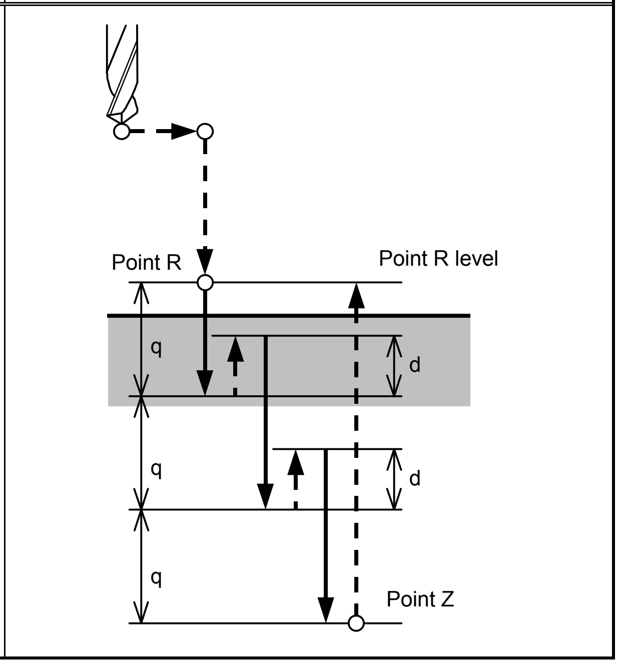
5.1.1 High-Speed Peck Drilling Cycle (G73)

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)

G73 (G98)	G73 (G99)
	

Explanation

- Operations

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

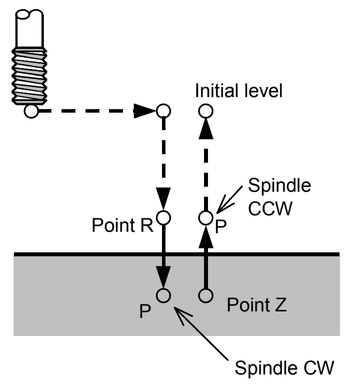
This cycle performs left-handed tapping. In the left-handed tapping cycle, when the bottom of the hole has been reached, the spindle rotates clockwise.

Format

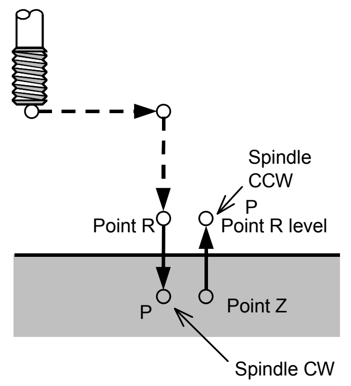
G74 X_ Y_ Z_ R_ P_ Q_ F_ K_ ;

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
P_	: Dwell time
Q_	: Depth of cut for each cutting feed (bit 6 (PCT) of parameter No. 5104 = "1")
F_	: Cutting feedrate
K_	: Number of repeats (if required)

G74 (G98)



G74 (G99)



Explanation

- Operations

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

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

- Spindle rotation

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

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

- Q command

After setting bit 6 (PCT) of parameter No. 5104 to 1, add address Q to the ordinary tapping cycle command format and specify the depth of cut for each tapping.

In the peck tapping cycle, the tool is retracted to point R for each tapping. In the high-speed peck tapping cycle, the tool is retracted by the retraction distance specified for parameter No. 5213 in advance. Which operation is to be performed can be selected by setting bit 5 (PCP) of parameter No. 5200.

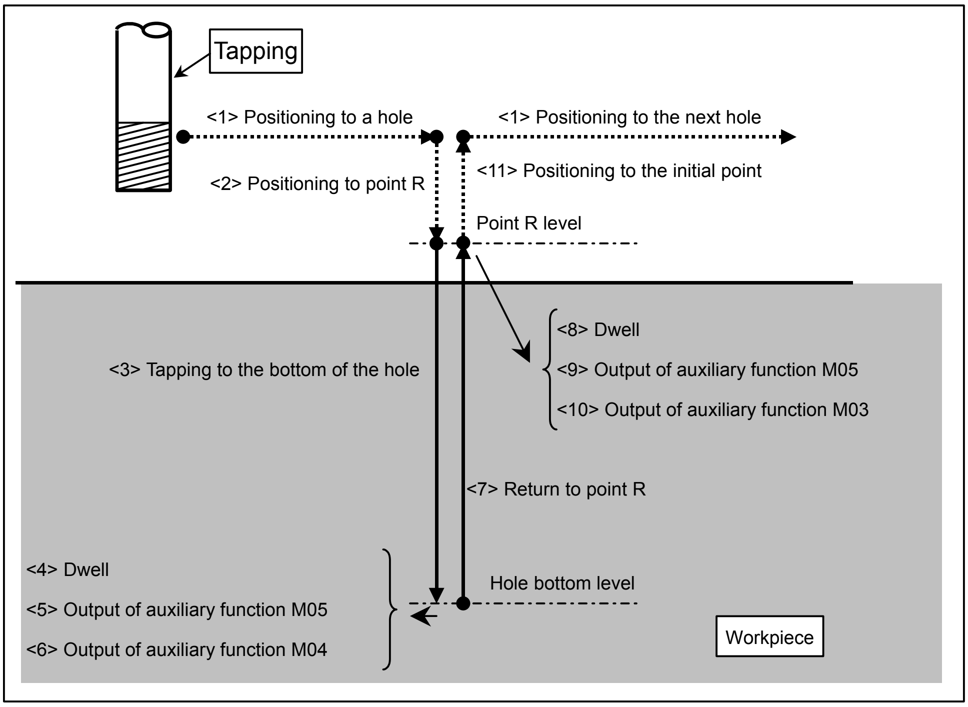
Operation

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

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

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

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



Peck tapping cycle

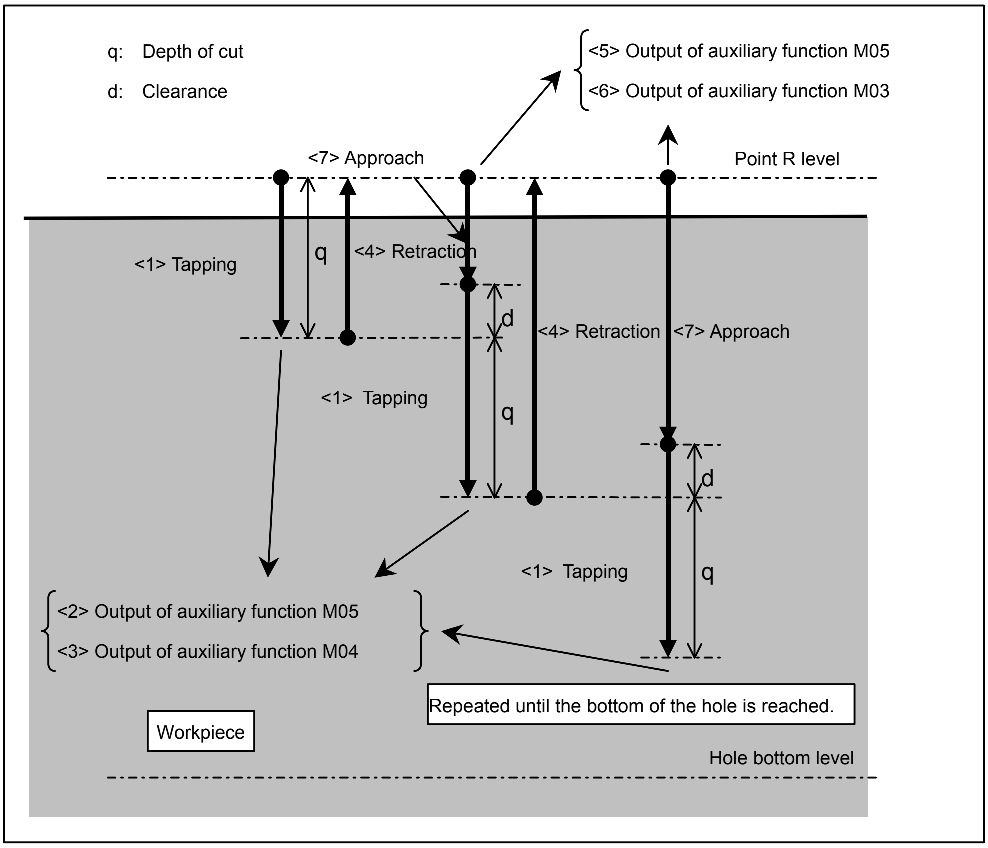
When bit 6 (PCT) of parameter No. 5104 is set to 1 and bit 5 (PCP) of parameter No. 5200 is set to 1, the peck tapping cycle is used.
Step 3 of the tapping cycle operation described above changes as follows:

3-1. The tool cuts the workpiece by the depth of cut q specified by address Q.
3-2. Auxiliary function M05 (spindle stop) is output, and the machine enters the FIN wait state.
3-3. When FIN is returned, auxiliary function M04 (reverse spindle rotation) is output, and the machine enters the FIN wait state.
3-4. When FIN is returned, the tool is retracted to point R in cutting feed.
3-5. Auxiliary function M05 (spindle stop) is output, and the machine enters the FIN wait state.

3-6. When FIN is returned, auxiliary function M03 (forward spindle rotation) is output, and the machine enters the FIN wait state.
3-7. When FIN is returned, the tool moves to the position the clearance d (parameter No. 5213) apart from the previous cutting point in cutting feed (approach).

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

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



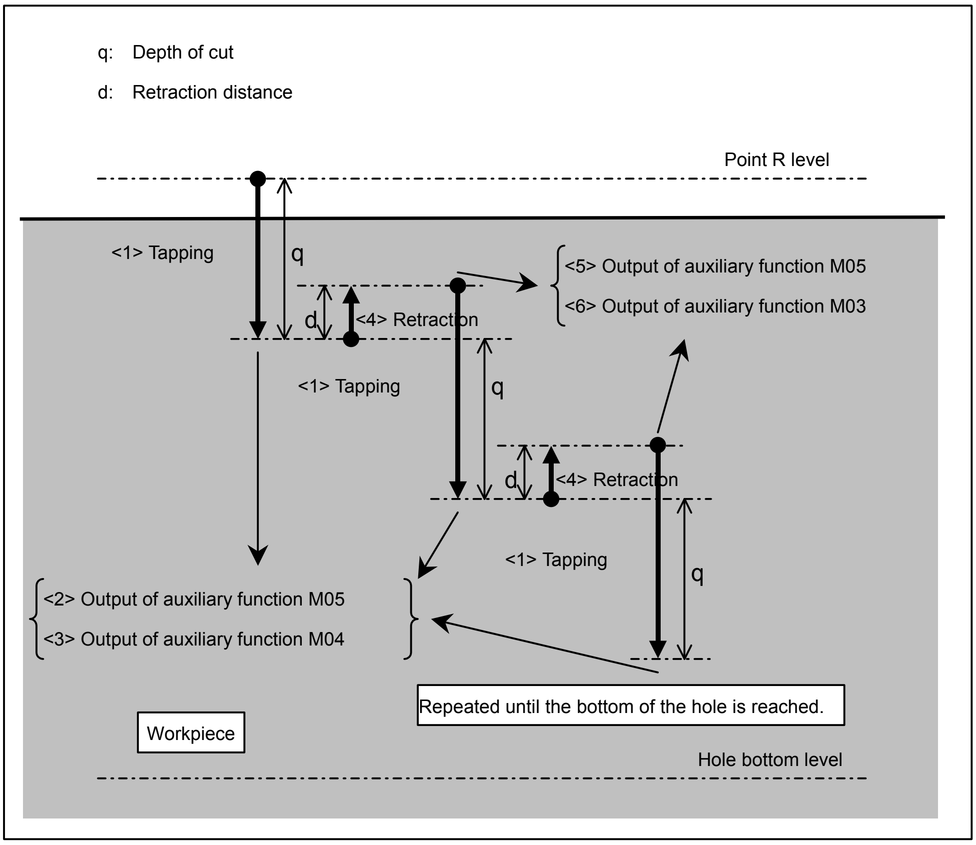
High-speed peck tapping cycle

When bit 6 (PCT) of parameter No. 5104 is set to 1 and bit 5 (PCP) of parameter No. 5200 is set to 0, the high-speed peck tapping cycle is used.
Step 3 of the tapping cycle operation described above changes as follows:

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

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

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



Notes

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

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

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

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

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

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

- 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.

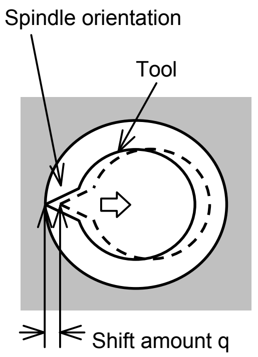
5.1.3 Fine Boring Cycle (G76)

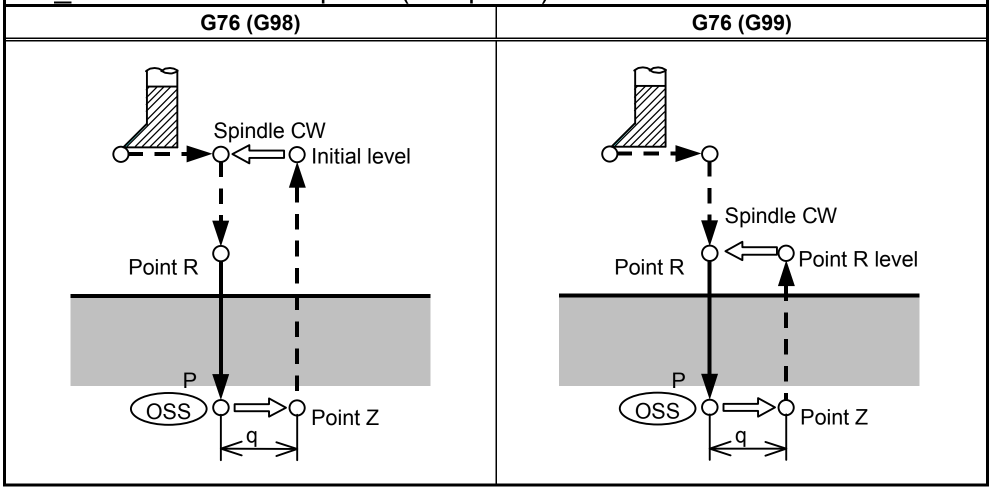
The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.

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)

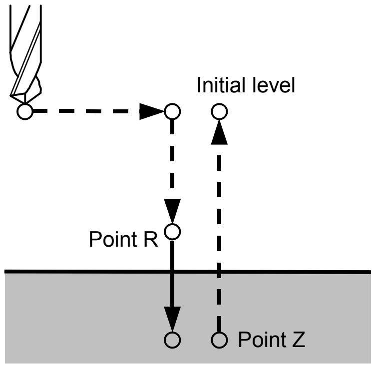
This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

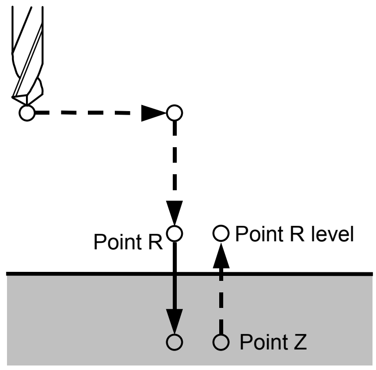
G81 X_ Y_ Z_ R_ F_ K_ ;

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)

G81 (G98)



G81 (G99)



Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

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

- Tool offset

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

Example

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G81 X300. 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.

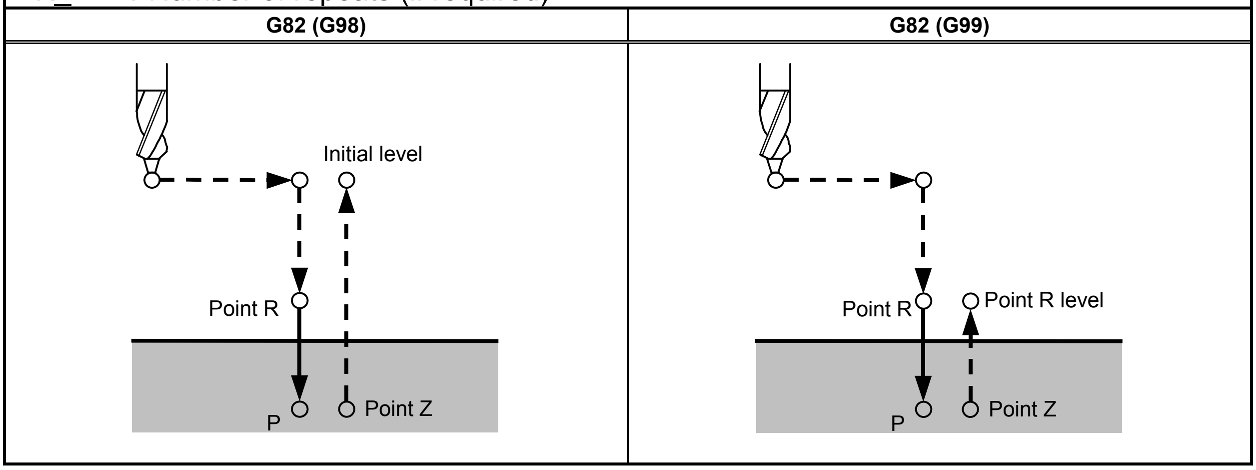
5.1.5 Drilling Cycle Counter Boring Cycle (G82)

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.
```

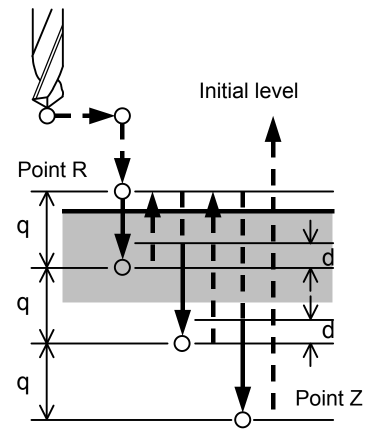
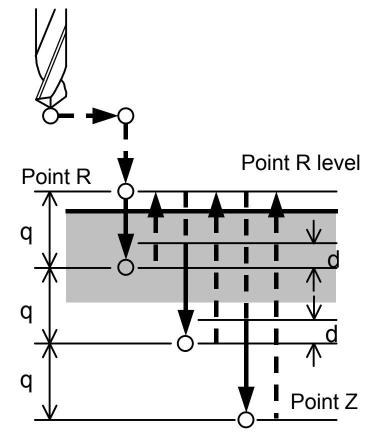
5.1.6 Peck Drilling Cycle (G83)

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)

G83 (G98)	G83 (G99)
	

Explanation

- Operations

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value. In the second and subsequent cutting feeds, rapid traverse is performed up to a 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. 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.

5.1.7 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.

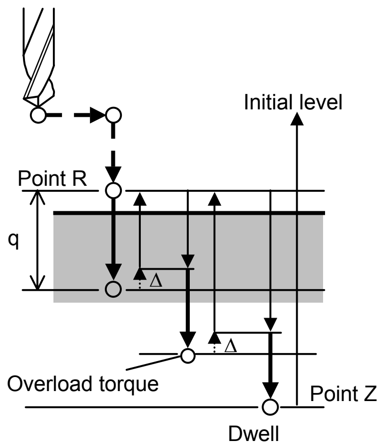
> **NOTE**
> When using the small-hole peck drilling cycle, set bit 4 (SPK) of parameter No. 8132 to "1".

Format

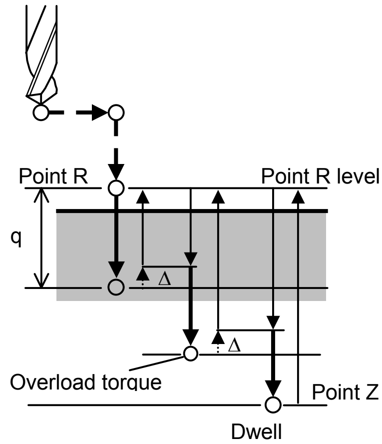
G83 X_ Y_ Z_ R_ Q_ F_ I_ K_ P_ ;

- 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 No.5172 and No.5173 are assumed as defaults.)
- K_ : Number of times the operation is repeated (if required)
- P_ : Dwell time at the bottom of the hole
  (If this is omitted, P0 is assumed as the default.)

G83 (G98)



G83 (G99)



Δ: Initial clearance when the tool is retracted to point R and the clearance from the bottom of the hole in the second or subsequent drilling (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
( )

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)
Repeated until point Z is reached	→	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, the signal indicating that the small–hole peck drilling cycle is in progress is output after the tool is positioned at the hole position along the axes not used for drilling. Signal output continues during positioning to point R along the drilling axis and terminates upon a return to point R or the initial level. 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 No.5166 and No.5167, specify the respective rates of change applied when the skip signal is detected and when it is not detected in the previous drilling operation.

Cutting feedrate = F × α

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

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

Spindle speed = S × γ

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

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

- Advance and retraction

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

- Specifying address I

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

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

Both commands indicate a speed of 1000 mm/min.

Address I specified with G83 in the continuous-state mode continues to be valid until G80 is specified or until a reset occurs.

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

- Functions that can be specified

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

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

- Single block

When single-block operation is enabled, drilling is stopped after each retraction. Also, a single block stop is performed by setting 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 No.5170 and No.5171. Parameters No.5170 and No.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.

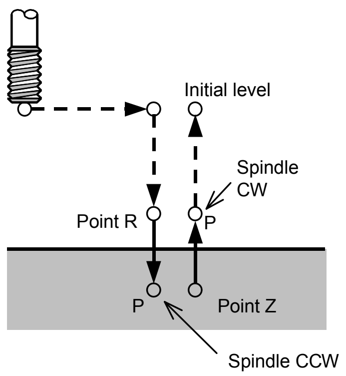
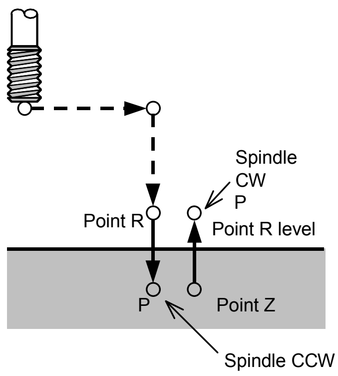
5.1.8 Tapping Cycle (G84)

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_ 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
P_ : Dwell time
Q_ : Depth of cut for each cutting feed (bit 6 (PCT) of parameter No. 5104 = "1")
F_ : Cutting feedrate
K_ : Number of repents (if required)

G84 (G98)	G84 (G99)
	

Explanation

- Operations

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

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

- Spindle rotation

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

- Q command

See “Left-Handed Tapping Cycle (G74)” above.

- 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. ; 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.
```

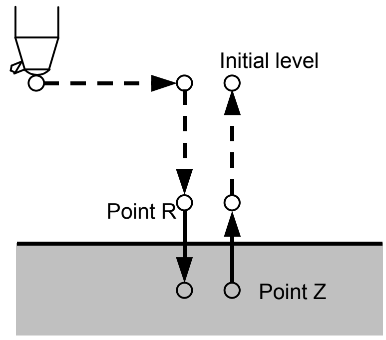
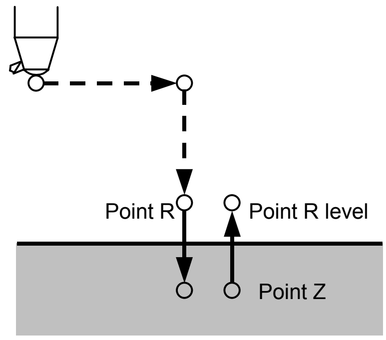
5.1.9 Boring Cycle (G85)

This cycle is used to bore a hole.

Format

G85 X_ Y_ Z_ R_ F_ K_ ;

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

G85 (G98)	G85 (G99)
	

Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

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

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

Example

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G85 X300. 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.

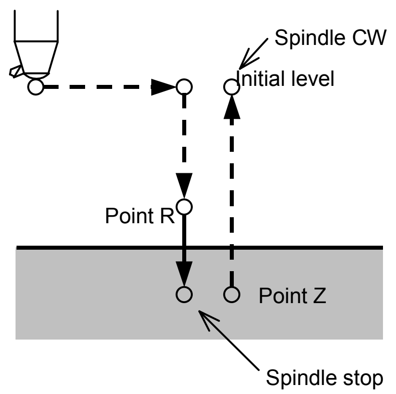
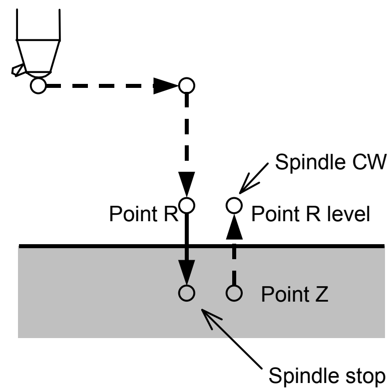
5.1.10 Boring Cycle (G86)

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)

G86 (G98)	G86 (G99)
	

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

- Spindle rotation
Before specifying G86, use an auxiliary function (M code) to rotate the spindle.

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

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

- Tool offset

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

Example

```
M3 S2000 ;                                   Cause the spindle to start rotating.
G90 G99 G86 X300. 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.
```

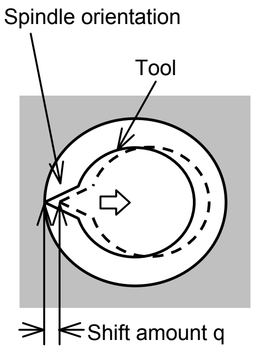
5.1.11 Back Boring Cycle (G87)

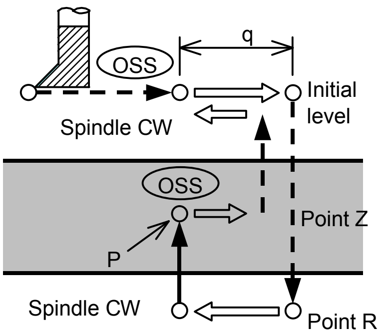
This cycle performs accurate boring.

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)

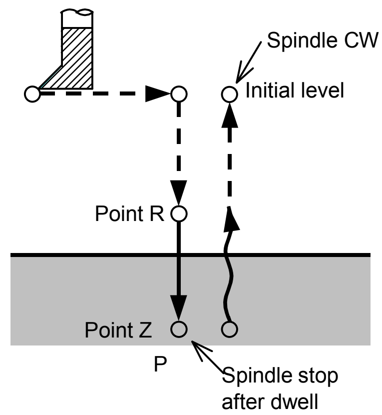
This cycle is used to bore a hole.

Format

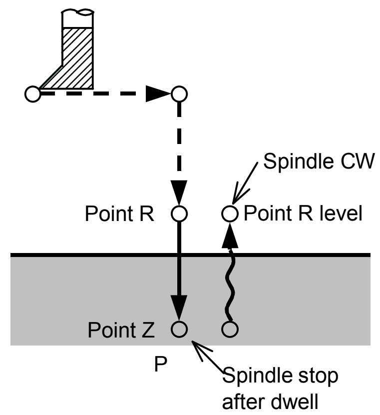
G88 X_ Y_ Z_ R_ P_ F_ K_ ;

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)

G88 (G98)



G88 (G99)



Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

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

- Cancel

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

- Tool offset

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

Example

```
M3 S2000 ;                   Cause the spindle to start rotating.
G90 G99 G88 X300. 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)

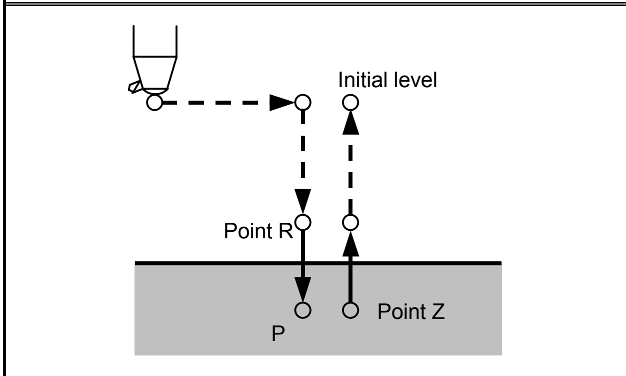
This cycle is used to bore a hole.

Format

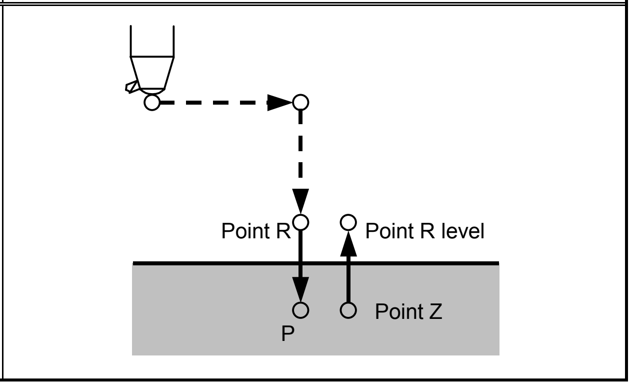
G89 X_ Y_ Z_ R_ P_ F_ K_ ;

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)

G89 (G98)



G89 (G99)



Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

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

- Cancel

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

- Tool offset

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

Example

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G89 X300. 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

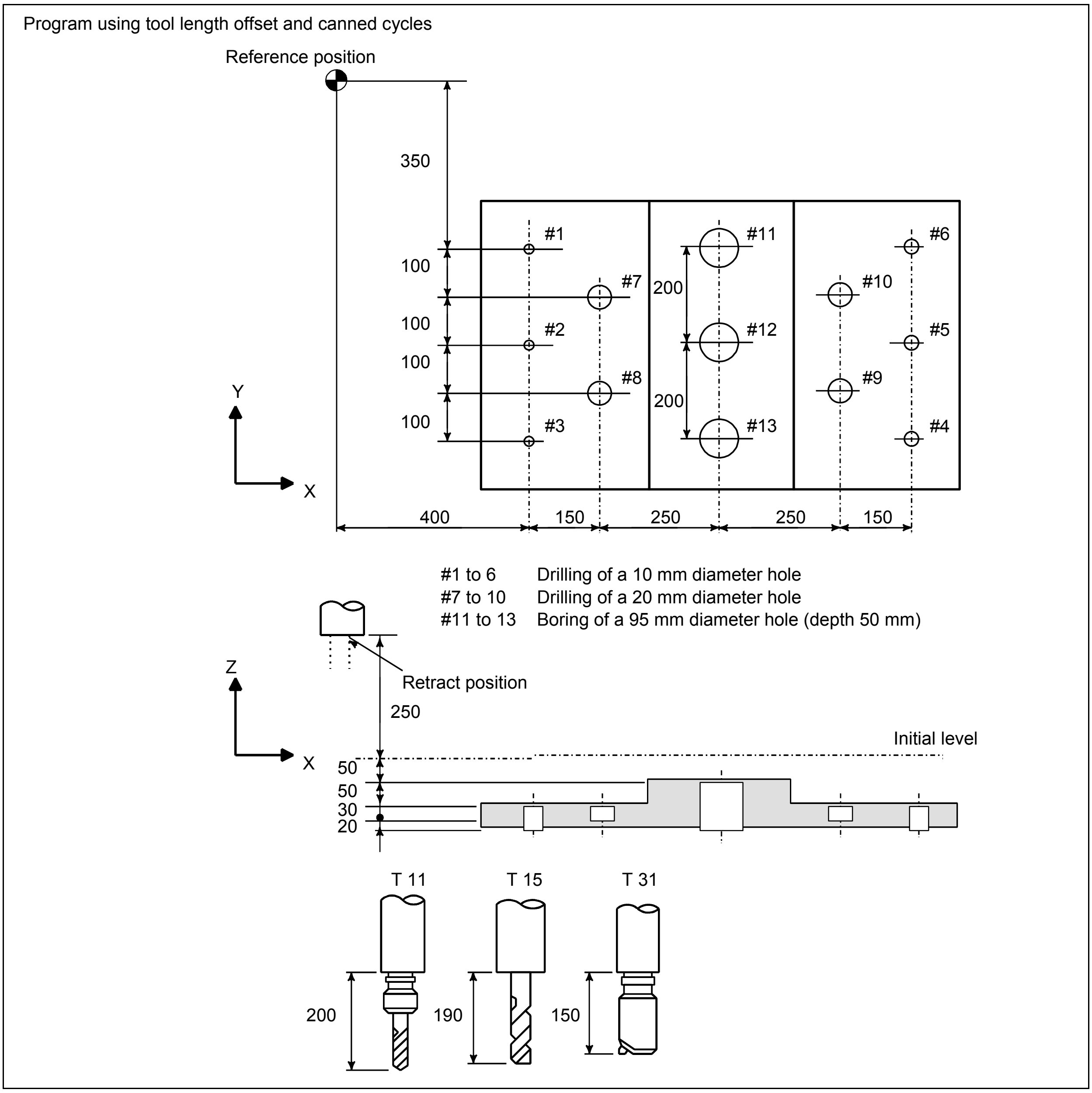


Fig. 5.1.15 (a) Example for using canned cycles for drilling

5.2 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.

5.2.1 Rigid Tapping (G84)

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

Format

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

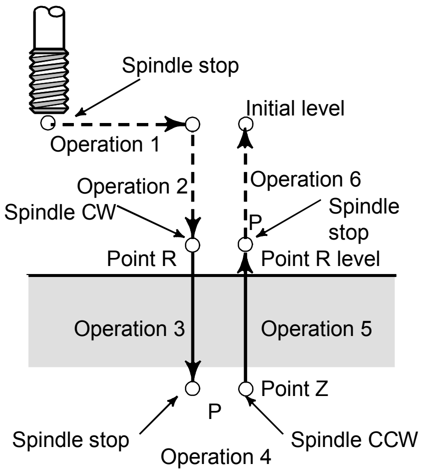
X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_	: The distance from the initial level to point R level
P_	: Dwell time at the bottom of the hole and at point R when a return is made
F_	: Cutting feedrate
K_	: Number of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

G84.2 X_ Y_ Z_ R_ P_ F_ L_ ;

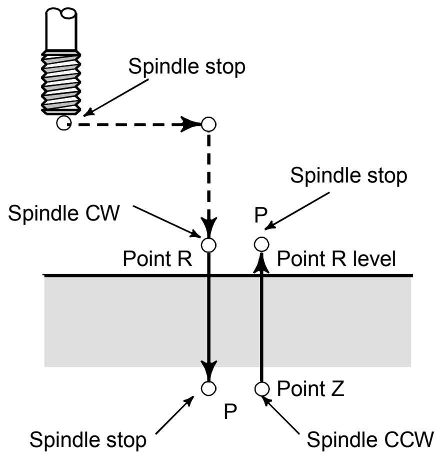
(Series 10/11 format)

L_	: Number of repeats (if required)

G84 (G98)



G84 (G99)



Explanation

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

- Rigid mode

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

- Specify M29 S***** before a tapping command.
- Specify M29 S***** in a block which contains a tapping command.
- Specify G84 for rigid tapping (parameter G84 No. 5200 #0 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 10/11 format command
Rigid tapping can be performed using Series 10/11 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

See “Override during Rigid Tapping” below for details.

- 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.

- 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 display 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) 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

5.2.2 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

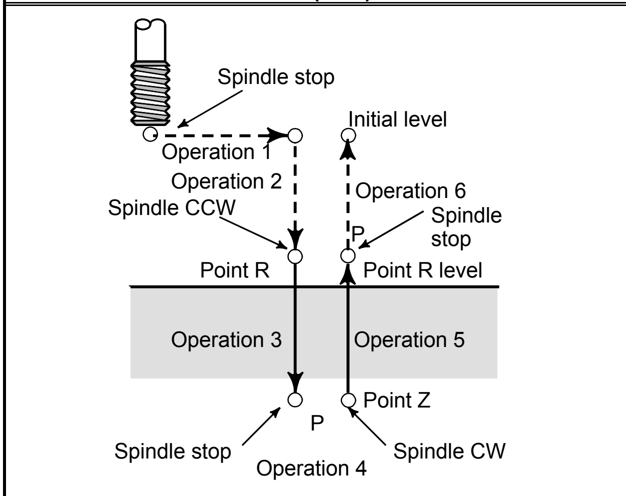
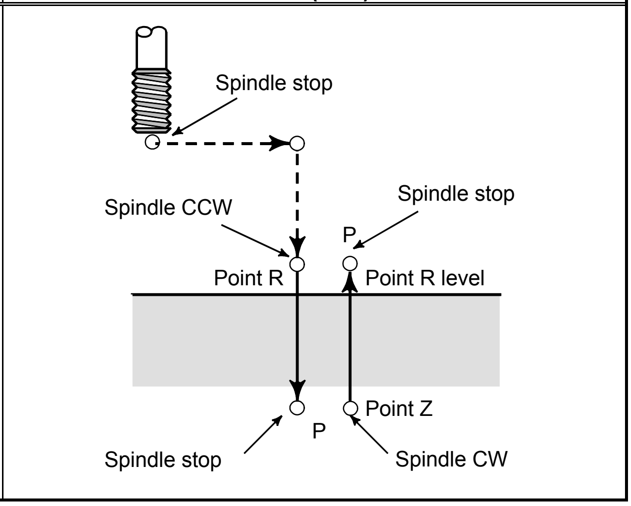
G74 X_ Y_ Z_ R_ P_ F_ K_ ;

- 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 10/11 format)

- L_ : Number of repeats (if required)

G74 (G98)	G74 (G99)
	

Explanation

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

- 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. (parameter G84 (No. 5200#0) set to1).

- 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 10/11 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

See “Override during Rigid Tapping” below for details.

- 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.

- 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 display 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) 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

5.2.3 Peck Rigid Tapping Cycle (G84 or G74)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.
In this cycle, cutting is performed several times until the bottom of the hole is reached. Two peck tapping cycles are available: High-speed peck tapping cycle and standard peck tapping cycle. These cycles are selected using the PCP bit (bit 5) of parameter 5200.

Format

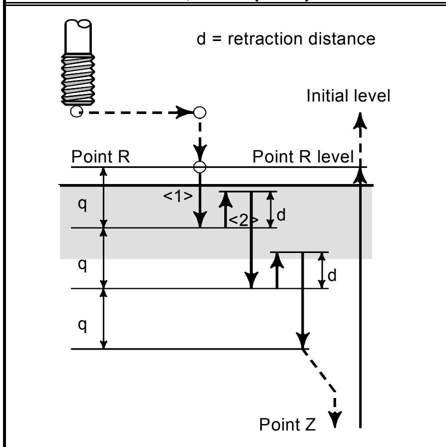
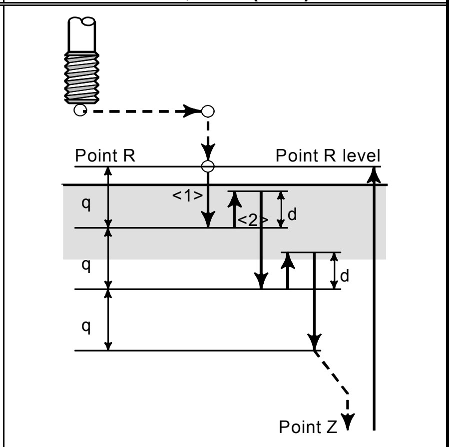
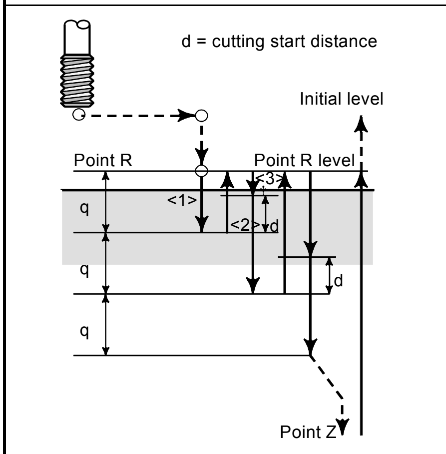
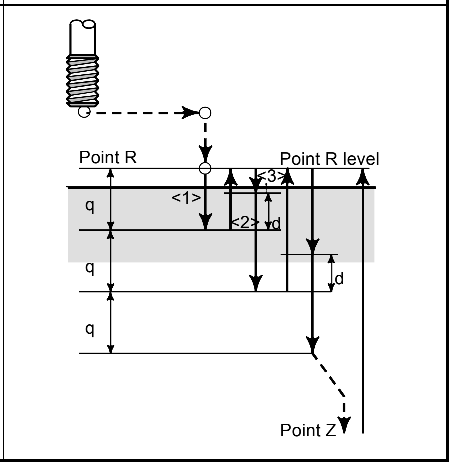
G84 (or G74) X_ Y_ Z_ R_ P_ Q_ F_ K_ ;

- 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 10/11 format)

- L_ : Number of repeats (if required)

G84, G74 (G98)	G84, G74 (G99)
	
	

- High-speed peck tapping cycle (Parameter PCP(No.5200#5)=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 (Parameter PCP(No.5200#5)=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 DOV bit (bit 4) of parameter 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 DOV bit (bit 4) of parameter 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 DOV bit (bit 4) of parameter 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 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

See “Override during Rigid Tapping” below for details.

- 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.

- 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 display 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) 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.

- d (parameter No.5213)

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

5.2.4 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.

5.2.5 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

5.2.5.1 Extraction override

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

Explanation

- Specifying the override in the parameter

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

- Specifying the override in a program

Set bit 4 (DOV) of parameter No. 5200 and bit 4 (OV3) of parameter No. 5201 to 1. The spindle speed at extraction can be specified in the program.
Specify the spindle speed at extraction using address "J" in the block in which rigid tapping is specified.
Example) To specify 1000 min-1 for S at cutting and 2000 min-1 for S at extraction

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

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

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

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

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

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

Command \ Parameter setting		DOV = 1, OV3 = 1	DOV = 1, OV3 = 0	DOV = 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		

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

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

NOTE

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

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

3 The maximum override is obtained using the following equation so that the spindle speed to which override at extraction is applied do not exceed the maximum used gear speed (specified in parameters Nos. 5241 to 5243). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

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

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

5.2.5.2 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
- 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 5243). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

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

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

5.3 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

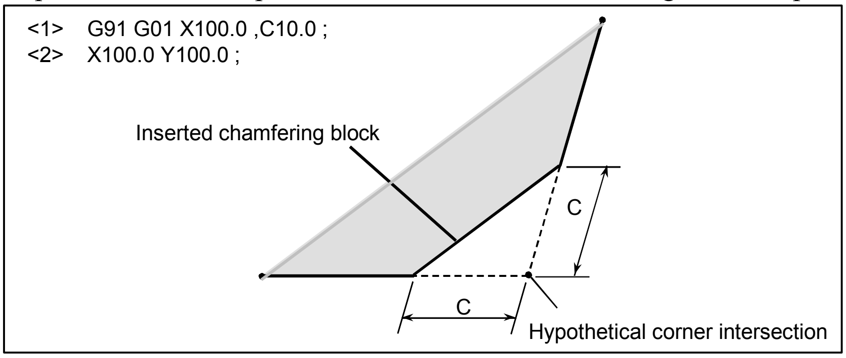
, C_	Chamfering
, R_	Corner R

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.

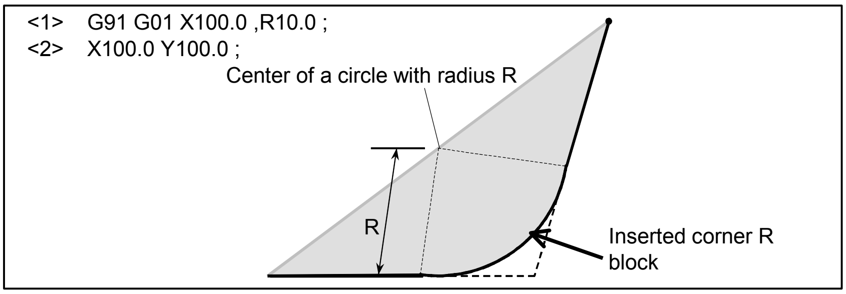
- Chamfering

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.



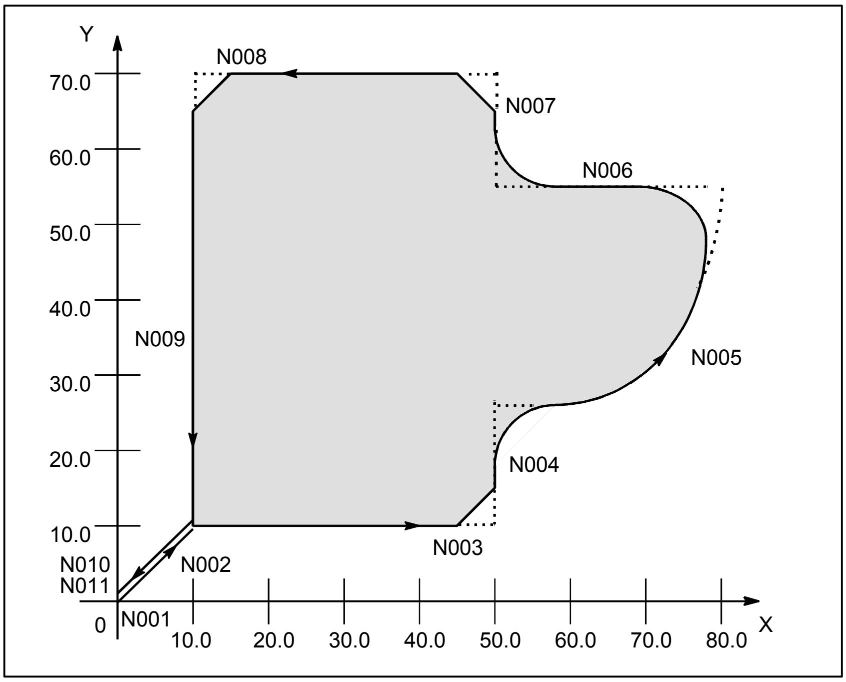
- Corner R

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 Y50.0 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.

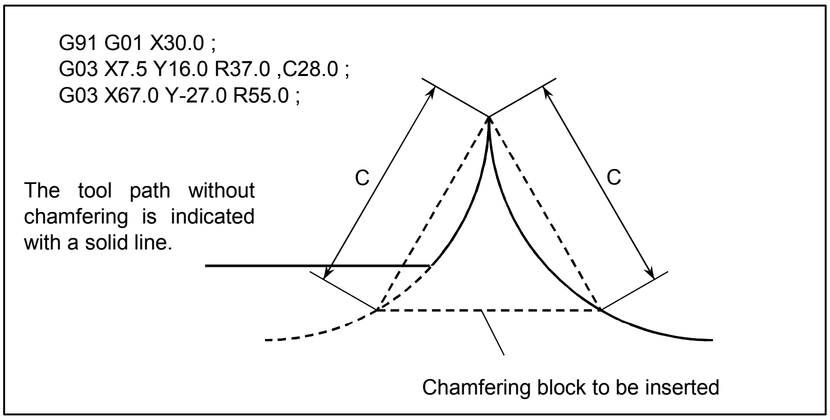


Fig 5.3 (a) Exceeding the move range

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

- 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.

- Unusable G codes

The following G codes are unusable in the same block as for chamfering or corner R commands or with a block for chamfering or corner R inputs that define continuous figures.

- G codes (except G04) in group 00
- G68 in group 16

- Threading

If ",C" or ",R" is specified in a threading command block, the alarm PS0050 is issued.

NOTE

When ",C" and ",R" are specified in the same block, the address specified last is valid.

5.4 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 .

> **NOTE**
> To enable the index table indexing function, reset bit 0 (ITI) of parameter No. 5501 to "0" and set bit 3 (IXC) of parameter No. 8132 to "1".

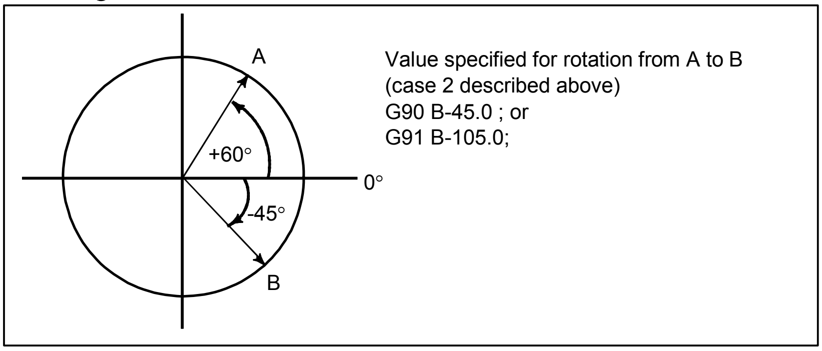
Explanation

- Indexing position

Specify an indexing position with address A, B, or C (set to bit 0 of parameter ROTx 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 (Bit 4 (G90) of parameter No.5500 =1)
2. Absolute or incremental value depending on the specified G code: G90 or G91 (Bit 4 (G90) of parameter No.5500 =0)

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) (Auxiliary 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 of parameter ABS 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, 3, and 4 of parameter ABS, INC,G90 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 of parameter ABS 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 of parameter ABS 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**
> If a reset is made during indexing of the index table, a reference position return must be made before each time the index table is indexed subsequently.

> **NOTE**
> 1 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 display 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 5.4 (a) Index indexing function and other functions

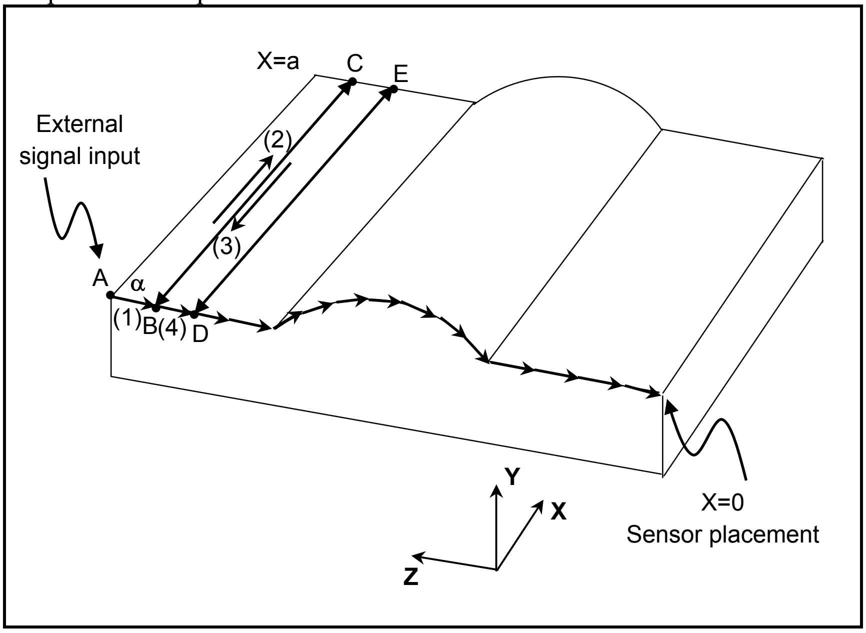
Item	Explanation
Relative position display	This value is rounded down when bit 1 of parameter REL No.5500 specifies this option.
Absolute position display	This value is rounded down when bit 2 of parameter ABS No.5500 specifies this option.
Movement in the machine coordinate system (G53)	Impossible to move
Single direction positioning	Impossible to specify
2nd auxiliary function (B code)	Possible with any address other than B that of the indexing axis.
Operations while moving the indexing axis	Unless otherwise processed by the machine, feed hold, interlock and emergency stop can be executed. Machine lock can be executed after indexing is completed.
SERVO OFF signal	Disabled The indexing axis is usually in the servo-off state.
Incremental commands for indexing the index table	The workpiece coordinate system and machine coordinate system must always agree with each other on the indexing axis (the workpiece zero point offset value is zero.).

Item	Explanation
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.

5.5 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.



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 figure above.
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 figure above.

> **NOTE**
> In-feed control function is optional function.

Format

G161 R_ ;

Profile

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

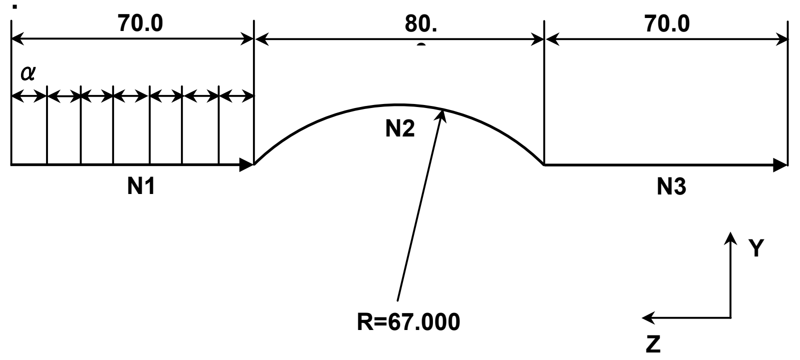
In this operation mode, block overlap is disabled.

- 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

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



The program above causes the machine to move by 10.000 along the machining profile in the figure above 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.

5.6 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
- 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.

5.6.1 Plunge Grinding Cycle (G75)

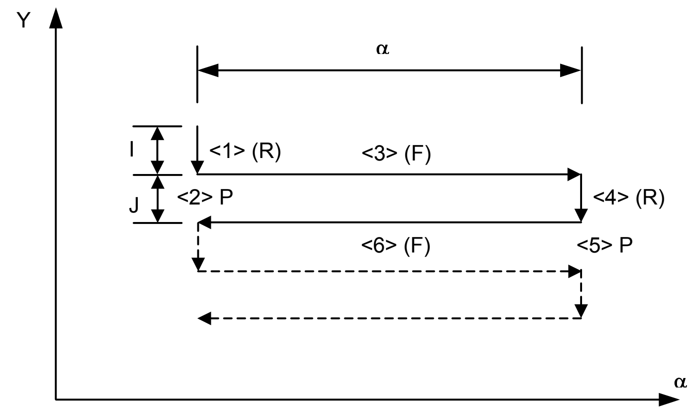
A plunge grinding cycle can be executed.

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

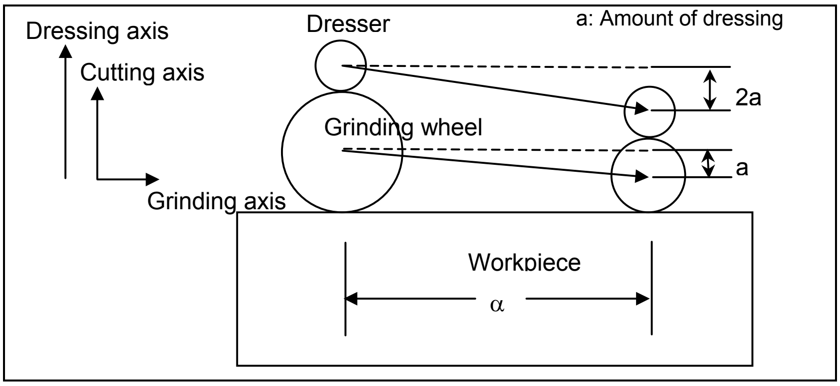
Performs a dwell for the time specified with P.

<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.



> **NOTE**
> Continuous dressing function is optional function.

Limitations

- Cutting axis

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

- Grinding axis

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

- Dressing axis

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

- α,I,J,K

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

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

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

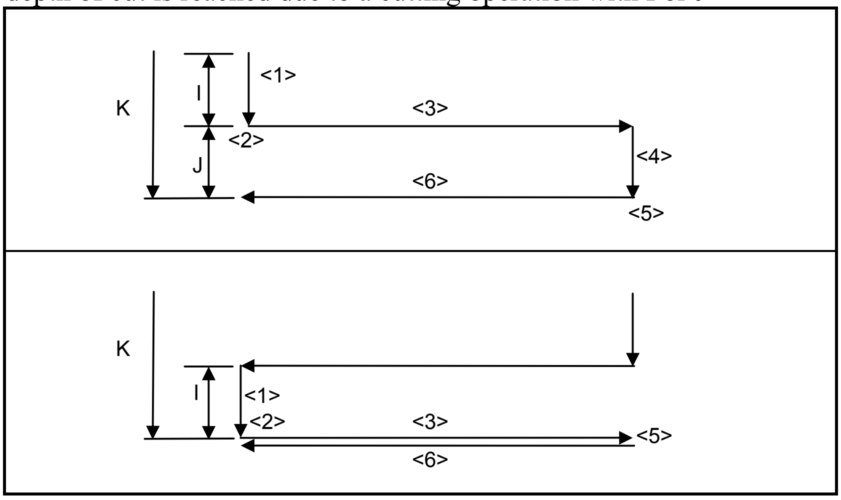
- Clearing

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

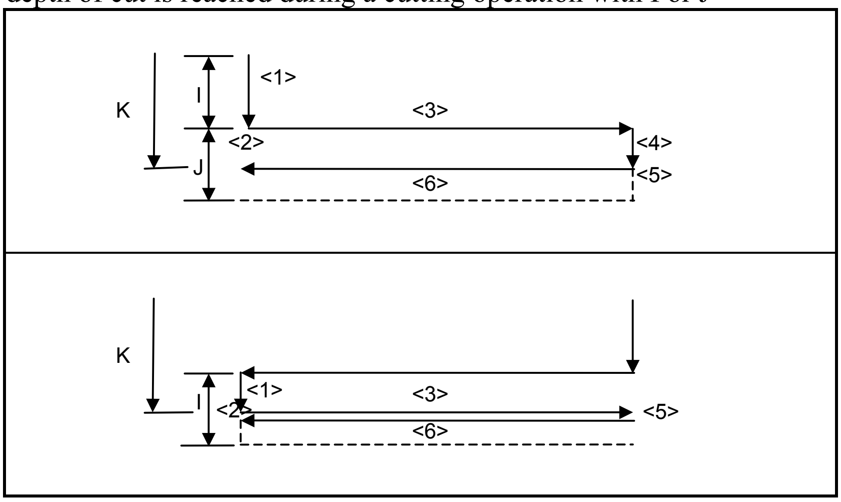
- Operation to be performed if the total depth of cut is reached

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

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



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



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

5.6.2 Direct Constant-Dimension Plunge Grinding Cycle (G77)

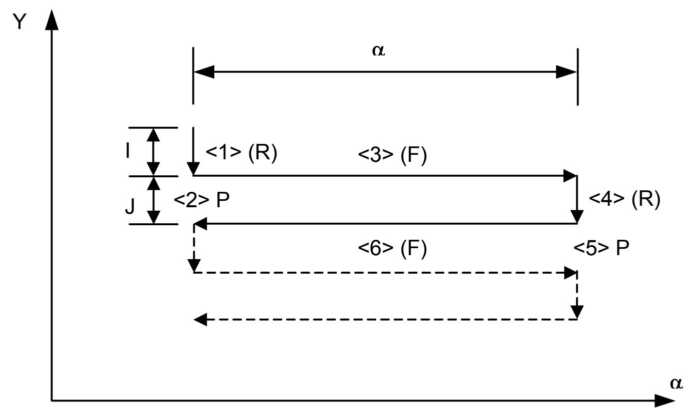
A direct constant-dimension plunge grinding cycle can be performed.

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.

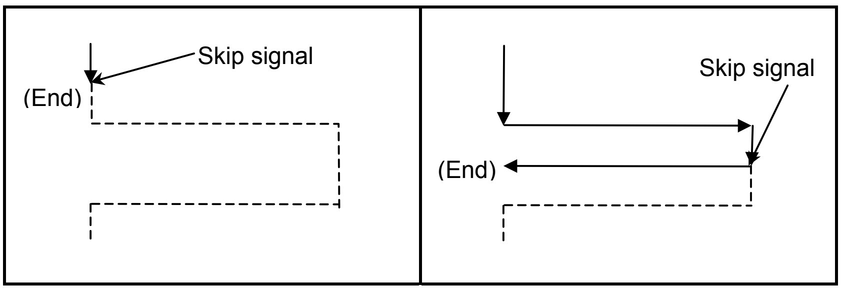
- Continuous dressing

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

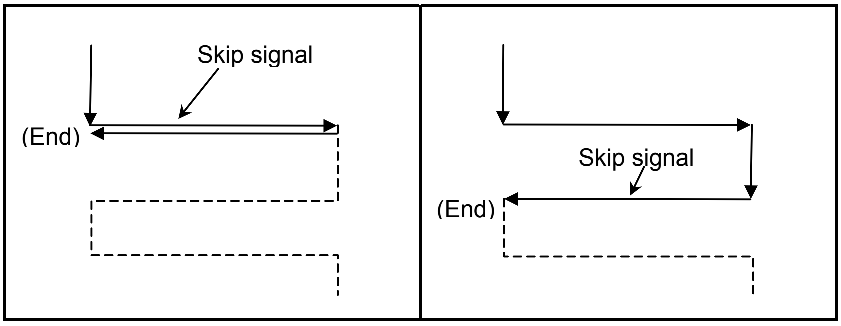
With G77, by inputting a skip signal in a cycle, it is possible to end the cycle after interrupting the current operation sequence (or after ending the current operation sequence).
The following shows the operation to be performed when a skip signal is input in each operation sequence.

- 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.

5.6.3 Continuous-feed Surface Grinding Cycle (G78)

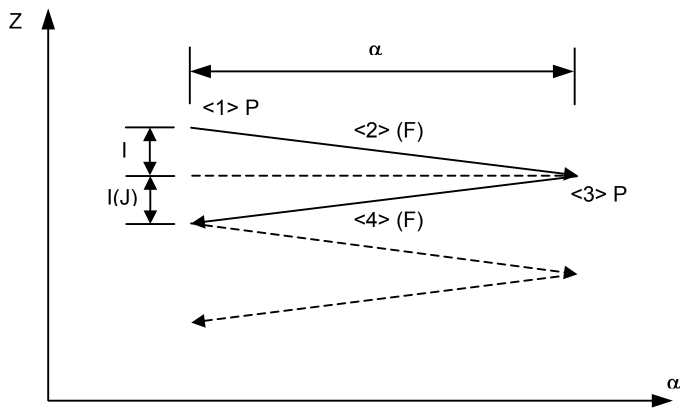
A continuous-feed surface grinding cycle can be performed.

Format

G78 I_ (J_) K_ α_ F_P_ L_ ;

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

G78



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.

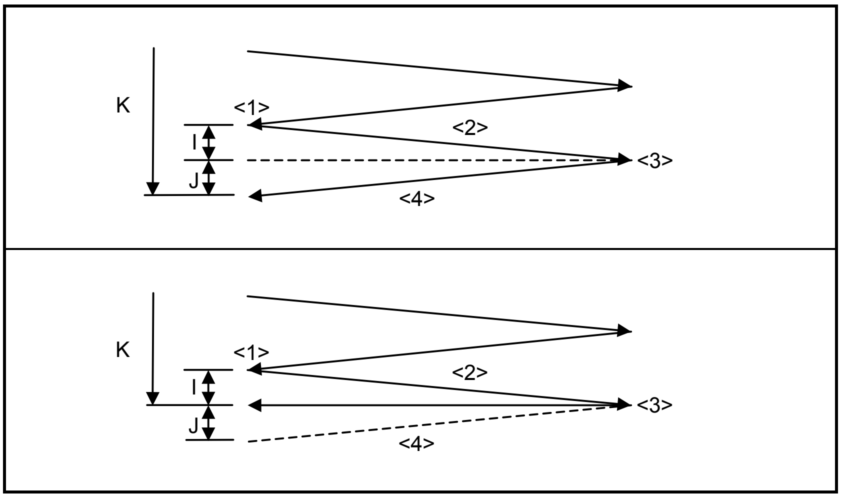
- Clearing

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.

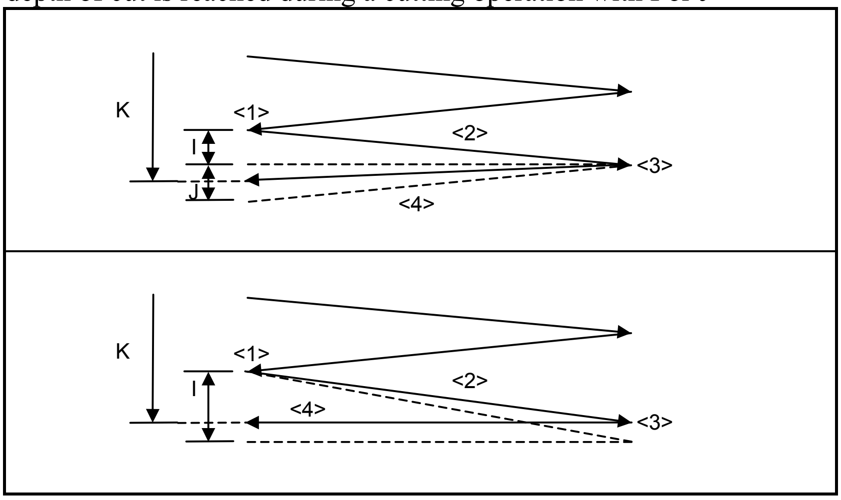
- Operation to be performed if the total depth of cut is reached

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.

5.6.4 Intermittent-feed Surface Grinding Cycle (G79)

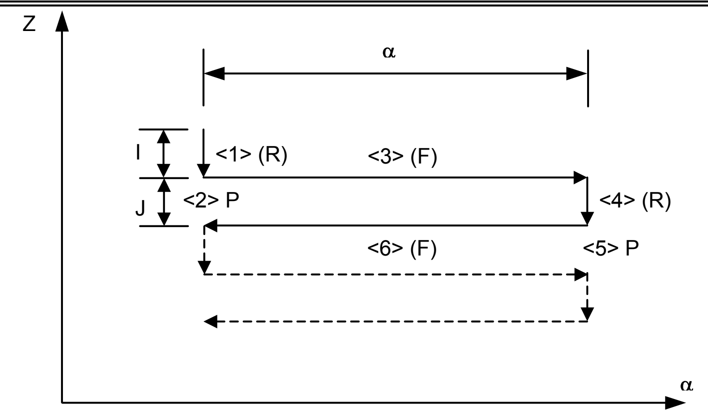
An intermittent-feed surface grinding cycle can be performed.

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

6 COMPENSATION FUNCTION

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

6.1 TOOL LENGTH COMPENSATION (G43, G44, G49)

This function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.
Specify the direction of offset with G43 or G44. Select a tool length compensation value from the offset memory by entering the corresponding address and number (H code).

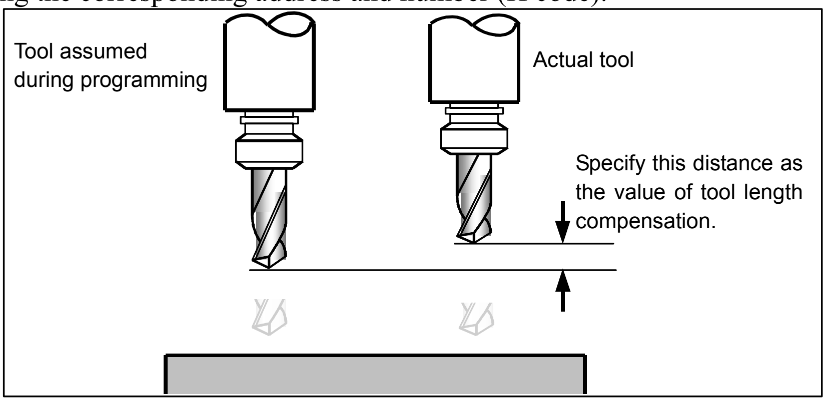


Fig. 6.1 (a) Tool length compensation

6.1.1 Overview

The following three methods of tool length compensation can be used, depending on the axis along which tool length compensation can be made.

- Tool length compensation A
  Compensates for the difference in tool length along the basic Z-axis.
- Tool length compensation B
  Compensates for the difference in tool length in the direction normal to a selected plane.
- Tool length compensation C
  Compensates for the difference in tool length along a specified axis.

Format

Type	Format	Description
Tool length compensation A	**G43 Z_ H_ ;** **G44 Z_ H_ ;**	G43 : Positive offset G44 : Negative offset G17 : XY plane selection G18 : ZX plane selection G19 : YZ plane selection α : Address of a specified axis H : Address for specifying the tool length compensation value X, Y, Z : Offset move command
Tool length compensation B	**G17 G43 Z_ H_ ;** **G17 G44 Z_ H_ ;** **G18 G43 Y_ H_ ;** **G18 G44 Y_ H_ ;** **G19 G43 X_ H_ ;** **G19 G44 X_ H_ ;**	
Tool length compensation C	**G43 α_H_ ;** **G44 α_H_ ;**	
Tool length compensation cancel	**G49 ; or H0 ;**	

Explanation

- Selection of tool length compensation

Select tool length compensation A, B, or C, by setting bits 1 (TLB) and 0 (TLC) of parameter No.5001 .

Parameter No.5001		Type
Bit 1 (TLB)	**Bit 0 (TLC)**	
0	0	Tool length compensation A
1	0	Tool length compensation B
0/1	1	Tool length compensation C

- Direction of the offset

When G43 is specified, the tool length compensation value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.

When the specification of an axis is omitted, a movement is made by the tool length compensation value.

G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

- Specification of the tool length compensation value

The tool length compensation value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the moving command in the program.

Example

```
  :
H1 ;       The offset value of offset number 1 is selected.
  :
G43 Z_ ;   Offset is applied according to the offset value of offset number 1.
  :
H2 ;       Offset is applied according to the offset value of offset number 2.
  :
H0 ;       Offset is applied according to the offset value 0.
  :
H3 ;       Offset is applied according to the offset value of offset number 3.
  :
G49 ;      Offset is canceled.
  :
H4 ;       The offset value of offset number 4 is selected.
  :
```

A tool length compensation value is to be set in the offset memory corresponding to an offset number.

⚠ **WARNING**

When another offset number is specified, the tool length compensation value just changes to a new value. The new tool length compensation value is not added to the old tool length compensation value.
H1 : Tool length compensation value 20.0
H2 : Tool length compensation value 30.0
G90 G43 Z100.0 H1 ; Z will move to 120.0
G90 G43 Z100.0 H2 ; Z will move to 130.0

NOTE

The tool length compensation value corresponding to offset No. 0, that is, H0 always means 0. It is impossible to set any other tool length compensation value to H0.

- Performing tool length compensation along two or more axes

Tool length compensation B can be executed along two or more axes when the axes are specified in two or more blocks.
By setting bit 3 (TAL) of parameter No. 5001 to 1, tool length compensation C can also can be executed along two or more axes when the axes are specified in two or more blocks. If no axis is specified in the same block, the alarm PS0027 is issued. If two or more axes are specified in the same block, the alarm PS0336 is issued.

Example 1

When tool length compensation B is executed along the X-axis and Y-axis

G19 G43 H_ ; Offset in X axis

G18 G43 H_ ; Offset in Y axis

Example 2

When tool length compensation C is executed along the X-axis and Y-axis

G43 X_ H_ ; Offset in X axis

G43 Y_ H_ ; Offset in Y axis

Example 3

When an alarm is issued with tool length compensation C

G43 X_ Y_ H_ ; An alarm (PS0336) occurs

- Tool length compensation cancel

To cancel tool length compensation, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

NOTE

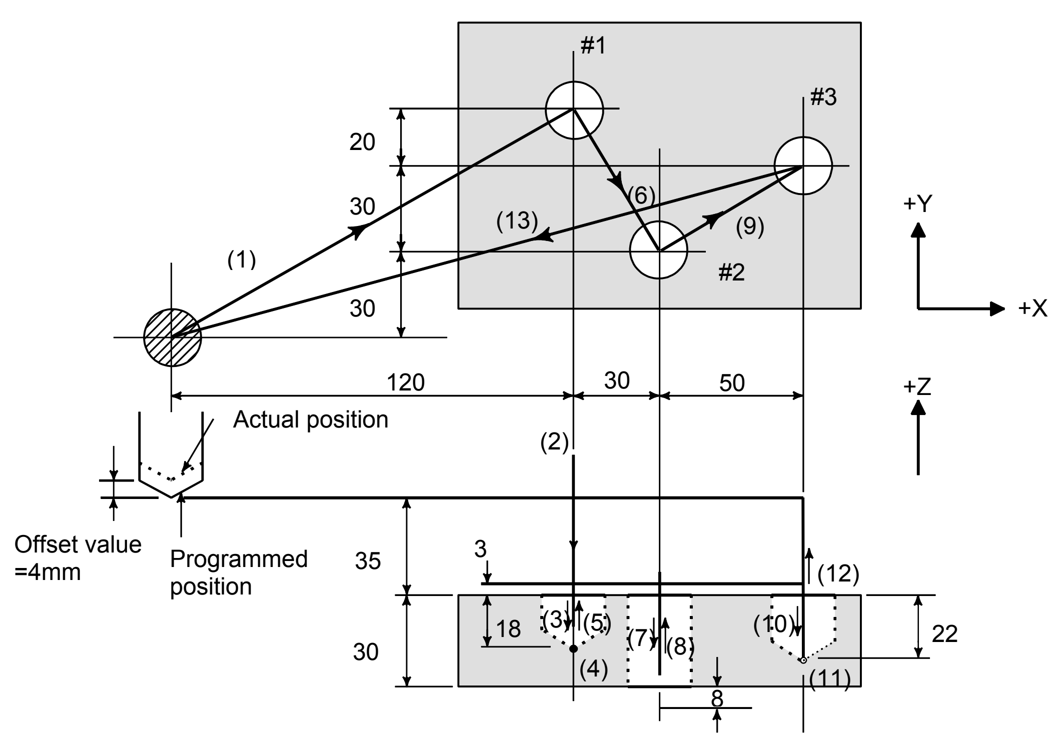
1 If offset is executed along two or more axes, offset along all axes is canceled by specifying G49. If H0 is used to specify cancellation, offset along only the axis normal to a selected plane is canceled in the case of tool length compensation B, or offset along only the last axis specified by G43 or G44 is canceled in the case of tool length compensation C.

2 If offset is executed along three or more axes, and offset along all axes is canceled using G49, the alarm PS0015 (TOO MANY SIMULTANEOUS AXES) may be issued. By using H0 together, for example, cancel offset so that the number of simultaneously controlled axes (the number of axes along which movements are made simultaneously) does not exceed the allowable range of the system.

3 When H is specified as an address for setting a compensation number in cutter compensation (G40, G41, or G42) (bit 2 (OFH) of parameter No.5001 = “1”), G49 (tool length compensation cancel) is performed in the block if G49 (tool length compensation cancel) is specified in the same block as for G40 (cutter compensation cancel).

Example

Tool length compensation (in boring holes #1, #2, and #3)



Program
H1=-4.0 (Tool length compensation value)
N1 G91 G00 X120.0 Y80.0 ; .... (1)
N2 G43 Z-32.0 H1 ; .... (2)
N3 G01 Z-21.0 F1000 ; .... (3)
N4 G04 P2000 ; .... (4)
N5 G00 Z21.0 ; .... (5)
N6 X30.0 Y-50.0 ; .... (6)
N7 G01 Z-41.0 ; .... (7)
N8 G00 Z41.0 ; .... (8)
N9 X50.0 Y30.0 ; .... (9)
N10 G01 Z-25.0 ; .... (10)
N11 G04 P2000 ; .... (11)
N12 G00 Z57.0 H0 ; .... (12)
N13 X-200.0 Y-60.0 ; .... (13)
N14 M2 ;

Notes

- Command for setting a workpiece coordinate system in the tool length compensation mode

Executing a workpiece coordinate system setting G code command (G92) presets a coordinate system in such a way that the specified position will be a pre-compensation position.
However, this G code cannot be used together with a block where tool length compensation vectors vary. Refer to Notes in "Work Coordinate System Setting" of the Operator’s Manual (Common) for details.

- Bit 2 (OFH) of parameter No.5001

If bit 2 (OFH) of parameter No. 5001 is set, cutter compensation takes precedence over tool length compensation. Concrete explanations follow:

If OFH = "0":
- Processing is carried out properly according to a selected modal state (G43, G44, or G49).

If OFH = "1":
- In a block where G40, G41, or G42 is specified, tool length compensation is disabled.
- In the G40 mode, processing is carried out properly according to a selected modal state (G43, G44, or G49).
- In the G41 and G42 modes, tool length compensation is enabled only in a block in which G43, G44, or G49 is specified. No compensation amount is updated only with the H code.
  G49 is enabled if G49 is specified in the same block as for G40, however.

6.1.2 G53, G28, and G30 Commands in Tool Length Compensation Mode

This section describes the tool length compensation cancellation and restoration performed when G53, G28, or G30 is specified in tool length compensation mode. Also described is the timing of tool length compensation.

Explanation

- Tool length compensation vector cancellation

When G53, G28, or G30 is specified in tool length compensation mode, tool length compensation vectors are canceled as described below. However, the previously specified modal G code remains displayed; modal code display is not switched to G49.

(1) When G53 is specified

Command	Specified axis	Operation
G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Not canceled
G49 G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Canceled

(IP_ : Dimension word)

> ⚠ **CAUTION**
> If tool length compensation is applied along multiple axes, the offset vector along the axis specified by G53 is canceled.

(2) When G28 or G30 is specified

Command	Specified axis	Operation
G28 IP_	Tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
	Other than tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
G49 G28 IP_	Tool length compensation axis	Canceled when a movement is made to an intermediate point.
	Other than tool length compensation axis	Canceled when a movement is made to an intermediate point.

(IP_ : Dimension word)

> ⚠ **CAUTION**
> If tool length compensation is applied along multiple axes, the offset vector along the axis on which a reference position return operation has been performed is canceled.

- Tool length compensation vector restoration

Tool length compensation vectors, canceled by specifying G53, G28, or G30 in tool length compensation mode, are restored as described below.

Type	Parameter EVO (No.5001#6)	Restoration condition
A/B	0	The H command or G43 (G44) is specified.
	1	Restored by the next buffered block.
C		The H command or G43 (G44) IP_ is specified.

(IP_ : Dimension word)

> ⚠ **CAUTION**
> If a tool length compensation vector is restored only with H_, G43, or G44 when tool length compensation is applied along multiple axes, the tool length compensation vector along only the axis normal to a selected plane is restored in the case of tool length compensation B, or the tool length compensation vector along only the last axis for which tool length compensation is specified is restored in the case of tool length compensation C. The tool length compensation vector along any other axes is not restored.

> **NOTE**
> In a block in which G40, G41, or G42 is specified, no tool length compensation vector is restored.

6.2 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 parameter TOS (parameter No. 5006#6). 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 parameters TLC and TLB (No. 5001#0, #1).

- 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 TAL (No. 5001#3) 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), and if the start or cancellation (G43, G44, G49, or H0) of a tool length compensation is specified in cutter compensation mode (G41,G42), 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.

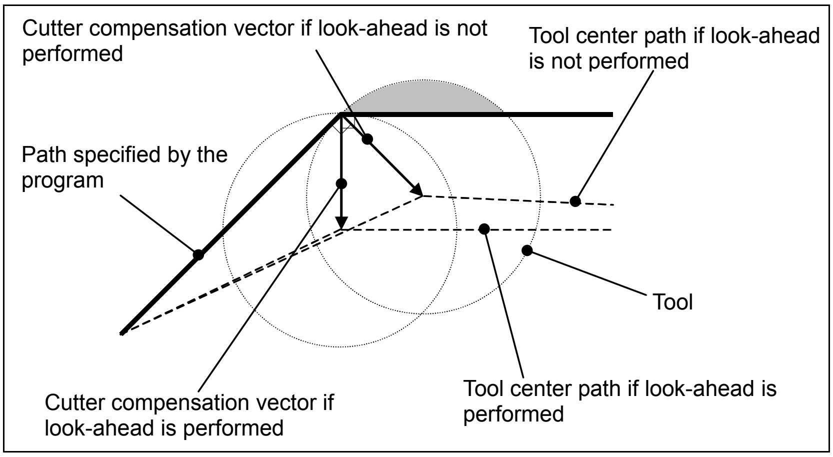
Example in which overcutting occurs in cutter compensation)

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 is a command for starting tool length compensation. However, blocks N2 and the subsequent blocks are preread because the current mode is not cutter compensation. 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), it is possible to select the operation to be performed if the tool length compensation is changed (*1) in cutter compensation (G41,G42) and in tool length (G43,G44), 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 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 to change the direction of tool length compensation
- Tool compensation amount changed using the offset screen, G10 command, system variable, or window function when bit 6 (EVO) of parameter No. 5001 = "1"
- Restoration of a tool length compensation vector that was temporarily canceled using G53, G28, or G30 during tool length compensation

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

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

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

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

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

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

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

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

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

⚠ **CAUTION**

1 Specifying tool length compensation (a shift type) first and then executing an incremental programming causes the tool length compensation value to be reflected in the coordinates only, not in the travel distance of the machine; executing an absolute programming causes the tool length compensation value to be reflected in both the movement of the machine and the coordinates.
2 If a programmable mirror image is effective, the tool length compensation is applied in the specified direction.
3 No scaling magnification is applied to the tool length compensation value.
4 No coordinate system rotation is applied to the tool length compensation value. Tool length compensation is effective in the direction in which the offset is applied.
5 With the WINDOW command, changing parameter TOS during automatic operation does not cause the tool length compensation type to be changed.
6 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.
7 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.

⚠ CAUTION

8 If reference position return (G28 or G30) has been specified, tool length compensation is canceled for the axis specified at the time of positioning on the reference point; however, tool length compensation is not canceled for an un-specified axis. If reference position return has been specified in the same block as that containing tool length compensation cancel (G49), tool length compensation is canceled for both the specified and un-specified axes at the time of positioning on the mid-point.

9 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.

10 The tool length compensation vector canceled by specifying G53, G28, or G30 during tool length compensation is restored as described below:

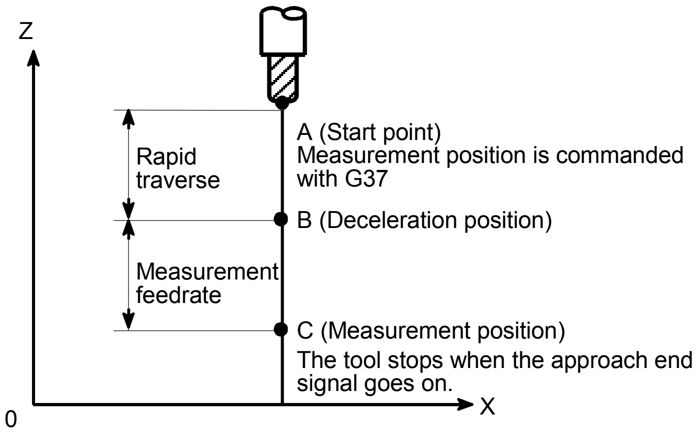
- For tool length compensation types A and B, if parameter EVO (No. 5001#6) is 1, the vector is restored in the block buffered next; for all of tool length compensation types A, B, and C, it is restored in a block containing an H, G43, or G44 command if parameter is 0.

11 When a tool length compensation shift type is used, if the start or cancellation of a tool length compensation or other command is specified cutter 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 cutter compensation mode or at a location where machining is not affected.

6.3 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.



Compensation value = (Current compensation value) + [(Coordinates of the point at which the tool is stopped) - (Coordinates of the programmed measure ment position)]

Fig. 6.3 (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 federate halfway, then continuous to move it until the approach end signal from the measuring instrument is issued. When the tool nose reaches the measurement position, the measuring instrument sends an approach end signal to the CNC which stops the tool.

- Changing the offset value

The difference between the coordinates of the position at which the tool reaches for measurement and the coordinates specified by G37 is added to the current tool length compensation value. (If parameter MDC (No. 6210#6) is 1, it is subtracted.)
Offset value =
(Current compensation value) + [(Coordinates of the position at which the tool reaches for measurement) - (Coordinates specified by G37)]
These offset values can be manually changed from MDI.

- Alarm

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

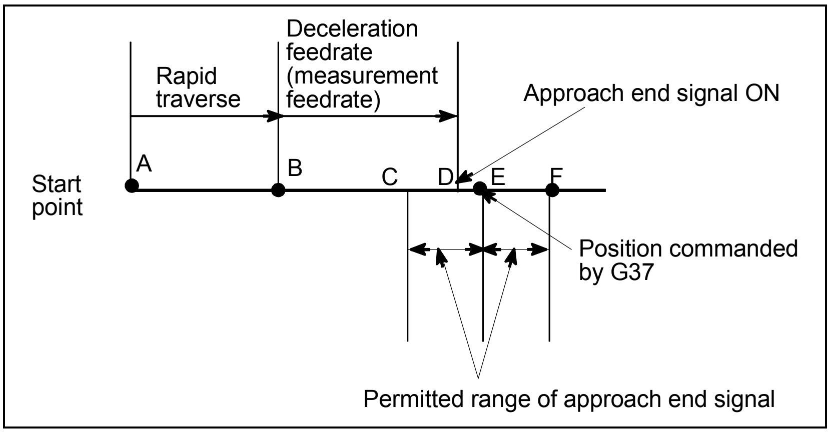


Fig. 6.3 (b) Tool movement to the measurement position

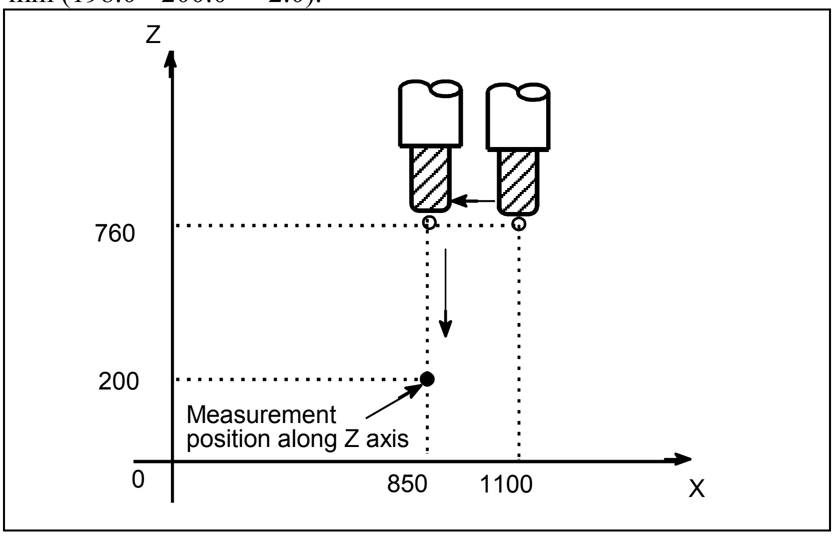
> ⚠ **CAUTION**
> When a manual movement is inserted into a movement at a measurement federate, return the tool to the position before the inserted manual movement for restart.

> **NOTE**
> 1 When an H code is specified in the same block as G37, an alarm is generated. Specify H code before the block of G37.
> 2 The measurement speed (FP), γ, and ε are set as parameters (FP: No. 6241, γ: No. 6251, ε: No. 6254) by the machine tool builder. Make settings so that e are always positive and γ are always greater than ε.
> 3 When tool offset memory A is used, the offset value is changed. When tool offset memory C is used, the tool wear compensation value for the H code is changed.
> 4 A delay or variation in detection of the measurement position arrival signal is 0 to 2 msec on the CNC side excluding the PMC side. Therefore, the measurement error is the sum of 2 msec and a delay or variation (including a delay or variation on the receiver side) in propagation of the skip signal on the PMC side, multiplied by the feedrate set in parameter No. 6241.
> 5 A delay or variation in time after detection of the measurement position arrival signal until a feed stops is 0 to 8 msec. To calculate the amount of overrun, further consider a delay in acceleration/deceleration, servo delay, and delay on the PMC side.

Example

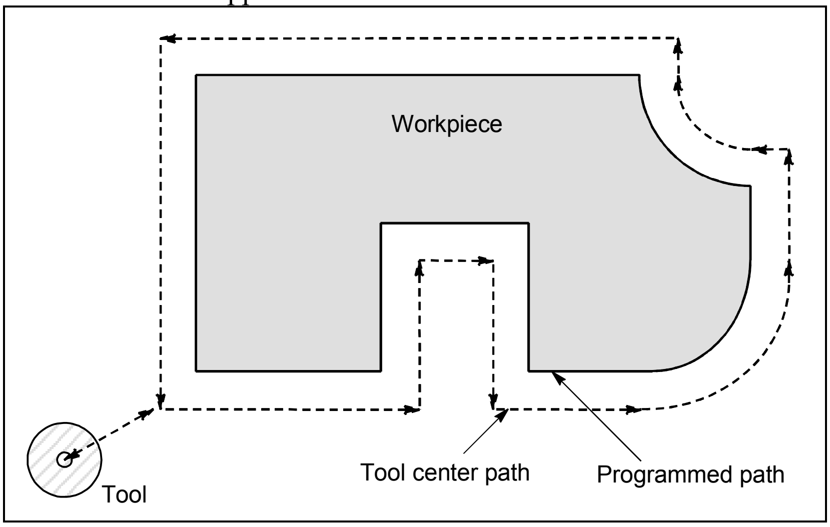
G92 Z760.0 X1100.0 ;	Sets a workpiece coordinate system with respect to the programmed absolute zero point.
G00 G90 X850.0 ;	Moves the tool to X850.0. That is the tool is moved to a position that is a specified distance from the measurement position along the Z-axis.
H01 ;	Specifies offset number 1.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

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



6.4 TOOL OFFSET (G45 - 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

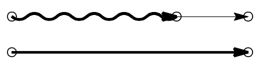
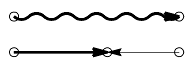
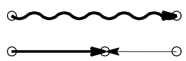
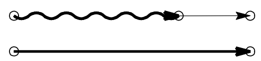
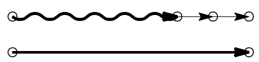
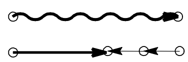
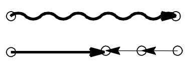
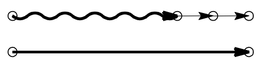
* If bit 2 (OFH) of parameter No. 5001 ="0", setting bit 5 (TPH) of parameter No. 5001 to "1" enables address H to be used as a code for specifying a tool position offset value.

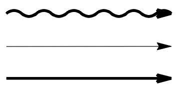
Explanation

- Increase and decrease

As shown in Table 6.4 (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 6.4 (a) Increase and decrease of the tool travel distance

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G45	Start point End point 	Start point End point 
G46	Start point End point 	Start point End point 
G47	Start point End point 	Start point End point 
G48	Start point End point 	Start point End point 



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

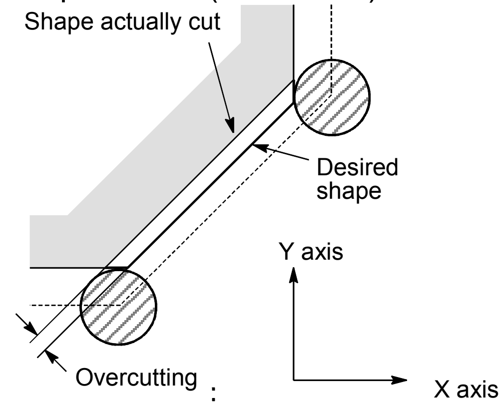
Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.
Tool offset values can be set within the following range:

D0 always indicates a tool offset value of zero.

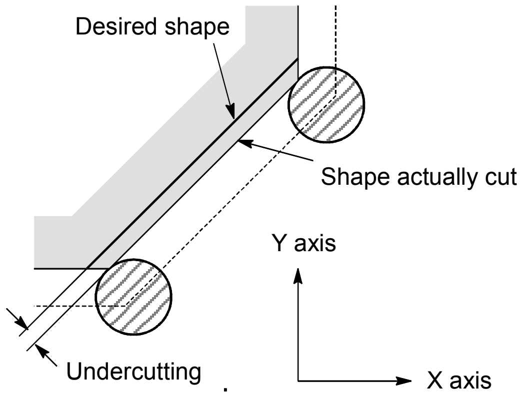
* If bit 2 (OFH) of parameter No. 5001 ="0", setting bit 5 (TPH) of parameter No. 5001 to "1" enables address H to be used as a code for specifying a tool position offset value.

⚠ CAUTION

1 When G45 to G48 is specified to n axes (n=1-4) simultaneously in a motion block, offset is applied to all n axes.
When the cutter is offset only for cutter radius or diameter in taper cutting, overcutting or undercutting occurs.
Therefore, use cutter compensation (G40 - G42) shown in II-6.4 or 6.6.




```
:
G01 X_ F_ ;
G47 X_ Y_ D_ ;
      Y_ ;
:
```

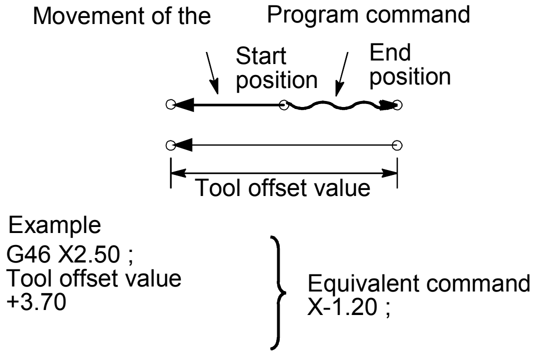



```
:
G01 G45 X_ F_ D_;
        X_ Y_ ;
G45 Y_ ;
:
```

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

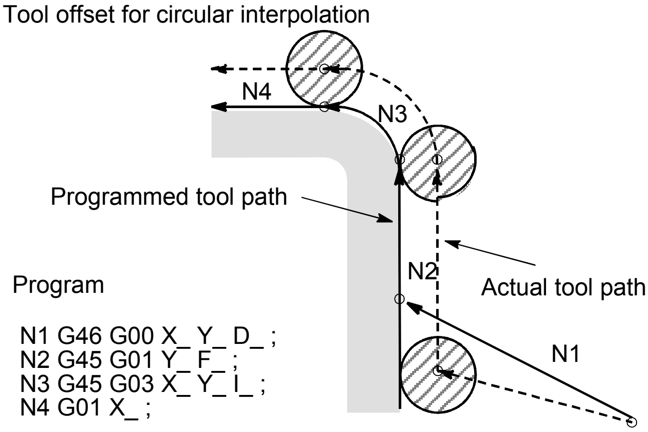
NOTE

1 When the specified direction is reversed by decrease as shown in the figure below, the tool moves in the opposite direction.



2 Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I, J and K 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

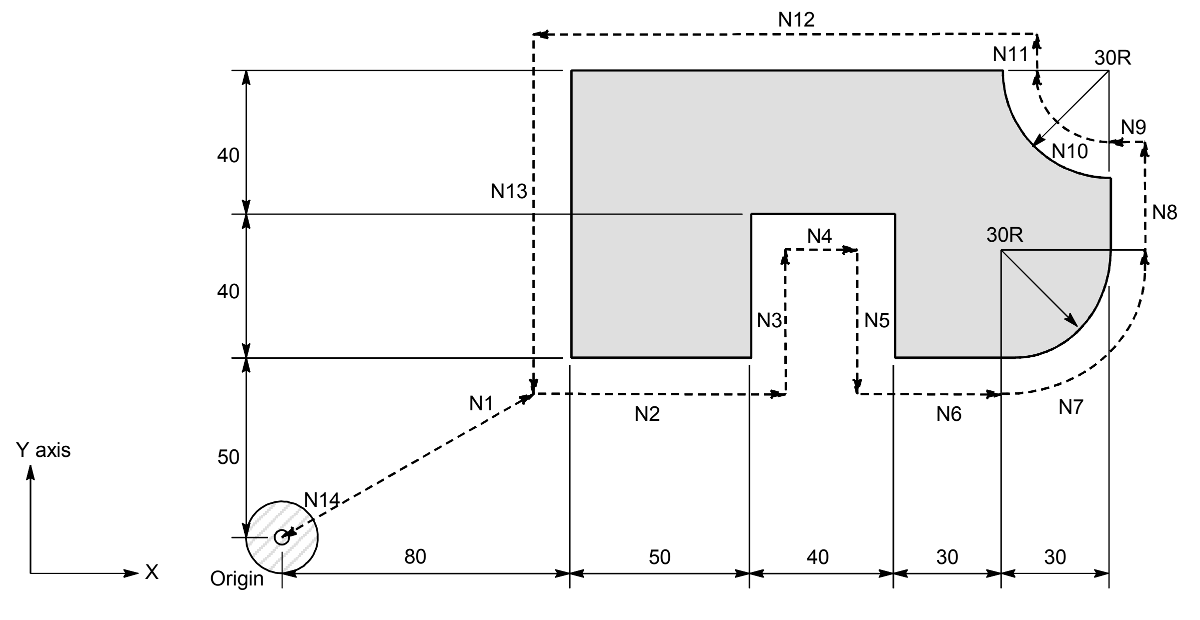


3 D code should be used in tool offset mode.

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 using tool offset



Tool diameter : 20ϕ
Offset No. : 01
Tool offset value : +10.0

Program

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

6.5 OVERVIEW OF CUTTER COMPENSATION (G40-G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 6.5 (a)).
To make an offset as large as the radius of the tool, CNC first creates an offset vector with a length equal to the radius of the tool (start-up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head positions to the center of the tool.
If a linear interpolation or circular interpolation command is specified after start-up, the tool path can be shifted by the length of the offset vector during machining.
To return the tool to the start point at the end of machining, cancel the cutter compensation mode.

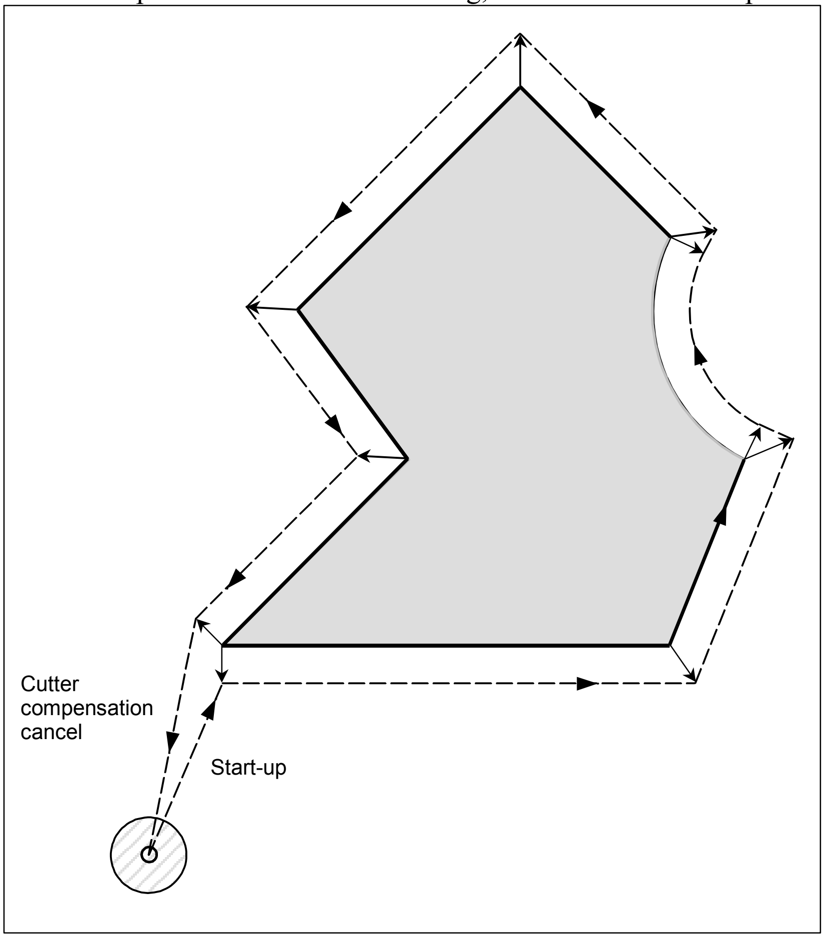


Fig. 6.5 (a) Outline of cutter compensation

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)

* Setting bit 2 (OFH) of parameter No. 5001 to "1" enables address H to be used as a code for specifying a cutter compensation amount. When bit 2 (OFH) of parameter No. 5001 = "1", if tool length compensation and cutter compensation are specified in the same block, the latter command takes precedence.

- 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, nonzero dimension word in the offset plane and command 0 in a non-D0 D code) is specified in the offset cancel mode, the CNC enters the offset mode. Moving the tool with this command is called start-up.
Specify positioning (G00) or linear interpolation (G01) for start-up.
If circular interpolation (G02, G03) is specified, alarm PS0034 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 command (G02 or G03) is 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.

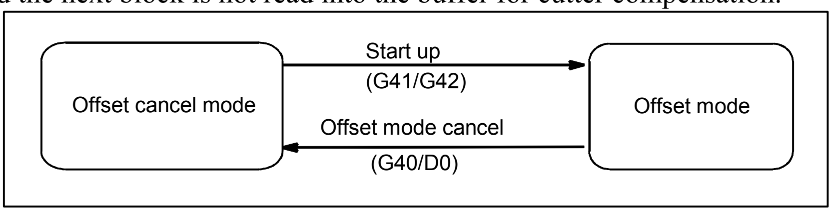


Fig. 6.5 (b) Changing the offset mode

- Change of the cutter compensation value

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

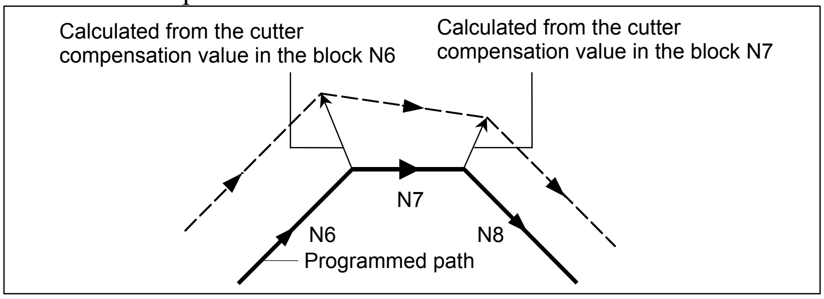


Fig. 6.5 (c) Changing the cutter compensation value

- 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. 6.5 (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.)

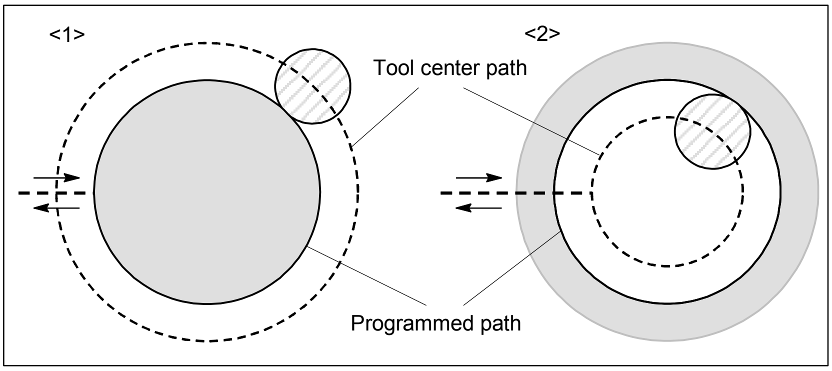


Fig. 6.5 (d) Tool center paths when positive and negative cutter compensation values are specified

- Cutter compensation value setting

Assign a cutter compensation values to the 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 1 (OFC) and 0 (OFA) parameter No. 5042.

Valid compensation range (metric input)

OFC	OFA	Range
0	1	±9999.99mm
0	0	±9999.999mm
1	0	±9999.9999mm

Valid compensation range (inch input)

OFC	OFA	Range
0	1	±999.999inch
0	0	±999.9999inch
1	0	±999.99999inch

The 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.

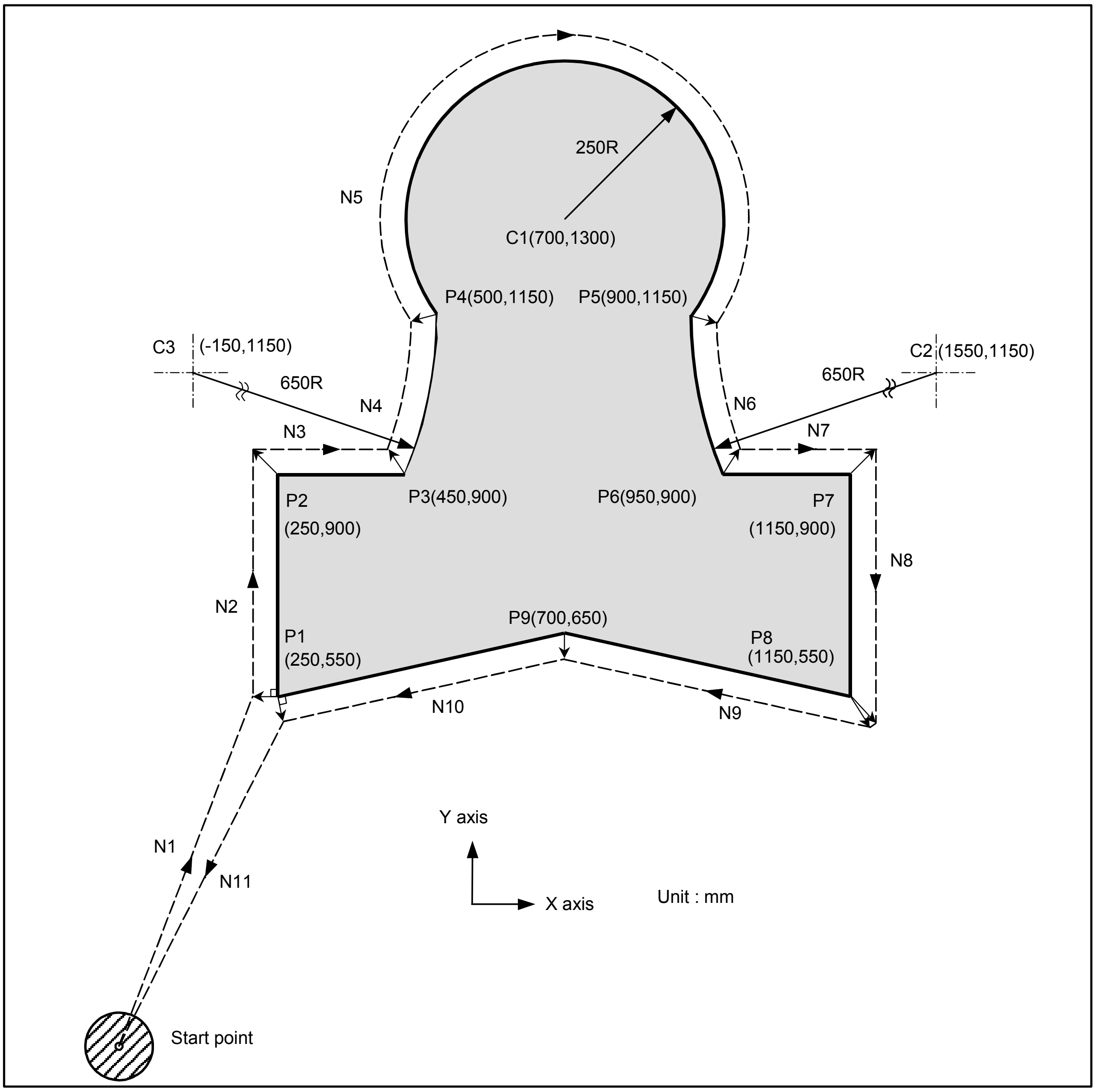
- Specifying a cutter compensation value

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



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

N1 G90 G17 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).

Notes

- Bit 2 (OFH) of parameter No.5001

If bit 2 (OFH) of parameter No. 5001 is set, cutter compensation takes precedence over tool length compensation. Concrete explanations follow:

If OFH = "0":
- Processing is carried out properly according to a selected modal state (G43, G44, or G49).

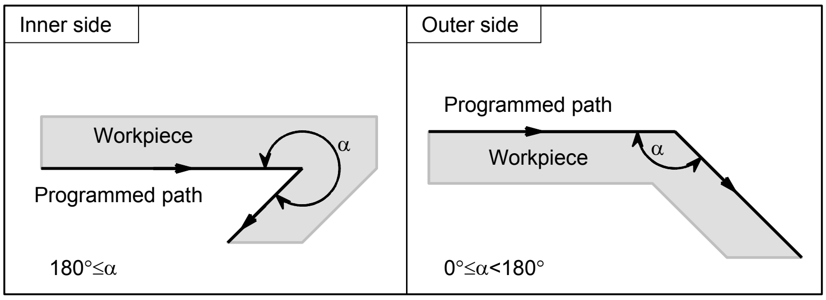
If OFH = "1":
- In a block where G40, G41, or G42 is specified, tool length compensation is disabled.
- In the G40 mode, processing is carried out properly according to a selected modal state (G43, G44, or G49).
- In the G41 and G42 modes, tool length compensation is enabled only in a block in which G43, G44, or G49 is specified. No compensation amount is updated only with the H code.
  G49 is enabled if G49 is specified in the same block as for G40, however.

6.6 DETAILS OF CUTTER COMPENSATION

6.6.1 Overview

- Inner side and outer side

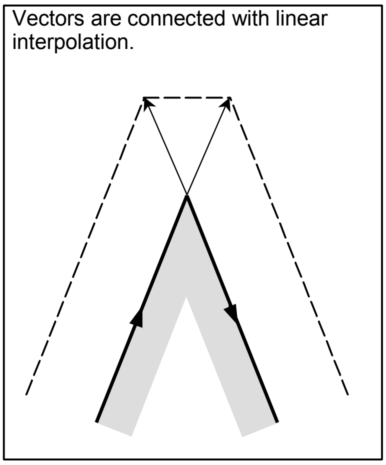
When an angle of intersection of the tool paths specified with move commands for two blocks on the workpiece side is over 180°, it is referred to as "inner side." When the angle is between 0° and 180°, it is referred to as "outer side."



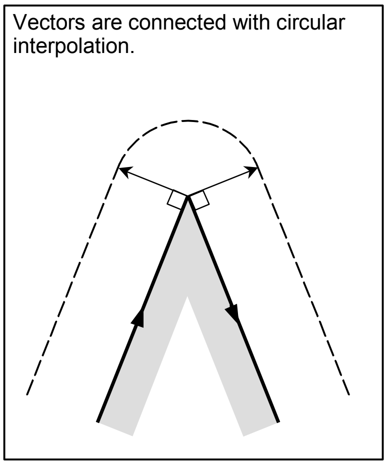
- Outer corner connection method

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 parameter CCC (No. 19607#2).

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



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



- 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

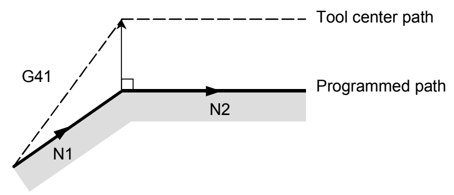
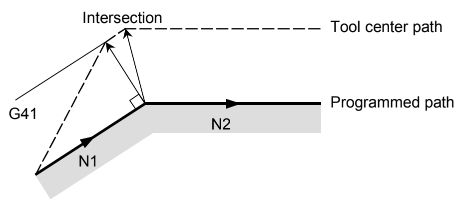
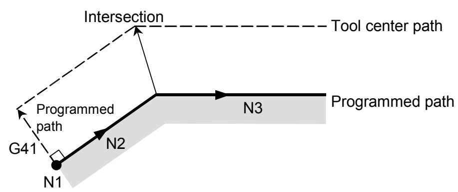
When a block which satisfies all the following conditions is executed in cancel mode, the CNC enters the cutter compensation mode. Control during this operation is called start-up.

<1> G41 or G42 is contained in the block, or has been specified to place the CNC in the cutter compensation mode.
<2> 0 < compensation number of cutter compensation ≤ maximum compensation number
<3> Positioning (G00) or linear interpolation (G01) mode
<4> A compensation plane axis command with a travel distance of 0 (except start-up type C) is specified.

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

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

Table 6.6.1 (a) Start-up/cancel operation

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector is output, which is vertical to the block subsequent to the start-up block and the block preceding the cancel block. 
0	1	Type B	A compensation vector is output, which is vertical to the start-up block and the cancel block. An intersection vector is also output. 
1	0 1	Type C	When the start-up block and the cancel block are blocks without tool movement, the tool moves by the cutter compensation value in the direction vertical to the block subsequent to the start-up block and the block preceding the cancel block.  For a block with tool movement, the tool follows the SUP setting: If it is 0, type A is assumed and if 1, type B is assumed.

- Reading input commands in cutter compensation mode

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

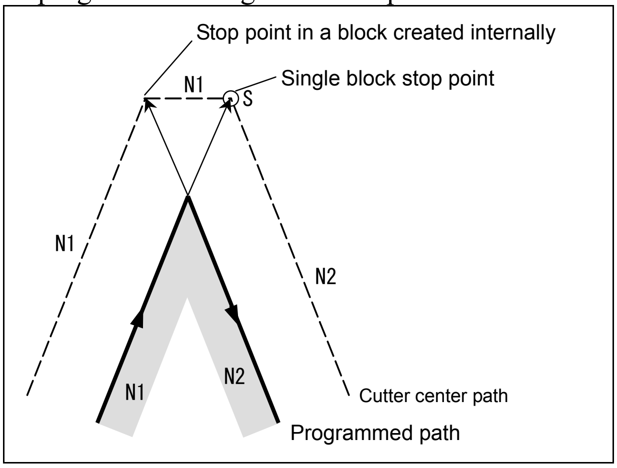
As the setting of parameter (No. 19625), that is, the number of blocks to read, increases, it is possible to predict overcutting (interference) for up to more subsequent commands. Increases in blocks to read and analyze, however, cause reading and analysis to take more time.

- Ending (canceling) cutter compensation

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

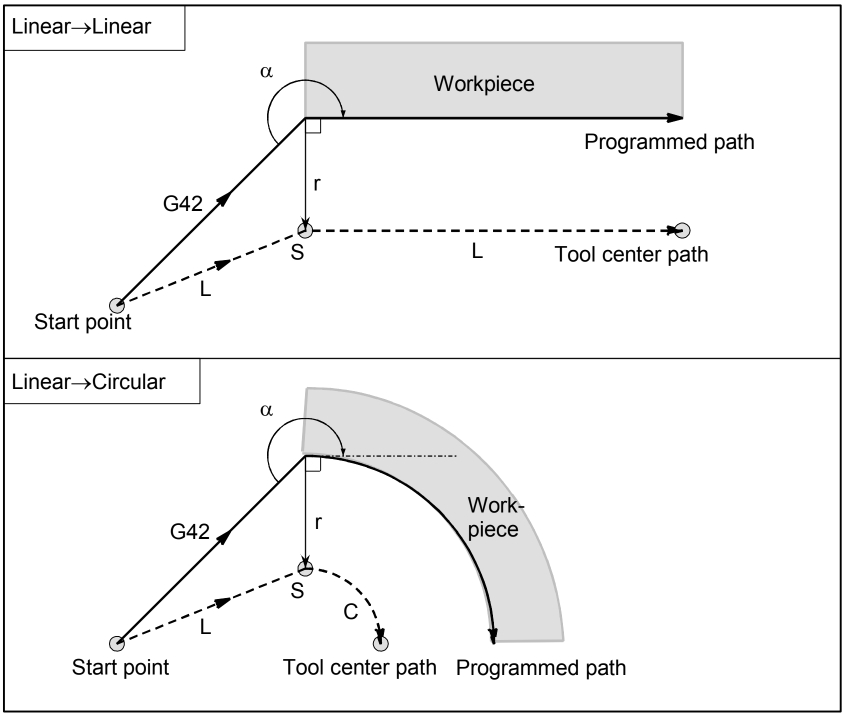
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves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

6.6.2 Tool Movement in Start-up

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

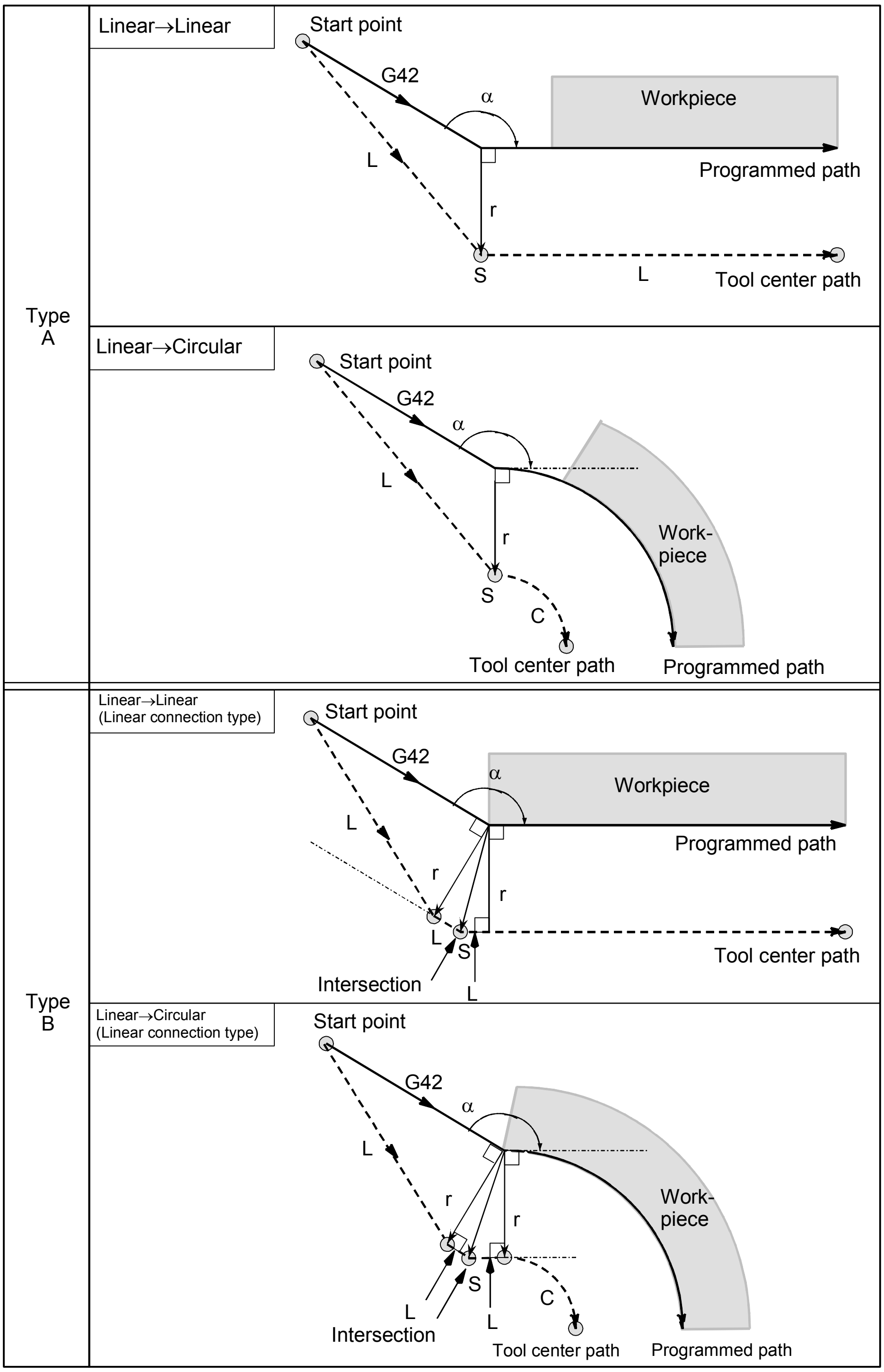
Explanation

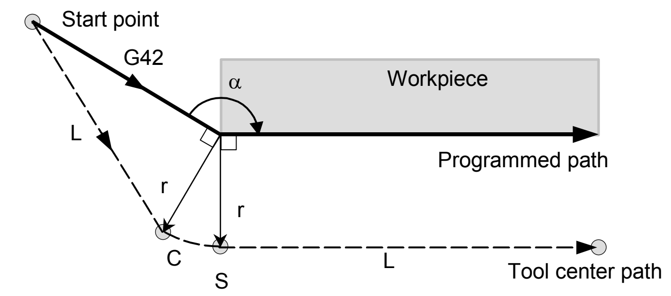
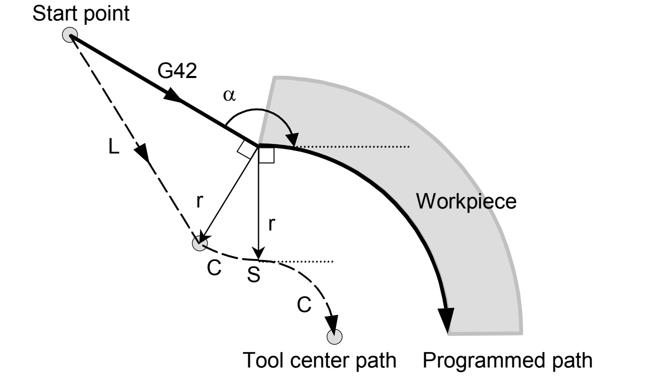
- Tool movement around an inner side of a corner (180°≤ α)



- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an obtuse angle (90°≤ α<180°)

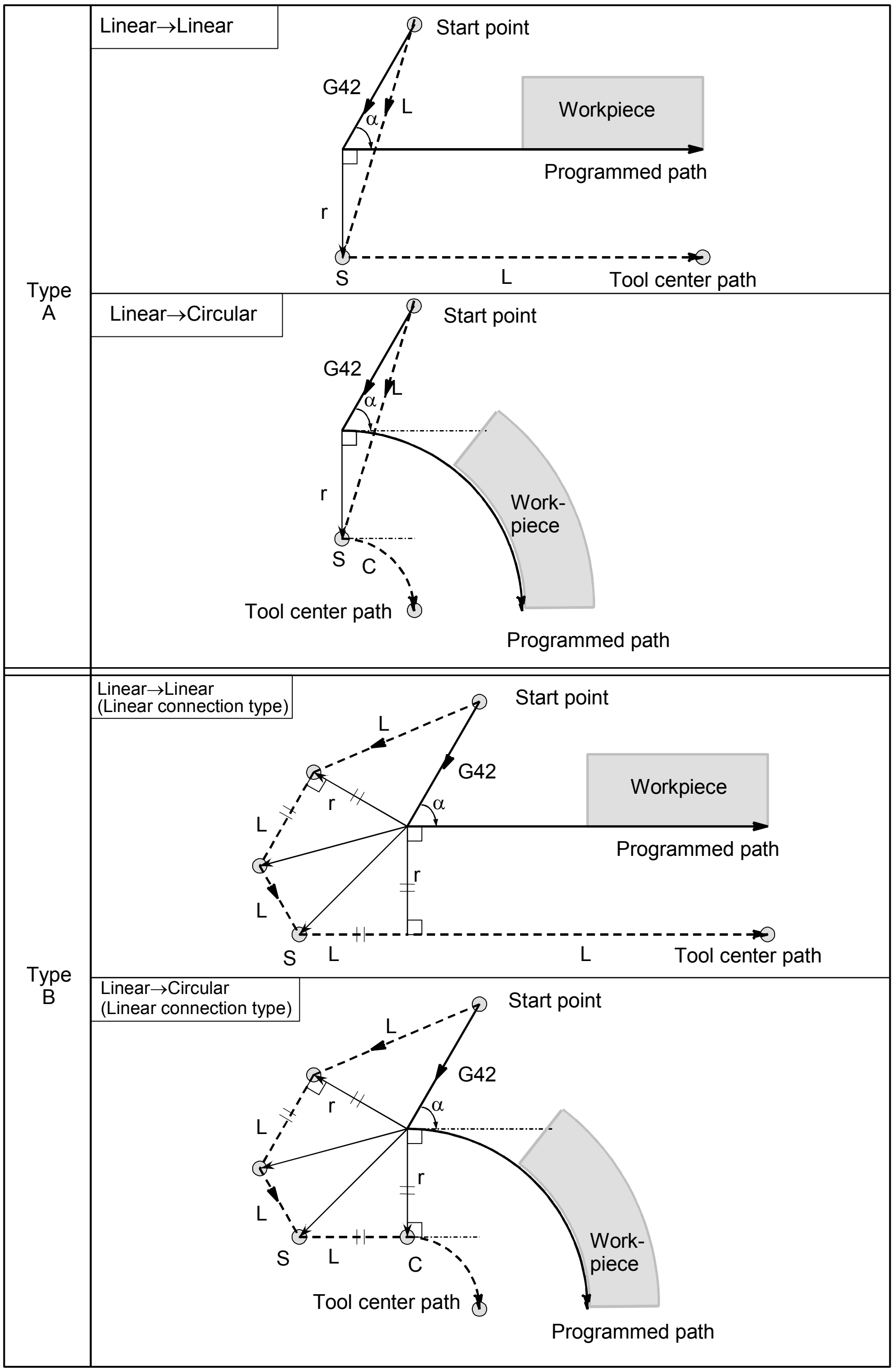
Tool path in start-up has two types A and B, and they are selected by parameter SUP (No.5003#0).

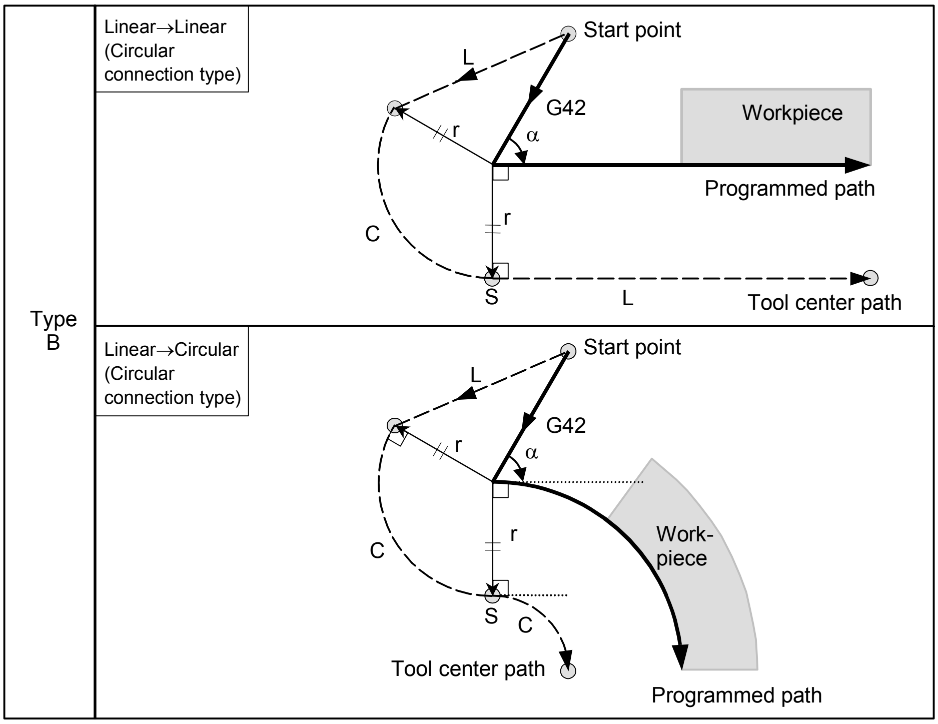


Type B	Linear→Linear (Circular connection type)	
	Linear→Circular (Circular connection type)	

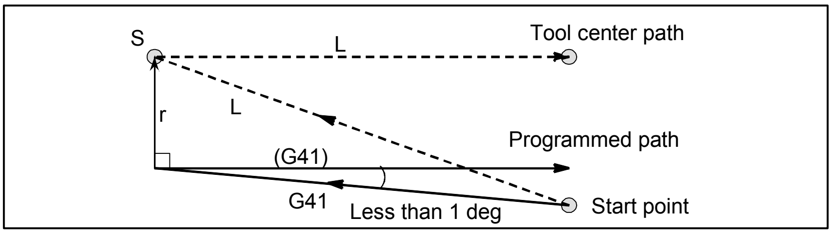
- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle (α<90°)

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





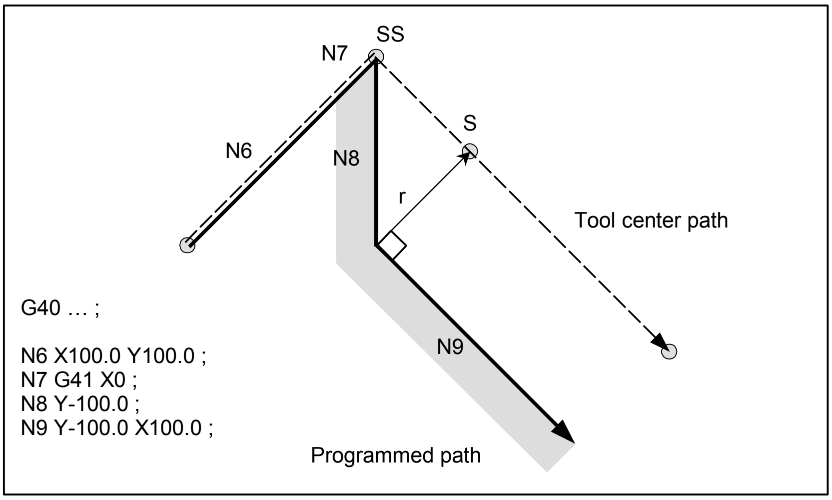
- Tool movement around the outside linear → linear at an acute angle less than 1 degree (α<1°)



- A block without tool movement specified at start-up

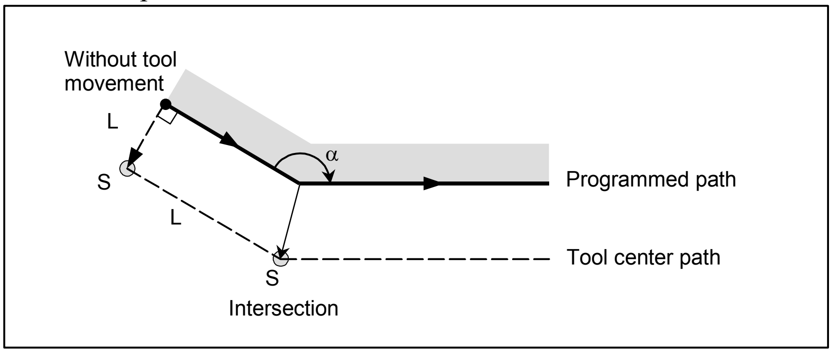
For type A and B

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



For type C

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



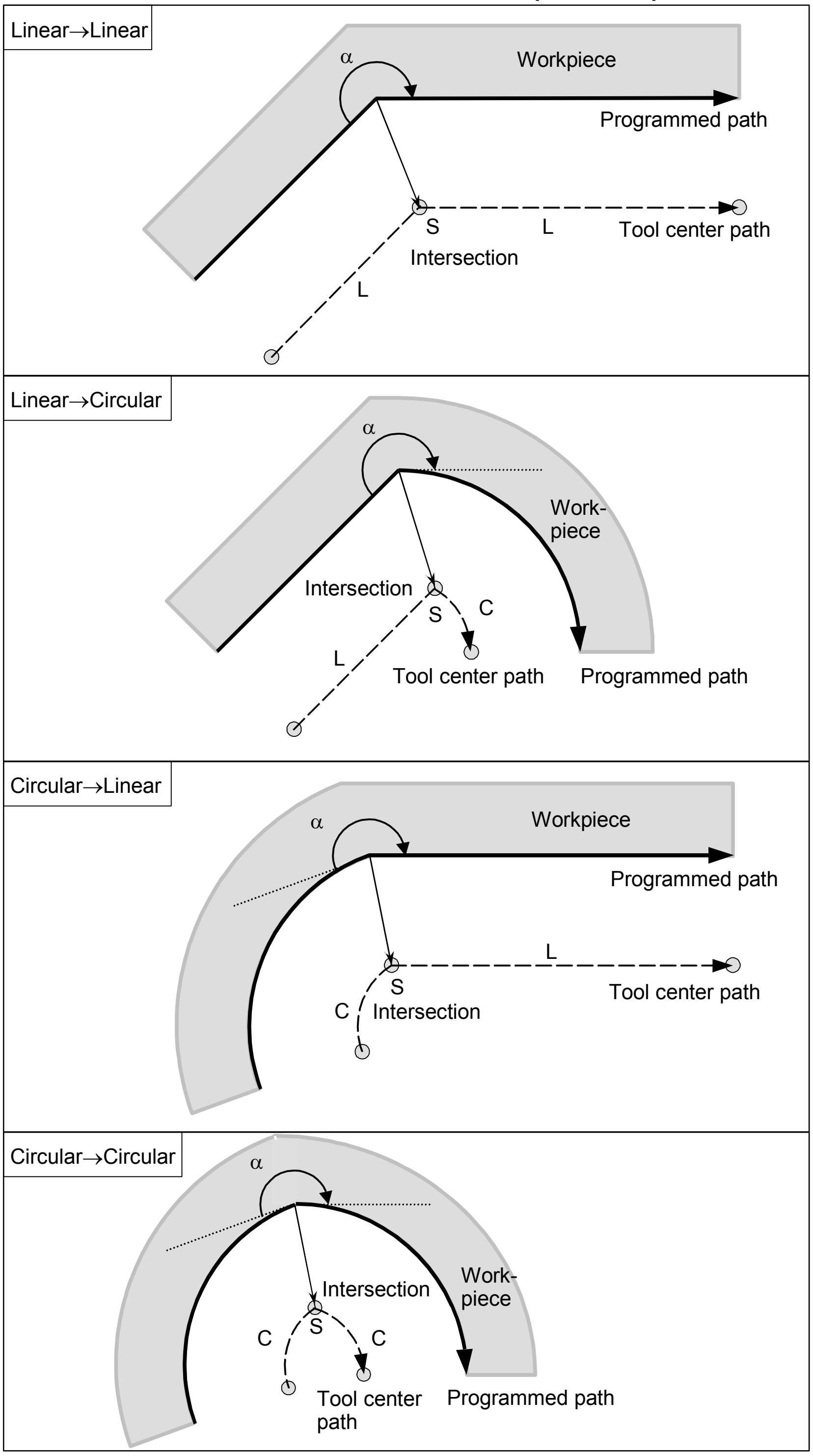
6.6.3 Tool Movement in Offset Mode

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.

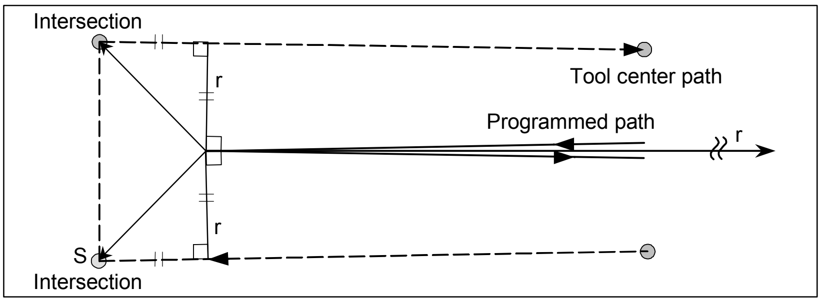
> **NOTE**
> The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the item, “interference check.”

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

- Tool movement around the inside of a corner (180°≤ α)

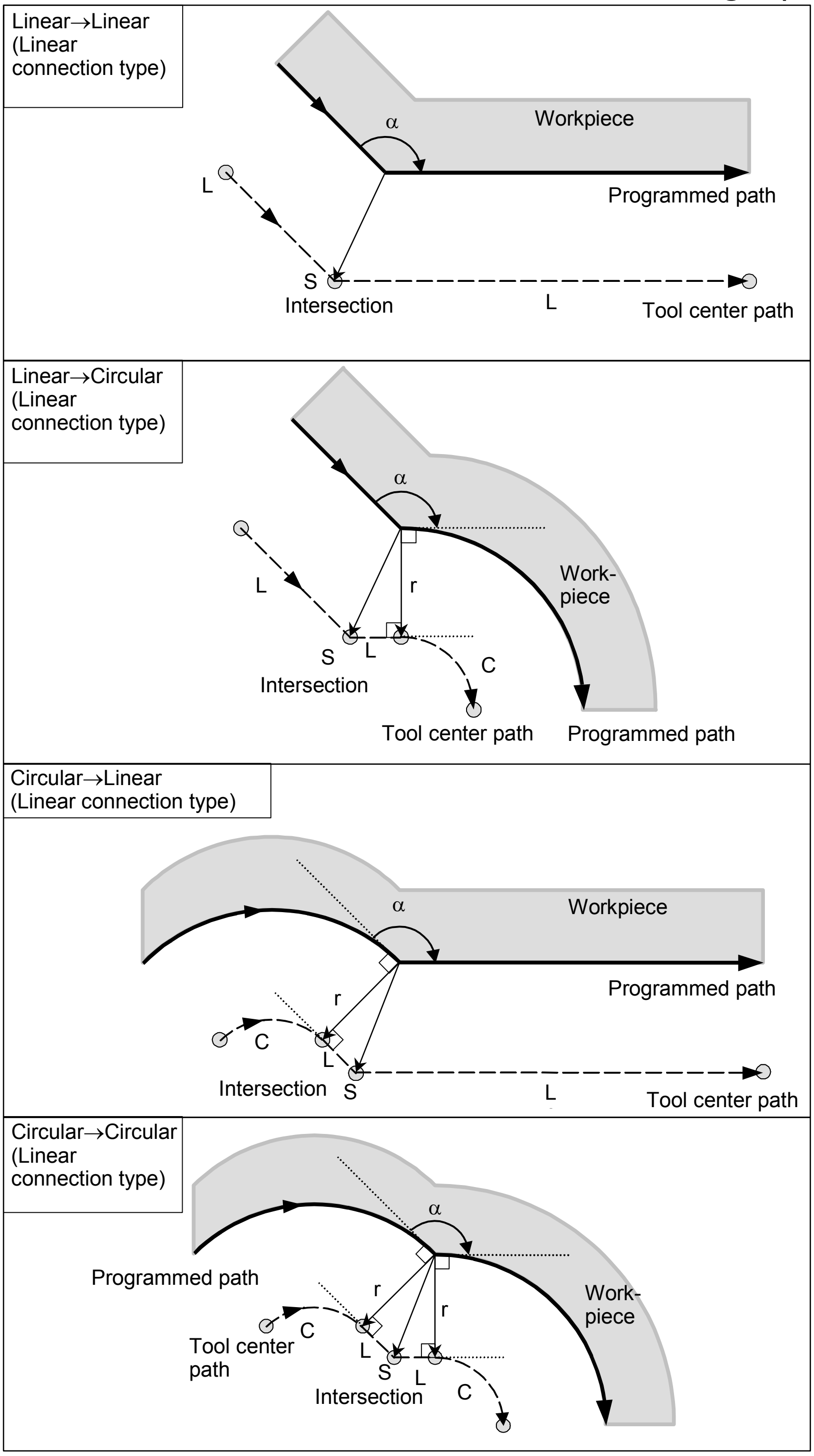


- Tool movement around the inside (α<1°) with an abnormally long vector, linear $\rightarrow$ linear

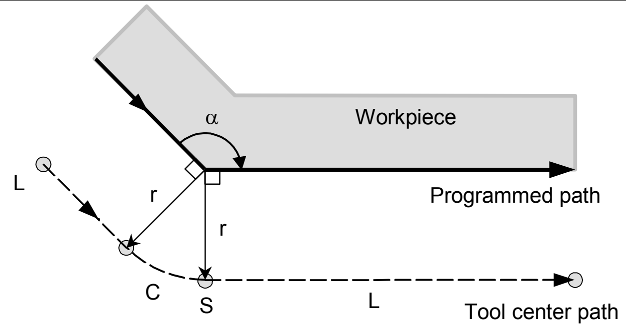


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

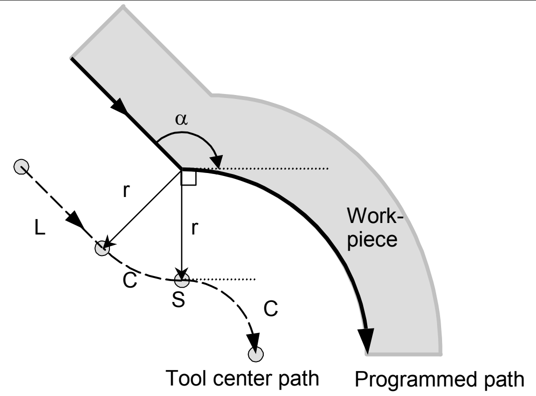
- Tool movement around the outside corner at an obtuse angle (90°≤α<180°)



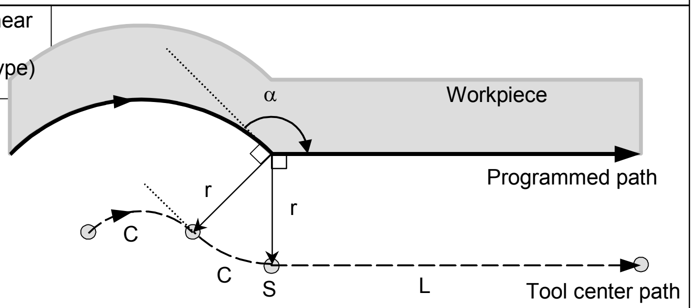
Linear→Linear (Circular connection type)



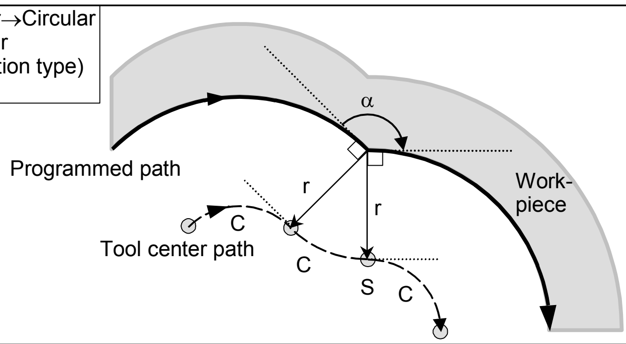
Linear→Circular (Circular connection type)



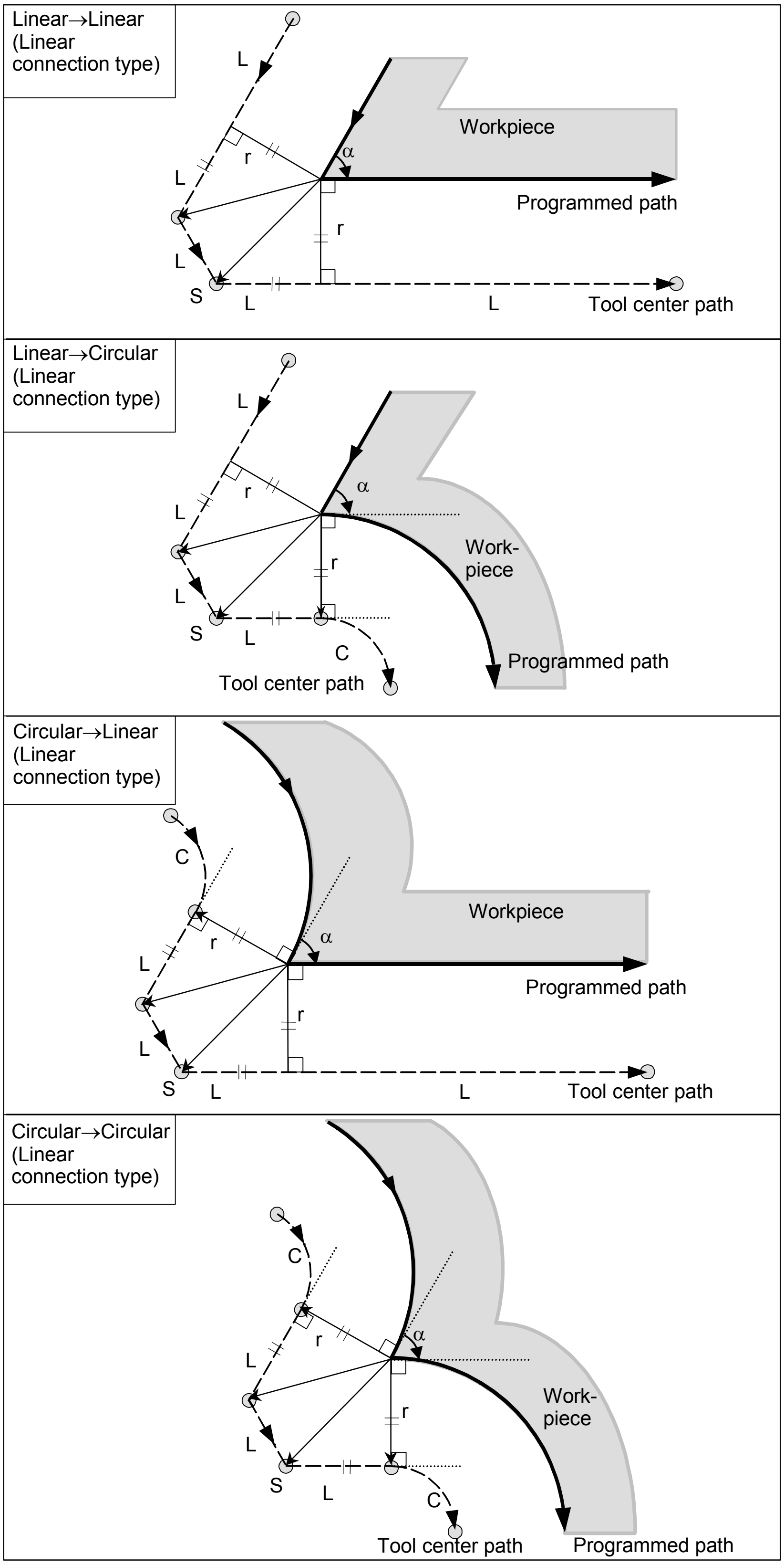
Circular→Linear (Circular connection type)



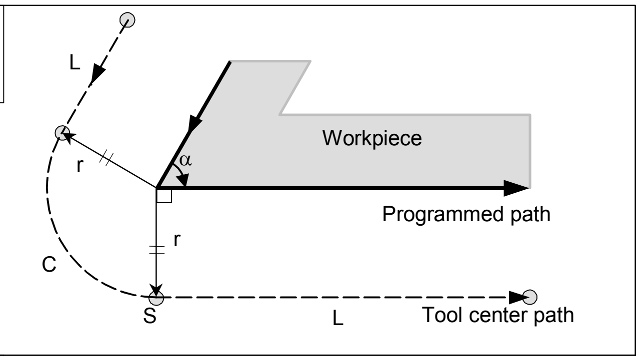
Circular→Circular (Circular connection type)



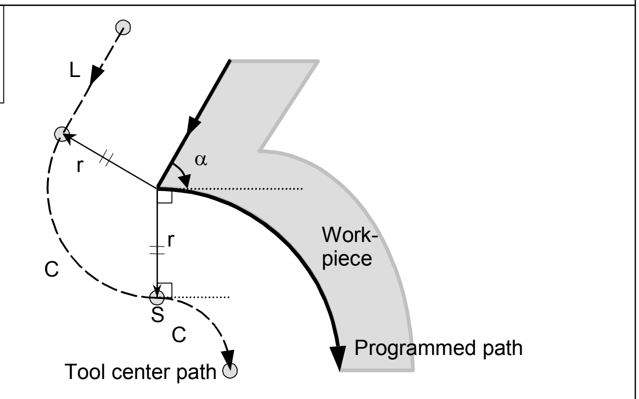
- Tool movement around the outside corner at an acute angle (α<90°)



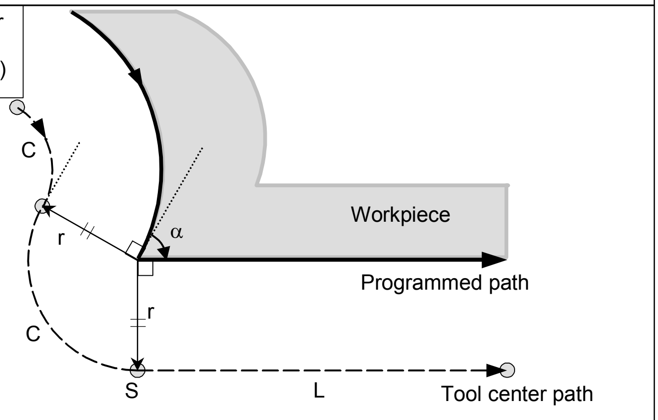
Linear→Linear
(Circular
connection type)



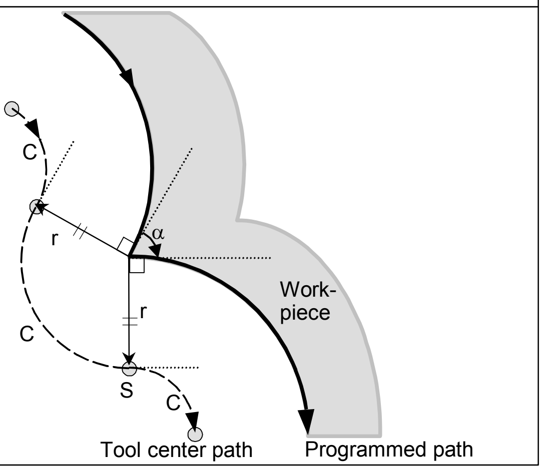
Linear→Circular
(Circular
connection type)



Circular→Linear
(Circular
connection type)



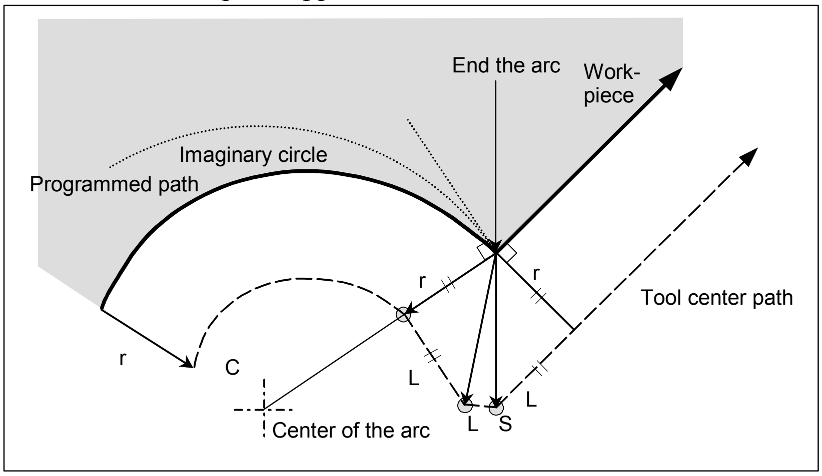
Circular→Circular
(Circular
connection type)



- When it is exceptional

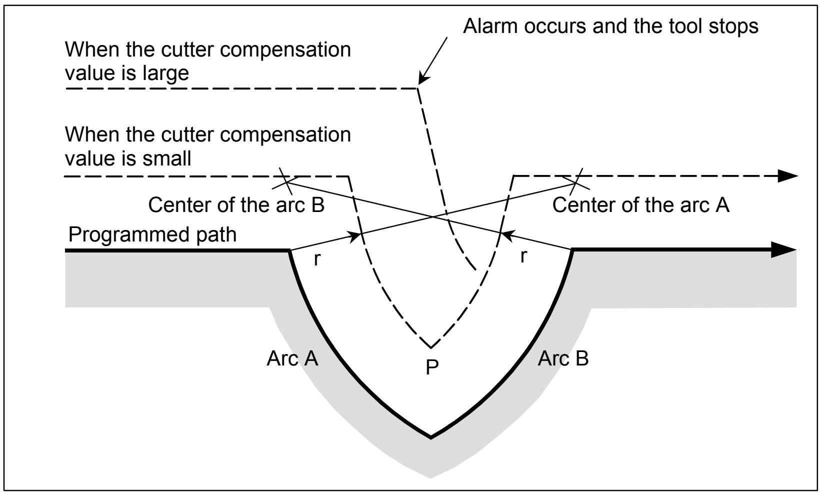
End point for the arc is not on the arc

If the end of a line leading to an arc is not on the arc as illustrated below, 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.



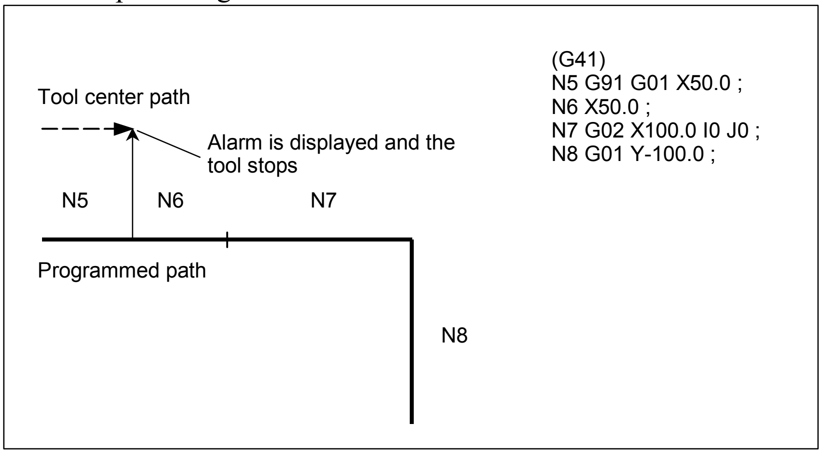
There is no inner intersection

If the cutter compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for cutter compensation. When this is predicted, PS0033 occurs at the end of the previous block and the tool is stopped.
In the Example shown below, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for cutter 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

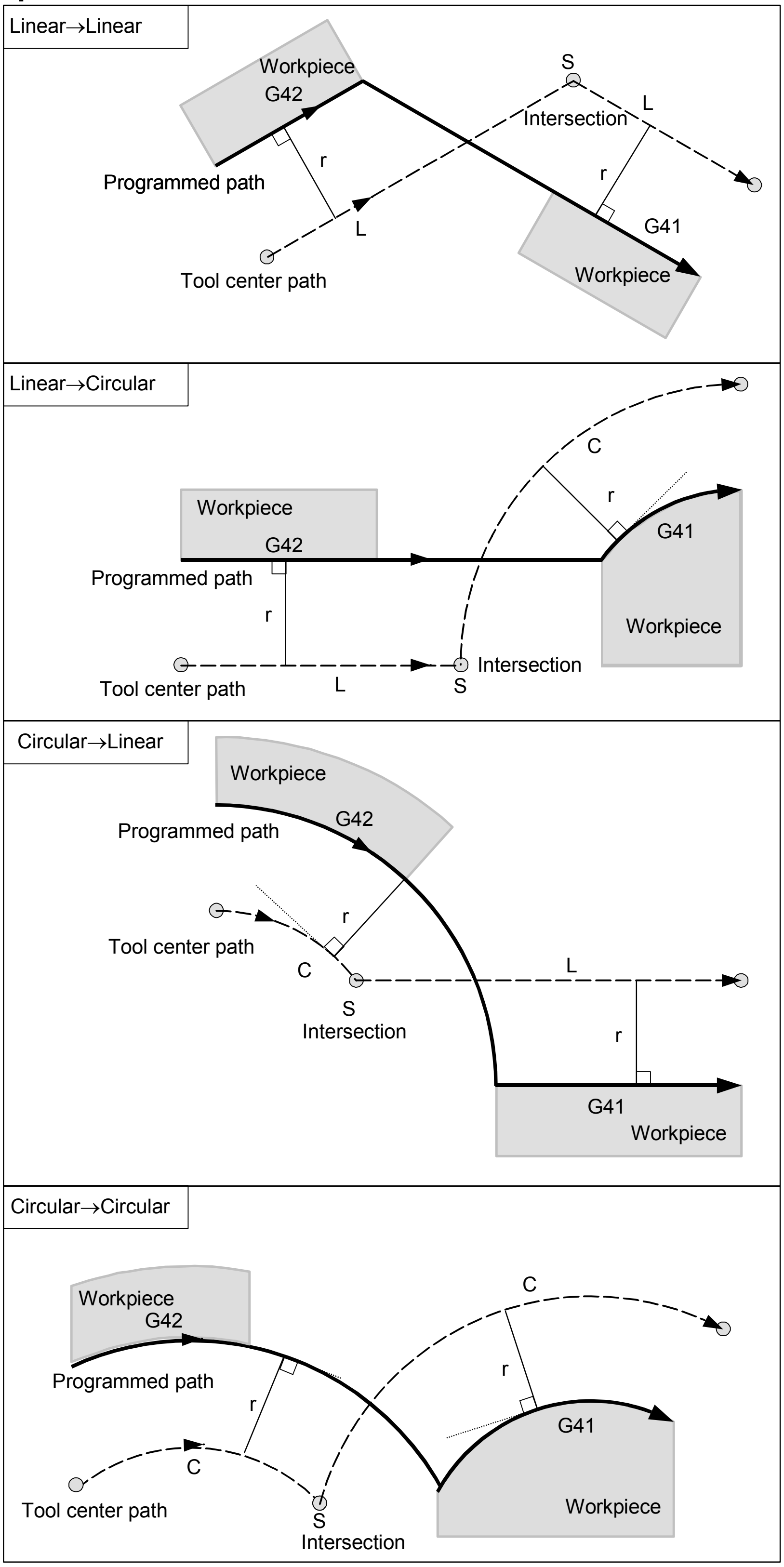
The offset direction is decided by G codes (G41 and G42) for cutter compensation and the sign of the compensation value as follows.

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

The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block.

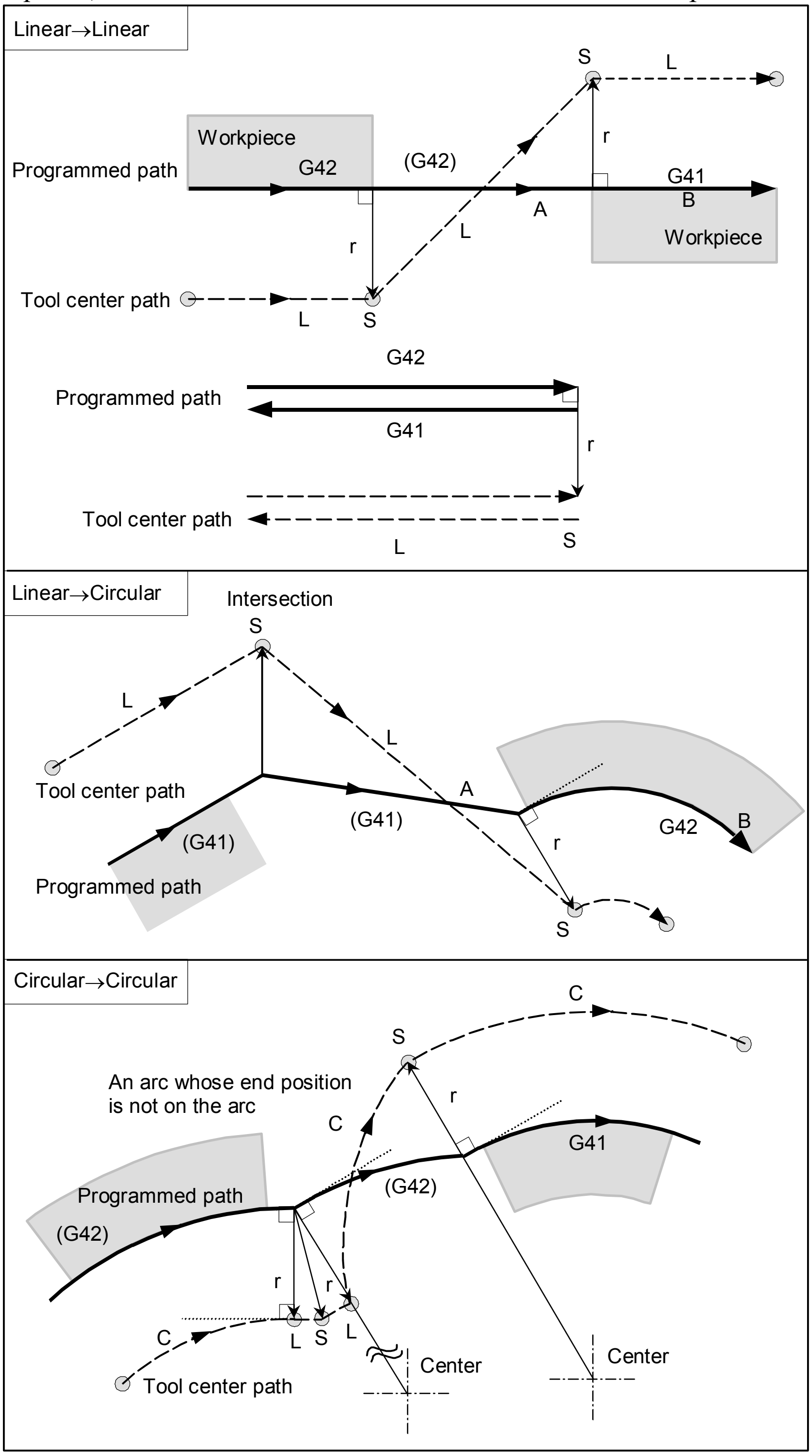
However, the change is not available in the start-up block and the block following it.

- Tool center path with an intersection



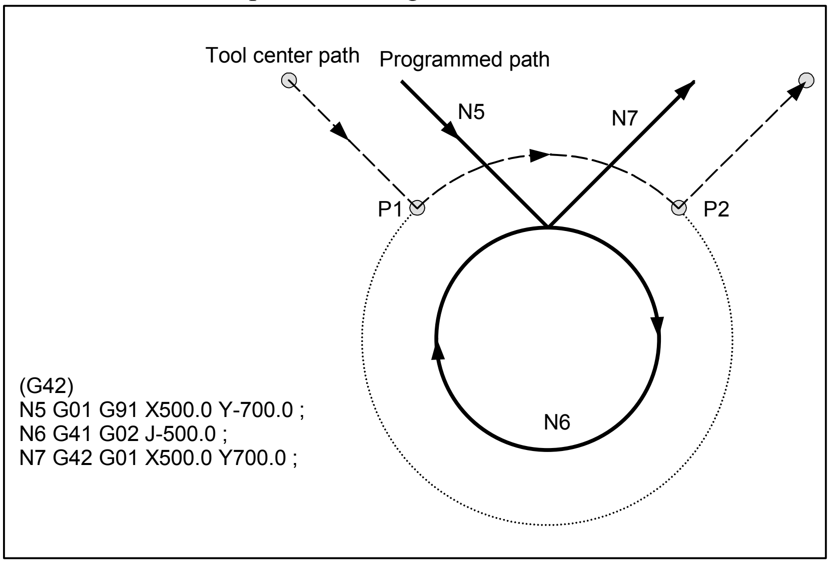
- Tool center path without an intersection

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



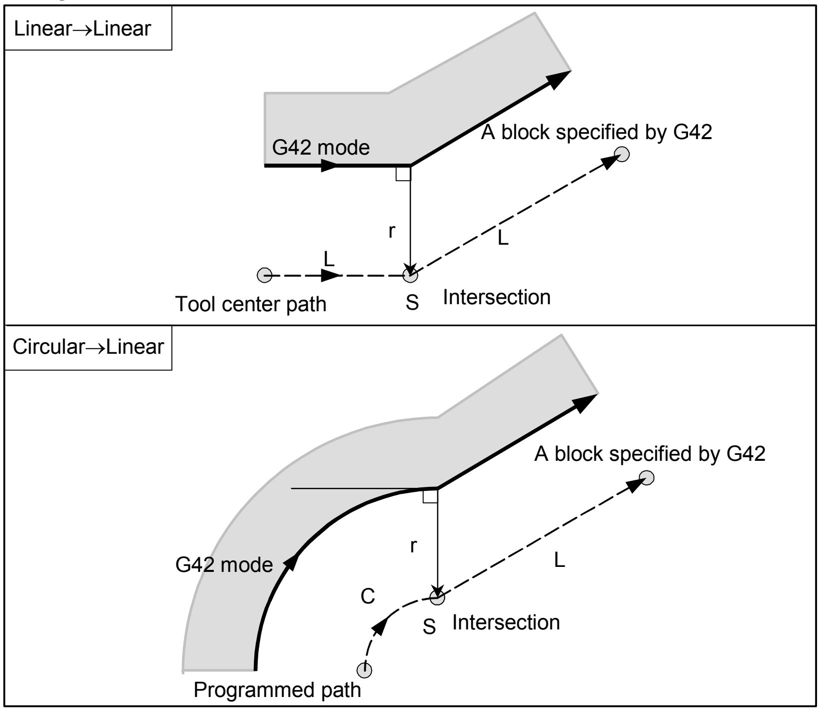
- The length of tool center path larger than the circumference of a circle

Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I, J, and K this situation can occur.
In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from P_1 to P_2 as shown. Depending on the circumstances, an alarm may be displayed due to the item, "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.



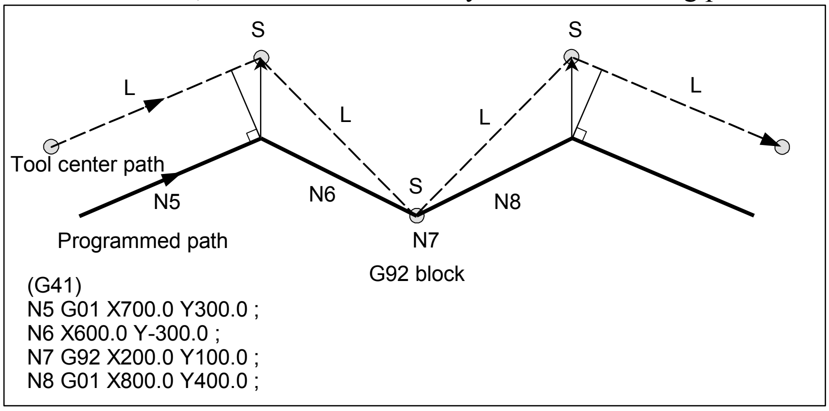
- Cutter compensation G code in the offset mode

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



- Command canceling the offset vector temporarily

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



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

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

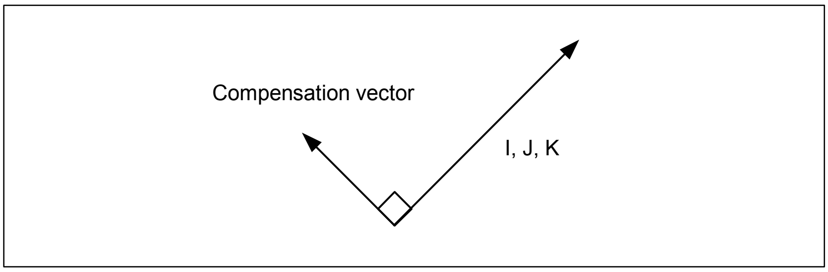
At the start of cutter compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This makes it possible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

IJ type vector (XY plane)

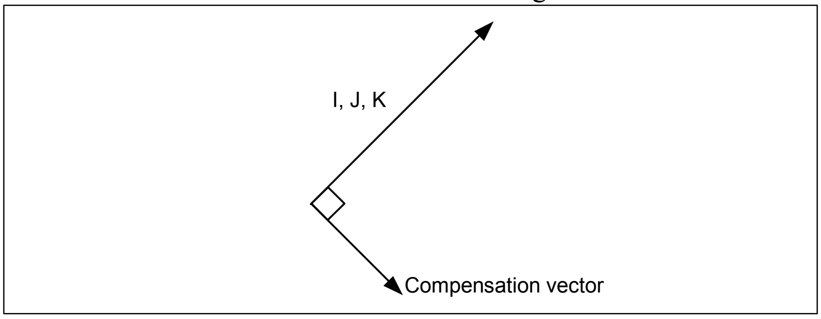
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) As shown in the figure below, it is assumed that the compensation vector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

Offset vector direction

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



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

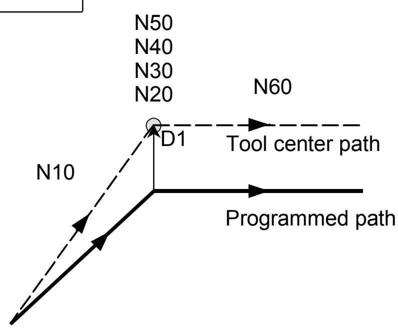


Example

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

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

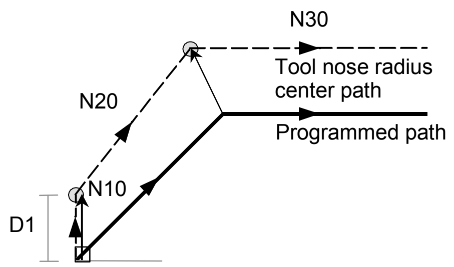
Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.



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

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

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

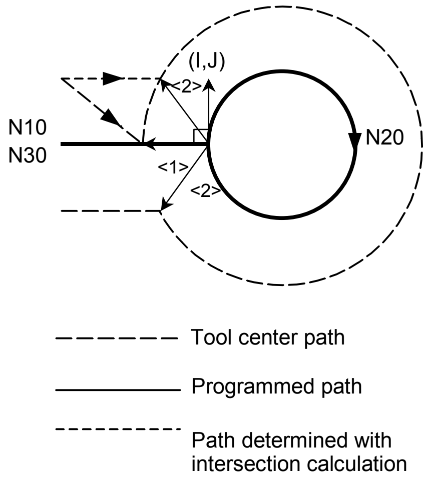


If I and J are specified at the start of compensation (with tool movement)

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

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

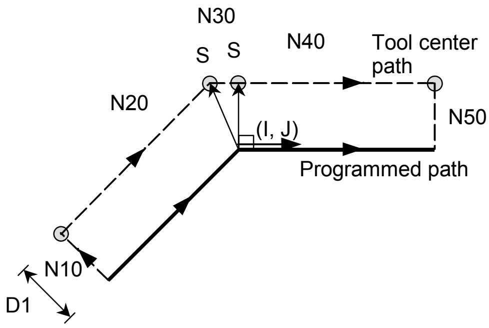
<1> IJ type vector
<2> Vector determined with intersection calculation



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

Start-up/cancel type C

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

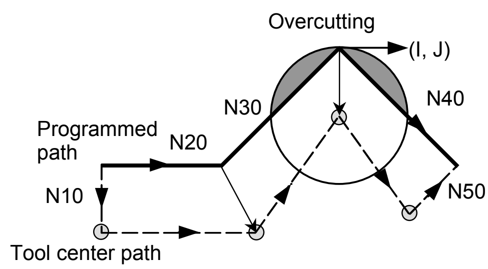


Limitation

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

Start-up/cancel
Type C

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



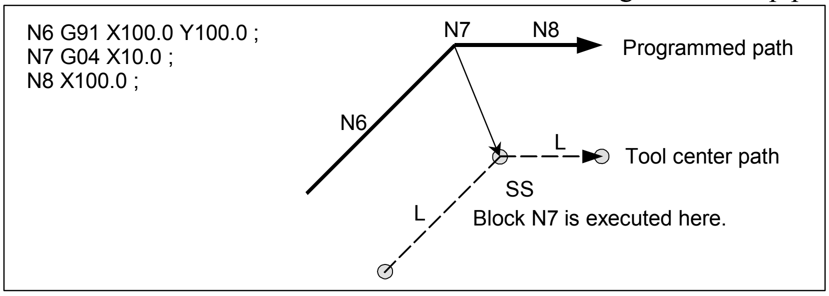
- A block without tool movement

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

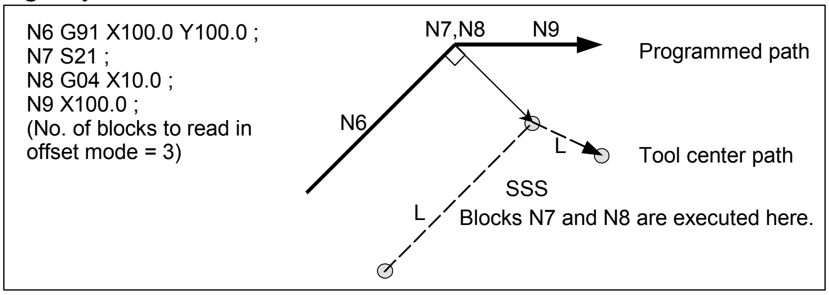
M05 ;	:	M code output
S21 ;	:	S code output
G04 X10.0 ;	:	Dwell
G22 X100000 ;	:	Machining area setting
G10 L11 P01 R10.0 ;	:	Cutter compensation value setting/changing
(G17) Z200.0 ;	:	Move command not included in the offset plane.
G90 ;, O10 ;, N20 ;	:	G, O, and N codes only
G91 X0 ;	:	Move distance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

- A block without tool movement specified in offset mode

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

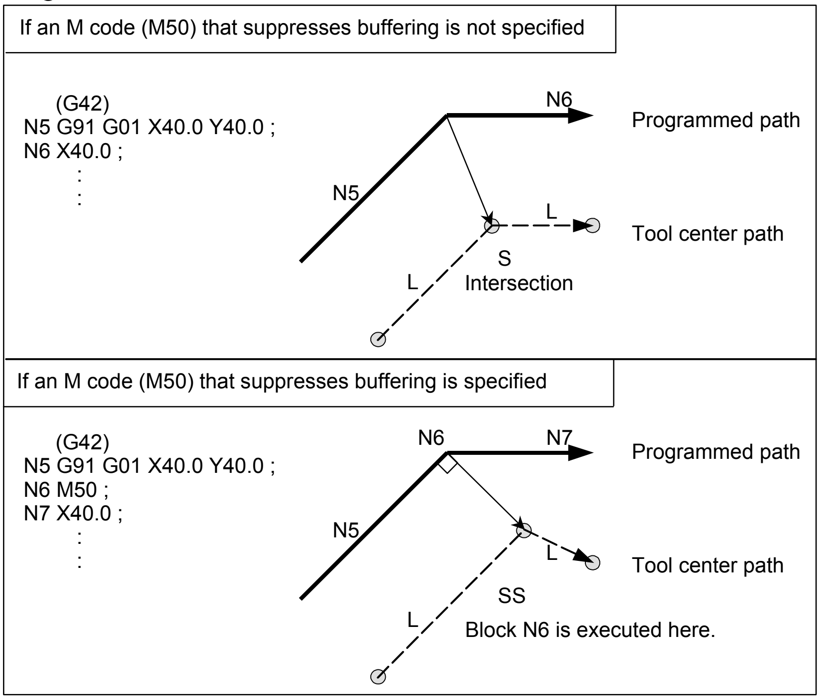


In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter (No. 19625)). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier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



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

If an M/G code that suppresses buffering is specified in offset mode, it is no longer possible to read and analyze subsequent blocks regardless of the number of blocks to read in offset mode, which is determined by parameter (No. 19625). Then, intersection calculation and a interference check, described later, are no longer possible. If this occurs, overcutting may occur because a vertical vector is output in the immediately preceding block.

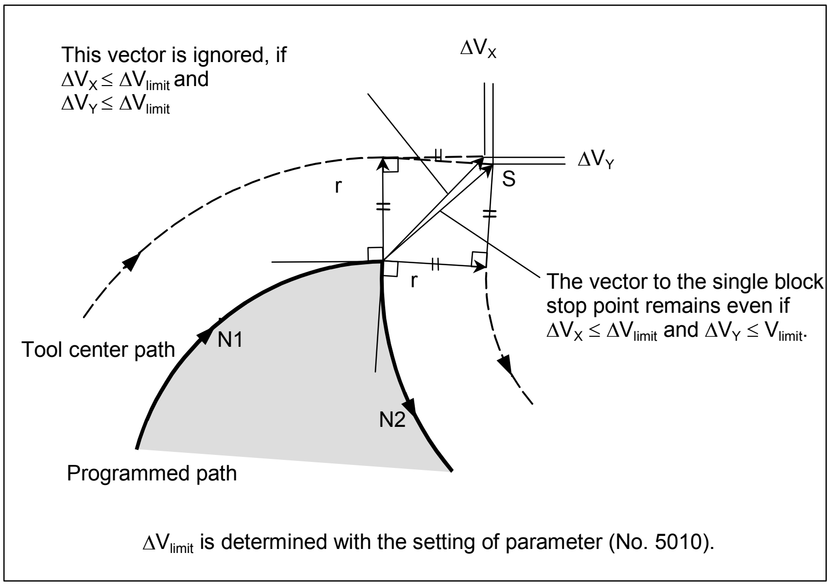


- Workpiece coordinate system or local coordinate system command in the offset mode

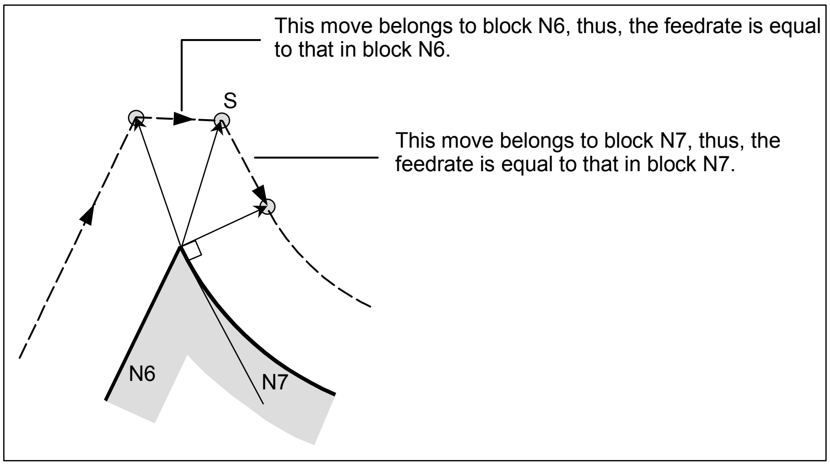
If the local coordinate system (G52) or workpiece coordinate system (G92) is specified in the cutter compensation (G41 or G42) mode, G52 or G92 is assumed to be a buffering masked G code. The subsequent blocks are not executed until the G52 or G92 block is executed.

- Corner movement

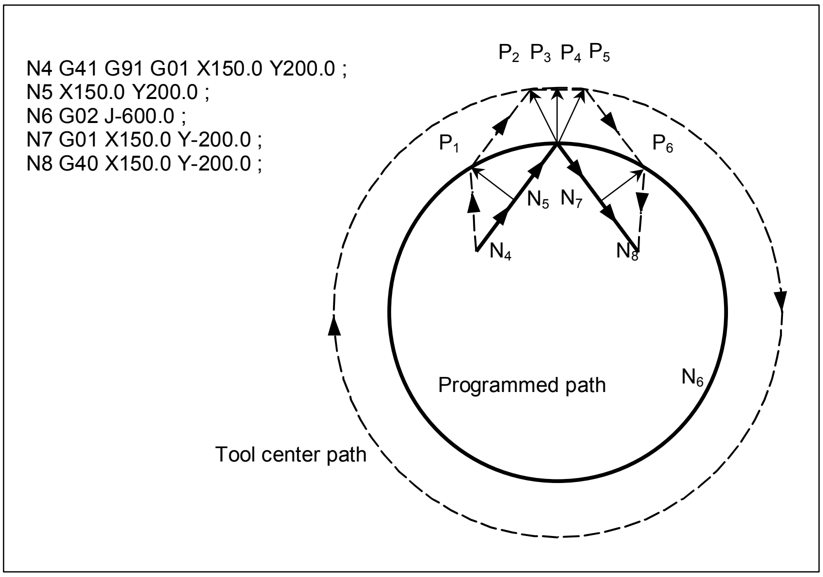
When two or more offset vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.
If these vectors almost coincide with each other (the distance of corner movement between the vectors is judged short due to the setting of parameter (No. 5010)), corner movement is not performed. In this case, the vector to the single block stop point takes precedence and remains, while other vectors are ignored. This makes it possible to ignore the very small movements arising from performing cutter compensation, thereby preventing velocity changes due to interruption of buffering.



If the vectors are not judged to almost coincide (therefore, are not erased), movement to turn around the corner is performed. The corner movement that precedes the single block stop point belongs to the previous block, while the corner movement that succeeds the single block stop point belongs to the latter block.



However, if the path of the next block is semicircular or more, the above function is not performed.
The reason for this is as follows:



If the vector is not ignored, the tool path is as follows:
$P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow$ (Circle) $\rightarrow P_4 \rightarrow P_5 \rightarrow P_6$
But if the distance between P_2 and P_3 is negligible, the point P_3 is ignored. Therefore, the tool path is as follows:
$P_2 \rightarrow P_4$
Namely, circle cutting by the block N6 is ignored.

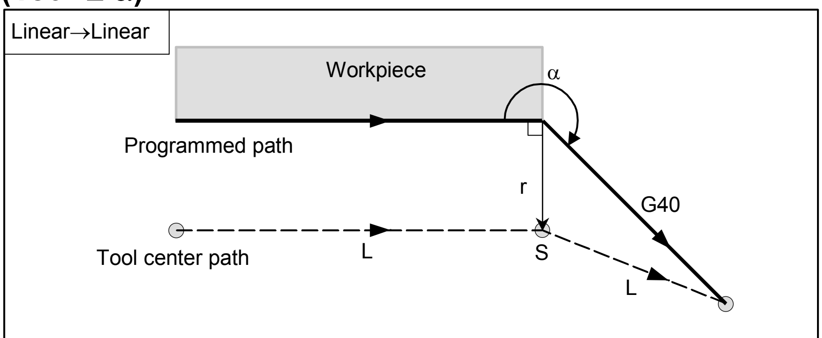
- Interruption of manual operation

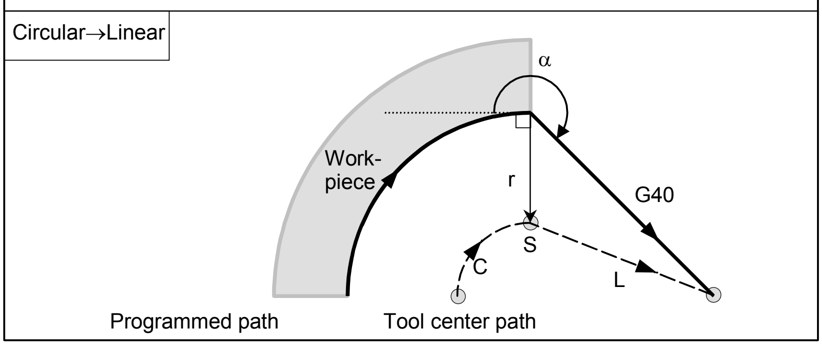
For manual operation during the offset mode, see "Manual Absolute ON and OFF."

6.6.4 Tool Movement in Offset Mode Cancel

Explanation

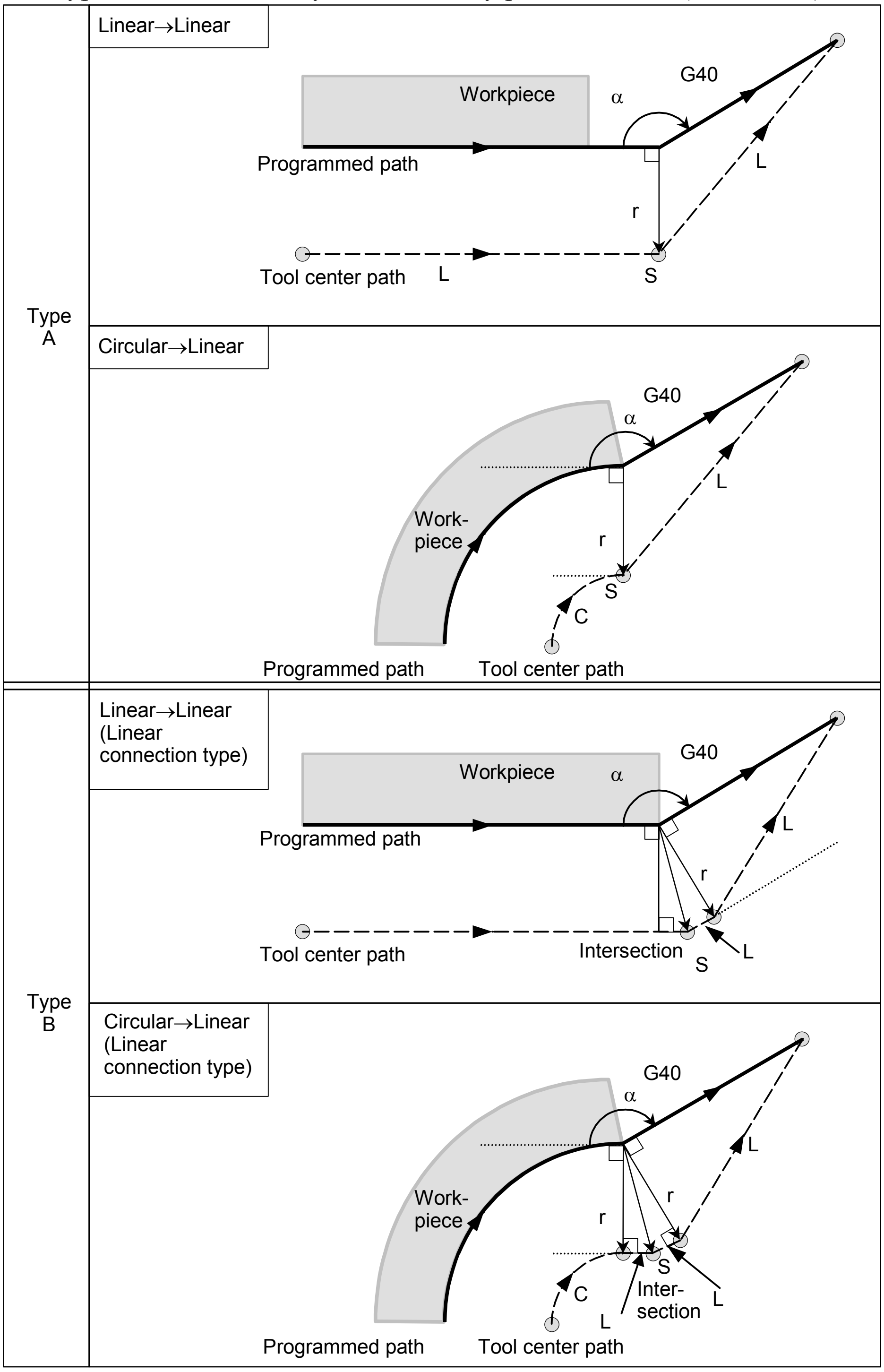
- If the cancel block is a block with tool movement, and the tool moves around the inside (180° ≤ α)

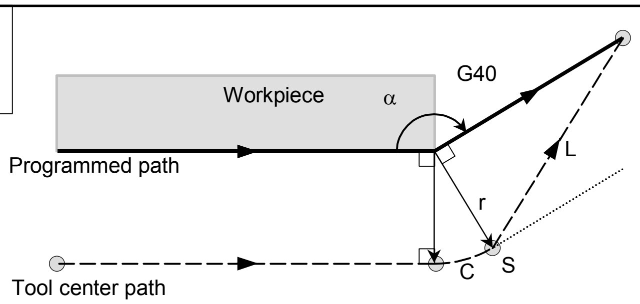
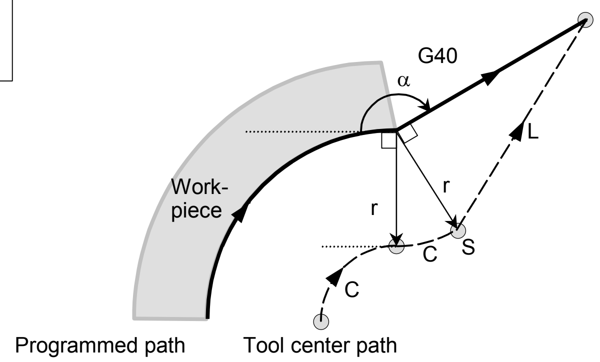




- If the cancel block is a block with tool movement, and the tool moves around the outside at an obtuse angle (90° ≤ α < 180°)

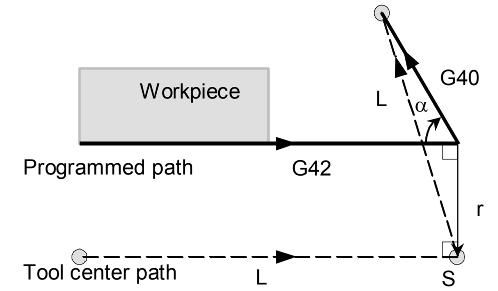
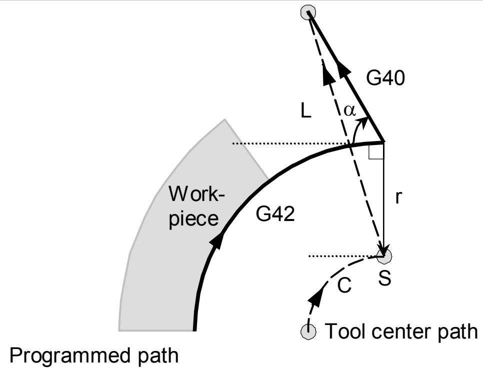
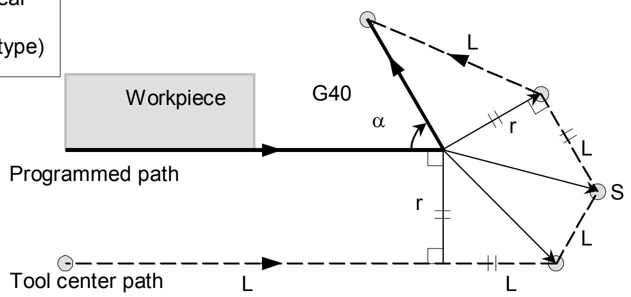
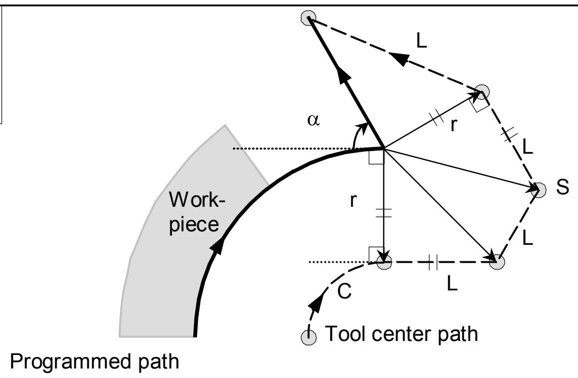
Tool path has two types A and B, and they are selected by parameter SUP (No.5003#0).

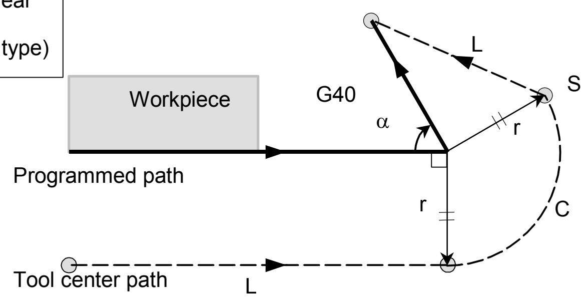
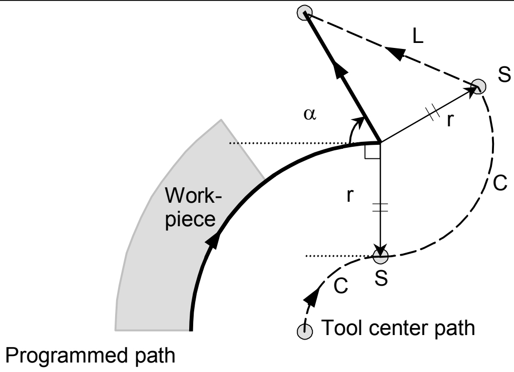


<table>
<tr><td rowspan="2">Type B</td><td>Linear→Linear (Circular connection type)</td><td>
</td></tr>
<tr><td>Circular→Linear (Circular connection type)</td><td>
</td></tr>
</table>

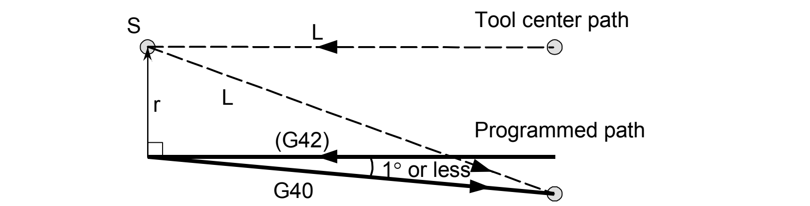
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle (α<90°)

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

Type A	Linear→Linear	
	Circular→Linear	
Type B	Linear→Linear (Linear connection type)	
	Circular→Linear (Linear connection type)	

Type B	Linear→Linear (Circular connection type)	
	Circular→Linear (Circular connection type)	

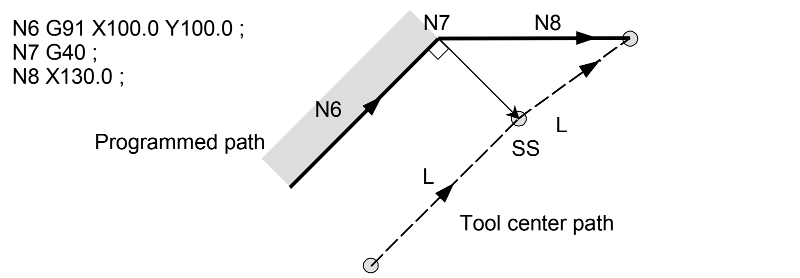
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle of 1 degree or less in a linear → linear manner (α≤1°)



- A block without tool movement specified together with offset cancel

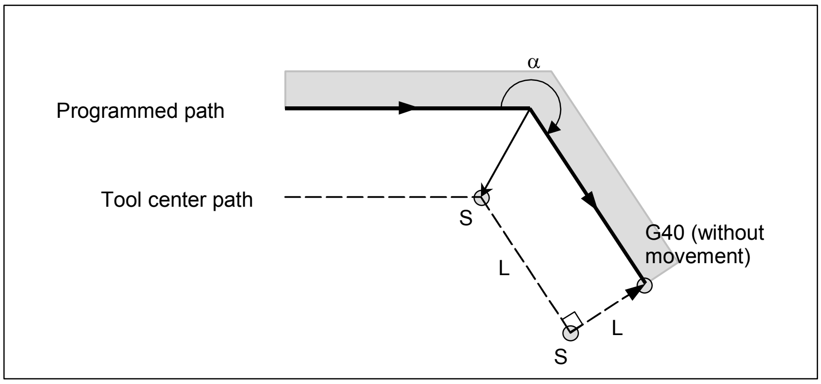
For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the cutter 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.



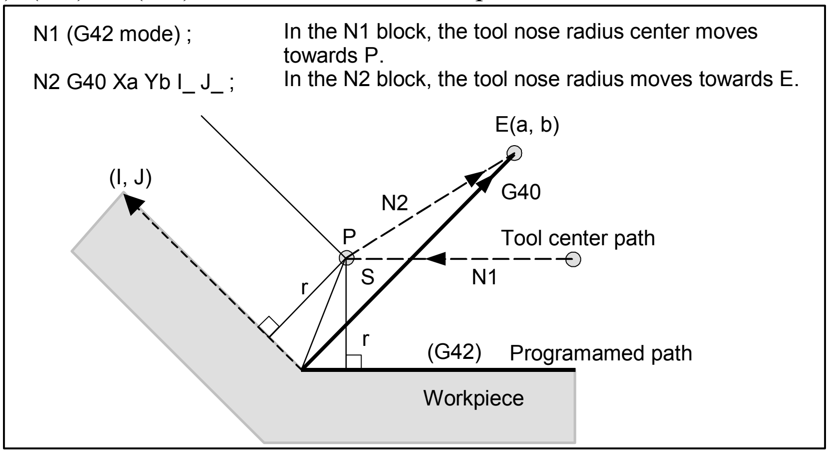
For type C

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

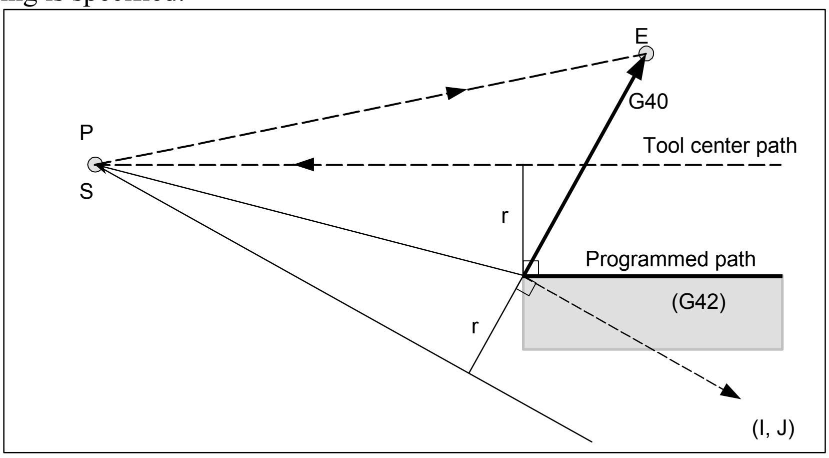


- Block containing G40 and I_J_K_
The previous block contains G41 or G42

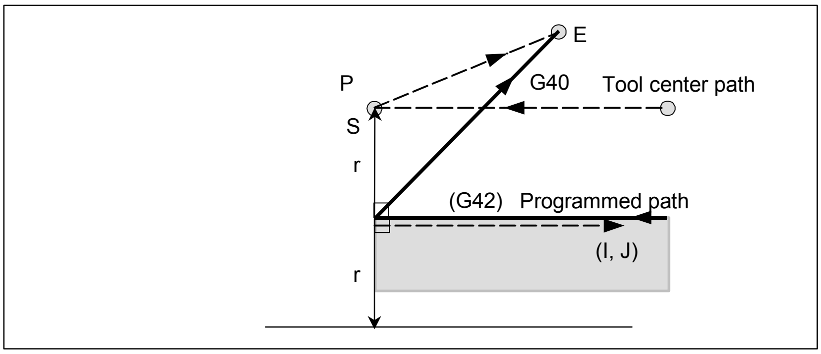
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end point determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified.

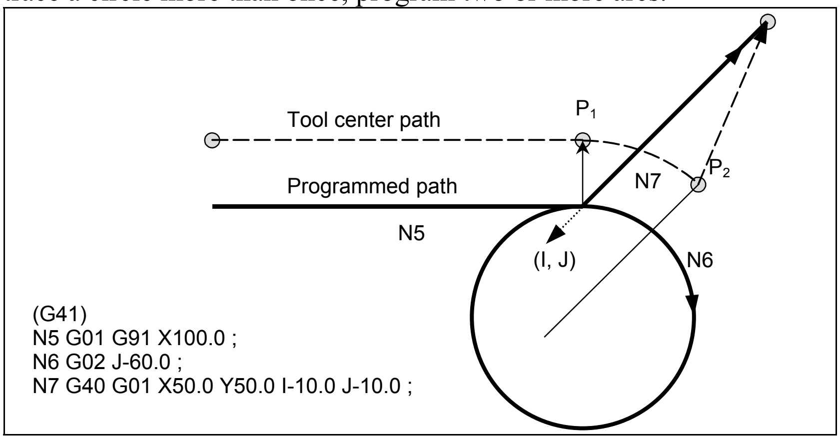


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



- 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 P_1 to P_2. The interference check function described below may raise an alarm.
To make the tool trace a circle more than once, program two or more arcs.



6.6.5 Prevention of Overcutting Due to Cutter 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.

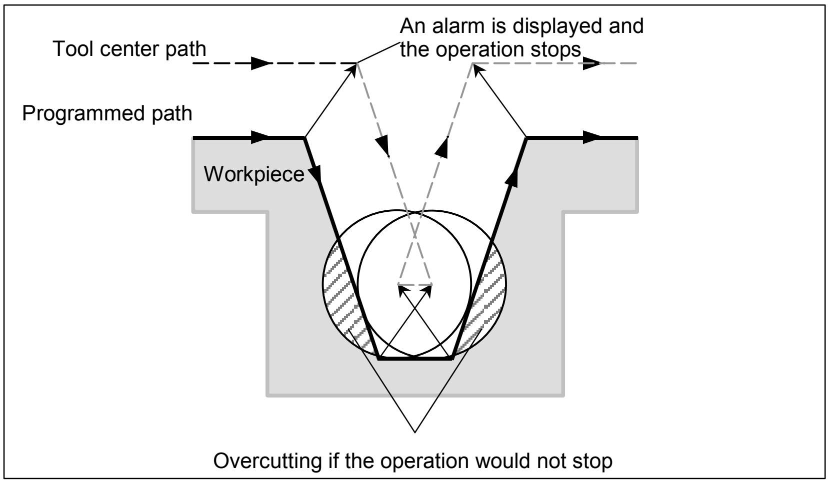


Fig. 6.6.5 (a) Machining a groove smaller than the diameter of the tool

- 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.

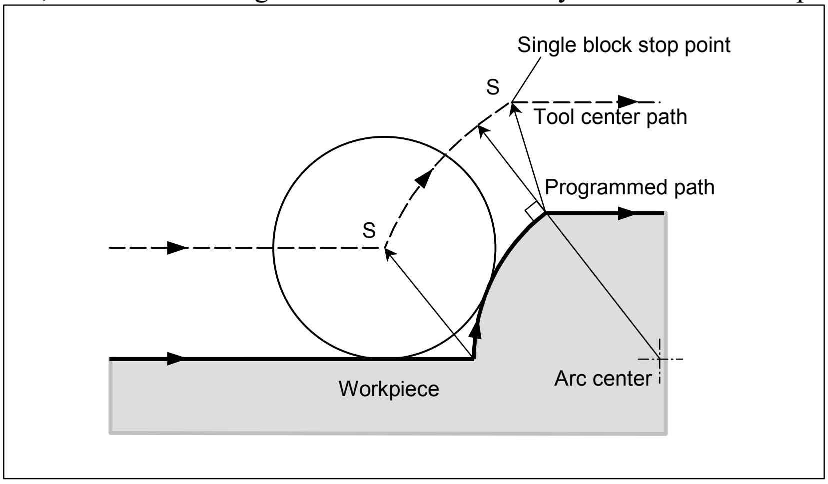


Fig. 6.6.5 (b) Machining a step larger than the tool radius

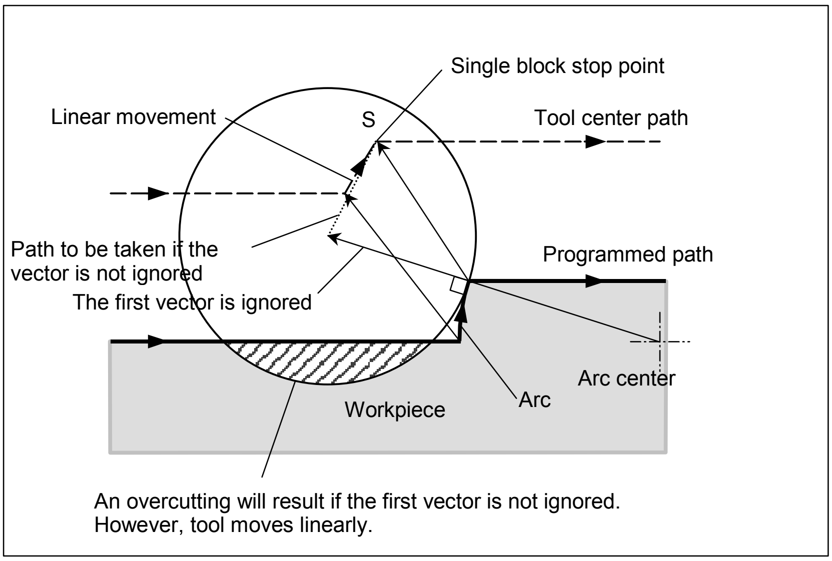
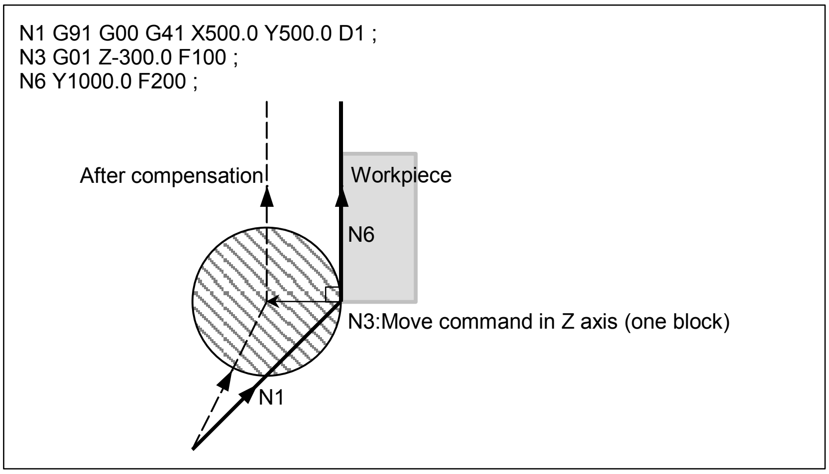


Fig. 6.6.5 (c) Machining a step smaller than the tool radius

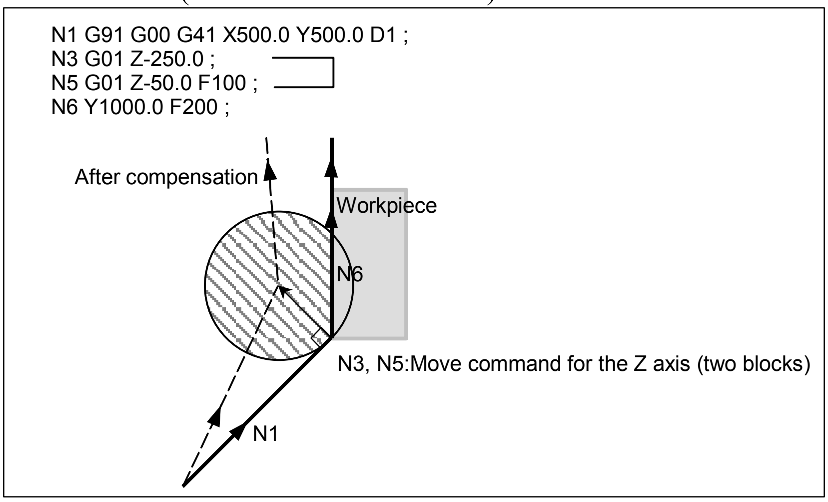
- Starting compensation and cutting along the Z-axis

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 following program, assuming the number of blocks to read in cutter compensation mode (parameter (No. 19625)) to be 3.



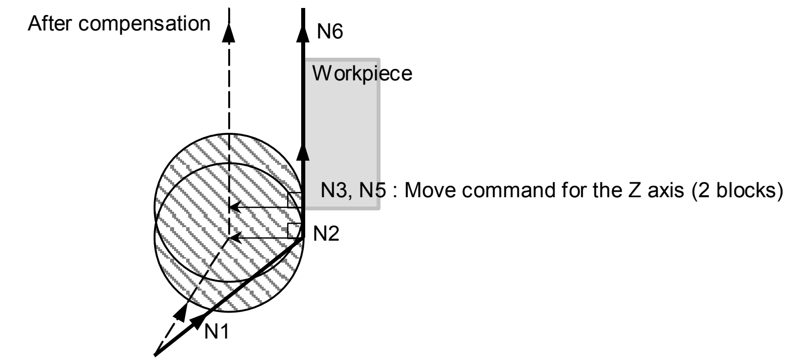
In the program example above, when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the figure above.
Then, suppose that the block N3 (move command in Z axis) is divided into N3 and N5.



At this time, because the number of blocks to read is 3, blocks up to N5 can be read at the start of N1 compensation, but block N6 cannot be read. As a result, compensation is performed only on the basis of the information in block N1, and a vertical vector is created at the end of the compensation start block. Usually, therefore, overcutting will result as shown in the figure above.

In such a case, it is possible to prevent overcutting by specifying a command with the exactly the same direction as the advance direction immediately before movement along the Z axis beforehand, after the tool is moved along the Z axis using the above rule.

```
N1 G91 G00 G41 X500.0 Y400.0 D1 ;
N2 Y100.0 ;
N3 Z-250.0 ;
N5 G01 Z-50.0 F100 ;
N6 Y1000.0 F200 ;
```



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. Parameter CNC (No. 5008#1) and parameter CNV (No. 5008#3) 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.

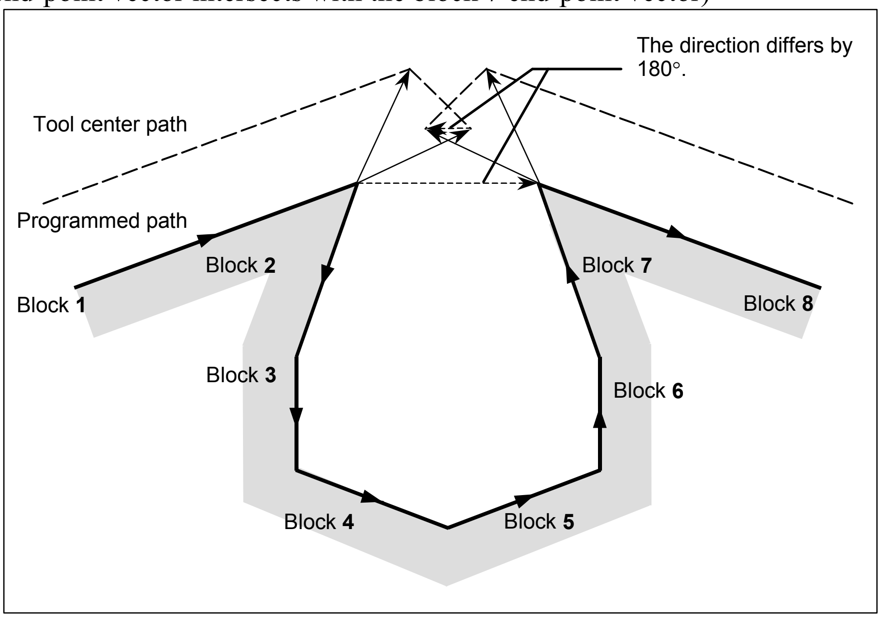
- Interference reference <1> (direction check)

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:

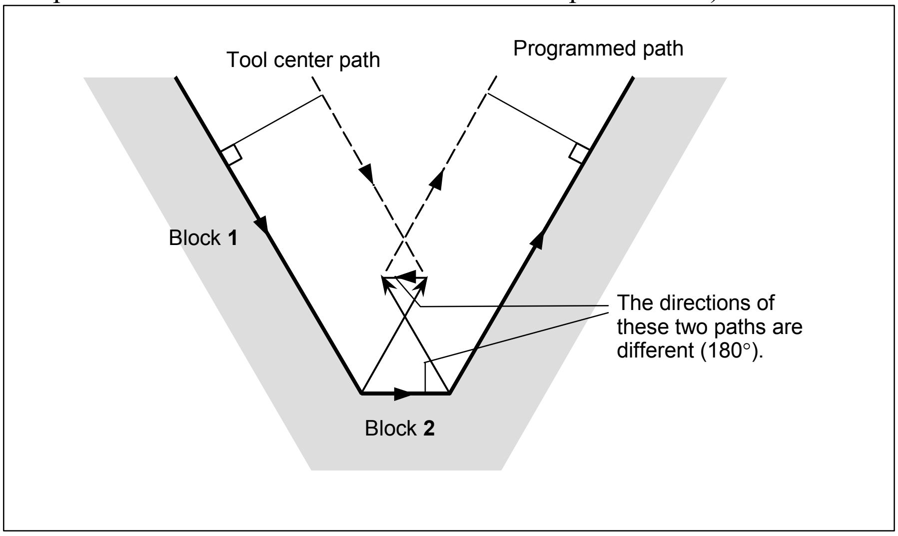
(Block 1 - block 2) and (block N-2 - block N-1)
(Block 1 - block 2) and (block N-3 - block N-2)
:
:
(Block 1 - block 2) and (block 2 - block 3)

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)



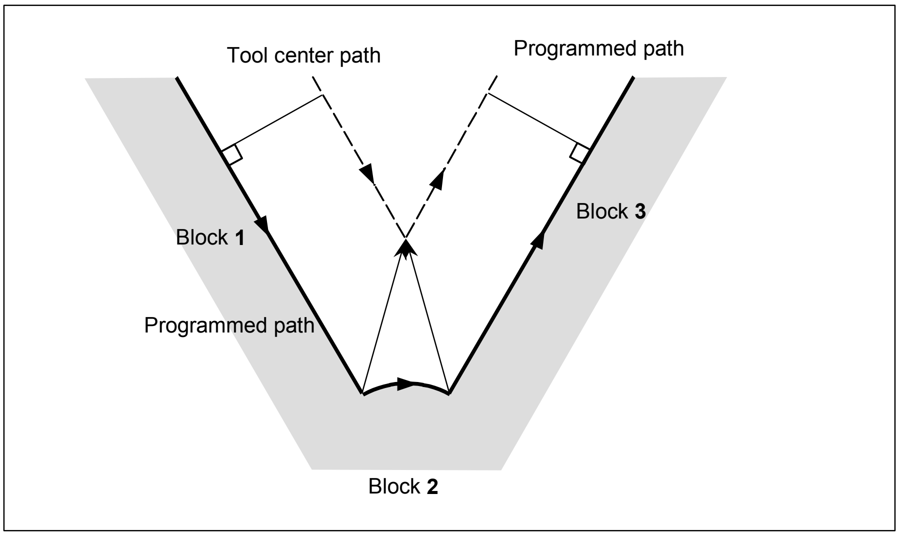
Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 2 end-point vector)



- Interference reference <2> (circular angle check)

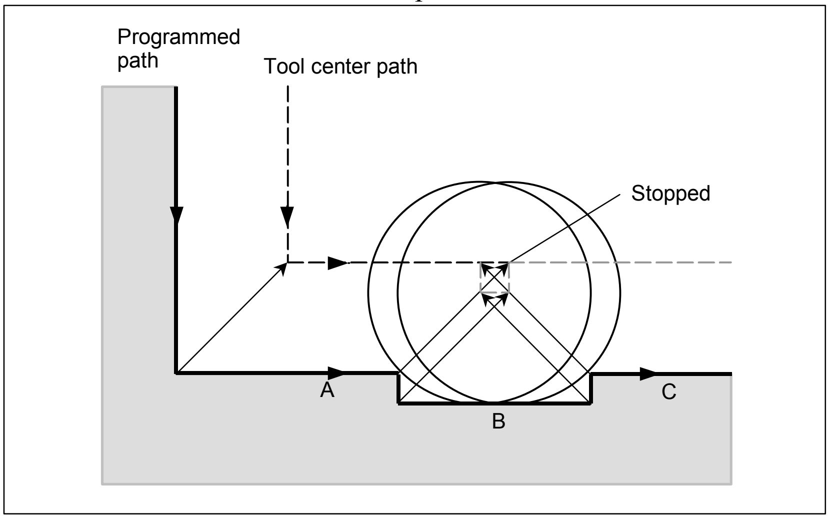
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed 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^{o} or greater, the blocks are judged to interfere. This is called a circular angle check.

Example of <2> (if block 2 is circular and the start point of the post-compensation arc coincide with the end point)



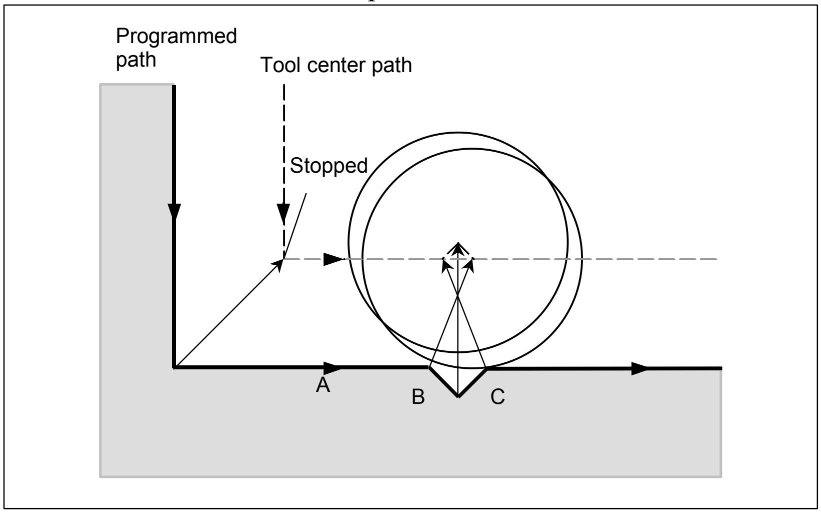
- When interference is assumed although actual interference does not occur

<1> Depression which is smaller than the cutter 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 cutter 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

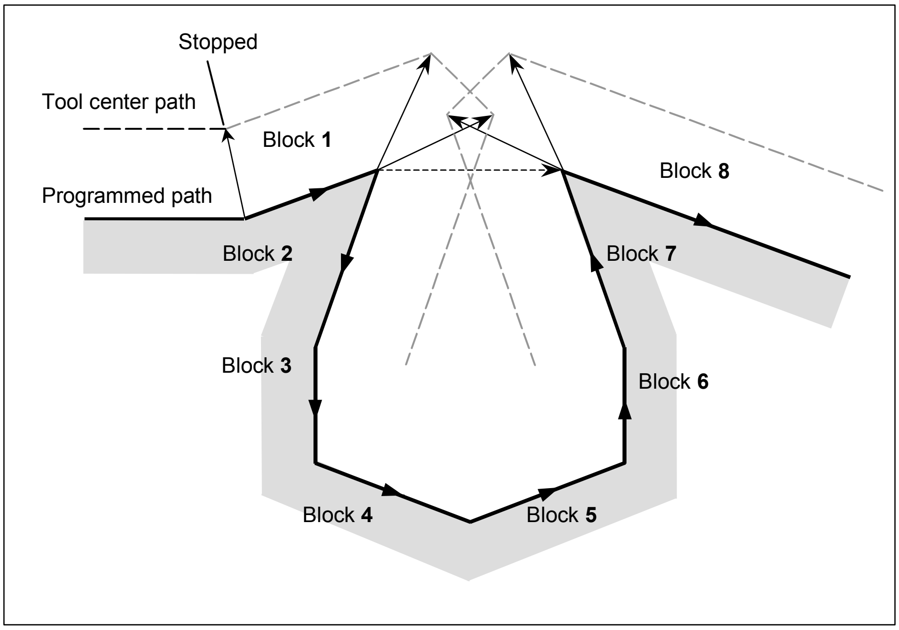
The operation to be performed if an interference check judges that an interference (due to overcutting) occurs can be either of the following two, depending on the setting of parameter CAV (No. 19607#5).

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

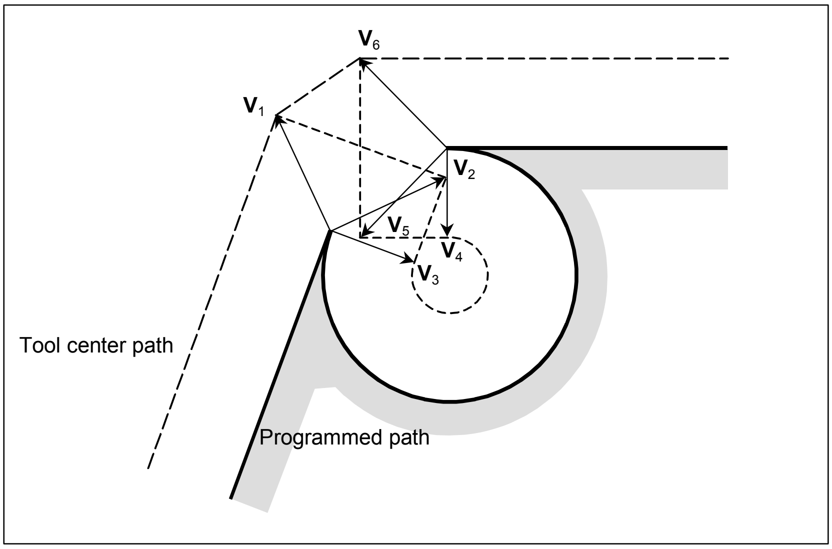
- Interference other than those between adjacent three blocks

If the end-point vector of block 1 and the end-point vector of block 7 are judged to interfere as shown in the figure, an alarm will occur before the execution of block 1 so that the tool stops. In this case, the vectors will not be erased.

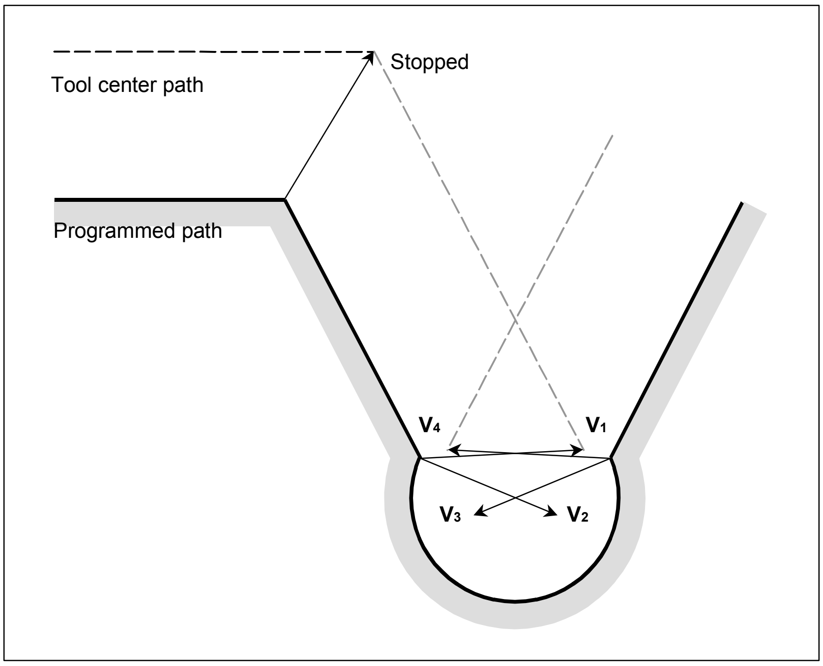


- Interference between adjacent three blocks

If an interference is judged to occur between adjacent three blocks, the interfering vector, as well as any vectors existing inside of it, is erased, and a path is created to connect the remaining vectors. In the Example shown in the figure below, V_2 and V_5 interfere, so that V_2 and V_5 are erased, so are V_3 and V_4, which are inside of them, and V_1 is connected to V_6. The operation during this time is linear interpolation.



If, after vector erasure, the last single vector still interferes, or if there is only one vector at the beginning and it interferes, an alarm will occur immediately after the start of the previous block (end point for a single block) and the tool stops. In the Example shown in the figure below, V_2 and V_3 interfere, but, even after erasure, an alarm will occur because the final vectors V_1 and V_4 interfere.



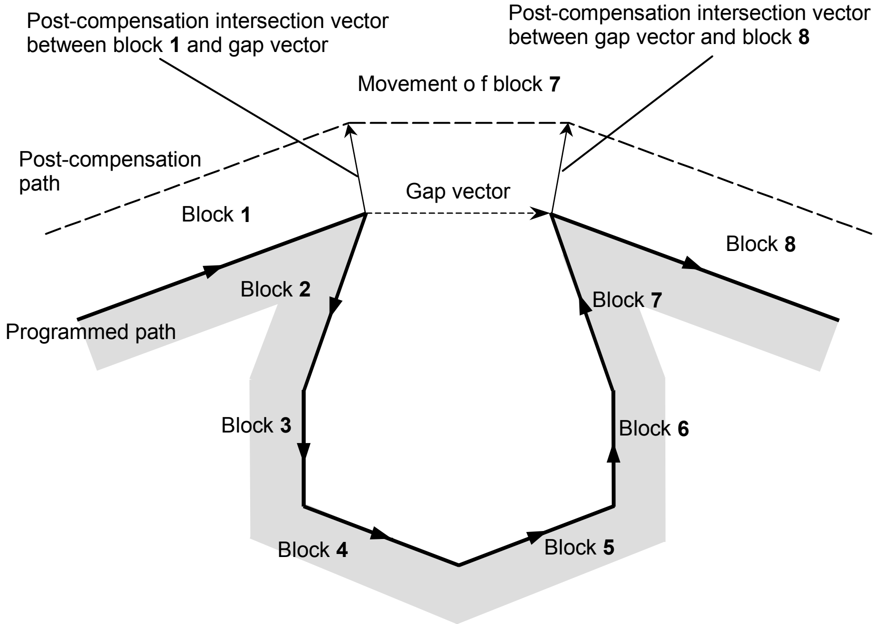
6.6.6.3 Interference check avoidance function

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.

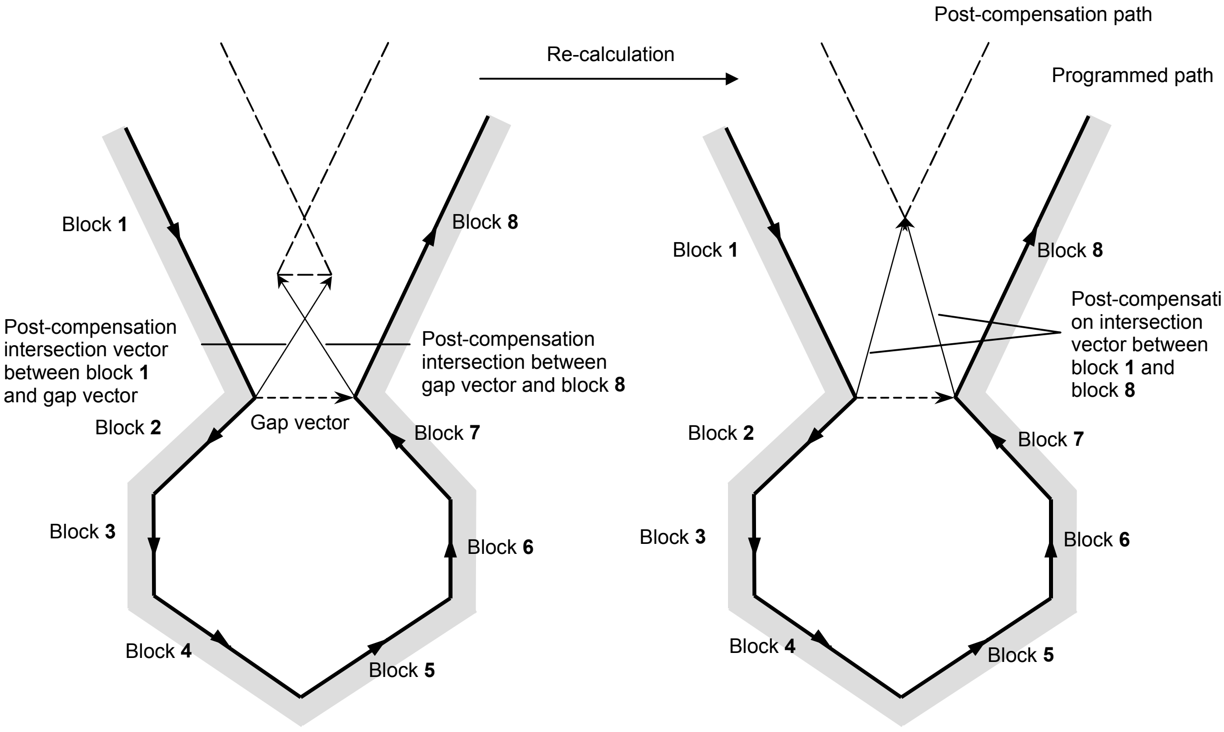
- Interference avoidance method

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.



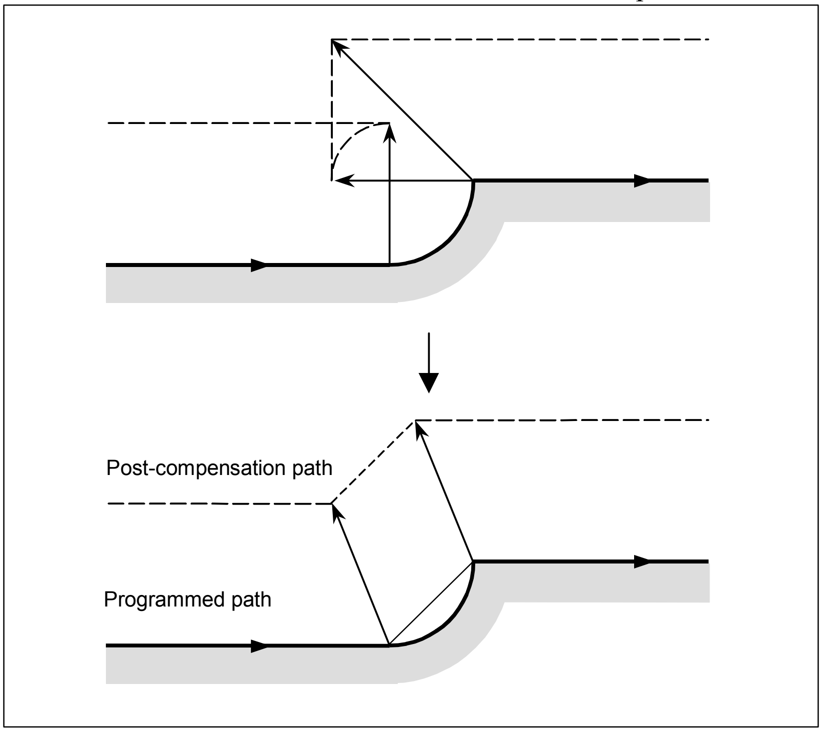
In this case, the post-compensation end points of blocks **2** to **6** coincide with the end point of block **1**. Thus, after compensation, blocks **2** to **6** will be blocks without tool movement.

If the post-compensation intersection vector of (block 1 - gap vector) and the post-compensation intersection vector of (gap vector - block N) further intersect, vector erasure is first performed in the same way as in "Interference between adjacent three blocks". If the last vectors that remains still intersects, the post-compensation intersection vector of (block 1 - block N) is re-calculated.



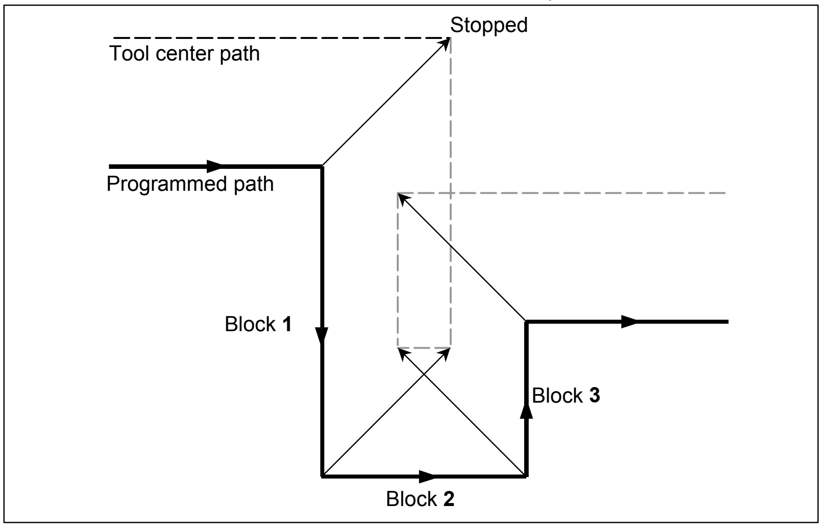
In this case, the post-compensation end points of blocks **2** to **7** coincide with the end point of block **1**. Thus, after compensation, blocks **2** to **7** will be blocks without tool movement.

If the cutter compensation value is greater than the radius of the specified arc as shown in the figure below, and a command is specified which results in compensation with respect to the inside of the arc, interference is avoided by performing intersection calculation with an arc command being assumed a linear one. In this case, avoided vectors are connected with linear interpolation.

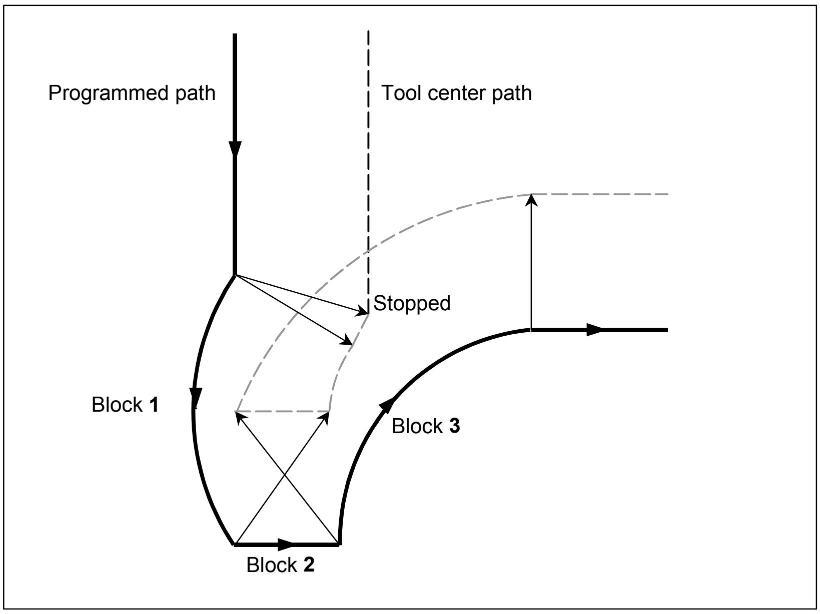


- If no interference avoidance vector exists

If the parallel pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are parallel to each other, no intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop.

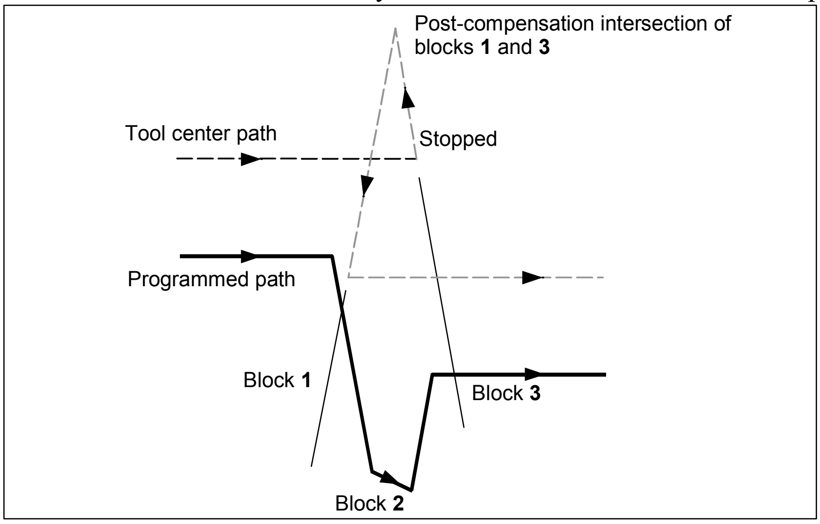


If the circular pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are circular, no post-compensation intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop, as in the previous example.

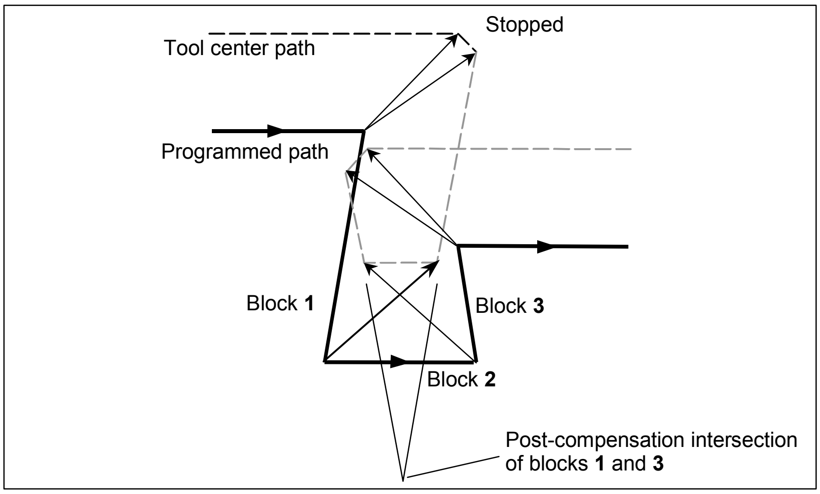


- If it is judged dangerous to avoid interference

If the acute-angle pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the movement direction of the post-avoidance path extremely differs from the previously specified direction. If the post-avoidance path extremely differs from that of the original command (90° or greater or 270° or less), interference avoidance operation is judged dangerous; an alarm will occur immediately before block 1 and the tool will stop.

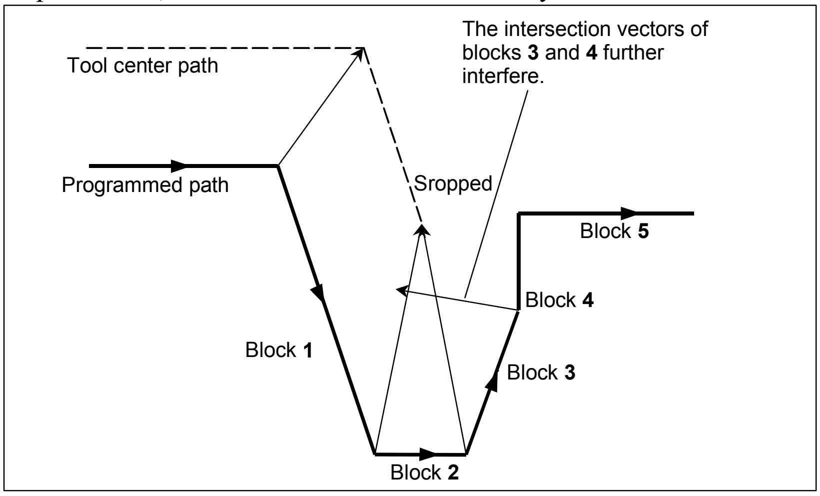


If a pocket in which the bottom is wider than the top, such as that shown in the figure, is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the relation between blocks 1 and 3 is judged an outer one, the post-avoidance path results in overcutting as compared with the original command. In such a case, interference avoidance operation is judge dangerous; an alarm will occur immediately before block 1 and the tool will stop.



- If further interference with an interference avoidance vector occurs

If the pocket shown in the figure is to be machined, if the number of blocks to read is 3, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, however, the end-point vector of block 3 that is to be calculated next further interferes with the previous interference avoidance vector. If a further interference occurs to the interference avoidance vector once created and output, the movement in the block will not be performed; an alarm will occur immediately before the block and the tool will stop.



NOTE

1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting parameter NAA (No. 19607#6) appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", however, it is not possible to avoid an alarm regardless of the setting of this parameter.

2 If a single block stop occurs during interference avoidance operation, and an operation is performed which differs from the original movement, such as manual intervention, MDI intervention, cutter 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 Cutter Compensation for Input from MDI

Explanation

- MDI operation

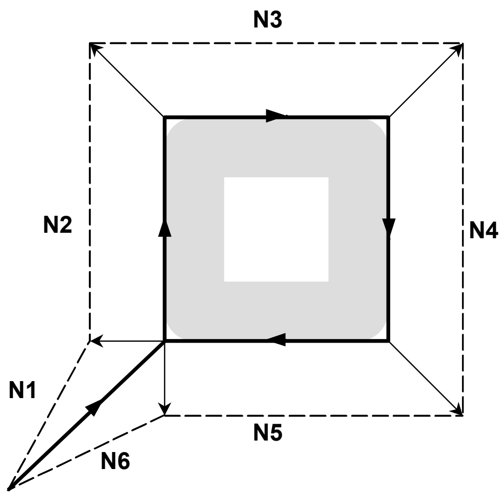
During MDI operation, that is, if a program command is specified in MDI mode in the reset state to make a cycle start, intersection calculation is performed for compensation in the same way as in memory operation/DNC operation. Compensation is performed in the same way if a subprogram is called from program memory due to MDI operation.

MDI command

```
G90 G00 X0 Y0 ;
M98 P9000 ;
M02 ;
```

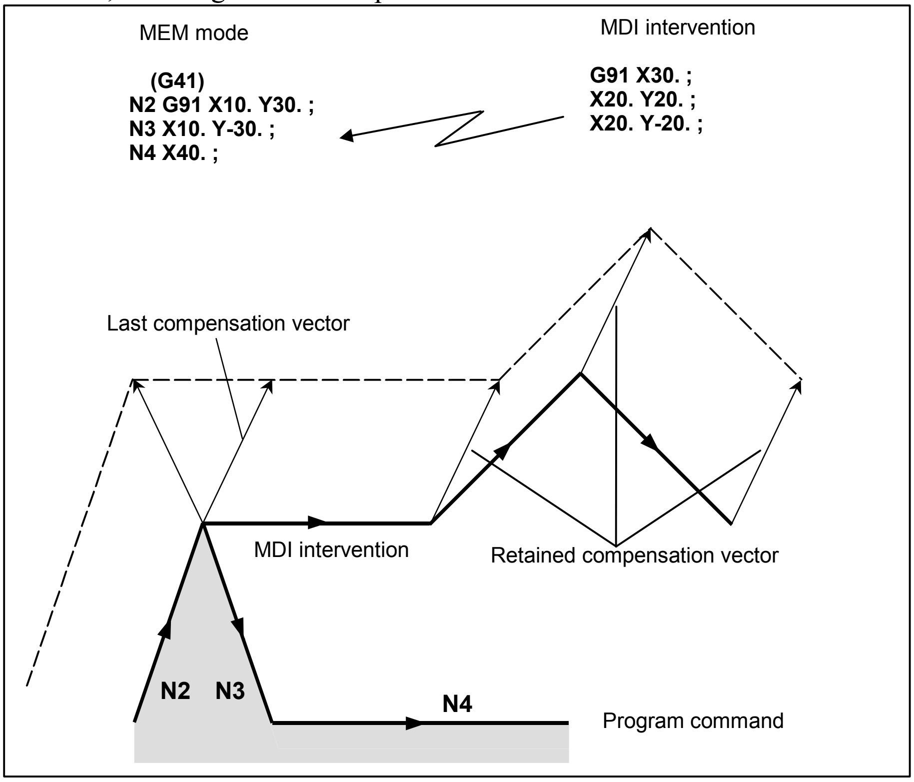
Subprogram in program memory

```
O9000 ;
N1 G41 G17 G91 G01 X10. Y10.
    D1 ;
N2 Y15. ;
N3 X15. ;
N4 Y-15. ;
N5 X-15. ;
N6 G40 X-10. Y-10. ;
M99 ;
```



- 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 CORNER CIRCULAR INTERPOLATION (G39)

By specifying G39 in offset mode during cutter compensation, corner circular interpolation can be performed. The radius of the corner circular interpolation equals the compensation value.

Format

In offset mode

G39 ;

or

$$\mathbf{G39}\left\{\begin{matrix}\mathbf{I\_\ J\_}\\ \mathbf{I\_\ K\_}\\ \mathbf{J\_\ K\_}\end{matrix}\right\}\ ;$$

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

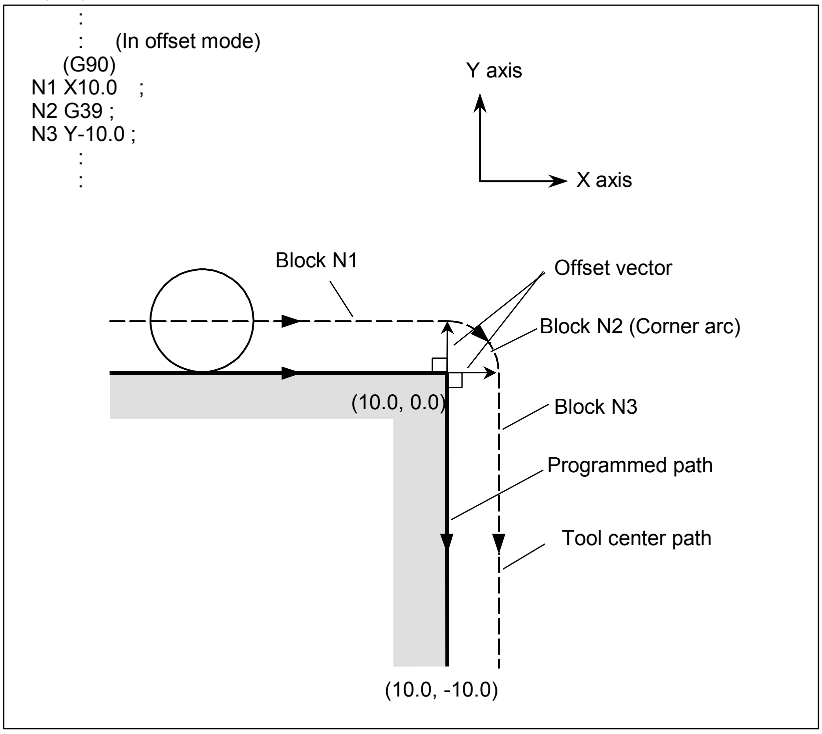
In an inner corner block, G39 cannot be specified. Otherwise, overcutting will occur.

- Corner arc velocity

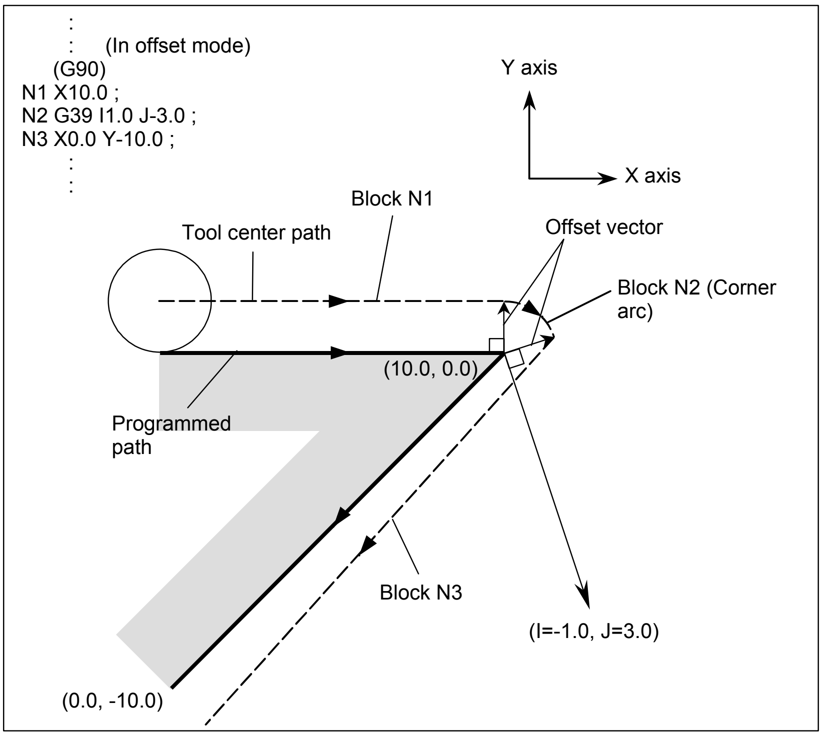
If a corner arc is specified with G39 in G00 mode, the corner arc block velocity will be that of the F command previously specified. If G39 is specified in a state in which no F command has never been specified, the velocity of the corner arc block will be that specified with parameter (No. 1411).

Example

- G39 without I, J, or K



- G39 with I, J, and K



6.8 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. 6.8 (a)).

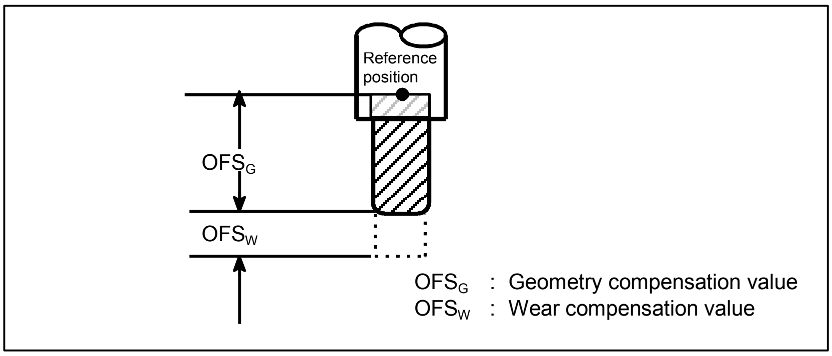


Fig. 6.8 (a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the MDI panel (see section III-1.1.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, cutter compensation, or the tool offset.
Two types of tool compensation memories are available according to the compensation value configuration: tool compensation memory A and C. One of the types can be selected (bit 6 (NGW) of parameter No.8136).

Explanation

- Tool compensation memory A (bit 6 (NGW) of parameter No.8136 = 1)

In tool compensation memory A, memory for geometry compensation and memory for wear compensation are not distinguished from each other. So, the sum of geometry compensation and wear compensation values is to be set in the compensation memory. Moreover, no distinction is made between memory for cutter compensation (for D code) and memory for tool length compensation (for H code).

Setting example

Compensation number	Compensation value (geometry+wear)	Common to D code/H code
001	10.000	For D code
002	20.000	For D code
003	100.000	For H code
:	:	:

- Tool compensation memory C (bit 6 (NGW) of parameter No.8136 = 0)

In tool compensation memory C, memory for geometry compensation and memory for wear compensation are prepared separately. So, geometry compensation values and wear compensation values can be set separately. Moreover, memory for cutter compensation (for D code) and memory for tool length compensation (for H code) are prepared separately.

Setting example

Compensation number	D code		H code	
	For geometry compensation	For wear compensation	For geometry compensation	For wear compensation
001	10.000	0.100	100.000	0.100
002	20.000	0.200	200.000	0.300
:	:	:	:	:

- Unit and valid range of tool compensation values

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

Unit and valid range of tool compensation values (metric input)

OFC	OFA	Unit	Valid range
0	1	0.01mm	±9999.99mm
0	0	0.001mm	±9999.999mm
1	0	0.0001mm	±9999.9999mm

Unit and valid range of tool compensation values (inch input)

OFC	OFA	Unit	Valid range
0	1	0.001inch	±999.999inch
0	0	0.0001inch	±999.9999inch
1	0	0.00001inch	±999.99999inch

- Number of tool compensation data items

Using the bit 5 (NDO) of parameter No. 8136 enables the total number of items of tool compensation data to be specified as either 400 (bit 5 (NDO) of parameter No. 8136 = "0") or 32 (bit 5 (NDO) of parameter No. 8136 = "1").

Format

The format for programming depends on the type of tool compensation memory.

For tool compensation memory A

G10 L11 P_ R_ ;

P_ : Tool compensation number
R_ : Tool compensation value

For tool compensation memory C

G10 L_ P_ R_ ;

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

P_ : Tool compensation number
R_ : Tool compensation value

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.

6.9 SCALING (G50, G51)

Overview

A programmed figure can be magnified or reduced (scaling).
Two types of scaling are available, one in which the same magnification rate is applied to each axis and the other in which different magnification rates are applied to different axes.
The magnification rate can be specified in the program.
Unless specified in the program, the magnification rate specified in the parameter is applied.

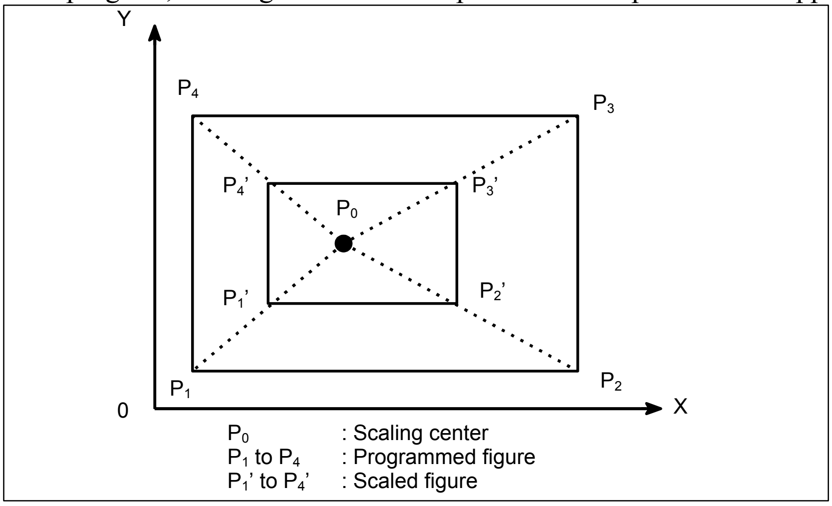


Fig. 6.9 (a) Scaling

> **NOTE**
> To enable scaling, set bit 5 (SCL) of parameter No. 8132 to "1".

Format

Scaling up or down along all axes at the same rate of magnification
(When parameter XSC (No. 5400#6) = 0)

Format	Meaning of command
G51 IP_P_ ; Scaling start } Scaling is effective. (Scaling mode) **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **P_** : Scaling magnification

Scaling up or down along each axes at a different rate of magnification (mirror image)
(When parameter XSC (No. 5400#6) = 1)

Format	Meaning of command
G51 IP_I_J_K_; Scaling start } Scaling is effective. (Scaling mode) **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **I_J_K_** : Scaling magnification for basic 3 axes (X, Y, and Z axes) respectively

> ⚠ **CAUTION**
> 1 Specify G51 in a separate block.
> 2 After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

NOTE

1 Entering electronic calculator decimal point input mode (bit 0 (DPI) of parameter No. 3401 = 1) does not cause the units of the magnification rates P, I, J, and K to change.

2 Setting the least input increment equal to 10 times the least command increment (bit 7 (IPR) of parameter No. 1004 = 1) does not cause the units of the magnification rates P, I, J, and K to change.

3 An attempt to specify 0 as a magnification rate causes alarm PS0142 to occur in a G51 block.

Explanation

- Axis for which scaling is to be enabled

For the axis for which scaling is to be enabled, set bit 0 (SCL) of parameter No. 5401 to 1.

- Minimum unit of scaling magnification

Least input increment of scaling magnification is: 0.001 or 0.00001.
It is 0.00001 (one hundred thousandth) if bit 7 (SCR) of parameter No. 5400 is 0 and 0.001 if it is 1.

- Scaling center

Even in incremental command (G91) mode, the scaling center coordinates IP_ specified in the G51 block are assumed those of an absolute position.
If the scaling center coordinates are omitted, the position assumed when G51 is specified is assumed the scaling center.

⚠ **CAUTION**

With the move command subsequent to the G51 block, execute an absolute (G90 mode) position command.
If no absolute position command is executed after the G51 block, the position assumed when G51 is specified is assumed the scaling center; once an absolute position command is executed, the scaling center assumes the coordinates specified in the G51 block, after that block.

- Scaling along each axis at the same rate of magnification

Set bit 6 (XSC) of parameter No. 5400 to 0.
If the scaling magnification P is not specified, the magnification set in parameter No. 5411 is used.
Decimal point input is not accepted as the magnification P. If decimal point input is made, alarm PS0007 will occur.
A negative value cannot be specified as the magnification P. If a negative value is specified, alarm PS0006 will occur.
The allowable magnification range is from 0.00001 to 9999.99999.

- Scaling of each axis, programmable mirror image (negative magnification)

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. The axis subject to the mirror image is the one that contains the scaling center.
Set bit 6 (XSC) of parameter No. 5400to 1 to validate each axis scaling (mirror image).
Using I, J, and K, specify the scaling magnifications for the basic 3 axes (X to Z axes). Use parameter No. 1022 to specify which axes to use as the basic 3 axes. For those of the X to Z axes for which I, J, and K are not specified and for axes other than the basic 3 axes, the magnification set with parameter No. 5421 is used.
A value other than 0 must be set to parameter No. 5421.
Decimal point programming can not be used to specify the rate of magnification (I, J, K).
Magnification can be set within the range of ±0.00001 to ±9999.99999.

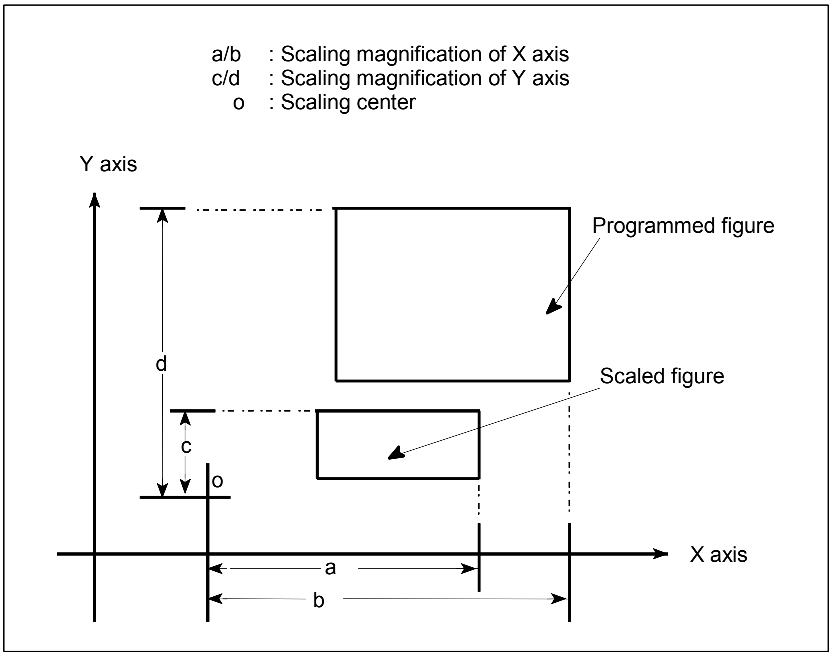


Fig. 6.9 (b) Scaling of each axis

> ⚠ **CAUTION**
> Specifying the following commands at the same time causes them to be executed in the order indicated below:
> <1> Programmable mirror image (G51.1)
> <2> Scaling (G51) (including a mirror image with a negative magnification)
> <3> Mirror image due to the external switch of the CNC or the settings of the CNC
> In this case, the programmable mirror image is effective to the scaling center and magnification as well.
> To specify G51.1 and G51 at the same time, specify them in this order; to cancel them, specify them in the reverse order.

- Scaling of circular interpolation

Even if different magnifications are applied to each axis in circular interpolation, the tool will not trace an ellipse.

G90 G00 X0.0 Y100.0 Z0.0;
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
(A magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.)
G02 X100.0 Y0.0 I0 J–100.0 F500;

Above commands are equivalent to the following command:

G90 G00 X0.0 Y100.0 Z0.0;
G02 X200.0 Y0.0 I0 J–100.0 F500;
(Because the end point is not on an arc, spiral interpolation is assumed.)

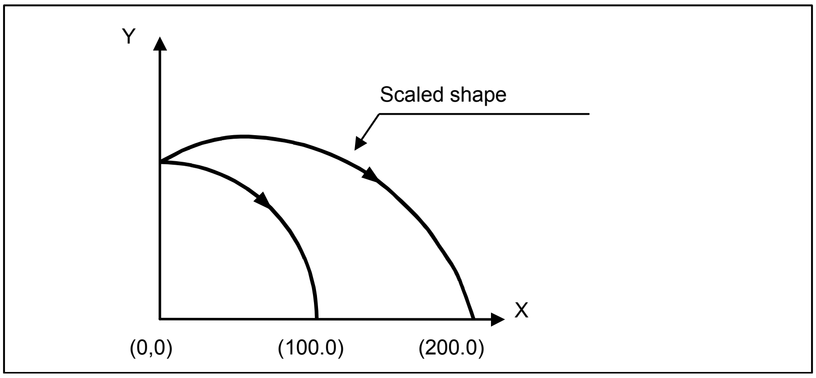


Fig. 6.9 (c) Scaling for circular interpolation1

Even for an R-specified arc, scaling is applied to each of I, J, and K after the radius value (R) is converted into a vector in the center direction of each axis.
If, therefore, the above G02 block contains the following R-specified arc, the operation will be same as that in which I and J are specified.

G02 X100.0 Y0.0 R100.0 F500 ;

- Scaling and coordinate system rotation

If both scaling and coordinate system rotation are specified at the same time, scaling is performed first, followed by coordinate system rotation. In this case, scaling is effective to the rotation center as well.
To specify both of them, specify scaling first and then coordinate system rotation. To cancel them, specify them in the reverse order.

Example
Main program

```
O1
G90 G00 X20.0 Y10.0 ;
M98 P1000 ;
G51 X20.0 Y10.0 I3000 J2000 ;        (x 3 in the X direction and x 2 in the Y direction)
M98 P1000 ;
G17 G68 X35.0 Y20.0 R30. ;
M98 P1000 ;
G69 ;
G50 ;
M30 ;
```

Subprogram

```
O1000 ;
G01 X20.0 Y10.0 F500 ;
G01 X50.0 ;
G01 Y30.0 ;
G01 X20.0 ;
G01 Y10.0 ;
M99 ;
```

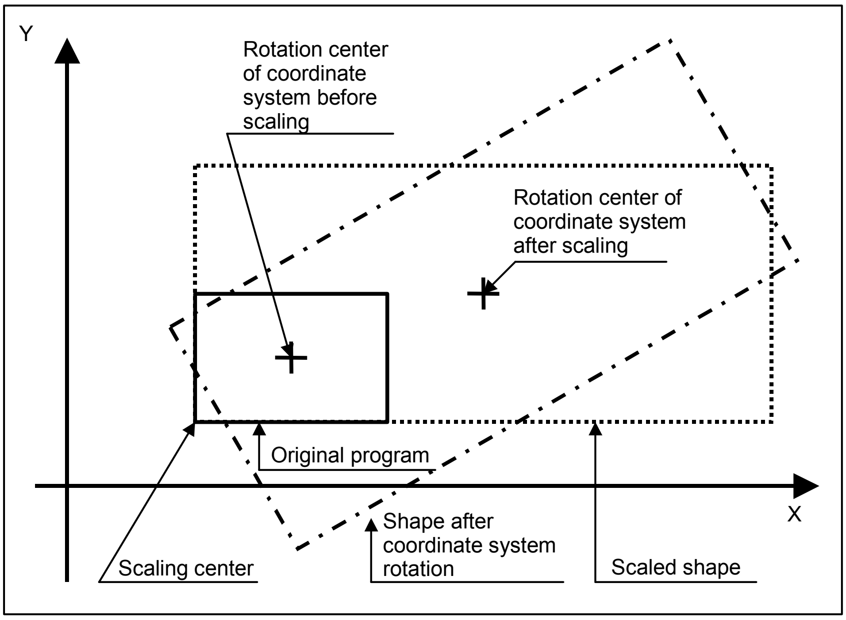


Fig. 6.9 (d) Scaling and coordinate system rotation

- Scaling and optional chamfering/corner R

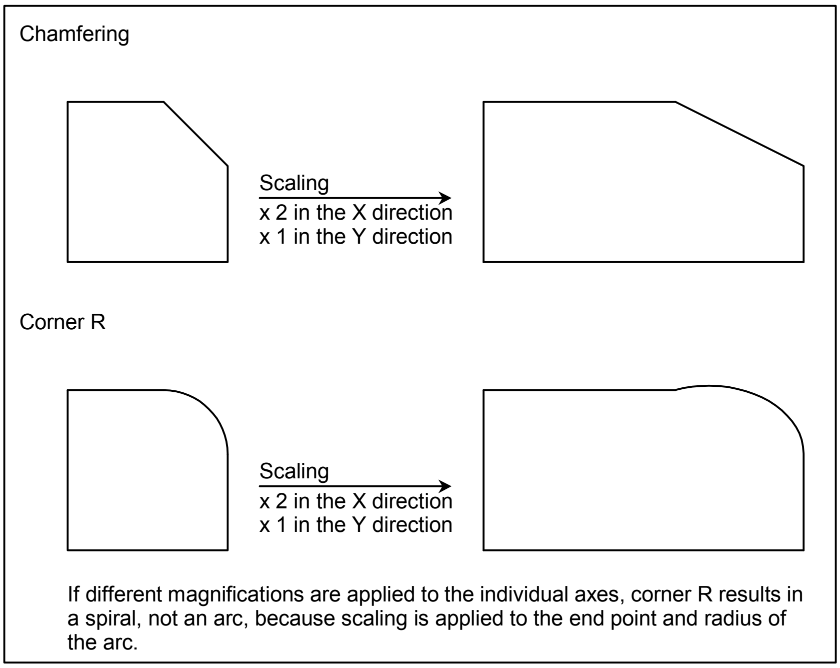


Fig. 6.9 (e) Scaling and optional chamfering/corner R

Limitation

- Tool compensation

This scaling is not applicable to cutter compensation values, tool length compensation values, and tool offset values (Fig. 6.9 (f)).

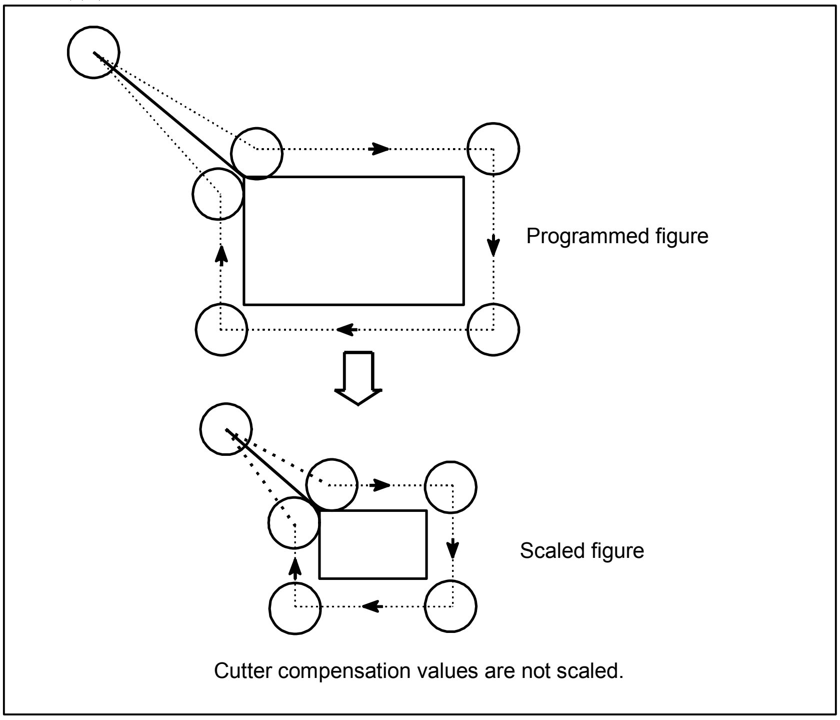


Fig. 6.9 (f) Scaling during cutter compensation

- Invalid scaling

Scaling is not applied to the travel distance during canned cycle shown below.

- Cut-in value Q and retraction value d of peck drilling cycle (G83, G73).
- Fine boring cycle (G76)
- Shift value Q of X and Y axes in back boring cycle (G87).

In manual operation, the travel distance cannot be increased or decreased using the scaling.

> ⚠ **CAUTION**
> 1 If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
> 2 Before specifying the G code for reference position return (G27, G28, G29, G30, etc.) or coordinate system setting (G52 to G59, G92, etc.), cancel the scaling mode. If it is specified without canceling scaling, the alarm PS0412 is issued.
> 3 If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In this case, the block is regarded as a no movement block, and therefore, it may affect the tool movement by cutter compensation. See the description of cutter compensation.
> 4 Refrain from scaling on a rotation axis for which the rollover function is enabled. Otherwise, the tool may rotate in a short-cut manner, possibly resulting in unexpected movement.

NOTE

1 The position display represents the coordinate value after scaling.
2 When a mirror image was applied to one axis of the specified plane, the following results:
   (1) Circular command .... Direction of rotation is reversed.
   (2) Tool radius · tool nose radius compensation
   .... Offset direction is reversed.
   (3) Coordinate system rotation .... Rotation angle is reversed.

Example

Sample program of a scaling in each axis

```
O1;
G51 X20.0 Y10.0 I750 J250; (× 0.75 in the X direction, × 0.25 in the Y direction)
G00 G90 X60.0 Y50.0;
G01 X120.0 F100;
G01 Y90;
G01 X60;
G01 Y50;
G50;
M30;
```

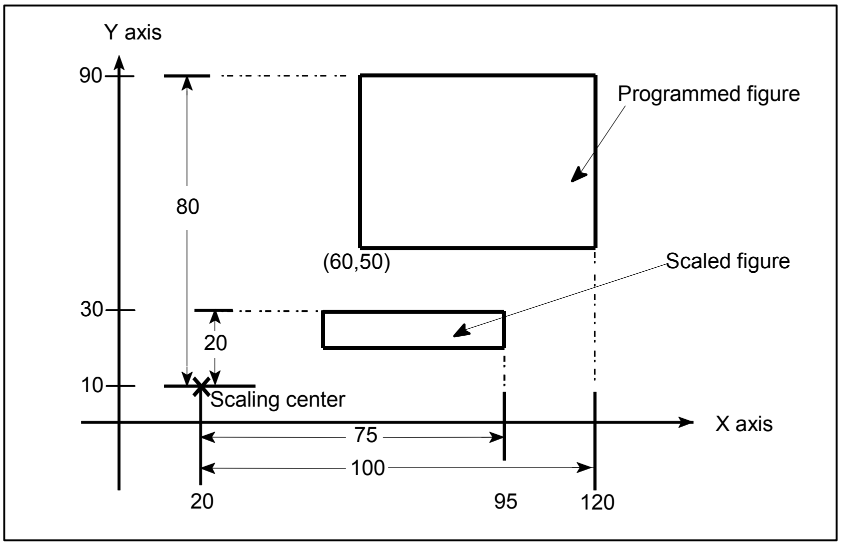


Fig. 6.9 (g) Program example of scaling in each axis

6.10 COORDINATE SYSTEM ROTATION (G68, G69)

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation.

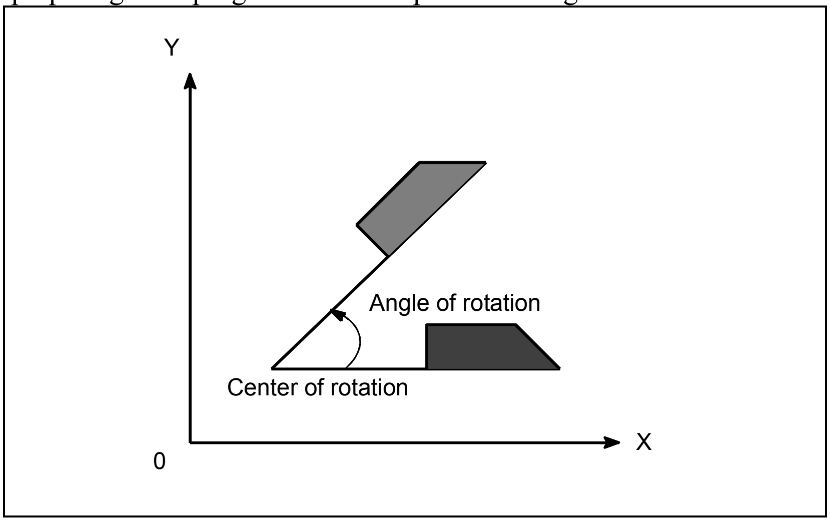


Fig. 6.10 (a) Coordinate system rotation

Format

Format	
{G17 / G18 / G19} G68 α_β_ R_ ;	Start rotation of a coordinate system.
:	Coordinate system rotation mode (The coordinate system is rotated.)
G69 ;	Coordinate system rotation cancel command
Meaning of command	
G17 (G18 or G19) :	Select the plane in which contains the figure to be rotated.
α_β_	Absolute programming for two of the X_, Y_, and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68
R_	Angular displacement with a positive value indicates counter clockwise rotation. Parameter RIN (No. 5400#0) 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

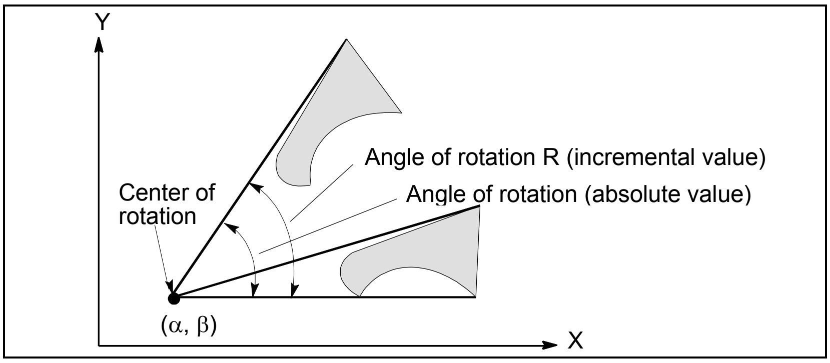


Fig. 6.10 (b) Coordinate system rotation

> **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 parameter FRD (No. 11630#0) 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

Cutter compensation, tool length compensation, tool offset, and other compensation operations are executed after the coordinate system is rotated.

Limitation

- Commands related to reference position return and the coordinate system

In coordinate system rotation mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling coordinate system rotation mode. If it is specified without canceling scaling, the alarm PS0412 is issued.

- 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

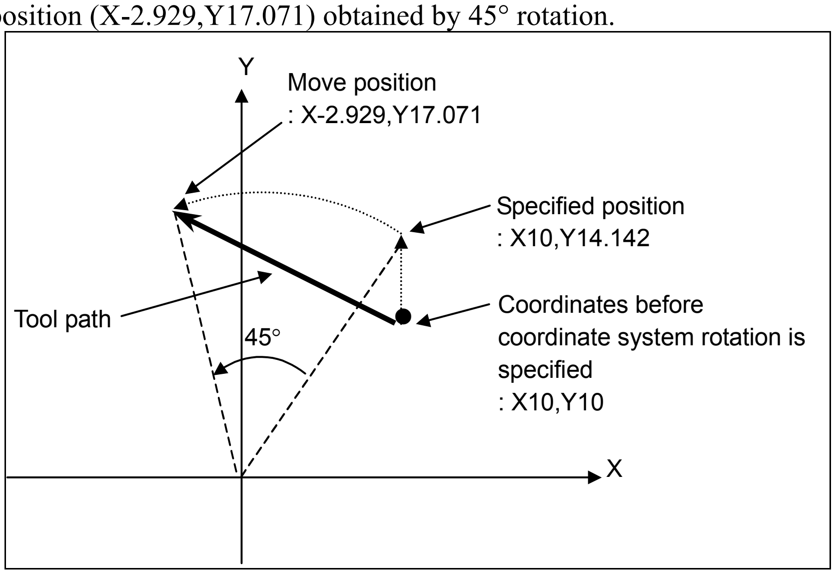
With the parameter below, a move position in the case where one axis is specified in the absolute mode can selected. If two axes are specified, a movement is made to the same position, regardless of the setting of the parameter.

Parameter AX1 (No. 11600#5)
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.
(FS0*i*-compatible specification)
This parameter changes the handling of coordinates on axes not specified, so that a position to be reached by movement changes.

(Example)

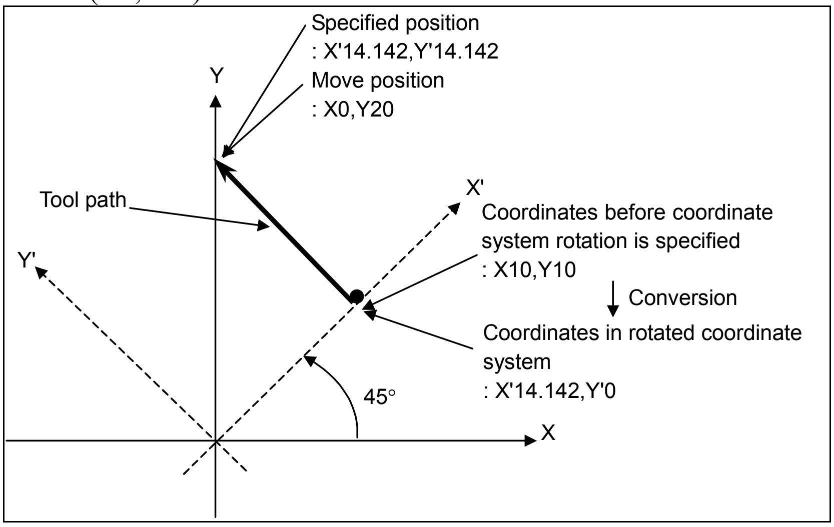
```
G90 G0 X0 Y0
G01 X10. Y10. F6000
G68 X0 Y0 R45...........Specifies coordinate system rotation.
Y14.142........................Specifies one axis ....(1)
G69
```

When parameter AX1 (No. 11600#5) = 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 parameter AX1 (No. 11600#5) = 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 ;
```

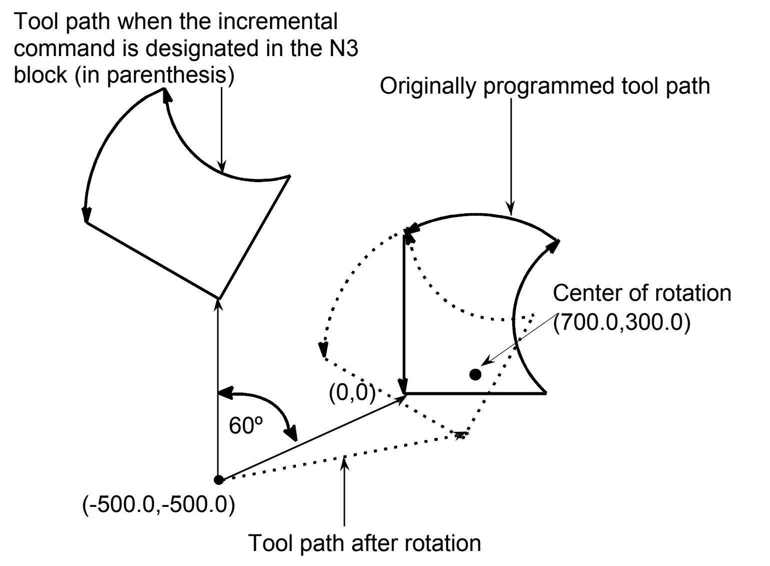


Fig. 6.10 (c) Absolute/incremental programming during coordinate system rotation

- Cutter compensation and coordinate system rotation

It is possible to specify G68 and G69 in cutter compensation mode.
The rotation plane must coincide with the plane of cutter compensation.

```
N1 G92 X0 Y0 G69 G01 ;
N2 G42 G90 X1000 Y1000 F1000 D01 ;
N3 G68 R-30000 ;
N4 G91 X2000 ;
N5 G03 Y1000 R1000 J500 ;
N6 G01 X-2000 ;
N7 Y-1000 ;
N8 G69 G40 G90 X0 Y0 M30 ;
```

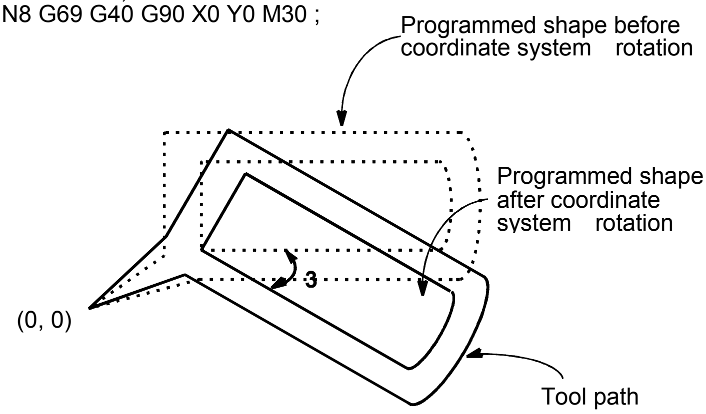


Fig. 6.10 (d) Cutter compensation and coordinate system rotation

- 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.

1. When the system is not in cutter compensation mode, specify the commands in the following order :
   G51 ; Scaling mode start
   G68 ; Coordinate system rotation mode start
   :
   G69 ; Coordinate system rotation mode cancel
   G50 ; Scaling mode cancel

2. When the system is in cutter compensation, specify the commands in the following order (Fig.6.10(e)) :
   (cutter compensation cancel)
   G51 ; Scaling mode start
   G68 ; Coordinate system rotation start
   :
   G41 ; Cutter compensation mode start
   :

```
G92 X0 Y0 ;
G51 X300.0 Y150.0 P500 ;
G68 X200.0 Y100.0 R45.0 ;
G01 X400.0 Y100.0 ;
Y100.0 ;
X-200.0 ;
Y-100.0 ;
X200.0 ;
```

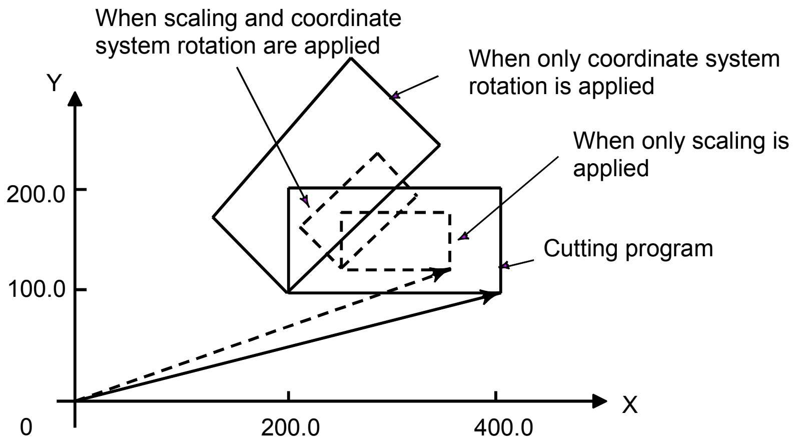


Fig. 6.10 (e) Scaling and coordinate system rotation in cutter compensation mode

- 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 displancement is treated as an absolute or incremental value depending on the specified G code (G90 or G91).

```
    G92 X0 Y0 G69 G17;
    G01 F200 H01 ;
    M98 P2100 ;
    M98 P072200 ;
    G00 G90 X0 Y0 M30 ;

O 2200 G68 X0 Y0 G91 R45.0 ;
    G90 M98 P2100 ;
    M99 ;

O 2100 G90 G01 G42 X0 Y-10.0 ;
    X4.142 ;
    X7.071 Y-7.071 ;
    G40 ;
    M99 ;
```

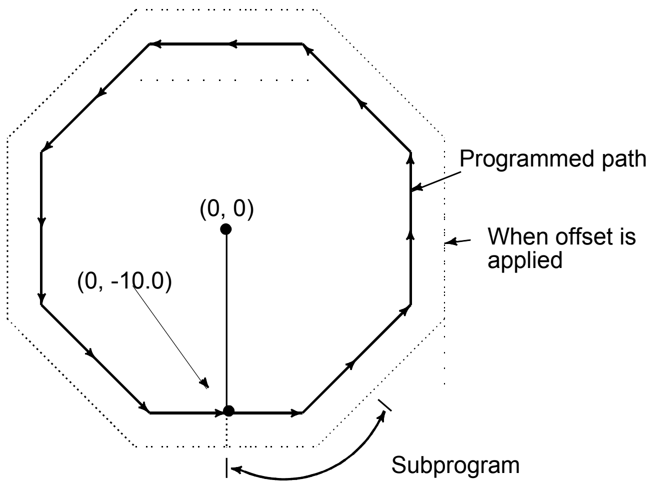


Fig. 6.10 (f) Coordinate system rotation command

6.11 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1)

Overview

When a tool with a rotation axis (C-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C-axis is always perpendicular to the tool path (Fig. 6.11 (a)).

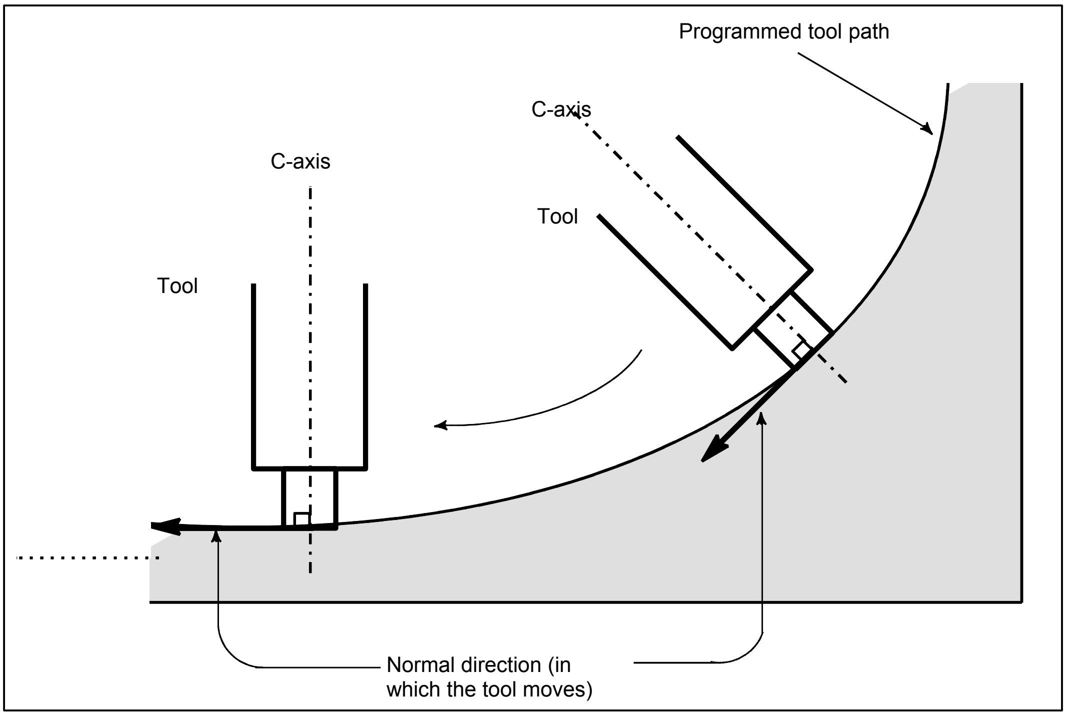


Fig. 6.11 (a) Sample Movement of the tool

Format

G41.1 ; Normal direction control, left

G42.1 ; Normal direction control, right

G40.1 ; Cancel normal direction control

e normal direction control, left (G41.1) command is used when the workpiece is on the right side of the tool as viewed while you are looking into the tool's way.
Once either G41.1 or G42.1 is issued, normal direction control is enabled (normal direction control mode).
Issuing G40.1 cancels the normal direction control mode.

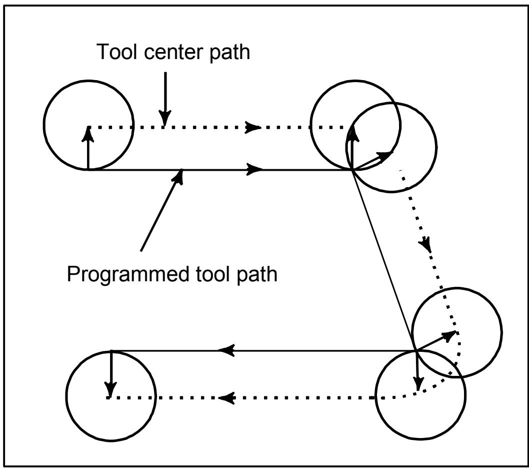


Fig. 6.11 (b) Normal direction control, left (G41.1)

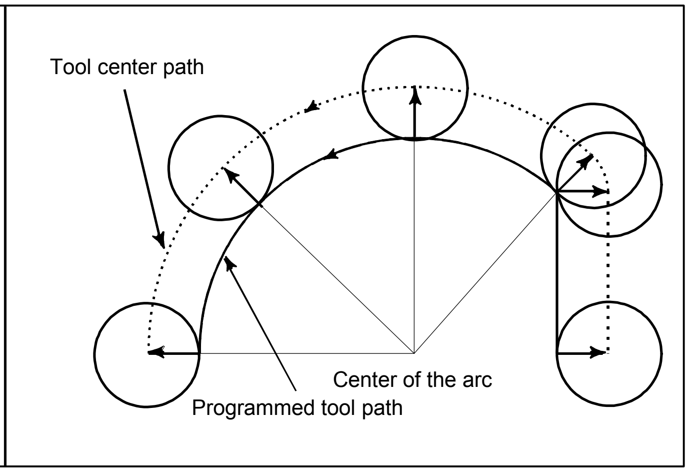


Fig. 6.11 (c) Normal direction control, right (G42.1)

Explanation

- Angle of the C axis

When viewed from the center of rotation around the C-axis, the angular displacement about the C-axis is determined as shown in Fig. 6.11 (d). The positive side of the X-axis is assumed to be 0, the positive side of the Y-axis is 90°, the negative side of the X-axis is 180°, and the negative side of the Y-axis is 270°.

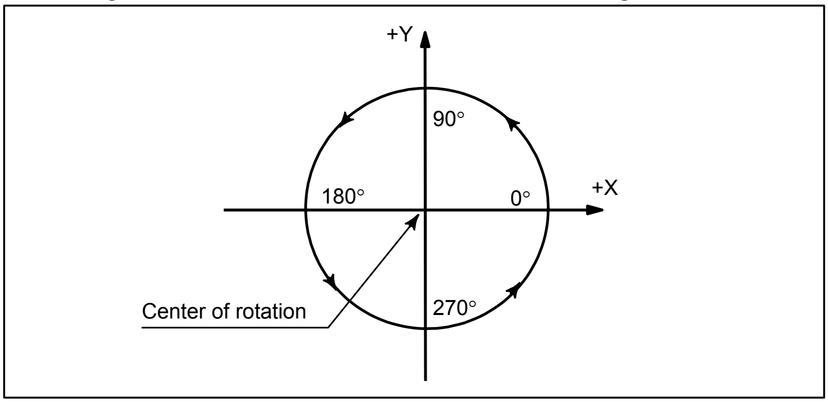


Fig. 6.11 (d) Angle of the C axis

- Normal direction control of the C axis

When the cancel mode is switched to the normal direction control mode, the C-axis becomes perpendicular to the tool path at the beginning of the block containing G41.1 or G42.1.

In the interface between blocks in the normal direction control mode, a command to move the tool is automatically inserted so that the C-axis becomes perpendicular to the tool path at the beginning of each block. The tool is first oriented so that the C-axis becomes perpendicular to the tool path specified by the move command, then it is moved along the X- and Y axes.

In the cutter compensation mode, the tool is oriented so that the C-axis becomes perpendicular to the tool path created after compensation.

In single-block operation, the tool is not stopped between a command for rotation of the tool and a command for movement along the X- and Y-axes. A single-block stop always occurs after the tool is moved along the X- and Y-axes.

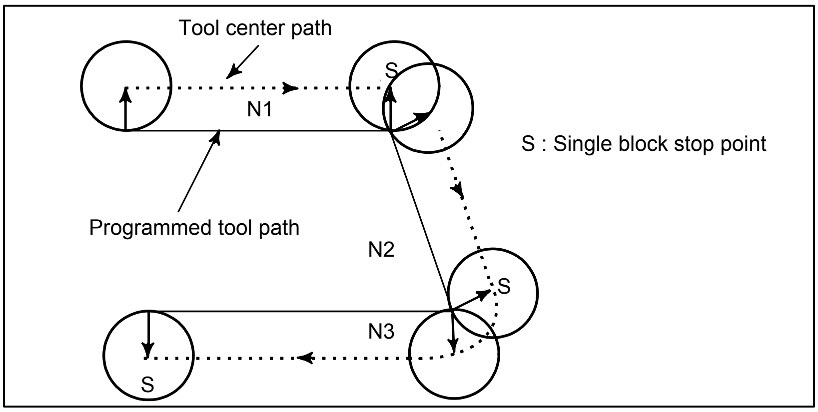


Fig. 6.11 (e) Point at which a single-block stop occurs in the normal direction control mode

Before circular interpolation is started, the C-axis is rotated so that the C-axis becomes normal to the arc at the start point. During circular interpolation, the tool is controlled so that the C-axis is always perpendicular to the tool path determined by circular interpolation.

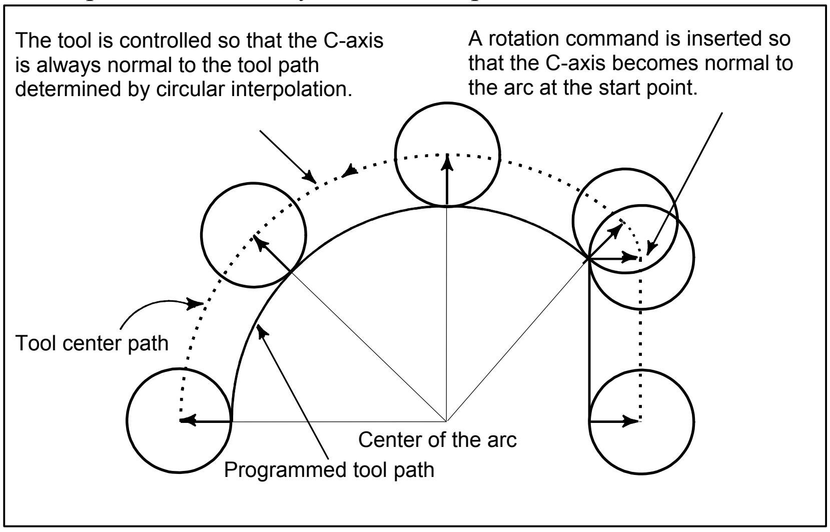


Fig. 6.11 (f) Normal direction control of the circular interpolation

> **NOTE**
> During normal direction control, the C axis always rotates through an angle less than 180 deg. I.e., it rotates in whichever direction provides the shorter route.

- C axis feedrate

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X-and Y-axes in rapid traverse (G00) mode, the rapid traverse feedrate is applied.
The feedrate of the C axis during circular interpolation is defined by the following formula.

$$F \times \frac{\text{Amount of movement of the C axis (deg)}}{\text{Length of arc (mm or inch)}} \quad \text{(deg/min)}$$

F : Feedrate (mm/min or inch/min) specified by the corresponding block of the arc
Amount of movement of the C axis :
The difference in angles at the beginning and the end of the block.

NOTE

If the feedrate of the C axis exceeds the maximum cutting speed of the C axis specified to parameter No. 1430, the feedrate of each of the other axes is clamped to keep the feedrate of the C axis below the maximum cutting speed of the C axis.

- Normal direction control axis

A C-axis to which normal-direction control is applied can be assigned to any axis with parameter No. 5480.

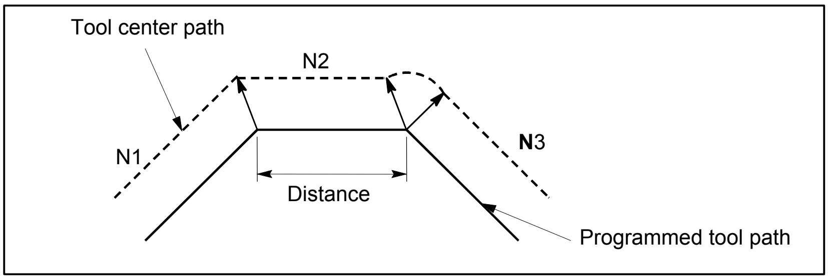
- Angle for which figure insertion is ignored

When the rotation angle to be inserted, calculated by normal-direction control, is smaller than the value set with parameter No. 5482, the corresponding rotation block is not inserted for the axis to which normal-direction control is applied. This ignored rotation angle is added to the next rotation angle to be inserted, the total angle being subject to the same check at the next block.
If an angle of 360 degrees or more is specified, the corresponding rotation block is not inserted.
If an angle of 180 degrees or more is specified in a block other than that for circular interpolation with a C-axis rotation angle of 180 degrees or more, the corresponding rotation block is not inserted.

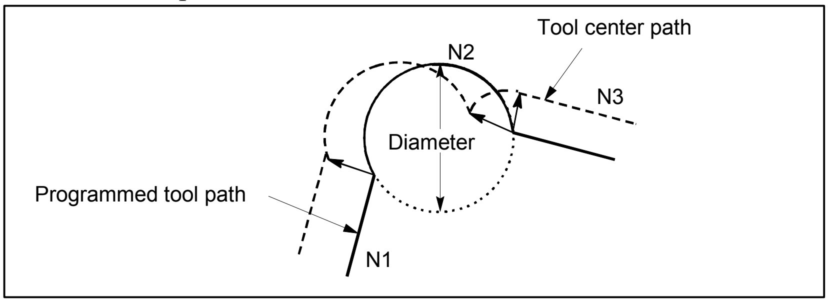
- Movement for which arc insertion is ignored

Specify the maximum distance for which machining is performed with the same normal direction as that of the preceding block.

- Linear movement
  When distance N2, shown below, is smaller than the set value, machining for block N2 is performed using the same direction as that for block N1.



- Circular movement
  When the diameter of block N2, shown below, is smaller than the set value, machining for block N2 is performed using the same normal direction as that for block N1. And control as compensation along the circular arc is not performed.



NOTE

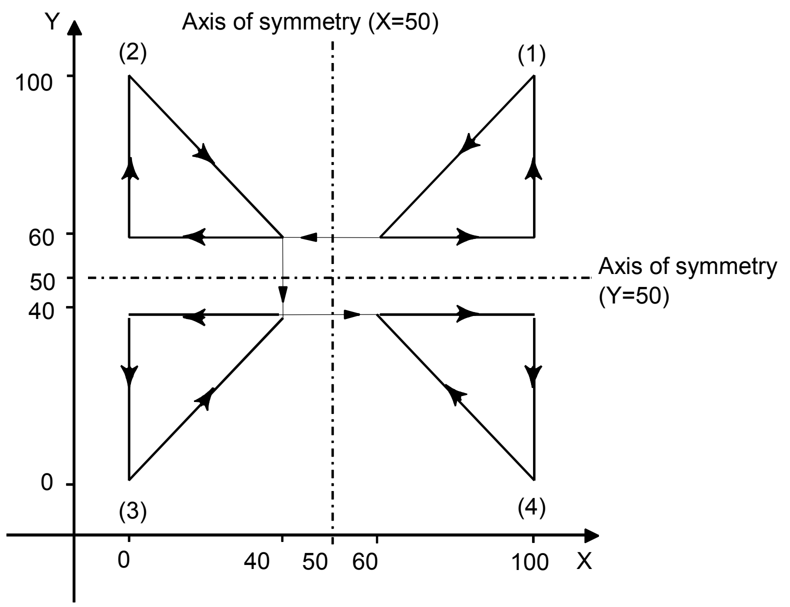
1 Do not specify any command to the C axis during normal direction control. Any command specified at this time is ignored.

NOTE

2 Before processing starts, it is necessary to correlate the workpiece coordinate of the C axis with the actual position of the C axis on the machine using the coordinate system setting (G92) or the like.
3 The helical cutting option is required to use this function. Helical cutting cannot be specified in the normal direction control mode.
4 Normal direction control cannot be performed by the G53 move command.
5 The C-axis must be a rotation axis.

6.12 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1)

A mirror image of a programmed command can be produced with respect to a programmed axis of symmetry (Fig. 6.12 (a)).



(1) Original image of a programmed command
(2) Image symmetrical about a line parallel to the Y-axis and crossing the X-axis at 50
(3) Image symmetrical about point (50, 50)
(4) Image symmetrical about a line parallel to the X-axis and crossing the Y-axis at 50

Fig. 6.12 (a) Programmable mirror image

Format

G51.1 IP_ ;		Setting a programmable image
:		A mirror image of a command specified in
:		these blocks is produced with respect to
:		the axis of symmetry specified by G51.1
:		IP_;.
G50.1 IP_ ;		Canceling a programmable mirror image
IP_ :	Point (position) and axis of symmetry for producing a mirror image when specified with G51.1. Axis of symmetry for producing a mirror image when specified with G50.1. Point of symmetry is not specified.	

Explanation

- Mirror image by setting

If the programmable mirror image function is specified when the command for producing a mirror image is also selected by a CNC external switch or CNC setting (see III-4.8 in Operator's Manual (Common to T/M series.)), the programmable mirror image function is executed first.

- Mirror image on a single axis in a specified plane

Applying a mirror image to one of the axes on a specified plane changes the following commands as follows :

Command	Explanation
Circular command	G02 and G03 are interchanged.
Cutter compensation	G41 and G42 are interchanged.
Coordinate system rotation	CW and CCW (directions of rotation) are interchanged.

Limitation

- Scaling and coordinate system rotation

Processing proceeds from program mirror image to scaling and coordinate system rotation in the stated order. The commands should be specified in this order, and, for cancellation, in the reverse order. Do not specify G50.1 or G51.1 during scaling or coordinate system rotation mode.

- Commands related to reference position return and coordinate system

In programmable mirror image 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 the programmable mirror image mode. If it is specified without canceling the mode, the alarm PS0412 is issued.

7 MEMORY OPERATION USING Series 10/11 PROGRAM FORMAT

Overview

Memory operation of the program registered in Series 10/11 program format is possible by setting the setting parameter FCV (bit 1 of parameter No. 0001) to 1.

Explanation

Data formats for tool radius compensation, subprogram call, and canned cycles are different between the Series 0*i* and Series 10/11. The Series 10/11 program formats can be processed for memory operation.
Other data formats must comply with the Series 0*i*. When a value out of the specified range for the Series 0*i* is registered, an alarm occurs. Functions not available in the Series 0*i* cannot be used for memory operation.

- Address for the tool radius compensation offset number

Offset numbers are specified by address D in the Series 10/11.
When an offset number is specified by address D, the modal value specified by address H is replaced with the offset number specified by address D.

- Subprogram call

If a subprogram number of more than four digits is specified, the four low-order digits are regarded as the subprogram number.
If no repeat count is specified, 1 is assumed.

Table 7 (a) Subprogram call program format

CNC	Program format
Series 10/11	**M98 P**○○○○ **L**○○○○ **;** P : Subprogram number L : Repetition count (1 to 9999)
Series 0*i*	**M98 P**○○○ □□□□ **;** Repetition count (1 to 9999) Subprogram number

- Address for the canned cycle repetition count for drilling

The Series 10/11 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 10/11	L
Series 0*i*	K

8 AXIS CONTROL FUNCTIONS

Chapter 8, "AXIS CONTROL FUNCTIONS", consists of the following sections:

8.1 ELECTRONIC GEAR BOX (G80, G81 (G80.4, G81.4))

8.1.1 Electronic Gear Box

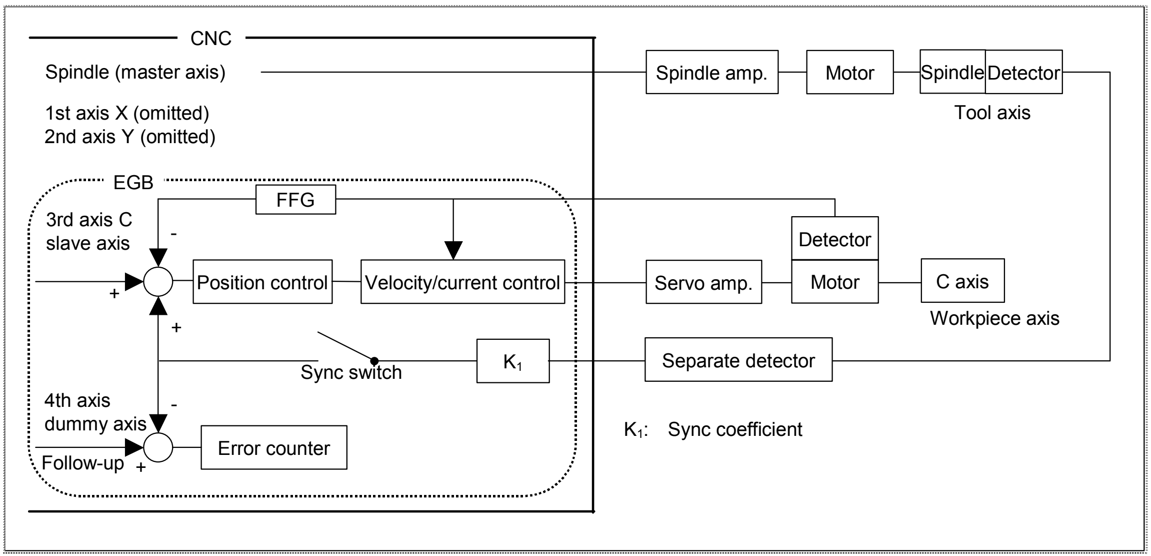
Overview

This function synchronizes the revolutions of the workpiece axis connected to the servo motor with the revolutions of the tool axis (grinding stone/hob) connected to the spindle motor so as to machine (grind/cut) gears as in the hobbing machine function. The synchronization ratio can be specified by a program. Synchronization of the tool axis and the workpiece axis by this function is directly controlled by a digital servo, so the workpiece axis can track changes in the speed of the tool axis with no error, thereby achieving high-precision machining of gears. In the following descriptions, the electric gear box is called the EGB. For details on the conditions that need be met to set the workpiece axis and tool axis, refer to the manual provided by the machine tool builder.

NOTE
Electronic gear box is optional function.

- Example of controlled axis configuration

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis.)



Format

	Parameter EFX (No.7731#0)=0	Parameter EFX (No.7731#0)=1
Start of synchronization	**G81 T__ (L__) (Q__ P__) ;**	**G81.4 T__ (L__) (Q__ P__) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**

T : Number of teeth (Specifiable range: 1 to 1000)
L : Number of hob threads (Specifiable range: -200 to +200)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No.7701.
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)

* When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.
A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

- Synchronous control

(1) Start of synchronization
If G81 is issued so that the machine enters synchronization mode, the synch switch of the EGB function is closed, and the synchronization of the tool and workpiece axes is started. During synchronization, the rotation about the tool and workpiece axes is controlled so that the relationship between T (number of teeth) and L (number of hob threads) is maintained. During synchronization, the synchronization relationship is maintained regardless of whether the operation is automatic or manual.
Specify P and Q to use helical gear compensation.

If only either P or Q is issued, alarm PS1594 is generated.
If, during synchronization, G81 is issued again without synchronization cancellation, alarm PS1595 is generated if ECN, bit 3 of parameter No. 7731, is 0. If ECN, bit 3 of parameter No. 7731, is 1, helical gear compensation is conducted with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued, and if T and L commands are not issued and only P and Q commands are issued, helical gear compensation is conducted with the synchronization coefficient kept intact. This allows consecutive fabrication of helical gears and super gears.

(2) Start of tool axis rotation
When the rotation of the tool axis starts, the rotation of the workpiece axis starts so that the synchronous relationship specified in the G81 block can be maintained.
The rotation direction of the workpiece axis depends on the rotation direction of the tool axis. That is, when the rotation direction of the tool axis is positive, the rotation direction of the workpiece axis is also positive; when the rotation direction of the tool axis is negative, the rotation direction of the workpiece axis is also negative. However, by specifying a negative value for L, the rotation direction of the workpiece axis can be made opposite to the rotation direction of the tool axis.
During synchronization, the machine coordinates of the workpiece axis and EGB axis are updated as synchronous motion proceeds. On the other hand, a synchronous move command has no effect on the absolute and relative coordinates.

(3) Termination of tool axis rotation
In synchronism with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the command below after the spindle stops, synchronization is canceled, and the EGB synchronization switch is opened.

(4) Cancellation of synchronization
When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization. Subsequently, absolute commands for the workpiece axis will be enabled.
For a rotation axis, the amount of travel during synchronization, as rounded to 360-degree units is added to the absolute coordinate.
In the G80 block, only O and N addresses can be specified.
By setting HBR, bit 0 of parameter No. 7700, to 0, it is possible to cancel synchronization with a reset. If the manual absolute switch is ON, the absolute coordinates are updated.

Synchronization is automatically canceled under the following conditions:
<1> An emergency stop is applied.
<2> A servo alarm is generated.
<3> Alarm PW0000 (POWER MUST BE OFF) is generated.
<4> An IO alarm is generated.

⚠ CAUTION

1. Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2. Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3. During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronous mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis.
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6 In synchronous mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
8 If TDP, bit 0 of parameter No. 7702, is 1, the permissible range of T is 0.1 to 100 (1/10 of the specified value).
9 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No.7701, is 0; if bit 3 (LZR) of parameter No.7701, is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
10 Feed per revolution is performed on the feedback pulses on the spindle. By setting ERV, bit 0 of parameter No. 7703, to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
11 Actual cutting feedrate display does not take synchronization pulses into consideration.
12 In EGB synchronization mode, AI contour control mode is temporarily canceled.

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

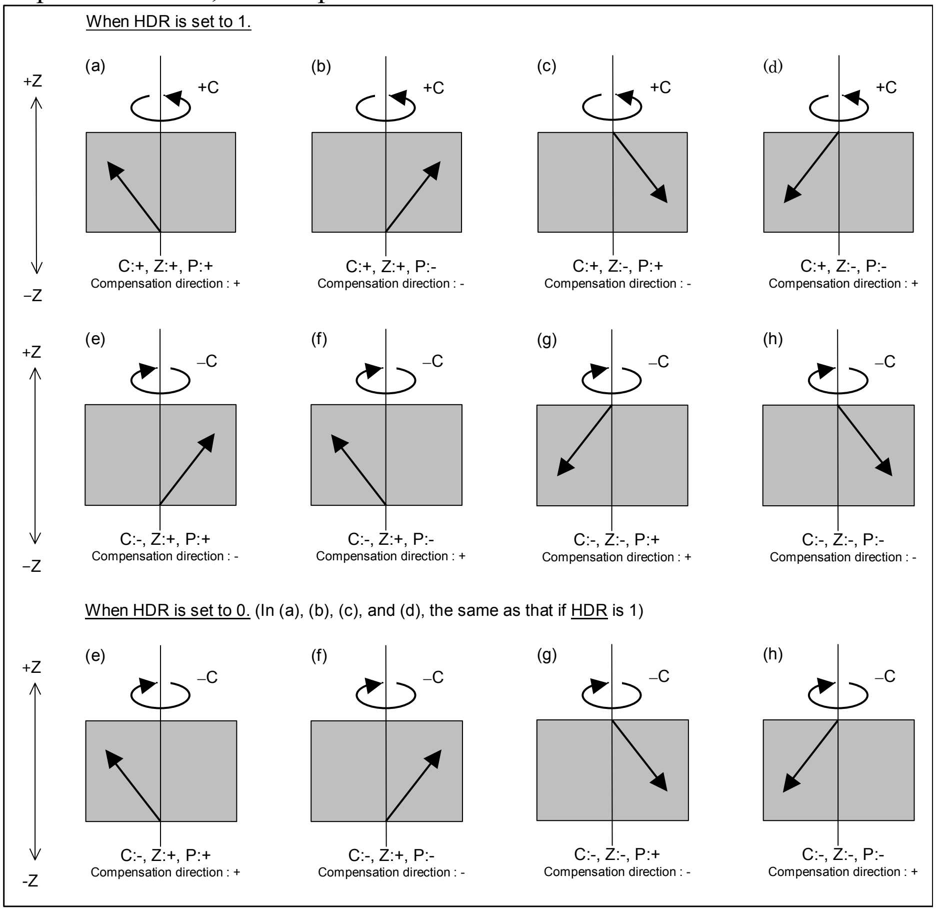
where

Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81
P : Signed gear helix angle (deg)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch (inch-1)
Use P, T, and Q specified in the G81 block.

In helical compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical compensation.

- Direction of helical gear compensation

The direction depends on HDR, bit 2 of parameter No. 7700.



- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (Kn/Kd) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_n}{K_d} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where

L : Number of hob threads

T : Number of teeth

α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)

β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

Kn / Kd is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:

$-2147483648 \le K_n \le 2147483647$

$1 \le K_d \le 65535$

When this restriction is not satisfied, the alarm (PS1596) is issued when G81 is specified.

Example

O1000 ;

N0010 M19 ;	Tool axis orientation
N0020 G28 G91 C0 ;	Reference position return on the workpiece axis
N0030 G81 T20 L1 ;	Synchronous start on tool and workpiece axes
	(Rotation about the workpiece axis by 18° per rotation about the tool axis)
N0040 S300 M03 ;	Rotation about the tool axis at 300min^{-1}

```
N0050 G01 X_  F_ ;  Movement along the X-axis (cut)
N0060 G01 Z_  F_ ;  Movement along the Z-axis (machining)
                 ;  If necessary, axis commands such as C, X, and Z commands are allowed.
N0100 G01 X_  F_ ;  Movement along the X-axis (escape)
N0110 M05 ;         Stop on the tool axis
N0120 G80 ;         Synchronous cancellation on tool and workpiece axes
N0130 M30 ;
```

- Retract function

(1) Retract function with an external signal

When the retract switch on the machine operator's panel is turned on, retraction is performed with the retract amount set in parameter No. 7741 and the feedrate set in parameter No. 7740.
No movement is performed along an axis for which 0 is set as the retract amount.
For the retract switch, refer to the relevant manual provided by the machine tool builder.

(2) Retract function with an alarm

If, during EGB synchronization or automatic operation, a CNC alarm is issued, retraction is performed with the retract amount set in parameter No. 7741 and the speed set in parameter No. 7740.
This can prevent the tool and the object being machined from damage if a servo alarm is generated.
No movement is performed along an axis for which 0 is set as the retract amount.
For the retract switch, refer to the relevant manual provided by the machine tool builder.

Conditions under the retract function with an alarm

The conditions under which the retract function with a servo or spindle alarm can be changed using the settings of ARE, bit 1 of parameter No. 7703, and bit 2 (ARO) of parameter No. 7703.
The table below lists parameter settings and corresponding conditions.

ARE	ARO	Condition
1	0	EGB synchronization is in progress.
1	1	Both EGB synchronization and automatic operation are in progress.
0	0	Either EGB synchronization or automatic operation is in progress.
0	1	

⚠ CAUTION

1 Retraction is performed at the speed specified in parameter No. 7740.
2 Feed hold is not effective to movement during retraction.
3 Feedrate override is not effective to movement during retraction.

NOTE

1 During a retract operation, an interlock is effective to the retract axis.
2 During a retract operation, a machine lock is effective to the retract axis.
3 The retraction direction depends on the movement direction of the machine, regardless of whether an mirror image (signal and setting) is enabled or disabled. (No mirror image can be applied to the updating of absolute coordinates.)
4 If retraction is performed during automatic operation, automatic operation is halted simultaneously with a retract operation, but it is at the end of the retract operation that the operation state switches to the automatic operation halt state.
5 It is not possible to perform automatic operation during retraction.
6 The acceleration/deceleration of a retract operation is in the acceleration/deceleration state at the start of retraction.

NOTE

7 Retract movement is performed with non-linear type positioning.

8 If, during a retract operation, a reset or an emergency stop is made, the operation is interrupted.

9 If, during a retract operation on multiple axes, an OT alarm or a malfunction prevention alarm is issued for a retract axis, the operation stops only on the axis for which the alarm is issued if bit 4 (RTS) of parameter No. 7731 is 0. If bit 4 (RTS) of parameter No. 7731 is 1, the retract operation is interrupted on all axes. If a servo alarm or non-axis malfunction prevention alarm is issued, the retract operation is interrupted on all axes regardless of the setting of bit 4 (RTS) of parameter No. 7731.

10 To enable the retract function with an alarm, bit 3 (ART) of parameter No.7702, must be set.

11 The retract function with an alarm does not perform a retract operation on the retract axis if an overtravel alarm or a servo alarm is generated on the retract axis.

12 If a new alarm is issued during retraction with the retract function with an alarm, a retract operation is not performed.

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

Press function key to display or set tool compensation values and other data.
This section explains how to display and specify the following:

1. Tool compensation value
2. Tool length measurement

Refer to the Operator's Manual (Common to Lathe System/Machining Center System) (B-64304EN) for explanations about how to display or specify the other types of data.

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.
There are two tool offset memory types, A and C.

Procedure for setting and displaying the tool compensation value

Procedure

1 Press function key .

2 Press chapter selection soft key [OFFSET] or press function key several times until the tool compensation screen is displayed.
The screen varies according to the type of tool compensation memory.

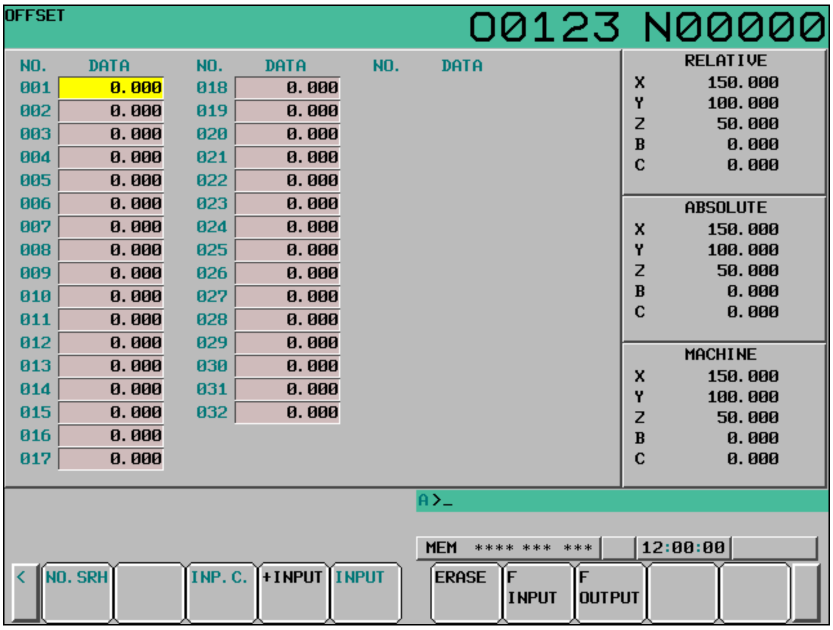


Fig. 1.1.1 (a) Tool compensation memory A (10.4-inch)

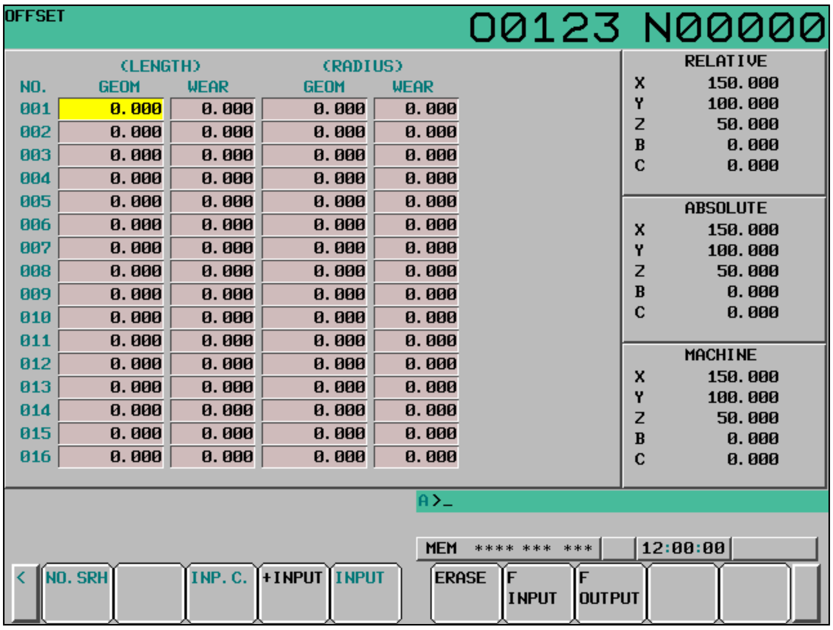


Fig. 1.1.1 (b) Tool compensation memory C (10.4-inch)

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].

4 To set a compensation value, enter a value and press soft key [INPUT].
To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [+INPUT]. Or, enter a new value and press 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 and C, which are classified as follows:

Tool compensation memory A

D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated the same.

Tool compensation memory C

D codes and H codes are treated differently. Tool geometry compensation and tool wear compensation are treated differently.

Bit 6 (NGW) of parameter No. 8136 can be used to specify whether to use tool offset memory C ("0" for specifying to use it and "1" for specifying not to use it). If tool offset memory C is not used, tool offset memory A is used.

- Number of tool offset values

Bit 5 (NDO) of parameter No. 8136 can be used to specify whether to use 400 tool offset values ("0" for specifying to use 400 tool offset values and "1" for specifying not to use them). If the number of tool offset values to be used is not 400, the number of tool offset values to be used is 32.

- 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).

In this case, it is possible to prohibit any range of tool offset values from being entered from the MDI by setting the start tool offset value number in parameter No. 3294 and the quantity of offset values counted from the beginning of the range in parameter No.3295.

If an attempt is made to enter tool offset values including those prohibited, the following occur:

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.

Bit 7 (NTL) of parameter No. 8136 can be used to specify whether to use tool length measurement ("0" for specifying to use it and "1" for specifying not to use it).

Procedure for tool length measurement

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.

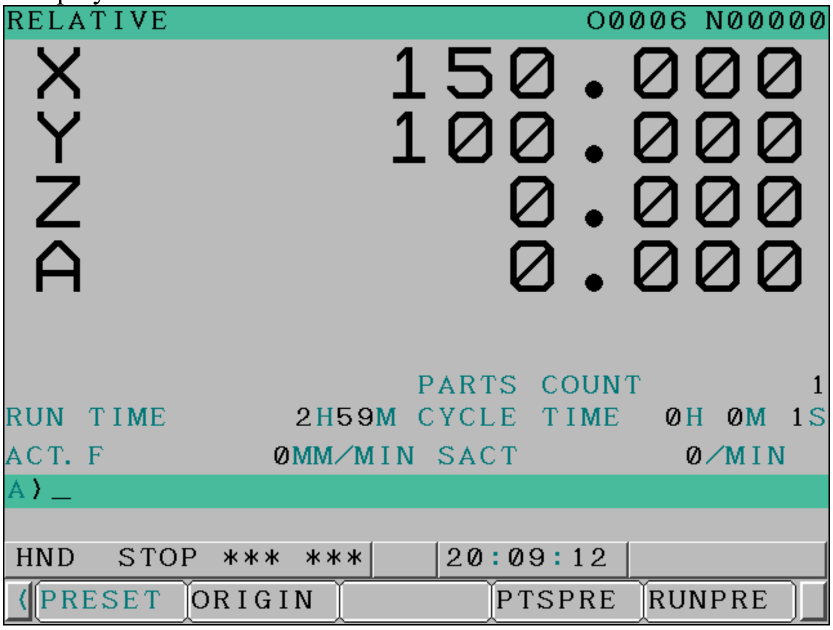


Fig. 1.1.2 (a) Current position display screen (8.4-inch)

3 Reset the relative coordinate for the Z-axis to 0.

4 Press function key several times until the tool compensation screen is displayed.

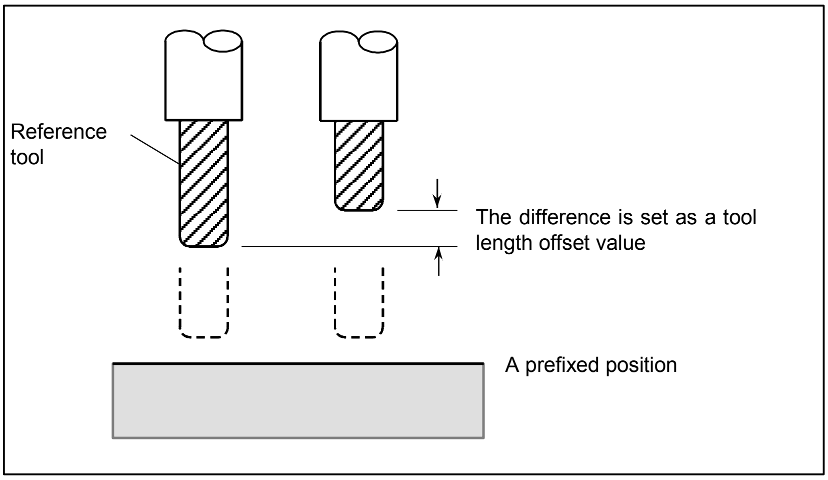
5 Use manual operation to move the tool to be measured until it touches the same specified position. The difference between the length of the reference tool and the tool to be measured is displayed in the relative coordinates on the screen.

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 Z. If either X or Y key is depressed instead of Z key, the X or Y axis relative coordinate value is input as an tool length compensation value.

8 Press the soft key [INP.C.]. The Z axis relative coordinate value is input and displayed as an tool length compensation value.

Fig. 1.1.2 (b) Tool compensation memory (8.4-inch)



1.1.3 Machining Level Selection

1.1.3.1 Smoothing level selection

An intermediate smoothing level between the parameters for smoothing level 1 and the parameters for smoothing level 10 set on the machining parameter tuning screen (smoothing) can be selected. As shown in the Fig. 1.1.3.1 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

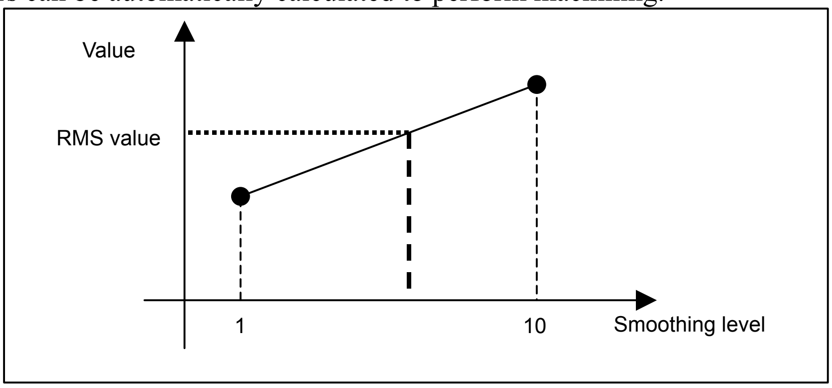
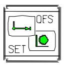


Fig. 1.1.3.1 (a) Image of “level”

Procedure for smoothing level selection

1 Select the MDI mode.

2 Press function key .

3 Press soft key [PRECI LEVEL].

4 Press soft key [SMOOTH LEVEL].

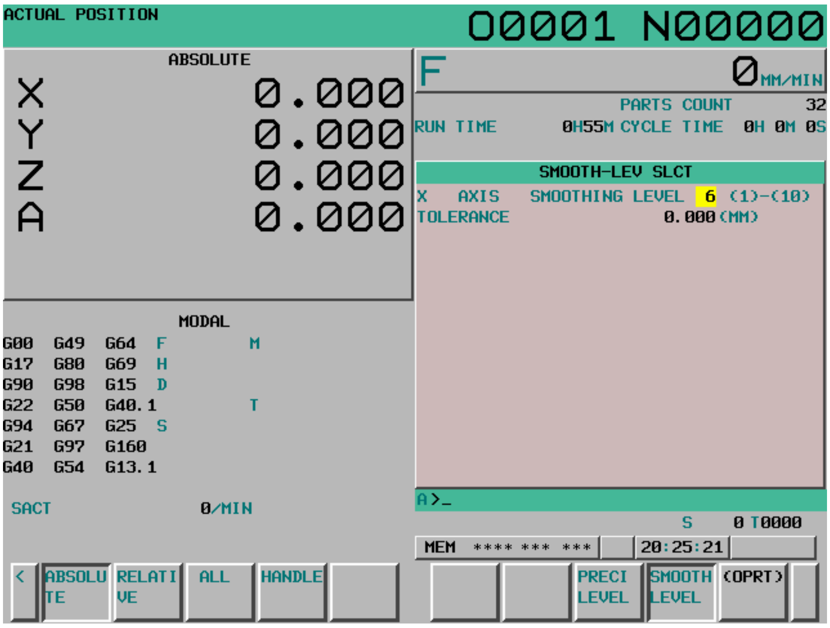
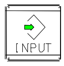


Fig. 1.1.3.1 (b) Smoothing level selection screen

5 To change the smoothing level, key in a desired smoothing level (1 to 10), then press the 

key on the MDI panel.

6 When the smoothing level is changed, a RMS value is obtained from the smoothing level 1 parameter set and smoothing level 10 parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

7 If there is an axis in addition to the currently displayed axes, press page key 

or 

several times to display the screen for the axis.

1.1.3.2 Precision level selection

For details of precision level selection, See Subsection III.12.3.10, “Precision Level Selection”.

Manual name	Item name
OPERATOR’S MANUAL (B-64304EN)	III.12.3.10 Precision Level Selection

1.1.4 Machining Quality Level Selection

Machining quality level selection allows the precision level and smoothing level to be adjusted intuitively and easily.
To display the machining quality level selection screen, set bit 6 (QLS) of parameter No. 11350 to 1.

> **NOTE**
> The machining quality level selection screen cannot be displayed on the 8.4-inch display unit.
> On these display units, only the machining level selection screen can be displayed.

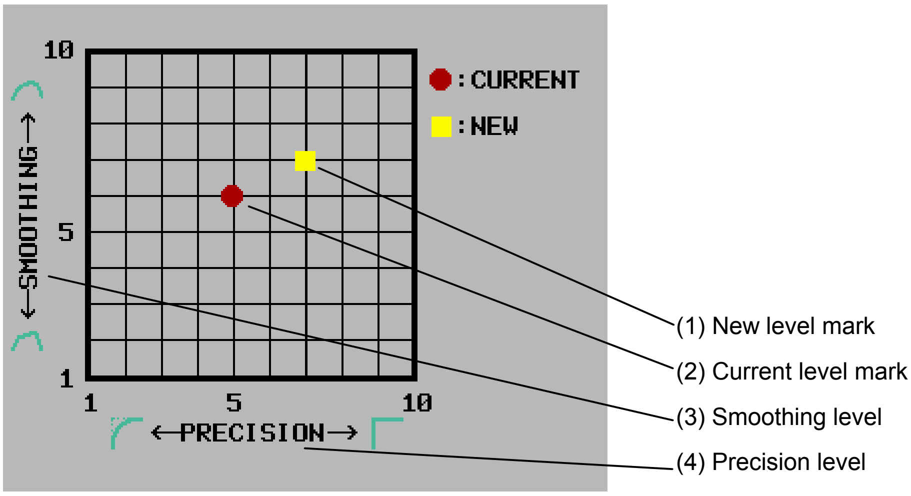


Fig. 1.1.4 (a) Quality level selection

(1) New level mark
Yellow square: Indicates the setting to be selected. (Cursor position)

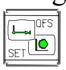
(2) Current level mark
Red circle: Indicates the current setting.

(3) Smoothing level
Vertical axis: Indicates the smoothing level (1 to 10).

(4) Precision level
Horizontal axis: Indicates the precision level (1 to 10).

Procedure for machining quality level selection screen

1 Enable parameter writing.

2 Press function key .

3 Press soft key [QUALITY SELECT].

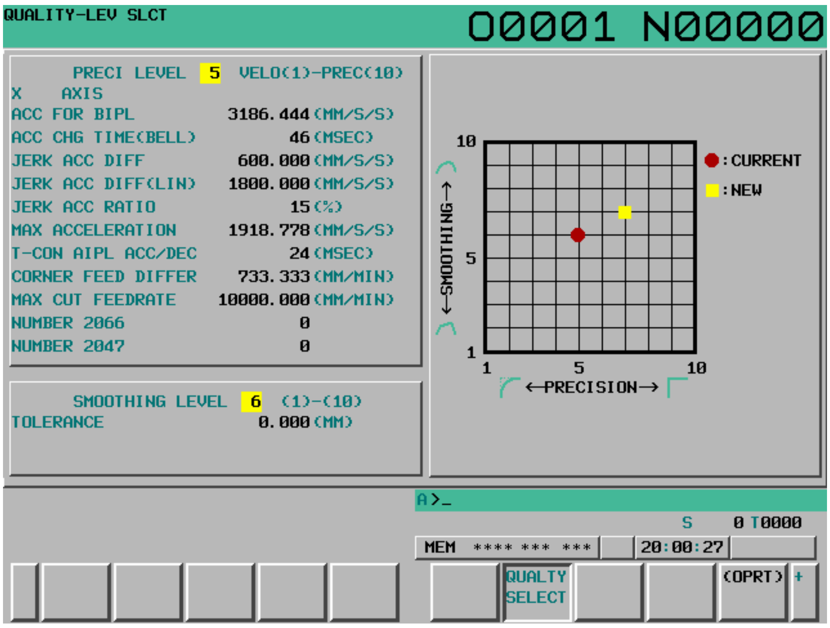
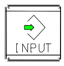


Fig. 1.1.4 (b) Machining quality level selection screen

4 Use cursor keys to move the new level mark and select the level.
(The new level mark moves.)

5 Press soft key [APPLY] or MDI key  to set the level.
 (The current level mark moves to the position of the new level mark.)
 Whether to enable or disable MDI key operation can be switched by setting the relevant parameter.
6 The set precision level and smoothing level are reflected in each setting on the PRECI LEVEL and SMOOTHING LEVEL screens displayed on the left side of the screen.
7 When the precision level or smoothing level is changed, an RMS value is obtained using the parameter settings for precision levels 1 and 10 and smoothing levels 1 and 10 and effective parameters are changed. For the changed parameters, see the description of the machining parameter tuning screen. When there is an axis other than the currently displayed axis, press a page key several times to display the screen for the desired axis.

2 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.
This chapter explains the following types of automatic operation:

2.1 RETRACE

Overview

The tool can retrace the path along which the tool has moved so far (reverse execution). Furthermore, the tool can move along the retraced path in the forward direction (forward reexecution). After forward reexecution is performed until the tool reaches the position at which reverse execution started, machining is continued as programmed.

Procedure

- Forward execution → reverse execution

To perform forward execution of a program, set the “REVERSE” switch on the machine operator’s panel to off, then perform a cycle start operation. If the “REVERSE” switch on the machine operator’s panel is set to on, reverse execution or the end of reverse execution results.
To perform reverse execution of a program, use one of the following three methods:

1) Set the “REVERSE” switch on the machine operator’s panel to on during forward execution of a block.
2) Perform a single block stop operation during forward execution, then set the “REVERSE” switch on the machine operator’s panel to on.
3) Perform a feed hold stop operation during forward execution, then set the “REVERSE” switch on the machine operator’s panel to on.

When method 1) is used, reverse execution starts after the end of the block being executed (after execution up to the single block stop position). Reverse execution does not start as soon as the “REVERSE” switch on the machine operator’s panel is set to on.

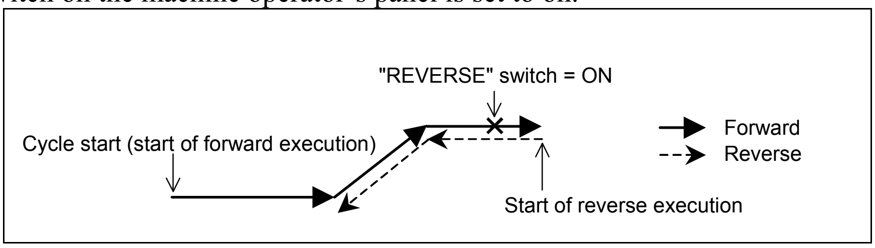


Fig. 2.1 (a)

When method 2) is used, performing a cycle start operation starts reverse execution from the position at which a single block stop takes place.

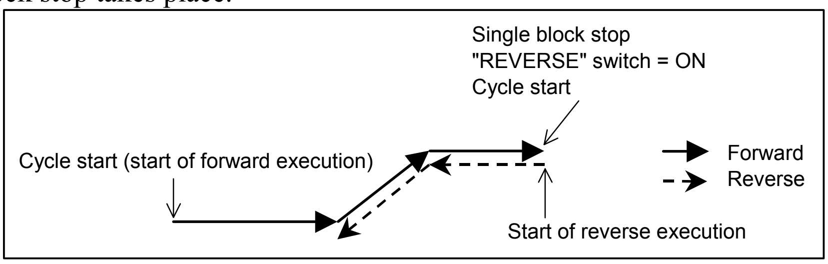


Fig. 2.1 (b)

When method 3) is used, performing a cycle start operation starts reverse execution from the position at which a feed hold stop takes place.

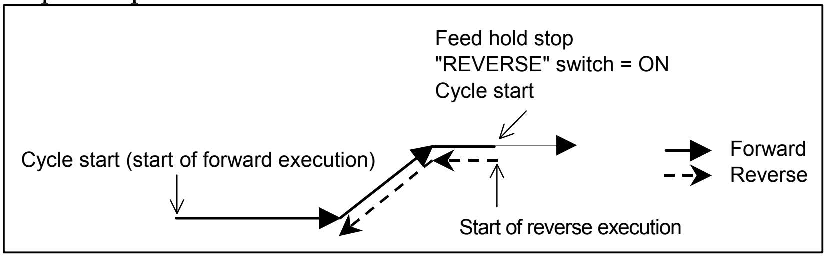


Fig. 2.1 (c)

- Reverse execution → forward reexecution

To perform forward reexecution of a program, use one of the following three methods:

1) Set the "REVERSE" switch on the machine operator's panel to off during reverse execution of a block.
2) Set the "REVERSE" switch on the machine operator's panel to off after a single block stop takes place during reverse execution.
3) Set the "REVERSE" switch on the machine operator's panel to off after a feed hold stop takes place during reverse execution.

When method 1) is used, forward reexecution starts after the block being executed ends (after execution up to the position at which a single block stop takes place). Forward reexecution does not start as soon as the "REVERSE" switch on the machine operator's panel is set to off.

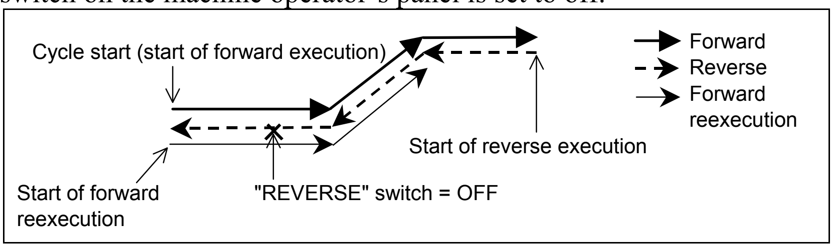


Fig. 2.1 (d)

When method 2) is used, performing a cycle start operation starts forward reexecution from the position at which a single block stop takes place.

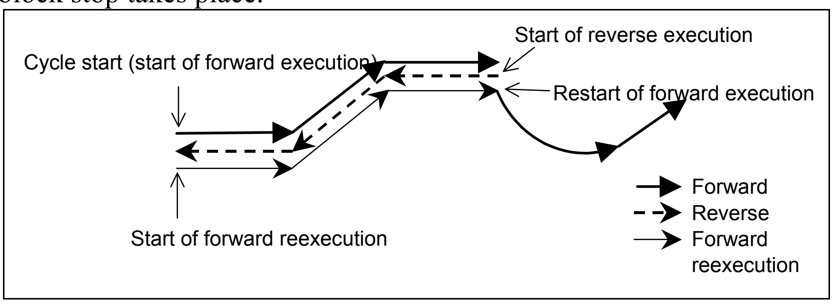


Fig. 2.1 (e)

When method 3) is used, performing a cycle start operation starts forward reexecution from the position at which a feed hold stop takes place.

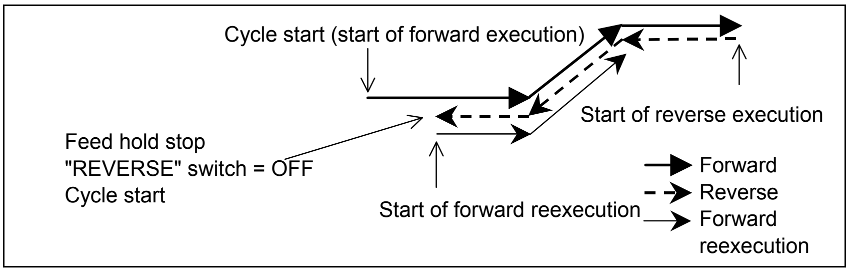


Fig. 2.1 (f)

- Reverse execution → end of reverse execution → forward reexecution

When a block to be executed is no longer present during reverse execution (when reverse execution has been performed up to the block where forward execution started, or when forward execution has not yet been performed), the reverse execution end state is entered, and operation stops.
Even when a cycle start operation is performed while the "REVERSE" switch on the machine operator's panel is held on, operation is not performed, and the reverse execution end state is maintained. Forward reexecution (or forward execution) is started by setting the "REVERSE" switch on the machine operator's panel to off then performing a cycle start operation.

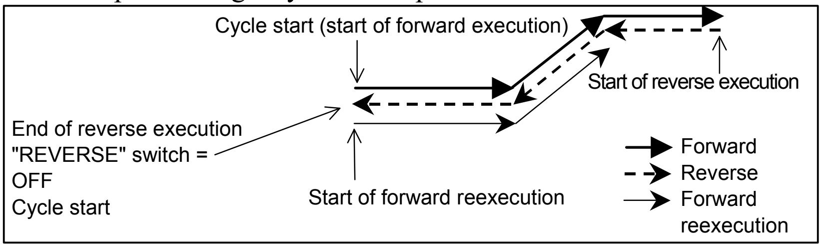


Fig. 2.1 (g)

- Forward reexecution → forward execution

After forward reexecution is performed up to the block at which reverse execution started, forward execution starts automatically, and commands are read from the program again and executed. No particular operation is required.

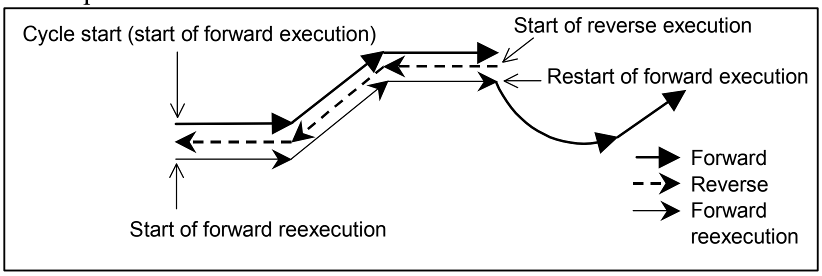


Fig. 2.1 (h)

If reverse execution was performed after feed hold stop, forward reexecution ends when the feed hold stop position is reached, then forward execution is performed. Also if single block operation was performed, forward reexecution ends at the single block stop position.

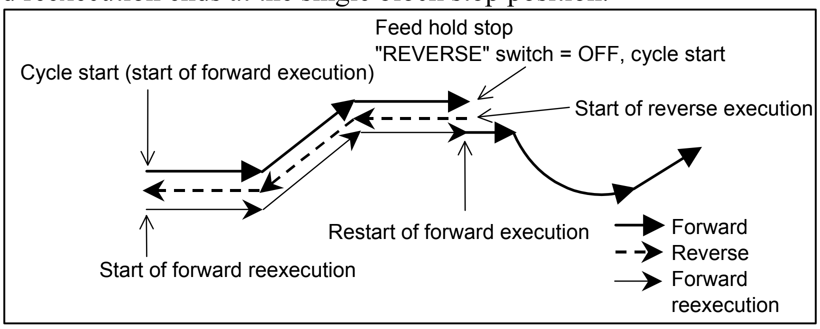


Fig. 2.1 (i)

Explanation

- Reverse execution and forward execution

Usually in automatic operation, a program is executed in the programmed order. This is called forward execution. This function allows a program executed by forward execution to be executed in the reverse direction. This is called reverse execution. Reverse execution allows the tool to retrace the path along which the tool has moved during forward execution.
Reverse execution of a program can be performed only for blocks that have been executed by forward execution.
Furthermore, in single block operation, reverse execution can also be performed on a block-by-block basis .

- Forward reexecution

Blocks that have been executed by reverse execution can be reexecuted in the forward direction up to the block from which reverse execution started. This is called forward reexecution. Forward reexecution allows the tool to retrace the same tool path as in forward execution until the position at which reverse execution started is reached.
After the block from which reverse execution started is reached, the program is executed again in the programmed order (forward execution).
Furthermore, in single block operation, forward reexecution can also be performed on a block-by-block basis.

- End of reverse execution

When a block to be executed is no longer present during reverse execution (when stored blocks have all been executed during reverse execution, or when forward execution has not yet been performed), operation stops. This is called the end of reverse execution.

- Status indication

During reverse execution, characters "RVRS" blink on the screen. During forward reexecution, characters "RTRY" blink to indicate that forward reexecution is in progress. The "RTRY" indication is kept blinking until the block at which reverse execution started is reached and normal operation starts (until forward execution is restarted).
When a block to be executed is no longer present during reverse execution, or if an attempt is made to perform reverse execution for a block that cannot be executed by reverse execution, characters "RVED" blink, notifying the user that reverse execution can no longer be performed.

- Number of blocks that can be executed by reverse execution

Up to about 100 blocks can be executed by reverse execution. Depending on the specified program, the maximum number of executable blocks may decrease.

- Reset

A reset operation (the RESET key on the MDI unit, the external reset signal, or the reset & rewind signal) clears the blocks stored for reverse execution.

- Feedrate

A feedrate to be applied during reverse execution can be specified in parameter No. 1414. If this parameter is set to 0, the feedrate in reverse execution is assumed to be the same as that in forward execution. Rapid traverse, however, is performed always at the rapid traverse rate, regardless of the setting of this parameter.
The feedrate in forward reexecution is always the same as that in forward execution.
In reverse execution or forward reexecution, feedrate override, rapid traverse override, and dry run are allowed.

- Start of reverse execution or forward reexecution after the end of a block

In a block for rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), dwelling (G04), skip cutting (G31), or an auxiliary function in an automatic operation mode (memory operation, part program operation, or MDI operation), reverse execution or forward reexecution can be started. However, reverse execution and forward reexecution do not start as soon as the reverse execution signal status is changed. When the block has ended, that is, after a movement, dwelling, or an auxiliary function is completed, reverse execution or forward reexecution starts.

- Start of reverse execution or forward reexecution after feed hold stop

When a feed hold stop operation is performed during execution of rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), or skip cutting (G31), then the reverse execution signal status is changed and operation is restarted, reverse execution or forward reexecution can be started immediately from the stop position. This cannot be performed when dwelling (G04) or an auxiliary function is being executed.

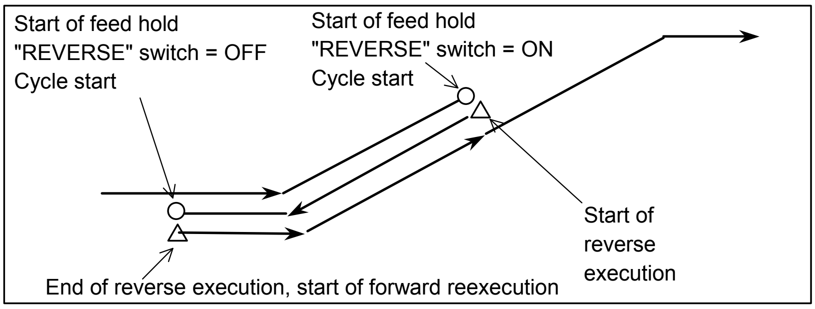


Fig. 2.1 (j)

When reverse execution is started after feed hold stop operation, the part from the start point of that block to the feed hold stop position is stored as one block. Therefore, when forward reexecution is performed with the single block switch set to 1, a single block stop takes place as soon as the position at which reverse execution started is reached.

- Start of reverse execution or forward reexecution after single block stop

After a single block stop takes place, reverse execution or forward reexecution can be started immediately when the reverse execution signal status is changed and restart operation is performed.

Limitation

- Blocks that cannot be executed by reverse execution

In the modes listed below, reverse execution cannot be performed.
When one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.

- Cylindrical interpolation (G07.1,G107)
- Polar coordinate command (G16)
- Thread cutting (G33)
- Single direction positioning (G60)
- Tapping mode (G63)
- Tapping cycle (G84,G74)
- Rigid tapping cycle (G84,G74,G84.2,G84.3)
- Fine boring cycle (G76)
- Back boring cycle (G87)

It is impossible to perform reverse execution for blocks specifying the commands listed below. If one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.

Some of these commands turn a mode on and off. It is possible to start reverse execution and perform forward reexecution in a mode set by such a command. However, if a block that turns the mode on or off is reached during reverse execution, the reverse execution ends at that block, and "RVED" is displayed.

- Functions related AI contour control (G05.1)
- HRV3 on/off (G05.4)
- Inch/metric conversion (G20, G21)
- Stored stroke check on/off (G22, G23)
- Functions related reference position return (G27, G28, G29, G30)
- Index table indexing
- Cs contouring control

- Manual intervention

To execute a program in the reverse direction after a feed hold stop or single block stop, when manual intervention is performed after the stop, make a return to the original position and then turn on the reverse signal. Movement by manual intervention is ignored during reverse execution and forward reexecution.
If manual intervention is performed during reverse execution or forward reexecution, the amount of manual intervention is added to the coordinate system at a restart after a stop due to a feed hold or single block during forward execution after the end of forward reexecution. Whether to add the amount of manual intervention follows the manual absolute switch.

- Single block stop position

A block that is internally generated by the control unit is also treated as one block during reverse execution.

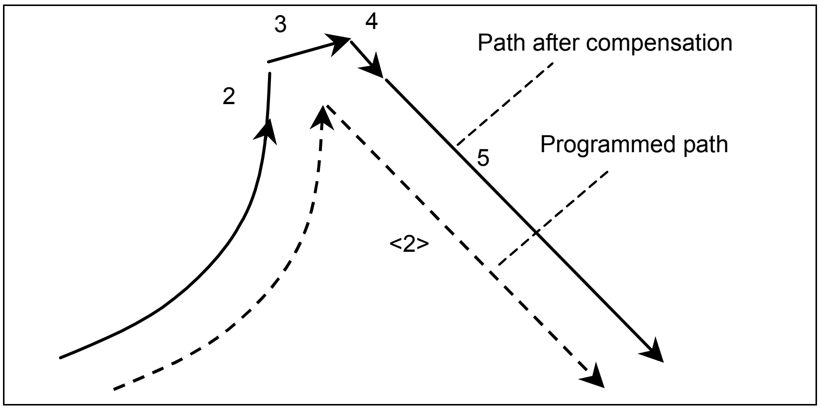


Fig. 2.1 (k) Path when cutter compensation is applied

In the above example, the program specifies two blocks, but in actual operation, move commands for five blocks are generated.
In such a case, positions at which a single block stop takes place may differ between forward execution and reverse execution.

- Positioning (G00)
When non-linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 0), the tool path in reverse execution and that in forward execution do not match. The tool path in forward reexecution is the same as that in forward execution.
When linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 1), the tool path in reverse execution is the same as that in forward execution.

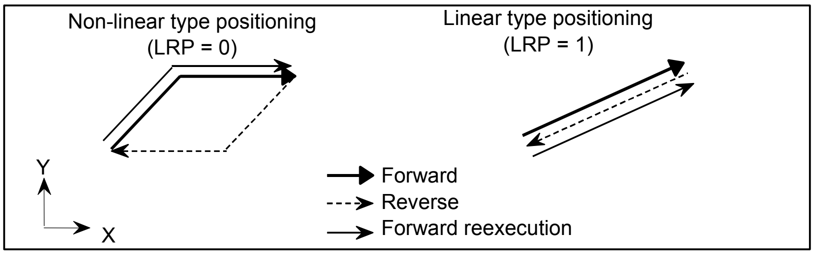


Fig. 2.1 (l)

- Dwell command (G04)
During reverse execution or forward reexecution, the dwell command (G04) is executed in the same way as in normal operation.

- Programmable data input (G10)
Tool compensation values, parameters, pitch error data, workpiece origin offsets, and tool life management values set or modified by programmable data input (G10) are ignored during reverse execution and forward reexecution.

- Skip function (G31) and automatic tool length compensation (G37)
The skip signal and the automatic tool length measurement signal are ignored during reverse execution and forward reexecution. During reverse execution and forward reexecution, the tool moves along the path that the tool has actually passed during forward execution.

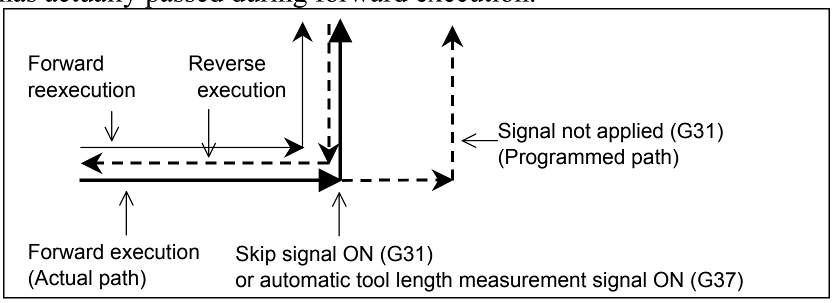


Fig. 2.1 (m)

- Setup of a coordinate system (G92, G54 to G59, G54.1P_, G52, and G92.1)
When setup of a coordinate system (G92, G54 to G59, G54.1P_, and G52) is specified during reverse execution, the indicated current position may differ from the position indicated during forward execution. However, the actual machine position does not differ.

- Mirror image
When a block to which a mirror image is applied by programmable mirror image (G50.1, G51.1) is executed during reverse execution, the tool moves along the actual path resulting from the application of a mirror image in the reverse direction.
When a mirror image is applied to a block by setting or a machine signal, the block with the mirror image not applied is stored. Mirror image application by setting or a machine signal is enabled also during reverse execution and forward reexecution. Therefore, during reverse execution and forward reexecution, the mirror image by setting data or machine signal must be turned on and off so that this on/off status and the on/off status during forward execution match.

- Changing offsets

Even when cutter compensation data or tool length offsets are changed during reverse execution or forward reexecution, the change in compensation or offset data does not become valid until forward reexecution ends and normal operation starts. Until then, the tool moves with the offset data that was applied when the block was executed for the first time during forward execution.

- Feedrate clamp

During reverse execution or forward reexecution, feedrate clamp is not performed with parameter No. 1420 (rapid traverse rate) or parameters Nos. 1430 and 1432 (maximum cutting feedrate). It is executed with parameter No. 1414 or at the feedrate assumed during forward execution.
If, for example, the parameters above are set to smaller values during reverse execution or forward reexecution, clamp is not performed with these values, but with parameter No. 1414 or at the feedrate assumed during forward execution.
For clamp at the feedrate assumed during backward execution or forward reexecution, change the feedrate with the external deceleration or override signal.

- Interrupt type custom macro

(1) Do not initiate any interrupt during reverse execution.
(2) Do not execute an interrupted block and the interrupt program in reverse execution.

- Tool management function

The tool life is not counted during reverse execution and forward reexecution.

- Inverse time feed (G93)

If a nonzero value is set as the feedrate to be applied during reverse execution in parameter No. 1414, a block that moves the tool by inverse time feed during forward execution is executed at the parameter-set feedrate (feed per minute) during reverse execution.
If the feedrate during reverse execution (parameter No. 1414) is not set (= 0), the same feedrate as applied during forward execution is used.

- Maximum spindle speed clamp (G92Sxxxx)

Clamping at a maximum spindle speed specified during reverse execution becomes valid. This means that if G92Sxxxx appears during reverse execution, the spindle speed is clamped at Sxxxx in the subsequent reverse execution. As a result, the clamp speed may differ between reverse execution and forward execution even when the same block is executed. The spindle speed is clamped when the G96 mode is set.

- Auxiliary functions

M, S, T, and the second auxiliary function (B function) are output directly also during reverse execution and forward reexecution.
When specified together with a move command in the same block, M, S, T, and the second auxiliary function (B function) are output with the move command at the same time during forward execution, reverse execution, and forward reexecution. Therefore, the output positions of M, S, T, and the second auxiliary function (B function) during reverse execution differ from those during forward execution and forward reexecution.

- Custom macro operation

Custom macro operations are ignored during reverse execution and forward reexecution.

- Tool retract and recover function

For retract operation and repositioning operation by the tool retract and recover function, reverse execution cannot be performed. Retract operation and repositioning operation are ignored during reverse execution and forward reexecution.

- AI contour control

During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI contour control is disabled.

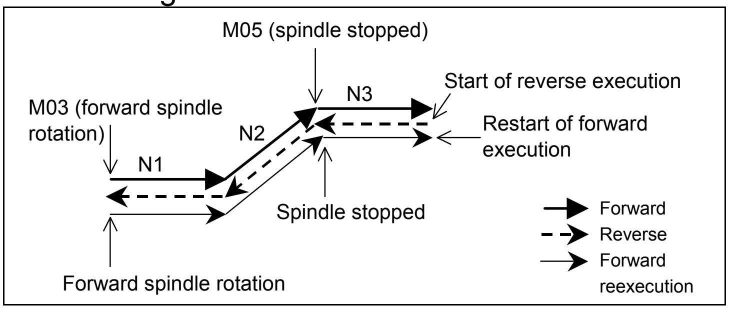
- Display

During reverse execution and forward reexecution, the modal display and the display of the currently executed program are not updated; information obtained at the start of reverse execution is maintained.

Warning

⚠ WARNING

1 Auxiliary functions are output directly even during reverse execution and forward reexecution. Accordingly, the execution status of an auxiliary function during forward execution may be reversed during reverse execution.
Example:
When forward rotation of the spindle (M03) and stop (M05) are specified
When N3 is executed during reverse execution, M05 is output. So, when N2 and N1 are executed during reverse execution, operation is performed with the spindle stopped.
When N1 is executed during forward reexecution, M03 is output. So, when N1 and N2 are executed during forward reexecution, operation is performed with the spindle rotating in the forward direction.



2 To perform reverse execution after a feed hold stop or single block stop operation, be sure to restore the original position if manual intervention has been performed after the stop , then set the "REVERSE" switch to on. Movements made by manual intervention are ignored during reverse execution and forward reexecution. (The same operation as in the manual absolute off state takes place.)
If manual intervention is performed during reverse execution or forward reexecution, the amount of manual intervention is added to the coordinate system at a restart after a stop due to a feed hold or single block during forward execution after the end of forward reexecution. Whether to add the amount of manual intervention follows the manual absolute switch.

APPENDIX

A PARAMETERS

This manual describes all parameters indicated in this manual.
For those parameters that are not indicated in this manual and other parameters, refer to the parameter manual.

Appendix A, "PARAMETERS", consists of the following sections:

A.1 DESCRIPTION OF PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
0001							FCV	

[Input type] Setting input
[Data type] Bit path

#1 FCV Program format
0: Series 0 standard format
(This format is compliant with the Series 0*i*-C.)
1: Series 10/11 format

> **NOTE**
> 1 Programs created in the Series 10/11 program format can be used for operation on the following functions:
> 1 Subprogram call M98,M198
> 2 Drilling canned cycle
> G80 to G89 (T series)
> G73, G74, G76, G80 to G89(M series)
> 2 When the program format used in the Series 10/11 is used for this CNC, some limits may add. Refer to the Operator’s Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
1004	IPR							

[Input type] Parameter input
[Data type] Bit path

#7 IPR Whether the least input increment for each axis is set to a value 10 times as large as the least command increment is specified, in increment systems of IS-B or IS-C at setting mm.
0: The least input increment is not set to a value 10 times as large as the least command increment.
1: The least input increment is set to a value 10 times as large as the least command increment.
If IPR is set to 1, the least input increment is set as follows:

Input increment	Least input increment
IS-B	0.01 mm, 0.01 deg, or 0.0001 inch

Input increment	Least input increment
IS-C	0.001 mm, 0.001 deg, or 0.00001 inch

> **NOTE**
> For IS-A, the least input increment cannot be set to a value 10 times as large as the least command increment.
> The least input increment is not multiplied by 10 also when the calculator-type decimal point input (bit 0 (DPI) of parameter No. 3401) is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1013							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 Increment system of each axis

Increment system	#1 ISCx	#0 ISAx
IS-A	0	1
IS-B	0	0
IS-C	1	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 (parameter No. 1020) can be arbitrarily selected from 'A', 'B', 'C', 'U', 'V', 'W', 'X', 'Y', and 'Z'.

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

> **NOTE**
> 1 The same axis name cannot be set for multiple axes.
> 2 When the 2nd auxiliary function is provided (when bit 2 (BCD) of parameter No. 8132 is 1), if the address (parameter No. 3460) that specifies the 2nd auxiliary function is used as an axis name, the 2nd auxiliary function is disabled.

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, tool radius/tool nose radius compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotation axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

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 one-digit F code feed may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.
Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0			LRP	

[Input type] Parameter input
[Data type] Bit path

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.

#4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

1410	Dry run rate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
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)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
When the machine doesn't need to change cutting feedrate frequently during cutting, a cutting feedrate can be specified in the parameter. This eliminates the need to specify a cutting feedrate (F command) in the NC program.

The feedrate set in this parameter is valid from when the CNC enters the clear state (when bit 6 (CLR) of parameter No. 3402 is 1) due to power-on or a reset to when the feedrate is specified by a program command (F command). After the feedrate is specified by a program command (F command), the feedrate is valid. For details on the clear state, refer to Appendix in the Operator's Manual (B-64304EN).

1414	Feedrate for retrace

[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 a cutting feedrate for retrace operation of Retrace function. When 0 is set, a retrace operation is performed at a programmed feedrate.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

	#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				JGLx			CTBx	CTLx

[Input type] Parameter input

[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

NOTE
When using bell-shaped acceleration/deceleration after interpolation, set this parameter to 0 and set bit 1 (CTBx) of parameter No. 1610 to select bell-shaped acceleration/deceleration after interpolation.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration after interpolation
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell-shaped acceleration/deceleration after interpolation

#1 CTBx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (depending on the setting in CTLx, bit 0 of parameter No.1610)
1: Bell-shaped acceleration/deceleration is applied.

NOTE
This parameter is valid only when the bell-shaped acceleration/deceleration after cutting feed interpolation function is used. When this function is not used, the acceleration/deceleration is determined according to bit 0 (CTLx) of parameter No. 1610 regardless of the setting of this parameter.

#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)

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)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

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.

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/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (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.

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					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.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 32, 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 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

When the subscription of an axis name is not set in a 2-path system, the subscription of an axis name is automatically set to the path number. To hide the subscription of an axis name, set the parameter of the subscription of an axis name to the ASCII code (32) of a space.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Input type] Parameter input
[Data type] Bit path

#0 WOF Setting the tool offset value (tool wear offset) by MDI key input is:
0: Not disabled.
1: Disabled. (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)

> **NOTE**
> When tool offset memory A is selected, the tool offset set in the parameter WOF is followed even if geometric compensation.

#1 GOF Setting the tool geometry offset value by MDI key input is:
0: Not disabled.
1: Disabled. (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled

3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to Tool compensation count - 1

When the modification of tool offset values by MDI key input is to be disabled using bit 0 (WOF) of parameter No.3290 and bit 1 (GOF) of parameter No.3290, parameter Nos.3294 and 3295 are used to set the range where such modification is disabled. In parameter No.3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No.3295, set the number of such values. In the following cases, however, none of the tool offset values may be modified:

- When 0 or a negative value is set in parameter No.3294
- When 0 or a negative value is set in parameter No.3295
- When a value greater than the maximum tool offset number is set in parameter No.3294

In the following case, a modification to the values ranging from the value set in parameter No.3294 to the maximum tool offset number is disabled:

When the value of parameter No.3294 added to the value of parameter No.3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI panel, the warning "WRITE PROTECT" is issued.

[Example]

When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:

- Bit 1 (GOF) of parameter No.3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No.3290 = 1 (to disable tool wear offset value modification)
- Parameter No.3294 = 51
- Parameter No.3295 = 10

If the setting of bit 0 (WOF) of parameter No.3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3401								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.

G19	G18	G17, G18, or G19 mode
0	0	G17 mode (X-Y plane)
0	1	G18 mode (Z-X plane)
1	0	G19 mode (Y-Z plane)

#3 G91 When the power is turned on or when the control is cleared
0: G90 mode (absolute command)
1: G91 mode (incremental command)

#6 CLR Reset button on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the 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

C23 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 RESET key of the MDI panel, the external reset signal, the reset & rewind signal, or the emergency stop signal.
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)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
When a circular interpolation command is executed, the tolerance for the radius between the start point and the end point is set.

> **NOTE**
> When the setting is 0, the difference between the arc radius values is not checked.

	#7	#6	#5	#4	#3	#2	#1	#0
5000							MOF	SBK

[Input type] Setting input
[Data type] Bit path

#0 SBK With a block created internally for cutter compensation or tool nose radius compensation:
0: A single block stop is not performed.
1: A single block stop is performed.

This parameter is used to check a program including cutter compensation/tool nose radius compensation.

#1 MOF When the tool length compensation shift type (bit 6 (TOS) of parameter No. 5006 is set to 1) is used, if the tool length compensation amount is changed(NOTE 2) in the tool length compensation mode when look-ahead blocks are present(NOTE 1):

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 command for the compensation axis are specified.

NOTE

1 "When look-ahead blocks are present" means as follows:

- The modal G code of the G codes (such as tool nose radius compensation) of group 07 is other than G40.

  One look-ahead block during automatic operation and multiple look-ahead blocks in the AI advanced preview control/AI contour control mode are not included in the state "when look-ahead blocks are present".

2 Changes in tool length compensation amount are as follows:

- When the tool length compensation number is changed by H code
- 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.
- When a tool length compensation vector canceled temporarily during tool length compensation by G53, G28, or G30 is recovered

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO	TPH	EVR	TAL	OFH	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

#2 OFH In cutter compensation (G40, G41, or G42), the address used to specify a compensation number is:
0: Address D.
1: Address H.

> **NOTE**
> When this parameter is 1, if tool length compensation and cutter compensation are specified in the same block, cutter compensation is prioritized.

#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 cutter 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.

#5 TPH In tool offsets (G45, G46, G47, or G48), the address used to specify a compensation number is:
0: Address D.
1: Address H.

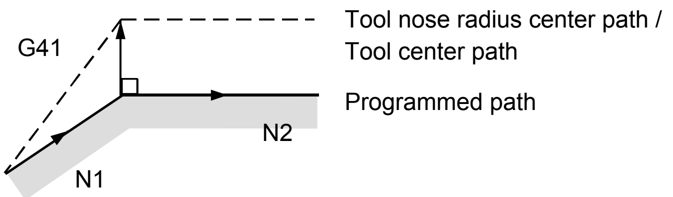
> **NOTE**
> This parameter is valid when bit 2 (OFH) of parameter No. 5001 is 0.

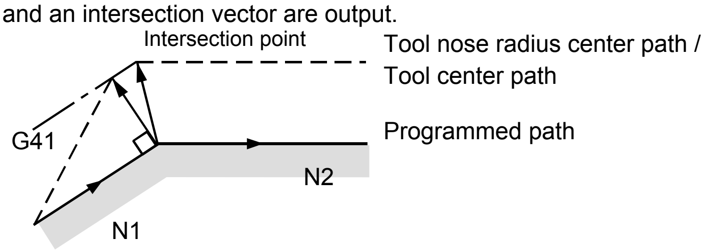
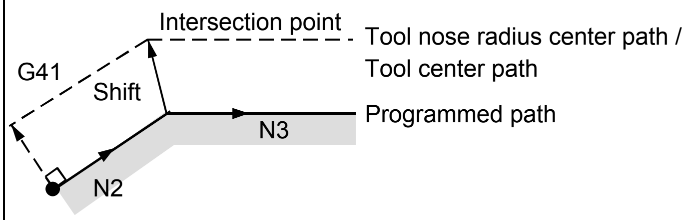
#6 EVO If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):
0: The new value becomes valid in a block where G43, G44, or an H code is specified next.
1: The new value becomes valid in a block where buffering is 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 cutter compensation or tool nose radius compensation.

<table>
<tr><th>SUV</th><th>SUP</th><th>Type</th><th>Operation</th></tr>
<tr><td>0</td><td>0</td><td>Type A</td><td>A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output.

</td></tr>
</table>

SUV	SUP	Type	Operation
0	1	Type B	A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output. 
1	0 1	Type C	When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.  When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.

NOTE

When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS0*i*-TC is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5005			QNI					

[Input type] Parameter input
[Data type] Bit path

#5 QNI With the tool length measurement function, a tool compensation number is selected by:
0: Operation through the MDI panel by the operator (selection based on cursor operation).
1: Signal input from the PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
5006		TOS						

[Input type] Parameter input
[Data type] Bit

#6 TOS Set a tool length compensation operation.
0: Tool length compensation is performed by an axis movement.
1: Tool length compensation is performed by shifting the coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
5008					CNV		CNC	

[Input type] Parameter input
[Data type] Bit path

#1 CNC
#3 CNV These bits are used to select an interference check method in the cutter compensation or tool nose radius compensation mode.

CNV	CNC	Operation
0	0	Interference check is enabled. The direction and the angle of an arc are checked.
0	1	Interference check is enabled. Only the angle of an arc is checked.
1	-	Interference check is disabled.

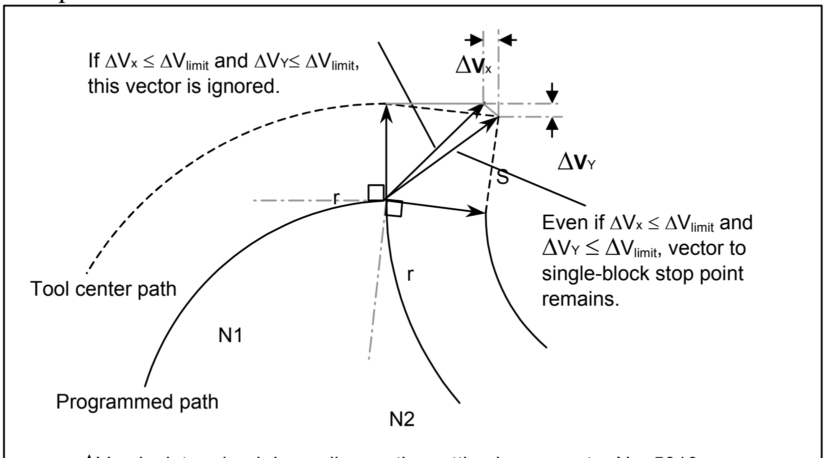
For the operation taken when the interference check shows the occurrence of an reference (overcutting) , see the description of bit 5 (CAV) of parameter No. 19607.

> **NOTE**
> Checking of only the direction cannot be set.

5010	Limit for ignoring the small movement resulting from cutter or tool nose radius compensation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

When the tool moves around a corner in cutter compensation or tool nose radius compensation mode, the limit for ignoring the small travel amount resulting from compensation is set. This limit eliminates the interruption of buffering caused by the small travel amount generated at the corner and any change in feedrate due to the interruption.



ΔV_{limit} is determined depending on the setting in parameter No. 5010.

	#7	#6	#5	#4	#3	#2	#1	#0
5042							OFC	OFA

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OFA
#1 OFC These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFC	OFA	Unit	Valid data range
0	1	0.01mm	±9999.99mm
0	0	0.001mm	±9999.999mm
1	0	0.0001mm	±9999.9999mm

For inch input

OFC	OFA	Unit	Valid data range
0	1	0.001inch	±999.999inch
0	0	0.0001inch	±999.9999inch
1	0	0.00001inch	±999.99999inch

	#7	#6	#5	#4	#3	#2	#1	#0
5101								FXY

[Input type] Parameter input
[Data type] Bit path

#0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:
0: In case of the Drilling canned cycle:
Z-axis at all times.
In case of the Grinding canned cycle:
G75,G77 command :Y-axis
G78,G79 command :Z-axis
1: Axis selected by the program

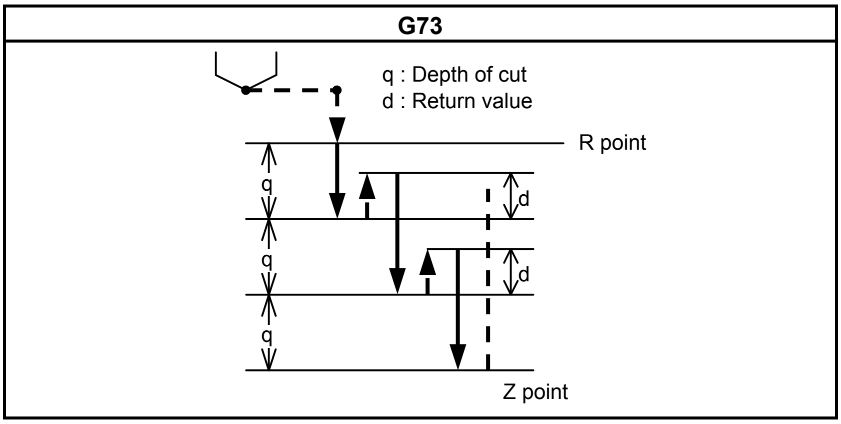
	#7	#6	#5	#4	#3	#2	#1	#0
5105								SBC

[Input type] Parameter input
[Data type] Bit path

#0 SBC In each of a drilling canned cycle, chamfering/corner rounding cycle,
0: A single block stop is not carried out.
1: A single block stop is carried out.

5114	Return value of high-speed peck drilling cycle

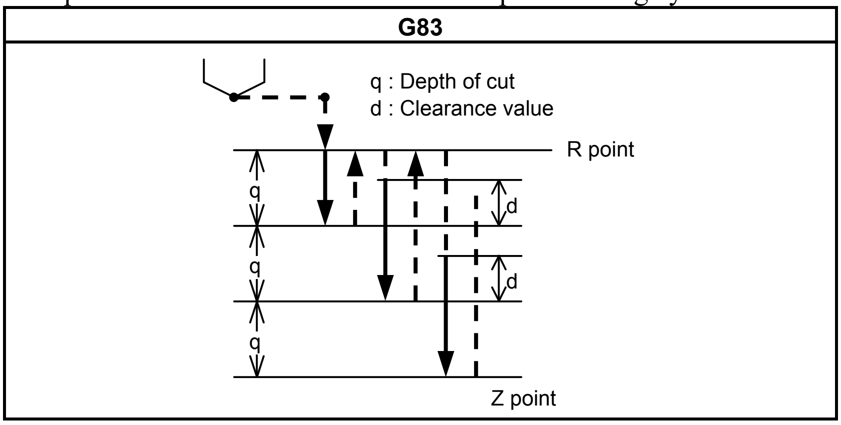
[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the return value in high-speed peck drilling cycle.



5115	Clearance value in a peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a clearance value in a peck drilling cycle.



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] -5 to 5

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)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

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)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

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)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the 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, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5177	Grinding axis number of 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, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5178	Grinding axis number of 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, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

5179	Grinding axis number of 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, PS0456 alarm is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, PS0456 alarm is also issued.

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, PS0456 alarm 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 PS0456 alarm 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, PS0456 alarm 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 PS0456 alarm 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, PS0456 alarm 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 PS0456 alarm 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, PS0456 alarm 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 PS0456 alarm is also issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5200		FHD	PCP	DOV				G84

[Input type] Parameter input
[Data type] Bit path

#0 G84 Method for specifying rigid tapping:
0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

#4 DOV Override during extraction in rigid tapping:
0: Invalidated
1: Validated (The override value is set in parameter No.5211. However, set an override value for rigid tapping return in parameter No.5381.)

#5 PCP Address Q is specified in a tapping cycle/rigid tapping:
0: A high-speed peck tapping cycle is assumed.
1: A peck tapping cycle is assumed.

> **NOTE**
> In a tapping cycle, this parameter is valid when bit 6 (PCT) of parameter No. 5104 is 1. When bit 6 (PCT) of parameter No. 5104 is 0, a (high-speed) peck tapping cycle is not assumed.

#6 FHD Feed hold and single block in rigid tapping:
0: Invalidated
1: Validated

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU			

[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
5202		OVE						

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#6 OVE The specification range of extraction override command (address J) by rigid tapping program specification is:
0: 100% to 200%.
1: 100% to 2000%.

> **NOTE**
> 1 To enable the extraction override command (address J) by program specification, set bit 4 (OV3) of parameter No.5201 to 1.
> 2 When this parameter is set to 1, the operation equivalent to that of the FS0*i*-C is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

[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.

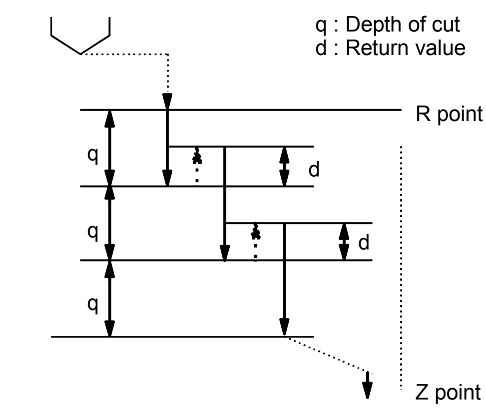
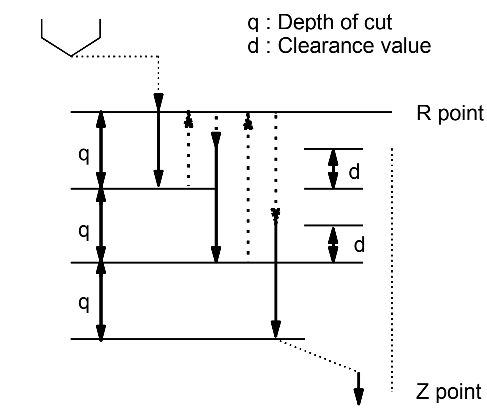
5211	Override value during rigid tapping extraction

[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 or clearance in peck tapping cycle

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the drilling axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the escape value of a high-speed peck tapping cycle or the clearance value of a peck tapping cycle.

When the parameter PCP (bit 5 of No.5200) is set to 0.	When the parameter PCP (bit 5 of No.5200) is set to 1.
	

NOTE
1 In a tapping cycle, this parameter is valid when bit 6 (PCT) of parameter No. 5104 is 1.
2 For the diameter axis, set this parameter using the diameter value.

5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 9999

Spindle position coder gear ratio
1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999

Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.
Set the same value for both parameter No.5241 and parameter No.5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200 will be issued.

5321	Spindle backlash in rigid tapping (first-stage gear)
5322	Spindle backlash in rigid tapping (second-stage gear)
5323	Spindle backlash in rigid tapping (third-stage gear)

[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	SCR	XSC						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)

#6 XSC The setting of a scaling magnification (axis-by-axis scaling) is:
0: Disabled.
1: Enabled.

#7 SCR Scaling (G51) magnification unit:
0: 0.00001 times (1/100,000)
1: 0.001 times

	#7	#6	#5	#4	#3	#2	#1	#0
5401								SCLx

[Input type] Parameter input

[Data type] Bit axis

#0 SCLx Scaling on this axis:
0: Invalidated
1: Validated

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.

5411	Scaling (G51) magnification

[Input type] Setting input
[Data type] 2-word path
[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No.5400)
[Valid data range] 1 to 999999999

This parameter sets a scaling magnification when axis-by-axis scaling is disabled (with bit 6 (XSC) of parameter No. 5400 set to 0). If no scaling magnification (P) is specified in the program, the setting of this parameter is used as a scaling magnification.

> **NOTE**
> When bit 7 (SCR) of parameter No.5400 is set to 1, the valid data range is 1 to 9999999.

5421	Scaling magnification for each axis

[Input type] Setting input
[Data type] 2-word axis
[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No.5400)
[Valid data range] -999999999 to –1, 1 to 999999999

This parameter sets a scaling magnification for each axis when axis-by-axis scaling is enabled (with bit 6 (XSC) of parameter No. 5400 set to 1). For the first spindle to the third spindle (X-axis to Z-axis), the setting of this parameter is used as a scaling magnification if scaling magnifications (I, J, K) are not specified in the program.

> **NOTE**
> When bit 7 (SCR) of parameter No.5400 is set to 1, the valid data ranges are -9999999 to -1 and 1 to 9999999.

	#7	#6	#5	#4	#3	#2	#1	#0
5431								MDL

[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 MDL The G60 code (one-direction positioning) is:
0: One-shot G code (group 00).
1: Modal G code (group 01).

5440	Positioning direction and overrun distance in single directional positioning

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the positioning direction and overrun distance in single directional positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.
Overrun distance>0: The positioning direction is positive (+).
Overrun distance<0: The positioning direction is negative (-).
Overrun distance=0: Single directional positioning is not performed.

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
[Minimum 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
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

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
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. (Set bit 2 (ABS) of parameter No.5500, to 1.)

#4 G90 A command for the index table indexing axis is:
0: Assumed to be an absolute or incremental command according to the mode.
1: Always assumed to be an absolute command.

#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 (IXS) of parameter No.5502 is followed.
1: The commands are executed.

NOTE
Even when this parameter is set to 1, an alarm (PS1564) 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.

> **NOTE**
> To enable the index table indexing function, set bit 3 (IXC) of parameter No. 8132 to 1 in addition to this parameter. The index table indexing function is enabled only when both ITI and IXC are enabled.

	#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) 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) 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. Rotation in the negative direction is performed only when the M code set in this parameter is specified together with a movement command.

> **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
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the 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.

> **NOTE**
> When the setting is 0, specification can be performed regardless of the minimum angle.

	#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)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
These parameters set the relevant feedrate during measurement of automatic tool 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)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the relevant γ value during automatic tool length measurement.

NOTE
1 For the M series, 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)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the relevant ε value during automatic tool length measurement.

NOTE
1 For the M series, 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

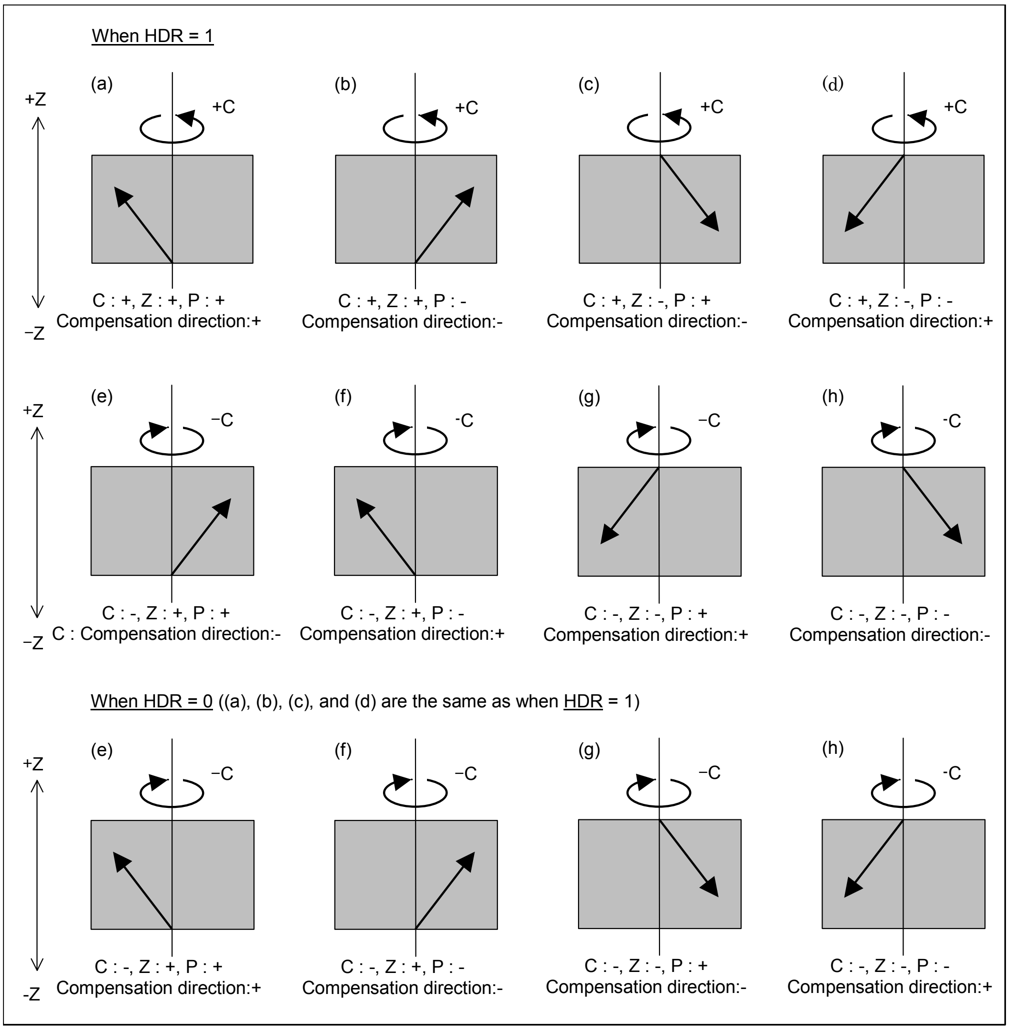
#1 ABS For the move command after manual intervention in the manual absolute on state:
0: Different paths are used in the absolute (G90) and incremental (G91) modes.
1: The same path (path in the absolute mode) is used in the absolute (G90) and incremental (G91) modes.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						HDR		HBR

[Input type] Parameter input
[Data type] Bit path

#0 HBR When the electronic gear box (EGB) function is used, performing a reset:
0: Cancels the synchronous mode (G81).
1: Does not cancel the synchronous mode. The mode is canceled only by the G80 command.

#2 HDR Direction of helical gear compensation (usually, set 1.)
(Example) To cut a left-twisted helical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.



	#7	#6	#5	#4	#3	#2	#1	#0
7701					LZR			

[Input type] Parameter input
[Data type] Bit path

#3 LZR When L (number of hob threads) = 0 is specified at the start of EGB synchronization (G81):
0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702					ART			TDP

[Input type] Parameter input
[Data type] Bit path

#0 TDP The specifiable number of teeth, T, of the electronic gear box (G81) is:
0: 1 to 1000
1: 0.1 to 100 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 1000 can be specified.

#3 ART The retract function executed when an alarm is issued is:
0: Disabled.
1: Enabled.
When an alarm is issued, a retract operation is performed with a set feedrate and travel distance (parameter Nos. 7740 and 7741).

> **NOTE**
> If a servo alarm is issued for other than the axis along which a retract operation is performed, the servo activating current is maintained until the retract operation is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
7703						ARO	ARE	ERV

[Input type] Parameter input
[Data type] Bit path

#0 ERV During EGB synchronization (G81), feed per revolution is performed for:
0: Feedback pulses.
1: Pulses converted to the speed for the workpiece axis.

#1 ARE In the retract function by an alarm, retract operation is:
0: Performed during EGB synchronization or automatic operation (automatic operation signal = 1).
1: Determined by the setting of parameter ARO.

#2 ARO The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization.
1: EGB synchronization and automatic operation (automatic operation signal OP = 1).

> **NOTE**
> This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 1.

The following table lists the parameter settings and corresponding operation.

ARE	ARO	Operation
1	0	During EGB synchronization
1	1	During EGB synchronization and automatic operation
0	0	During EGB synchronization or automatic operation
0	1	

NOTE
Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when an alarm is issued).

	#7	#6	#5	#4	#3	#2	#1	#0
7731				RTS	ECN		EHF	EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

NOTE
When this parameter is set to 0, no drilling canned cycle can be used.

#1 EHF Feed-forward control for the axial feed axis for helical compensation is:
0: Enabled only during cutting.
1: Always enabled in the G81 synchronous mode.
Usually, set 0.
Feed-forward control is usually enabled in the cutting feed mode. When this parameter is set to 1, feed-forward control is always enabled for the axial feed axis for helical compensation during synchronization by the command (G81) for a hobbing machine.
When bit 3 (FFR) of parameter No. 1800 is set to 1, feed-forward control is always enabled regardless of the setting of this parameter.

#3 ECN During EGB synchronization:
0: G81 cannot be specified again. (An alarm (PS1595) occurs.)
1: G81 can be specified.

#4 RTS When an OT alarm or axis type malfunction protection alarm is issued during EGB retract operation:
0: Only the axis for which the alarm is issued is stopped.
1: All axes are stopped.

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets the feedrate during retraction for each axis.

7741	Retracted distance

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the retracted distance for each axis.

7772	Number of position detector pulses per rotation about the tool axis

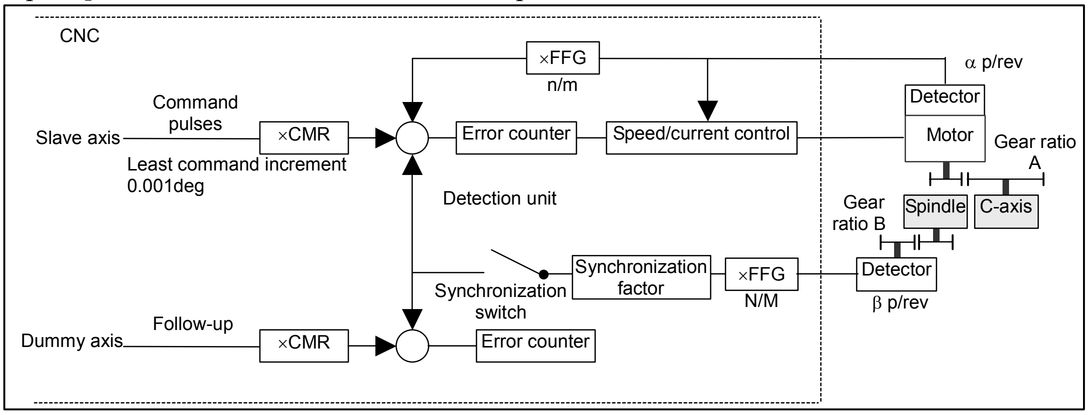
[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the tool axis (on the spindle side), for the position detector.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

NOTE
Specify the number of feedback pulses per rotation about the tool axis for the position detector, considering the gear ratio with respect to the position coder.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the workpiece axis (on the slave side), for the position detector.
Set the number of pulses output by the detection unit.
Set parameters Nos. 7772 and 7773 when using the G81 EGB synchronization command.

[Example 1] When the EGB master axis is the spindle and the EGB slave axis is the C-axis



Gear ratio of the spindle to the detector B:
1/1 (The spindle and detector are directly connected to each other.)

Number of detector pulses per spindle rotation β: 80,000 pulses/rev
(Calculated for four pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis: 1/1
Gear ratio of the C-axis A: 1/36 (One rotation about the C-axis to 36 motor rotations)
Number of detector pulses per C-axis rotation α: 1,000,000 pulses/rev
C-axis CMR: 1
C-axis FFG n/m: 1/100

In this case, the number of pulses per spindle rotation is:
$80000 \times 1/1 = 80000$
Therefore, set 80000 for parameter No. 7772.
The number of pulses per C-axis rotation in the detection unit is:
$1000000 \div 1/36 \times 1/100 = 360000$
Therefore, set 360000 for parameter No. 7773.

[Example 2] When the gear ratio of the spindle to the detector B is 2/3 for the above example (When the detector rotates twice for three spindle rotations)
In this case, the number of pulses per spindle rotation is:

$$80000 \times \frac{2}{3} = \frac{160000}{3}$$

160000 cannot be divided by 3 without a remainder. In this case, change the setting of parameter No. 7773 so that the ratio of the settings of parameters Nos. 7772 and 7773 indicates the value you want to set.

$$\frac{\text{No.7772}}{\text{No.7773}} = \frac{160000/3}{360000} = \frac{160000}{360000 \times 3} = \frac{160000}{1080000}$$

Therefore, set 160000 for parameter No. 7772 and 1080000 for parameter No. 7773.
As described above, all the settings of parameters Nos. 7772 and 7773 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings. For example, you may set 16 for parameter No. 7772 and 108 for parameter No. 7773 for this case.

	#7	#6	#5	#4	#3	#2	#1	#0
8132			SCL	SPK	IXC			

[Input type] Parameter input
[Data type] Bit

#3 IXC Index table indexing is:
0: Not Used.
1: Used.

> **NOTE**
> When enabling the index table indexing function, set bit 0 (ITI) of parameter No. 5501 to 0 in addition to this parameter. The index table indexing function is enabled only when both ITI and IXC are enabled.

#4 SPK Small-hole peck drilling cycle is:
0: Not Used.
1: Used.

#5 SCL Scaling is:
0: Not Used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8136		NGW						

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#6 NGW Tool offset memory C (M series) or tool geometry/wear compensation (T series) is:
0: Used.
1: Not Used.

	#7	#6	#5	#4	#3	#2	#1	#0
11600			AX1					

[Input type] Parameter input
[Data type] Bit path

#5 AX1 When coordinate rotation mode is in effect, if one axis is commanded in absolute mode:
0: A commanded position is calculated in the coordinate system before it is rotated.
1: The coordinate system is rotated and then a shift to a commanded position occurs on the rotated coordinate system.
(Parameter to have compatibility with FS0*i*-C)

	#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 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 check alarm function)

1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)

For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

#6 NAA When the interference check avoidance function considers that an avoidance operation is dangerous or that a further interference to the interference avoidance vector occurs:

0: An alarm is issued.
When an avoidance operation is considered to be dangerous, the alarm (PS5447) is issued.
When a further interference to the interference avoidance vector is considered to occur, the alarm (PS5448) is issued.

1: No alarm is issued, and the avoidance operation is continued.

> ⚠ **CAUTION**
> When this parameter is set to 1, the path may be shifted largely. Therefore, set this parameter to 0 unless special reasons are present.

19625	Number of blocks to be read in the cutter compensation/tool nose radius compensation mode

[Input type] Setting input
[Data type] Byte path
[Valid data range] 3 to 8

This parameter sets the number of blocks to be read in the cutter compensation/tool nose radius compensation mode. When a value less than 3 is set, the specification of 3 is assumed. When a value greater than 8 is set, the specification of 8 is assumed. As a greater number of blocks are read, an overcutting (interference) forecast can be made with a command farther ahead. However, the number of blocks read and analyzed increases, so that a longer block processing time becomes necessary.

Even if the setting of this parameter is modified in the MDI mode by stopping in the cutter compensation/tool nose radius compensation mode, the setting does not become valid immediately. Before the new setting of this parameter can become valid, the cutter compensation mode must be canceled, then the mode must be entered again.

A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit Bit machine group Bit path Bit axis Bit spindle	0 or 1	
Byte Byte machine group Byte path Byte axis Byte spindle	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Word Word machine group Word path Word axis Word spindle	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
2-word 2-word machine group 2-word path 2-word axis 2-word spindle	0 to ±999999999	Some parameters handle these types of data as unsigned data.
Real Real machine group Real path Real axis Real spindle	See the Standard Parameter Setting Tables.	

NOTE

1. Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
2. For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group. For the 0*i* -D/0*i* Mate-D, the maximum number of machine groups is always 1.
3. For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
4. For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
5. For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
6. The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

> **NOTE**
> 1 Values are rounded up or down to the nearest multiples of the minimum data unit.
> 2 A valid data range means data input limits, and may differ from values representing actual performance.
> 3 For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value" in the "OPERATOR'S MANUAL" (B-64304EN).

(A) Length and angle parameters (type 1)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
inch	IS-A	0.001	-99999.999 to +99999.999
	IS-B	0.0001	-99999.9999 to +99999.9999
	IS-C	0.00001	-9999.99999 to +9999.99999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
inch	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-A	0.01	0.0 to +999000.00
	IS-B	0.001	0.0 to +999000.000
	IS-C	0.0001	0.0 to +99999.9999
inch/min	IS-A	0.001	0.0 to +96000.000
	IS-B	0.0001	0.0 to +9600.0000
	IS-C	0.00001	0.0 to +4000.00000

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.001	0.000 to +999000.000
inch/min	IS-C	0.0001	0.0000 to +9600.0000

(D)Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 deg./sec^2	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
inch/sec^2	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.001	0.000 to +999999.999
inch/min	IS-C	0.0001	0.0000 to +99999.9999

B DIFFERENCES FROM SERIES 0*i*-C

Appendix B, "Differences from Series 0*i*-C", consists of the following sections:

B.1 SETTING UNIT

B.1.1 Differences in Specifications

Function	Explanation
Diameter/radius specification in the move command for each axis	- Make a selection using bit 3 (DIAx) of parameter No. 1006. **<u>Bit 3 (DIAx) of parameter No. 1006</u>** The move command for each axis specifies: 0: Radius. 1: Diameter. With Series 0*i*-C, in order for an axis whose diameter is specified to travel the specified distance, it is necessary not only to set 1 in bit 3 (DIAx) of parameter No. 1006 but also to make either of the following two changes: - Reduce the command multiplier (CMR) to half. (The detection unit does not need to be changed.) - Reduce the detection unit to half, and double the flexible feed gear (DMR). With Series 0*i*-D, by contrast, just setting 1 in bit 3 (DIAx) of parameter No. 1006 causes the CNC to reduce the command pulses to half, eliminating the need to make the changes described above (when the detection unit is not changed). Note that, when the detection unit is reduced to half, both the CMR and DMR need to be doubled.

B.1.2 Differences in Diagnosis Display

None.

B.2 AUTOMATIC TOOL OFFSET

B.2.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Operation of the current offset for the measurement result	- Added to the current offset.	- Select whether to add or subtract, by using bit 6 (MDC) of parameter No. 6210. **<u>Bit 6 (MDC) of parameter No. 6210</u>** The measurement result of automatic tool length measurement (system M) or automatic tool compensation (system T) is: 0: Added to the current offset. 1: Subtracted from the current offset.

Function	Series 0*i*-C	Series 0*i*-D
Setting of the feedrate for measurement	- Set the value in parameter No. 6241. This is a parameter common to the measuring position reached signals (XAE, YAE, and ZAE).	- **Parameter No. 6241** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6242** This is a parameter for the measuring position reached signals (XAE2 and GAE2). - **Parameter No. 6243** This is a parameter for the measuring position reached signals (XAE3 and GAE3). **NOTE** When 0 is set in parameter Nos. 6242 and 6243, the value in parameter No. 6241 becomes valid.
Setting of the γ value	- Set the value in parameter No. 6251. This is a parameter common to the measuring position reached signals (XAE, YAE, and ZAE).	- **Parameter No. 6251** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6252** This is a parameter for the measuring position reached signals (XAE2 and GAE2). - **Parameter No. 6253** This is a parameter for the measuring position reached signals (XAE3 and GAE3). **NOTE** When 0 is set in parameter No. 6252 and 6253, the value in parameter No. 6251 becomes valid.
Setting of the ε value	- Set the value in parameter No. 6254. This is a parameter common to the measuring position reached signals (XAE, YAE, and ZAE).	- **Parameter No. 6254** This is a parameter for the measuring position reached signals (XAE1 and GAE1). - **Parameter No. 6255** This is a parameter for the measuring position reached signals (XAE2 and GAE2). - **Parameter No. 6256** This is a parameter for the measuring position reached signals (XAE3 and GAE3). **NOTE** When 0 is set in parameter Nos. 6255 and 6256, the value in parameter No. 6254 becomes valid.

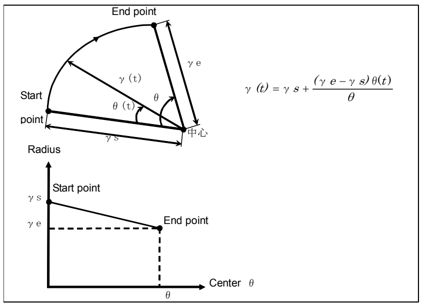
B.2.2 Differences in Diagnosis Display

None.

B.3 CIRCULAR INTERPOLATION

B.3.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Interpolation method when the arc end point is not on the arc</td><td colspan="2">If the difference between the radius values of the start point and end point of an arc is greater than the value set in parameter No. 3410, alarm PS0020 is issued. If the difference is smaller (the end point is not on the arc), circular interpolation is performed as follows.</td></tr>
<tr><td>- Circular interpolation is performed using the radius value of the start point and, when an axis reaches the end point, it is moved linearly.

<u>Parameter No. 3410</u>
In a circular interpolation command, set the limit allowed for the difference between the radius values of the start point and end point.</td><td>- Helical interpolation is performed as shown in the figure below.


In other words, the radius of the arc moves linearly according to the center angle θ(t). Specifying an arc where the arc radius of the start point differs from that of the end point enables helical interpolation. When performing helical interpolation, set a large value in parameter No. 3410 that specifies the limit for the arc radius difference.</td></tr>
</table>

B.3.2 Differences in Diagnosis Display

None.

B.4 HELICAL INTERPOLATION

B.4.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Specification of the feedrate	- Specify the feedrate along a circular arc. Therefore, the feedrate of the linear axis is as follows: $F \times \dfrac{\text{Length of linear axis}}{\text{Length of circular arc}}$	- Make a selection using bit 5 (HTG) of parameter No. 1403. 0: Same as left. 1: Specify a feedrate along the tool path including the linear axis. Therefore, the tangential velocity of the arc is expressed as follows: $F \times \dfrac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$ The velocity along the linear axis is expressed as follows: $F \times \dfrac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$ For details, refer to "HELICAL INTERPOLATION" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).
Helical cutting feedrate clamp	- Make a selection using bit 0 (HFC) of parameter No. 1404. 0: The feedrate of the arc and linear axes is clamped by parameter No. 1422 or No.1430. 1: The combined feedrate along the tool path including the linear axis is clamped by parameter No. 1422.	- Bit 0 (HFC) of parameter No. 1404 is not available. The feedrate of the arc and linear axes is clamped by parameter No. 1430.

B.4.2 Differences in Diagnosis Display

None.

B.5 SKIP FUNCTION

B.5.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Setting to enable the high-speed skip signal for normal skip (G31) when the multi-stage skip function is enabled</td><td>- Set 1 in bit 5 (SLS) of parameter No. 6200.</td><td>- Set 1 in bit 4 (HSS) of parameter No. 6200.</td></tr>
<tr><td colspan="2">
<table>
<tr><th rowspan="2">Multi-stage skip function</th><th rowspan="2">Command</th><th colspan="2">Parameter to decide on use of the high-speed skip signal</th></tr>
<tr><th>FS0i-C</th><th>FS0i-D</th></tr>
<tr><td>Disabled</td><td>G31 (normal skip)</td><td>HSS</td><td>HSS</td></tr>
<tr><td rowspan="2">Enabled</td><td>G31 (normal skip)</td><td><u>SLS</u></td><td><u>HSS</u></td></tr>
<tr><td>G31P1 to G31P4 (multi-stage skip)</td><td>SLS</td><td>SLS</td></tr>
</table>
</td></tr>
<tr><td>Target of acceleration/deceleration and servo system delay compensation</td><td>- Compensation is performed for the skip coordinates obtained when the high-speed skip signal is set to "1".</td><td>- Compensation is performed for the skip coordinates obtained when the skip or high-speed skip signal is set to "1".</td></tr>
<tr><td>Method of acceleration/deceleration and servo system delay compensation</td><td>- There are two ways to perform compensation, as follows.
[Compensating the value calculated from the cutting constant and servo constant]
Set 1 in bit 0 (SEA) of parameter No. 6201.
[Compensating the accumulated pulses and positional deviation due to acceleration/deceleration]
Set 1 in bit 1 (SEB) of parameter No. 6201.</td><td>- Bit 0 (SEA) of parameter No. 6201 is not available.
There is only one way to perform compensation, as follows.
[Compensating the accumulated pulses and positional deviation due to acceleration/deceleration]
Set 1 in bit 1 (SEB) of parameter No. 6201.</td></tr>
<tr><td>Skip cutting feedrate (normal skip)</td><td>- Feedrate specified by the F code in the program</td><td>- Depends on bit 1 (SFP) of parameter No. 6207. When 0 is set, the processing is the same as Series 0i-C.

<u>Bit 1 (SFP) of parameter No. 6207</u>
The feedrate during the skip function (G31) is:
0: Feedrate specified by the F code in the program.
1: Feedrate specified in parameter No. 6281.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Skip cutting feedrate (skip using the high-speed skip signal or multi-step skip)	- Feedrate specified by the F code in the program	- Depends on bit 2 (SFN) of parameter No. 6207. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 2 (SFP) of parameter No. 6207** When the skip function using the high-speed skip signal (1 is set in bit 4 (HSS) of parameter No. 6200) or the multi-step skip function is executed, the feedrate is: 0: Feedrate specified by the F code in the program. 1: Feedrate specified in parameter Nos. 6282 to 6285.
Axis to monitor to check whether the torque limit has been reached (torque limit skip)	- Depends on bit 3 (TSA) of parameter No. 6201. **Bit 3 (TSA) of parameter No. 6201** To check whether the torque limit has been reached, the torque limit skip function (G31 P99/98) monitors: 0: All axes. 1: Only the axis specified in the same block as G31 P99/98.	- Bit 3 (TSA) of parameter No. 6201 is not available. Only the axis specified in the same block as G31 P99/98 is monitored.
High-speed skip signal input for the G31 P99 command (torque limit skip)	As the skip signal for the G31 P99 command, the high-speed skip signal:	
	- Cannot be input.	- Can be input.
Setting of a positional deviation limit in the torque limit skip command (torque limit skip)	- No parameter is available dedicated to setting a positional deviation limit for the torque limit skip function.	- The value can be set in parameter No. 6287. **Parameter No. 6287** Set a positional deviation limit in the torque limit skip command for each axis.
When G31 P99/98 is specified without a torque limit being specified in advance (torque limit skip)	- The G31 P99/98 command is executed as is. (No alarm is issued.)	- Alarm PS0035 is issued.

B.5.2 Differences in Diagnosis Display

None.

B.6 MANUAL REFERENCE POSITION RETURN

B.6.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Conditions for performing manual reference position return during feed hold</td><td colspan="2">Manual reference position return is performed when automatic operation is halted (feed hold) and when any of the following conditions is met:
<Conditions>
(1) Travel distance is remaining.
(2) An auxiliary function (M, S, T, or B function) is being executed.
(3) A dwell, canned cycle, or other cycle is in progress.</td></tr>
<tr><td>- Depends on bit 2 (OZR) of parameter No. 1800.
[When OZR = 0]
Alarm PS0091 occurs, and manual reference position return is not performed.
[When OZR = 1]
Manual reference position return is performed without issuing an alarm.</td><td>- Bit 2 (OZR) of parameter No. 1800 is not available.
Alarm PS0091 occurs, and manual reference position return is not performed.</td></tr>
<tr><td>When inch/metric switch is done</td><td>- The reference position is lost.
(The reference position is not established.)</td><td>- The reference position is not lost.
(The reference position remains established.)</td></tr>
<tr><td>Reference position setting without dogs for all axes</td><td>- Set 1 in bit 1 (DLZ) of parameter No. 1002.</td><td>- Bit 1 (DLZ) of parameter No. 1002 is not available.
Reference position setting without dogs (bit 1 (DLZx) of parameter No. 1005) is set for all axes.</td></tr>
<tr><td>Function that performs reference position setting without dogs two or more times when the reference position is not established in absolute position detection</td><td>- Not available.</td><td>- Depends on bit 4 (GRD) of parameter No. 1007.

<u>Bit 4 (GRD) of parameter No. 1007</u>
For the axis on which absolute values are detected, when correspondence between the machine position and the position by the absolute position detector is not completed, the reference position setting without dogs is:
0: Not performed two or more times.
1: Performed two or more times.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Behavior when manual reference position return is started on a rotation axis with the deceleration dog pressed before a reference position is established	- Does not depend on bit 0 (RTLx) of parameter No. 1007. Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090.	- [Rotation axis type = A and bit 0 (RTLx) of parameter No. 1007 = 0] Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090. [Rotation axis type = A and bit 0 (RTLx) of parameter No. 1007 = 1] Movement is made at the rapid traverse feedrate until the grid is established. If the deceleration dog is released before the grid is established, one revolution is made at the rapid traverse feedrate, thus establishing the grid. Pressing the deceleration dog again establishes the reference position. [Rotation axis type = B] Does not depend on bit 0 (RTLx) of parameter No. 1007. Movement is made at the reference position return feedrate FL even if the grid is not established. Releasing the deceleration dog before the grid is established causes alarm PS0090.
Reference position shift function setting	- The function is enabled for all axes by setting 1 in bit 2 (SFD) of parameter No. 1002.	- Bit 2 (SFD) of parameter No. 1002 is not available. Set bit 4 (SFDx) of parameter No. 1008 for each axis.
Setting of whether to preset the coordinate system upon high-speed manual reference position return	- Not available. The coordinate system is not preset.	- Depends on bit 1 (HZP) of parameter No. 1206. **Bit 1 (HZP) of parameter No.1206** Upon high-speed manual reference position return, the coordinate system is: 0: Preset. 1: Not preset (FS0*i*-C compatible specification).

Function	Series 0*i*-C	Series 0*i*-D
G28/G30 command in the coordinate system rotation, scaling, or programmable mirror image mode	- Not available. Cancel the mode before executing the command.	- The command can be executed only when all of the conditions described below are met. Otherwise, alarm PS0412 occurs. <Conditions> [Conditions required before specifying the command] (1) An absolute command is specified for the target axis of coordinate system rotation, scaling, or programmable mirror image. (2) Tool length compensation has not been performed for the target axis of coordinate system rotation, scaling, or programmable mirror image when it is moved by reference position return. (3) Tool length compensation has been canceled. [Conditions required when specifying the command] (4) In an incremental command, the travel distance of the middle point is 0. [Conditions required after specifying the command] (5) The first move command specified for the target axis of coordinate system rotation, scaling, or programmable mirror image is an absolute command.

B.6.2 Differences in Diagnosis Display

None.

B.7 WORKPIECE COORDINATE SYSTEM

B.7.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Change in absolute position display when the workpiece zero point offset value is changed	- Make a selection using bit 5 (AWK) of parameter No. 1201. **Bit 5 (AWK) of parameter No. 1201** When the workpiece zero point offset value is changed: 0: Changes the absolute position display when the program executes the block that is buffered next. 1: Changes the absolute position display immediately. In either case, the changed value does not take effect until the block that is buffered next.	- Bit 5 (AWK) of parameter No. 1201 is not available. The tool always behaves as when AWK is set to 1.

B.7.2 Differences in Diagnosis Display

None.

B.8 LOCAL COORDINATE SYSTEM

B.8.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Clearing of the local coordinate system after servo alarm cancellation	- The processing is determined by the settings of bit 5 (SNC) and bit 3 (RLC) of parameter No. 1202. **Bit 3 (RLC) of parameter No. 1202** Upon reset, the local coordinate system is: 0: Not canceled. 1: Canceled. **Bit 5 (SNC) of parameter No. 1202** After servo alarm cancellation, the local coordinate system is: 0: Cleared. 1: Not cleared. **NOTE** When the RLC bit of the parameter is set to 1, the local coordinate system is cleared, even if the SNC bit of the parameter is set to 1.	- The processing is determined by the settings of bit 7 (WZR) of parameter No. 1201, bit 3 (RLC) of parameter No. 1202, bit 6 (CLR) of parameter No. 3402, and bit 6 (C14) of parameter No. 3407. Bit 5 (SNC) of parameter No. 1202 is not available. **Bit 7 (WZR) of parameter No. 1201** If the CNC is reset by the reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 0, the G code of group number 14 (workpiece coordinate system) is: 0: Placed in the reset state. 1: Not placed in the reset state. **NOTE** When bit 6 (CLR) of parameter No. 3402 is set to 1, the processing depends on the setting of bit 6 (C14) of parameter No. 3407. **Bit 3 (RLC) of parameter No. 1202** Upon reset, the local coordinate system is: 0: Not canceled. 1: Canceled. **NOTE** - When bit 6 (CLR) of parameter No. 3402 is set to 0 and bit 7 (WZR) of parameter No. 1201 is set to 1, the local coordinate system is canceled, regardless of the setting of this parameter. - When bit 6 (CLR) of parameter No. 3402 is set to 1 and bit 6 (C14) of parameter No. 3407 is set to 0, the local coordinate system is canceled, regardless of the setting of this parameter. **Bit 6 (CLR) of parameter No. 3402** The reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal places the local coordinate system in: 0: Reset state. 1: Clear state. **Bit 6 (C14) of parameter No. 3407** If the CNC is reset by the reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 1, the G code of group number 14 (workpiece coordinate system) is: 0: Placed in the clear state. 1: Not placed in the clear state.

Function	Series 0*i*-C	Series 0*i*-D
Operation with the local coordinate system setting (G52)	- Make a selection using bit 4 (G52) of parameter No. 1202. **Bit 4 (G52) of parameter No. 1202** 1) If there are two or more blocks that are not moved before G52 is specified during cutter compensation, or if G52 is specified after the cutter compensation mode is turned off, with the offset vector maintained, the local coordinate system setting is performed: 0: Without considering the cutter compensation vector. 1: Considering the cutter compensation vector. 2) When G52 is specified, the local coordinate system setting is performed for: 0: All axes. 1: Only those axes whose command addresses are found in the G52-specified block.	- Bit 4 (G52) of parameter No. 1202 is not available. The tool always behaves as when G52 is set to 1.

B.8.2 Differences in Diagnosis Display

None.

B.9 Cs CONTOUR CONTROL

B.9.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
In-position check when the Cs contour control mode is off	- The in-position check is not made.	- Make a selection using bit 2 (CSNs) of parameter No. 3729. **Bit 2 (CSNs) of parameter No. 3729** When the Cs contour control mode is off, the in-position check is: 0: Made. 1: Not made. When 1 is set in this parameter, the processing is the same as Series 0*i*-C.

B.9.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Position error display for Cs contour control	For the first spindle, diagnosis display No. 418 is used. For the second spindle, diagnosis display No. 420 is used.	For both the first and second spindles, diagnosis display No. 418 (spindle) is used.

B.10 SERIAL/ANALOG SPINDLE CONTROL

B.10.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Spindle number of the analog spindle	- When one serial spindle and one analog spindle are simultaneously controlled in one path (serial/analog spindle control), the spindle number of the analog spindle is as follows.	
	Third spindle	Second spindle For details about the parameters and other settings, refer to "SERIAL/ANALOG SPINDLE CNOTROL" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).

B.10.2 Differences in Diagnosis Display

None.

B.11 CONSTANT SURFACE SPEED CONTROL

B.11.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Constant surface speed control with no position coder	- This is an optional function for the T series. It is not available with the M series.	- This is a basic function for both M series and T series. It can be used by enabling constant surface speed control (setting 1 in bit 0 (SSC) of parameter No. 8133) and setting 1 in bit 2 (PCL) of parameter No. 1405.
	- Using bit 0 (PSSCL) of parameter No. 1407, select whether to enable or disable the axis feedrate clamp in feed per revolution when the spindle speed is clamped by the maximum spindle speed set in parameter No. 3772. **Bit 0 (PSSCL) of parameter No. 1407** In constant surface speed control with no position coder, when the spindle speed is clamped by the maximum spindle speed parameter, the axis feedrate in feed per revolution is: 0: Not clamped. 1: Clamped. When 1 is set in this parameter, select the spindle to be used for feed per revolution by using the position coder selection signal. (To use the position coder selection signal requires enabling multi-spindle control.)	- Bit 0 (PSSCL) of parameter No. 1407 is not available. The axis feedrate is always clamped. Using the position coder selection signal, select the spindle to be used for feed per revolution. (To use the position coder selection signal requires enabling multi-spindle control.) The M series does not support the multi-spindle control function. Therefore, the second spindle cannot be used for feed per revolution.

B.11.2 Differences in Diagnosis Display

None.

B.12 TOOL FUNCTIONS

B.12.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Specification of a G code of the 00 group and a T code in the same block	- Not allowed.	- Not allowed. Specifying a G code in this way causes alarm PS0245.

Function	Series 0*i*-C	Series 0*i*-D
Behavior when G49 and G40 are specified in the same block	- Make a selection using bit 6 (GCS) of parameter No. 5008. **<u>Bit 6 (GCS) of parameter No. 5008</u>** When G49 (tool length compensation cancellation) and G40 (cutter compensation cancellation) are specified in the same block: 0: Tool length compensation is canceled in the next block. 1: Tool length compensation is canceled in the block in which the command is specified.	- Bit 6 (GCS) of parameter No. 5008 is not available. The tool always behaves as when 1 is set in bit 6 (GCS) of parameter No. 5008. (Tool length compensation is canceled in the command block.)
Specification of the tool length compensation amount (Select the compensation amount number with H code.)	- Depends on whether the order of compensation amount numbers specified by the H code is that of tool length compensation types A, B, and C, whether the cutter compensation mode is on or off, and the setting of bit 2 (OFH) of parameter No. 5001. For details, refer to Section 14.1, "TOOL LENGTH COMPENSATION", in "OPERATOR'S MANUAL" (B-64124EN).	- Not dependent on the conditions described at left. In Series 0*i*-D, the H code is used to specify the compensation amount number (select the compensation amount), and G43, G44, and G49 are used to select whether to enable or disable tool length compensation. For details, refer to Section 6.1, "TOOL LENGTH COMPENSATION", in "OPERATOR'S MANUAL (MACHINING CENTER)" (B-64304EN-2).
Restoration of the tool length compensation vector canceled by specifying G53, G28, or G30 during tool length compensation	- The restoration conditions differ depending on the setting of bit 2 (OFH) of parameter No. 5001, as well as on whether the cutter compensation mode is on or off. For details, refer to Section 14.1, "TOOL LENGTH COMPENSATION", in "OPERATOR'S MANUAL" (B-64124EN).	- Not dependent on the setting of bit 2 (OFH) of parameter No. 5001 or the cutter compensation mode. Depends only on the setting of bit 6 (EVO) of parameter No. 5001. **<u>Bit 6 (EVO) of parameter No. 5001</u>** For tool length compensation type A or B, if the tool compensation amount is changed during the offset mode (G43 or G44), the vector is restored in: 0: Subsequent block containing a G43 or G44 command or a H code. 1: Block buffered next.

B.12.2 Differences in Diagnosis Display

None.

B.13 TOOL COMPENSATION MEMORY

B.13.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Unit and range of tool compensation values</td><td>- The unit and range of tool compensation values are determined by the setting unit.</td><td>- Set the unit and range using bit 0 (OFA) and bit 1 (OFC) of parameter No. 5042.

<u>Bit 0 (OFA) and bit 1 (OFC) of parameter No. 5042</u>
Select the setting unit and range of tool offset values.

Metric input
<table>
<tr><th>OFC</th><th>OFA</th><th>Unit</th><th>Range</th></tr>
<tr><td>0</td><td>1</td><td>0.01mm</td><td>±9999.99mm</td></tr>
<tr><td>0</td><td>0</td><td>0.001mm</td><td>±9999.999mm</td></tr>
<tr><td>1</td><td>0</td><td>0.0001mm</td><td>±9999.9999mm</td></tr>
</table>
Inch input
<table>
<tr><th>OFC</th><th>OFA</th><th>Unit</th><th>Range</th></tr>
<tr><td>0</td><td>1</td><td>0.001inch</td><td>±999.999inch</td></tr>
<tr><td>0</td><td>0</td><td>0.0001inch</td><td>±999.9999inch</td></tr>
<tr><td>1</td><td>0</td><td>0.00001inch</td><td>±999.99999inch</td></tr>
</table></td></tr>
<tr><td>Automatic conversion of tool compensation values upon inch/metric switch</td><td>- Make a selection using bit 0 (OIM) of parameter No. 5006.

<u>Bit 0 (OIM) of parameter No. 5006</u>
Upon inch/metric switch, automatic conversion of tool compensation values is:
0: Not performed.
1: Performed.
If the setting of this parameter is changed, set the tool compensation data again.</td><td>- Bit 0 (OIM) of parameter No. 5006 is not available.
Tool compensation values are always converted automatically.</td></tr>
</table>

B.13.2 Differences in Diagnosis Display

None.

B.14 CUSTOM MACRO

B.14.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Keep-type common variable (#500 to #999)	- The default value is <null>.	- The default value is 0.
	- The Series 0*i*-D function (described at right) is not available.	- The range specified by parameter Nos. 6031 and 6032 can be made write-protected (read-only).

Function	Series 0*i*-C	Series 0*i*-D
System variable to read machine coordinates #5021 to #5025	- Machine coordinates are always read in machine units (output units).	- Machine coordinates are always read in input units. Example) When the setting unit is IS-B, the input unit is the inch, the machine unit is the millimeter, and the coordinate value of the X axis (first axis) is as follows: Machine coordinate = 30.000 (mm) Since the value of #5021 is read in input units (inches), #5021 is 1.1811.
Logical operations in an if statement	- Logical operations can be used by setting 1 in bit 0 (MLG) of parameter No. 6006. **Bit 0 (MLG) of parameter No. 6006** In an if statement in a custom macro, logical operations: 0: Cannot be used. (P/S alarm No. 114 is issued.) 1: Can be used.	- Bit 0 (MLG) of parameter No. 6006 is not available. Logical operations can always be used.
Behavior of the GOTO statement when a sequence number is not found at the start of the block	- The command after the sequence number of the block (to the right of the sequence number) is executed.	- If a move command is specified before the sequence number (left side), alarm PS0128 is issued. If no move command is specified before the sequence number (left side), a block containing a sequence number is executed from the beginning.
	* Use a sequence number at the start of a block.	
Behavior of "GOTO 0" when there is a sequence number	- The program jumps to the block containing the sequence number.	- No jump occurs. Alarm PS1128 is issued.
	* Do not use a sequence number.	
When another NC command is found in a G65 block or in an M code block where a macro is called by an M code Example) G01 X100. G65 P9001 ;	- In a program like the one shown in the example, G01 changes the G code group to 01, while the move command X100. is not executed. X100. is regarded as an argument of G65.	- A program like the one shown in the example cannot be executed. Alarm PS0127 is issued. A G65 code or an M code that calls a macro must be specified at the beginning of a block (before all other arguments).

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Behavior when subprogram call using an M code and subprogram call using an T code are done</td><td colspan="2">- When the machine is run under the conditions and program described below:
[Conditions]
- Subprogram call by T code is enabled (bit 5 (TCS) of parameter No. 6001 is set to 1).
- The M code that calls subprogram No. 9001 is M06 (parameter No. 6071 is set to 6).
[Program]
O0001;
T100; (1)
M06 T200; (2)
T300 M06; (3)
M30;
%</td></tr>
<tr><td>In FS0i-C, blocks (1) to (3) of the program causes the machine to behave as follows:
1) Calls and executes O9000.
2) Outputs T200 and waits for FIN. Upon receipt of the FIN signal, the machine calls and executes O9001.
3) Outputs T300 and waits for FIN. Upon receipt of the FIN signal, the machine calls and executes O9001.</td><td>In FS0i-D, blocks (1) to (3) of the program causes the machine to behave as follows:
1) Calls and executes O9000.
2) Issues alarm PS1091.
3) Issues alarm PS1091 (when the program is run with block (2) deleted).</td></tr>
<tr><td rowspan="2">Block containing "M98 Pxxxx" or "M99" without any addresses other than O, N, P, and L</td><td>- Bit 4 (NPS) of parameter No. 3450 is not available. The block is always treated as a macro statement. (Single block stop is not performed.)</td><td>- Bit 4 (NPS) of parameter No. 3450 is not available. The block is always treated as a macro statement. (Single block stop is not performed.)</td></tr>
<tr><td colspan="2">* For details about macro and NC statements, refer to Section 14.5, "MACRO STATEMENTS AND NC STATEMENTS", in "OPERATOR'S MANUAL" (B-64304EN).</td></tr>
<tr><td>Subprogram and macro calls</td><td colspan="2">- The call nesting level differs as follows.
<table>
<tr><th rowspan="2">Model
Call method</th><th colspan="2">Series 0i-C</th><th colspan="2">Series 0i-D</th></tr>
<tr><th>Independent nesting level</th><th>Total</th><th>Independent nesting level</th><th>Total</th></tr>
<tr><td>Macro call (G65/G66)</td><td>4 in all</td><td rowspan="2">(G65/G66/M98) 8 in all</td><td>5 in all</td><td rowspan="2">(G65/G66/M98) 15 in all</td></tr>
<tr><td>Subprogram call (M98)</td><td>4</td><td>10</td></tr>
</table></td></tr>
<tr><td>Local variable clear operation by reset</td><td>- Make a selection using bit 7 (CLV) of parameter No. 6001.

<u>Bit 7 (CLV) of parameter No. 6001</u>
When reset, the local variables in the custom macro are:
0: Cleared to <null>.
1: Not cleared.</td><td>- Bit 7 (CLV) of parameter No. 6001 is not available.
Local variables are always cleared to <null> when reset.</td></tr>
</table>

B.14.2 Differences in Diagnosis Display

None.

B.14.3 Miscellaneous

Series 0*i*-D allows you to customize the specifications related to the maximum and minimum variable values and accuracy by using bit 0 (F0C) of parameter No. 6008. When 1 is set in bit 0 (F0C) of parameter No. 6008, the specifications are the same as Series 0*i*-C. For details, refer to Section II-14, "CUSTOM MACRO", in "OPERATOR'S MANUAL" (B-64304EN).

B.15 INTERRUPTION TYPE CUSTOM MACRO

B.15.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Interruption type custom macro in DNC operation	- Not available.	- Available.
Program restart	- When an interruption type custom macro is executed during return operation in dry run after search operation invoked by program restart:	
	The interruption type custom macro is executed after all axes have restarted.	Alarm DS0024 is issued.

B.15.2 Differences in Diagnosis Display

None.

B.16 PROGRAMMABLE PARAMETER INPUT (G10)

B.16.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Parameter input mode setting	- Specify G10 L50.	- Specify G10 L52.

B.16.2 Differences in Diagnosis Display

None.

B.17 AI ADVANCED PREVIEW CONTROL /AI CONTOUR CONTROL

B.17.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Function name	Some function names have been changed as follows.	
	- Automatic corner deceleration	- Speed control based on the feedrate difference on each axis
	- Arc radius-based feedrate clamp	- Speed control with acceleration in circular interpolation

Function	Series 0*i*-C	Series 0*i*-D
Setting to enable bell-shaped acceleration/deceleration in rapid traverse	- Setting 1 in bit 6 (RBL) of parameter No. 1603 enables bell-shaped acceleration/deceleration in rapid traverse.	- Bit 6 (RBL) of parameter No. 1603 is not available. Bell-shaped acceleration/deceleration in rapid traverse is enabled by setting the time constant of bell-shaped acceleration/deceleration after interpolation in rapid traverse in parameter No. 1621 or the acceleration change time of bell-shaped acceleration/deceleration before interpolation in rapid traverse in parameter No. 1672.
Selection of acceleration/deceleration before interpolation in rapid traverse or acceleration/deceleration after interpolation in rapid traverse	- The combination of bit 1 (AIR) of parameter No. 7054 and bit 1 (LRP) of parameter No. 1401 determines acceleration/deceleration before interpolation or acceleration/deceleration after interpolation.	- Bit 1 (AIR) of parameter No. 7054 is not available. The combination of bit 5 (FRP) of parameter No. 19501 and bit 1 (LRP) of parameter No. 1401 determines acceleration/deceleration before interpolation or acceleration/deceleration after interpolation. For details, refer to "PARAMETER MANUAL" (B-64310EN).
Setting of acceleration for look-ahead linear acceleration/deceleration before interpolation	- Set acceleration by specifying the maximum cutting feedrate for linear acceleration/deceleration before interpolation in parameter No. 1770 and the time to elapse before reaching the maximum cutting feedrate for linear acceleration/deceleration before interpolation in parameter No. 1771.	- Parameter Nos. 1770 and 1771 are not available. In parameter No. 1660, set the maximum permissible cutting feedrate for acceleration/deceleration before interpolation for each axis.
Time constant setting of linear/bell-shaped acceleration/deceleration after interpolation in cutting feed common to all axes	- Set the value in parameter No. 1768.	- Parameter No. 1768 is not available. Set the time constant for each axis in parameter No. 1769.
Time constant setting of exponential acceleration/deceleration after interpolation in cutting feed for each axis	- Set the value in parameter No. 1762. (To set the value for linear or bell-shaped acceleration/deceleration, use parameter No. 1769.)	- Parameter No. 1762 is not available. Set the value in parameter No. 1769. (Use parameter No. 1769 for any acceleration/deceleration type - linear, bell-shaped, or exponential.)
Automatic corner deceleration based on angle difference	- Setting 0 in bit 4 (CSD) of parameter No. 1602 enables the function. Set the lower limit speed in parameter No. 1777 and the critical angle between the two blocks in parameter No. 1779.	- Automatic corner deceleration based on angle difference is not available. Therefore, bit 4 (CSD) of parameter No. 1602 and parameter Nos. 1777 and 1779 are not available.

Function	Series 0*i*-C	Series 0*i*-D
Permissible speed difference common to all axes for automatic corner deceleration based on angle difference (speed control based on the feedrate difference on each axis)	- Set the value in parameter No. 1780.	- Parameter No. 1780 is not available. Set the permissible speed difference for each axis in parameter No. 1783.
Setting of arc radius-based feedrate clamp (speed control with acceleration in circular interpolation)	- Set the upper limit of the feedrate and the corresponding arc radius value in parameter Nos. 1730 and 1731, respectively.	- Parameter Nos. 1730 and 1731 are not available. Set the permissible acceleration for each axis in parameter No. 1735.
Setting of the maximum cutting feedrate common to all axes	- Set the value in parameter No. 1431.	- Parameter No. 1431 is not available. Set the maximum cutting feedrate for each axis in parameter No. 1432.
Rapid traverse block overlap	- Disabled in the advanced preview control (T series), AI advanced preview control (M series), or AI contour control (M series) mode.	- Enabled only when acceleration/deceleration after interpolation is used in the advanced preview control (T series), AI advanced preview control (M series), or AI contour control (M series) mode.
Function name	Some function names have been changed as follows.	
	- Acceleration-based feedrate clamp	- Speed control with the acceleration on each axis
Setting of acceleration-based feedrate clamp (speed control with the acceleration on each axis)	- Set the permissible acceleration by specifying the time to elapse before reaching the maximum cutting feedrate in parameter No. 1785. The maximum cutting feedrate set in parameter No. 1432 is used.	- Parameter No. 1785 is not available. Set the permissible acceleration for each axis in parameter No. 1737.

Differences regarding AI contour control

Function	Series 0*i*-C	Series 0*i*-D
Time constant of acceleration/deceleration in rapid traverse in the AI contour control mode	- Set parameter Nos. 1773 and 1774. If these parameters are not set, parameter Nos. 1620 and 1621 are used.	- Parameter Nos. 1773 and 1774 are not available. In the case of acceleration/deceleration before interpolation in rapid traverse, set parameter Nos. 1660 and 1672. In the case of acceleration/deceleration after interpolation in rapid traverse, set parameter Nos. 1620 and 1621.
Setting to enable look-ahead bell-shaped acceleration/deceleration before interpolation	- Setting 1 in bit 7 (BEL) of parameter No. 1603 enables bell-shaped acceleration/deceleration before interpolation.	- Bit 7 (BEL) of parameter No. 1603 is not available. Setting the acceleration change time of bell-shaped acceleration/deceleration before interpolation in parameter No. 1772 enables bell-shaped acceleration/deceleration before interpolation.

B.17.2 Differences in Diagnosis Display

None.

B.18 MACHINING CONDITION SELECTION FUNCTION

B.18.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Parameters set by "acceleration/deceleration before interpolation" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1770] Maximum cutting feedrate in linear acceleration/deceleration before interpolation [Parameter No. 1771] Time before the maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1770) is reached	- The following parameters are set according to the precision level: [Parameter No. 1660] Maximum permissible cutting feedrate in acceleration/deceleration before interpolation on each axis (Series 0*i*-D does not have parameter Nos. 1770 and 1771.)
Parameter 1 set by "permissible acceleration" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1730] Upper limit of the feedrate by arc radius-based feedrate clamp [Parameter No. 1731] Arc radius corresponding to the upper limit of the feedrate by arc radius-based feedrate clamp (parameter No. 1730)	- The following parameters are set according to the precision level: [Parameter No. 1735] Permissible acceleration in speed control with acceleration in circular interpolation (Series 0*i*-D does not have parameter Nos. 1730 and 1731. Also, "arc radius-based feedrate clamp" has been renamed "speed control with acceleration in circular interpolation".)
Parameter 2 set by "permissible acceleration" (machining parameter adjustment screen)	- The following parameters are set according to the precision level: [Parameter No. 1432] Maximum cutting feedrate [Parameter No. 1785] Time before the maximum cutting feedrate (parameter No. 1432) is reached (Set this to determine the permissible acceleration for acceleration-based feedrate clamp.)	- The following parameters are set according to the precision level: [Parameter No. 1737] Permissible acceleration for speed control with the acceleration on each axis (Series 0*i*-D does not have parameter No. 1785. Also, "acceleration-based feedrate clamp" has been renamed "speed control with the acceleration on each axis".)

B.18.2 Differences in Diagnosis Display

None.

B.19 AXIS SYNCHRONOUS CONTROL

B.19.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Function name	- Quick synchronous control	- Axis synchronous control
Setting to perform synchronous operation all the time	- Not available.	- Depends on bit 5 (SCA) of parameter No. 8304 for the slave axis. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 5 (SCA) of parameter No. 8304** In axis synchronous control: 0: Synchronous operation is performed when the axis synchronous control selection signal SYNCx or axis synchronous control manual feed selection signal SYNCJx for the slave axis is set to "1". 1: Synchronous operation is performed all the time. Synchronous operation is performed regardless of the setting of the SYNCx or SYNCJx signal.
Setting to move multiple slave axes in synchronism with the master axis	- Not available.	- Available. This is possible by setting the same master axis number in parameter No. 8311 for the multiple slave axes.
Setting of the same name for the master and slave axes	- The same name cannot be set for the master and slave axes.	- The same name can be set for the master and slave axes. In that case, however, automatic operation cannot be performed in normal operation; only manual operation is allowed. (No alarm is caused even if an attempt to perform automatic operation is made.)
Setting of axes for which to perform simple synchronous control (axis synchronous control)	- The master axis number set in parameter No. 8311 must be smaller than the slave axis number.	- The master axis number set in parameter No. 8311 may or may not be smaller than the slave axis number.

Function	Series 0*i*-C	Series 0*i*-D
Synchronization error check based on positional difference	- The servo positional difference between the master and slave axes is monitored, and alarm PS0213 is issued if the difference exceeds the limit value set in parameter No. 8313 when the number of synchronized axis pairs is one or the limit value set in parameter No. 8323 for the master axis when the number of synchronized axis pairs is two. - The data range of parameter No. 8323 is as follows: [Data range] 0 to 32767	- The servo positional difference between the master and slave axes is monitored, and alarm DS0001 is issued if the difference exceeds the limit value set in parameter No. 8323 for the slave axis. At the same time, the signal for indicating a positional difference error alarm for axis synchronous control SYNER<F403.0> is output. Parameter No. 8313 is not available. Regardless of the number of pairs, set the limit value in parameter No. 8323. - The data range of parameter No. 8323 is as follows: [Data range] 0 to 999999999
Synchronization error check based on machine coordinates	- The machine coordinates of the master and slave axes are compared and, if the difference is greater than the value set in parameter No. 8314 for the master axis, alarm SV0407 is issued and the motor is stopped immediately. - The data range of parameter No. 8314 is as follows: [Data range] 0 to 32767	- The machine coordinates of the master and slave axes are compared and, if the difference is greater than the value set in parameter No. 8314 for the slave axis, alarm SV0005 is issued and the motor is stopped immediately. - The data range of parameter No. 8314 is as follows: [Data range] 0 or positive 9 digits of the minimum unit of data. (For IS-B, 0.0 to +999999.999)
Setting of synchronization establishment	- Synchronization establishment is enabled by setting 1 in bit 7 (SOF) of parameter No. 8301 when the number of synchronized axis pairs is one or by setting 1 in bit 7 (SOF) of parameter No. 8303 for the master axis when the number of synchronized axis pairs is two.	- Synchronization establishment is enabled by setting 1 in bit 7 (SOF) of parameter No. 8303 for the slave axis. (Bit 7 (SOF) of parameter No. 8301 is not available. Regardless of the number of pairs, set 1 in bit 7 (SOF) of parameter No. 8303.)
Timing of synchronization establishment	- Synchronization establishment is performed when: 1. Power is turned on when the absolute position detector is used. 2. Emergency stop is canceled.	- Synchronization establishment is performed when: 1. Power is turned on when the absolute position detector is used. 2. Manual reference position return operation is performed. 3. The state of servo position control is changed from off to on. (This occurs when emergency stop, servo alarm, servo off, etc. is canceled. Note, however, that synchronization establishment is not performed at the time of axis removal cancellation.)

Function	Series 0*i*-C	Series 0*i*-D
Maximum compensation for synchronization	- Set the value in parameter No. 8315 when the number of synchronized axis pairs is one or in parameter No. 8325 for the master axis when the number of synchronized axis pairs is two. If the compensation amount exceeds the values set in the relevant parameter, alarm SV0410 occurs. - The data unit and data range of parameter Nos. 8315 and 8325 are as follows: [Data unit] Detection unit [Data range] 0 to 32767	- Set the value in parameter No. 8325 for the slave axis. If the compensation amount exceeds the values set in this parameter, alarm SV0001 occurs. (Parameter No. 8315 is not available. Regardless of the number of pairs, set the value in parameter No. 8325.) - The data unit and data range of parameter No. 8325 are as follows: [Data unit] Machine unit [Data range] 0 or positive 9 digits of the minimum unit of data. (For IS-B, 0.0 to +999999.999)
Automatic setting for grid position matching	- Enable automatic setting for grid position matching by setting 1 in bit 0 (ATE) of parameter No. 8302 when the number of synchronized axis pairs is one or in bit 0 (ATE) of parameter No. 8303 when the number of synchronized axis pairs is two. - Start automatic setting for grid position matching by setting 1 in bit 1 (ATS) of parameter No. 8302 when the number of synchronized axis pairs is one or in bit 1 (ATS) of parameter No. 8303 when the number of synchronized axis pairs is two.	- Set 1 in bit 0 (ATE) of parameter No. 8303 for the slave axis to enable automatic setting for grid position matching. (Bit 0 (ATE) of parameter No. 8302 is not available. Regardless of the number of pairs, set the value in bit 0 (ATE) of parameter No. 8303.) - Set 1 in bit 1 (ATS) of parameter No. 8303 for the slave axis to start automatic setting for grid position matching. (Bit 1 (ATS) of parameter No. 8302 is not available. Regardless of the number of pairs, set the value in bit 1 (ATS) of parameter No. 8303.)
Difference between the master axis reference counter and slave axis reference counter obtained through automatic setting for grid positioning	- Set the value in parameter No. 8316 when the number of synchronized axis pairs is one or in parameter No. 8326 for the master axis.	- Set the value in parameter No. 8326 for the slave axis. (Parameter No. 8316 is not available. Regardless of the number of pairs, set the value in parameter No. 8326.)
Time from the servo preparation completion signal SA <F000.6> being set to 1 until torque difference alarm detection is started	- Set the value in parameter No. 8317 when the number of synchronized axis pairs is one or in parameter No. 8327 for the master axis when the number of synchronized axis pairs is two.	- Set the value in parameter No. 8327 for the slave axis. (Parameter No. 8317 is not available. Regardless of the number of pairs, set the value in parameter No. 8327.)

Function	Series 0*i*-C	Series 0*i*-D
Setting to use the external machine coordinate system shift function for the slave axis	- When 1 is set in bit 3 (SSE) of parameter No. 8302, setting an external machine coordinate system shift for the master axis causes the slave axis to shift as well. This parameter is used for all the pairs.	- Bit 3 (SSE) of parameter No. 8302 is not available. By setting 1 in bit 7 (SYE) of parameter No. 8304 for the slave axis, the slave axis is shifted as well when an external machine coordinate system shift is set for the corresponding master axis. This parameter is used individually for each slave axis.
Setting to prevent slave axis movement from being added to the actual feedrate display	- Setting 1 in bit 7 (SMF) of parameter No. 3105 prevents slave axis movement from being added to the actual feedrate display. This parameter is used for all the pairs.	- Bit 7 (SMF) of parameter No. 3105 is not available. Setting 0 in bit 2 (SAF) of parameter No. 8303 prevents slave axis movement from being added to the actual feedrate display. (Note that the meaning of the value is the opposite from bit 7 (SMF) of parameter No. 3105.) This parameter is used individually for each slave axis.
Change of the synchronization state during a program command	- Specify an M code that is not to be buffered. Using this M code, change the input signal - SYNCx<G138> or SYNCJx<G140> - from the PMC side.	- Specify an M code that changes the synchronization state (parameter No. 8337 or 8338). By changing the input signal - SYNCx<G138> or SYNCJx<G140> - from the PMC side using this M code, it is possible to change the synchronization state during a program command. **Parameter No. 8337** Specify an M code that changes synchronous operation to normal operation. **Parameter No. 8338** Specify an M code that changes normal operation to synchronous operation.
Automatic slave axis parameter setting	- This function is enabled by setting 1 in bit 4 (SYP) of parameter No. 8303 for the master axis.	- Bit 4 (TRP) of parameter No. 12762 is not available. This function is enabled by setting 1 in bit 4 (SYP) of parameter No. 8303 for the master and slave axes.
Mirror image for the slave axis	- A mirror image cannot be applied to a slave axis during simple synchronous control. It can be applied only to the T series.	- By setting parameter No. 8312 for the slave axis, a mirror image can be applied to a slave axis during simple synchronous control. **Parameter No. 8312** This parameter sets mirror image for the slave axis. When 100 or a more value is set with this parameter, the mirror image function is applied to synchronous control.

Function	Series 0*i*-C	Series 0*i*-D
Setting to cancel the check of positional difference between the master and slave axes during synchronization establishment	- Depends on bit 5 (SYE) of parameter No. 8301. **Bit 5 (SYE) of parameter No. 8301** During synchronization establishment, the positional difference limit is: 0: Checked. 1: Not checked.	- Not available. Therefore, bit 5 (SYE) of parameter No. 8301 is not available. Since the positional difference is always checked, parameter No. 8318 is not available, either. **Parameter No. 8318** Set the time from the synchronization establishment function outputting a compensation pulse to the slave axis until the check of the positional difference limit between the master and slave axes starts.

B.19.2 Differences in Diagnosis Display

Item	Series 0*i*-C	Series 0*i*-D
Positional difference between the master and slave axes	- This item is displayed in diagnosis No. 540 for the master axis when the number of synchronized axis pairs is one or in diagnosis No. 541 for the master axis when the number of synchronized axis pairs is two.	- This item is displayed in diagnosis No. 3500 for the slave axis. (Regardless of the number of pairs, the item is displayed in diagnosis No. 3500.)

B.20 ARBITRARY ANGULAR AXIS CONTROL

B.20.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Angular and perpendicular axes when an invalid value is set in parameter No. 8211 or 8212</td><td colspan="2">
<table>
<tr><th rowspan="2"></th><th colspan="2">Series 0i-C</th><th colspan="2">Series 0i-D</th></tr>
<tr><th>Angular axis</th><th>Perpendicular axis</th><th>Angular axis</th><th>Perpendicular axis</th></tr>
<tr><td>M series</td><td>Y axis (2nd axis)</td><td>Z axis (3rd axis)</td><td>Y-axis of the basic three axes (axis with 2 set in parameter No. 1022)</td><td>Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)</td></tr>
</table>
</td></tr>
<tr><td>Reference position return completion signal ZP for the perpendicular axis moved with the angular axis <Fn094, Fn096, Fn098, Fn100></td><td>- Select the signal using bit 3 (AZP) of parameter No. 8200.
When the bit is set to 0, ZP is not set to "0". (The signal is not cleared.)
When the bit is set to 1, ZP is set to "0". (The signal is cleared.)</td><td>- Bit 3 (AZP) of parameter No. 8200 is not available.
ZP is always set to "0". (The signal is cleared.)</td></tr>
<tr><td>When an angular axis is specified individually in machine coordinate system selection (G53) during arbitrary angular axis control</td><td>- Select the perpendicular axis operation using bit 6 (A53) of parameter No. 8201.
When the bit is set to 0, the perpendicular axis is also moved.
When the bit is set to 1, only the angular axis is moved.</td><td>- Bit 6 (A53) of parameter No. 8201 is not available.
Only the angular axis is always moved.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
G30 command during arbitrary angular axis control	- Select the operation using bit 0 (A30) of parameter No. 8202. When the bit is set to 0, the operation is for the perpendicular coordinate system. When the bit is set to 1, the operation is for the angular coordinate system.	- Bit 0 (A30) of parameter No. 8202 is not available. The operation is always for the angular coordinate system.

B.20.2 Differences in Diagnosis Display

None.

B.21 RUN HOUR AND PARTS COUNT DISPLAY

B.21.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Data range of the M code that counts the number of machined parts	**Parameter No. 6710** The data range of the M code that counts the number of machined parts is as follows.	
	- 0 to 255	- 0 to 99999999 (8 digits)
Data range of the number of parts required	**Parameter No. 6713** The data range of the number of parts required is as follows.	
	- 0 to 9999	- 0 to 999999999 (9 digits)
Data range of the number and total number of parts machined	**Parameter No. 6711** Number of parts machined	**Parameter No. 6712** Total number of parts machined
	The data range is as follows.	
	- 0 to 99999999 (8 digits)	- 0 to 999999999 (9 digits)
Data range of the power-on period, time during automatic operation, cutting time, input signal TMRON on time, and one automatic operation time	**Parameter No. 6750** Integrated value of power-on period **Parameter No. 6752** Integrated value of time during automatic operation **Parameter No. 6756** Integrated value of time when input signal TMRON (G053.0) is on	**Parameter No. 6754** Integrated value of cutting time **Parameter No. 6758** Integrated value of one automatic operation time
	The data range is as follows.	
	- 0 to 99999999 (8 digits)	- 0 to 999999999 (9 digits)

B.21.2 Differences in Diagnosis Display

None.

B.22 MANUAL HANDLE FEED

B.22.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Handle pulses exceeding the rapid traverse rate</td><td colspan="2">If manual handle feed exceeding the rapid traverse rate is specified, whether to ignore or accumulate handle pulses exceeding the rapid traverse feedrate can be set as follows.</td></tr>
<tr><td>- Depends on bit 4 (HPF) of parameter No. 7100. The amount of pulses to be accumulated is set in parameter No. 7117.</td><td>- Bit 4 (HPF) of parameter No. 7100 is not available. Whether to ignore or accumulate excess handle pulses is determined by the amount to be accumulated that is set in parameter No. 7117.
[When parameter No. 7117 = 0]
Ignored.
[When parameter No. 7117 > 0]
Accumulated in the CNC without being ignored.</td></tr>
<tr><td>Permissible amount of pulses for manual handle feed</td><td>- The value range of parameter No. 7117 is 0 to 99999999 (8 digits).</td><td>- The value range of parameter No. 7117 is 0 to 999999999 (9 digits).</td></tr>
<tr><td rowspan="5">Value range of the magnification parameter for manual handle feed</td><td>- For parameter Nos. 7113, 7131, 7133, and 12350, magnification ranges from 1 to 127.
For parameter Nos. 7114, 7132, 7134, and 12351, magnification ranges from 1 to 1000.</td><td>- For parameter No. 7113, 7114, 7131, 7132, 7133, 7134, 12350, and 12351, magnification ranges from 1 to 2000.</td></tr>
<tr><td colspan="2"><u>Parameter No. 7113</u>
Magnification when manual handle feed amount selection signals MP1 = 0 and MP2 = 1
<u>Parameter No. 7114</u>
Magnification when manual handle feed amount selection signals MP1 = 1 and MP2 = 1
[When bit 5 (MPX) of parameter No. 7100 = 0]
Magnification common to all the generators in the path
[When bit 5 (MPX) of parameter No. 7100 = 1]
Magnification used by the first generator in the path</td></tr>
<tr><td colspan="2"><u>Parameter No. 7131</u>
Magnification when manual handle feed amount selection signals MP21 = 0 and MP22 = 1
<u>Parameter No. 7132</u>
Magnification when manual handle feed amount selection signals MP21 = 1 and MP22 = 1
When bit 5 (MPX) of parameter No. 7100 is set to 1, the magnification used by the second generator in the path applies.</td></tr>
<tr><td colspan="2"><u>Parameter No. 7133</u>
Magnification when manual handle feed amount selection signals MP31 = 0 and MP32 = 1
<u>Parameter No. 7134</u>
Magnification when manual handle feed amount selection signals MP31 = 1 and MP32 = 1
When bit 5 (MPX) of parameter No. 7100 is set to 1, the magnification used by the third generator in the path applies.</td></tr>
<tr><td colspan="2"><u>Parameter No. 12350</u>
Magnification when per-axis manual handle feed amount selection signals MP1 = 0 and MP2 = 1
<u>Parameter No. 12351</u>
Magnification when per-axis manual handle feed amount selection signals MP1 = 1 and MP2 = 1</td></tr>
<tr><td>Number of manual pulse generators used</td><td>- Set the value in parameter No. 7110.</td><td>- Parameter No. 7110 is not available. Up to two generators can be used without setting the parameter.</td></tr>
</table>

B.22.2 Differences in Diagnosis Display

None.

B.23 PMC AXIS CONTROL

B.23.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Relationship with synchronous control (synchronous control of synchronous/composite control)</td><td>- PMC axis control can be applied for any axis other than a synchronous slave axis.</td><td>- PMC axis control cannot be applied for any axis under synchronous control.</td></tr>
<tr><td>Relationship with the feed-forward and advanced preview feed-forward functions</td><td>- Enable or disable the functions by using bit 7 (NAH) of parameter No. 1819, bit 3 (G8C) of parameter No. 8004, and bit 4 (G8R) of parameter No. 8004 in combination.</td><td>- Neither the feed-forward nor advanced preview feed-forward function is available for an axis under PMC axis control.
Bit 3 (G8C) and bit 4 (G8R) of parameter No. 8004 are not available.</td></tr>
<tr><td>Data range of rapid traverse rate for rapid traverse (00h), 1st to 4th reference position return (07h to 0Ah), and machine coordinate system selection (20h)</td><td>- The data range is as follows.
<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Valid data range</th><th rowspan="2">Unit of data</th></tr>
<tr><th>IS-A, IS-B</th><th>IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Millimeter machine</td><td>30 to 15000</td><td>30 to 12000</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>30 to 6000</td><td>30 to 4800</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>30 to 15000</td><td>30 to 12000</td><td>deg/min</td></tr>
</table></td><td>- 1 to 65535
The data unit is as follows.
<table>
<tr><th colspan="2" rowspan="2"></th><th>Data unit</th><th rowspan="2">Unit</th></tr>
<tr><th>IS-A to IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Metric machine</td><td>1</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>0.1</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>1</td><td>deg/min</td></tr>
</table></td></tr>
<tr><td>Data range of total moving distance for rapid traverse (00h), cutting feed - feed per minute (01h), cutting feed - feed per revolution (02h), and skip - feed per minute (03h)</td><td>- The data range is as follows.
<table>
<tr><th>Input increment</th><th>IS-B</th><th>IS-C</th><th>Unit</th></tr>
<tr><td>mm unit input
deg unit input</td><td>±99999.999</td><td>±9999.9999</td><td>mm
deg</td></tr>
<tr><td>inch unit input</td><td>±9999.9999</td><td>±999.99999</td><td>inch</td></tr>
</table></td><td>- The data range is as follows.
<table>
<tr><th>IS-A</th><th>IS-B,IS-C</th></tr>
<tr><td>-99999999 to 99999999 (8 digits)</td><td>-999999999 to 999999999 (9 digits)</td></tr>
</table>
The data unit is the minimum setting unit for the corresponding axis. (See the table below.)
<table>
<tr><th>Setting unit</th><th>Minimum data unit</th></tr>
<tr><td>IS-A</td><td>0.01</td></tr>
<tr><td>IS-B</td><td>0.001</td></tr>
<tr><td>IS-C</td><td>0.0001</td></tr>
</table></td></tr>
<tr><td>Data range of cutting feedrate for rapid traverse (01h) and skip - feed per minute (03h)</td><td>- 1 to 65535
The specified feedrate must be within the range shown in the table below.
<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Valid data range</th><th rowspan="2">Unit of data</th></tr>
<tr><th>IS-B</th><th>IS-C</th></tr>
<tr><td rowspan="2">Linear axis</td><td>Millimeter machine</td><td>1 to 100000</td><td>0.1 to 12000.0</td><td>mm/min</td></tr>
<tr><td>Inch machine</td><td>0.01 to 4000.00</td><td>0.01 to 480.000</td><td>inch/min</td></tr>
<tr><td colspan="2">Rotation axis</td><td>1 to 100000</td><td>0.1 to 12000.0</td><td>deg/min</td></tr>
</table></td><td>- 1 to 65535</td></tr>
<tr><td>Function to increase the specification unit by a factor of 200 for continuous feed (06h)</td><td>- Not available.</td><td>- By setting 1 in bit 2 (JFM) of parameter No. 8004, it is possible to increase the specification unit by a factor of 200.

<u>Bit 2 (JFM) of parameter No. 8004</u>
Set the specification unit of feedrate data for specifying the continuous feed command for PMC axis control.
<table>
<tr><th>Increment system</th><th>Bit 2 (JFM) of No. 8004</th><th>Millimeter input (mm/min)</th><th>Inch input (inch/min)</th><th>Rotation axis (min^{-1})</th></tr>
<tr><td rowspan="2">IS-B</td><td>0</td><td>1</td><td>0.01</td><td>0.00023</td></tr>
<tr><td>1</td><td>200</td><td>2.00</td><td>0.046</td></tr>
<tr><td rowspan="2">IS-C</td><td>0</td><td>0.1</td><td>0.001</td><td>0.000023</td></tr>
<tr><td>1</td><td>20</td><td>0.200</td><td>0.0046</td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Maximum feedrate for continuous feed (06h)</td>
<td>- When an override of 254% is applied
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr>
<tr><td>1 time</td><td>166458 mm/min</td><td>1664.58 inch/min</td><td>16645 mm/min</td><td>166.45 inch/min</td></tr>
<tr><td>10 times</td><td>1664589 mm/min</td><td>16645.89 inch/min</td><td>166458 0mm/min</td><td>1664.58 inch/min</td></tr>
</table>
- When override is canceled
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input</th><th>Inch input</th><th>Metric input</th><th>Inch input</th></tr>
<tr><td>1 time</td><td>65535 mm/min</td><td>655.35 inch/min</td><td>6553 mm/min</td><td>65.53 inch/min</td></tr>
<tr><td>10 times</td><td>655350 mm/min</td><td>6553.50 inch/min</td><td>65535 mm/min</td><td>655.35 inch/min</td></tr>
</table></td>
<td>- When an override of 254% is applied
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th></tr>
<tr><td>1 time</td><td>166458</td><td>1664.58</td><td>16645</td><td>166.46</td></tr>
<tr><td>10 times</td><td>999000</td><td>16645.89</td><td>99900</td><td>1664.58</td></tr>
<tr><td>200 times</td><td>999000</td><td>39330.0</td><td>99900</td><td>3933.0</td></tr>
</table>
- When override is canceled
<table>
<tr><th></th><th colspan="2">IS-B</th><th colspan="2">IS-C</th></tr>
<tr><th></th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th><th>Metric input (mm/min)</th><th>Inch input (inch/min)</th></tr>
<tr><td>1 time</td><td>65535</td><td>655.35</td><td>6553</td><td>65.53</td></tr>
<tr><td>10 times</td><td>655350</td><td>6553.5</td><td>65535</td><td>655.35</td></tr>
<tr><td>200 times</td><td>999000</td><td>39330.0</td><td>999000</td><td>3933.0</td></tr>
</table></td></tr>
<tr><td rowspan="2">Minimum unit of feedrate for the speed command (10h)</td>
<td colspan="2">The minimum unit of feedrate is given by the expressions shown below. The value must be specified as an integer. No finer value may be specified.
A calculation is made according to IS-B.
Fmin: Minimum feedrate unit
P: Number of pulses per revolution of a detector for speed feedback</td></tr>
<tr><td>- Fmin = P ÷ 7500 (mm/min)</td><td>- Fmin = P ÷ 1000 (mm/min)</td></tr>
<tr><td rowspan="2">Speed specification in the speed command (10h)</td>
<td colspan="2">A speed is specified according to the expressions shown below.
A calculation is made according to IS-B.
F: Speed command (integer)
N: Servo motor speed (min^{-1})
P: Number of pulses per revolution of a detector for speed feedback</td></tr>
<tr><td>- F = N × P ÷ 7500 (mm/min)</td><td>- F = N × P ÷ 1000 (mm/min)</td></tr>
<tr><td>Setting range of torque data for torque control (11h)</td>
<td>- The setting range is as follows.
<table>
<tr><th>Valid data range</th><th>Unit</th></tr>
<tr><td>-99999999 to +99999999</td><td>0.0000 1 Nm</td></tr>
</table></td>
<td>- The setting range is as follows.
<table>
<tr><th>Valid data range</th><th>Unit</th></tr>
<tr><td>-999999999 to +999999999 (9 digits)</td><td>0.0000 1 Nm</td></tr>
</table></td></tr>
<tr><td>Note on executing an absolute command from the program for an axis subject to PMC axis control during automatic operation</td>
<td colspan="2">- [For Series 0i-D]
When you switch to PMC axis control to execute a move command during automatic operation and then switch back to NC axis control to execute an absolute command from the program for the moved axis, that PMC command needs to be executed using a non-buffering M code.

For example, when an absolute command is executed in a N40 block after PMC control is applied to Y axis, as in the example below, PMC axis control needs to be executed in a non-buffering M code (N20 block).

O0001 ;
N10 G94 G90 G01 X20. Y30. F3000 ;
N20 M55 ; → Executes PMC axis control for the Y axis.
N30 X70. ;
N40 Y50. ;
N50 M30 ;

Execute PMC axis control as follows.
1. After the output of the auxiliary function strobe signal MF for M55, start PMC axis control.
2. Upon completion of PMC axis control, input the completion signal FIN for M55.
- [For Series 0i-C]
Control does not need to be executed using a non-buffering M code.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Acceleration/deceleration control for an axis synchronized with external pulses using external pulse synchronization (0Bh, 0Dh to 0Fh)	- Depends on bit 2 (SUE) of parameter No. 8002. **Bit 2 (SUE) of parameter No. 8002** With the external pulse synchronization command for PMC axis control, the acceleration/deceleration of the axis synchronized with external pulses is: 0: Controlled (exponential acceleration/deceleration). 1: Not controlled.	- Bit 2 (SUE) of parameter No. 8002 is not available. The acceleration/deceleration of the axis synchronized with external pulses is controlled (exponential acceleration/deceleration).
Inch/metric conversion for a linear axis controlled only by PMC axis control	- Depends on bit 0 (PIM) of parameter No. 8003. **Bit 0 (PIM) of parameter No. 8003** When the axis controlled only by PMC axis control (see parameter No. 1010) is a linear axis, inch/metric input: 0: Influences the axis. 1: Does not influence the axis.	- Bit 0 (PIM) of parameter No. 8003 is not available. Parameter No. 1010 is not available, either. For a linear axis controlled only by PMC axis control, set rotation axis type B (set 1 in both bit 1 and bit 0 of parameter No. 1006) to avoid the influence of inch/metric input.
Setting to change all axes to CNC axes or PMC axes	- Depends on bit 1 (PAX) of parameter No. 8003. **Bit 1 (PAX) of parameter No. 8003** When 0 is set as the number of CNC control axes (parameter No. 1010), all axes are changed to: 0: CNC axes. 1: PMC axes.	- Bit 1 (PAX) of parameter No. 8003 is not available. Parameter No. 1010 is not available, either. There is no parameter to change all axes to PMC axes.
If the PMC issues an axis control command for an axis when the tool is waiting for the auxiliary function completion signal after moving that axis according to a move command and an auxiliary function specified from the CNC side	- Depends on bit 0 (CMV) of parameter No. 8004. **Bit 0 (CMV) of parameter No. 8004** If the PMC issues an axis control command for an axis when the tool is waiting for the auxiliary function completion signal after moving that axis according to a move command and an auxiliary function specified from the CNC side: 0: Alarm PS0130 is issued. 1: The axis control command from the PMC side is executed.	- Bit 0 (CMV) of parameter No. 8004 is not available. The axis control command from the PMC side is executed.
If the CNC issues a command for an axis when that axis is being moved by the axis control command from the PMC side	- Depends on bit 1 (NMT) of parameter No. 8004. **Bit 1 (NMT) of parameter No. 8004** If the CNC issues a command for an axis when that axis is being moved by the axis control command from the PMC side: 0: Alarm PS0130 is issued. 1: A command that does not involve moving the axis is executed without an alarm.	- Bit 1 (NMT) of parameter No. 8004 is not available. A command that does not involve moving the axis is executed without an alarm. (If the command involves moving the axis, alarm PS0130 is issued.)

Function	Series 0*i*-C	Series 0*i*-D
Setting of diameter/radius specification for the amount of travel and feedrate when diameter programming is specified for a PMC-controlled axis	- This item is determined by using bit 7 (NDI) of parameter No. 8004 and bit 1 (CDI) of parameter No. 8005 in combination.	- Bit 7 (NDI) of parameter No. 8004 is not available. The item is determined by bit 1 (CDI) of parameter No. 8005. **Bit 1 (CDI) of parameter No. 8005** In PMC axis control, when diameter programming is specified for a PMC-controlled axis: 0: The amount of travel and feedrate are each specified with a radius. 1: The amount of travel is specified with a diameter while the feedrate is specified with a radius.
Individual output of the auxiliary function	- Depends on bit 7 (MFD) of parameter No. 8005. **Bit 7 (MFD) of parameter No. 8005** The individual output of the auxiliary function for PMC axis control function is: 0: Disabled. 1: Enabled.	- Bit 7 (MFD) of parameter No. 8005 is not available. The individual output of the auxiliary function for PMC axis control function is enabled.
Function to exert position control for the speed command (10h)	- Depends on bit 4 (EVP) of parameter No. 8005. **Bit 4 (EVP) of parameter No. 8005** The speed of PMC axis control is specified by: 0: Speed command. 1: Position command.	- Depends on bit 4 (EVP) of parameter No. 8005. Note that, for the EVP=1 setting to take effect, 1 must be set in bit 2 (VCP) of parameter No. 8007. **Bit 2 (VCP) of parameter No. 8007** The speed command in PMC axis control is: 0: FS10/11 type. 1: FS0 type.
In-position check for an axis controlled only by PMC axis control	- Depends on bit 2 (IPA) of parameter No. 8006. **Bit 2 (IPA) of parameter No. 8006** In the case of an axis controlled only by PMC axis control (see parameter No. 1010), in-position check is: 0: Performed when no move command is specified for the PMC axis. 1: Always not performed.	- Bit 2 (IPA) of parameter No. 8006 is not available. Parameter No. 1010 is not available, either. The check is performed when no move command is specified for the PMC axis. Otherwise, the processing is determined by bit 6 (NCI) of parameter No. 8004. **Bit 6 (NCI) of parameter No. 8004** When the PMC-controlled axis is decelerated, in-position check is: 0: Performed. 1: Not performed.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>No in-position check signal for a PMC-controlled axis and no in-position check signals for individual axes</td><td>- Depends on bit 0 (NIS) of parameter No. 8007.

<u>Bit 0 (NIS) of parameter No. 8007</u>
For in-position check for a PMC axis, the no in-position check signal NOINPS<G023.5> and no in-position check signals for individual axes NOINP1<G359> to NOINP5<G359> are:
0: Disabled.
1: Enabled.</td><td>- Bit 0 (NIS) of parameter No. 8007 is not available.
The no in-position check signal NOINPS<G023.5> and no in-position check signals for individual axes NOINP1<G359> to NOINP5<G359> are disabled for in-position check for a PMC axis.</td></tr>
<tr><td>Minimum speed for rapid traverse override in PMC axis control</td><td>- Set the value in parameter No. 8021.</td><td>- Parameter No. 8021 is not available.
The minimum speed for rapid traverse override cannot be set.</td></tr>
<tr><td>Operation when instructing in machine coordinate system selection (20h) to the axis to which roll-over is effective</td><td>- Depends on bit 1 (RAB) of parameter No. 1008.

<u>Bit 1 (RAB) of parameter No. 1008</u>
In the absolute commands, the axis rotates in the direction:
0: In which the distance to the target is shorter.
(Specified by the shortest path)
1: Specified by the sign of command value.</td><td>- Depends on bit 1 (RAB) of parameter No. 1008 and bit 4 (R20) of parameter No. 8013.
<table>
<tr><td></td><td></td><td colspan="2">Bit 4 (R20) of parameter No.8013</td></tr>
<tr><td></td><td></td><td>0</td><td>1</td></tr>
<tr><td rowspan="2">Bit 1 (RAB) of parameter No.1008</td><td>0</td><td>Direction of the shortest path</td><td>Direction of the shortest path</td></tr>
<tr><td>1</td><td>Direction of sign of the amount of the movement to be made</td><td>Direction of sign of command value</td></tr>
</table></td></tr>
</table>

B.23.2 Differences in Diagnosis Display

None.

B.24 EXTERNAL SUBPROGRAM CALL (M198)

B.24.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Address P format when calling a subprogram on the memory card (file number specification/program number specification)</td><td>- Depends on bit 2 (SBP) of parameter No. 3404.

<u>Bit 2 (SBP) of parameter No. 3404</u>
In the external device subprogram call M198, address P is specified using:
0: File number.
1: Program number.</td><td>- To call a subprogram, the program number must always be specified in address P.
When calling a subprogram on the memory card, the processing is not dependent on the setting of bit 2 (SBP) of parameter No. 3404.</td></tr>
<tr><td rowspan="2">Multiple call alarm</td><td colspan="2">If a subprogram called by an external subprogram call specifies a further external subprogram call, the following alarms are issued, respectively:</td></tr>
<tr><td>- Alarm PS0210</td><td>- Alarm PS1080</td></tr>
<tr><td>External subprogram call in MDI mode</td><td>- Enabled.</td><td>- Depends on bit 1 (MDE) of parameter No. 11630.

<u>Bit 1 (MDE) of parameter No. 11630</u>
In MDI mode, an external device subprogram call (M198 command) is:
0: Disabled. (Alarm PS1081 is issued.)
1: Enabled.</td></tr>
</table>

B.24.2 Differences in Diagnosis Display

None.

B.25 SEQUENCE NUMBER SEARCH

B.25.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Return from a subprogram to the calling program's block that has a specified sequence number
Sequence number search when (M99 Pxxxxx) is executed</td><td>- The calling program is searched from the beginning, and control is returned to the first block found to have sequence number Nxxxxx.</td><td>- The calling program is searched in a forward direction from the block that called the subprogram, and control is returned to the first block found to have sequence number Nxxxxx.
If the specified sequence number is not found, the calling program is searched from the beginning, and control is returned to the first block found to have sequence number Nxxxxx.</td></tr>
<tr><td colspan="2">Example) Main program
O0001 ;
N100 ; (1)
N100 ; (2)
M98 P9001 ;
N100 ; (3)
N100 ; (4)
M30 ;
Sub program
O9001 ;
M99 P100 ;
- [For Series 0i-C]
Control is returned to block (1).
- [For Series 0i-D]
Control is returned to block (3).

⚠ WARNING
Be sure to avoid writing two or more identical sequence numbers in a program. Doing so may cause the search to find unintended blocks.</td></tr>
</table>

B.25.2 Differences in Diagnosis Display

None.

B.26 STORED STROKE CHECK

B.26.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Stored stroke check immediately following powering on	- This function is always enabled for all axes.	- It is possible to select whether to enable or disable the function on an axis-by-axis basis using bit 0 (DOT) of parameter No. 1311. **Bit 0 (DOT) of parameter No. 1311** The stored stroke limit check immediately following powering on is: 0: Disabled. 1: Enabled. **NOTE** This function stores machine coordinates using software and therefore imposes a burden on the system. Disable the function for those axes that do not require it. Movements made while the power is off are not reflected on the machine coordinate system immediately after powering on.
	- Machine coordinates are set upon powering on. Absolute and relative coordinates are not set. (They are set when the absolute position detector is provided.)	- Machine coordinates are set upon powering on. Absolute and relative coordinates are set based on these machine coordinates.
Overtravel alarm	- Stored stoke check 2 does not support bit 7 (BFA) of parameter No. 1300. Therefore, if an interference alarm occurs, the tool stops after entering the prohibited area. This makes it necessary to make the prohibited area slightly larger than actually necessary.	- Stored stoke check 2 also supports bit 7 (BFA) of parameter No. 1300. Setting 1 in BFA allows the tool to stop before entering the prohibited area, thus eliminating the need to make the prohibited area slightly larger than actually necessary. **Bit 7 (BFA) of parameter No. 1300** If a stored stoke check 1, 2, or 3 alarm occurs, if an interference alarm occurs with the inter-path interference check function (T series), or if an alarm occurs with chuck/tail stock barrier (T series), the tool stops: 0: After entering the prohibited area. 1: Before entering the prohibited area.
Operation continuation after automatic alarm cancellation when a soft OT1 alarm is issued during the execution of an absolute command in automatic operation	- When the operation is resumed, the tool moves the remaining travel distance of the block that caused the soft OT. Therefore, the program can be continued if the tool is moved through manual intervention beyond the remaining travel distance.	- When the operation is resumed, the tool moves toward the end point of the block that caused the soft OT, causing another soft OT and making it impossible to continue the program. For details, refer to "STORED STROKE CHECK 1" in "CONNECTION MANUAL (FUNCTION)" (B-64303EN-1).

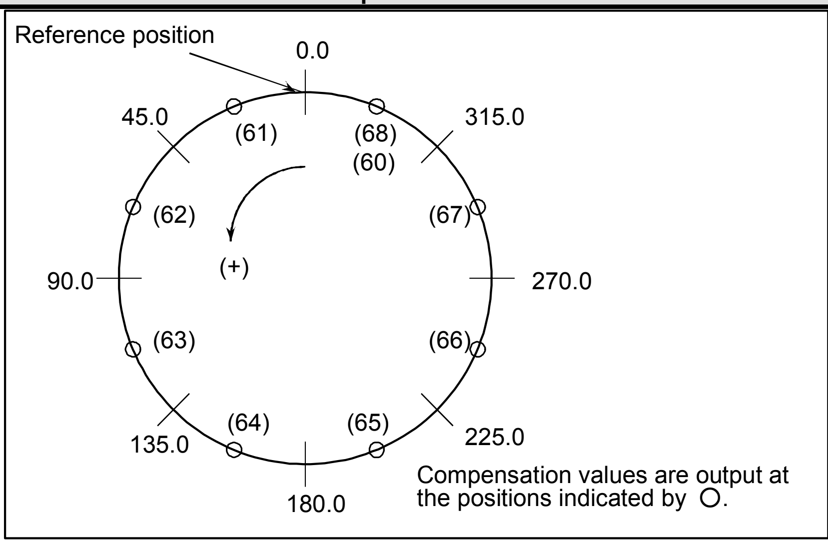
Function	Series 0*i*-C	Series 0*i*-D
Block that judges the distance to the stored stroke limit in AI advanced preview control or AI contour control mode	- A selection can be made using bit 5 (ODA) of parameter No. 7055. **Bit 5 (ODA) of parameter No. 7055** The distance to the stored stroke limit in AI advanced preview control or AI contour control mode is judged with respect to: 0: Axes specified in the current and next blocks. 1: Axes specified in the current block.	- Bit 5 (ODA) of parameter No. 7055 is not available. The distance is always judged with respect to the axes specified in the current block.

B.26.2 Differences in Diagnosis Display

None.

B.27 STORED PITCH ERROR COMPENSATION

B.27.1 Differences in Specifications

Function	Explanation
Value of parameter No. 3621 for the setting of a rotary axis (type A)	 - Amount of movement per rotation: 360° - Interval between pitch error compensation positions: 45° - Number of the compensation position of the reference position: 60 In the above case, the values of the parameters are as follows. <table><tr><th>Parameter</th><th>Series 0i-C</th><th>Series 0i-D</th></tr><tr><td>No. 3620: Number of the compensation position of the reference position</td><td>60</td><td>60</td></tr><tr><td>No. 3621: Smallest compensation position number</td><td>60</td><td>61</td></tr><tr><td>No. 3622: Largest compensation position number</td><td>68</td><td>68</td></tr><tr><td>No. 3623: Compensation magnification</td><td>1</td><td>1</td></tr><tr><td>No. 3624: Interval between compensation positions</td><td>45000</td><td>45000</td></tr><tr><td>No. 3625: Amount of movement per rotation</td><td>360000</td><td>360000</td></tr></table>The value of parameter No. 3621 is as follows. Series 0*i*-C = Number of the compensation position of the reference position (parameter No. 3620) Series 0*i*-D = Number of the compensation position of the reference position (parameter No. 3620) + 1

B.27.2 Differences in Diagnosis Display

None.

B.28 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION

B.28.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Behavior of the manual screen erasure function ("<CAN> + function key") when an alarm is issued	- When an alarm is issued (including one associated with the other path), the manual screen erasure function is enabled. ("<CAN> + function key" erases the screen.)	- When an alarm is issued (including one associated with the other path), the manual screen erasure function is disabled. ("<CAN> + function key" does not erase the screen.)
Redisplay of the screen upon mode switching	- When the operation mode is switched while the screen is erased:	
	The screen is not redisplayed. (The screen remains erased.) Please set "1" to screen clear invalidation signal *CRTOF<G0062.1> to redisplay the screen when operation mode is switched.	The screen is redisplayed.
Function key input when the screen is erased or displayed	- Select the behavior using bit 2 (NFU) of parameter No. 3209. **Bit 2 (NFU) of parameter No. 3209** When a function key is pressed to erase or display the screen for the screen erasure or automatic screen erasure function, the screen change using a function key is: 0: Performed. 1: Not performed.	- Bit 2 (NFU) of parameter No. 3209 is not available. The tool always behaves as when 1 is set in bit 2 (NFU) of parameter No. 3209.
Time before the automatic screen erasure function starts	- Set the value in parameter No.3123.	
	The value range is 1 to 255 (minutes).	The value range is 1 to 127 (minutes).
Redisplay of the screen upon external message	- When the external message is input while the screen is erased:	
	The screen is redisplayed.	The screen is not redisplayed. (The screen remains erased.) Please set "1" to screen clear invalidation signal *CRTOF<G0062.1> to redisplay the screen when external message is input.

B.28.2 Differences in Diagnosis Display

None.

B.29 RESET AND REWIND

B.29.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Modal data when reset during the execution of a block</td><td colspan="2">- If reset occurs during the execution of a block, the states of the modal G codes and modal addresses (N, F, S, T, M, etc.) specified in that block are handled as follows.</td></tr>
<tr><td>Maintained.</td><td>Not maintained. The states return to those of the modal data specified in the preceding blocks.
(The modal data is updated after the specified block is fully executed.)

Example) If reset occurs before positioning is completed in the N2 block in the program shown below, the T code and offset return to the data of the preceding tool (T0101) data.

N1 G00 X120. Z0. T0101 ;
;
N2 G00 X180. Z20. T0202 ;
;</td></tr>
<tr><td>Information in a block that is pre-read when a reset is made during an automatic operation (contents of the buffer)</td><td>- The information in the block may or may not be held depending on whether MDI mode is in progress.
<u>In MDI mode</u>
The information in the block is held.
<u>In modes other than MDI mode</u>
The information in the block is not held.</td><td>- The information in the block is not held regardless whether MDI mode is in progress.</td></tr>
</table>

B.29.2 Differences in Diagnosis Display

None.

B.30 MANUAL ABSOLUTE ON AND OFF

B.30.1 Differences in Specifications

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Absolute coordinates during automatic tool compensation change</td><td colspan="2">- If tool compensation is automatically changed when the manual absolute signal *ABSM(Gn006.2) is set to 1, absolute coordinates are handled as follows.</td></tr>
<tr><td>Absolute coordinates are not changed.</td><td>Absolute coordinates are changed according to the amount of tool compensation resulting from the coordinate shift.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Operation at manual absolute on	- When the block intervened manually ends, the tool is at the position which shifts by manual intervention. (Fig.1) (Even incremental command and absolute command, the result is the same)	- In case of incremental command and bit 1 (ABS) of parameter No. 7001 is set to 0, when the block intervened manually ends, the tool is at the position which shifts by manual intervention. (Fig.1) - In case of absolute command or bit 1 (ABS) of parameter No. 7001 is set to 1, when the block intervened manually ends, the tool is at the programmed position. (Fig.2)

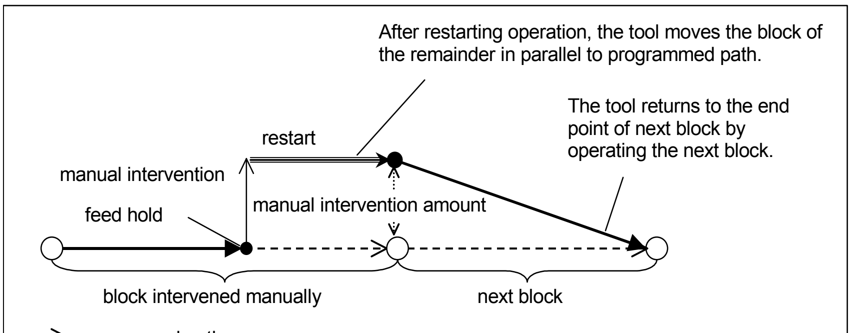


Fig. 1

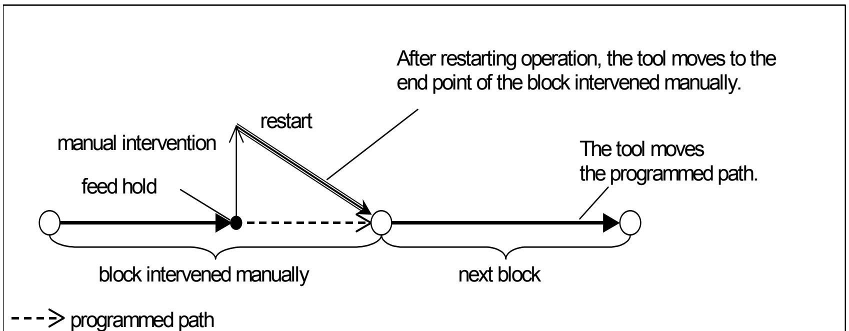


Fig. 2

B.30.2 Differences in Diagnosis Display

None.

B.31 EXTERNAL DATA INPUT

B.31.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Number of external alarm messages and message length	- [Number of messages that can be set at a time] Up to 4 messages [Length of a message] Up to 32 characters	- [Number of messages that can be set at a time] Depends on bit 1 (M16) of parameter No. 11931. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 1 (M16) of parameter No. 11931** The maximum number of external alarm messages or external operator messages that can be displayed in connection with external data input or external messages is: 0: 4. 1: 16. [Length of a message] Up to 32 characters
Display format of external alarm messages	- [Alarm numbers that can be sent] 0 to 999 [How to distinguish these numbers from general alarm numbers] Add 1000 to the number sent	- Depends on bit 0 (EXA) of parameter No. 6301. **Bit 0 (EXA) of parameter No. 6301** Select the external alarm message specification. 0: The alarm numbers that can be sent range from 0 to 999. The CNC displays an alarm number, with 1000 added to the number following the character string "EX". 1: The alarm numbers that can be sent range from 0 to 4095. The CNC displays an alarm number, with the character string "EX" added in front of it.
Number of external operator messages and message length	- Depends on bit 0 (OM4) of parameter No. 3207. **Bit 0 (OM4) of parameter No. 3207** The external operator message screen can display: 0: Up to 256 characters in up to 1 message. 1: Up to 64 characters in up to 4 messages.	- Bit 0 (OM4) of parameter No. 3207 is not available. [Number of messages that can be set at a time] Depends on bit 1 (M16) of parameter No. 11931. Select either up to 4 or 16 messages. [Length of a message] 256 characters or less

Function	Series 0*i*-C	Series 0*i*-D
Display format of external operator messages	- [Message numbers that can be sent] 0 to 999 [How to distinguish these numbers from alarm and other numbers] **Messages from 0 to 99** The message is displayed on the screen along with the number. The CNC adds 2000 to this number for distinction. **Messages from 100 to 999** Only the message is displayed on the screen without the number.	- Depends on bit 1 (EXM) of parameter No. 6301. When 0 is set, the processing is the same as Series 0*i*-C. **Bit 1 (EXM) of parameter No. 6301** Select the external operator message specification. 0: The message numbers that can be sent range from 0 to 999. A message from 0 to 99 is displayed on the screen along with the number. The CNC adds 2000 to this number for distinction. As for the messages from 100 to 999, only the message is displayed on the screen without the number. 1: The message numbers that can be sent range from 0 to 4095. A message from 0 to 99 is displayed on the screen along with the number. The CNC adds the character string "EX" in front of the number. As for the messages from 100 to 4095, only the message is displayed on the screen without the number.
Data range of external operator message numbers	**Parameter No. 6310** The data range of external operator message numbers is as follows.	
	- 0 to 1000	- 0 to 4096
When an external program number search is done with 0 set as the program number	- An alarm is not issued; the search is not done, either.	- Alarm DS0059 is issued.
Input of an external tool offset for an invalid function compensation value	- The input is ignored without issuing an alarm.	- Alarm DS1121 is issued.

B.31.2 Differences in Diagnosis Display

None.

B.32 DATA SERVER FUNCTION

B.32.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Memory operation mode	- The memory operation mode is not supported.	- In the memory operation mode, the following operations can be performed for a program registered with the data server: 1. Select the program on the data server as the main program and run it in the memory mode. 2. Call a subprogram or custom macro in the same directory as the main program on the data server. 3. Edit the program, including inserting, deleting, and replacing words.

B.32.2 Differences in Diagnosis Display

None.

B.33 POWER MATE CNC MANAGER

B.33.1 Differences in Specifications

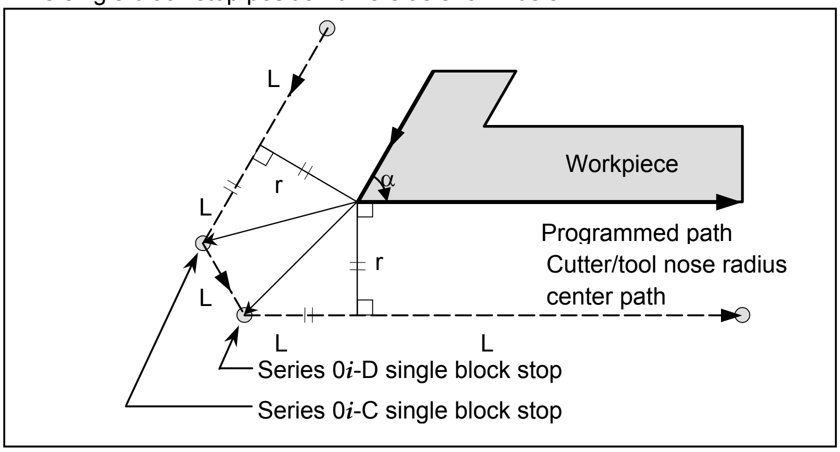
Function	Series 0*i*-C	Series 0*i*-D
4-slave display function	- By setting 1 in bit 0 (SLV) of parameter No. 0960, it is possible to split the screen into four windows, enabling up to four slaves to be displayed. **Bit 0 (SLV) of parameter No. 0960** When Power Mate CNC Manager is selected, the screen: 0: Displays one slave. 1: Is split into four windows, enabling up to four slaves to be displayed.	- Bit 0 (SLV) of parameter No. 0960 is not available. One slave is always displayed. When there is more than one slave, you switch the active slave by using the relevant soft key.

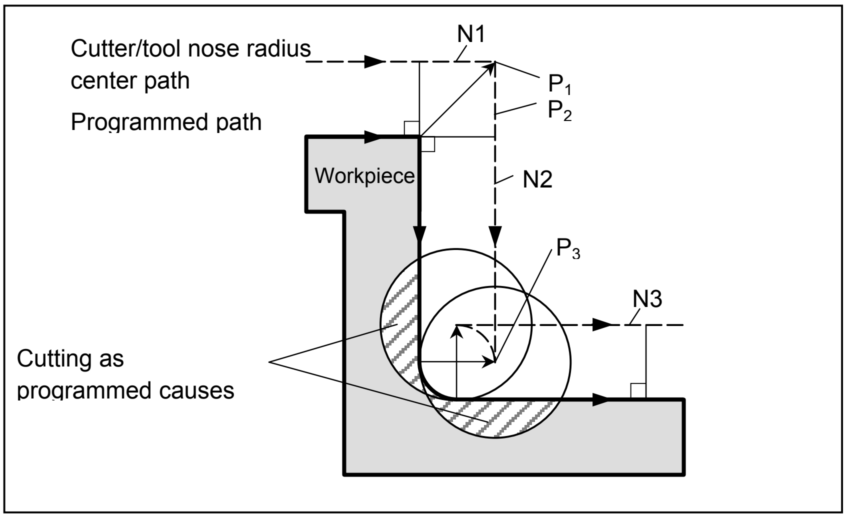
B.33.2 Differences in Diagnosis Display

None.

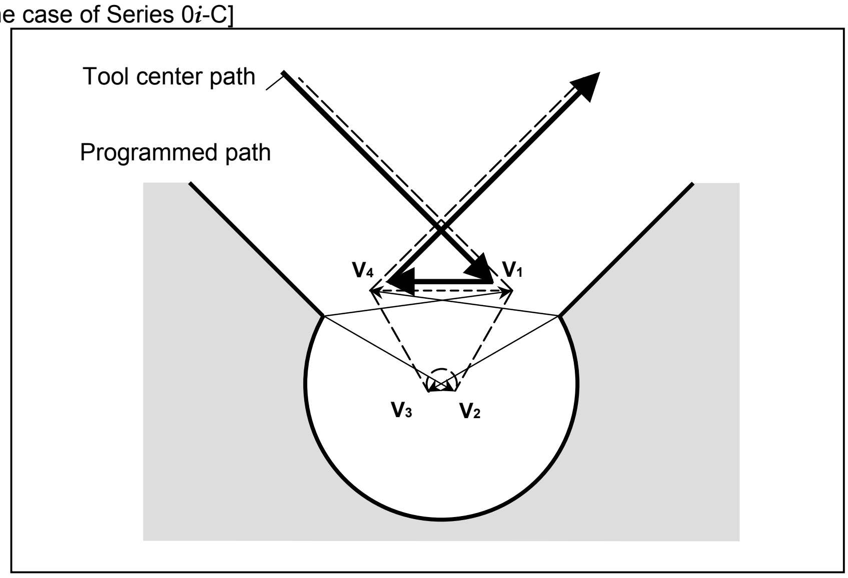
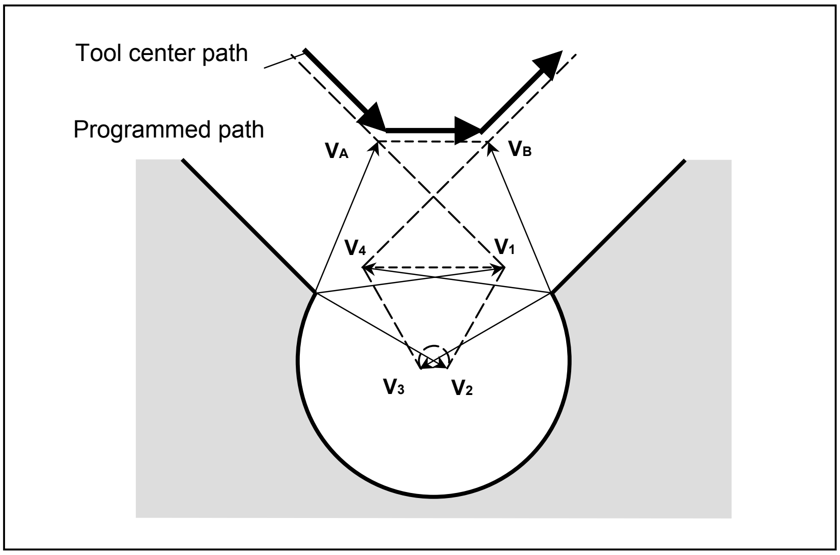
B.34 CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION

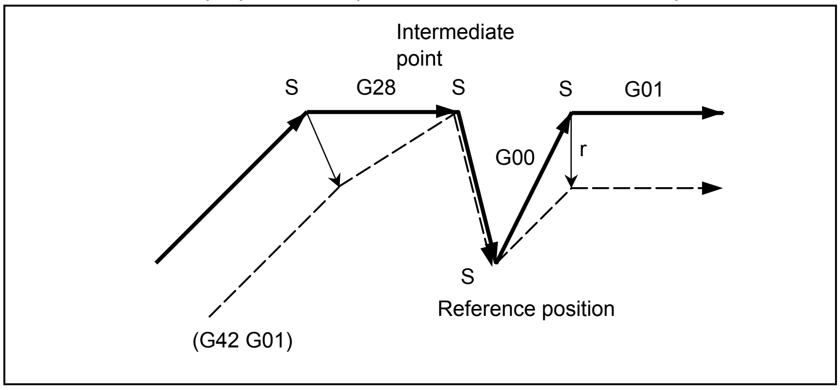
B.34.1 Differences in Specifications

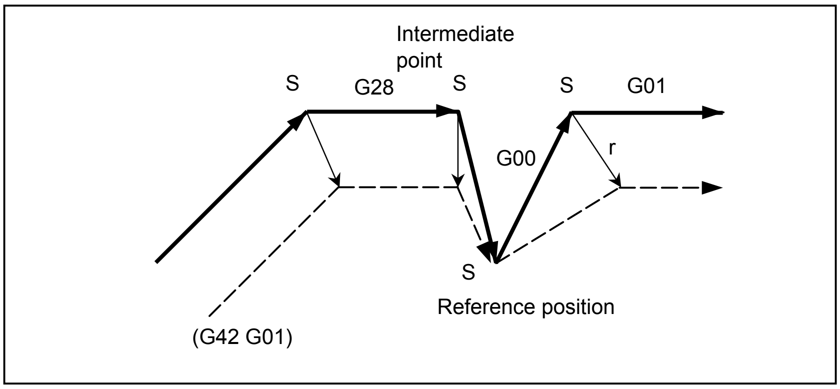
<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>Cutter compensation/tool nose radius compensation</td><td colspan="2">- In Series 0i-D, the cutter compensation C (M series) and tool-nose radius compensation (T series) functions of Series 0i-C are collectively referred to as cutter compensation/tool nose radius compensation.</td></tr>
<tr><td>Corner circular interpolation (G39)</td><td>- Enabled by setting 1 in bit 2 (G39) of parameter No. 5008.</td><td>- Available.
It is included in cutter compensation/tool nose radius compensation.
Since corner circular interpolation (G39) is always enabled, bit 2 (G39) of parameter No. 5008 is not available.</td></tr>
<tr><td>Cutter compensation/tool nose radius compensation in MDI operation</td><td>- Neither cutter compensation C nor tool nose radius compensation is available in MDI operation.</td><td>- Cutter compensation/tool nose radius compensation is also available in MDI operation.</td></tr>
<tr><td>Single block stop position during the cutter compensation/tool nose radius compensation mode</td><td colspan="2">- The single block stop position differs as shown below.

</td></tr>
<tr><td>Function to change the compensation direction intentionally (IJ type vector, KI type vector, and JK type vector)</td><td>- Not available.</td><td>- At the start of or during the cutter compensation/tool nose radius compensation mode, specify I, J, or K in a G00 or G01 block. This makes the compensation vector at the end point of the block perpendicular to the direction specified by I, J, or K. This way, you can change the compensation direction intentionally.</td></tr>
</table>

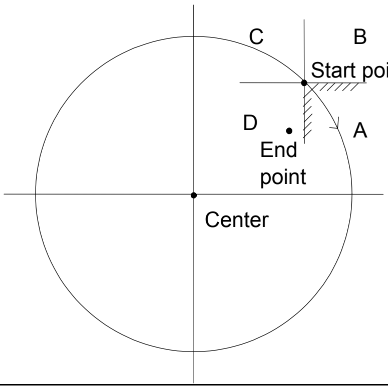
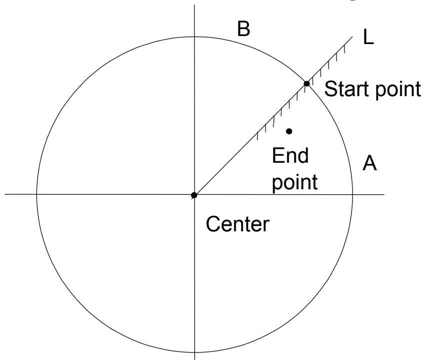
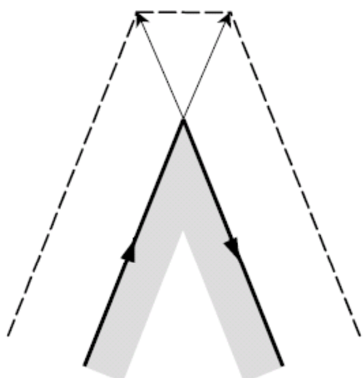
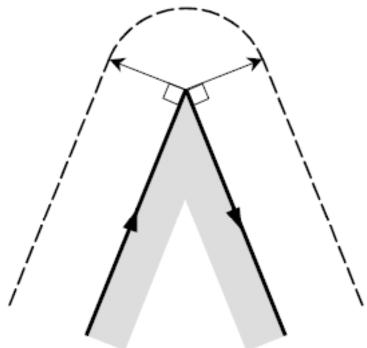
<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Stop position upon an overcutting alarm</td><td colspan="2">- If the specified radius value for circular interpolation is smaller than that for cutter compensation/tool nose radius compensation, as in the example below, performing compensation inwardly through cutter compensation/tool nose radius compensation causes overcutting, generating an alarm and stopping the tool. The stop position differs.

</td></tr>
<tr><td colspan="2">[When single block stop occurs in the preceding block in Series 0i-C]
Since the tool moves until it reaches the end point of the block (P_3 in the figure), overcutting may result.
[When single block stop does not occur in the preceding block in Series 0i-C]
The tool stops immediately after executing the block (P_2 in the figure).
[In the case of Series 0i-D]
Since the tool stops at the start point of the block (P_1 in the figure), regardless of the single block state, overcutting can be prevented.</td></tr>
<tr><td>Single block stop in a block created internally for cutter compensation/tool nose radius compensation</td><td>- Not available.</td><td>- Depends on bit 0 (SBK) of parameter No. 5000.

<u>Bit 0 (SBK) of parameter No. 5000</u>
In a block created internally for cutter compensation/tool nose radius compensation, single block stop is:
0: Not performed.
1: Performed.
This parameter is used to check a program including cutter compensation/tool nose radius compensation.</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Setting to disable interference checking and to delete interfering vectors	- Set 1 in bit 0 (CNI) of parameter No. 5008. In the example below, an interference check is made on the vectors inside V_1 and V_4, and the interfering vectors are deleted. As a result, the tool center path is from V_1 to V_4.	- Not available. (Bit 0 (CNI) of parameter No. 5008 is not available.) To prevent overcutting, the interference check avoidance function (bit 5 (CAV) of parameter No. 19607) is used. In the example below, interference occurs between V_1 and V_4 and between V_2 and V_3. Therefore, vectors V_A and V_B are created. The tool center path is from V_A to V_B.
	[In the case of Series 0*i*-C]  [In the case of Series 0*i*-D] 	
Number of blocks to be read in the cutter compensation/tool nose radius compensation mode	- Always 3 blocks	- The number can be set in parameter No. 19625. The specifiable range is 3 to 8 blocks. If the parameter is not set (0 is set), the same number as Series 0*i*-C (3 blocks) is assumed.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td>When circular interpolation is specified that causes the center to coincide with the start or end point during the cutter compensation/tool nose radius compensation mode</td><td>- Alarm PS0038 is issued, and the tool stops at the end point of the block preceding the circular interpolation block.</td><td>- Alarm PS0041 is issued, and the tool stops at the start point of the block preceding the circular interpolation block.</td></tr>
<tr><td rowspan="2">Behavior when automatic reference position return is specified during the cutter compensation/tool nose radius compensation mode</td><td>- Depends on bit 2 (CCN) of parameter No. 5003.</td><td>- Bit 2 (CCN) of parameter No. 5003 is not available. The tool always behaves as when CCN is set to 1.</td></tr>
<tr><td colspan="2">[When CCN = 0]
The offset vector is canceled when the tool moves to the middle point.
Also, the start-up operation is performed from the reference position.


[When CCN = 1 or for Series 0i-D]
The offset vector is not canceled when the tool moves to the middle point; it is canceled when the tool moves to the reference position.
Also, the tool moves from the reference position to the next intersection point.

</td></tr>
</table>

Function	Series 0*i*-C	Series 0*i*-D
Travel distance judgment method for circular interpolation in cutter compensation/tool nose radius compensation	- Depends on bit 5 (QCR) of parameter No. 5008.	- Bit 5 (QCR) of parameter No. 5008 is not available. The tool always behaves as when QCR is set to 1.
	[When QCR = 0]  	[When QCR = 1 or for Series 0*i*-D] 
	If the end point is on side A when viewed from the start point, the travel distance is small. If it is on side B, C, or D, the tool has traveled almost one round.	If the end point is on side A of line L connecting the start point and center, the travel distance is small. If it is on side B, the tool has traveled almost one round.
Compensation vector connection method when the tool travels around an external corner during the cutter compensation/tool nose radius compensation mode	- Connected by linear interpolation.	- Depends on bit 2 (CCC) of parameter No. 19607.
	[When CCC = 0 or for Series 0*i*-C] Connect vectors by linear interpolation 	[When CCC = 1] Connect vectors by circular interpolation 

B.34.2 Differences in Diagnosis Display

None.

B.35 CANNED CYCLE FOR DRILLING

B.35.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
M05 output in a tapping cycle	- Make a selection using bit 6 (M5T) of parameter No. 5101. **Bit 6 (M5T) of parameter No. 5101** When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G74 with the M series, or G84/G88 with the T series): 0: M05 is output before output of M04 or M03. 1: M05 is not output before output of M04 or M03.	- Make a selection using bit 3 (M5T) of parameter No. 5105. **Bit 3 (M5T) of parameter No. 5105** When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G74 with the M series, or G84/G88 with the T series): 0: M05 is output before output of M04 or M03. 1: M05 is not output before output of M04 or M03.
Behavior when K0 is specified for the number of repetitions K	- Drilling operation is not performed, and only drilling data is stored.	- Make a selection using bit 4 (K0D) of parameter No. 5105. **Bit 4 (K0D) of parameter No. 5105** When K0 is specified in a drilling canned cycle (G80 to G89): 0: Drilling operation is not performed, and only drilling data is stored. 1: One drilling operation is performed.
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **Bit 1 (NRF) of parameter No. 3700** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.
Forward/retraction feedrate for the small-hole peck drilling cycle (G83)	- When the I command (forward/retraction feedrate) is omitted and 0 is set in parameter Nos. 5172 and 5173, the forward/retraction feedrate is as follows.	
	0	Same feedrate as that specified by the F command
Tool retraction direction in a fine boring cycle (G76) or back boring cycle (G87)	- Set the direction using bit 5 (RD2) and bit 4 (RD1) of parameter No. 5101 in combination.	- Bit 5 (RD2) and bit 4 (RD1) of parameter No. 5101 is not available. Set the direction in axis-type parameter No. 5148.

<table>
<tr><th>Function</th><th>Series 0i-C</th><th>Series 0i-D</th></tr>
<tr><td rowspan="2">Address Q command in a high-speed peck drilling cycle (G73), peck drilling cycle (G83), or small-hole peck drilling cycle (G83)</td><td colspan="2">- In a high-speed peck drilling cycle (G73), peck drilling cycle (G83), and small-hole peck drilling cycle (G83), when the address Q (amount of each-time cutting) command is not specified or Q0 is specified:</td></tr>
<tr><td>Select the operation using bit 1 (QZA) of parameter No. 5103.

<u>Bit 1 (QZA) of parameter No. 5103</u>
0: The tool repeats the upward and downward movement at the same position without cutting.
1: P/S alarm No. 045 is issued.</td><td>Bit 1 (QZA) of parameter No. 5103 is not available.
The tool always behaves as when 1 is set in bit 1 (QZA) of parameter No. 5103. (Alarm PS0045 is issued.)</td></tr>
<tr><td>Tool length compensation (G43 or G44) in a canned cycle when tool length compensation type C is selected (1 is set in bit 0 (TLC) of parameter No. 5001)</td><td>- Select the axis for which to enable tool length compensation, by using bit 4 (TCE) of parameter No. 5006.

<u>Bit 4 (TCE) of parameter No. 5006</u>
When tool length compensation (G43 or G44) is specified in a canned cycle, tool length compensation is enabled for:
0: Axis selected according to tool length compensation type C.
1: Drilling axis.</td><td>- Bit 4 (TCE) of parameter No. 5006 is not available.
The tool always behaves as when 1 is set in bit 4 (TCE) of parameter No. 5006.</td></tr>
</table>

B.35.2 Differences in Diagnosis Display

None.

B.36 CANNED GRINDING CYCLE

B.36.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Grinding axis specification	- The grinding axis is the X or Z axis.	- Set the grinding axes for the individual canned grinding cycles in parameter Nos. 5176 to 5179. If the same axis number as the cutting axis is specified in any of these parameters, or if a canned grinding cycle is executed when 0 is set, alarm PS0456 is issued.
Behavior of the first positioning command (G00) for a Cs contour control axis in a canned cycle	- The behavior can be selected using bit 1 (NRF) of parameter No. 3700. **<u>Bit 1 (NRF) of parameter No. 3700</u>** After a serial spindle is changed to a Cs contour control axis, the first move command: 0: Performs the normal positioning operation after executing the reference position return operation. 1: Performs the normal positioning operation.	- While bit 1 (NRF) of parameter No. 3700 exists, the normal positioning operation is performed in a canned cycle, regardless of the setting of this parameter bit.

Function	Series 0*i*-C	Series 0*i*-D
Dressing axis specification	- The dressing axis is always the fourth axis.	- Set the dressing axes for the individual canned grinding cycles in parameter Nos. 5180 to 5183. If the same axis number as the cutting axis or grinding axis is specified in any of these parameters, or if a canned grinding cycle is executed when 0 is set, alarm PS0456 is issued.

B.36.2 Differences in Diagnosis Display

None.

B.37 SINGLE DIRECTION POSITIONING

B.37.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Behavior when linear interpolation type positioning is used with mirror image	If positioning of linear interpolation type is used (1 is set in bit 1 (LRP) of parameter No. 1401), and if the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, the following alarms are issued, respectively.	
	- Alarm PS5254	- Alarm DS0025

B.37.2 Differences in Diagnosis Display

None.

B.38 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING

B.38.1 Differences in Specifications

Function	Series 0*i*-C	Series 0*i*-D
Optional angle chamfering and corner rounding commands for a plane including a parallel axis	- Not available. Alarm PS0212 is issued.	- Available.
Single block operation	- Single block stop is not performed at the start point of an inserted optional angle chamfering or corner rounding block.	- Whether to perform single block stop at the start point of an inserted block depends on bit 0 (SBC) of parameter No. 5105. **Bit 0 (SBC) of parameter No. 5105** In a drilling canned cycle, chamfer cycle/corner rounding (T series) or optional angle chamfering/corner rounding cycle (M series): 0: Single block stop is not performed. 1: Single block stop is performed.

Function	Series 0*i*-C	Series 0*i*-D
Negative value specified in a ,C_ or ,R_ command	- The value is regarded as positive.	- Alarm PS0006 is issued.
Number of dwells to be inserted between two blocks for which to perform optional angle chamfering or corner rounding	- Not limited.	- Only one block can be inserted. Inserting more than one block causes alarm PS0051.
DNC operation	- Optional angle chamfering and corner rounding are not available in DNC operation.	- Optional angle chamfering and corner rounding are also available in DNC operation.

B.38.2 Differences in Diagnosis Display

None.

INDEX

<O>

<P>

<R>

<S>

<T>

<W>

Revision Record

FANUC Series 0*i*-MODEL D/Series 0*i* Mate-MODEL D OPERATOR'S MANUAL (For Machining Center System) (B-64304EN-2)

Edition	Date	Contents	Edition	Date	Contents
02	Aug., 2010	Total revision			
01	Jun., 2008	————			