FANUC Series 0*i*-MODEL F

For Machining Center System

OPERATOR'S MANUAL

B-64604EN-2/01

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

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

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

SAFETY PRECAUTIONS

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

CONTENTS

DEFINITION OF WARNING, CAUTION, AND NOTE

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.

⚠ **WARNING**

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

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

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

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

⚠ **CAUTION**

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

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

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

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

NOTE

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

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

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

⚠ **WARNING**

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

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

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

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

⚠ **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.

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

⚠ **CAUTION**

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

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

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

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

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

WARNINGS AND CAUTIONS RELATED TO HANDLING

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

⚠ **WARNING**

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

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

3 **Manual numeric command**
When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid.
Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

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

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

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

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

⚠ **WARNING**

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

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

⚠ **CAUTION**

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

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

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

4 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.

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

WARNINGS RELATED TO DAILY MAINTENANCE

⚠ **WARNING**

1 **Memory backup battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.
Refer to the Section “Method of replacing battery” in the OPERATOR’S MANUAL (Common to Lathe System/Machining Center System) for details of the battery replacement procedure.

⚠ **WARNING**

2 **Absolute pulse coder battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.
Refer to the FANUC SERVO MOTOR αi series Maintenance Manual for details of the battery replacement procedure.

⚠ **WARNING**

3 Fuse replacement

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

TABLE OF CONTENTS

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.

NOTE
1 This manual describes the functions that can operate in the CNC model for machining center system (path control type). For other functions not specific to the lathe system, refer to the Operator's Manual (Common to Lathe System/Machining Center System) (B-64604EN).
2 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-64610EN).
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.
3 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

This manual describes the models indicated in the table below.
In the text, the abbreviations indicated below may be used.

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

NOTE
1 For explanatory purposes, the following descriptions may be used according to the CNC model :
- 0i-MF : Machining center system (M series)
2 For the FANUC Series 0i-MODEL F, parameters need to be set to enable or disable some basic functions. For these parameters, refer to "PARAMETERS OF 0i-F BASIC FUNCTIONS" in the PARAMETER MANUAL (B-64610EN).

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*- MODEL F

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

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64602EN	
CONNECTION MANUAL (HARDWARE)	B-64603EN	
CONNECTION MANUAL (FUNCTION)	B-64603EN-1	
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64604EN	
OPERATOR'S MANUAL (For Lathe System)	B-64604EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64604EN-2	*
MAINTENANCE MANUAL	B-64605EN	
PARAMETER MANUAL	B-64610EN	
Programming		
Macro Executor PROGRAMMING MANUAL	B-63943EN-2	
Macro Compiler PROGRAMMING MANUAL	B-66263EN	
C Language Executor PROGRAMMING MANUAL	B-63943EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64513EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-63993EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN	
DeviceNet Board CONNECTION MANUAL	B-64043EN	
CC-Link Board CONNECTION MANUAL	B-64463EN	
Operation guidance function		
MANUAL GUIDE *i* (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE *i* (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN-1	
MANUAL GUIDE 0*i* OPERATOR'S MANUAL	B-64434EN	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64483EN-2	

Related manuals of SERVO MOTOR 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

Manual name	Specification number
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* series FANUC AC SERVO MOTOR 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

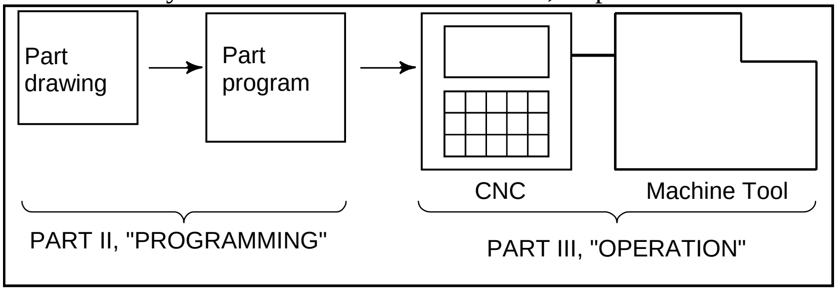
The above servo motors and the corresponding spindles can be connected to the CNC covered in this manual. In the α*i* SV, α*i* SP, α*i* PS, and β*i* SV series, however, they can be connected only to 30 *i*-B-compatible versions. In the β*i* SVSP series, they cannot be connected.
This manual mainly assumes 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 process / Cutting procedure	1 End face cutting	2 Outer diameter cutting	3 Grooving
1. Cutting method : Rough Semi Finish			
2. Cutting tools			
3. Cutting conditions : Feedrate Cutting depth			
4. Tool path			

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.

⚠ **CAUTION**

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**

1 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.

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

II. PROGRAMMING

1 GENERAL

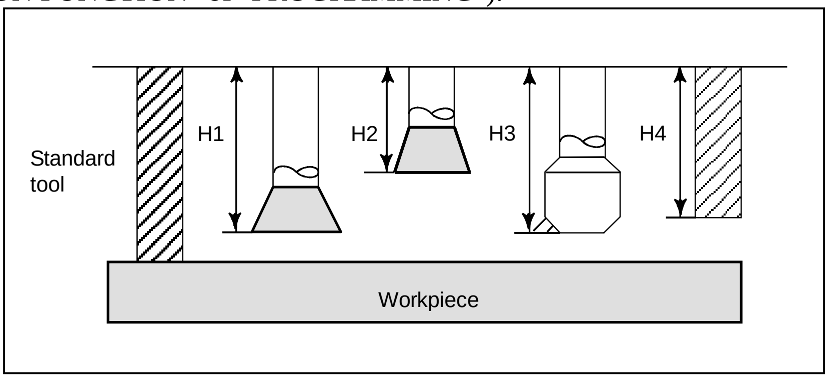
Chapter 1, "GENERAL", consists of the following sections:

1.1 TOOL FIGURE AND TOOL MOTION BY PROGRAM

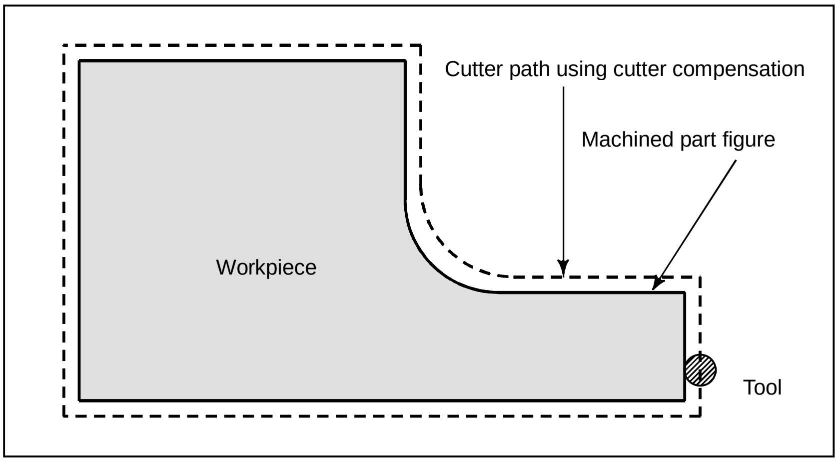
Explanation

- 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 (Refer to Chapter, "SETTING AND DISPLAYING DATA" in "OPERATION"), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation (Refer to Chapter, "COMPENSATION FUNCTION" of "PROGRAMMING").



- 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 (Refer to Chapter, "SETTING AND DISPLAYING DATA" in "OPERATION"), the tool can be moved by cutter radius apart from the machining part figure. This function is called cutter compensation (Refer to Chapter, “COMPENSATION FUNCTION” of "PROGRAMMING").

2 PREPARATORY FUNCTION (G FUNCTION)

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

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

(Example)
G01 and G00 are modal G codes in group 01.

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

Explanation

1. When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with as indicated in Table 2 (a).
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by bit 7 (G23) of parameter No. 3402. However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting bit 0 (G01) of parameter No. 3402.
   (5) The user can select G90 or G91 by setting bit 3 (G91) of parameter No. 3402.
   When G code system B or C is used in the lathe system, setting bit 3 (G91) of parameter No. 3402 determines which code, either G90 or G91, is effective.
   (6) In the machining center system, the user can select G17, G18, or G19 by setting bits 1 (G18) and 2 (G19) of parameter No. 3402.
2. G codes other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010, “IMPROPER G-CODE” occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle for drilling.
6. G codes are indicated by group.
7. The group of G60 is switched according to the setting of the bit 0 (MDL) of parameter No. 5431. (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

Table 2 (a) G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G04	00	Dwell	
G04.1		G code preventing buffering	
G05.1		AI Advanced Preview Control / AI contour control / Nano smoothing	
G05.4		HRV3 on/off	
G07.1 (G107)		Cylindrical interpolation	
G08		AI Advanced Preview Control / AI contour control (advanced preview control compatible command)	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G11		Programmable data input mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20 (G70)	06	Input in inch	
G21 (G71)		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G25	19	Spindle speed fluctuation detection off	
G26		Spindle speed fluctuation detection on	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G28.2		In-position check disable reference position return	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.2		In-position check disable 2nd, 3rd, or 4th reference position return	
G31		Skip function	
G31.8		EGB-axis skip	
G33	01	Threading	
G37	00	Automatic tool length measurement	
G38		Tool radius/tool nose radius compensation : preserve vector	
G39		Tool radius/tool nose radius compensation : corner circular interpolation	
G40	07	Tool radius/tool nose radius compensation : cancel	
G41		Tool radius/tool nose radius compensation : left	
G42		Tool radius/tool nose radius compensation : right	
G40.1	18	Normal direction control cancel mode	
G41.1		Normal direction control on : left	
G42.1		Normal direction control on : right	
G43	08	Tool length compensation +	
G44		Tool length compensation -	
G43.7		Tool offset	
G45	00	Tool offset : increase	
G46		Tool offset : decrease	
G47		Tool offset : double increase	
G48		Tool offset : double decrease	
G49 (G49.1)	08	Tool length compensation cancel	

Table 2 (a) G code list

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

Table 2 (a) G code list

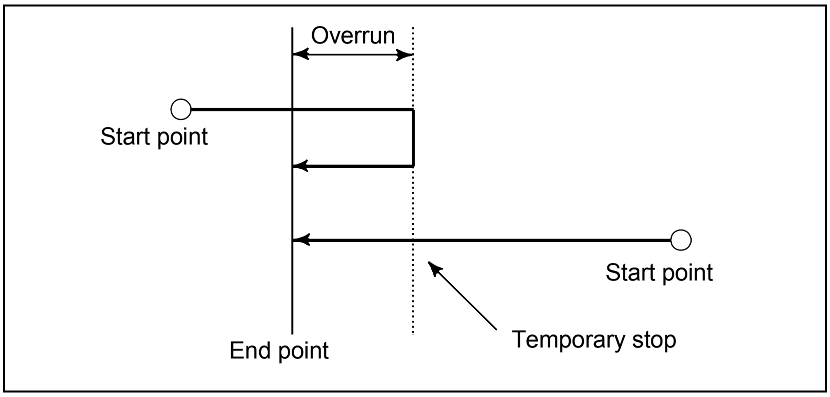
G code	Group	Function
G82	09	Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS10/11 format)
G84.3		Left-handed rigid tapping cycle (FS10/11 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring 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
G96.1	00	Spindle indexing execution (waiting for completion)
G96.2		Spindle indexing execution (not waiting for completion)
G96.3		Spindle indexing completion check
G96.4		SV speed control mode ON
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G107	00	Cylindrical interpolation
G160	20	In-feed control cancel
G161		In-feed control

3 INTERPOLATION FUNCTION

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;   }  Single direction positioning
G60 Y100;   }
G04 X10;
G00 X0Y0;
```

When modal G60 command is used.

```
G90G60;        Single direction positioning mode start
X0Y0;   }
X100;   }      Single direction positioning
Y100;   }
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 (Fig. 3.1 (a)), single direction positioning is performed independently along each axis.

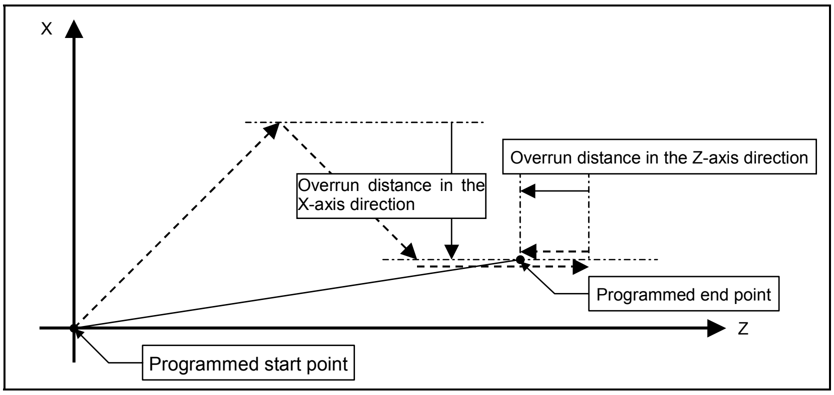


Fig. 3.1 (a)

- **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.

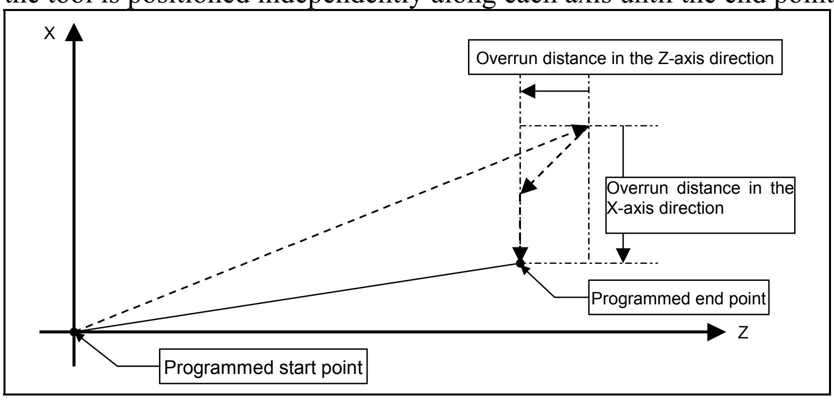


Fig. 3.1 (b)

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 (DS0025)"G60 CANNOT BE EXECUTED" 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.

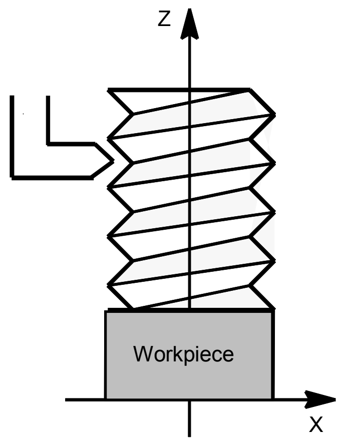
3.2 THREADING (G33)

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)

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. If the spacing between adjacent inflection points or programmed points is not constant, this function can generate a smoother curve.
The interpolation of the curve reduces the segment approximation error, and the nano-interpolation makes the cutting surface smoother.

> **NOTE**
> This function is an optional function.
> To use this function, the both options for "AI contour control II" and this function are required.

Format

G5.1 Q3 Xp0 Yp0 Zp0 [α0] [β0] ; : 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
α, β : Rotary 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
                       - Machining condition selecting function
G5.1 Q3 X0 Y0 Z0;
…
G5.1 Q0; ----------    Nano smoothing mode off
                       AI contour control mode off
…
M30;
```

Explanation

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

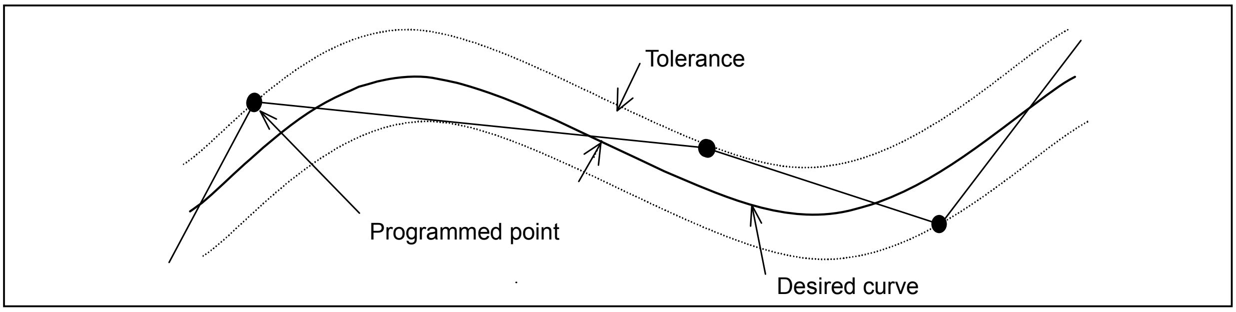


Fig. 3.3 (a)

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.

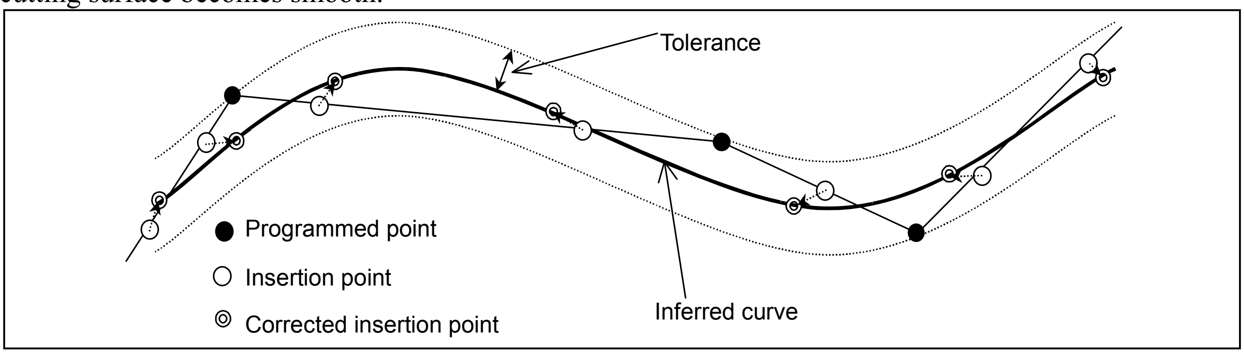


Fig. 3.3 (b)

- 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.

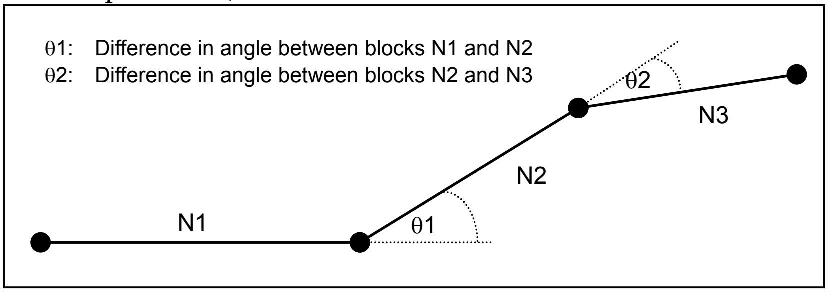


Fig. 3.3 (c)

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.

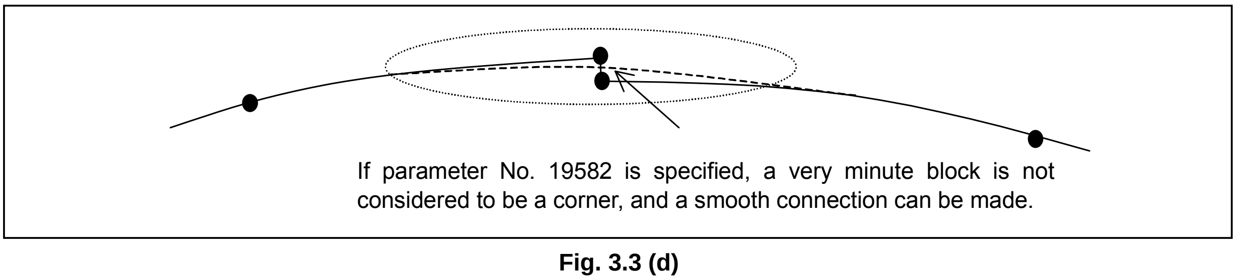


Fig. 3.3 (d)

- Automatically turning on and off AI contour control with nano smoothing

Specifying G5.1 Q3 also enables nano smoothing and AI contour control to be turned on at the same time. The automatic velocity control by AI contour control reduces impacts on the mechanical system. Specifying G5.1 Q0 cancels the nano smoothing and the AI contour control 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 or inverse time feed
  - Tool radius compensation cancel
  - Canned cycle cancel
  - Scaling cancel
  - Macro modal call cancel
  - Constant surface speed control cancel
  - Cutting mode
  - Coordinate system rotation/3-dimensional coordinate system conversion 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

Diagnosis 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(G49.1),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,G93 : Feed per minute or inverse time feed
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, "ILLEGAL COMMAND IN NANO SMOOTHING", 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, "CUSTOMMACRO INTERRUPT ENABLE IN NANO SMOOTHING", 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, "ILLEGAL RESTART(NANO SMOOTHING)", 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, "TOO MANY COMMAND BLOCK (NANO SMOOTHING)", 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, "CANNOT CONTINUE NANOSMOOTHING", 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) Retracting and returning the tool
The tool cannot be retracted or returned in the nano smoothing mode.

(3) Retracing
Retracing cannot be performed in the nano smoothing mode.

(4) Manual handle retrace
In nano smoothing mode, manual handle retrace cannot be performed.

(5) Active block cancel
The active block cancel function is temporarily disabled in the nano smoothing mode.

- Dynamic graphic display

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

3.4 SMART TOLERANCE CONTROL

Two functions that generate smooth machining path within specified tolerance, and realize high-speed and high-precision machining are introduced as follows.

> **NOTE**
> "Smart tolerance control" is an optional function.
> To use this function, the both options for "AI contour control II" and this function are required.
> To use the "Smoothing small line segments" for this function, in addition to these options, the option for "Nano smoothing" are required.

Making corner path into curve

In the conventional AI contour control, direction and curvature of specified path are discontinuous at each joint of linear interpolation blocks and circular interpolation blocks. This function makes direction and curvature of corner paths continuous, and corner paths are made into curves so that the precision at each joint of linear interpolation blocks and circular interpolation blocks is within the tolerance specified parameter No. 19596 or G code.
Therefore, setting the machining precision gets easier and high-precision machining is possible regardless of feedrate.

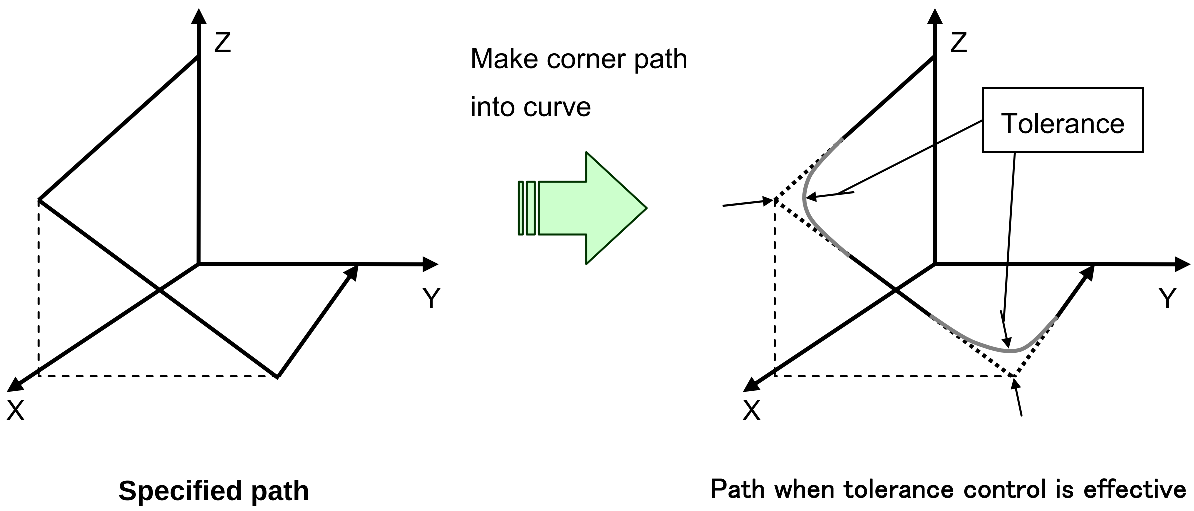


Fig.3.4 (a) Making corner path into curve

The following blocks are made into curves in “making corner path into curve”.

- Linear interpolation – Linear interpolation
- Linear interpolation – Circular interpolation
- Circular interpolation – Circular interpolation

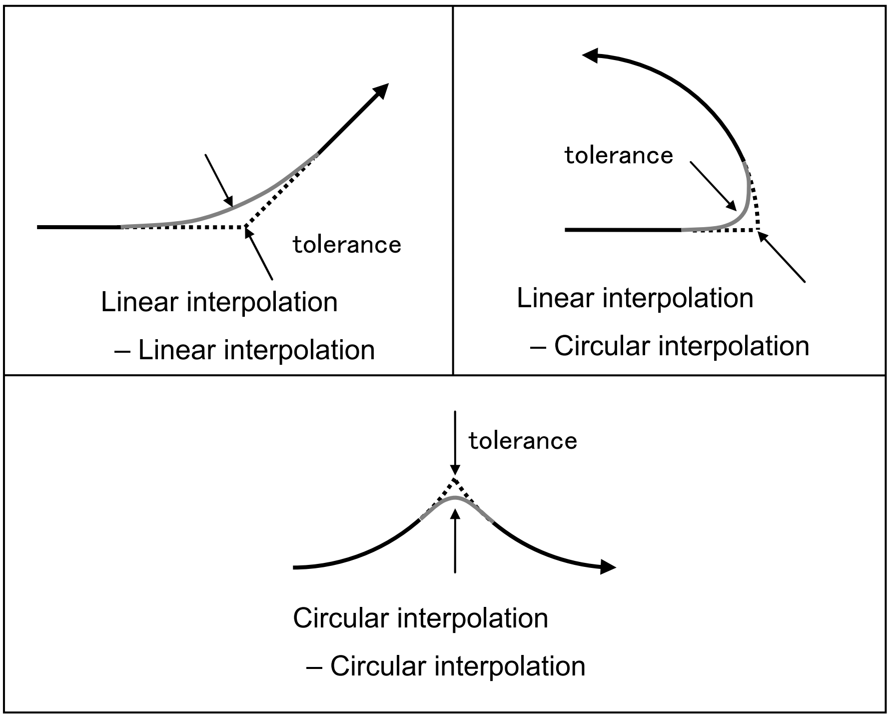


Fig.3.4 (b) Conversion to curves at each blocks

The options for smart tolerance control and AI contour control II are required.

Smoothing small line segments

This function makes a path defined by small line segments into a curvature, furthermore it makes smooth the connection of curvatures.

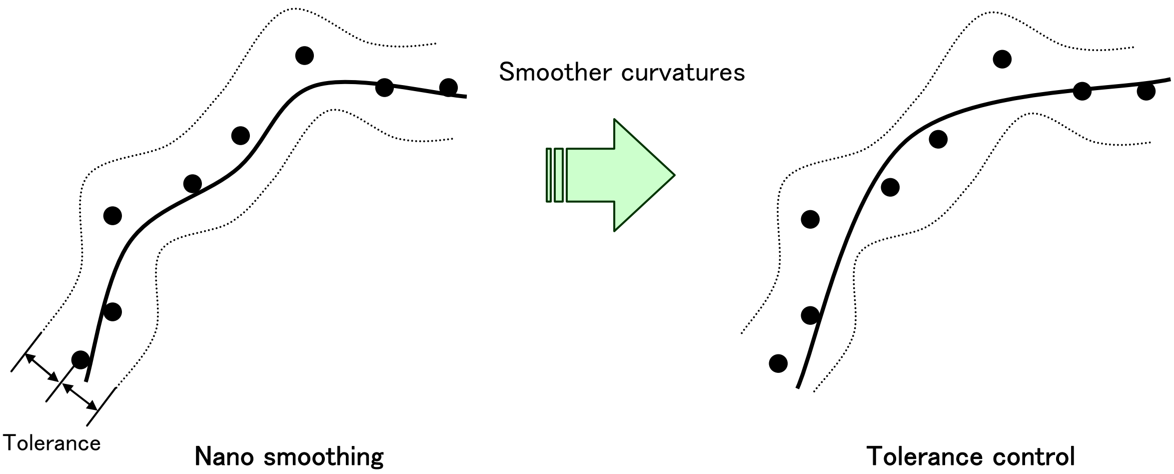


Fig.3.4 (c) Smoothing small line segments

In addition, conversion is effective at corners between the following blocks.

- Linear interpolation – Curve generated from small line segments
- Circular interpolation – Curve generated from small line segments

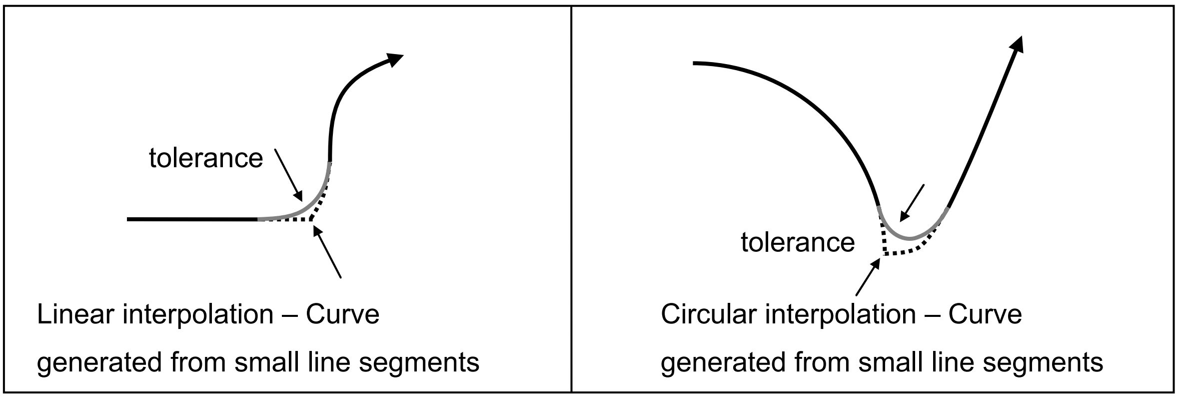


Fig.3.4 (d) Conversion to curves at corner of curves generated from small line segments

Nano smoothing options is required besides smart tolerance control and AI contour control II in order to smooth small line segments.

Format

G05.1 Q3 Xp0 Yp0 Zp0;	**Smart tolerance control mode on**
G10.8 L4 I_ Q_;	**Specify tolerance**
:	
G05.1 Q0;	**Smart tolerance control 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
I: Tolerance for linear axis at corners
Q: Tolerance for linear axis on curves

NOTE

1. Specify G05.1 alone in a block.
   (Avoid specifying any other G code in the same block)
2. Bit 0 (ATC) of parameter No. 19594 must be set to 1 when using this function.
3. Unit of "I" in G10.8 command depends on the increment system of the basic axis.
4. The value of parameter No. 19596 and No. 19597 are effective as tolerance between G05.1 Q3 command and G10.8 L4 command.
5. Smart tolerance control is disabled when the tolerance for linear axis is set to 0.
6. Smart tolerance control mode is activated at the start of automatic operation by setting bit 0 (CAT) of parameter No. 11785.

Option

The options for smart tolerance control and AI contour control II are required convert corner path into curve.
And the options for nano smoothing is also required besides smart tolerance control and AI contour control II in order to smooth small line segments.

Description of the Function

- Making corner path into curve

In normal AI contour control, smooth control at corners is possible by setting allowable feedrate difference (parameter No. 1783). On the other hand, in smart tolerance control mode, corner paths are converted into curve so that the path error at corners is within the specified tolerance. A direction and a curvature of corner path are seamlessly changing, and the appropriate feedrate is set at a corner so that the acceleration is less than parameter No. 19599. Therefore machining accuracy can be easily specified.

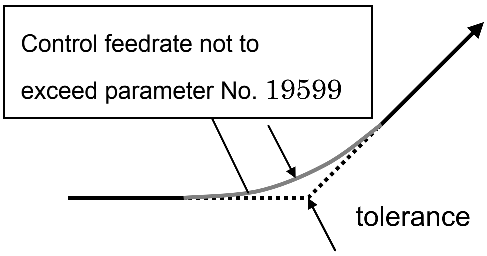


Fig.3.4 (a) Inner deviations at a corner

> **NOTE**
> The value of parameter No. 1737 is applied unless parameter No. 19599 is set.

- Smoothing in smart tolerance control mode

If length of linear blocks are less than the value set at parameter No. 19595, those blocks are treated as small line segment blocks, and smooth curve not depending on commanded points, which are within tolerance for linear axes and rotary axes from commanded points, are generated.
Machining gets smoother even if there are small discontinuous blocks in a program.

> **NOTE**
> 1 Inner deviation for neither acceleration / deceleration after interpolation nor delay of servo is included in accuracy of corners this function controls.
> 2 The number of look-ahead blocks may be decrease in this function because of generating corner paths.

- Automatically turning on AI contour control with smart tolerance control

Specifying G5.1 Q3 also enables smart tolerance control and AI contour control to be turned on at the same time.
The automatic velocity control by AI contour control reduces impacts on the mechanical system.

- Conditions for enabling smart tolerance control

Smart tolerance control is enabled when the following conditions are satisfied.
In a block that does not satisfy the conditions for enabling it, smart tolerance control is canceled, and it is judged
in the next block whether to perform smart tolerance control again.

- The mode is all of the following:
  - Cutting mode (G64)
  - Linear interpolation (G01), or circular interpolation (G02/G03)
  - Feed per minute (G94, however G98 in G code system A of T series)
  - Macro modal call cancel (G67)
  - Constant surface speed control cancel (G97)
  - Normal direction control cancel (G40.1)
  - Polar coordinate command cancel (G15)

- The block does not contain a one shot G code command.
- The block does not contain MST command.
- The block does not suppress look ahead (buffering).
- The block contains a move for at least a linear axis.
- The block contains a move command for only axes subject to smart tolerance control.
- The difference between a radius at the beginning point and a radius at the end point is less than 20μm.

- Conditions for enabling smoothing in smart tolerance control mode

Smoothing in smart tolerance control mode is effective on the following conditions besides conditions for enabling smart tolerance control. In the following description, "block length" apply to the basic three axes (or axes parallel to them) only, not rotation axes.

- Linear interpolation (G01)
- Block length is less than the value of parameter No. 19595.

- Conditions for canceling making corner path into curve

In smart tolerance control mode, making corner path into curve is canceled at the connection of blocks on the following conditions. In this case, movement of axes temporary stops at the connection of blocks.

- In case moving direction is inverted between linear interpolation blocks.
- In case moving direction is inverted between circular interpolations whose central coordinate values are exact the same.
- In case the setting tolerance is less than 10^{-5} (1 / 100000) of the length of the block just before the corner (arc length in case a circular interpolation block).

- Interlock

When an axis for smart tolerance control is interlocked, all axes are interlocked in the block which smart tolerance control is enabled, even if the interlocked axis is not commanded in the block.

- Axis moving signals

In the block which smart tolerance control is enabled, axis moving signals MV1 to MV8 <Fn102> for axes for smart tolerance control are set to one regardless of the movement.

- Use with other functions

In the case smart tolerance control is used with the following functions, it controls paths which each function is applied.

- Cutter compensation and tool nose radius compensation
- Tool length compensation
- Programmable mirror image
- Scaling
- Coordinate system rotation

Use with cutter compensation and tool nose radius compensation

In case using cutter compensation and tool nose radius compensation, smart tolerance control works for paths cutter compensation and tool nose radius compensation applied.

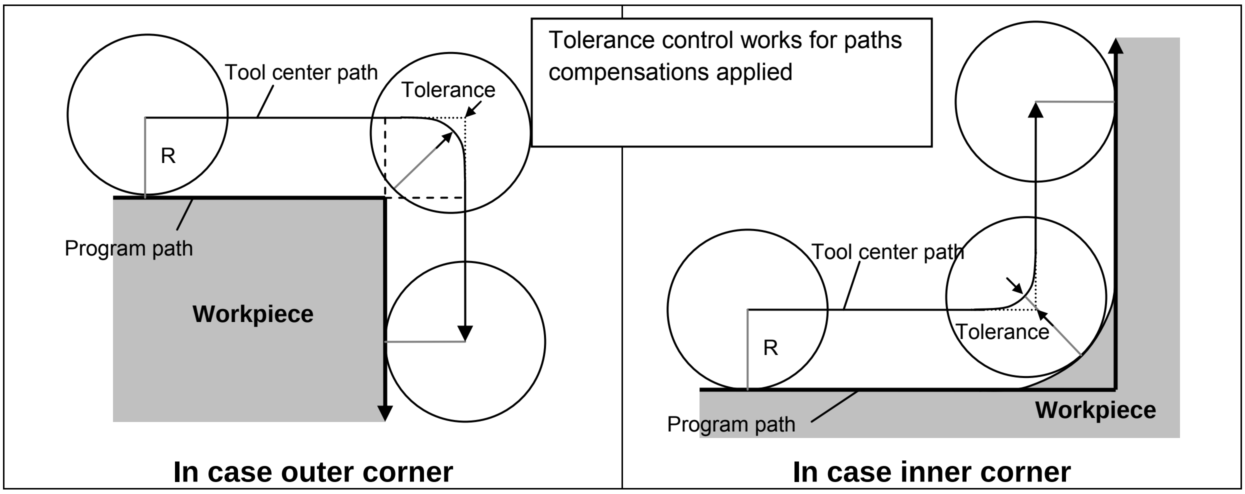
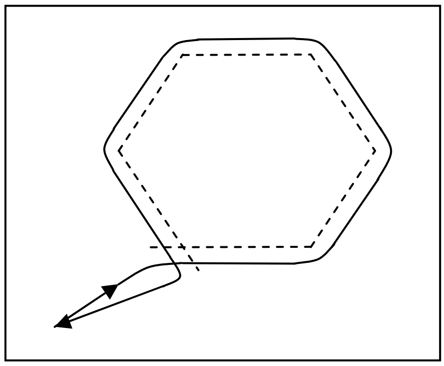


Fig.3.4 (b) Use with cutter compensation and tool nose radius compensation

Example

Here is an example for smart tolerance control.

```
O0010;
G28 G91 X0 Y0;
G05.1 Q3 X0 Y0 Z0;      (Smart tolerance control mode on)
G10.8 L4 I2.0;          (Specify tolerance)
G90 G01 G41 X12.0 Y11.340 D4 F1000;
X25.0 Y11.340;
X30.0 Y20.0;
X25.0 Y28.660;
X15.0 Y28.660;
X10.0 Y20.0;
X16.5 Y8.742;
G40 X0 Y0;
G05.1 Q0;               (Smart tolerance control mode off)
M30;
```



Note

- Functions Which Smart tolerance control Is Temporary Disabled

This function is temporary disabled when the following functions are used together.

Functions
Any kind of interpolations except liner interpolation and circular interpolation. (Include helical interpolation)
Feed per revolution
Inverse time feed
Constant surface speed control cancel
Normal direction control
Polar coordinate command
Tool length compensation command block
Canned cycle
In-position check disable reference position return

- Single block operation

When single block operation is carried out in the smart tolerance control mode, the operation stops at end of corners not at a programmed point.
Even in smart tolerance control mode, normal single-block operation is carried out for a block that does not satisfy the conditions of smart tolerance control mode.

- Background graphic display

The background graphic display function draws the path in smart tolerance control mode by linear interpolation.

- Functions Which Does Not Work with Smart tolerance control

This function doesn’t work with the following functions.

Functions	Option is disabled (1)	Alarm (2)
Nano smoothing	✓	
Macro modal call		PS2010
Interruption type custom macro		PS2012
Manual intervention with manual absolute mode on		PS2012
Program restart		PS2012
Quick program restart		PS2012

(1) Optional functions which are disabled when a bit 0 (ATC) of parameter No. 19594 is 1.
(2) Functions which issue alarms in case using with smart tolerance control.

3.4.1 Change Tolerance in Smart tolerance control Mode

The tolerance in smart tolerance control mode can be changed any time by specifying G10.8 L4. The appropriate tolerance can be used which depends on situations in a program.

Format

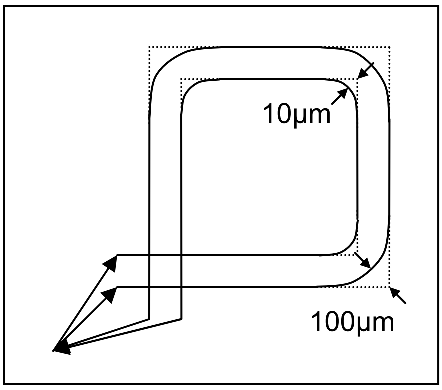
Change Tolerance in Smart tolerance control Mode

G10.8 L4 I_ Q_; Directly specify tolerance
G10.8 L4; Use paramter (No. 19596, No.11786) as default tolerance
I: Tolerance for linear axis at corners
Q: Tolerance for linear axis on curves

NOTE
1 Specify G10.8 alone in a block.
(Avoid specifying any other G code in the same block)
2 G10.8 L4 is a one-shot G code.
3 Unit of "I" and "Q" in G10.8 command depends on the increment system of the basic axis.
4 Smart tolerance control is ineffective when both I are set to 0.
5 Specifying a negative value to "I", or "Q" causes alarm PS2010, "ILL. COMMAND IN TOLERANCE CON.".
6 When G10.8 L4 specified not in smart tolerance control mode (G05.1 Q3), alarm PS0412, "ILLEGAL G CODE" is issued.
7 The value of parameter No. 19596, No. 19597, No. 11786 and No. 11787 do not change by specifying G10.8 L4.

Example

Here is an example for changing tolerance in smart tolerance control mode.




```
O0011;
G28 G91 X0 Y0;
G05.1 Q3 X0 Y0 Z0;              (Smart tolerance control mode on)
G10.8 L4 I0.1;                  (Tolerance is set to 100μm)
G90 G01 G41 X20.0 Y28.0 D6 F1000;
X47.0;
Y47.0;
X28.0;
Y20.0;
X0 Y0;
G40 X0 Y0;
G10.8 L4 I0.01;                 (Tolerance is set to 10μm)
G90 G01 G41 X20.0 Y30.0 D6 F1000;
X45.0;
Y45.0;
X30.0;
Y20.0;
G40 X0 Y0;
G05.1 Q0;                       (Smart tolerance control mode off)
M30;
```

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).

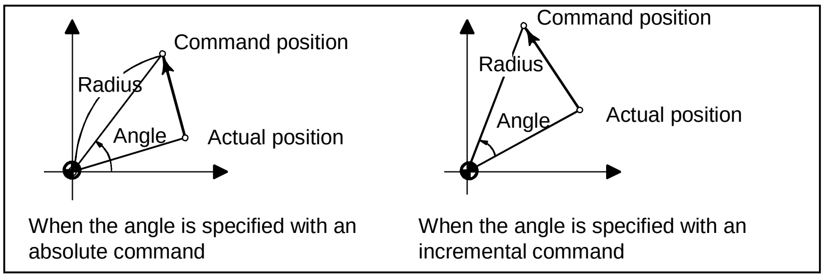
Format

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 program coordinate system as the origin of the polar coordinate system, from which a radius is measured.
G91 specifies the current position as the origin of the polar coordinate system, from which a radius is measured.
IP_ : Specifying the addresses of axes constituting the plane selected for the polar coordinate system, and their values
First axis : radius of polar coordinate
Second axis : angle of polar coordinate

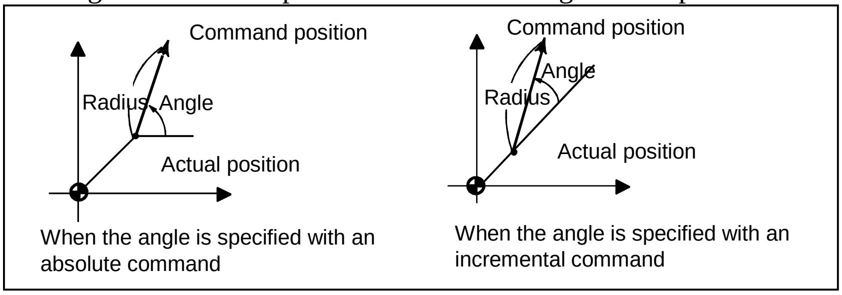
- Setting the origin of the program coordinate system as the origin of the polar coordinate system

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



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

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



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

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

The origin of the polar coordinate system

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

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

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

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

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

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

*2 This means the following.
(1) G16 or the selected plane 1st axis (radius) in G90 is commanded.
(2) The origin of the program coordinate system is set to the origin of the polar coordinate.
(3) Thereafter, the selected plane 2nd axis (angle) is commanded without the address of the selected plane 1st axis (radius).

[Example]
G16 .... The origin of the polar coordinate system is the origin of the program coordinate system.
G91 Y60.0 .... There is not the address of the selected plane 1st axis (radius) and there is the address of the selected plane 2nd axis (angle).

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

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

The radius and angle

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

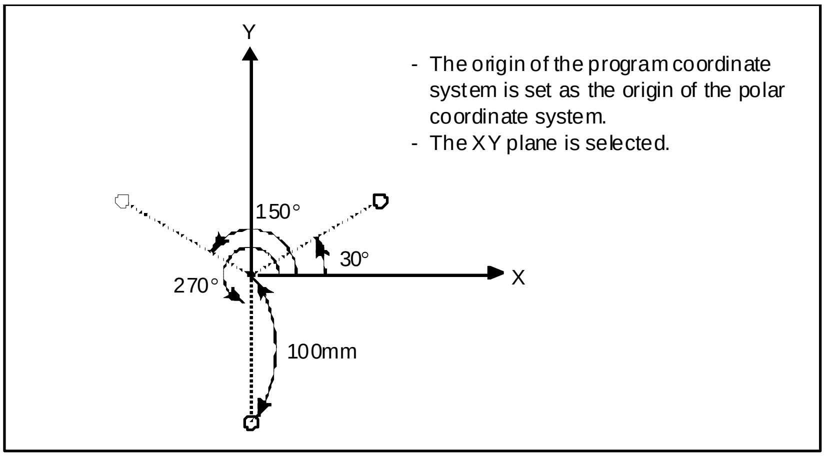
Table 4.1 (b) The radius and the angle

<table>
<tr><th></th><th>PCC = 0</th><th>PCC = 1</th></tr>
<tr><td>When G16 has been commanded</td><td rowspan="2">The radius and the angle become 0.
When the radius or the angle is commanded at the same time, the radius or the angle becomes the value specified in the command.
[Example]
G90 G00 X50.0 Y50.0
G16 .... The radius = 0, the angle = 0.
Y60.0.. The radius = 0, the angle = 60.0.
Therefore, the axes move to (X 0.0, Y 0.0).</td><td rowspan="2">The radius and the angle are calculated from the current position.
When the radius or the angle is commanded at the same time, the radius or the angle becomes the value specified in the command.
[Example]
G90 G00 X50.0 Y50.0
G16.... The radius = 70.710 , the angle = 45.0.
(from the current position (X 50.0, Y 50.0))
Y60.0 . The radius = 70.710 , the angle = 60.0.
Therefore, the axes move to (X 35.355, Y 61.237).</td></tr>
<tr><td>When Polar coordinate command has been commanded after reset</td></tr>
</table>

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

Example

Bolt hole circle



- Specifying angles and a radius with absolute programmings

N1 G17 G90 G16 ; Specifying the polar coordinate command and selecting the XY plane
Setting the origin of the program coordinate system as the origin of the polar coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;
Specifying a distance of 100 mm and an angle of 30 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 program coordinate system as the origin of the polar coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;
Specifying a distance of 100 mm and an angle of 30 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. The command value is not converted by the polar coordinate command.

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

- Rotary axis

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

- Function with limitation when using simultaneously

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

- Retrace
- Nano smoothing (M series)
- Inch/metric conversion

- Functions that cannot be used simultaneously

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

- AI Advanced Preview Control / AI contour control
- Cs contour control
- Optional angle chamfering and corner rounding

NOTE

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

5 FUNCTIONS TO SIMPLIFY PROGRAMMING

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.

> **NOTE**
> When bit 4 (NCD) of parameter No.8137 is 0, this function can be used.

Table 5.1 (a) Canned cycles for drilling

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

Explanation

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

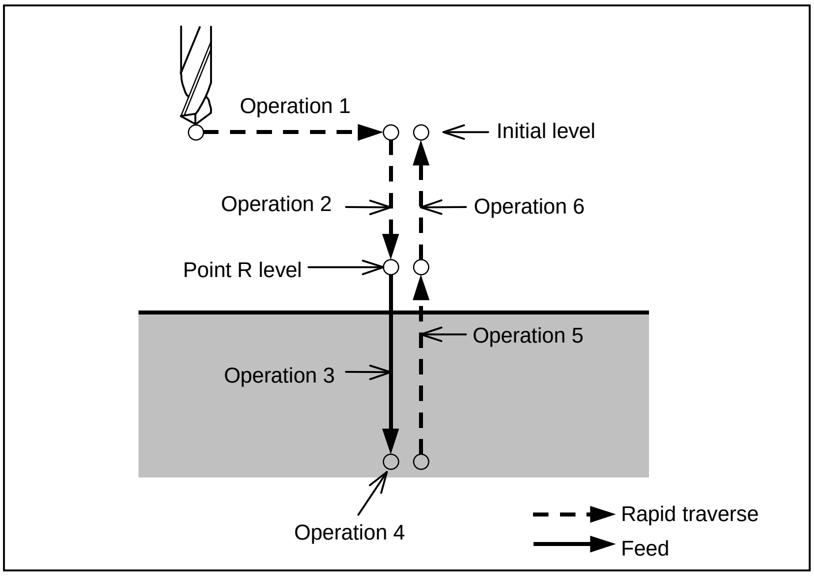


Fig. 5.1 (a) Operation sequence of canned cycle for drilling

- Positioning plane

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

- Drilling axis

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

Table 5.1 (b) Positioning plane and drilling axis

G code	Positioning plane	Drilling axis
G17	Xp-Yp plane	Zp
G18	Zp-Xp plane	Yp
G19	Yp-Zp plane	Xp

Xp: X axis or an axis parallel to the X axis
Yp: Y axis or an axis parallel to the Y axis
Zp: Z axis or an axis parallel to the Z axis

Example

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

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

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

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

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

- Travel distance along the drilling axis G90/G91

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

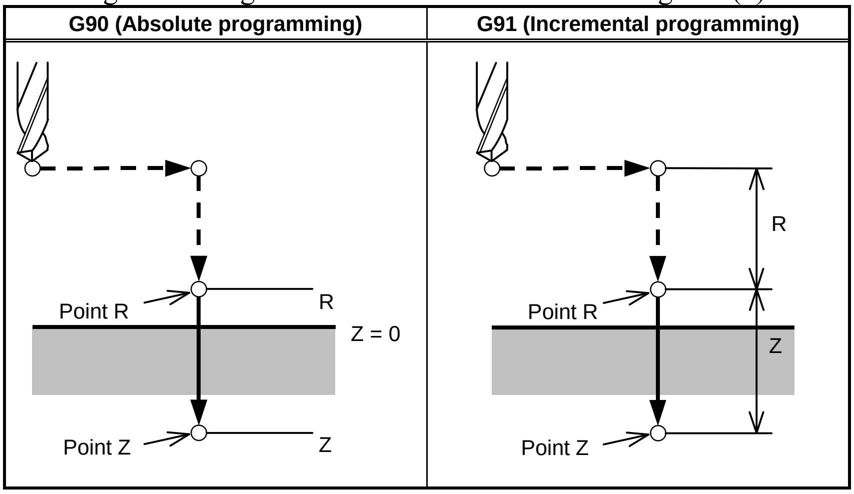


Fig. 5.1 (b) Absolute programming and incremental programming

- Drilling mode

G73, G74, G76, and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.
Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.
Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

- Return point level G98/G99

When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The operations performed when G98 and G99 are specified are shown in Fig. 5.1 (c). Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.
The initial level does not change even when drilling is performed in the G99 mode.

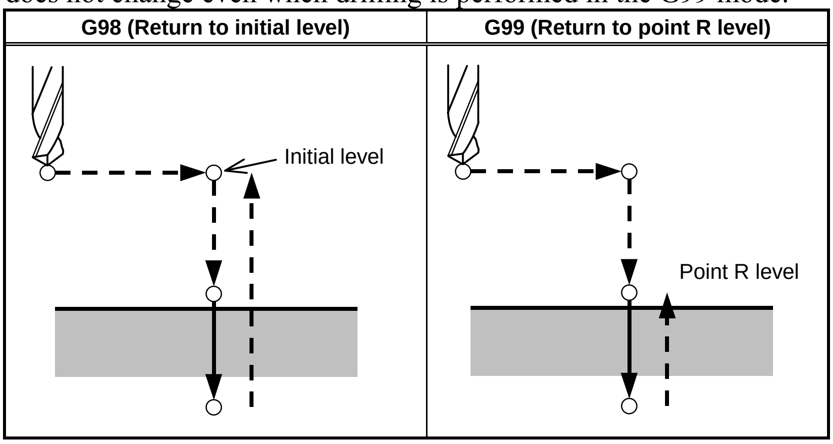


Fig. 5.1 (c) Initial level and point R level

- Repeat

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

Number of repeats K	The maximum command value = 9999

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

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

- Single block

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

- Cancel

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

<u>Group 01 G codes</u>

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
OSS	Oriented spindle stop (The spindle stops at a fixed rotation position)
⇨	Shift (rapid traverse G00)
P	Dwell

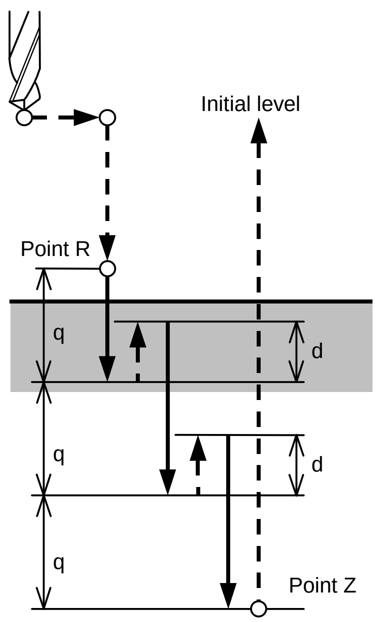
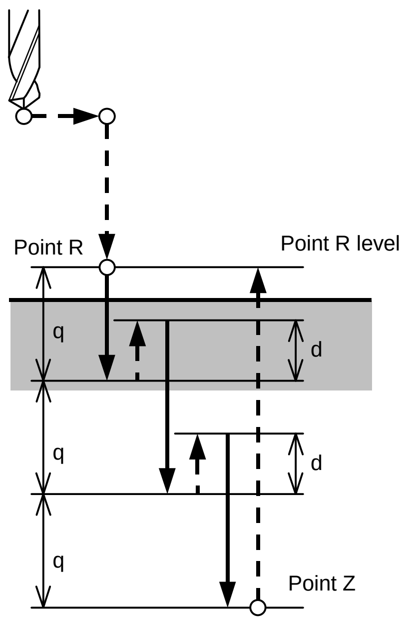
5.1.1 High-Speed Peck Drilling Cycle (G73)

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

Format

G73 X_ Y_ Z_ R_ Q_ 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 No. 5114.
The tool is retracted in rapid traverse.

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Q

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

- Cancel

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

- Tool offset

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

Example

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

5.1.2 Left-Handed Tapping Cycle (G74)

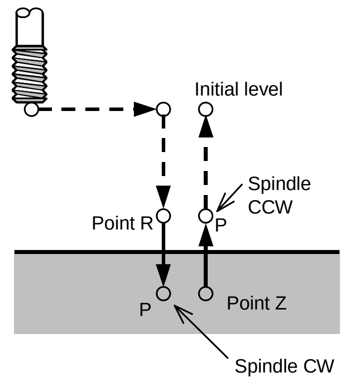
This cycle performs left-handed tapping. In the left-handed tapping cycle, when the bottom of the hole has been reached, the spindle rotates clockwise.

Format

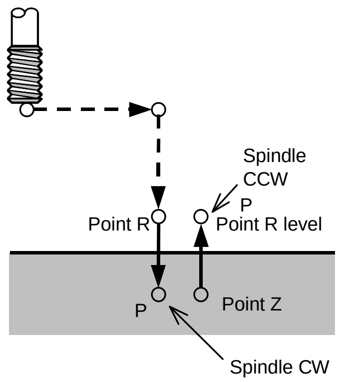
G74 X_ Y_ Z_ R_P_ F_ K_ ;

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

G74 (G98)



G74 (G99)



Explanation

- Operations

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

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

- Spindle rotation

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

- Q command

Refer to “Tapping Cycle (G84)”.

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

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

- Cancel

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

- Tool offset

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

Example

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

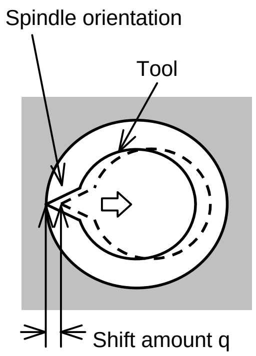
5.1.3 Fine Boring Cycle (G76)

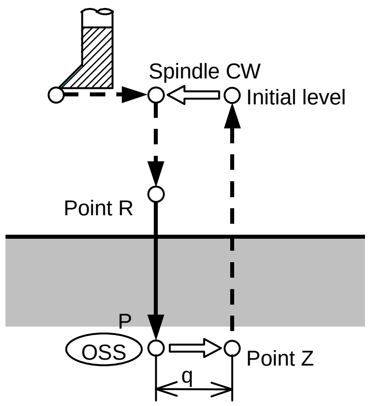
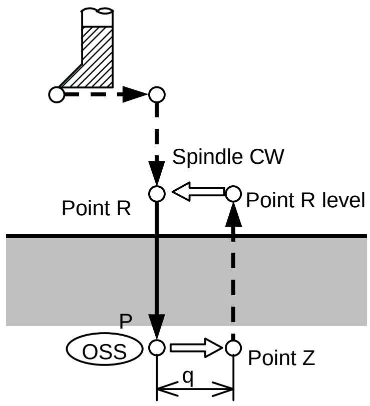
The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.

Format

G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

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



G76(G98)	G76(G99)
	

Explanation

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

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

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

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

Limitation

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

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

- P/Q

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in the parameter No.5148.
Specify P and Q in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data.

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

- Cancel

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

- Tool offset

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

Example

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

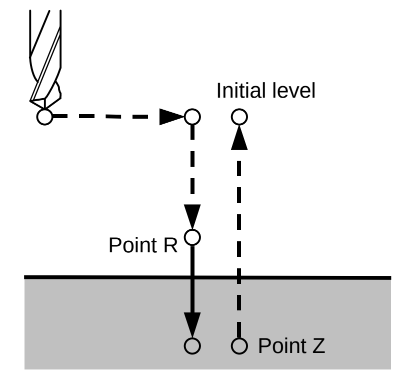
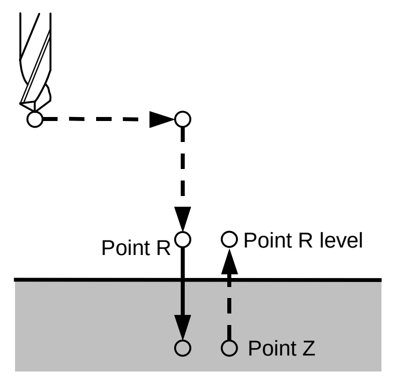
5.1.4 Drilling Cycle, Spot Drilling (G81)

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

Format

G81 X_ Y_ Z_ R_ F_ K_ ;

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

G81 (G98)	G81 (G99)
	

Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

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

- Tool offset

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

Example

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

5.1.5 Drilling Cycle Counter Boring Cycle (G82)

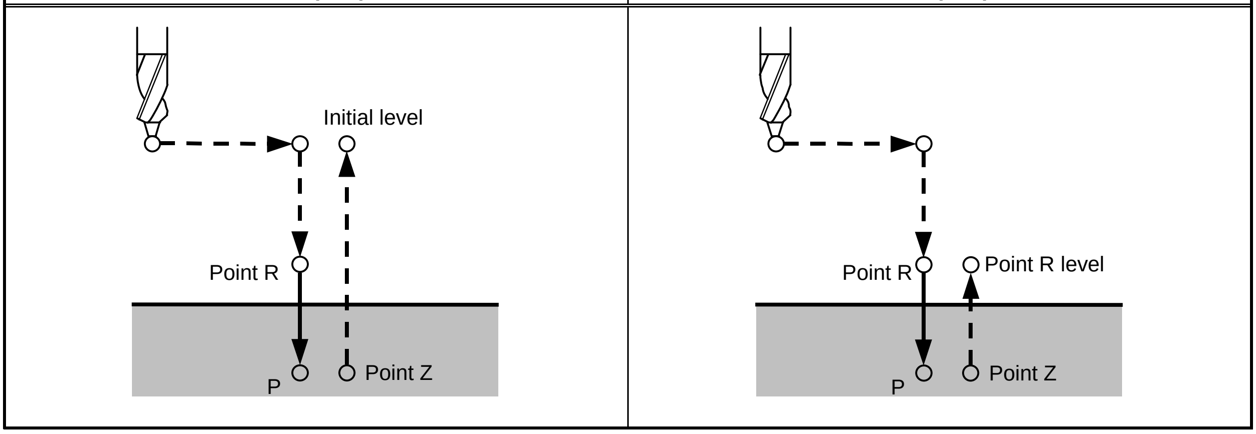
This cycle is used for normal drilling.
Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse.
This cycle is used to drill holes more accurately with respect to depth.

Format

G82 X_ Y_ Z_ R_ P_ F_ K_ ;

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

G82 (G98)	G82 (G99)



Explanation

- Operations

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

When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.

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

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

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

Limitation

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

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

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

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

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

Example

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

5.1.6 Peck Drilling Cycle (G83)

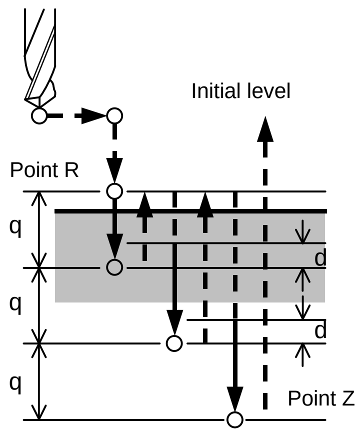
This cycle performs peck drilling.
It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Format

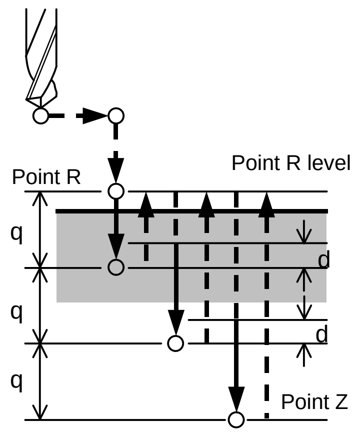
G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

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

5.1.7 Small-Hole Peck Drilling Cycle (G83)

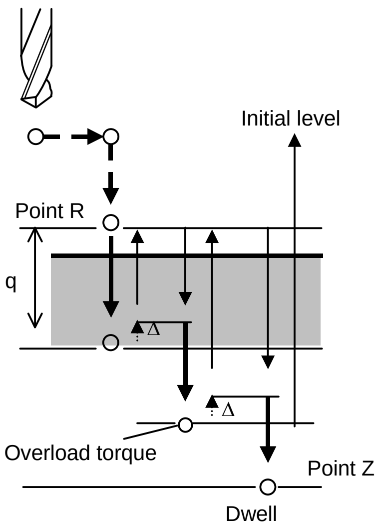
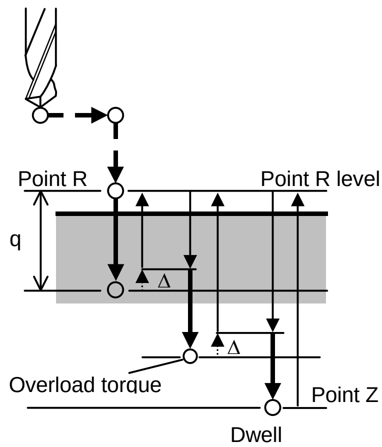
An arbor with the overload torque detection function is used to retract the tool when the overload torque detection signal (skip signal) is detected during drilling. Drilling is resumed after the spindle speed and cutting feedrate are changed. These steps are repeated in this peck drilling cycle.
The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

NOTE
When bit 4 (SPK) of parameter No.8132 is 1, this function can be used.

Format

G83 X_ Y_ Z_ R_ Q_ F_ I_ K_ P_ ;

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

G83 (G98)	G83 (G99)
	

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 No. 5174)
q: Depth of each cut
Path along which the tool travels at the rapid traverse rate
Path along which the tool travels at the programmed cutting feedrate
Path along which the tool travels at the forward or backward rate 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)

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 No. 5163 is specified, the system enters the mode for the small–hole peck drilling cycle.
This M code does not wait for FIN. Care must be taken when this M code is specified with another M code in the same block.
(Example) M03 M| | ; → Waits for FIN.
M| | M03 ; → Does not wait for FIN.

- Specifying a G code

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

- Signal indicating that the cycle is in progress

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

- Overload torque detection signal

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

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

- Changing the drilling conditions

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

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

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 No. 5168, the cutting feedrate is not changed.
The cutting feedrate can be increased up to the maximum cutting feedrate.

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

Spindle speed = S × γ

<First drilling> 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 bit 0 (SBC) of parameter No. 5105.

- Feedrate override

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

- Custom macro interface

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

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

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

NOTE

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

- Positioning to hole position

When positioning the axes to hole position (axes X and Y when XY plane is used) in Small-hole peck drilling cycle, the machining time can be shortened by the spindle is not stopped.

This function is enabled by the parameter SPH(No.5108#6).

Limitation

- Subprogram call

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

Example

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

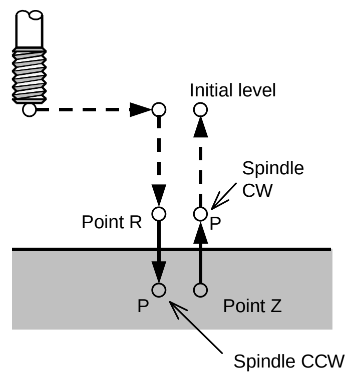
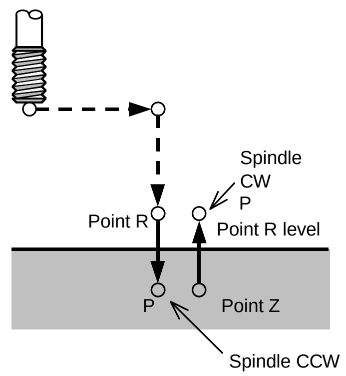
5.1.8 Tapping Cycle (G84)

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

Format

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

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

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.

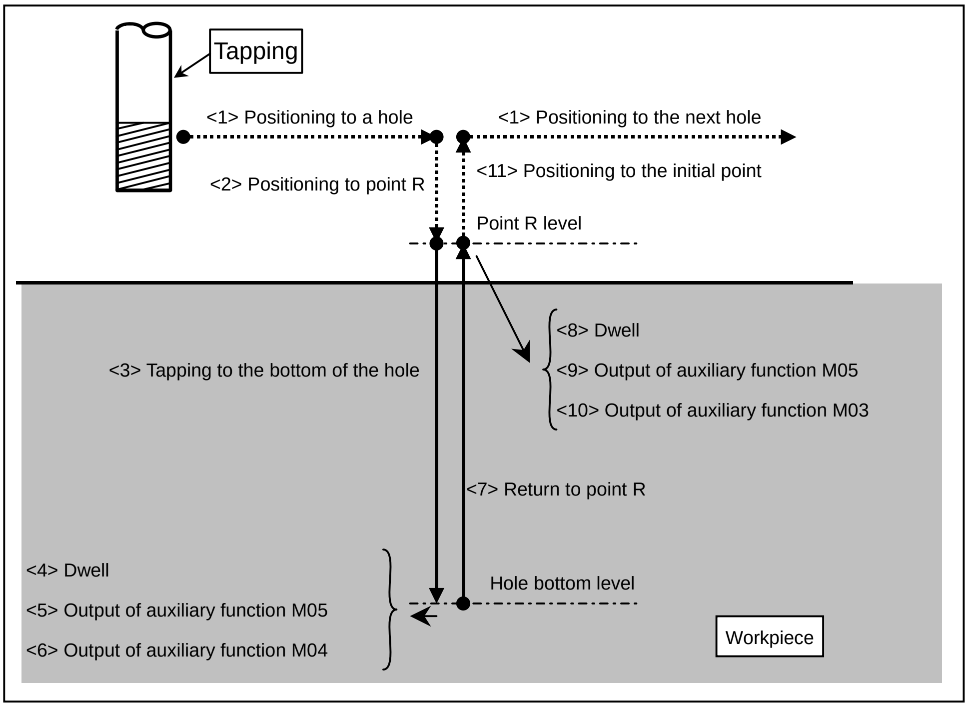
Operation

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

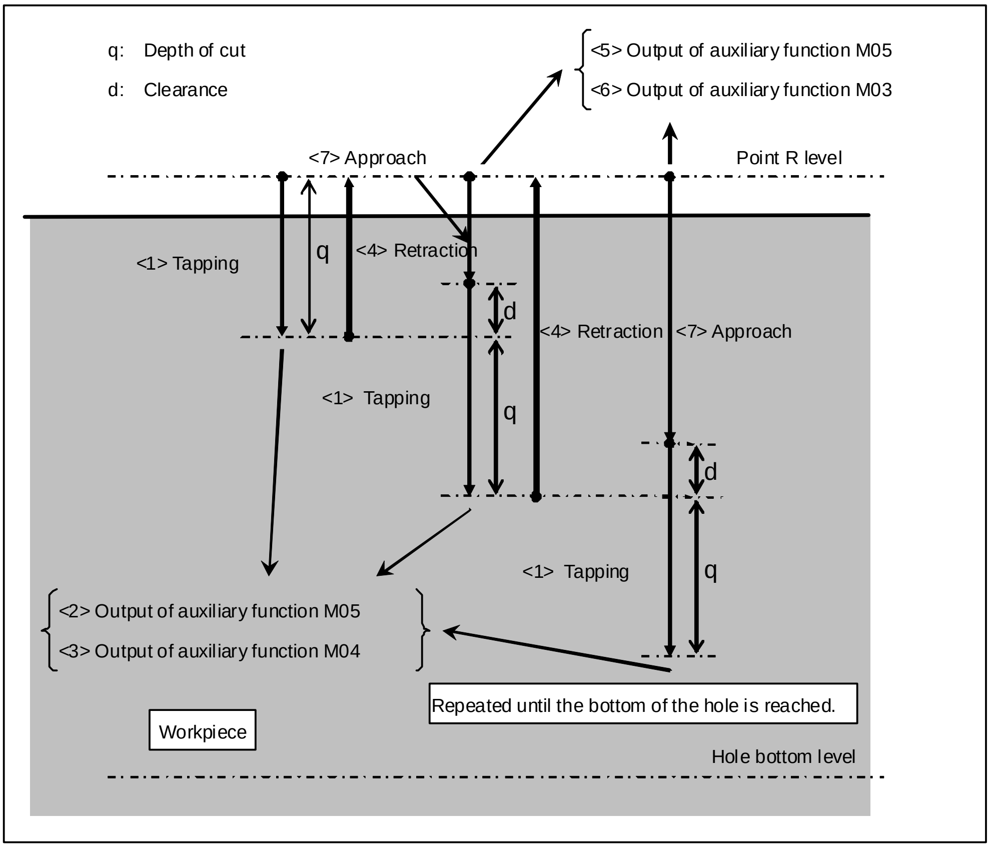
When bit 6 (PCT) of parameter No. 5104 is set to 1 and bit 5 (PCP) of parameter No. 5200 is set to 1, the peck tapping cycle is used.
Step 3 of the tapping cycle operation described above changes as follows:

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.



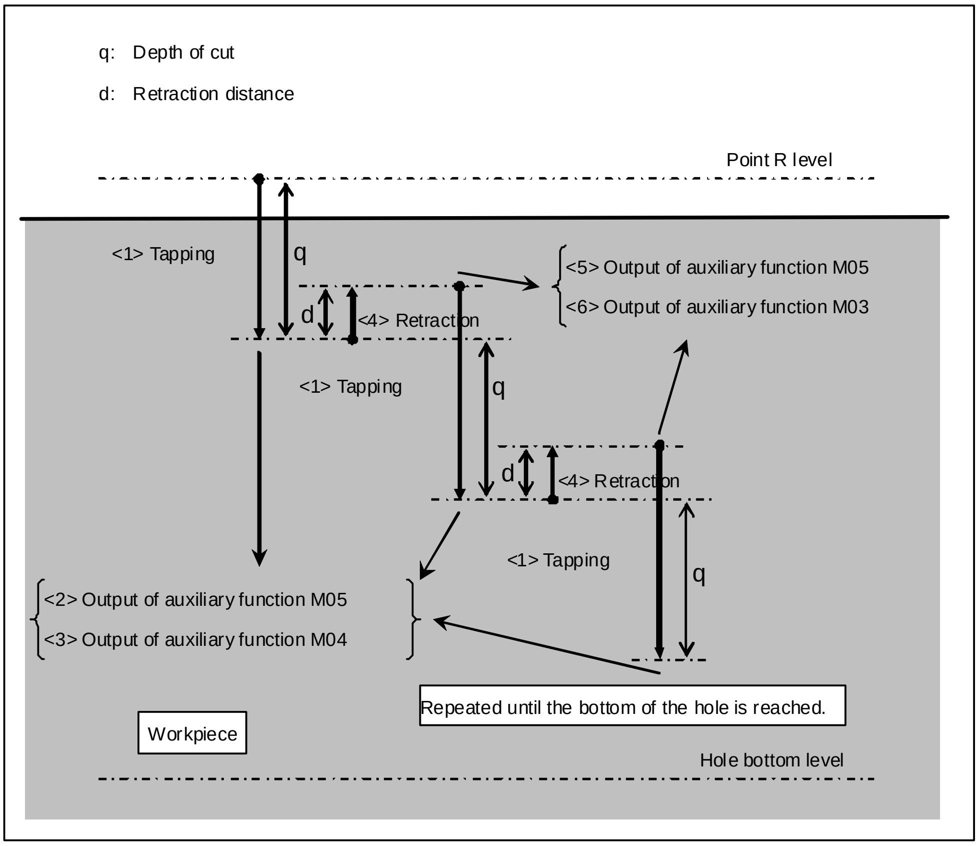
High-speed peck tapping cycle

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 X100.0 Y150.0 Z-100.0 <u>Q20.0</u> ;
   N20 X150.0 Y200.0 ; ← The peck tapping cycle is also performed in this block.
   N30 G80 ;

   Example 2
   N10 G83 X100.0 Y150.0 Z-100.0 <u>Q20.0</u> ;
   N20 G84 Z-100.0 ; ← The peck tapping cycle is also performed in this block.
   N30 G80 ;

   Example 3
   N10 G83 X100.0 Y150.0 Z-100.0 <u>Q20.0</u> ;
   N15 <u>G80</u> ; ← The canned cycle mode is canceled.
   N20 G84 Z-100.0 ;
   N30 G80 ;

Example 4
N10 G83 X100.0 Y150.0 Z-100.0 Q20.0 ;
N20 G84 Z-100.0 Q0 ; ←Q0 is added.
N30 G80 ;

2. The unit for the drilling axis is used as the unit of Q. Any sign is ignored.

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

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

- Cancel

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

Example

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

5.1.9 Boring Cycle (G85)

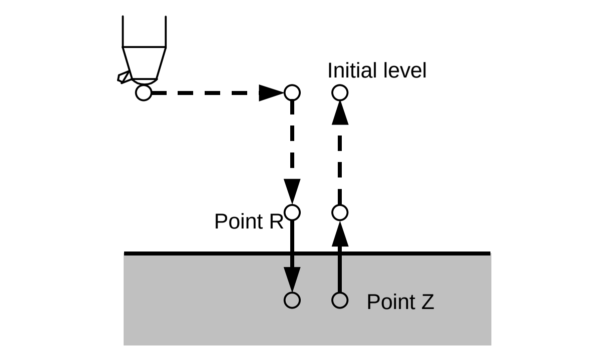
This cycle is used to bore a hole.

Format

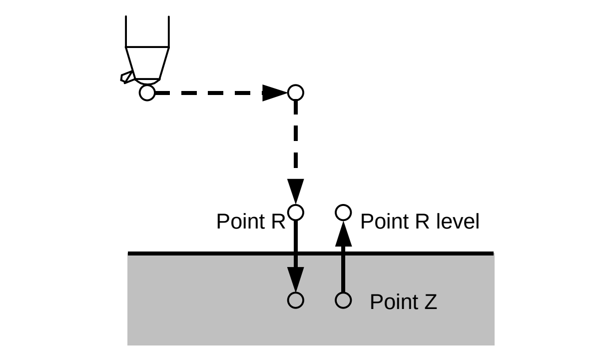
G85 X_ Y_ Z_ R_ F_ K_ ;

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

G85 (G98)



G85 (G99)



Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

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

- Tool offset

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

Example

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

5.1.10 Boring Cycle (G86)

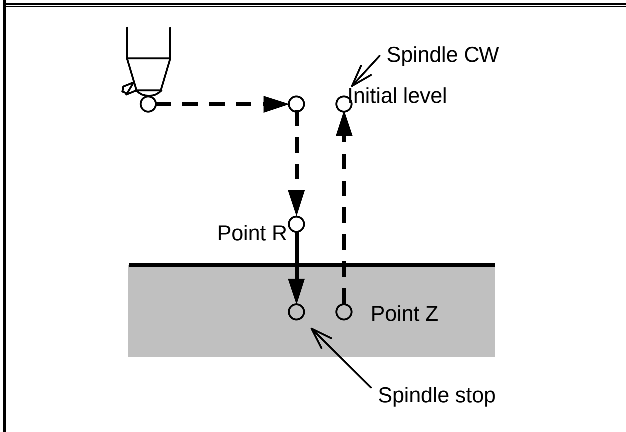
This cycle is used to bore a hole.

Format

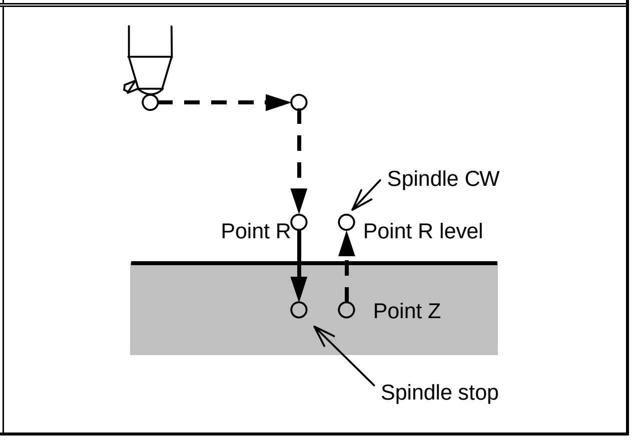
G86 X_ Y_ Z_ R_ F_ K_ ;

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

G86 (G98)



G86 (G99)



Explanation

- Operations

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

- Spindle rotation

Before specifying G86, use an auxiliary function (M code) to rotate the spindle.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation.

In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- Cancel

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

- Tool offset

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

Example

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

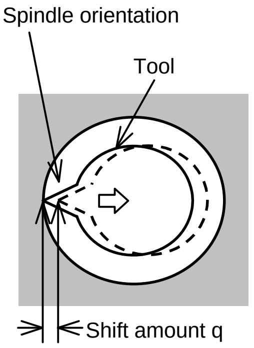
5.1.11 Back Boring Cycle (G87)

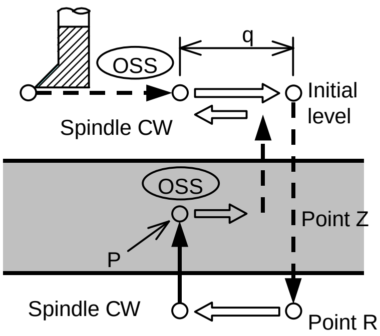
This cycle performs accurate boring.

Format

G87 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

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



G87 (G98)	G87 (G99)
	Not used

Explanation

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P/Q

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in the parameter No. 5148.
Specify P and Q in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data.

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

- Cancel

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

- Tool offset

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

Example

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

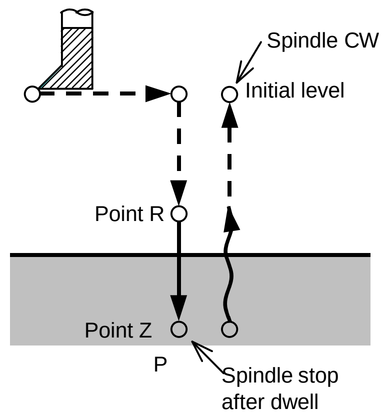
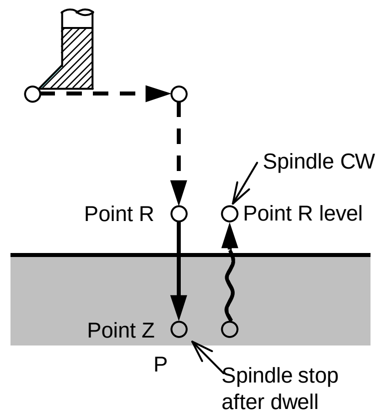
5.1.12 Boring Cycle (G88)

This cycle is used to bore a hole.

Format

G88 X_ Y_ Z_ R_ P_ F_ K_ ;

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

G88 (G98)	G88 (G99)
	

Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

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

- Cancel

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

- Tool offset

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

Example

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

5.1.13 Boring Cycle (G89)

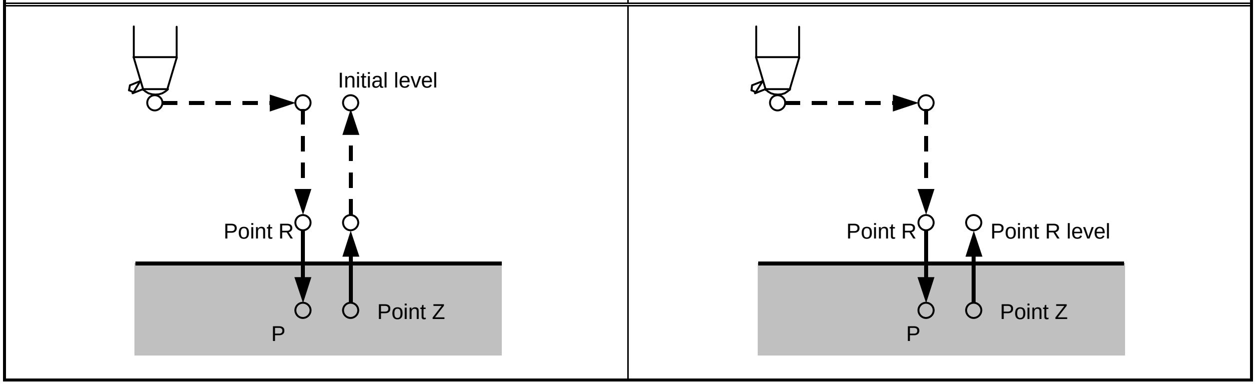
This cycle is used to bore a hole.

Format

G89 X_ Y_ Z_ R_ P_ F_ K_ ;

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

G89 (G98)	G89 (G99)



Explanation

- Operations

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

- Spindle rotation

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

- Auxiliary function

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

- Tool length compensation

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

Limitation

- Axis switching

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

- Drilling

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

- P

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

- Cancel

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

- Tool offset

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

Example

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

5.1.14 Canned Cycle Cancel for Drilling (G80)

G80 cancels canned cycles for drilling.

Format

G80 ;

Explanation

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

Example

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

5.1.15 Example for Using Canned Cycles for Drilling

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

Program example

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

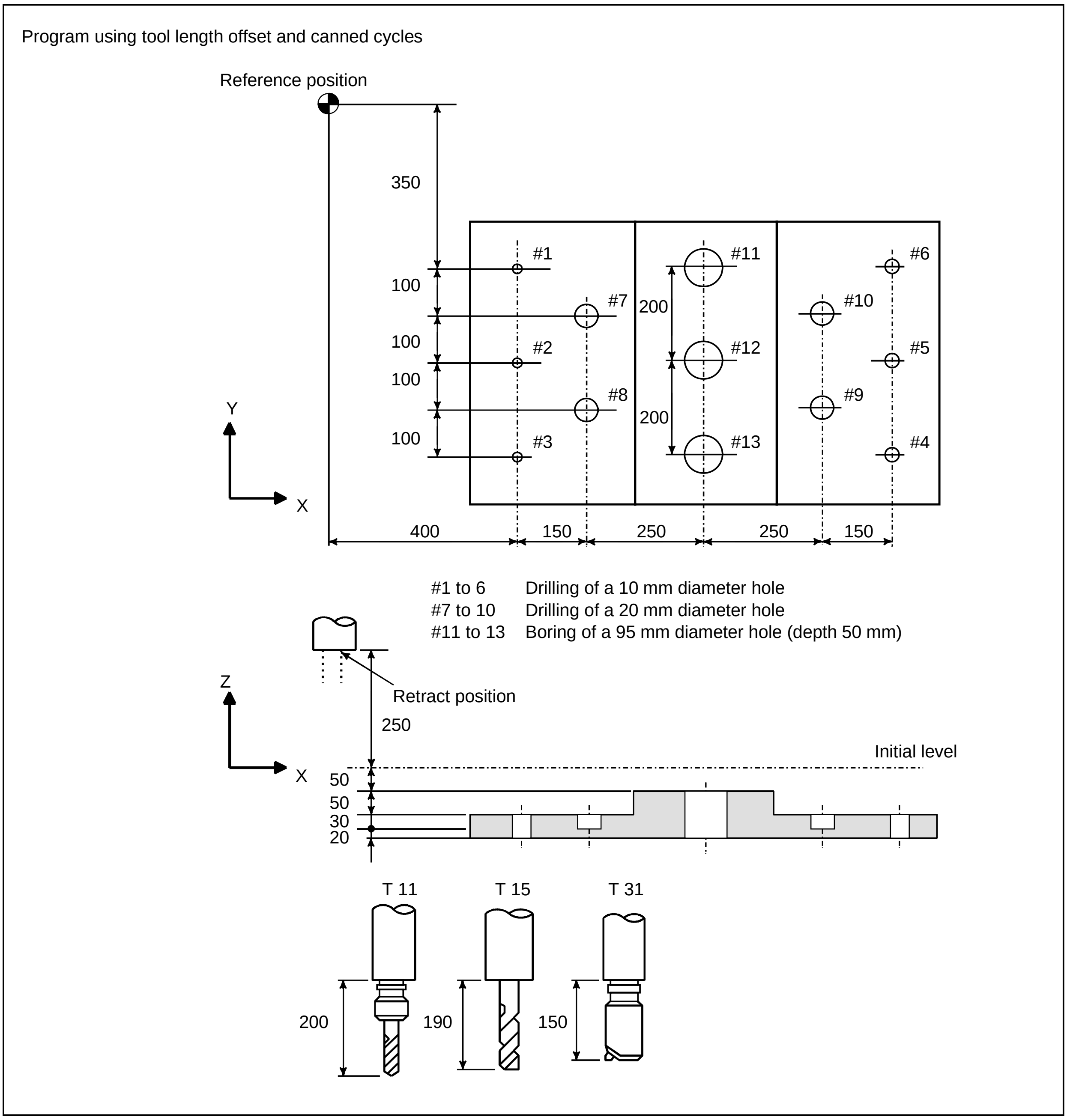


Fig. 5.1.15 (a) Example for using canned cycles for drilling

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

Overview

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

Explanation

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

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

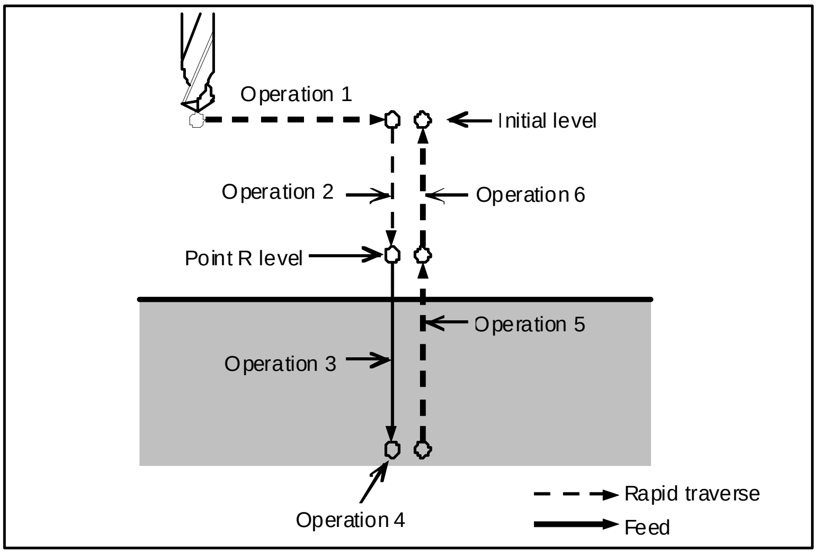


Fig. 5.1.16 (a) Operation sequence of canned cycle for drilling

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

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

Applied of speed-up of each command

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

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

Table of canned cycle for drilling (Series 10/11 format)

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

5.2 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.

NOTE
When bit 3 (NRG) of parameter No.8135 is 0, this function can be used.

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 sped up.

Format

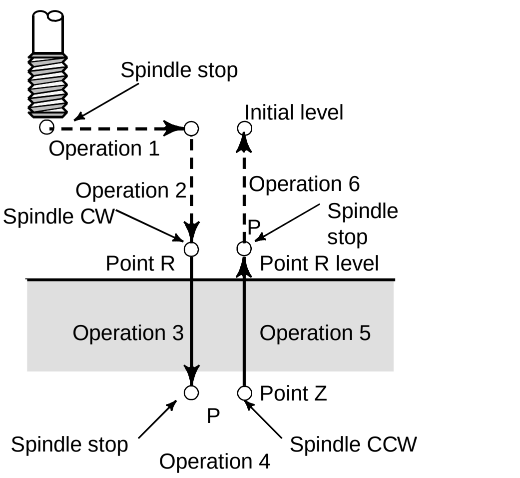
G84 X_ Y_ Z_ R_ P_ F_ K_ ;

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

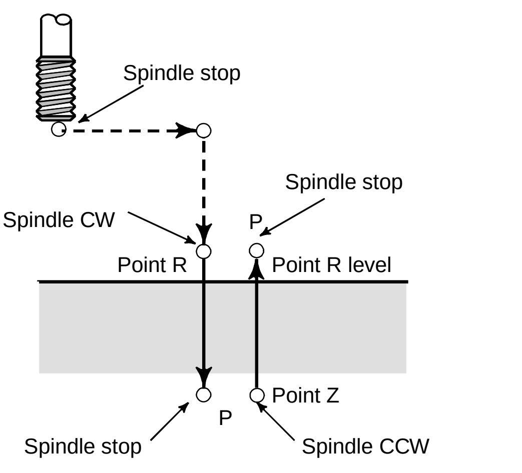
G84.2 X_ Y_ Z_ R_ P_ F_ L_ ;
(Series 10/11 format)

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 (bit 0 (G84) of parameter No. 5200 set to 1).

- Thread lead

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

- Tool length compensation

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

- Series 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

Details are given later.

- Dry run

Dry run can be executed also in G84 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock

Machine lock can be executed also in G84 (G74).
When G84 (G74) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

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

- Interlock
Interlock can also be applied in G84 (G74).

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

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

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

Limitation

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

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

- Distribution amount for the spindle
The maximum distribution amount is as follows (displayed on diagnosis data No. 451):
- For a serial spindle: 32,767 pulses per 8 ms

This amount is changed according to the gear ratio setting for the position coder or rigid tapping command. If a setting is made to exceed the upper limit, alarm PS0202 is issued.

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

- Unit of F command

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

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

- P

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03 or G60 (when the bit 0 (MDL) of parameter No. 5431 is set to 1)) and G84 in a single block. Otherwise, G84 will be canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Program restart

A program cannot be restarted during rigid tapping.

- Subprogram call

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

- Constant surface speed control

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

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.

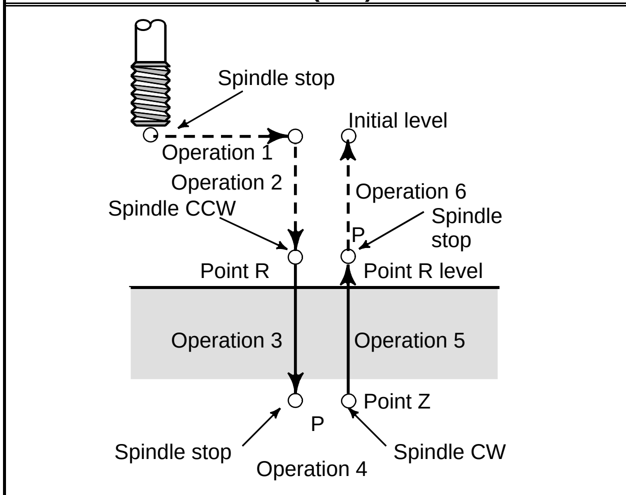
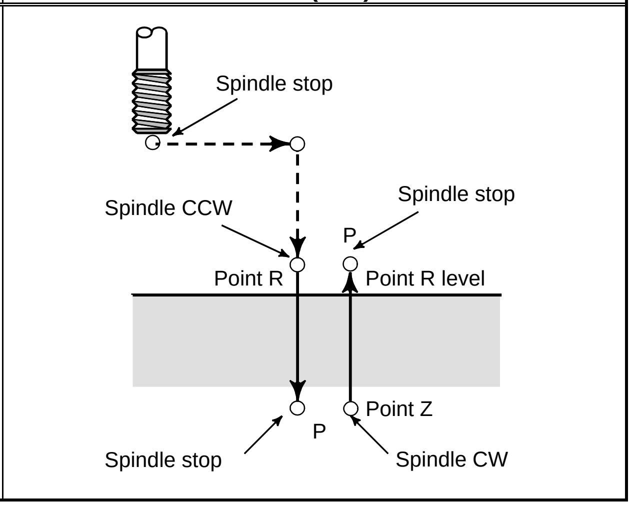
Format

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. (bit 0 (G84) of parameter No. 5200 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 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

Details are given later.

- Dry run
Dry run can be executed also in G84 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock
Machine lock can be executed also in G84 (G74).
When G84 (G74) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

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

- Interlock
Interlock can also be applied in G84 (G74).

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

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

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

Limitation

- Axis switching

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

- S command

- Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm PS0200.
- When the rigid tapping canned cycle is cancelled, the S command used for rigid tapping is cleared to S0.

- Distribution amount for the spindle

The maximum distribution amount is as follows (displayed on diagnosis data No. 451):

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

This amount is changed according to the gear ratio setting for the position coder or rigid tapping command. If a setting is made to exceed the upper limit, alarm PS0202 is issued.

- F command

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

- Unit of F command

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

- M29

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

- P

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

- Cancel

Do not specify a G code of the 01 group (G00 to G03 or G60 (when the bit 0 (MDL) of parameter No. 5431 is set to 1)) and G74 in a single block. Otherwise, G74 will be canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Subprogram call

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

- Constant surface speed control

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

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 bit 5 (PCP) of parameter No. 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)

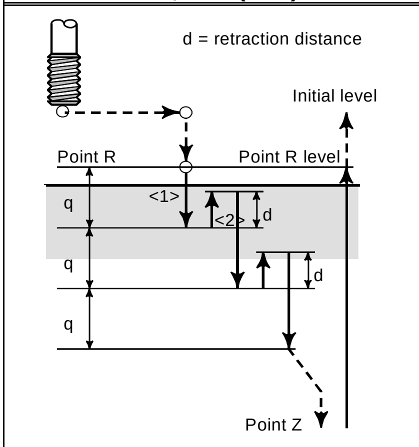
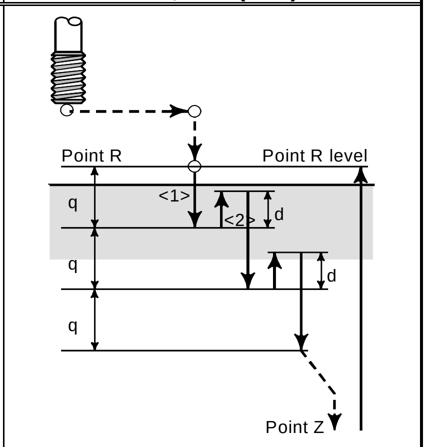
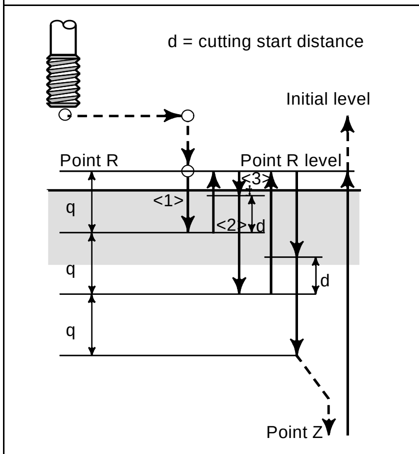
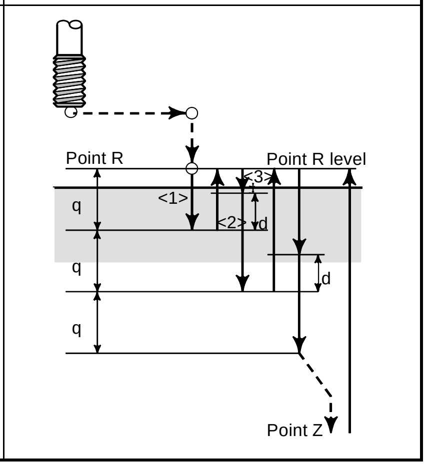
G84.2 (or G84.3) X_ Y_ Z_ R_ P_ Q_ F_ L_ ;

(Series 10/11 format)

L_	:	Number of repeats (if required)

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

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

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

Explanation

- High-speed peck tapping cycle

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

- Peck tapping cycle

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

- Acceleration/deceleration after interpolation

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

- Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

- Override

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

- Extraction override
- Override signal

Details are given later.

- Dry run

Dry run can be executed also in G84 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock

Machine lock can be executed also in G84 (G74).
When G84 (G74) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

- Reset

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

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

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

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

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

Limitation

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

- S command
- Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm PS0200.
- When the rigid tapping canned cycle is cancelled, the S command used for rigid tapping is cleared to S0.

- Distribution amount for the spindle
The maximum distribution amount is as follows (displayed on diagnosis data No. 451):
- For a serial spindle: 32,767 pulses per 8 ms
  This amount is changed according to the gear ratio setting for the position coder or rigid tapping command. If a setting is made to exceed the upper limit, alarm PS0202 is issued.

- F command
Specifying a value that exceeds the upper limit of cutting feedrate causes alarm PS0011.

- Unit of F command

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

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

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

- Cancel
Do not specify a group 01 G code (G00 to G03 or G60 (when the bit 0 (MDL) of parameter No. 5431 is set to 1)) and G84 in the same block. If they are specified together, G84 is canceled.

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

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

- Amount of return and cutting start distance

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

- Constant surface speed control

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

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%.

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

The override to be applied is determined according to the setting of parameters and that in the command as shown in the Table 5.2.5.1 (a).

Table 5.2.5.1 (a)

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		

NOTE

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

2 Do not use a minus sign in the value specified at address "J".
If a minus sign is used, a value outside the range between 100% to 200% is assumed.

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

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

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
(When the second feedrate override signal turns “1”, the second feedrate override is applied to the feedrate to which feedrate override is applied.)
- Canceling override using the override cancel signal

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

- At cutting

- When the override cancel signal is set to “0”: value specified by the override signal
- When the override cancel signal is set to “1”: 100%

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

NOTE

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

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

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

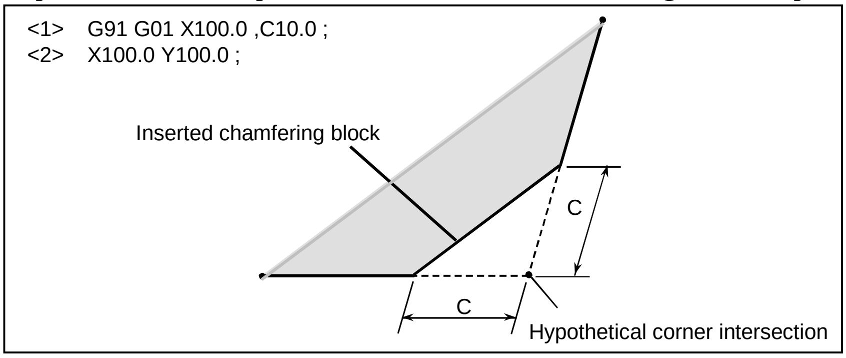
, C_	Chamfering
, R_	Corner R

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.

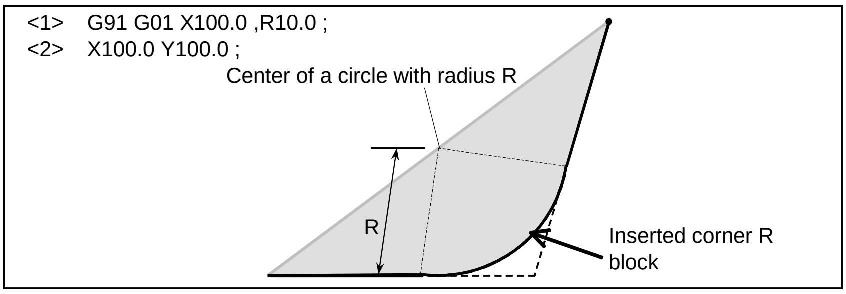
- Chamfering

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

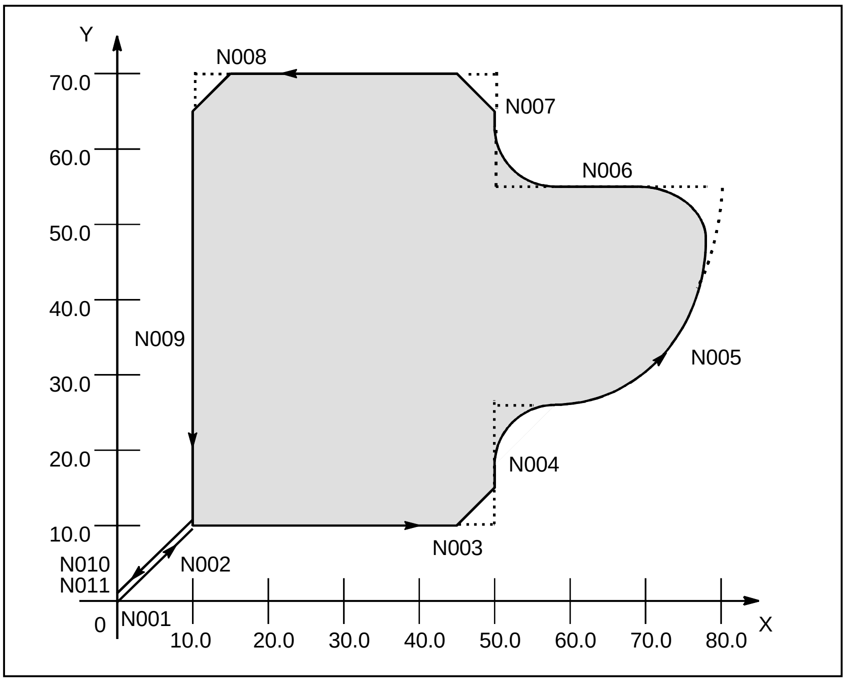
After R, specify the radius for corner R.



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 Y55.0 R30.0 ,R8.0 ;
N006 G01 X50.0 ,R8.0 ;
N007 Y70.0 ,C5.0 ;
```

N008 X10.0 ,C5.0 ;
N009 Y10.0 ;
N010 G00 X0 Y0 ;
N011 M0;



Limitation

- Invalid specification

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

- Next block

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

- Exceeding the move range

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

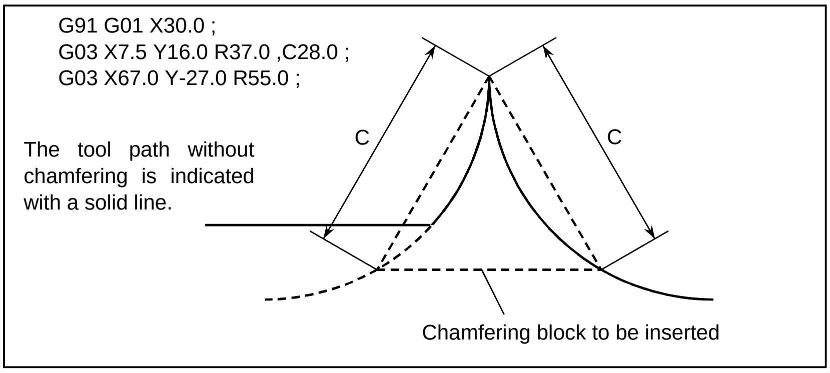


Fig 5.3 (a) Exceeding the move range

- 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.

NOTE

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

5.4 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**
> When bit 3 (IXC) of parameter No.8132 is 1, this function can be used.

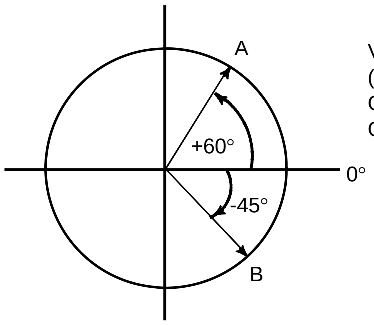
Explanation

- Indexing position

Specify an indexing position with address A, B, or C (set to bit 0 (ROTx) of parameter No. 1006).
The indexing position is specified by either of the following (depending on bit 4 of parameter G90 No.5500):

1. Absolute value only
2. Absolute or incremental value depending on the specified G code: G90 or G91

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



Value specified for rotation from A to B
(case 2 described above)
G90 B-45.0 ; or
G91 B-105.0;

- Direction and value of rotation

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

1. Using the auxiliary function specified in parameter No. 5511 (Address) (Indexing position) (Miscellaneous function); Rotation in the negative direction (Address) (Indexing position); Rotation in the positive direction (No auxiliary functions are specified.)
   An angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° when bit 2 (ABS) of parameter No. 5500 specifies this option.
   For example, when G90 B400.0 (auxiliary function); is specified at a position of 0 , the table is rotated by 40° in the negative direction.
2. Using no auxiliary functions
   By setting to bits 2 (ABS), 3 (INC), and 4 (G90) of parameter No. 5500, operation can be selected from the following two options.
   Select the operation by referring to the manual written by the machine tool builder.
   (1) Rotating in the direction in which an angular displacement becomes shortest
   This is valid only in absolute programming. A specified angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° when bit 2 (ABS) of parameter No. 5500 specifies this option.

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

(2) Rotating in the specified direction
In the absolute programming, the value set in bit 2 (ABS) of parameter No. 5500 determines whether an angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360°.
In the incremental programming, the angular displacement is not rounded down. For example, when G90 B720.0; is specified at a position of 0, the table is rotated twice in the positive direction, when the angular displacement is not rounded down.

- Feedrate

The table is always rotated around the indexing axis in the rapid traverse mode.
Dry runs cannot be executed for the indexing axis.

> ⚠ **WARNING**
> 1 If a reset is made during indexing of the index table, a reference position return must be made before each time the index table is indexed subsequently.
> 2 For a path on which the index table indexing function is not to be used, disable the index table indexing function (set bit 0 (ITI) of parameter No. 5501 to 0).

> **NOTE**
> 1 If an index table indexing axis and another controlled axis are specified in the same block either alarm PS1564 is issued or the command is executed, depending on bit 6 (SIM) of parameter No. 5500 and bit 0 (IXS) of parameter No. 5502.
> 2 The auxiliary function specifying a negative direction is processed in the CNC. The relevant M code signal and completion signal are sent between the CNC and the machine.
> 3 If a reset is made while waiting for completion of clamping or unclamping, the clamp or unclamp signal is cleared and the CNC exits the completion wait state.

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

5.5 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.

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

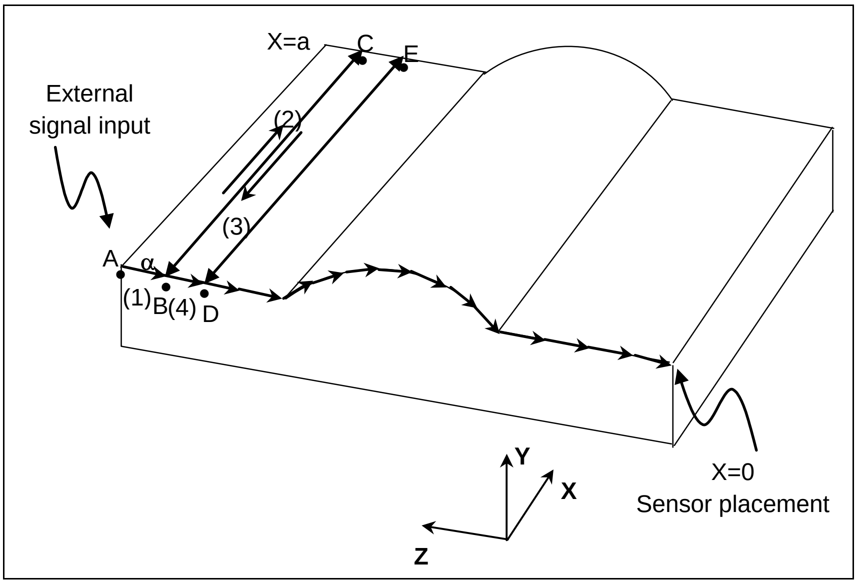


Fig. 5.5 (a)

For example, it is possible to machine a workpiece with a profile programmed with linear interpolation, circular interpolation, and linear interpolation on the YZ plane, such as that shown in the Fig. 5.5 (a).
A sensor is placed at a X = 0 position so that the switch on the machine operator's panel is input when the sensor detects the grinding wheel. When the program is started at point A, the machine is first placed in the state in which it waits for the input of the switch on the machine operator's panel.
Then, when the sensor detects the grinding wheel, the switch on the machine operator's panel is input, and the machine makes a cut by the constant amount α along the programmed profile on the specified YZ plane and moves to point B (operation (1)). The machine is then placed in the state in which it waits for the input of the switch on the machine operator's panel again, and performs a grinding operation along the X-axis. It grinds from point B to point C (operation (2)) and grinds back from point C to point B (operation (3)). When the machine returns to point B, the sensor detects the grinding wheel again, and the switch on the machine operator's panel is input, so that the machine makes a cut by the amount of α and moves to point D (operation (4)). At point D, the machine performs a grinding operation along the X-axis. Afterwards, each time the switch on the machine operator's panel is input, the machine makes a cut by the amount of α along the profile program, so that the workpiece is machined to a profile such as that shown in the Fig. 5.5 (a).

Format

```
G161 R_ ;
  Profile program

G160 ;
```

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

Explanation

- G161 R_

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

- Profile program

Program the profile of a workpiece on the YZ plane, using linear interpolation (G01) or circular interpolation (G02, G03). Multiple-block commands are possible.
When a profile program is started, the machine is placed in the state in which it waits for the input of the 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 depth 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, the specified depth of cut is not correct.

- Grinding operation

In this operation mode, a grinding operation that causes the machine to move to and from the grinding wheel cannot be specified in an NC program. Perform such an operation 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 ;
   :
```

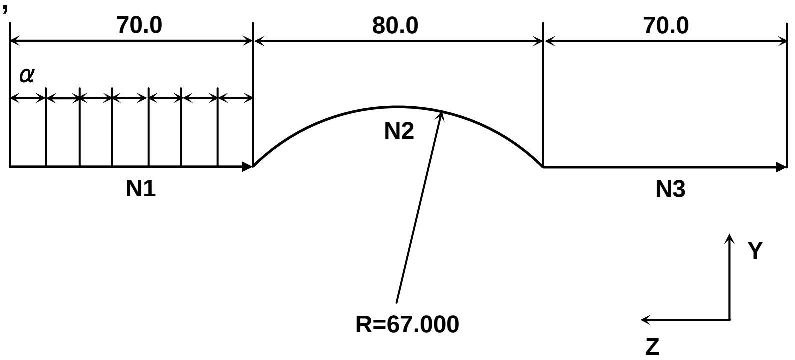


Fig. 5.5 (b)

The program above causes the machine to move by 10.000 along the machining profile in the Fig. 5.5 (b) each time the switch on the machine operator's panel is input.
α = Travel distance at each input of the switch on the machine operator's panel.
The feedrate is the one specified in the program with an F code.

Note

NOTE

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

5.6 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
- 3-dimensional coordinate conversion
- One-digit F code feed
- Tool length compensation

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

⚠ **WARNING**

The G codes for canned grinding cycles G75, G77, G78, and G79 are G codes of group 01. A G code for cancellation such as G80 used for a canned cycle for drilling is unavailable. By specifying a G code of group 00 other than G04, modal information such as a depth of cut is cleared but no canned grinding cycle can be canceled. To cancel a canned grinding cycle, a G code of group 01 other than G75, G77, G78, and G79 needs to be specified. So, when switching to another axis move command from canned grinding cycles, for example, be sure to specify a G code of group 01 such as G00 or G01 to cancel the canned grinding cycle. If another axis move command is specified without canceling the canned grinding cycle, an unpredictable operation can result because of continued cycle operation.

NOTE

1 This function is included in the option "Grinding function A" and "Grinding function B".
  To use this function, any one of the above option is required.
2 If the G code for a canned grinding cycle (G75, G77, G78, or G79) is specified, the canned grinding cycle is executed according to the values of I, J, K,   , 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_   _ R_ F_ P_ ;
  ; ← The canned grinding cycle is executed even if an empty block is specified.
  %
3 When switching from a canned cycle for drilling to a canned grinding cycle, specify G80 to cancel the canned cycle for drilling.
4 When switching from a canned grinding cycle to another axis move command, cancel the canned cycle according to the warning above.

5.6.1 Plunge Grinding Cycle (G75)

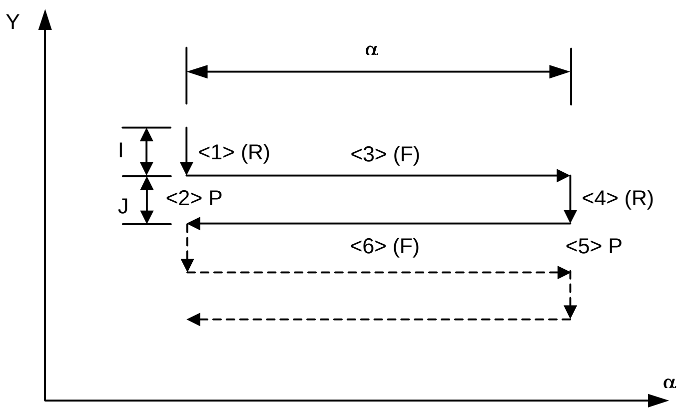
A plunge grinding cycle can be executed.

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.

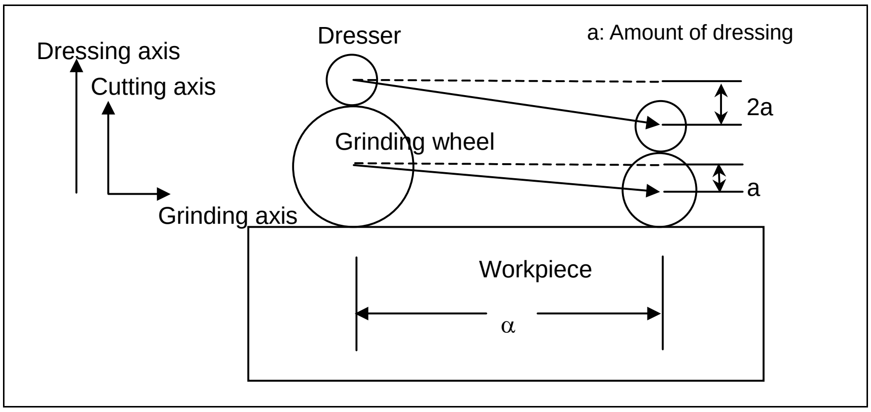
<6> Grinding (return direction)

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 with address L. Up to 400 offset numbers (L1 to L400) can be specified. Establish correspondence between compensation amounts and offset numbers, and set it in offset memory in advance, using the MDI unit.
No compensation operation is performed in the following cases:

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



Limitations

- Cutting axis

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

- Grinding axis

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

- Dressing axis

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

- α,I,J,K

α, I, J, and K commands are all incremental ones.

Spark-out (execution of movement in the grinding direction only) occurs in the following cases:

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

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

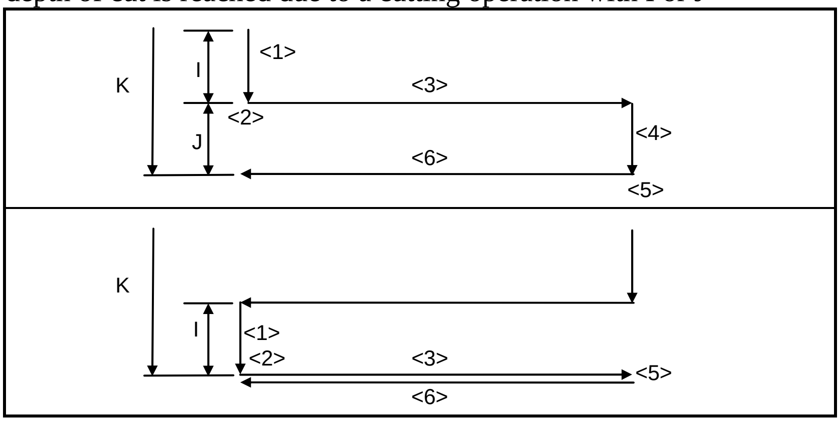
- Clearing

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

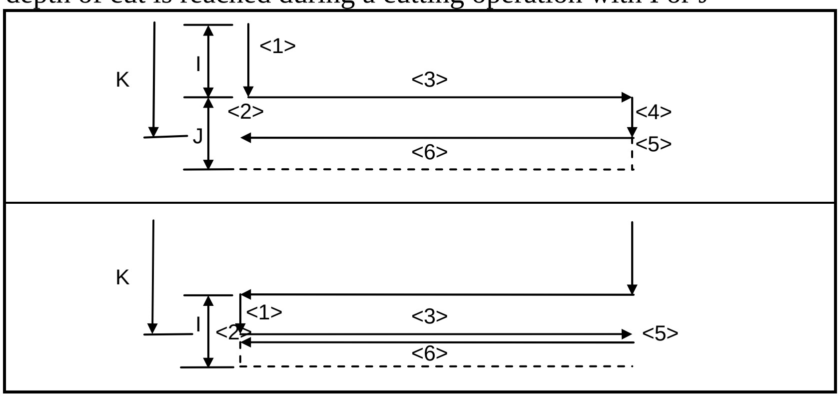
- Operation to be performed if the total depth of cut is reached

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

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



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



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

5.6.2 Direct Constant-Dimension Plunge Grinding Cycle (G77)

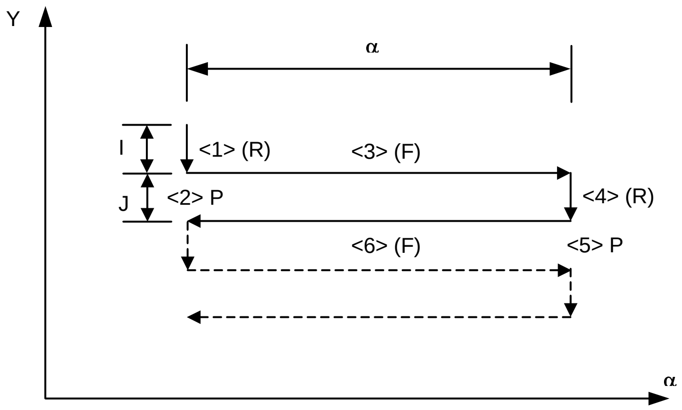
A direct constant-dimension plunge grinding cycle can be performed.

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

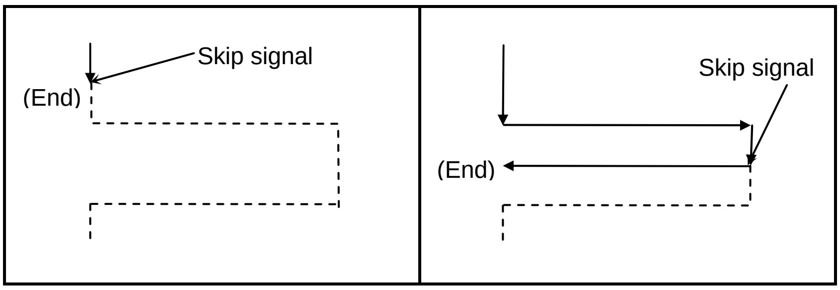
If the continuous dressing function is enabled, the grinding-wheel cut and the dresser cut are continuously compensated for according to the dressing amount specified with L during the execution of grinding. For details, see Explanation of G75.

- 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).

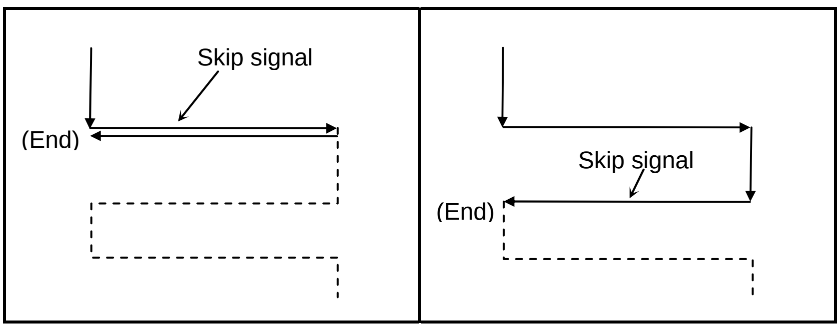
The following shows the operation to be performed when a skip signal is input in each operation sequence.

- 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)

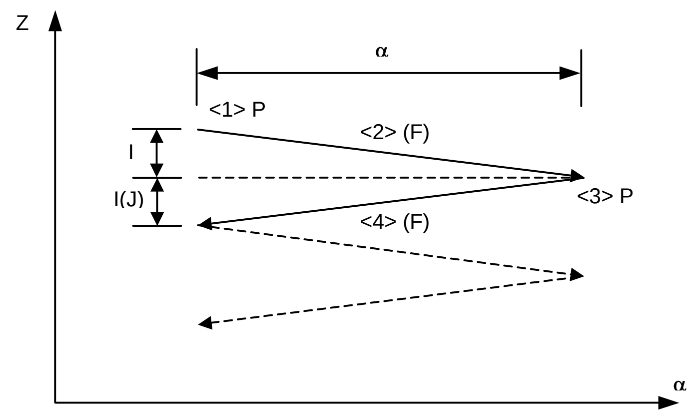
A continuous-feed surface grinding cycle can be performed.

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.

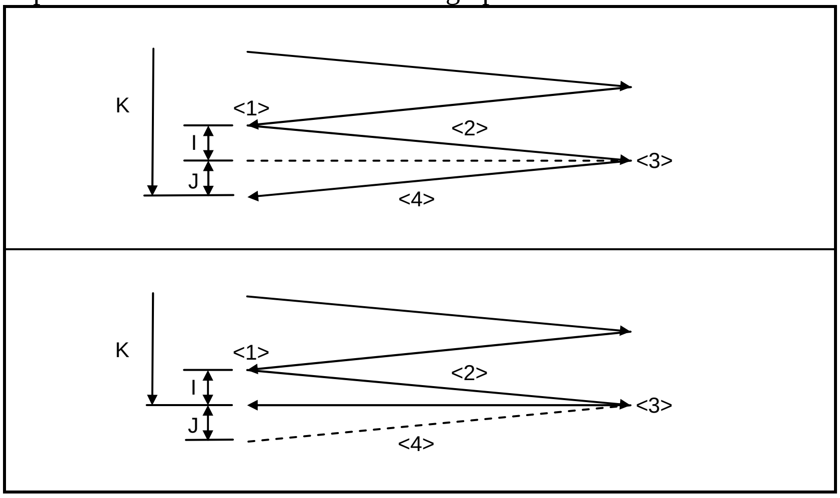
- Clearing

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.

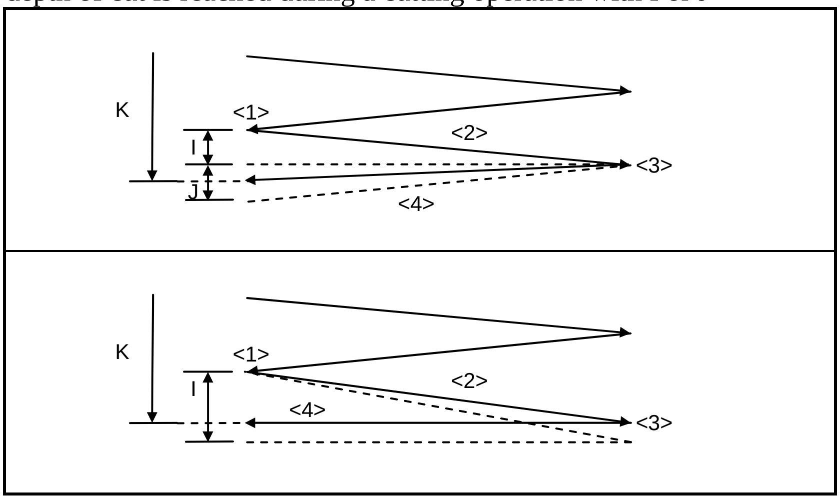
- Operation to be performed if the total depth of cut is reached

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)

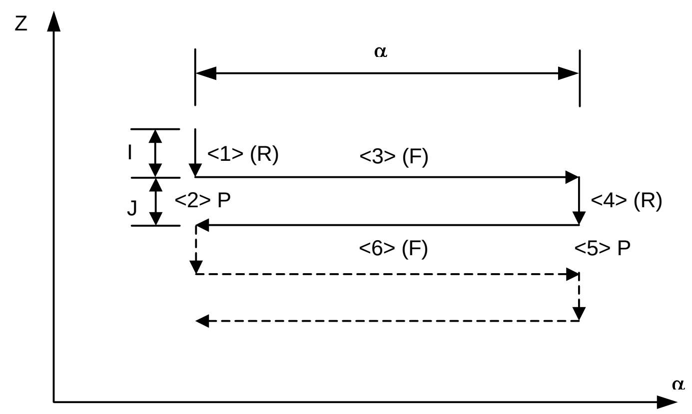
An intermittent-feed surface grinding cycle can be performed.

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.

NOTE

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

5.7 TILTED WORKING PLANE INDEXING

5.7.1 Tilted Working Plane Indexing

Overview

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. The feature coordinate system is defined in the workpiece coordinate system.
For explanations about the relationship between the feature coordinate system and workpiece coordinate system, see Fig. 5.7.1 (a).

NOTE
This function is an optional function.

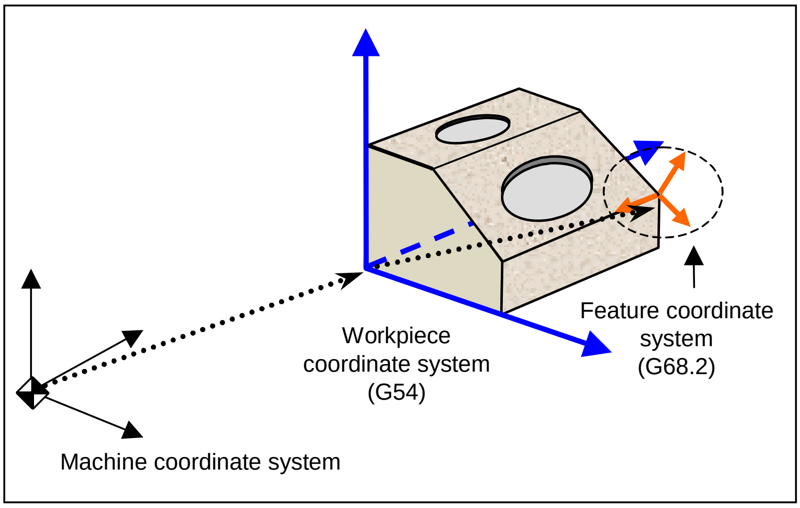


Fig. 5.7.1 (a) Feature coordinate system

The G68.2 command causes the programming coordinate system to switch to the feature coordinate system. The commands in all subsequent blocks are assumed to be specified in the feature coordinate system until G69 appears.
If G68.2 specifies the relationship between the feature coordinate system and the workpiece coordinate system, G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction even if no angle is specified for the rotary axis. (See Fig. 5.7.1 (c).)
For explanations about the tool axis direction, see Fig. 5.7.1 (b).

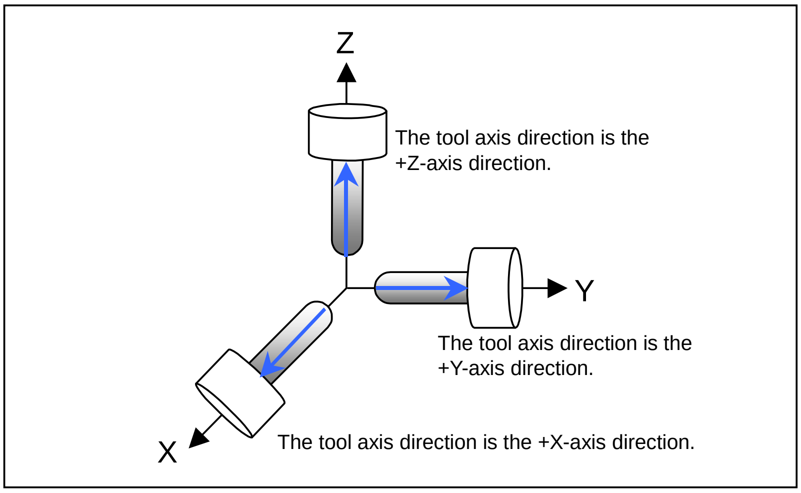
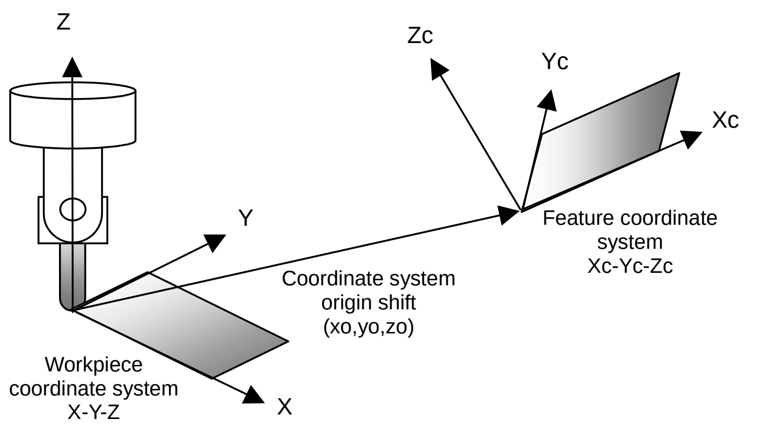


Fig. 5.7.1 (b) Tool axis direction

This function regards the direction normal to the machining plane as the +Z-axis direction of the feature coordinate system. After the G53.1 command, the tool is controlled so that it remains perpendicular to the machining plane.

- **Only G68.2 is specified**



- **G53.1 is specified after G68.2**

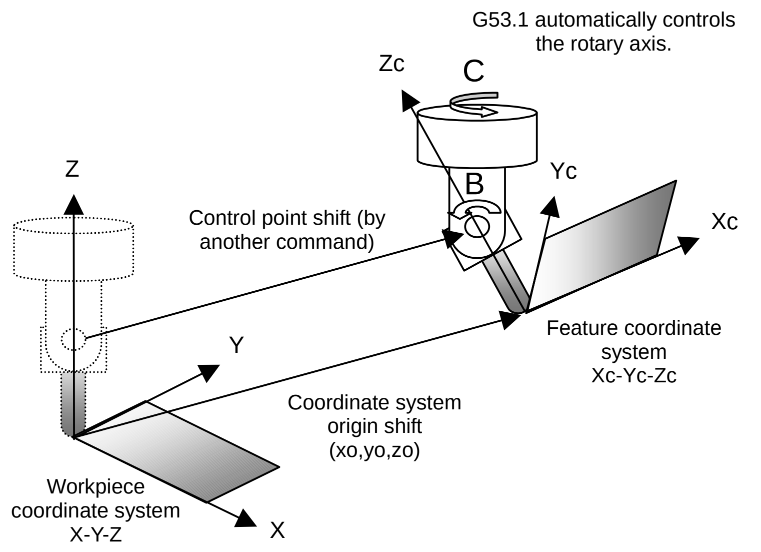


Fig. 5.7.1 (c) G68.2 and G53.1 commands

This function is applicable to the following machine configurations. (Refer to Fig. 5.7.1 (d).)
<1> Tool rotation type machine controlled with two tool rotation axes
<2> Table rotation type machine controlled with two table rotation axes
<3> Composite type machine controlled with one tool rotation axis and one rotary axis

The function can also be used for a machine configuration in which the rotary axis for controlling the tool does not intersect the rotary axis for controlling the table.

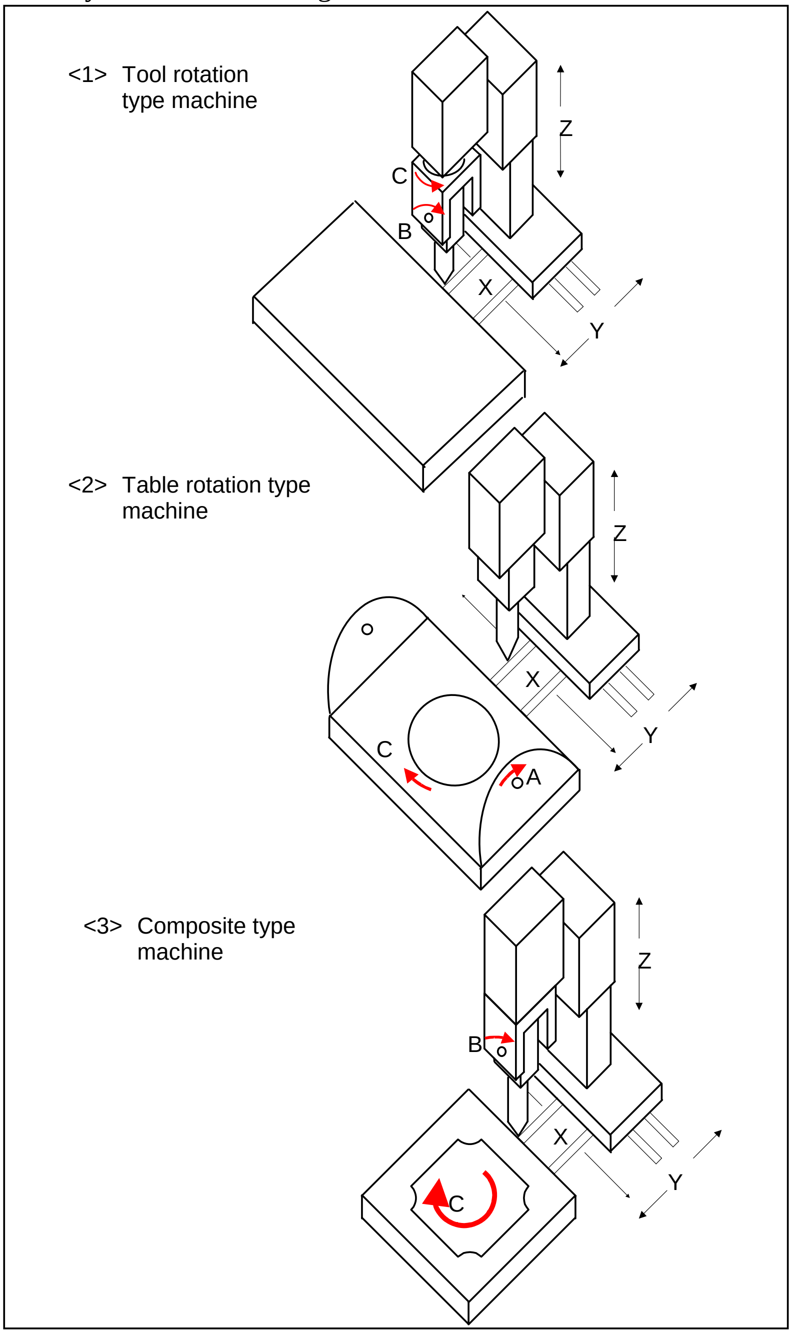


Fig. 5.7.1 (d) Three types of 5-axis machine

5.7.1.1 Tilted working plane indexing based on Eulerian angle

Format

- Tilted working plane indexing (G68.2)

M

G68.2 X $\underline{x_0}$ Y $\underline{y_0}$ Z $\underline{z_0}$ I$\underline{\alpha}$ J$\underline{\beta}$ K$\underline{\gamma}$; Tilted working plane indexing
G69 ; Cancels the tilted working plane indexing.

X,Y,Z: Feature coordinate system origin
The axes specified here are the three axes of the feature coordinate system. Specify the three basic axes or parallel axes set by parameter No. 1022. When specification is omitted, the X, Y, and Z of the basic three axes are assumed to be 0.
I,J,K: Euler's angle for determining the orientation of the feature coordinate system

- Tool axis direction control (G53.1)

G53.1 ; Controls the tool axis direction.

⚠ **CAUTION**
1 G53.1 must be specified in a block after the block that contains G68.2. An alarm occurs if G53.1 is specified without G68.2 being specified in a preceding block.
2 G53.1 must be specified in a block in which there is no other command.
3 The rotary axis moves at the maximum rapid traverse federate in the case of rapid traverse and at the specified federate in the case of cutting feed.

Explanation

- Coordinate conversion using an Euler's angle

Coordinate conversion by rotation is assumed to be performed around the workpiece coordinate system origin.
Let the coordinate system obtained by rotating the workpiece coordinate system around the Z-axis by an angle of α degrees be coordinate system 1. Similarly, let the coordinate system obtained by rotating coordinate system 1 around the X'-axis by an angle of β be coordinate system 2. The feature coordinate system is the coordinate system obtained by shifting the coordinate system that is obtained by rotating coordinate system 2 around the Z"-axis through an angle of γ degrees from the workpiece coordinate system origin by (Xo, Yo, Zo).
Fig. 5.7.1 (e) shows the relationship between the workpiece coordinate system and the feature coordinate system.
The figure also gives examples of displacement on the X-Y plane.

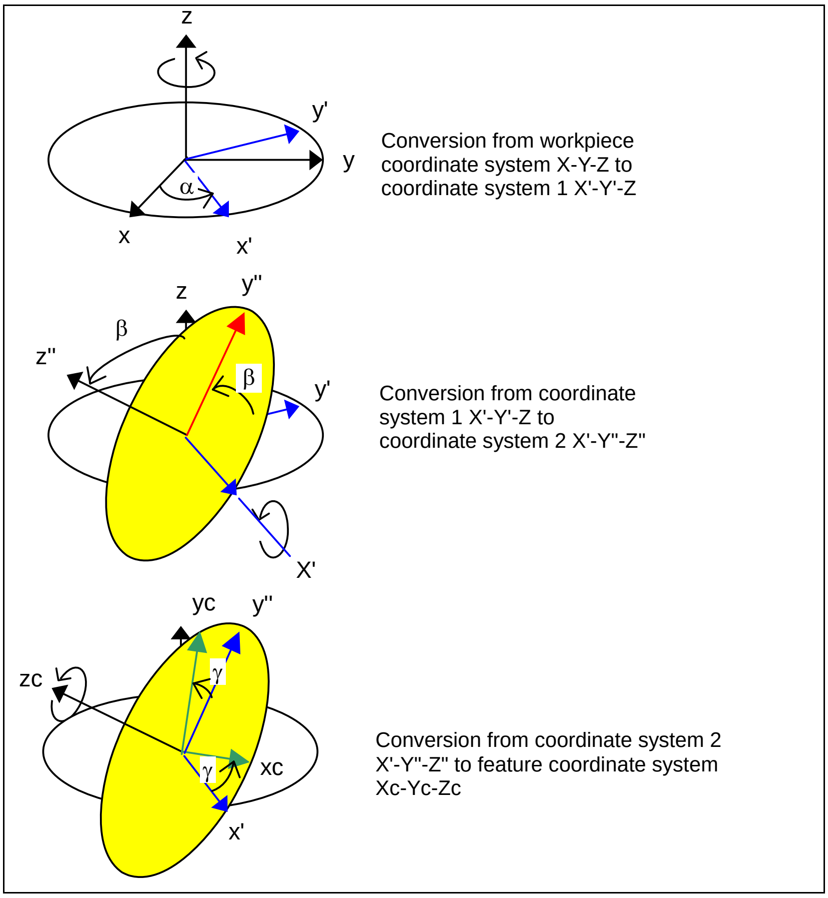


Fig. 5.7.1 (e) Coordinate conversion using an Euler's angle

- I0 J0 K0 command

When I0 J0 K0 is specified as an Euler's angle, the alarm PS5457, “G68.2 FORMAT ERROR” usually occurs. When bit 1 (ATW) of parameter No. 13451 is set to 1, the feature coordinate system with a tilted angle of 0 degree is used.

5.7.1.2 General specifications of the tilted working plane indexing

- Constant surface speed control

Constant surface speed control is exercised by using, as the reference, the machine axis specified in address P in a G96 block or the machine axis (not in the feature coordinate system but in the actually operating workpiece coordinate system) set in parameter No. 3770.

- Workpiece coordinate system selection command during the tilted working plane indexing

By executing the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 1, it is possible to change the workpiece coordinate system. In this case, the coordinate system zero point shift of the tilted working plane indexing is maintained.
If an attempt is made to execute the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 0, alarm PS5462, ”ILLEGAL COMMAND(G68.2/G69)” is issued.

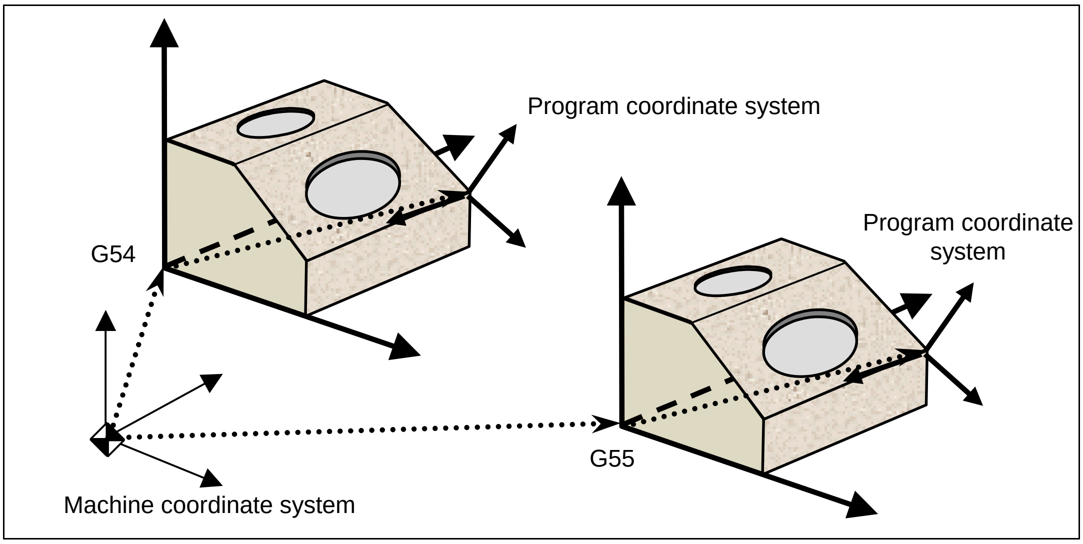


Fig. 5.7.1 (f)

- Minimum command unit of rotation angles

The minimum command unit of the rotation angles (I, J, K, and R) of the tilted working plane indexing is 0.001 degree regardless of the increment system. By setting bit 2 (TFR) of parameter No. 11630 to 1, the minimum command unit of the rotation angles can be set to 0.00001 degree.

- System variables of skip positions during the tilted working plane indexing

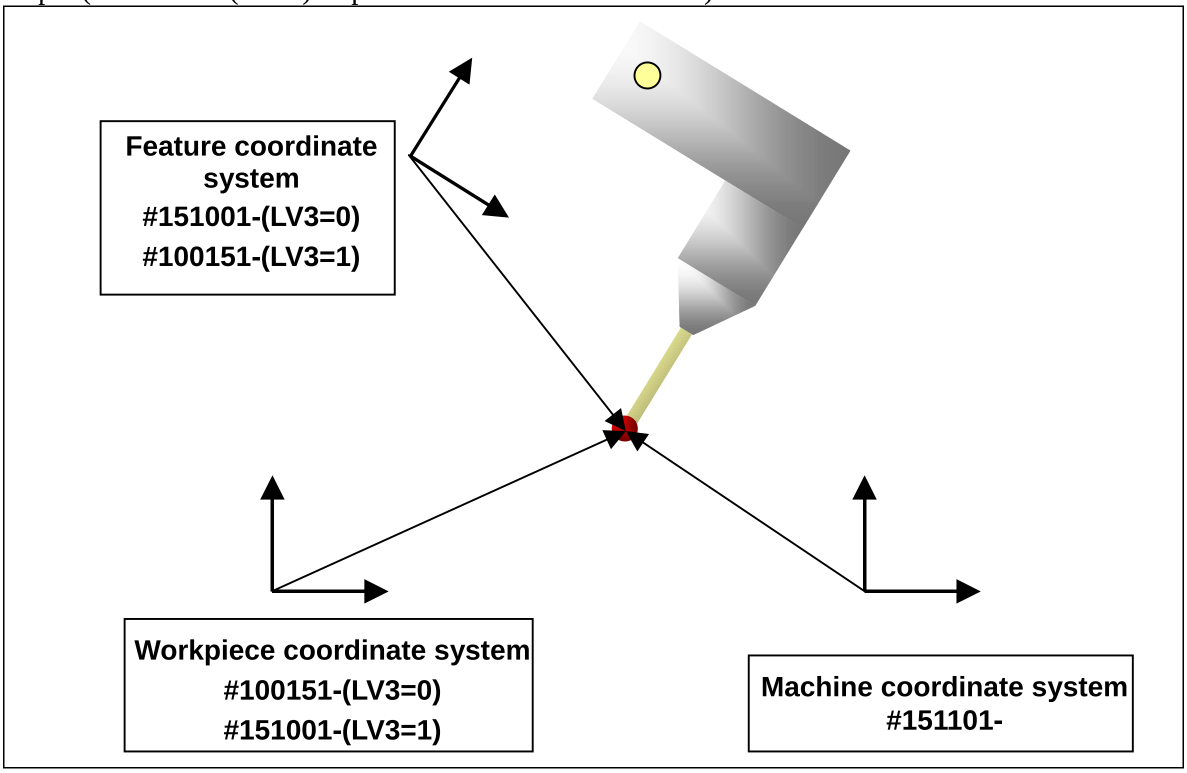
System variable number and coordinate system of skip are as Table5.7.1 (a). The coordinate system of #100105- and #151001- changes according to bit 5 (LV3) of parameter No.5400.

Table5.7.1 (a) Parameter LV3 and coordinate system of skip positions

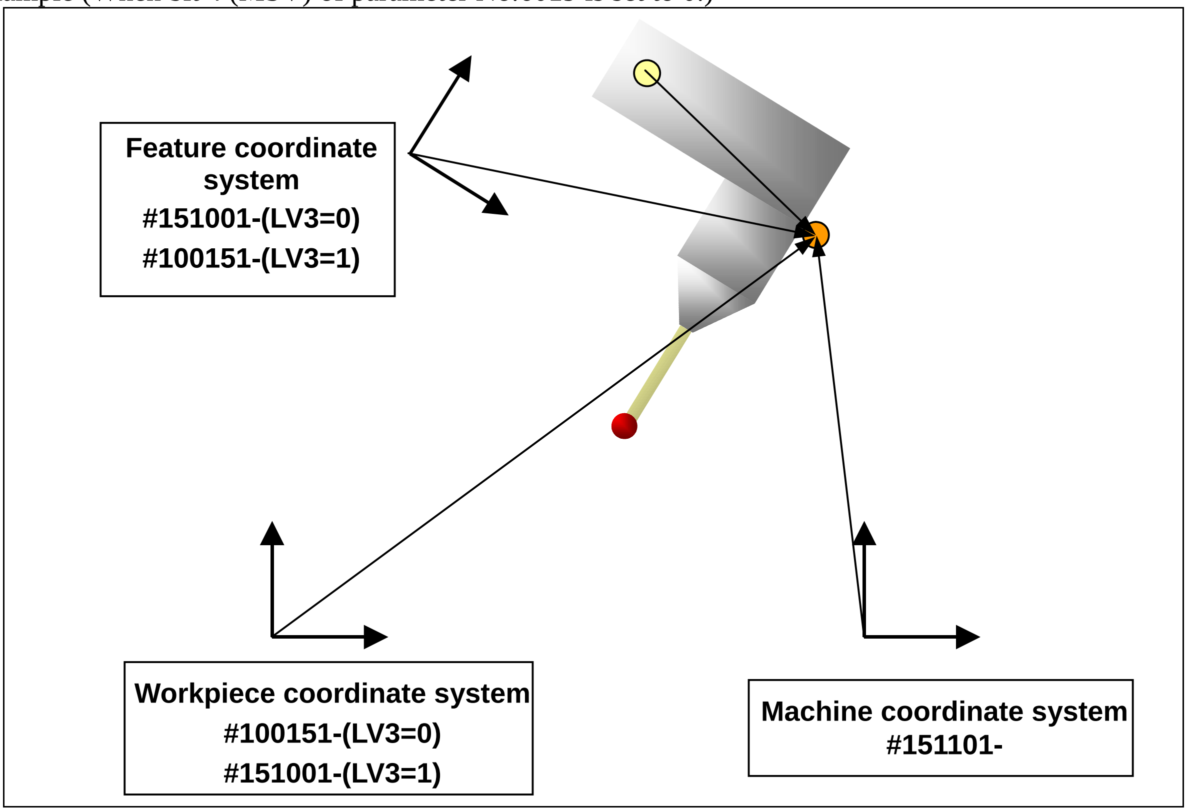
System variable number	Coordinate system of skip positions	
	Parameter LV3=0	Parameter LV3=1
#100151 -	Workpiece coordinate system	Feature coordinate system
#151001 -	Feature coordinate system	Workpiece coordinate system
#151101 -	Machine coordinate system	

Moreover, in case of machining center system, skip positions of tool tip position can be read by setting bit 4 (MSV) of parameter No. 6019.
When bit 4 (MSV) of parameter No.6019 is set to 0, system variable includes tool length compensation offset (control point position).
When bit 4 (MSV) of parameter No.6019 is set to 1, system variable does not include tool length compensation offset (tool tip position).

Example (When bit 4 (MSV) of parameter No.6019 is set to 1.)

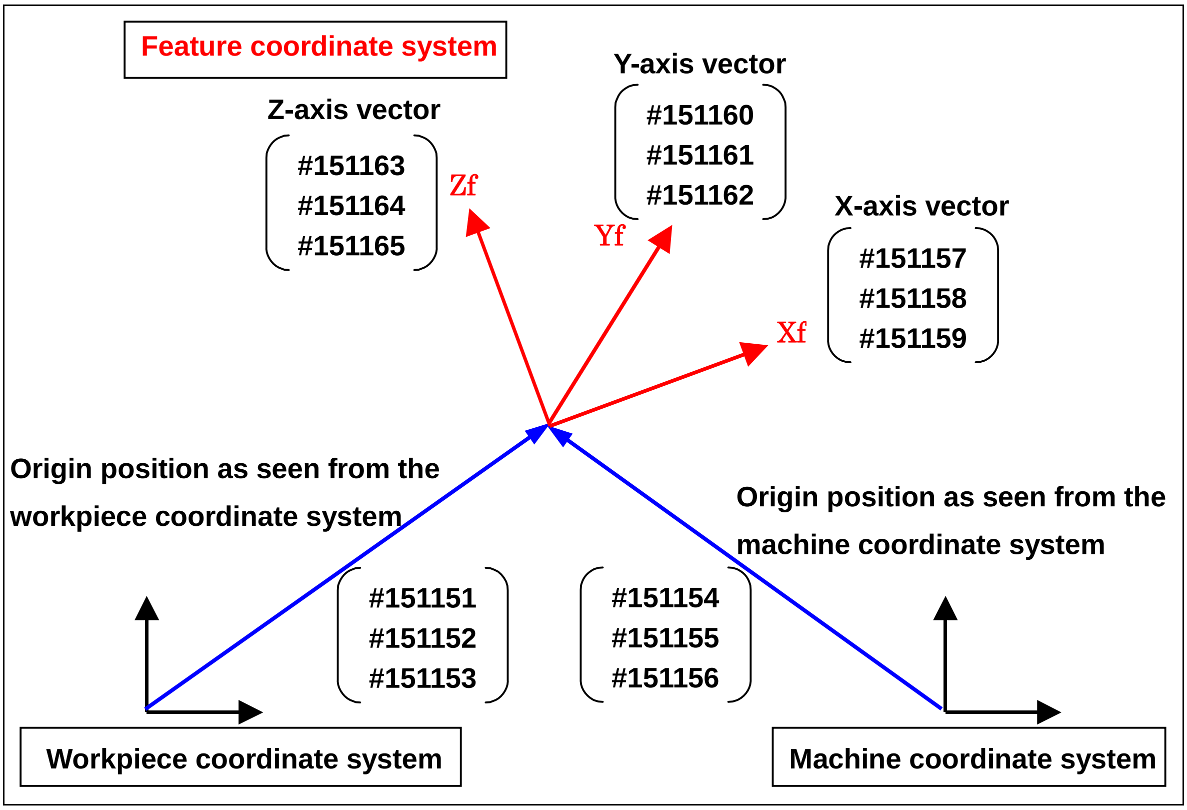


Example (When bit 4 (MSV) of parameter No.6019 is set to 0.)



- System variables of the feature coordinate system information

By using custom macro variables #151151 to #151165, the feature coordinate system information can be read.



System variable number	System variable name	Attribute	Description
#151151 to #151165	[#_FCOORD [n]]	R	The feature coordinate system information during the tilted working plane indexing. Note) Subscript n represents a compensation number (1 to 15).

R is attribute of a variable and indicates read-only.
Details of each variable are as follows.

System variable number	System variable name	Description
#151151	[#_FCOORD [1]]	Origin position X as seen from the machine coordinate system
#151152	[#_FCOORD [2]]	Origin position Y as seen from the machine coordinate system
#151153	[#_FCOORD [3]]	Origin position Z as seen from the machine coordinate system
#151154	[#_FCOORD [4]]	Origin position X as seen from the workpiece coordinate system
#151155	[#_FCOORD [5]]	Origin position Y as seen from the workpiece coordinate system
#151156	[#_FCOORD [6]]	Origin position Z as seen from the workpiece coordinate system
#151157	[#_FCOORD [7]]	X of X-axis vector as seen from the machine/workpiece coordinate system
#151158	[#_FCOORD [8]]	Y of X-axis vector as seen from the machine/workpiece coordinate system
#151159	[#_FCOORD [9]]	Z of X-axis vector as seen from the machine/workpiece coordinate system
#151160	[#_FCOORD [10]]	X of Y-axis vector as seen from the machine/workpiece coordinate system
#151161	[#_FCOORD [11]]	Y of Y-axis vector as seen from the machine/workpiece coordinate system
#151162	[#_FCOORD [12]]	Z of Y-axis vector as seen from the machine/workpiece coordinate system
#151163	[#_FCOORD [13]]	X of Z-axis vector as seen from the machine/workpiece coordinate system
#151164	[#_FCOORD [14]]	Y of Z-axis vector as seen from the machine/workpiece coordinate system
#151165	[#_FCOORD [15]]	Z of Z-axis vector as seen from the machine/workpiece coordinate system

When it is not in the tilted working plane indexing mode, all variables are set to 0.0.
The length of the each axis vector on the feature coordinate system (#151157～#151165) is 1. The vector variable is displayed by 9-digits in the decimal part

Example) Execute the block N20:O1234 of the following NC program.

O1234 ;	
N10 G54 X0.0 Y0.0 Z0.0 ;	Set the workpiece coordinate system (G54: X=100.0 Y=200.0 Z=300.0)
N20 G68.2 X5.0 Y10.0 Z15.0 I30.0 J0.0 K0.0;	Specifying the tilted working plane indexing.

The feature coordinate system made in N20 is the workpiece coordinate system that shifted X5.0 Y10.0 Z15.0 in parallel and rotated around the Z-axis by angle of 30 degree. The value of each system variable at this time is as follows.

System variable number	Value	System variable number	Value	System variable number	Value	System variable number	Value	System variable number	Value
#151151	105.0	#151154	5.0	#151157	0.866025404	#151160	-0.5	#151163	0.0
#151152	210.0	#151155	10.0	#151158	0.5	#151161	0.866025404	#151164	0.0
#151153	315.0	#151156	15.0	#151159	0.0	#151162	0.0	#151165	1.0

- Cutting feedrate clamp

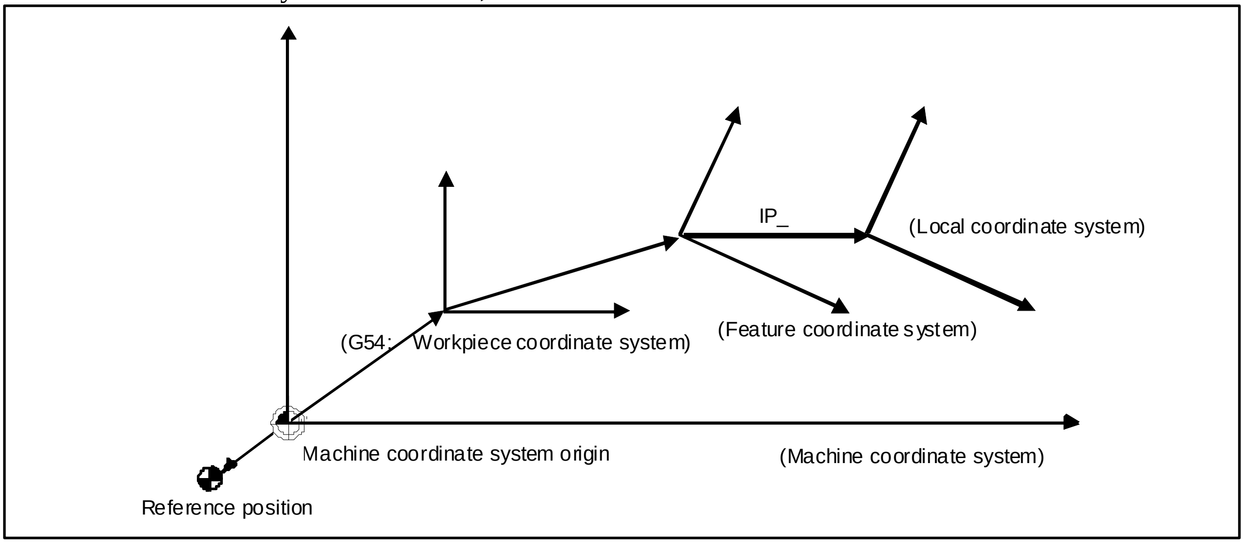
The cutting feedrate is clamped so that the feedrate of each real axis after the conversion by the tilted working plane indexing does not exceed the maximum cutting feedrate（Parameter No.1432 if acceleration/deceleration before interpolation is enabled and parameter No.1430 otherwise).

- Local Coordinate System

The local coordinate system is available to the feature coordinate system during the tilted working plane indexing.

X,Y,Z commands of the local coordinate system defines a local coordinate system that the feature coordinate system is translated in X,Y,Z direction.

Alarm PS5462 occurs when the tilted working plane indexing is specified on the condition that the offset of the local coordinate system is not zero,



- Absolute position display

The absolute coordinates based on the program or workpiece coordinate system can be displayed during the tilted working plane indexing. Specify a desired coordinate system in bit 6 (DAK) of parameter No. 3106.

- Distance to go display

The distance to go based on the program or workpiece coordinate system can be displayed during the tilted working plane indexing. Specify a desired coordinate system in bit 5 (D3D) of parameter No.19602.

5.7.1.3 Tilted working plane indexing based on roll-pitch-yaw

Overview

With the tilted working plane indexing, coordinate system conversion by rotation about the X-axis, Y-axis, and Z-axis of a workpiece coordinate system in this order can be used (roll-pitch-yaw).
The order of rotary axes can be specified using address Q.

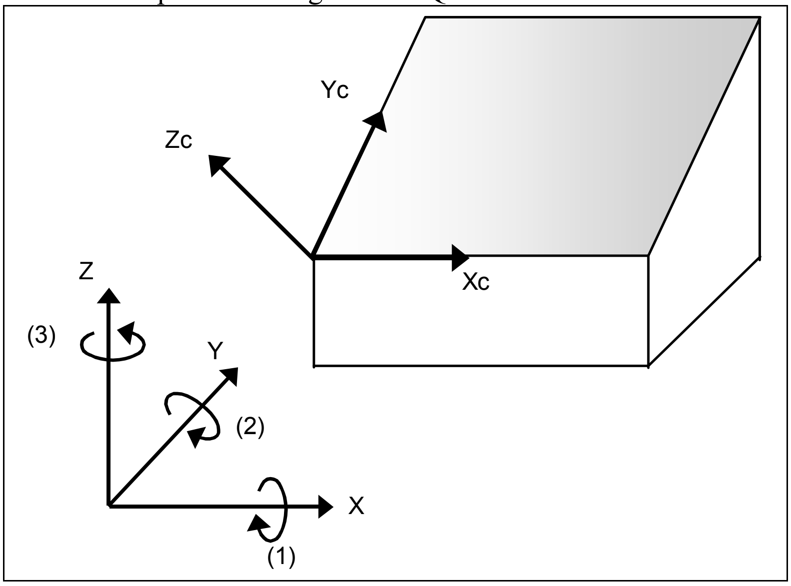


Fig. 5.7.1.3 (a)

Format

Format
G68.2 P1 Qq X_ Y_ Z_ Iα Jβ Kγ; Tilted working plane indexing
G69 ; Cancel tilted working plane indexing (M series).
Explanation of symbols
Q: Order in which axes are rotated X_ Y_ Z_: Origin of a feature coordinate system I: Angle of rotation about the X-axis (roll angle) J: Angle of rotation about the Y-axis (pitch angle) K: Angle of rotation about the Z-axis (yaw angle)

The values of address Q and the order in which axes are rotated are shown below.

Table 5.7.1.3 (a)

	First rotation axis	Second rotation axis	Third rotation axis
Q 123	X axis	Y axis	Z axis
Q 132	X axis	Z axis	Y axis
Q 213	Y axis	X axis	Z axis
Q 231	Y axis	Z axis	X axis
Q 312	Z axis	X axis	Y axis
Q 321	Z axis	Y axis	X axis

⚠ **CAUTION**

1 When address Q is omitted, the X-axis, Y-axis, and Z-axis are rotated in this order. (equivalent to Q123)

> ⚠ **CAUTION**
> 2 When address Q is set to a value other than the above, alarm PS5457, “G68.2/G68.3 FORMAT ERROR” is issued.

Explanation

Suppose that the coordinate system is rotated about (1) the X-axis, (2) the Y-axis, and (3) the Z-axis in this order.
A "workpiece coordinate system" rotated by angle α about the X-axis is "coordinate system 1".
"Coordinate system 1" rotated by angle β about the Y-axis is "coordinate system 2". "Coordinate system 2" rotated by angle γ about the Z-axis then shifted by (X_0,Y_0,Z_0) from the workpiece coordinate system origin is a "feature coordinate system".

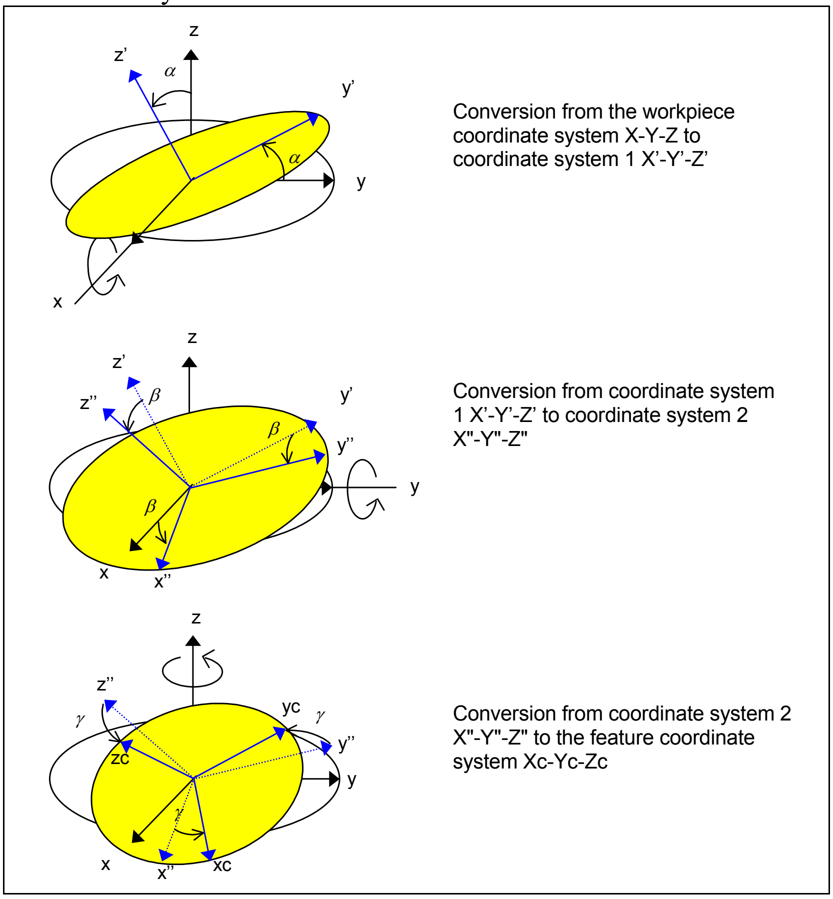


Fig. 5.7.1.3 (b)

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

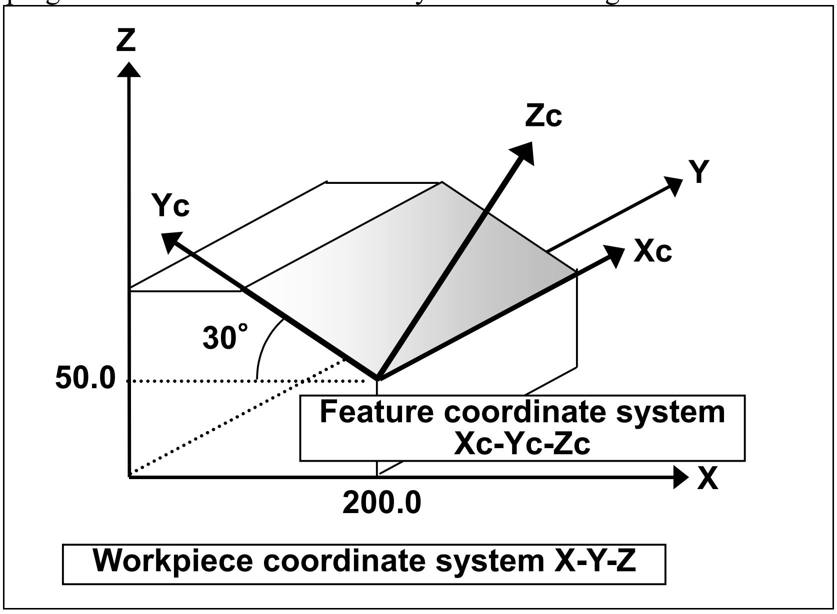


Fig. 5.7.1.3 (c)

-	Feature coordinate system origin	:	(200.0, 0, 50.0)
-	Order in which axes are rotated	:	X axis → Y axis → Z axis
-	Angle of rotation about the X-axis	:	30 degrees
-	Angle of rotation about the Y-axis	:	0 degree
-	Angle of rotation about the Z-axis	:	90 degrees

Example of a program

```
G68.2 P1 Q123 X200.0 Y0 Z50.0 I30.0 J0 K90.0 ;
G53.1 ;
   :
```

5.7.1.4 Tilted working plane indexing based on three points

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying three points in a feature coordinate system.

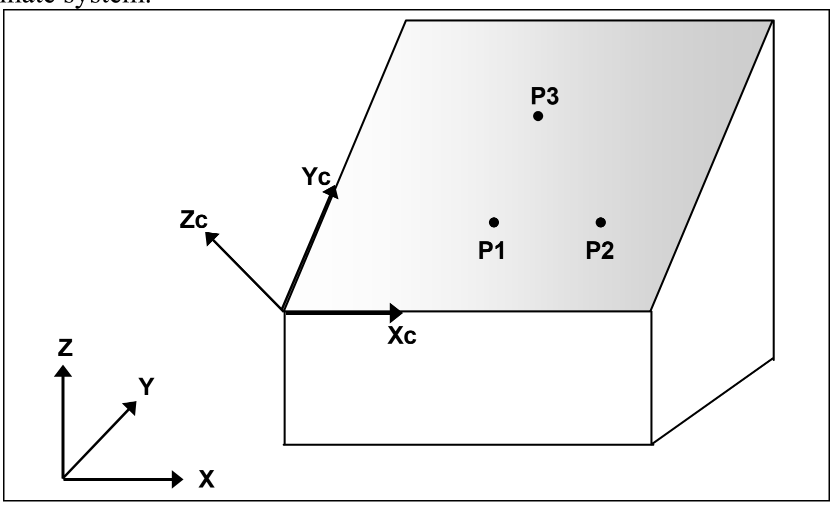


Fig. 5.7.1.4 (a)

Format

Format
G68.2 P2 Q0 X $\underline{x_0}$ Y $\underline{y_0}$ Z $\underline{z_0}$ Rα ; **G68.2 P2 Q1 X $\underline{x_1}$ Y $\underline{y_1}$ Z $\underline{z_1}$;** **G68.2 P2 Q2 X $\underline{x_2}$ Y $\underline{y_2}$ Z $\underline{z_2}$;** **G68.2 P2 Q3 X $\underline{x_3}$ Y $\underline{y_3}$ Z $\underline{z_3}$; Tilted working plane indexing** **G69 ; Cancel tilted working plane indexing (M series).**
Explanation of symbols
Q0 X x_0 Y y_0 Z z_0 : Amount of shift from the first point to the origin of a feature coordinate system By default, this value is 0. Q1 X x_1 Y y_1 Z z_1 : First point. (origin of a feature coordinate system) Q2 X x_2 Y y_2 Z z_2 : Second point. Q3 X x_3 Y y_3 Z z_3 : Third point. R : Angle of rotation about the Z-axis of a feature coordinate system By default, this value is 0. Any block in the G68.2 P2 command can be specified.

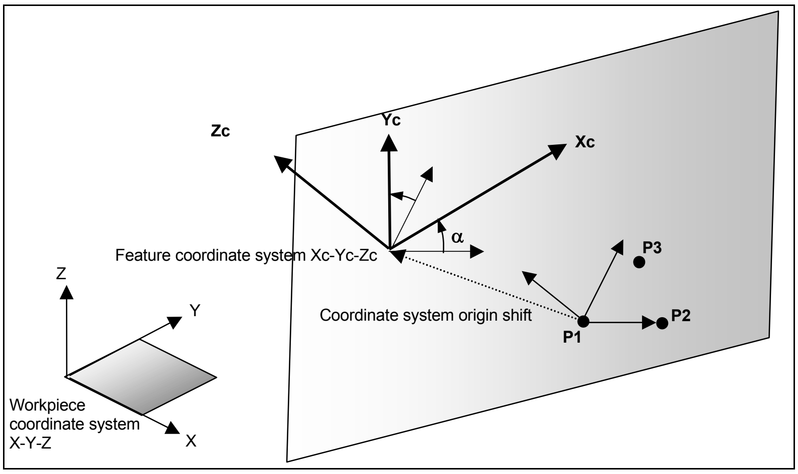


Fig. 5.7.1.4 (b)

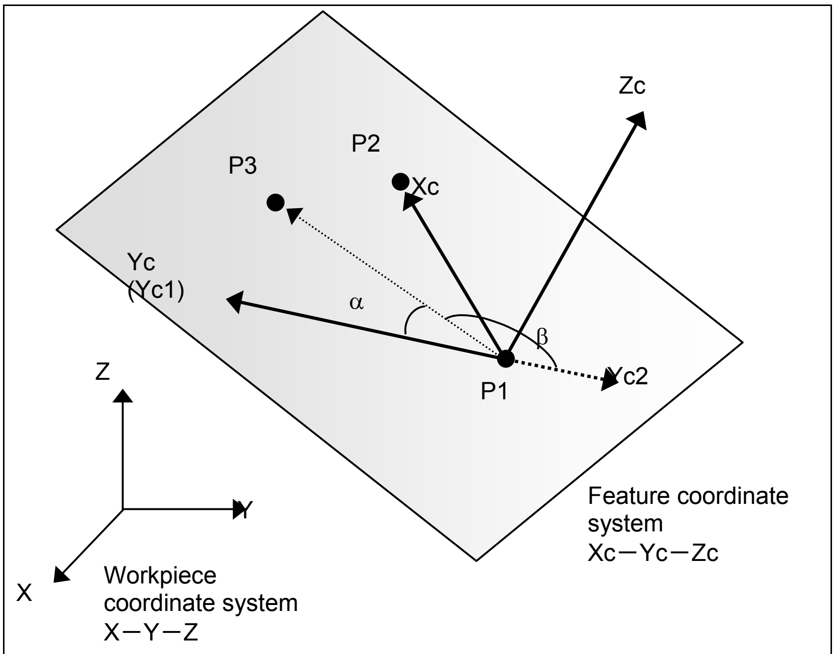
⚠ CAUTION

1 Three G68.2P2 commands (Q1, Q2, and Q3) determine a tilted plane. If the G68.2P2 commands are interrupted, alarm PS5457, "G68.2/G68.3 FORMAT ERROR" is issued.
2 When any of the following conditions is met or address Q is set to a value other than the above, alarm PS5457 is issued.
(1) When two or more points are duplicate (the plane is not determined.)
(2) When three points are placed on a line (the plane is not determined.)
(3) When the distance between a line passing two of the three points and the remaining point is smaller than the setting of parameter No. 11220 (the plane is unstable.)

Explanation

- Determination of a feature coordinate system

Three entered points are named P1, P2, and P3 in the order of entry.
The P1-to-P2 direction is defined as the X-axis of a feature coordinate system. Among the directions that are on the plane containing the three points and are normal to the X-axis of the feature coordinate system, the direction that makes a smaller angle with the P1→P3 vector is defined as the Y-axis of the feature coordinate system. The Z-axis of the feature coordinate system is defined according to the right-handed system.



On the plane containing three points, there are two directions normal to Xc: Yc1 and Yc2. Angles α and β formed relative to the P1→P3 vector satisfy α < β, so that Yc1 is selected as Yc.

Fig. 5.7.1.4 (c)

- Origin of the feature coordinate system

The origin of the feature coordinate system is the first specified point P1.
By setting an origin shift amount (G68.2 P2 Q0 X_Y_Z_), the origin of the feature coordinate system is shifted by (X,Y,Z) from P1. Specify (X,Y,Z) in the feature coordinate system.

- Angular displacement R

The angular displacement R is positive for clockwise rotation viewed in the Z-axis direction in the feature coordinate system.

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

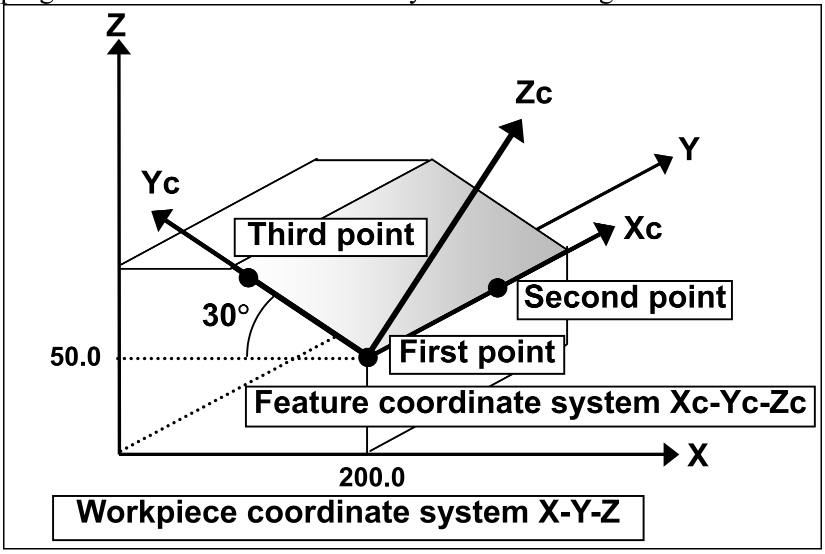


Fig. 5.7.1.4 (d)

- First point (origin of a feature coordinate system) : (200.0, 0, 50.0)
- Second point : (200.0, 100.0, 50.0)
- Third point : (26.795, 0, 150.0)

Example of a program

```
G68.2 P2 Q1 X200.0 Y0 Z50.0 ;
G68.2 P2 Q2 X200.0 Y100.0 Z50.0 ;
G68.2 P2 Q3 X26.795 Y0 Z150.0 ;
G53.1 ;
 . . .
```

5.7.1.5 Tilted working plane indexing based on two vectors

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying an X-axis direction vector and a Z-axis direction vector in the feature coordinate system.

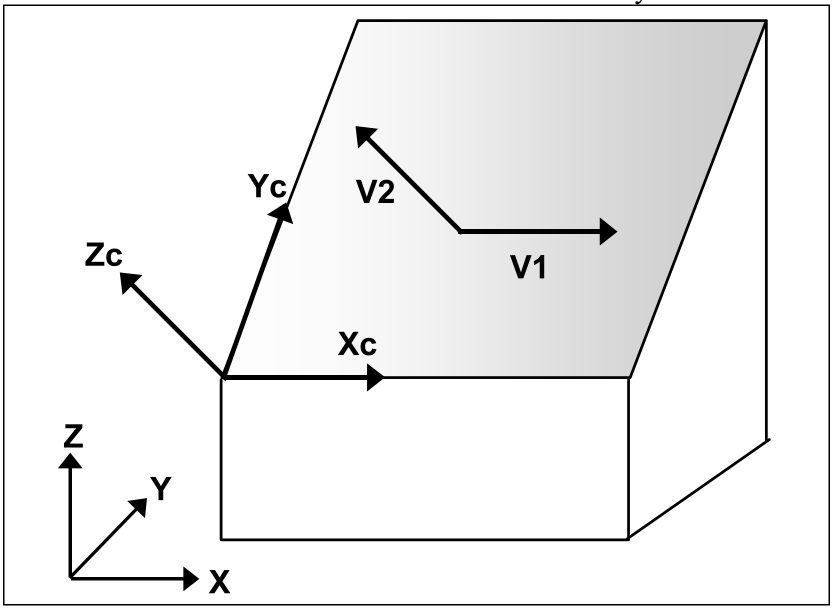


Fig. 5.7.1.5 (a)

Format

Format
G68.2 P3 Q1 X_ Y_ Z_ Iα_1 Jβ_1 Kγ_1 ; **G68.2 P3 Q2 Iα_2 Jβ_2 Kγ_2 ; Tilted working plane indexing** **G69 ; Cancel tilted working plane indexing (M series).**

Explanation of symbols	
X_ Y_ Z_:	Origin of a feature coordinate system (specifiable in the Q1 block)
Q1 Iα_1 Jβ_1 Kγ_1	: Feature coordinate system X-axis direction relative to the workpiece coordinate system (first vector)
Q2 Iα_2 Jβ_2 Kγ_2	: Feature coordinate system Z-axis direction relative to the workpiece coordinate system (second vector)

⚠ **CAUTION**

1 The two G68.2P3 commands (Q1 and Q2) determine a tilted plane. The G68.2P3 commands are interrupted, alarm PS5457, “G68.2/G68.3 FORMAT ERROR” is issued.
2 If the angle between the two vectors is 5 degrees or more off the 90 degrees, alarm PS5457 is issued.
3 If (I, J, K) is set to a 0 vector, alarm PS5457 is issued.

Explanation

- Determination of a feature coordinate system

The first vector is defined as the X-axis of the feature coordinate system, and the second vector is defined as the Z-axis of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

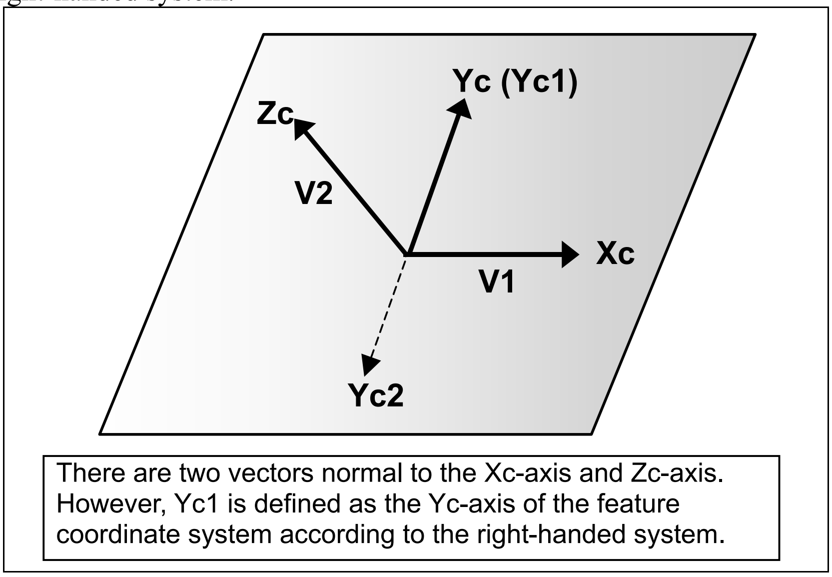


Fig. 5.7.1.5 (b)

- When the first and second vectors are not normal

When the first vector and second vector are not normal, the orthographical vector from the second vector to plane P normal to the first vector is defined as the Z-axis of the feature coordinate system.

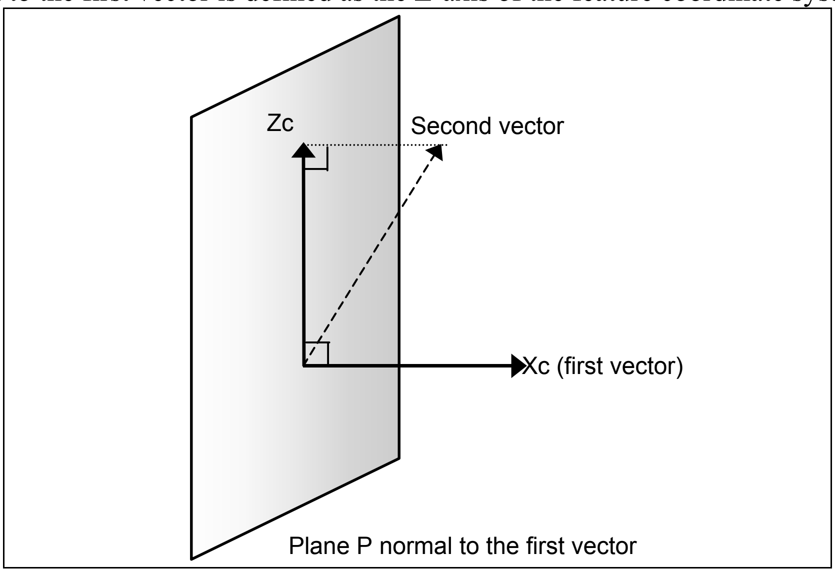


Fig. 5.7.1.5 (c)

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

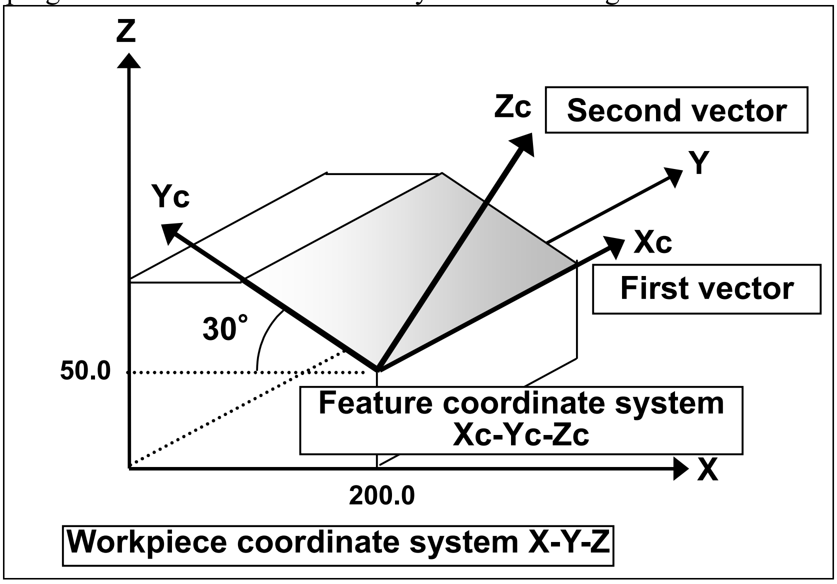


Fig. 5.7.1.5 (d)

Origin of a feature coordinate system	: (200.0, 0, 50.0)
X-axis direction of a feature coordinate system (First vector)	: (0, 1.0, 0)
Z-axis direction of a feature coordinate system (Second vector)	: (100.0, 0, 173.205)

Example of a program

```
G68.2 P3 Q1 X200.0 Y0 Z50.0 I0 J1.0 K0 ;
G68.2 P3 Q2 I100.0 J0 K173.205 ;
G53.1 ;
 . . .
```

5.7.1.6 Tilted working plane indexing based on projection angles

Overview

With the tilted working plane indexing, a tilted working plane can be specified based on projection angles.

A plane determined by vector A and vector B produced by rotating the X-axis vector and Y-axis vector of the workpiece coordinate system is defined to be a tilted working plane.

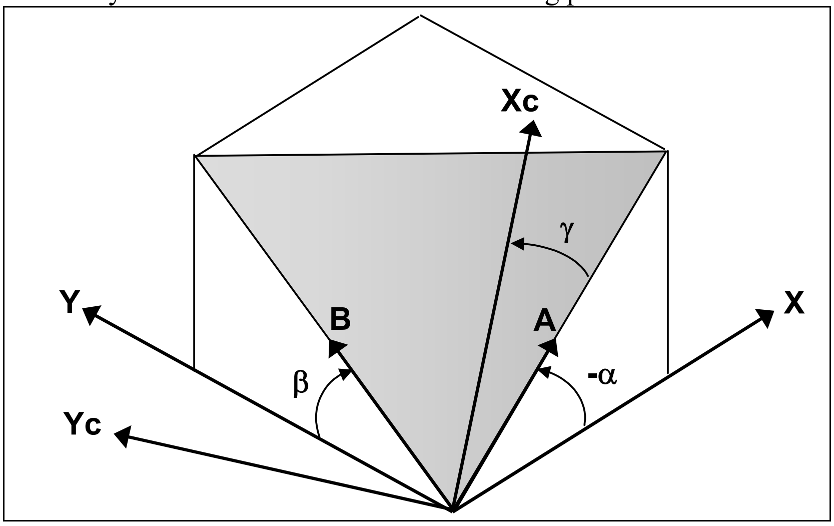


Fig. 5.7.1.6 (a)

Format

Format	
G68.2 P4 X_ Y_ Z_ Iα Jβ Kγ; Tilted working plane indexing	
G69 ; Cancel tilted working plane indexing (M series).	
Explanation of symbols	
X_ Y_ Z_	: Origin of a feature coordinate system
α	: Angle by which the X-axis vector is rotated about the Y-axis of the workpiece coordinate system
β	: Angle by which the Y-axis vector is rotated about the X-axis of the workpiece coordinate system
γ	: Angle of rotation about the Z-axis of the feature coordinate system

Explanation

- Determination of a feature coordinate system

The X-axis direction vector of the workpiece coordinate system rotated by α about the Y-axis of the workpiece coordinate system is defined as vector A. The Y-axis direction vector of the workpiece coordinate system rotated by β about the X-axis of the workpiece coordinate system is defined to be vector B.

The direction normal to plane P containing vector A and vector B (direction of the outer product of A × B) is defined to be the Z-axis direction of the feature coordinate system.

Vector A rotated by γ about the Z-axis of the feature coordinate system is defined to be the X-axis direction of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

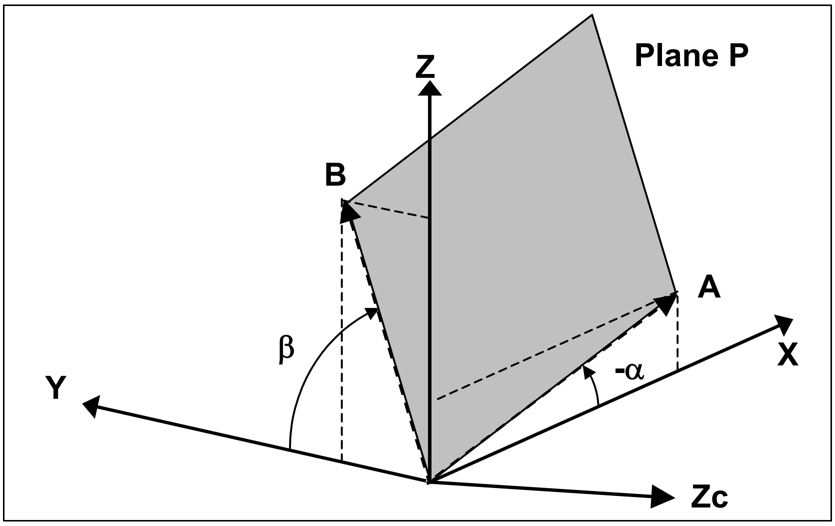


Fig. 5.7.1.6 (b)

By the third command angle α and second command angle β, the Z-axis of the feature coordinate system are determined.

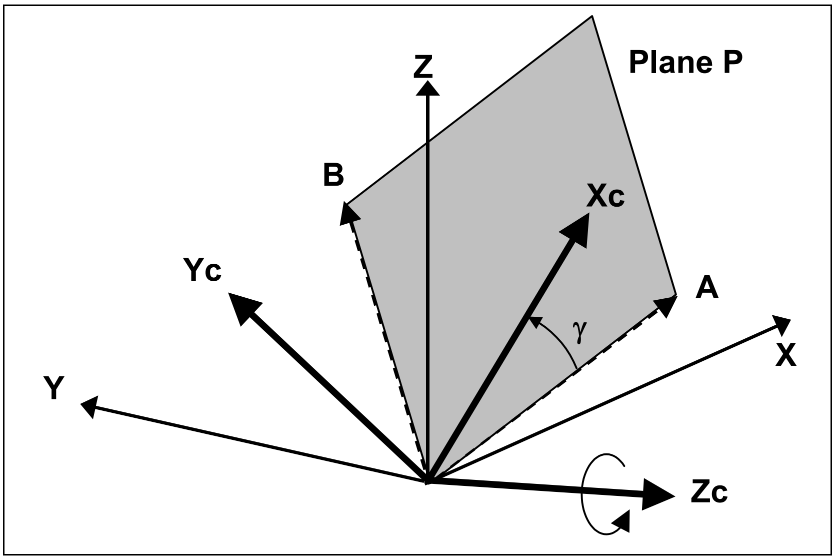


Fig. 5.7.1.6 (c)

By the third command angle γ, the X-axis and Y-axis of the feature coordinate system are determined.

> ⚠ **CAUTION**
> When vector A and vector B are considered to be parallel with each other (when the angle formed by the two vectors is smaller than 1°), alarm PS5457, “G68.2/G68.3 FORMAT ERROR” is issued.

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

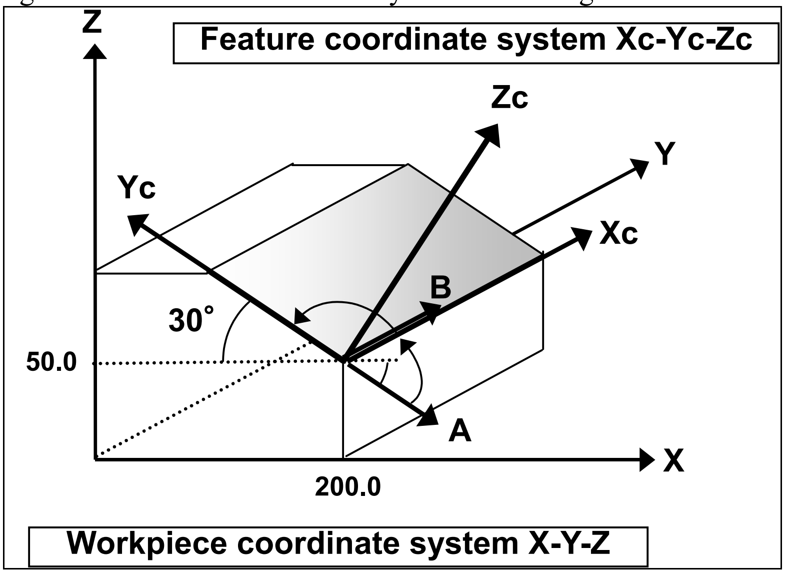


Fig. 5.7.1.6 (d)

- Origin of a feature coordinate system : (200.0, 0, 50.0)
- Angle by which the X-axis vector is rotated about the Y-axis of the workpiece coordinate system : 30 degrees
- Angle by which the Y-axis vector is rotated about the X-axis of the workpiece coordinate system : 0 degree
- Angle of rotation about the Z-axis of the feature coordinate system : 90 degrees

Example of a program

```
G68.2 P4 X200.0 Y0 Z50.0 I30.0 J0 K90.0 ;
G53.1 ;
    :
```

5.7.1.7 Tilted working plane indexing by tool axis direction

Overview

By specifying G68.3, a coordinate system (feature coordinate system) where the tool axis direction is the +Z-axis direction can be automatically specified. When a feature coordinate system is used, a program for cutting a hole or pocket in a plane tilted relative to the workpiece coordinate system can be made simpler. This function can automatically generate a feature coordinate system that is normal to the tool direction.

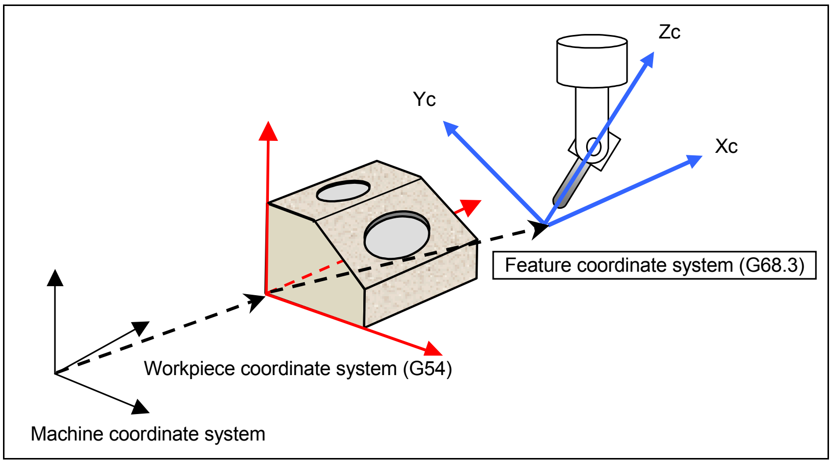


Fig. 5.7.1.7 (a) Feature coordinate system

When G68.3 is specified in a block, the coordinate system for programming is changed to a feature coordinate system. All commands after the block are regarded as commands in the feature coordinate system until G69 is specified.

Format

Format	
G68.3 X $\underline{x_0}$ Y $\underline{y_0}$ Z $\underline{z_0}$ Rα ;	**Tilted working plane indexing**
G68.3 P1 X $\underline{x_0}$ Y $\underline{y_0}$ Z $\underline{z_0}$;	**Tilted working plane indexing**
G69 ;	**Cancel tilted working plane indexing (M series).**
Explanation of symbols	
X,Y,Z	: Origin of a feature coordinate system (absolute) By default, the current position is used as the origin of the feature coordinate system.
R	: Angular displacement about the Z-axis in the feature coordinate system. The default is 0°.
P1	: The feature coordinate system is defined corresponding to tool rotation axis position.

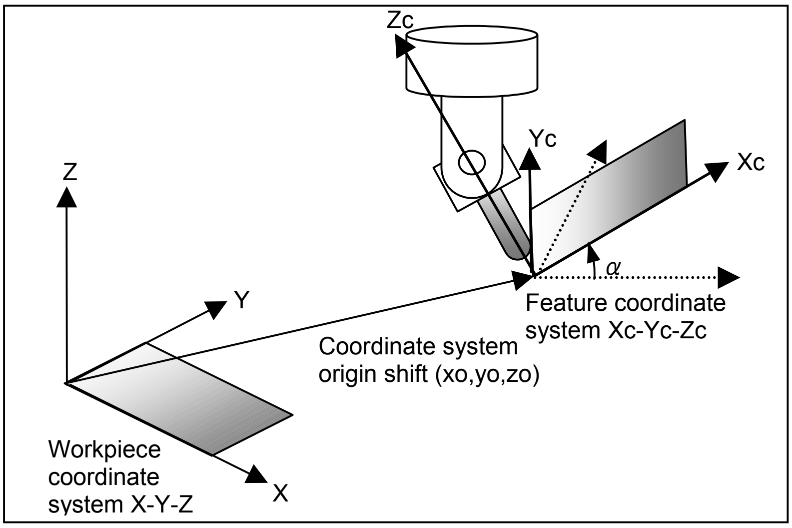


Fig. 5.7.1.7 (b) G68.3 command

Explanation

- Feature coordinate system

By specifying G68.3, a feature coordinate system with the tool axis direction being the +Z-axis direction can be created. The tool axis direction means the tool axis direction based on the rotation axis position reached by automatic operation or manual operation.

- Origin of a feature coordinate system

The origin of a feature coordinate system must be specified using an absolute command.
Even in the incremental command mode, the specified origin is regarded as an absolute position.
When 1 address or 2 addresses are omitted in X, Y, Z, alarm PS5457, "G68.2/G68.3 FORMAT ERROR" is issued.
When the origin of a feature coordinate system is not specified, the position when G68.3 is specified is used as the origin.

- Determination of a feature coordinate system

Determination of a feature coordinate system depends on whether P1 command is present.

(1) When G68.3 block does not include P1 command (G68.3)

When G68.3 is specified, the tool axis direction vector ($\vec{T}$) represents the +Z direction ($\vec{Zc}$) of the feature coordinate system.

The vector normal to a plane formed by the +Z direction ($\vec{Zc}$) of the feature coordinate system and the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) represents the +X direction ($\vec{Xc}$) of the feature coordinate system.

Expression: $\vec{Xc} = \vec{P} \times \vec{Zc}$

The vector normal to the +Z direction ($\vec{Zc}$) of the feature coordinate system and the +X direction ($\vec{Xc}$) of the feature coordinate system represents the +Y direction ($\vec{Yc}$) of the feature coordinate system.

Expression: $\vec{Yc} = \vec{Zc} \times \vec{Xc}$

When R is commanded, a coordinate system rotated by R around Zc from the above-mentioned coordinate system is the feature coordinate system.

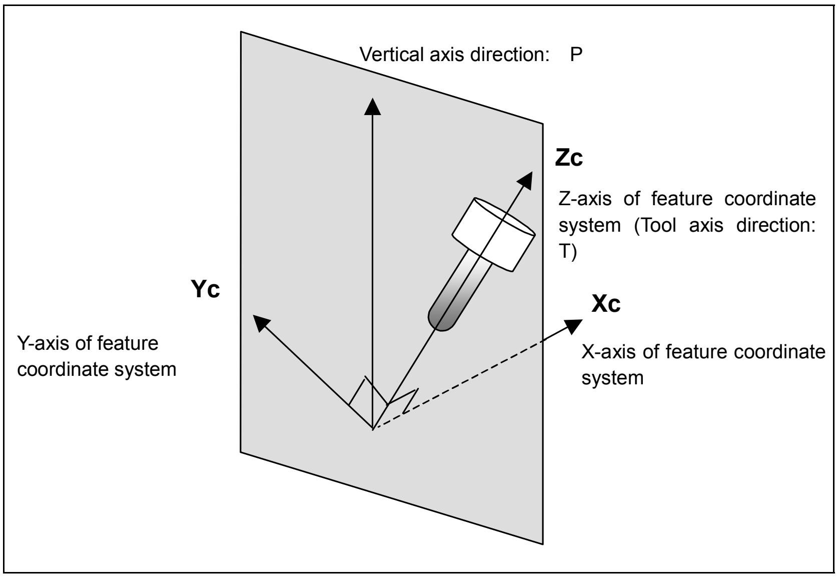


Fig. 5.7.1.7 (c) Determination of a feature coordinate system

When the tool axis direction vector ($\vec{T}$) is parallel with the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between the vector ($\vec{T}$) and the vector ($\vec{P}$) is equal to or less than the value of parameter No. 12322), the feature coordinate system Xc-Yc-Zc is as indicated below. By specifying angular displacement R, a feature coordinate system rotated about the Z-axis of this coordinate system can be specified.

Table 5.7.1.7 (a)

Parameter No. 12321	Z-axis of feature coordinate system Zc	X-axis of feature coordinate system Xc	Y-axis of feature coordinate system Yc
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

When 0 is set in parameter No. 12321, the vertical axis direction is the reference tool axis direction (parameter No. 19697).
If a value other than 0 through 3 is set in parameter No. 12321, alarm PS5459, "MACHINE PARAMETER INCORRECT" is issued.

> ⚠ **CAUTION**
> Tool axis direction is Z-axis direction of feature coordinate system regardless of the reference tool axis direction (parameter No. 19697).

- Angular displacement R

Angular displacement R is positive when a rotation is made clockwise viewed in the Z-axis direction of the feature coordinate system. The range of angular displacement R is: 0.0° ≤ R ≤ 360.0°.

(2) When G68.3 block includes P1 command (G68.3 P1)

"G68.3 P1" command defines the feature coordinate system corresponding to tool rotation axis position. The direction of the feature coordinate system is the direction of reference coordinate system rotated by tool rotation axes. The direction of this feature coordinate system is the same as the direction of tool axis direction feed / tool axis right-angle direction feed of 3-dimensional manual feed when the parameter FLL (No.12320#1) is set to zero.

The reference coordinate system of feature coordinate system (the feature coordinate system that is defined when absolute coordinate system of tool rotation axes is zero) is as follows by the parameter (No.19697) for reference tool axis direction.
The feature coordinate system defined by "G68.3 P1" command is the coordinate system that the reference coordinate system is rotated by tool rotation axis position and parameter RA (No.19698), RB (No.19699).

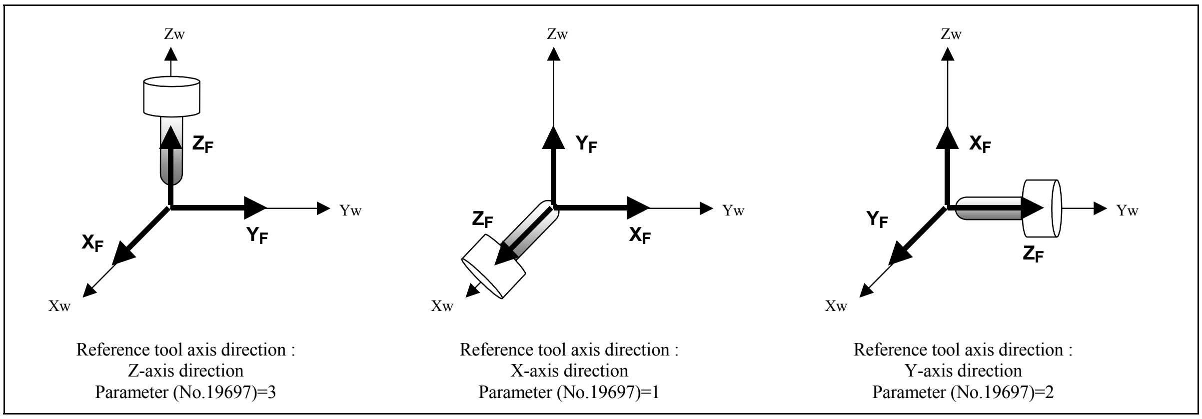
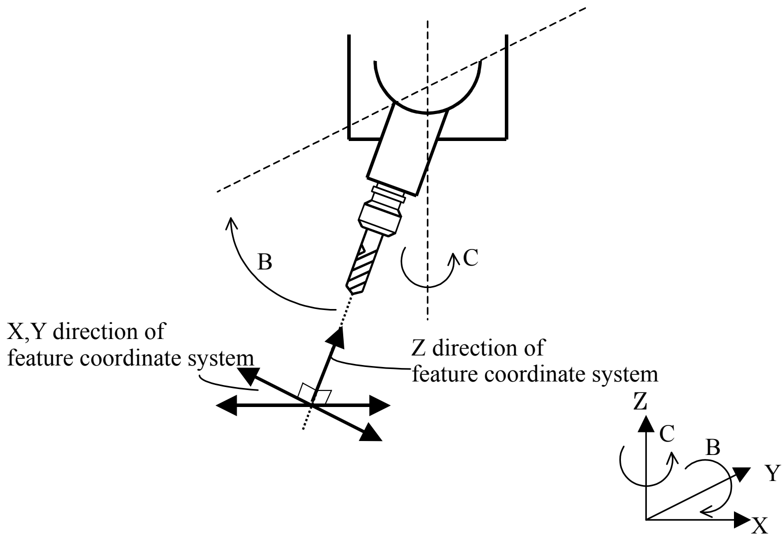


Fig. 5.7.1.7 (d) The reference coordinate system of feature coordinate system (G68.3 P1)



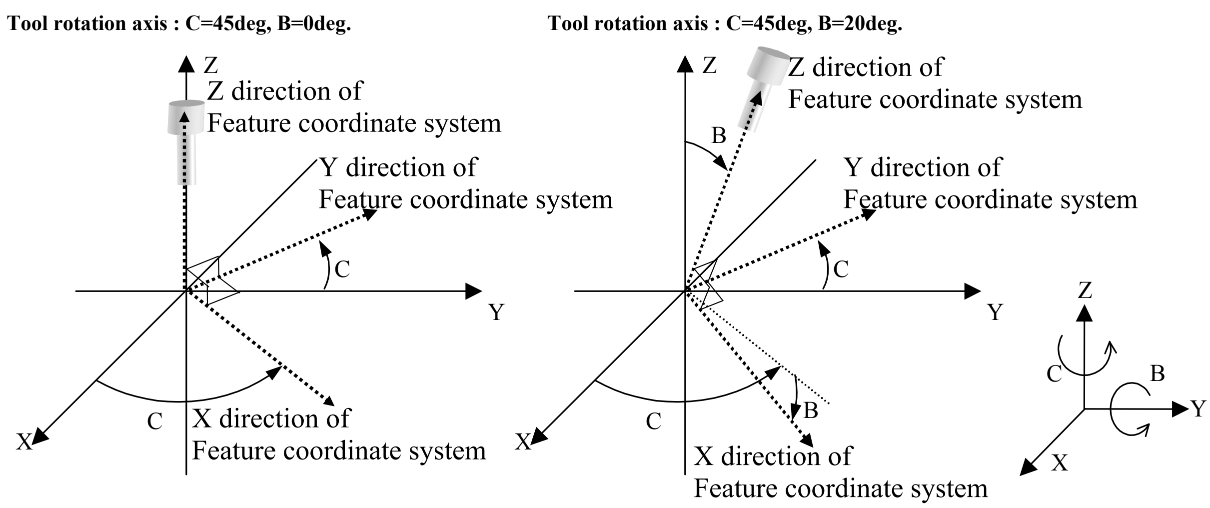


Fig. 5.7.1.7 (e) Example that reference tool axis direction is Z direction

- Machine of table rotation type

On a machine of table rotation type, the tool direction remains unchanged. So, a feature coordinate system based on the reference tool axis direction (parameter No. 19697) is set. However, the origin specification of the feature coordinate system and angular displacement R about the Z-axis are valid.

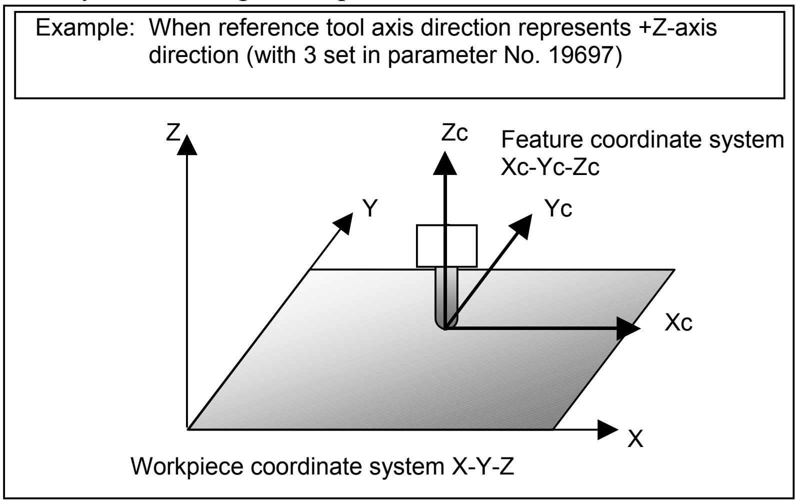


Fig. 5.7.1.7 (f)

- G53.1 / G53.6 command in G68.3 mode

When G53.1 or G53.6 command is executed in G68.3 mode, alarm PS5458, ”ILLEGAL USE OF G53.1/G53.6” occurs.

- Use in combination with tool length compensation

The G68.3 command can be specified even during tool length compensation.

- Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "BC type reference tool axis Z-axis".

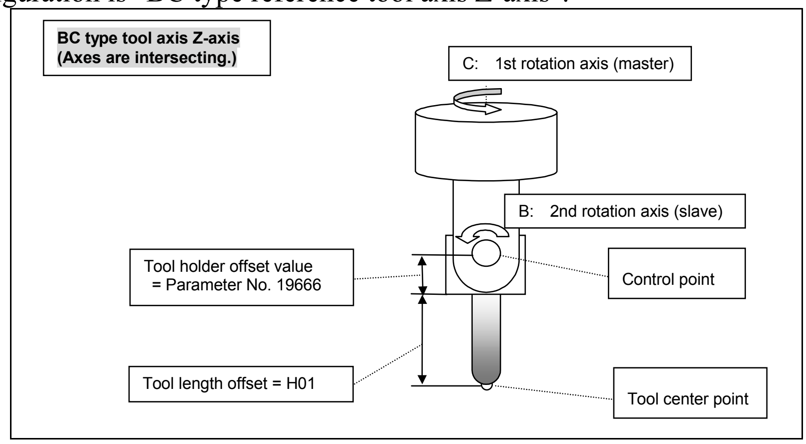


Fig. 5.7.1.7 (g)

Sample program 1

```
O0100 ;
N1 G55 ;
N2 G90 G01 X0Y0Z50.0 F1000 ;
N3 G43 H01 X0 Y0 Z0 ;
N4 B-45.0 ;
N5 G68.3 ;
:
```

N6 G69 ;
:

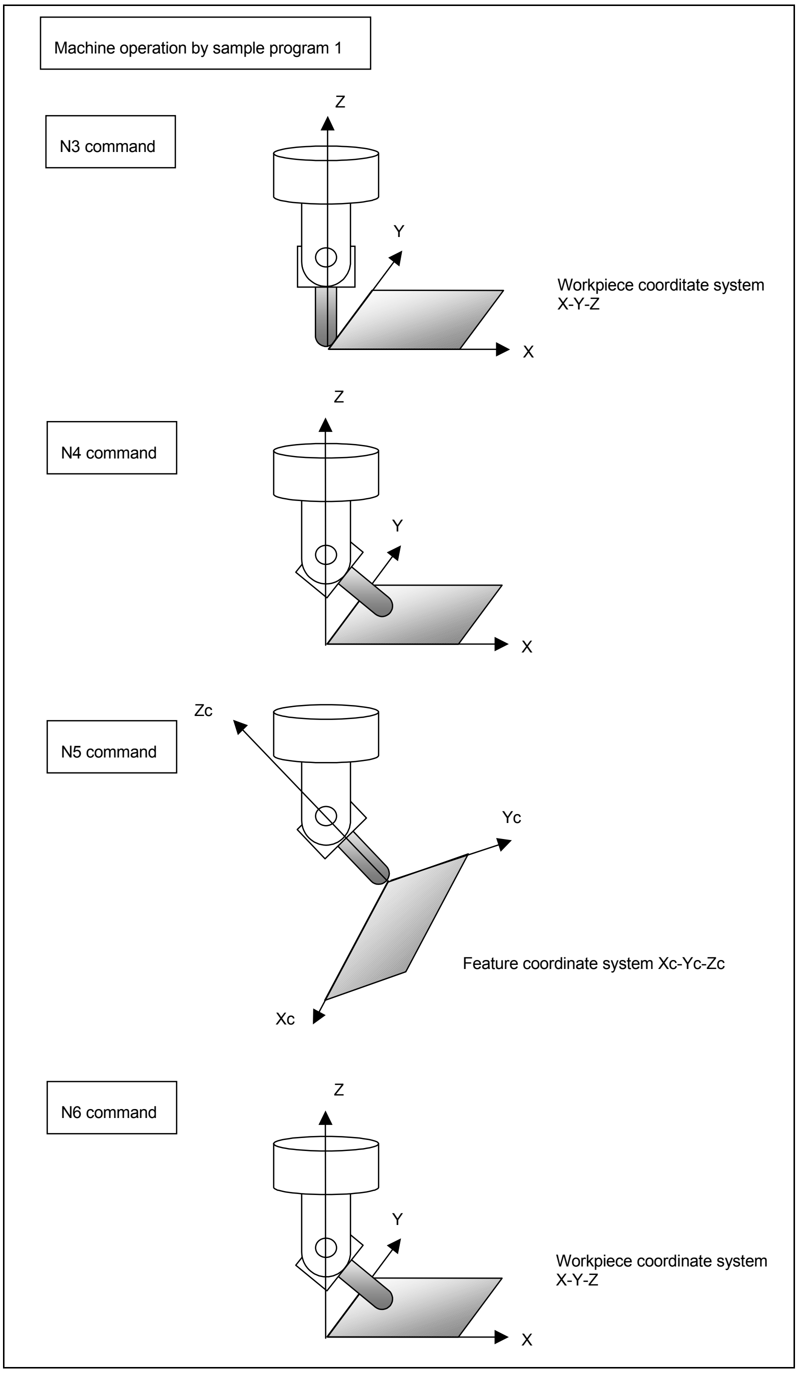
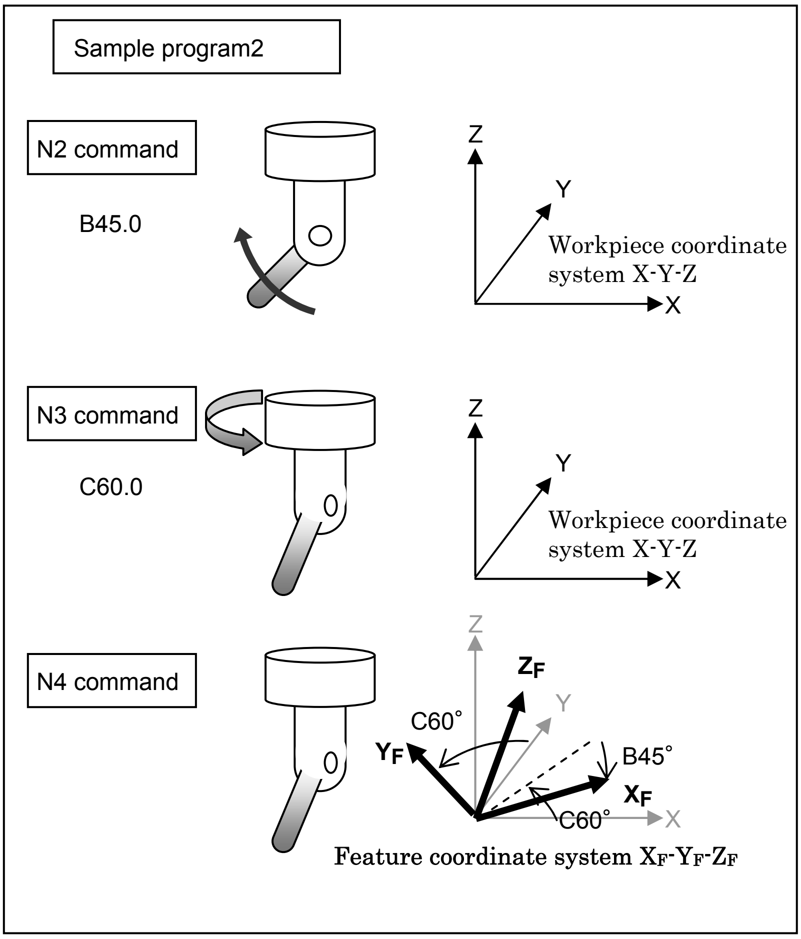


Fig. 5.7.1.7 (h)

N3 block: Performs tool length compensation in the workpiece coordinate system.
The tool center point moves to the origin of the workpiece coordinate system.
N4 block: Tilts the tool.
N5 block: Sets a feature coordinate system where the tool axis direction is the Z-axis direction and the tool center point is placed at the origin.
N6 block: Cancels the feature coordinate system to return to the workpiece coordinate system.

Sample program 2
O0100 ;
N1 G54 G90 G00 B0 C0 ;

N2 B45.0 ;
N3 C60.0 ;
N4 G68.3 P1 X0 Y0 Z0 ;



N2 block: Tilts the tool. (B45 deg)
N3 block: Tilts the tool. (C60 deg)
N4 block: The direction of the reference coordinate system of feature coordinate system is the direction of workpiece coordinate system because the reference tool axis direction is Z direction. The feature coordinate system is the coordinate system that the reference coordinate system is rotated by 45 deg. around Y direction and 60 deg. around Z direction.

X_F : The direction that X direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.
Y_F : The direction that Y direction of workpiece coordinate system is rotated by 60 deg. round Z direction of workpiece coordinate system.
Z_F : The direction that Z direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.

- Multiple G68.3

After the tool axis direction is changed in G68.3 mode, by specifying G68.3, a new feature coordinate system where the tool axis direction is the +Z-axis direction can be specified.

Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "AC type reference tool axis Z-axis".

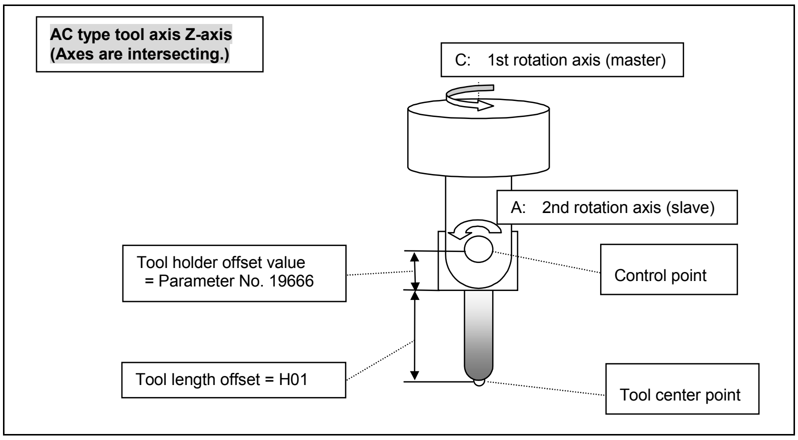


Fig. 5.7.1.7 (i)

Sample program 2

```
O0200 ;
N1 G55 ;
N2 G01 A90.0 F1000 ;
N3 G68.3 X0 Y0 Z0 R0;
:
N4 X10.0 Y0 Z0 ;
N5 C90.0;
N6 G68.3 X10.0 Y0 Z0 ;
:
N7 G69 ;
:
```

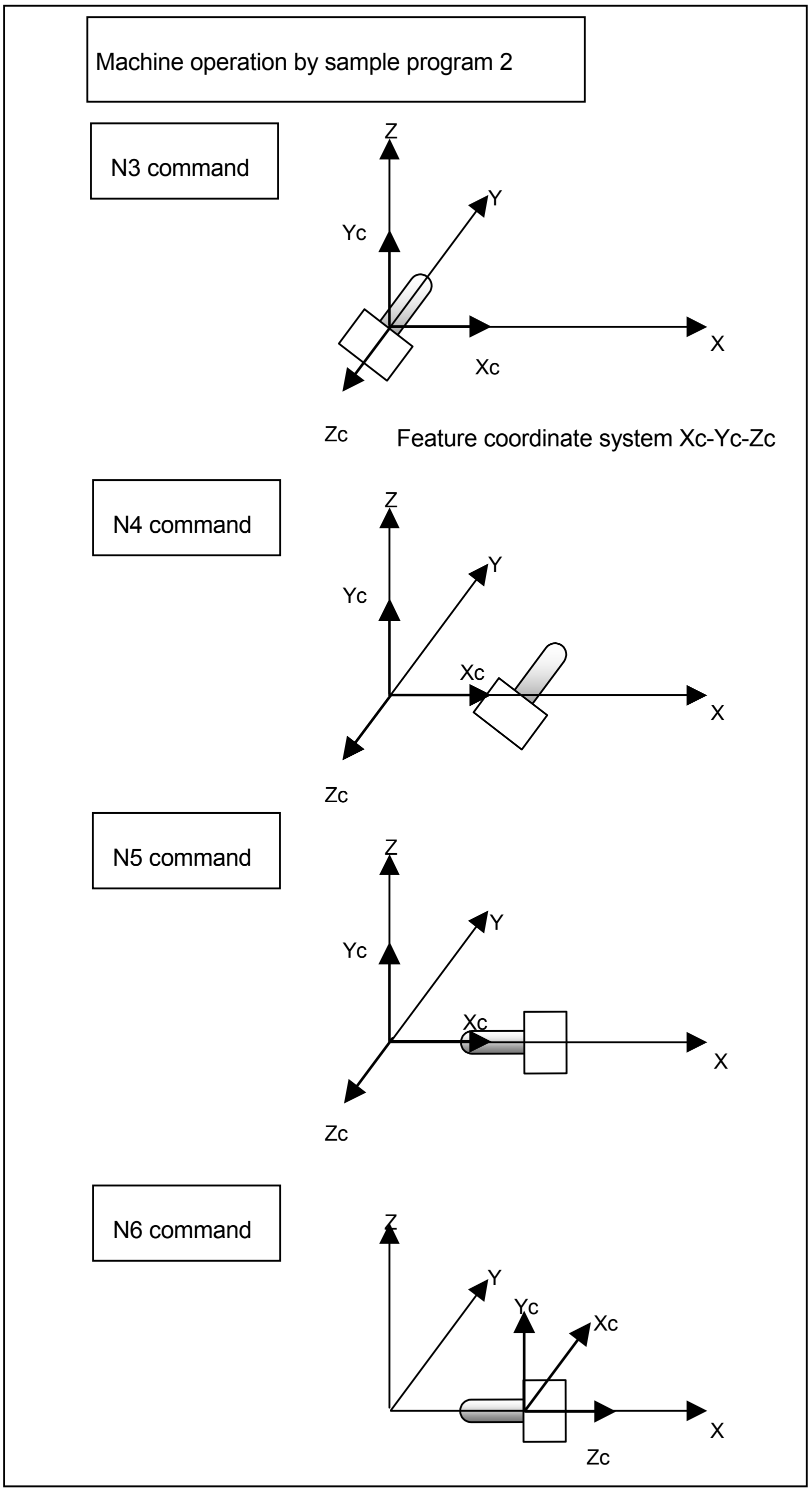


Fig. 5.7.1.7 (j)

N3 block: Sets a feature coordinate system according to the tool direction.
N4 block: Specifies coordinates in the feature coordinate system.
N5 block: Changes the tool direction.
N6 block: Sets a feature coordinate system according to the tool direction.

5.7.2 Multiple command of tilted working plane indexing

5.7.2.1 Absolute multiple command

By additionally specifying G68.2 in the tilted working plane indexing mode, a feature coordinate system produced by additionally applying coordinate system conversion to the workpiece coordinate system can be set. The workpiece coordinate system is resumed by specifying G69.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the workpiece coordinate system.

> **NOTE**
> Before specifying G68.2, cancel tool length compensation and tool radius compensation. If G68.2 is specified during tool length compensation or tool radius compensation, alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)" is issued.

Example of operation

An example of operation on a tool rotation type machine is explained below.
The machine configuration is "BC type with the reference tool axis being the Z-axis".

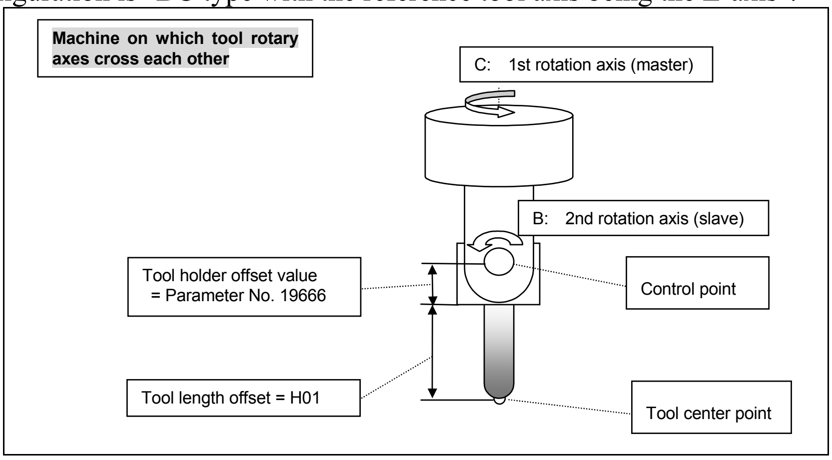


Fig. 5.7.2.1 (a)

Sample program 2

```
O0100 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
  :
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.2 X5.0 Y20.0 Z0 I90.0 J90.0 K0 ;
N7 G53.1;
  :
N8 G69 ;
  :
```

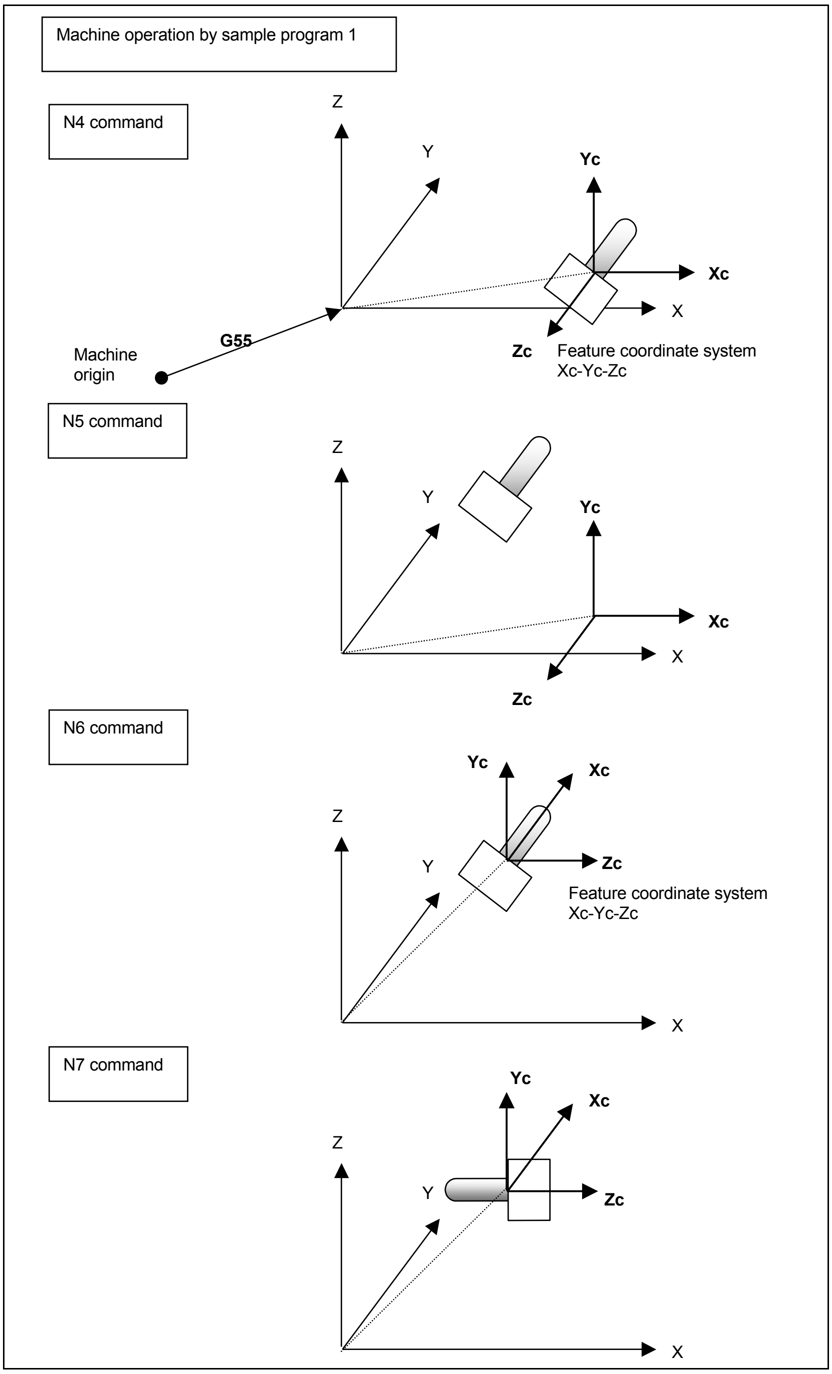


Fig. 5.7.2.1 (b)

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Sets a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

5.7.2.2 Incremental multiple command

By specifying G68.4, coordinate system conversion can be applied to the currently set feature coordinate system.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the immediately preceding feature coordinate system.

Table 5.7.2.2 (a)

Specification method	Incremental multiple command
Eulerian angle	G68.4
Roll-pitch-yaw	G68.4 P1
Three points	G68.4 P2
Two vectors	G68.4 P3
Projection angles	G68.4 P4

> **NOTE**
> Before specifying G68.4, cancel tool radius compensation. If G68.4 is specified during tool radius compensation, alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)" is issued.

Example of operation

An example of operation on a tool rotation type machine is explained below.
Rotary axis C rotates about the Z-axis (master axis).
Rotary axis B rotates about the Y-axis (slave axis).

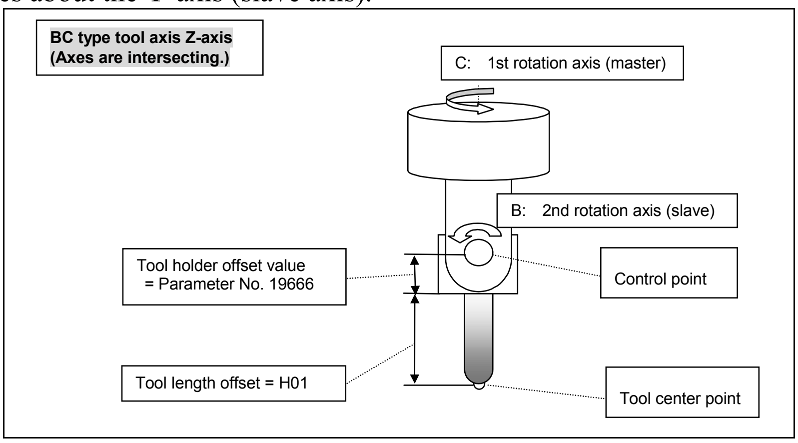


Fig. 5.7.2.2 (a)

Sample program 3

```
O0200 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
  :
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.4 X-15.0 Y0 Z-15.0 I90.0 J90.0 K-90.0 ;
N7 G53.1;
  :
N8 G69 ;
```

:

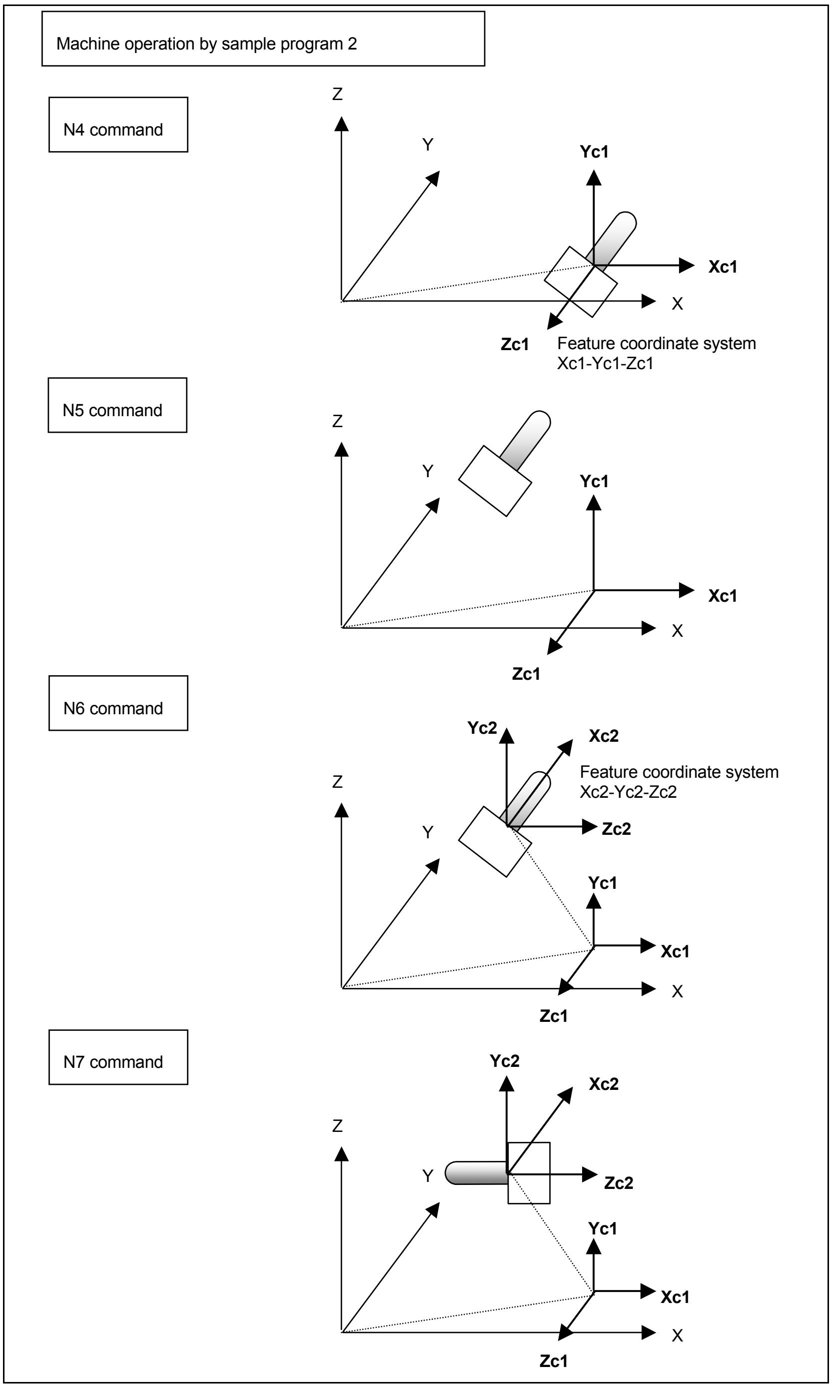


Fig. 5.7.2.2 (b)

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Applies coordinate system conversion to the feature coordinate system based on Eulerian angle to set a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

5.7.3 Tool Axis Direction Control

5.7.3.1 Tool axis direction control

G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction.

Example of operation

The following gives an operation example in the machine configuration below.

- Table rotation type
- Master axis: About the Y-axis (B axis)
- Slave axis: About the Z-axis (C axis)
- Reference tool axis direction: Z direction
- Increment system for the rotation axis: 1/10(IS-C)

Program example 1

G68.2 I90.0 J0.0005 K-90.0 (rotation by 0.0005 degree about the Y-axis)
G53.1

The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0010 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0005 degree
  C axis: 0.0000 degree

Program example 2

G68.2 I90.0 J10 K-90.0 (rotation by (minimum command unit of the rotation angles × 10) about the Y-axis)
G53.1

The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0100 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0001 degree
  C axis: 0.0000 degree

- Tool rotation type machine

The following paragraphs describe several cases of the tool rotation type machine operation.

Operation description 1:
When G43 (tool length compensation) is specified for a machine with its axes crossing one another

The G53.1 command, when specified after the G68.2 command, automatically controls the rotary axis in such a way that the tool axis will be oriented in the +Z direction of the feature coordinate system.

Example)

```
O100 (Sample Program1) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z30.0 F1000 ;
N5 G53.1 ;
N6 G43 H01 X0 Y0 Z0 ;
N7 . . .
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
In addition, the tool axis, tool rotation axis B, and tool rotation axis C cross one another.

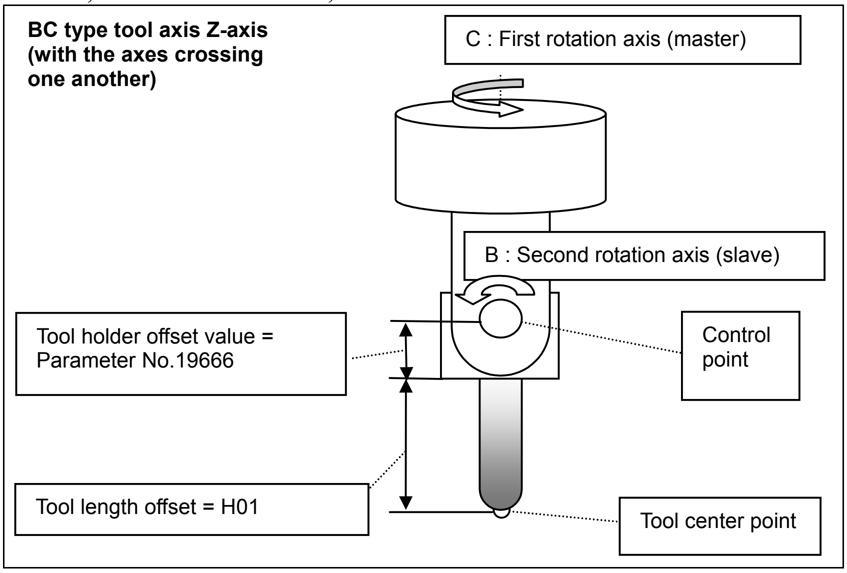


Fig. 5.7.3.1 (a)

Block N3 :	Defines a feature coordinate system in the workpiece coordinate system.
Block N4 :	Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 :	Exerts automatic control over the rotary axes.
Block N6 :	Performs tool length compensation in the feature coordinate system. The tool center point is shifted to the origin of the feature coordinate system.

Fig. 5.7.3.1 (b) shows the behavior of the machine when it runs sample program 1.

- **Sample program 1 (with axes crossing one another)**

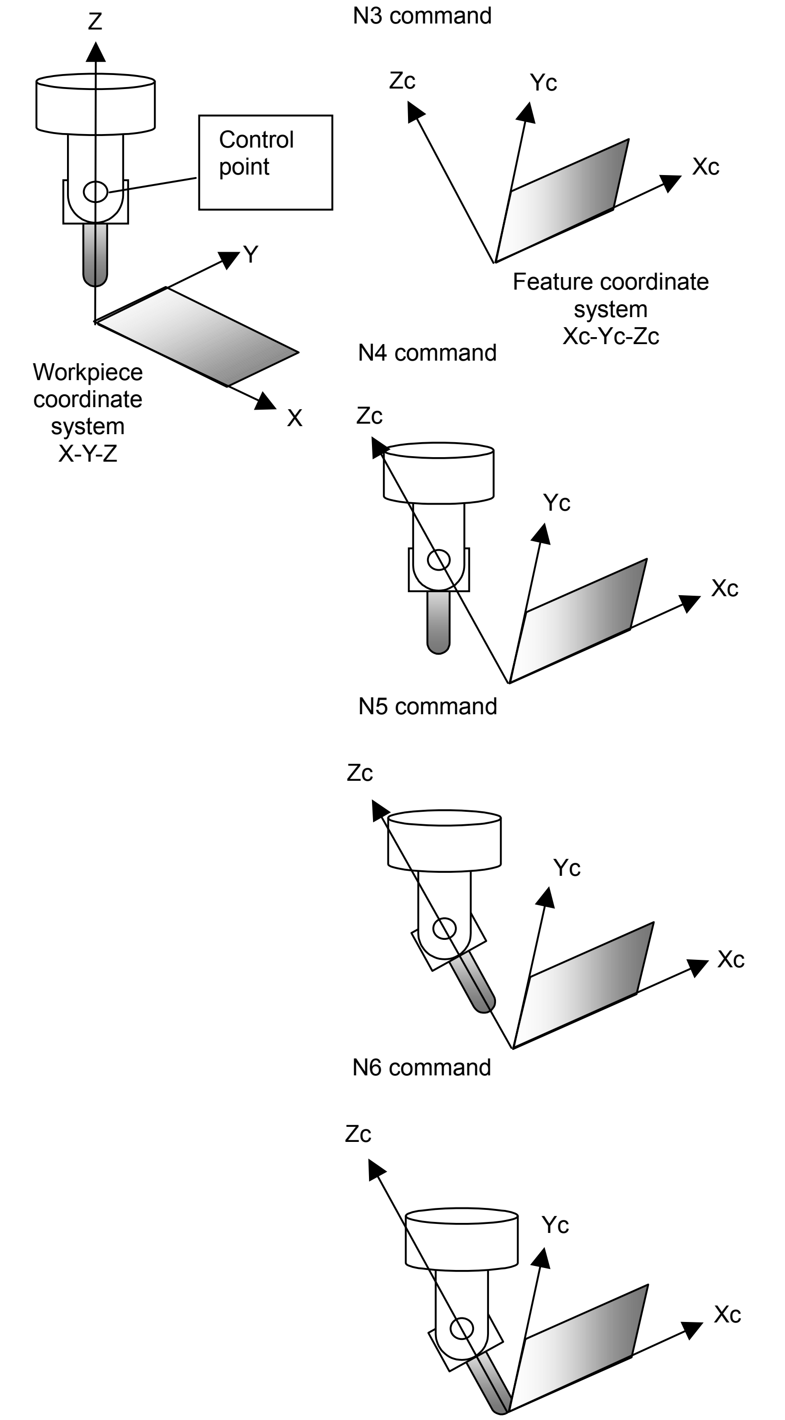


Fig. 5.7.3.1 (b) Tool axis direction control 1

Operation description 2:
When G43 (tool length compensation) is specified for a machine with no axis crossing

Here is the case where no axis of the machine crosses any other axis.
It is assumed that sample program 1 is used.

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
It is assumed, however, that the tool axis does not cross the B-axis while the B-axis and C-axis cross each other.

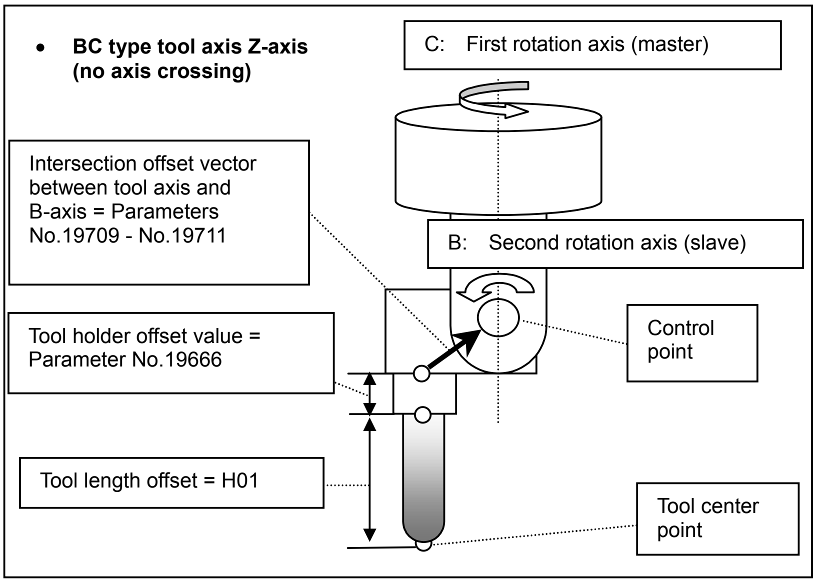


Fig. 5.7.3.1 (c)

Block N4 : Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 : Exerts automatic control over the rotary axes.
Block N6 : An intersection offset vector between the tool axis and the B-axis with automatic control for rotary axes taken into consideration is output in the feature coordinate system.
Performs tool length compensation in the feature coordinate system.
The tool center point is shifted to the origin of the feature coordinate system.

This is also true when the B-axis does not cross the C-axis.
For explanations about the offset to be applied when the B-axis does not cross the C-axis, see the descriptions about parameters No.19712, No.19713, and No.19714.

Fig. 5.7.3.1 (d) shows the behavior of the machine when it runs sample program 1.

- **Sample program 1 (no axis crossing)**

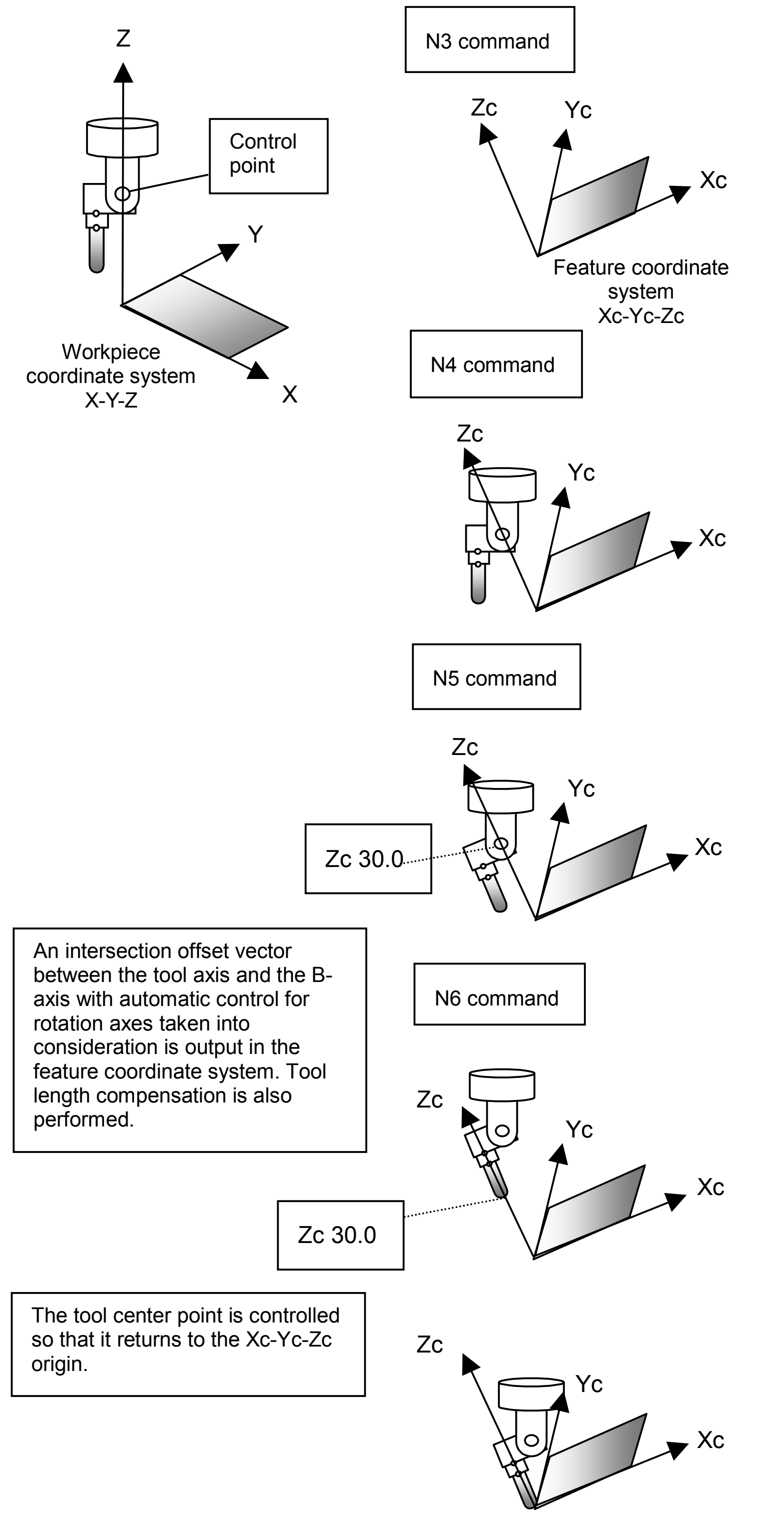


Fig. 5.7.3.1 (d) Tool axis direction control 2

Operation description 3:
When no G43 (tool length compensation) command is specified or if no G53.1 (tool axis direction control) command is specified

Sample program 2 of O200 is equivalent to sample program 1 except that sample program 2 has no tool length compensation command (G43).
Example)

```
O200 (Sample Program2) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G53.1 ;
N6 . . . ;
```

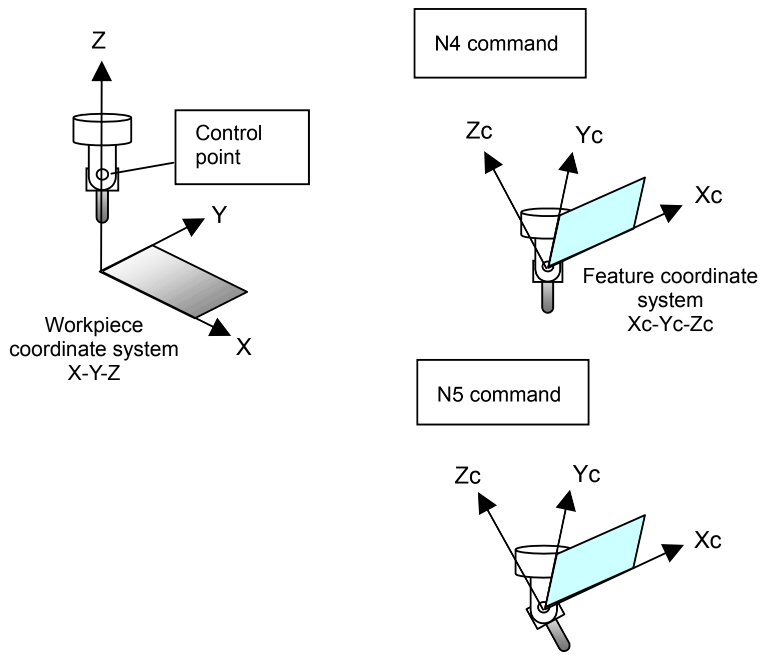
In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Fig. 5.7.3.1 (e) shows the behavior of the machine when it runs sample program 2.

Sample program 3 of O300 is equivalent to sample program 1 except that sample program 3 has no tool axis direction control command (G53.1).
Example)

```
O300 (Sample Program3) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G43 H01 ;
N6 . . . ;
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Tool length compensation is applied in the +Z-axis direction of the feature coordinate system.
Fig. 5.7.3.1 (f) shows the behavior of the machine when it runs sample program 3.

- **Sample program 2 (with axes crossing one another)**



- **Sample program 2 (no axis crossing)**

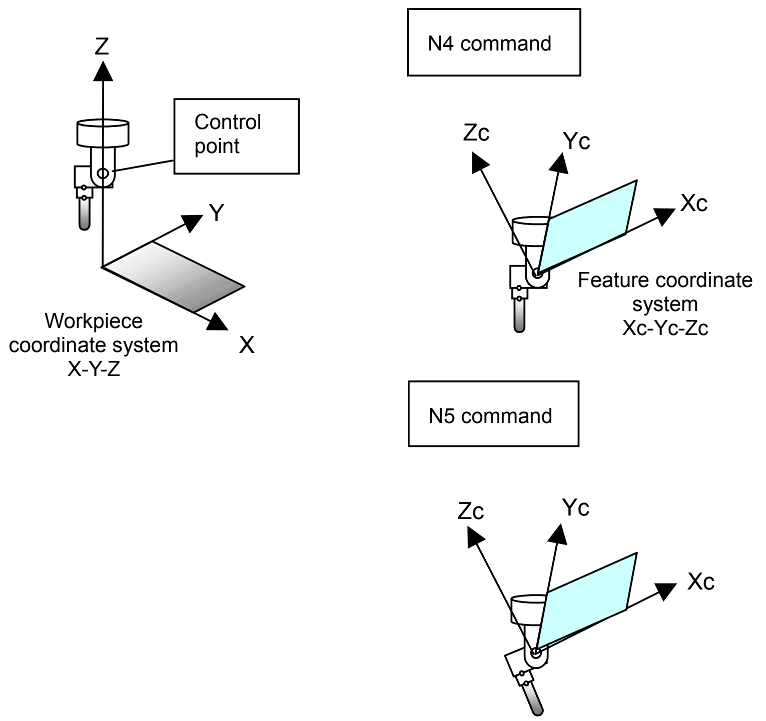
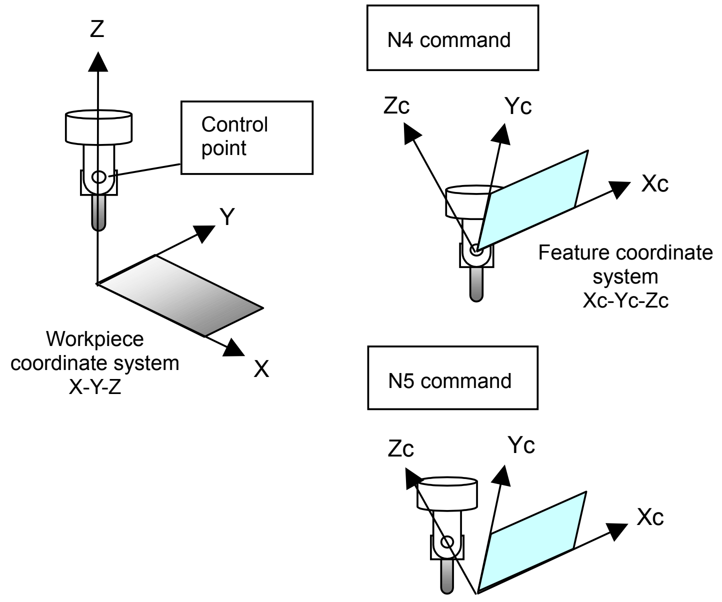


Fig. 5.7.3.1 (e) When the tool length compensation command is not specified

- **Sample program 3 (with axes crossing one another)**



- **Sample program 3 (no axis crossing)**

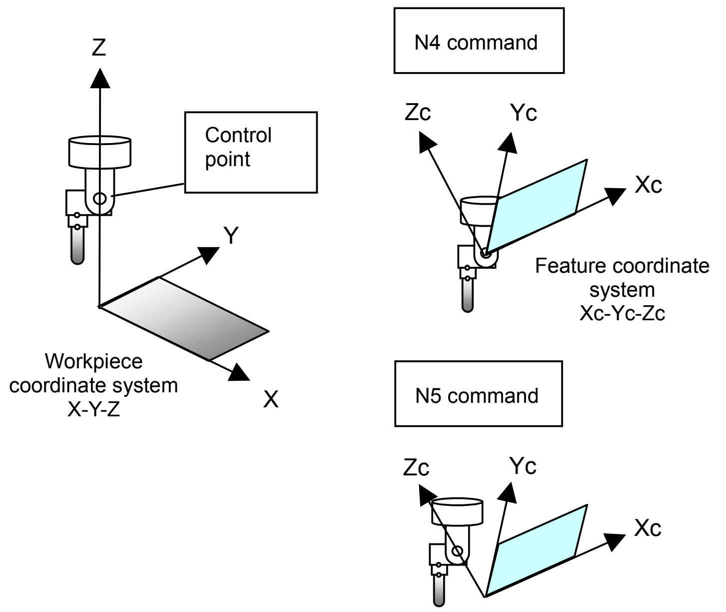


Fig. 5.7.3.1 (f) When the tool axis direction control command is not specified

- Composite type machine Basic operation

This function is also available for a composite type machine in which the tool head rotates on the tool rotation axis and the table rotates on the table rotation axis.
The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.
Given the A-axis and B-axis shown in Fig. 5.7.3.1 (g), control is performed in such a way that the A-axis rotates until Zc comes in the X-Z plane and the B-axis is controlled so that the tool axis is oriented toward the +Z-axis direction of the feature coordinate system.

- **Tool axis direction control for a composite type machine**

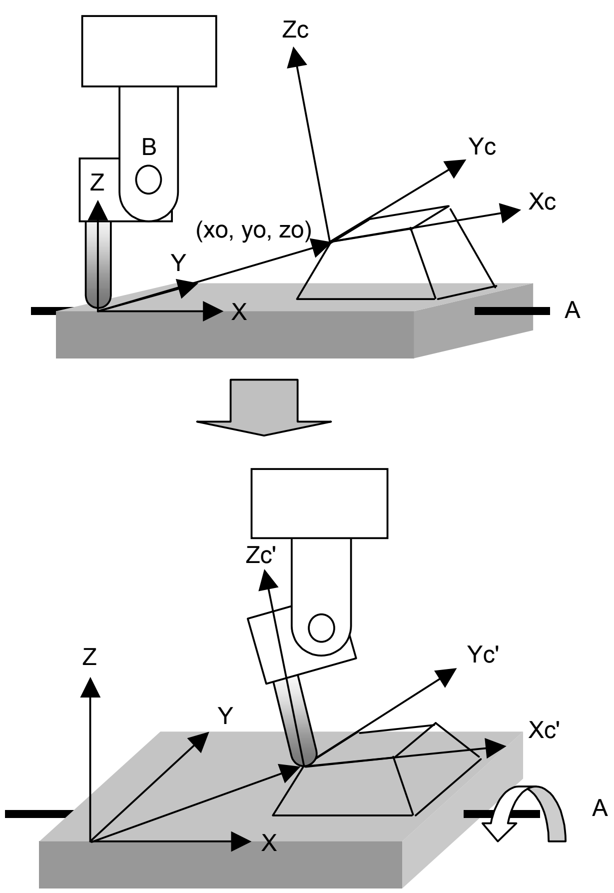


Fig. 5.7.3.1 (g) Composite type machine

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The composite type machine shown in Fig. 5.7.3.1 (g) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the tilted working plane indexing (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 5.7.3.1 (h).)
In the composite type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

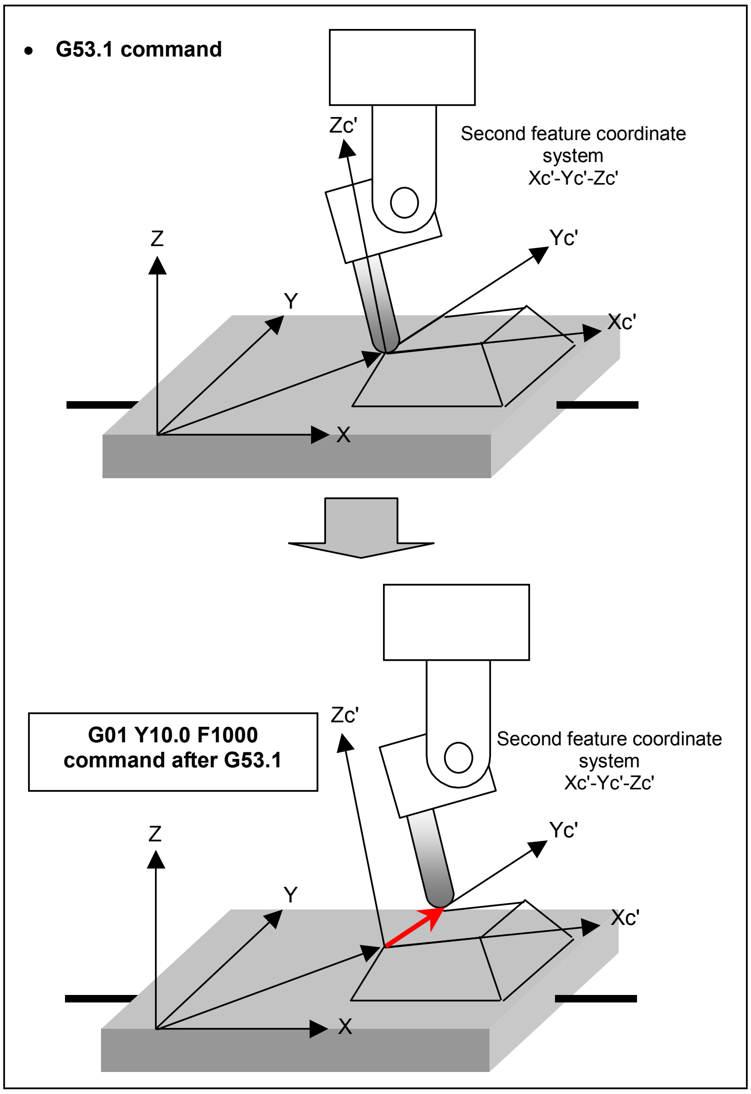


Fig. 5.7.3.1 (h) Resetting of the feature coordinate system

- Rotation direction of the table rotation axis

The composite type machine shown in Fig. 5.7.3.1 (g) is explained as an example.
Set parameter No.19684 to 1 if the rotation direction of the rotation table corresponding to the positive-direction move command is clockwise when viewed from the positive direction of the rotation center axis on which the table rotation axis rotates. If the rotation direction is counterclockwise, set parameter No.19684 to 0.
Let's take sample program 4 of O400 as an example, where the movement of the table is specified by G53.1.
If parameter No.19684 is set to 1, control is performed in such a way that the table is rotated to A-45.0.
If parameter No.19684 is set to 0, control is performed in such a way that the table is rotated to A45.0.
Example)

```
O400 (Sample Program4) ;
N1 G68.2 X100.0 Y100.0 Z0 I180.0 J45.0 K0 ;
N2 G53.1 ;
N3 . . . ;
```

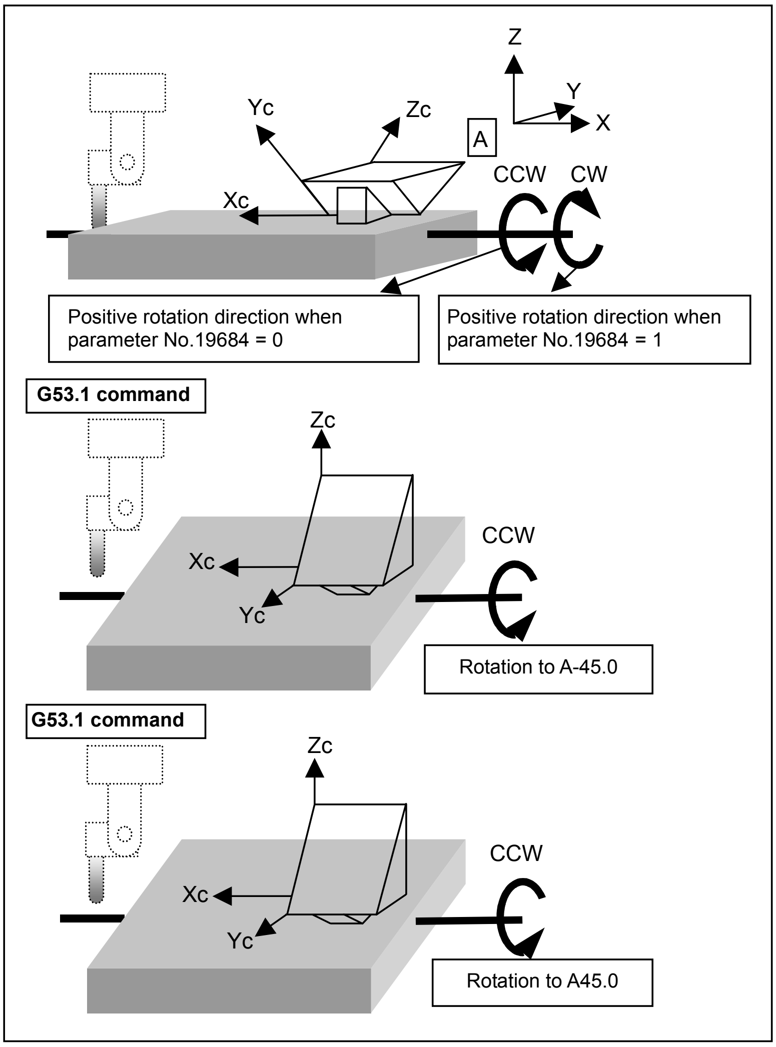


Fig. 5.7.3.1 (i) Rotation direction of the table rotation axis

- Table rotation type machine
Basic operation

This function is also usable for a table rotation type machine with two table rotation axes.
The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.
Given the A-axis and C-axis shown in Fig. 5.7.3.1 (j), the A-axis and C-axis rotate until Zc comes in the X-Z plane and the tool axis is directed toward the +Z-axis direction of the feature coordinate system.

- **Tool axis direction control for a table rotation type machine**

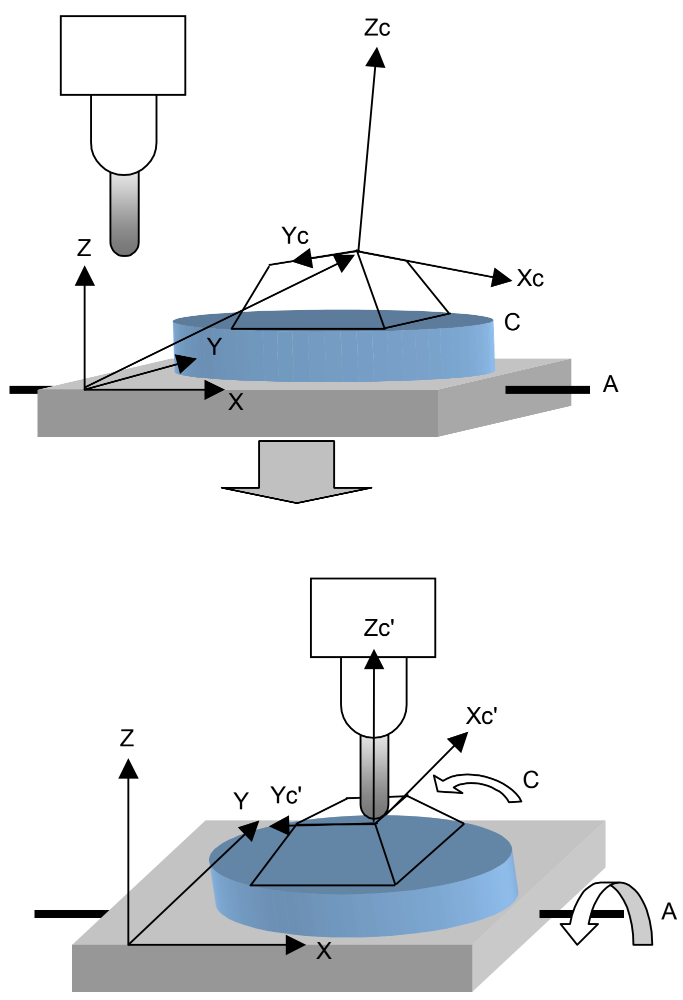


Fig. 5.7.3.1 (j) Table rotation type machine

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The table rotation type machine shown in Fig. 5.7.3.1 (j) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the tilted working plane indexing (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 5.7.3.1 (k).)
In the table rotation type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

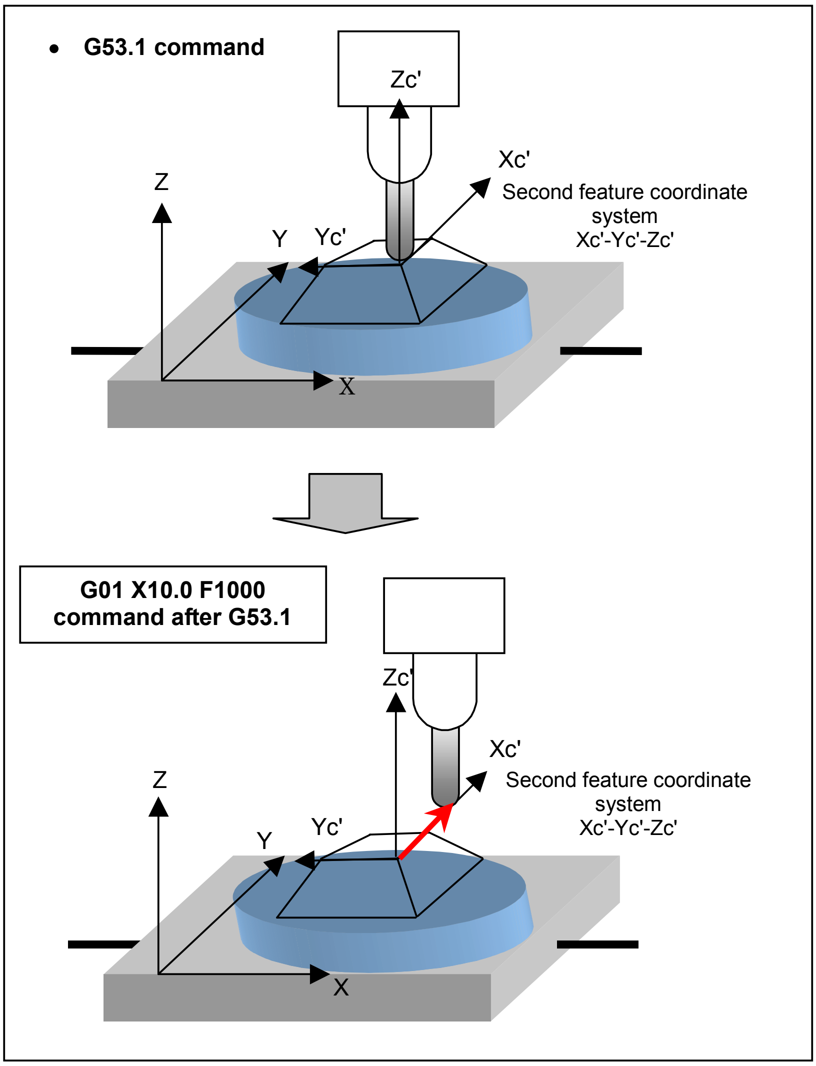


Fig. 5.7.3.1 (k) Resetting of the feature coordinate system

- Angle of the rotary axis

When tool axis direction control (G53.1) has been performed, more than two pairs of "computed angles" of the rotary axes usually exist.

The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the tool axis direction specified by G53.1.

The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.

When the upper and lower limits of the movement range of rotary axes are specified by parameters No. 19741 to No. 19744 at this time, a calculated angle that makes the two axes fall within the movement range is selected.

"Output judgment conditions"

Tool rotation type or table rotation type machine

<1> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller.

↓
↓ When the master axis moving angle is the same
↓

<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.

↓
↓ When the slave axis moving angle is the same
↓

<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

↓
↓ When the master axis angle is equally near to 0 degree
↓

<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Composite type machine

<1> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller.

↓
↓ When the table moving angle is the same
↓

<2> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.

↓
↓ When the tool moving angle is the same
↓

<3> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

↓
↓ When the master axis angle is equally near to 0 degree
↓

<4> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."
When bit 5 (PRI) of parameter No.19608 is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

The "movement judgement" process is explained below.
When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that the machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A $\theta 1$ degree; B $\phi 1$ degree)
(A $\theta 2$ degrees; B $\phi 2$ degrees) where $\theta 1 \leq \theta 2$ and $\phi 1 \leq \phi 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows (when bit 5 (PRI) of parameter No.19608 is 0).

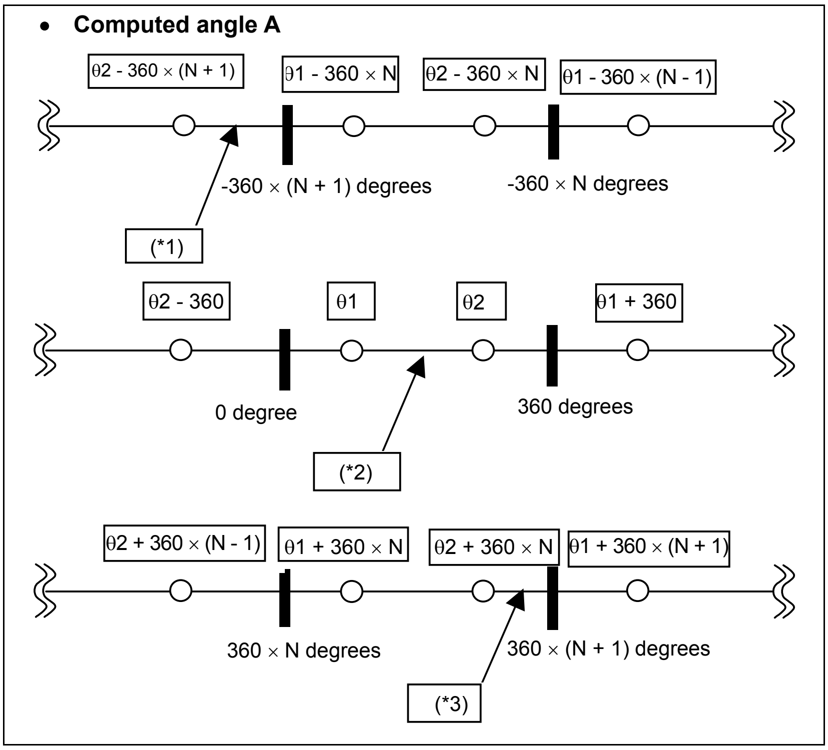


Fig. 5.7.3.1 (l) "Movement judgment"

When the PA angle is (*1):
The output angle is: (A θ2 - 360 × (N + 1) degrees; B ϕ2 degrees).
Namely, θ2 - 360 × (N + 1) degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A θ1 degrees; B ϕ1 degrees).
Namely, θ1 degrees is adopted that is nearer to the computed angle of A, and ϕ1, which is the same group as θ1, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A θ2 + 360 × N degrees; B ϕ2 degrees).
Namely, θ2 + 360 × N degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2. is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."
If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B.
Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

⚠ CAUTION

1. To use the rotary axis roll-over function, set parameter No. 1260 (amount of rotary axis movement per rotation) to 360 degrees.
2. A stroke limit before movement is applied to the rotary axis subject to tool axis direction control.

⚠ CAUTION

3 If the setting of the lower limit (parameters Nos. 19742 and 19744) is greater than that of the upper limit (parameters Nos. 19741 and 19743), alarm PS5459 , "MACHINE PARAMETER INCORRECT" is issued.

4 If there is no calculated angle that falls within the movement range because the movement range is too small, alarm PS5459 is issued.

5 When the parameters that specify the upper limit and lower limit of the movement range are set to 0, no movement range is assumed to be specified.

The "output angle" is explained below using a tool rotation type machine as an example. (Assume that bit 5 (PRI) of parameter No.19608 is 0.)
This example illustrates a machine having a "BC type tool axis Z."

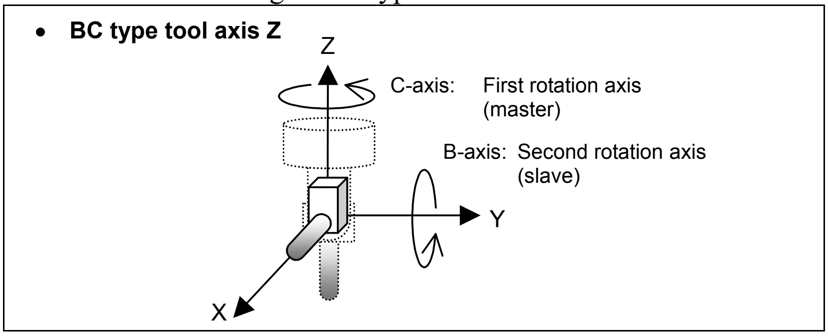


Fig. 5.7.3.1 (m) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the +X-axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C-axis angle is 0 degree and whose B-axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.
In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

- **BC type tool axis Z**

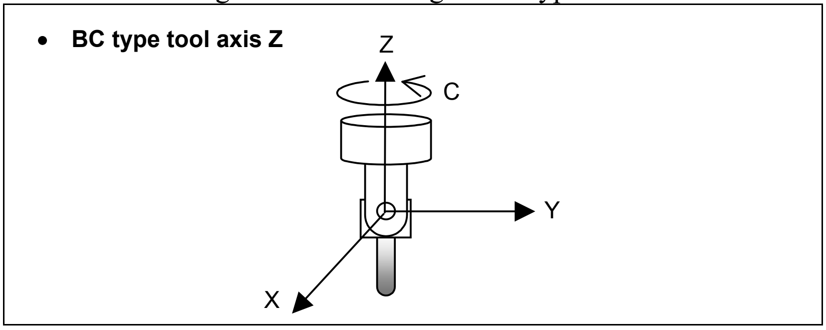


Fig. 5.7.3.1 (n) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

- Specification for rotary axes

The tilted working plane indexing assumes that positioning of a rotary axis is performed by tool axis direction control (G53.1) and then machining is performed without the rotary axis being moved.
When movement is made with the rotary axis specified directly, the movement of the rotary axis is not considered in the movement of the linear axis. Do not specify the movement of the rotary axis during execution of the tilted working plane indexing to ensure correct machining.

Second Rotation Axis Control in Tool Axis Direction Control Where the End Point is a Singular Point

When the end point of tool axis direction control (G53.1/G53.6) during execution of the tilted working plane indexing is a singular point, the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.
To enable this function, set bit 4 (CFW) of parameter No. 11221 to 1.

When bit 4 (CFW) of parameter No. 11221 is set to 0, if the end point of tool axis direction control (G53.1/G53.6) is a singular point, the second rotation axis does not operate and only the first rotation axis turns. As a result, the X- and Y-directions of the feature coordinate system depend on the second rotation axis immediately before tool axis direction control. (Fig. 5.7.3.1 (o), Fig. 5.7.3.1 (p))

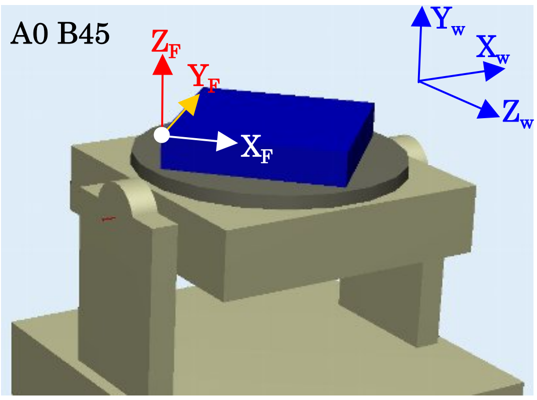


Fig. 5.7.3.1 (o) Before G53.1

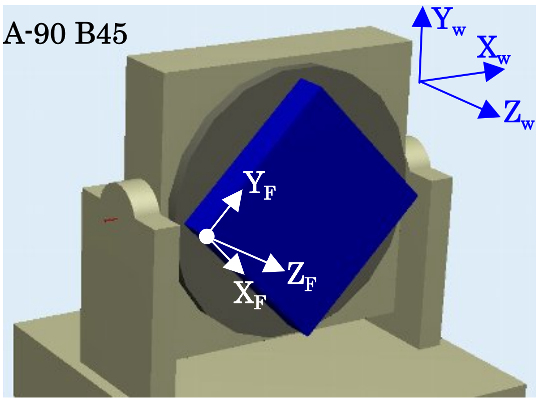


Fig. 5.7.3.1 (p) After G53.1
The second rotation axis (B-axis) does not turn.

When this function is enabled (bit 4 (CFW) of parameter No. 11221 = 1), the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system. (Fig. 5.7.3.1 (q),Fig. 5.7.3.1 (r))

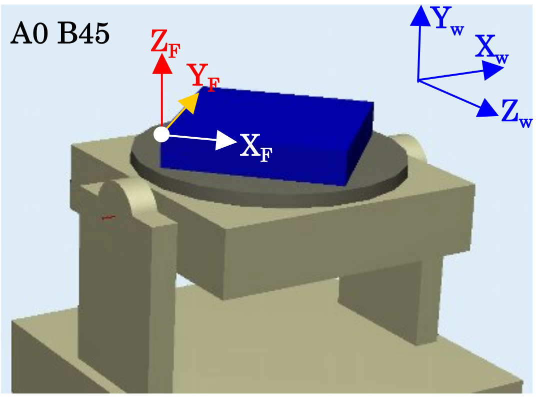


Fig. 5.7.3.1 (q) Before G53.1

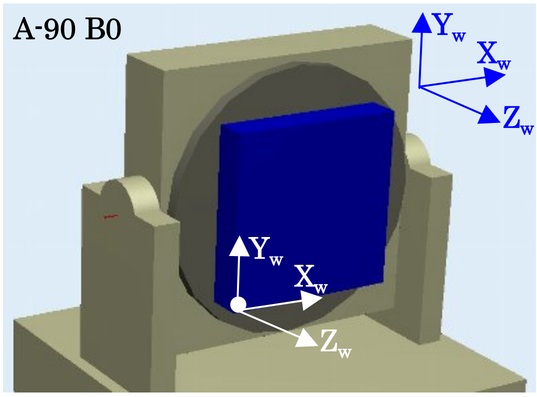


Fig. 5.7.3.1 (r) After G53.1
The second rotation axis (B-axis) moves so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.

5.7.3.2 Tool center point retention type tool axis direction control

In tilted working plane indexing, tool center point retention type tool axis direction control (G53.6) can be specified so that tool can be perpendicular to the tilted plane with the tool center point maintained on the workpiece. In tool center point retention type tool axis direction control, end point of tool center point can be specified. Then tool center point moves on the feature coordinate system fixed on the workpiece. The cycle time can be shortened by moving tool axis direction and tool center point simultaneously.
By specifying the distance between the tool center point to the rotation center with R, it is possible to move the tool so that it is perpendicular to the titled plane while retaining the rotation center shifted from the tool center point.

Format

M

G53.6 (H_) (R_) X_ Y_ Z_ ; Tool center point retention type tool axis direction control

H : Tool compensation number
R : Distance from the tool center point to the rotation center
(Specify R as a radius value.)
X_ Y_ Z_ : End point of tool center point

G53.6 is a one shot G code.
To use tool center point retention type tool axis direction control, a tool length offset number is required. If no H command is specified, the H code is regarded as modal information.
To shift the rotation center from the tool center point, enter the distance from the tool center point to the rotation center, using the R command. If no R command is specified, the distance is regarded as R0, and the tool moves while retaining the tool center position on the workpiece so that it is perpendicular to the tilted plane.
Specify end point of tool center point in feature coordinate system.

⚠ **CAUTION**

1. Specify tool center point retention type tool axis direction control (G53.6) in tool length compensation cancel mode (G49). If it is specified in a mode other than G49 mode, alarm PS5458, "ILLEGAL USE OF G53.1/G53.6" is issued.
2. If G68.2 is not specified before G53.6 is specified, alarm PS5458 is issued.
3. Specify G53.6 independently. If an axis movement command is specified in the same block, alarm PS5458 is issued.
4. The tool center point retention type tool axis direction control (G53.6) block becomes a block that suppresses buffering.
5. If tool center point retention type tool axis direction control does not have an H command, and the modal information for H is 0, alarm PS5458 is issued.
6. For the feedrate, the movement speed of the rotation axis is applied. During rapid traverse, it is regarded as the maximum rapid traverse rate, and as the specified speed during cutting feed.
7. Specify tool center point retention type tool axis direction control (G53.6) in cutter compensation cancel mode (G40). If it is specified in a mode other than G40 mode, alarm PS5458 is issued.
8. Specify tool center point retention type tool axis direction control (G53.6) in either G00 or G01 mode.
9. Specify tool center point retention type tool axis direction control (G53.6) with feed per minute or feed per revolution.
10. Do not perform manual intervention during tool center point retention type tool axis direction control (G53.6). Otherwise, alarm PS5458 is issued.
11. If the R command is specified, and if an exceedingly large value is input as R, alarm PS0143, "COMMAND DATA OVERFLOW" may be issued.

Example

Tool center point retention type tool axis direction control with R not specified

Fig. 5.7.3.2 (a) and Fig. 5.7.3.2 (b) show tool center point retention type tool axis direction control with R not specified. The tool moves so that it is perpendicular to the tilted plane while retaining the tool center point on the workpiece.

This function can be used by specifying G53.6 and specifying a tool length offset number with H. (If H has the modal information for the currently used tool, this function can be used without specifying H.)

```
O0002(TCP-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0
G53.6 H1
```

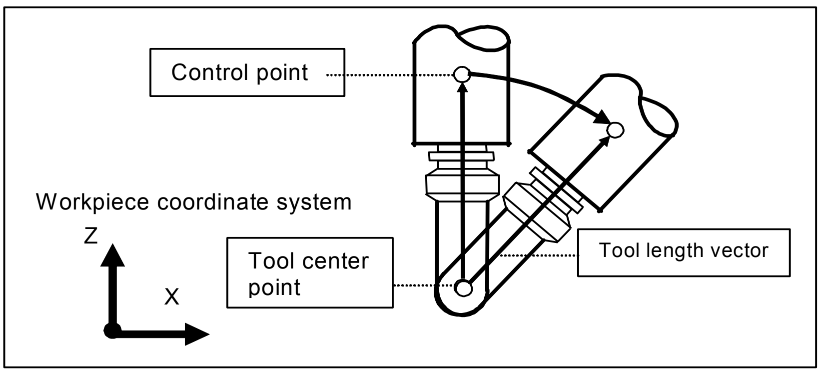


Fig. 5.7.3.2 (a) Operation of tool center point retention type tool axis direction control (tool rotation type)

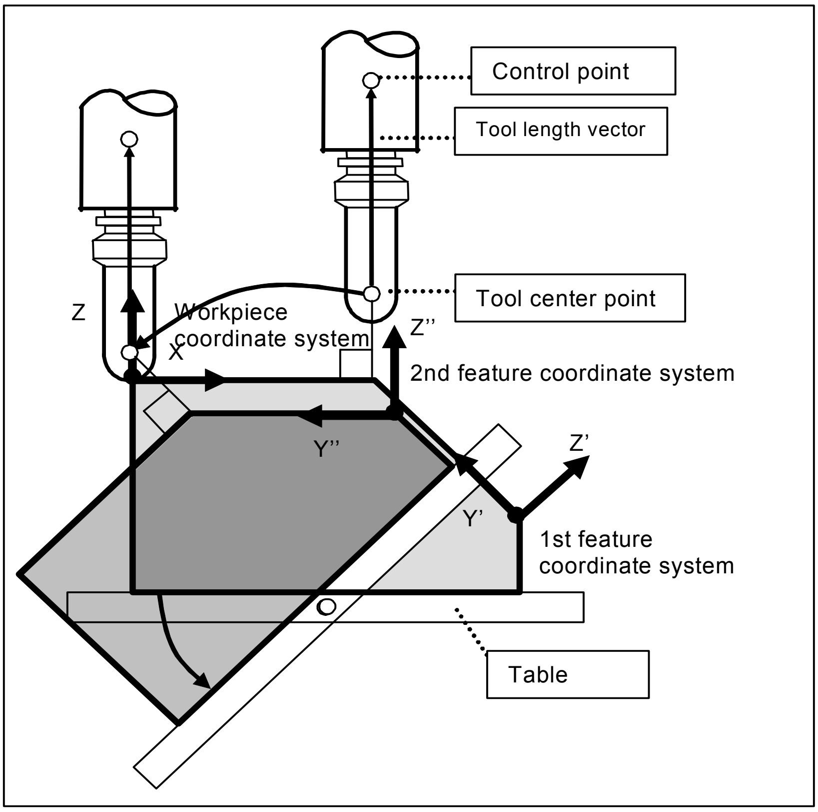


Fig. 5.7.3.2 (b) Operation of tool center point retention type tool axis direction control (table rotation type)

Fig. 5.7.3.2 (c) and Fig. 5.7.3.2 (d) show tool center point retention type tool axis direction control specified with end point of tool center point. Tool center point moves on the feature coordinate system fixed on the workpiece.

```
O0012(TCP-HOLD-TYPE-TOOL_ROT)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K-90.0
G53.6 H1 X100.0 Y0 Z0
```

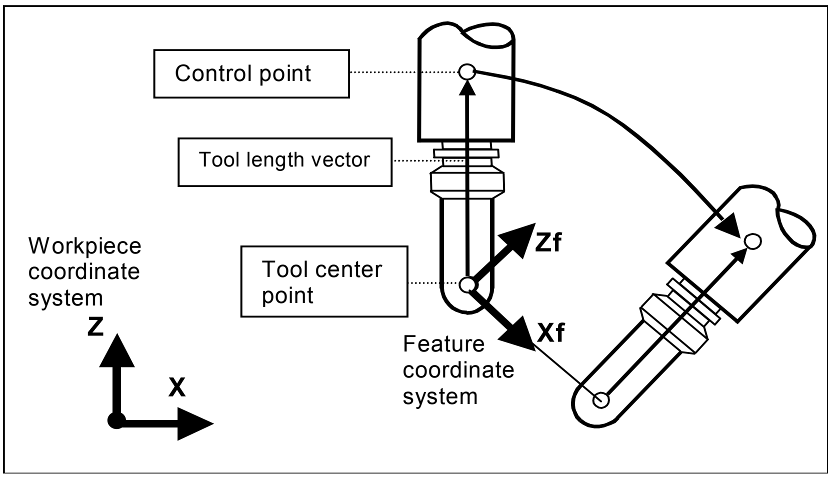


Fig. 5.7.3.2 (c) Operation of tool center point retention type tool axis direction control specified with end point of tool center point (tool rotation type)

O0022(TCP-HOLD-TYPE-TABLE_ROT)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K-90.0
G53.6 H1 X0 Y0 Z0

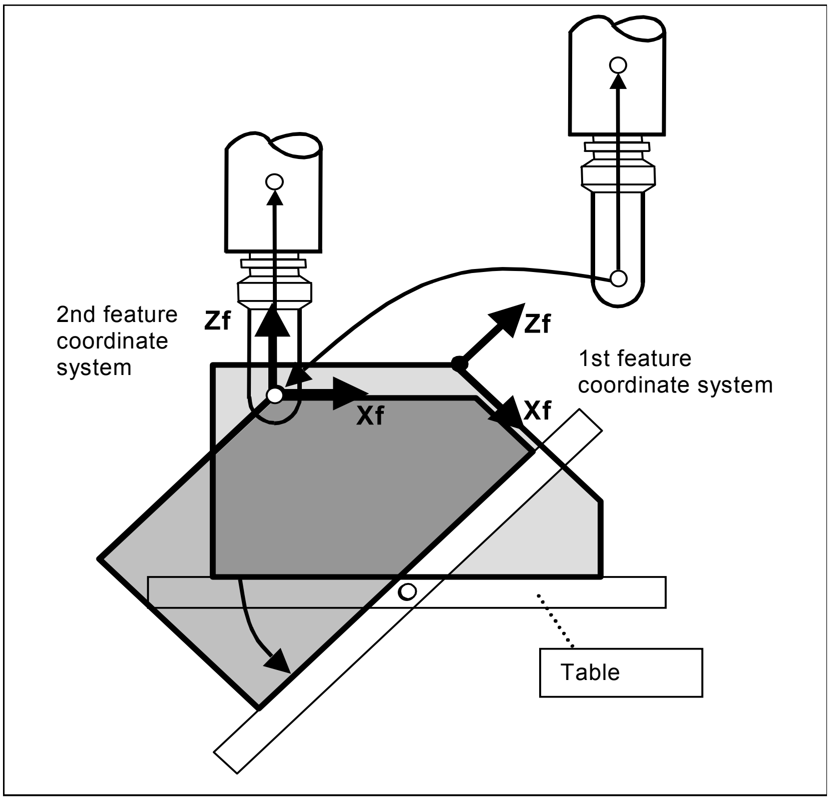


Fig. 5.7.3.2 (d) Operation of tool center point retention type tool axis direction control specified with end point of tool center point (table rotation type)

Rotation center compensation with the tool center point retention type
In tool center point retention type tool axis direction control, the rotation center can be shifted from the tool center point by specifying R.
Fig. 5.7.3.2 (e) and Fig. 5.7.3.2 (f) show cases in which the rotation center is shifted by specifying the distance from the tool center point to the workpiece with R.
By specifying this, the tool moves so that it is perpendicular to the titled plane while retaining the rotation center on the workpiece.
This function can be used by specifying G53.6 and specifying a tool length offset number with H and specifying the distance from the tool center point to the rotation center with R.

O0003(CENTER-OF-ROTATION-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0
G53.6 H1 R200.0

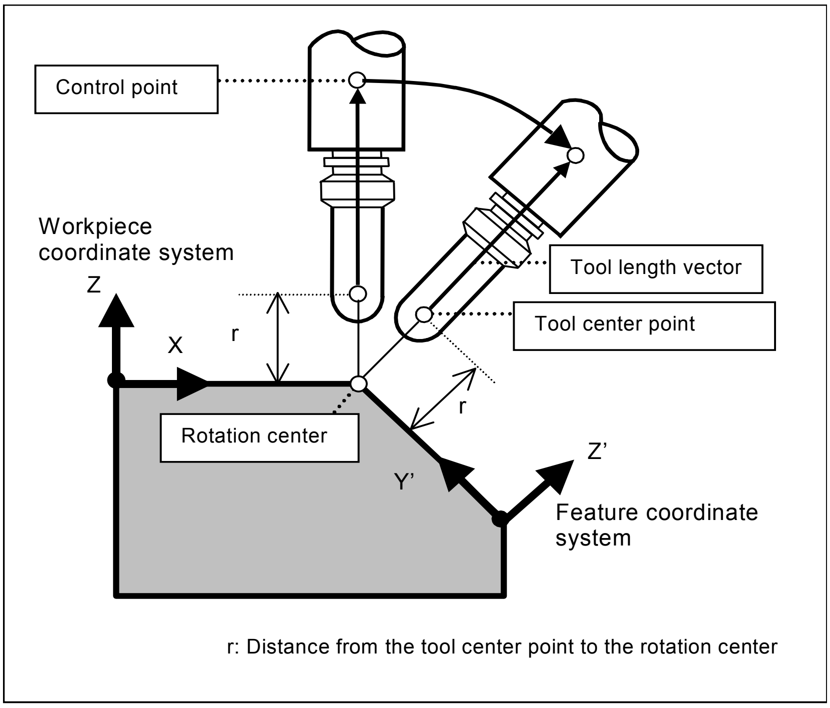


Fig. 5.7.3.2 (e) Operation of rotation center compensation with tool center point retention type tool axis direction control (tool rotation type)

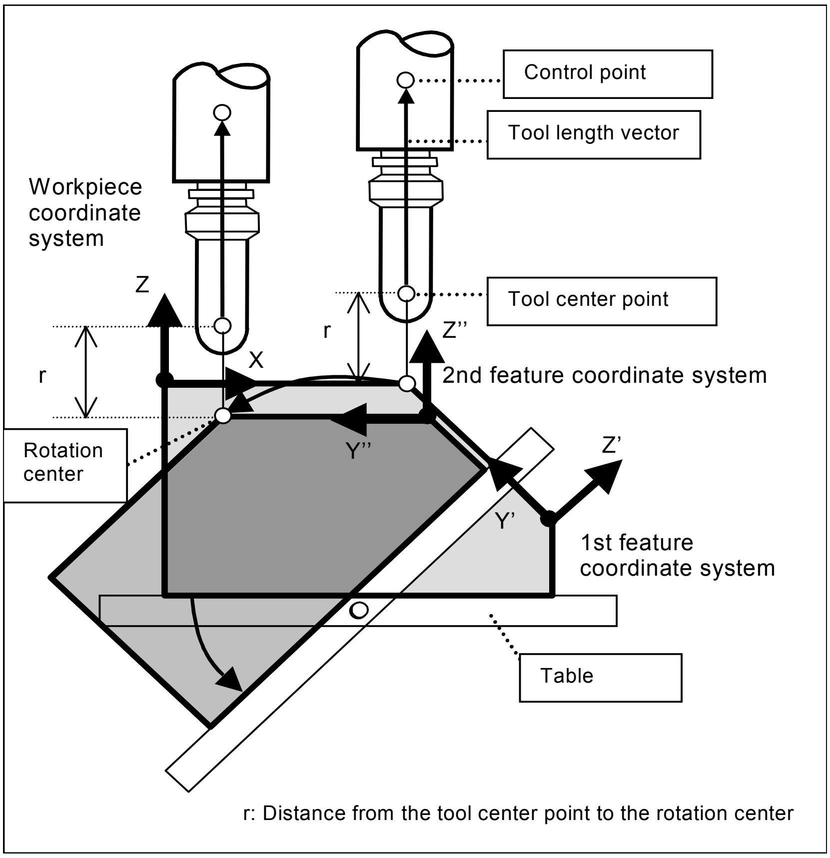


Fig. 5.7.3.2 (f) Operation of rotation center compensation with tool center point retention type tool axis direction control (table rotation type)

5.7.4 Tilted Working Plane Indexing in Tool Length Compensation

Overview

In tool length compensation (G43), G68.2/G68.4 (tilted working plane indexing) and G53.1 (tool axis direction control)/G53.6 (tool center point retention type tool axis direction control) can be specified.
Accordingly, the G68.2/G68.4 and G53.1/G53.6 commands can be used without canceling tool length compensation.

Explanation

- G68.2/G68.4 command in tool length compensation

The G68.2/G68.4 command can be executed in tool length compensation.
Absolute coordinates after the G68.2/G68.4 command are based on the position of the tool center point on the feature coordinate system.
When the tilted working plane indexing is executed with the tool or table tilted on the rotation axis, absolute coordinates are based on the position of the tool center point with the position of the rotation axis considered.
Accordingly, machining is allowed even when the tool axis direction is not the Z-axis direction on the feature coordinate system.

Example of operation 1

N10 G69 ;
N20 G54 **G43** H1 X0 Y0 Z0 ;
N30 **G68.2** X_ Y_ Z_ I90. 0J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N40 X100.0 Y0 Z0 ;

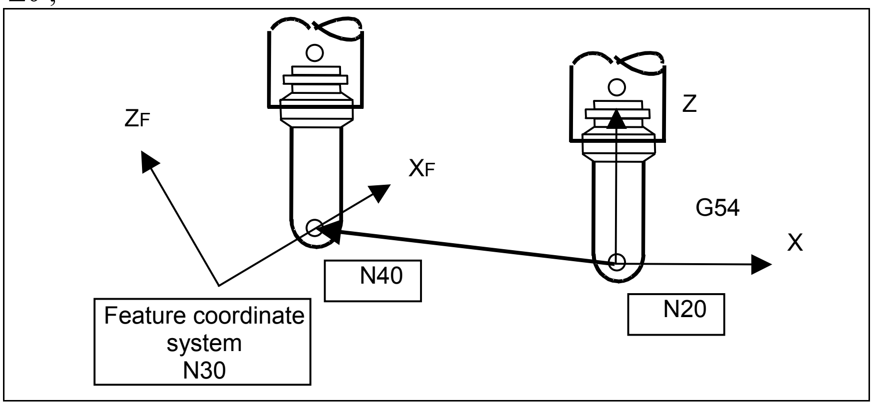


Fig. 5.7.4 (a) Example of operation 1

Example of operation 2

N10 G69 ;
N20 G54 **G43** H1 X0 Y0 Z0 B0 ;
N30 B20.0 ;
N40 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;

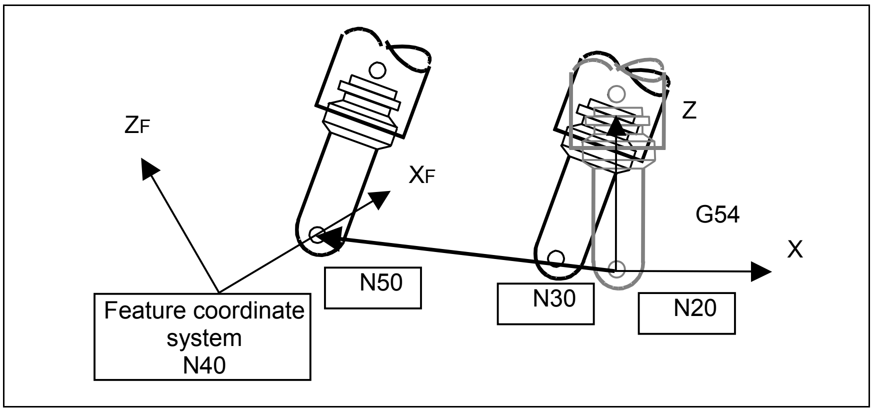


Fig. 5.7.4 (b) Example of operation 2 (tool rotation type)

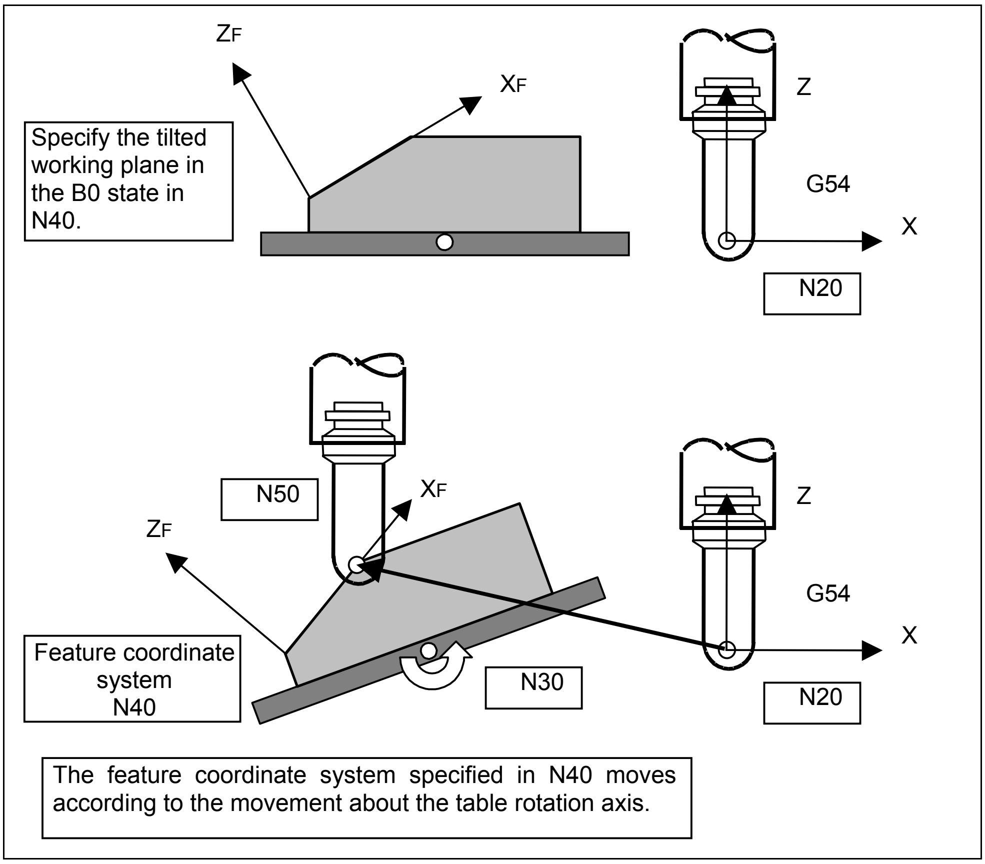


Fig. 5.7.4 (c) Example of operation 2 (table rotation type)

Example of operation 3

N30 **G43** X0 Y0 Z0 B20.0 ;
N40 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;
N60 B-20.0
N70 **G68.4** X_ Y_ Z_ I90.0 J40.0 K-90.0 ;（incremental multiple command: rotation by 40 degrees about the Y-axis）
N80 X100.0 Y0 Z0 ;

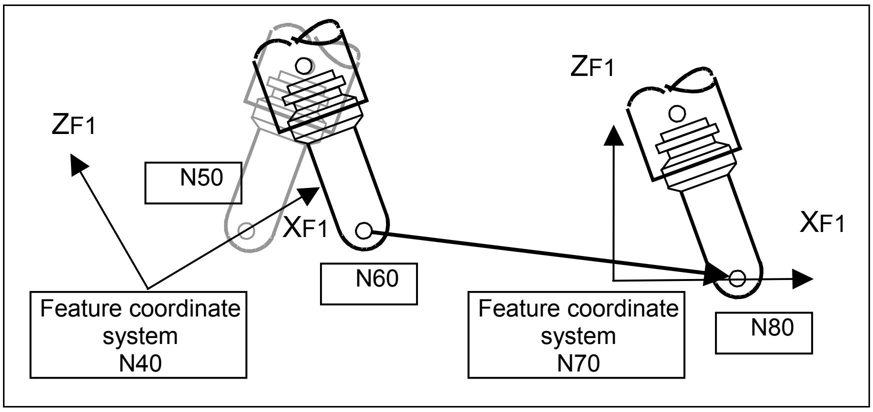


Fig. 5.7.4 (d) Example of operation 3 (tool rotation type)

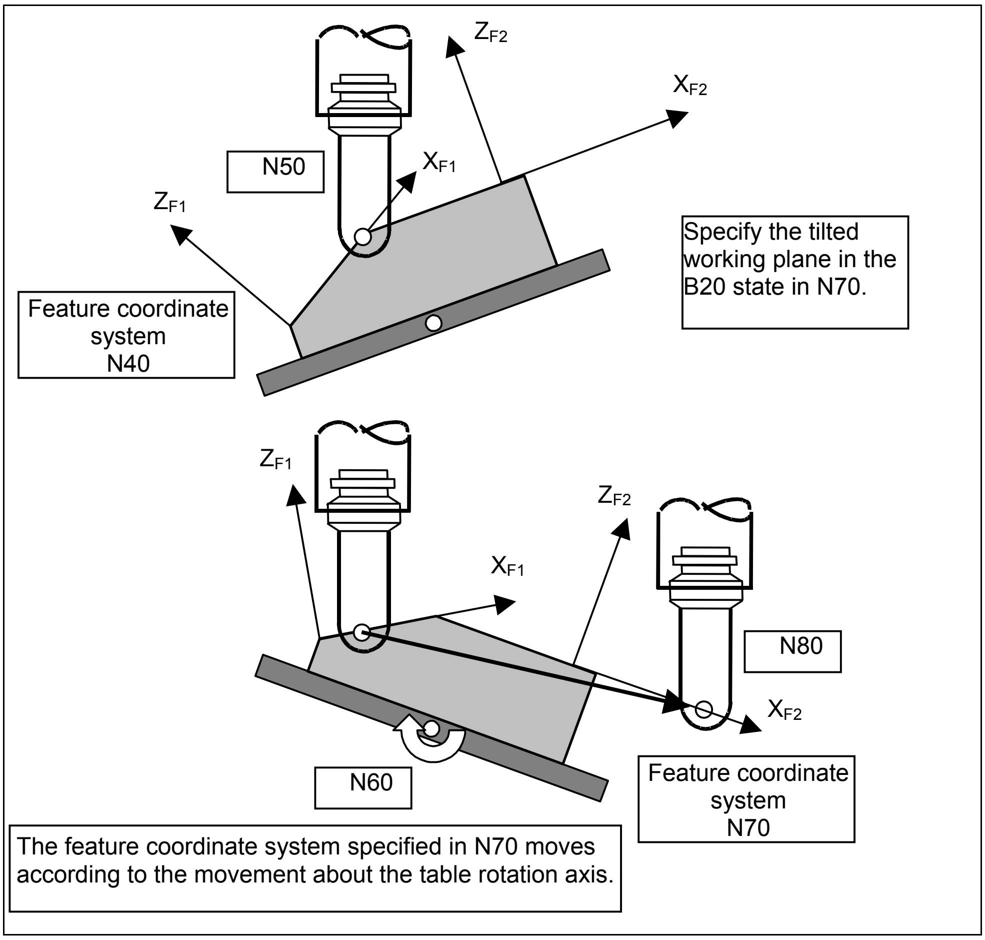


Fig. 5.7.4 (e) Example of operation 3 (table rotation type)

G53.1 command in tool length compensation

The G53.1 command can be executed in tool length compensation.
G53.1 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.
Absolute coordinates after the G53.1 command are based on the position of the tool center point on the feature coordinate system after the G53.1 command is specified.

Example of operation 4

N10 G54 **G43** H1 X_ Y_ Z_ ;
N20 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N30 **G53.1** ;

N40 X100.0 Y0 Z0 ;

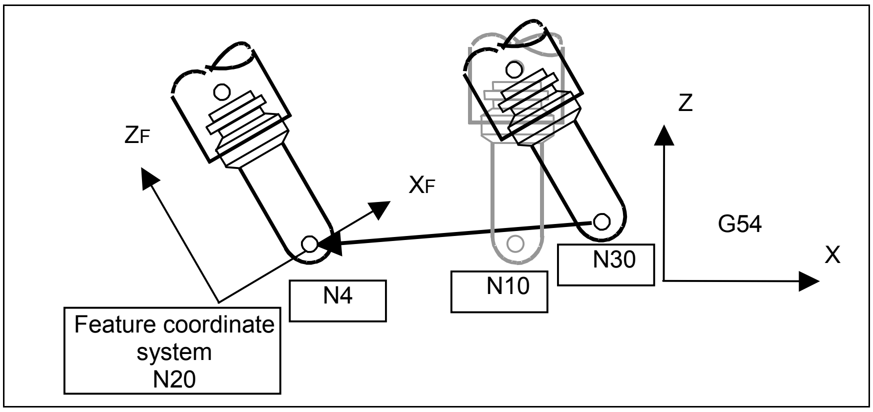


Fig. 5.7.4 (f) Example of operation 4 (tool rotation type)

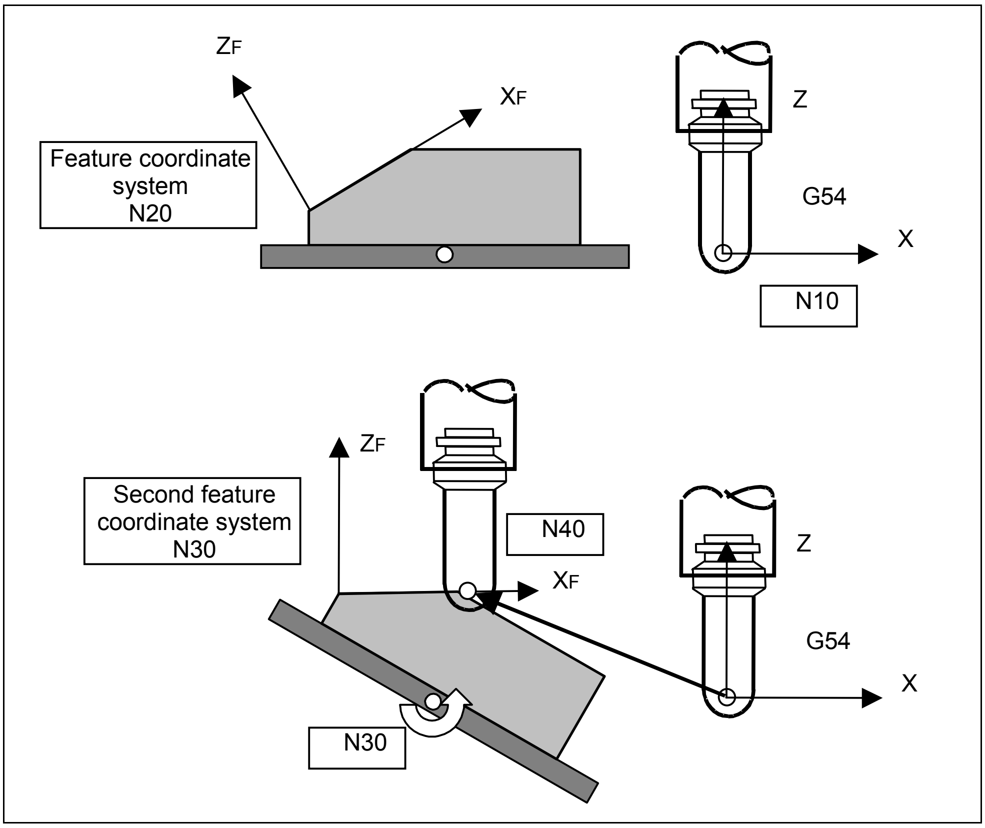


Fig. 5.7.4 (g) Example of operation 4 (table rotation type)

G53.6 command in tool length compensation

The G53.6 command can be executed in tool length compensation.
G53.6 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.

G69 command in tool length compensation

The G69 command can be executed in tool length compensation.
After G69 operation in tool length compensation, the tool length compensation vector represents the Z direction of the workpiece coordinate system regardless of the position of the rotation axis.

5.7.5 Restrictions of Tilted Working Plane Indexing

- Basic restrictions
The restrictions imposed on 3-dimensional coordinate conversion also apply to the tilted working plane indexing.

- Increment system
The same increment system must be used for the basic three axes used by this function.

- Rapid traverse command
Linear rapid traverse (bit 1 (LRP) of parameter No. 1401 = 1) must be set for the rapid traverse command.

- 3-dimensional coordinate system conversion
If an attempt is made to set a new coordinate system by performing 3-dimensional coordinate conversion in a feature coordinate system, an alarm is also raised.

- Positioning in the machine coordinate system
Positioning commands in the machine coordinate system such as G28, G30, and G53 operate in the machine coordinate system rather than the feature coordinate system.

- External mirror image
If an attempt is made to use this function and an external mirror image function simultaneously, this function takes effect before the external mirror image function.

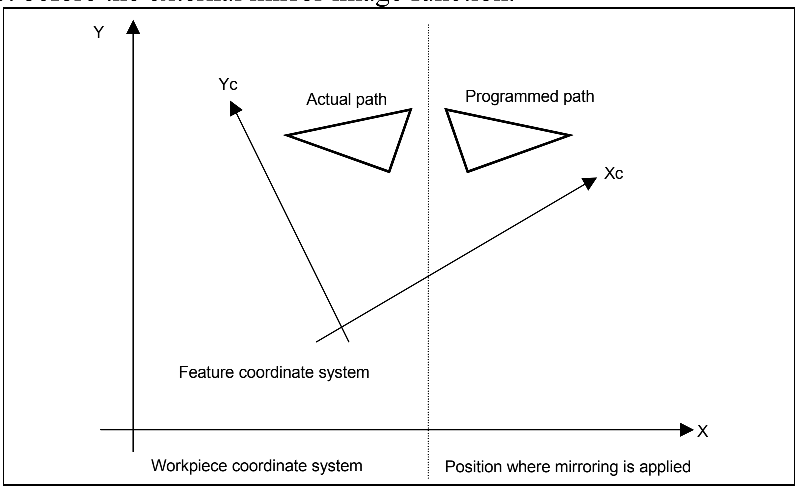


Fig. 5.7.5 (h)

- Tool center point retention type tool axis direction control
Performing a reset during tool center point retention type tool axis direction control results in the operation to be performed when the following parameters are set regardless of whether the reset is performed in the foreground or background. Thus, do not use the background during tool center point retention type tool axis direction control.

(1) Bit 2 (D3R) of parameter No. 5400 = 0: Clears the tilted working plane indexing.
(2) Bit 6 (CLR) of parameter No. 3402 = 1: Clears with a reset.
(3) Bit 0 (C08) of parameter No. 3407 = 0: Clears the G code in group 08.
(4) Bit 7 (C23) of parameter No. 3408 = 0: Clears the G code in group 23.
(5) Bit 7 (CFH) of parameter No. 3409 = 0: Clears the F, H, D, and T codes.
(6) Bit 6 (LVK) of parameter No. 5003 = 0: Clears the tool length compensation vector.

- Relationships with other modal commands

G41, G42, and G40 (cutter compensation), G43, G49 (tool length compensation), G51.1 and G50.1 (programmable mirror image), and canned cycle commands must have nesting relationships with G68.2.
In other words, first issue G68.2 when the modes mentioned above are off, turn the modes on and off, then issue G69.

- Manual reference position return

Manual reference position return in the tilted working plane indexing mode results in alarm PS5324, "REFERENCE RETURN INCOMPLETE". If you want to perform manual reference position return, cancel the tilted working plane indexing mode.

- Hypothetical axis of a table rotation axis

When a table rotation axis is set as a hypothetical axis, tilted working plane indexing is performed on the assumption that the table rotation axis is at 0 degrees.

- Specifiable G codes

The G codes that can be specified in the tilted working plane indexing mode are listed below.
Specifying a G code other than these codes results in alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)".

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- AI contour control, Nano smoothing OFF/ON (G05.1 Q0/Q1/Q3)
- Exact stop(G09)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Automatic return to reference position (G28)
- Movement from reference position (G29)
- 2nd, 3rd and 4th reference position return (G30)
- Skip function (G31)
- Cutter compensation : cancel (G40)
- Tool radius or tool nose radius compensation (G41/G42)
- Tool length compensation + (G43)
- Tool length compensation - (G44)
- Tool length compensation cancel (G49,G49.1)
- Scaling cancel (G50)
- Scaling(G51)
- Programmable mirror image cancel (G50.1)
- Local coordinate system (G52)
- Machine coordinate system setting (G53)
- Tool axis direction control (G53.1)
- Workpiece coordinate system selection (G54 to G59, G54.1)
- Exact stop (G61)
- Automatic corner override (G62)
- Tapping mode (G63)
- Cutting mode (G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Canned cycle for drilling (G73, G74, G76, G80 to G89)
- Absolute programming (G90)

- Incremental programming (G91)
- Inverse time feed (G93)
- Constant surface speed control (G96)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Programmable mirror image (G51.1)
- Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69)
- Feed per minute (G94)
- Feed per revolution (G95)

- Modal G codes that allow specification of a tilted working plane indexing

A tilted working plane indexing can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying the tilted working plane indexing results in alarm PS5462:

- Positioning (G00)
- Linear interpolation (G01)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Stored stroke check function (G22/G23)
- Cutter compensation : cancel (G40)
- Tool length compensation cancel (G49,G49.1)
- Scaling cancel (G50)
- Programmable mirror image cancel (G50.1)
- Workpiece coordinate system selection (G54 to G59, G54.1)
- Exact stop mode (G61)
- Automatic corner override (G62)
- Tapping mode (G63)
- Cutting mode (G64)
- Macro modal call A/B cancel (G67)
- Canned cycle cancel (G80)
- Absolute programming (G90)
- Incremental programming (G91)
- Inverse time feed (G93)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69)
- Feed per minute (G94)
- Feed per revolution (G95)

5.8 FIGURE COPYING (G72.1, G72.2)

Machining can be repeated after moving or rotating the figure using a subprogram.

NOTE
This function is an optional function.

Format

- Rotational copying

Xp-Yp plane (specified by G17) : **G72.1 P_ L_ Xp_Yp_R_ ;**
Zp-Xp plane (specified by G18) : **G72.1 P_ L_ Zp_Xp_R_ ;**
Yp-Zp plane (specified by G19) : **G72.1 P_ L_ Yp_Zp_R _;**

P :Subprogram number
L :Number of times the operation is repeated
Xp :Center of rotation on the Xp axis (Xp: X-axis or an axis parallel to the X-axis)
Yp :Center of rotation on the Yp axis (Yp: Y-axis or an axis parallel to the Y-axis)
Zp :Center of rotation on the Zp axis (Zp: Z-axis or an axis parallel to the Z-axis)
R :Angular displacement (A positive value indicates a counterclockwise angular displacement. Specify an incremental value.)

Specify a plane selection command (G17, G18, or G19) to select the plane on which the rotational copying is made.

- Linear copying

Xp-Yp plane (specified by G17) : **G72.2 P_ L_ I_ J_ ;**
Zp-Xp plane (specified by G18) : **G72.2 P_ L_ K_ I_ ;**
Yp-Zp plane (specified by G19) : **G72.2 P_ L_ J_ K_;**

P :Subprogram number
L :Number of times the operation is repeated
I :Shift along the Xp axis
J :Shift along the Yp axis
K :Shift along the Zp axis

Specify a plane selection command (G17, G18, or G19) to select the plane on which the linear copying is made.

Explanation

- First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a rotational or linear copying. If the first block contains only the program number such as O1234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1, 2, 3, ...) copying.
<u>Specify the first move command in the absolute mode.</u>

(Example of an incorrect program)

```
O1234 ;
G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

(Example of a correct program)

```
O1000 G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

- Combination of rotational and linear copying

The linear copying command can be specified in a subprogram for a rotational copying. Also, the rotational copying command can be specified in a subprogram for a linear copying.

- Subprogram call

In a subprogram for rotational or linear copying, M98 for calling another subprogram or G65 for calling a macro can be specified.

- Specifying the center of rotation

The center of rotation specified with G72.1 is processed as an absolute position even in the incremental mode.

- Specifying address

In a block with G72.1, addresses other than P, L, Xp, Yp, Zp, or R are ignored. The subprogram number (P), coordinates of the center of rotation (Xp, Yp, Zp), and angular displacement (R) must be specified.
In a block with G72.2, addresses other than P, L, I, J, or K are ignored.
The subprogram number (P) and shift (I, J, K) must be specified.

- Address P

If the subprogram number specified with P is not found, alarm PS0310, “FILE NOT FOUND” occurs. If P is not specified, alarm PS0076, “PROGRAM NOT FOUND” occurs.

- Address L

If L is omitted, the repetition count is assumed to be 1 and the subprogram is called only once.

- Increment in angular displacement or shift

In a block with G72.1, an increment in angular displacement is specified with address R. The angular displacement of the figure made by the n-th rotation is calculated as follows : R × (n - 1).
In a block with G72.2, an increment in shift is specified with addresses I, J, and K. The shift of the figure made by the n-th movement is calculated as follows : (Programmed shift) × (n - 1).

- Nesting level of a subprogram

If a subprogram is called by G72.1 or G72.2, the nesting level is increased by one in the same manner as when M98 is specified.

- Block end position

The coordinates of a figure moved rotationally or linearly (block end position) can be read from #5001 and subsequent system variables of the custom macro of rotational or linear copying.

- Disagreement between end point and start point

If the end point of the figure made by the n-th copy does not agree with the start point of the figure to be made by the next (n + 1) copy, the figure is moved from the end point to the start point, then copying is started. (Generally, this disagreement occurs if an incorrect angular displacement or shift is specified.)

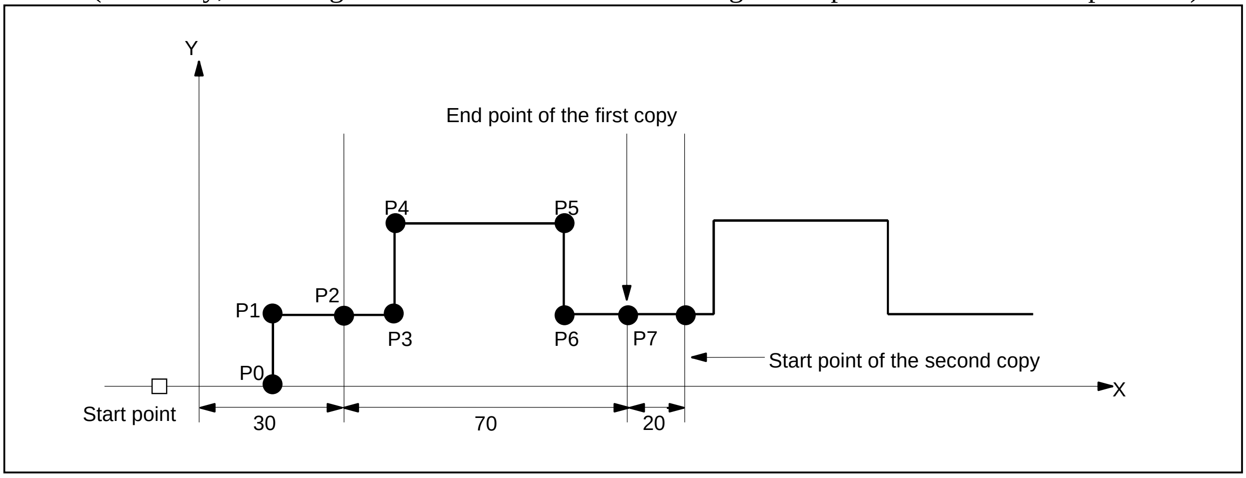


Fig. 5.8 (a)

```
Main program
O1000 ;
N10 G92 X-20.0 Y0.0 ;
N20 G00 G90 X0.0 Y0.0 ;
N30 G01 X20.0 Y0.0 F10 ;          (P0)
N40 Y20.0 ;                       (P1)
N50 X30.0 ;                       (P2)
N60 G72.2 P2000 L3 I90.0 J0.0 ;
                                  Although a shift of 70 mm was required, I90.0 was
                                  specified instead of I70.0. Since an incorrect shift was
                                  specified, the end point of the figure made by the n-th
                                  copy disagrees with the start point of the figure to be
                                  made by the next (n + 1) copy.

Subprogram
O2000 G90 G01 X40.0 ;             (P3)
N100 Y40.0 ;                      (P4)
N200 X80.0 ;                      P5)
N300Y20.0 ;                       (P6)
N400 X100.0 ;                     (P7)
N500 M99;
```

Limitation

- Specifying two or more commands to copy a figure

G72.1 cannot be specified more than once in a subprogram for making a rotational copying (If this is attempted, alarm PS0160, “MISMATCH WAITING M-CODE” will occur). G72.2 cannot be specified more than once in a subprogram for making a linear copying (If this is attempted, alarm PS0161, “ILLEGAL P OF WAITING M-CODE” will occur).

- Commands that must not be specified
Within a program that performs a rotational or linear copying, the following must not be specified:
- Command for changing the selected plane (G17 to G19)
- Command for specifying polar coordinates
- Reference position return command
- Coordinate system rotation, scaling, programmable mirror image
  The command for rotational or linear copying can be specified after a command for coordinate system rotation, scaling, or programmable mirror image is executed.

- Modes that must not be selected
Figure copying cannot be specified in the following modes.
- Tool offset
- Tilted working plane indexing command
- 3-dimensional coordinate system conversion

- Unit system
The two axes of the plane for copying a figure must have an identical unit system.

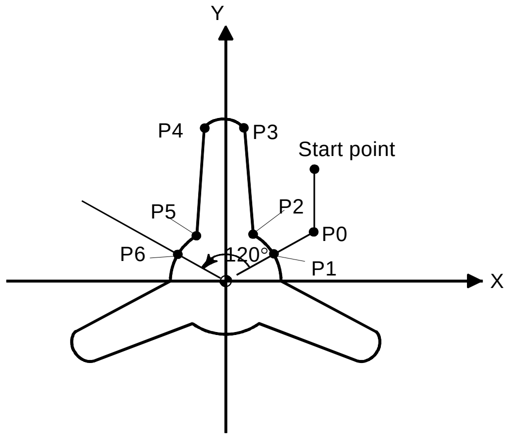
- Single block
Single-block stops are not performed in a block with G72.1 or G72.2.

- Specifying tool radius compensation and the workpiece coordinate system
In a subprogram for copying a figure, the G code for tool radius / tool nose radius compensation or compensation amount (H or D code) cannot be changed. G92 and G54 to G59 cannot be changed either. Those codes must be specified before figure copying is started.

- Copy axially excluding the axis direction of plane selection
Rotational copying and Linear copying for the copy axially excluding the axis direction of plane selection cannot be executed. At the Rotational copying, the rotation center axis command excluding the axis direction of plane selection (for example, Z command in Xp-Yp plane (specified by G17)) is ignored. At the Linear copying, the shift along the axis excluding the axis direction of plane selection (for example, K command in Xp-Yp plane (specified by G17)) is ignored.

Example

- Rotational copying

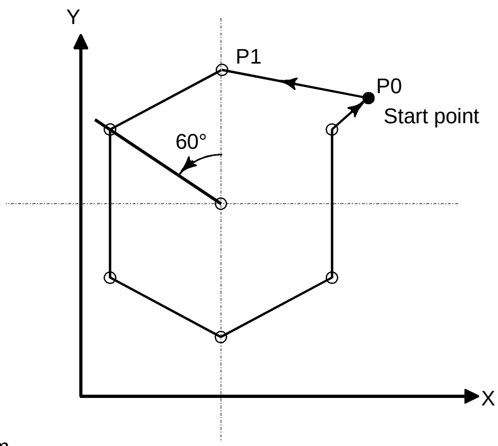



```
Main program
  O1000 ;
  N10 G92 X40.0 Y50.0 ;
  N20 G00 G90 X_ Y_ ;                    (P0)
  N30 G01 G17 G41 X_ Y_ D01 F10 ;        (P1)
  N40 G72.1 P2000 L3 X0 Y0 R120.0 ;
  N50 G40 G01 X_ Y_ I_ J_ ;              (P0)
  N60 G00 X40.0 Y50.0 ;
  N70 M30 ;

Sub program
  O2000 G03 X_ Y_ R30.0 ;                (P2)
  N100 G01 X_ Y_ ;                       (P3)
  N200 G03 X_ Y_ R10.0 ;                 (P4)
  N300 G01 X_ Y_ ;                       (P5)
  N400 G03 X_ Y_ R30.0 ;                 (P6)
  N500 M99;
```

Fig. 5.8 (b)

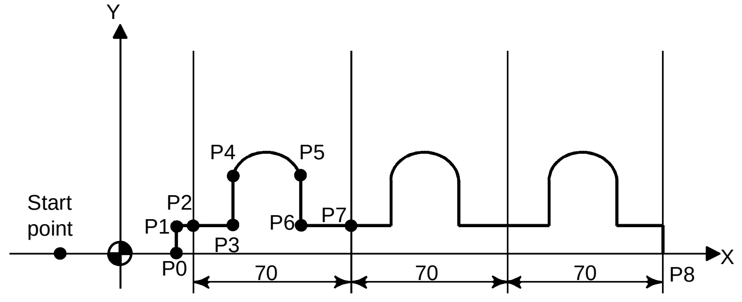
- Rotational copying (spot boring)




```
Main program
    O3000 ;
    N10 G92 G17 X80.0 Y50.0 ;            (P0)
    N20 G72.1 P4000 L6 X0 Y0 R60.0 ;
    N30 G80 G00 X80.0 Y50.0 ;            (P0)
    N40 M30 ;
Subprogram
    O4000 N100 G90 G81 X_ Y_ R_ Z_ F_ ;  (P1)
    N200 M99 ;
```

Fig. 5.8 (c)

- Linear copying

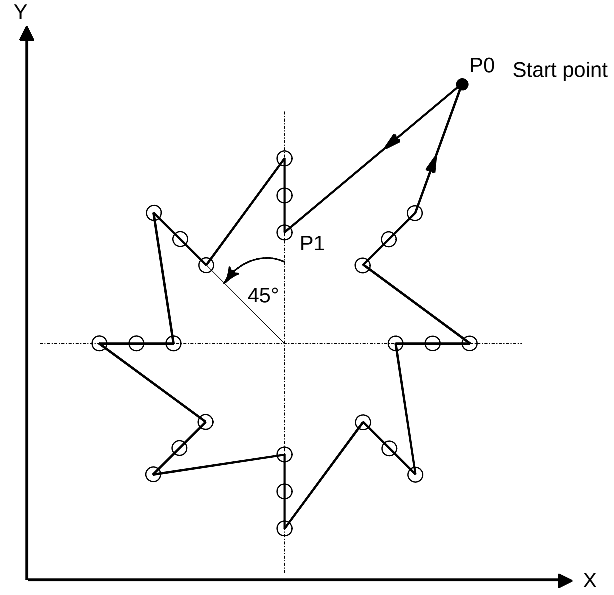


Main program
```
O1000 ;
N10 G92 X-20.0 Y0 ;
N20 G00 G90 X0 Y0 ;
N30 G01 G17 G41 X20. Y0 D01 F10 ;    (P0)
N40 Y20. ;                           (P1)
N50 X30. ;                           (P2)
N60 G72.2 P2000 L3 I70.0 J0 ;
N70 X_ Y_ ;                          (P8)
N80 X0 ;
N90 G00 G40 X-20.0 Y0 ;
N100 M30 ;
```

Subprogram
```
O2000 G90 G01 X_ ;                   (P3)
N100 Y_ ;                            (P4)
N200 G02 X_ I_ ;                     (P5)
N300 G01 Y_ ;                        (P6)
N400 X_ ;                            (P7)
N500 M99 ;
```

Fig. 5.8 (d)

- Combination of rotational copying and linear copying (bolt hole circle)



Main program

```
O1000 ;
N10 G92 G17 X100.0 Y80.0 ;            (P0)
N20 G72.1 P2000 X0 Y0 L8 R45.0 ;
N30 G80 G00 X100.0 Y80.0 ;            (P0)
N40 M30 ;
```

Subprogram (rotational copy)

```
O2000 N100 G72.2 P3000 I0 J_ L3 ;
N200 M99 ;
```

Subprogram (linear copy)

```
O3000 N110 G90 G81 X_ Y_ R_ Z_ F_ ;   (P1)
N210 M99 ;
```

Fig. 5.8 (e)

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).

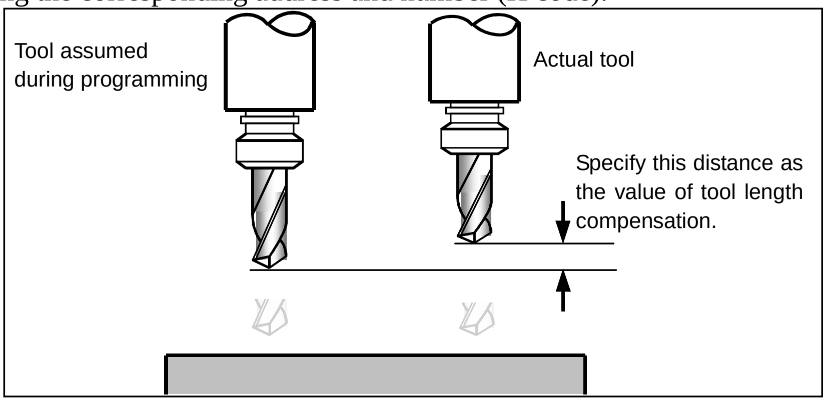


Fig. 6.1 (a) Tool length compensation

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 0 (TLC) and 1 (TLB) 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, “NO AXES COMMANDED IN G43/G44” is issued. If two or more axes are specified in the same block, the alarm PS0336, “TOOL COMPENSATION COMMANDED MORE TWO AXES” 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.

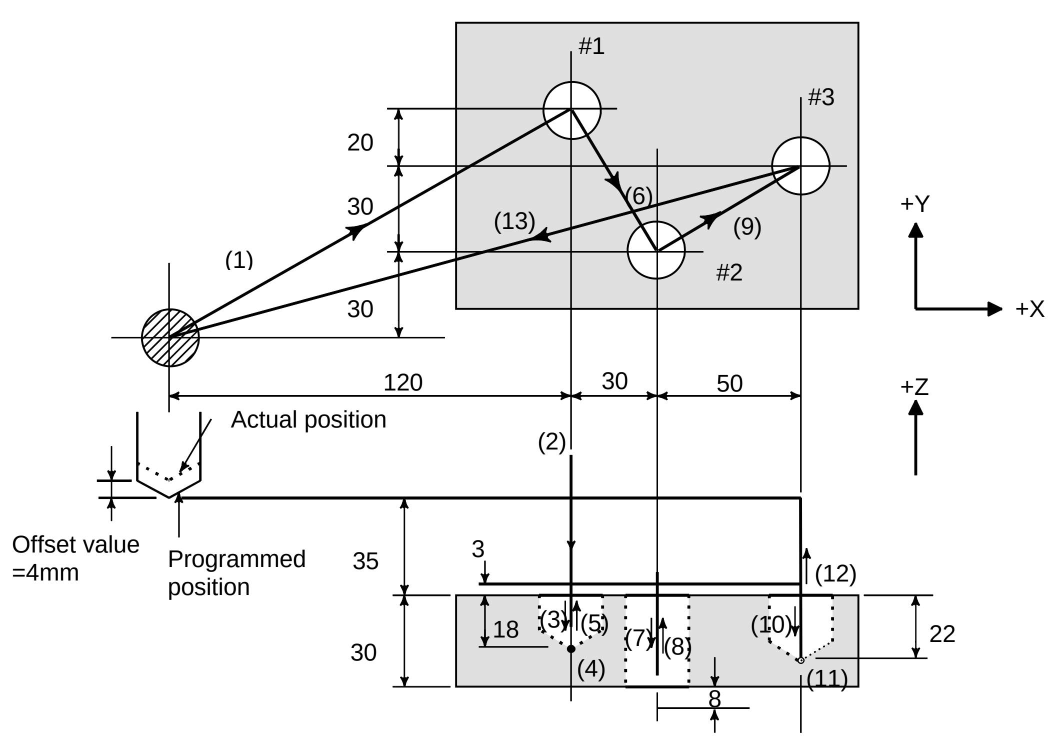
NOTE

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 If tool length compensation and 3-dimensional coordinate system conversion are canceled by a reset with 3-dimensional coordinate system conversion performed during tool length compensation, the direction of tool length compensation vector cancellation becomes incorrect. By setting bit 6 (LVK) of parameter No. 5003 to 1 and setting bit 2 (D3R) of parameter No. 5400 to 1, ensure that the tool length compensation vector and 3-dimensional coordinate system conversion are not canceled by a reset.

Example)
```
G43 H1 ;
G68 X_ Y_ Z_ I_ J_ K_ R_ ;
  :
  :   Ensure that tool length compensation and three-dimensional
  :   coordinate conversion are not canceled by a reset in this range.
G69 ;
G49 ;
```

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. For details, see “Notes” in Subsection 7.2.1, “Setting a Workpiece Coordinate System.”

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.
As mentioned in "WARNINGS AND CAUTIONS RELATED TO PROGRAMMING" on page S-3 of this manual, it is recommended to cancel tool length compensation mode before executing the G53, G28, and G30 commands.

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	Bit 6 (EVO) of parameter No. 5001	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**
> 1 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.
> 2 In the block in which the tool length compensation vector is restored, do not execute commands other than positioning with G00 or G01, G04, and a single block with EOB.

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 bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400. If no move command is specified together with the G43, G44, or G49 command, the tool will not move along the axis. If a move command is specified together with the G43, G44, or G49 command, the coordinate system will be shifted first, then the tool will move along the axis.

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

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

Format

- Tool length compensation A

G43 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the + side.
G44 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the - side.
G43 (or G44) : + (or -) side offset at which to start tool length compensation
H_ : Address specifying the tool length compensation value

- Tool length compensation B

G17 G43 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the + side.
G17 G44 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the - side.
G18 G43 Y_H_;
Shifts the coordinate system along the X axis by the compensation value, to the + side.
G18 G44 Y_H_;
Shifts the coordinate system along the X axis by the compensation value, to the - side.
G19 G43 X_H_;
Shifts the coordinate system along the Y axis by the compensation value, to the + side.
G19 G44 X_H_;
Shifts the coordinate system along the Y axis by the compensation value, to the - side.

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

- Tool length compensation C

G43 α_H_;
Shifts the coordinate system along a specified axis by the compensation value, to the + side.
G44 α_H_;
Shifts the coordinate system along a specified axis by the compensation value, to the - side.

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

- Tool length compensation cancel

G49; or H0; Tool length compensation cancel

G49 (or H0)	:	Tool length compensation cancel

Explanation

- Offset direction

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

- Specifying a tool length compensation value

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

- Compensation axis

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

- Specifying offset on two or more axes

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

To perform offset on X and Y axes

G19 G43 H_; Performs offset on the X axis.
G18 G43 H_; Performs offset on the Y axis.

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

- Tool length compensation cancel

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

Limitation

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

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

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

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

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

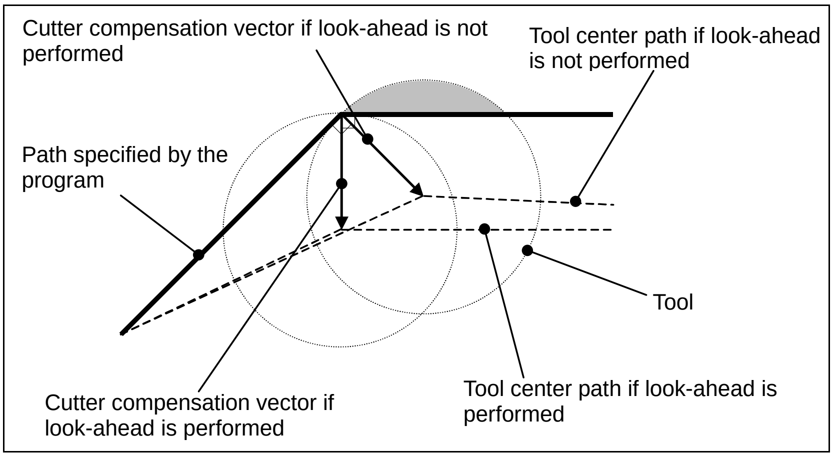
Example in which overcutting occurs in cutter compensation)

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

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

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

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



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

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

Example in which no overcutting occurs in cutter compensation (recommended)
Before cutter compensation mode, start tool length compensation.

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

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

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

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

- Bit 1 (MOF) of parameter No. 5000 = 0

The tool moves along the axis by the change in tool length compensation.

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

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

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

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

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

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

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

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

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

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

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

```
    :
G40 G49 G00 G90 X0 Y0 Z100. ;
```

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

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

⚠ **CAUTION**

1. Specifying tool length compensation (a shift type) first and then executing an incremental programming causes the tool length compensation value to be reflected in the coordinates only, not in the travel distance of the machine; executing an absolute programming causes the tool length compensation value to be reflected in both the movement of the machine and the coordinates.
2. If a programmable mirror image is effective, the tool length compensation is applied in the specified direction.
3. No scaling magnification is applied to the tool length compensation value.
4. No coordinate system rotation is applied to the tool length compensation value. Tool length compensation is effective in the direction in which the offset is applied.
5. 3-dimensional coordinate conversion is applied to tool length compensation. If tool length compensation is made effective to multiple axes, the tool length compensation must be canceled for one axis at a time.

⚠ **CAUTION**

6 With the WINDOW command, changing bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400 during automatic operation does not cause the tool length compensation type to be changed.

7 If offset has been performed on two or more axes with tool length compensation B, a G49 command causes the offset to be canceled on all axes; H0 causes the offset to be canceled only on the axis vertical to the specified plane.

8 If the tool length compensation value is changed by changing the offset number, this simply means that the value is replaced by a new tool length compensation value; it does not mean that a new tool length compensation value is added to the old tool length compensation.

9 If reference position return (G28, G30, or G30.1) has been specified, tool length compensation is canceled for the axis specified at the time of positioning on the reference point; however, tool length compensation is not canceled for an un-specified axis. If reference position return has been specified in the same block as that containing tool length compensation cancel (G49), tool length compensation is canceled for both the specified and un-specified axes at the time of positioning on the mid-point.

10 With a machine coordinate system command (G53), tool length compensation is canceled for the axis specified at the time of positioning on the specified point.

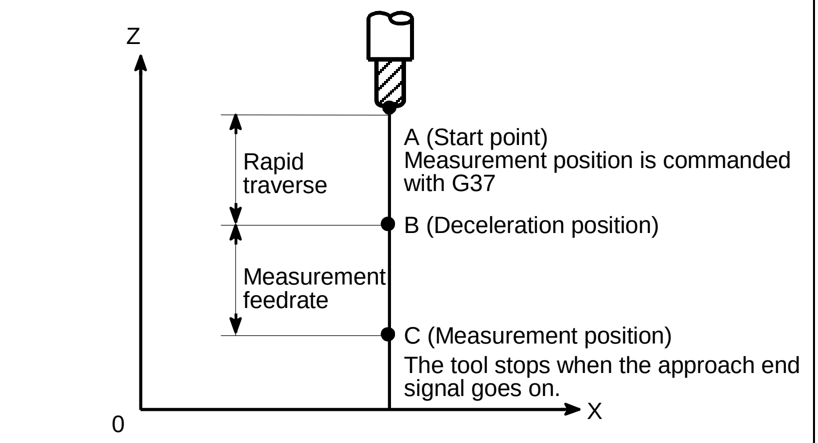
11 The tool length compensation vector canceled by specifying G53, G28, G30, or G30.1 during tool length compensation is restored as described below:

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

12 When a tool length compensation shift type is used, if the start or cancellation of a tool length compensation or other command is specified tool radius · tool nose radius compensation mode, look-ahead is not performed. As a result, overcutting or undercutting may occur before or after the block in which the start or cancellation is specified. Thus, specify the start and cancellation of tool length compensation before the entry to tool radius · tool nose radius compensation mode or at a location where machining is not affected.

6.3 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

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

Explanation

- Setting the workpiece coordinate system

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

- Specifying G37

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

- Changing the offset value

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

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

These offset values can be manually changed from MDI.

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

$$\text{Offset value} = -\text{(Tool geometry offset)} \begin{cases} +\text{(Current offset value)} & : \text{G43} \\ -\text{(Current offset value)} & : \text{G44} \\ 0 & : \text{G49} \end{cases}$$
$$+ [\text{(Coordinates of the position at which the tool reaches for measurement)} - \text{(Coordinates specified by G37)}]$$

- Alarm

When automatic tool length measurement is executed, the tool moves as shown in Fig. 6.3 (b). If the approach end signal turns 1 while the tool is traveling from point B to point C, an alarm occurs. Unless the approach end signal turns 1 before the tool reaches point F, the same alarm occurs. The alarm number is PS0080.

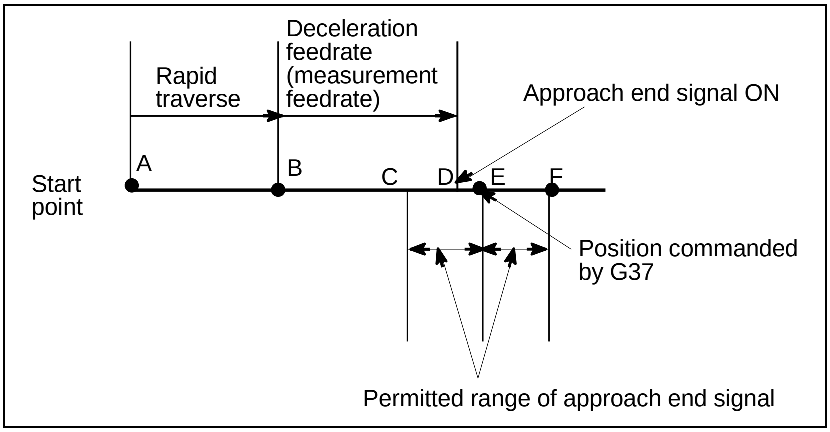


Fig. 6.3 (b) Tool movement to the measurement position

> ⚠ **WARNING**
> 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.

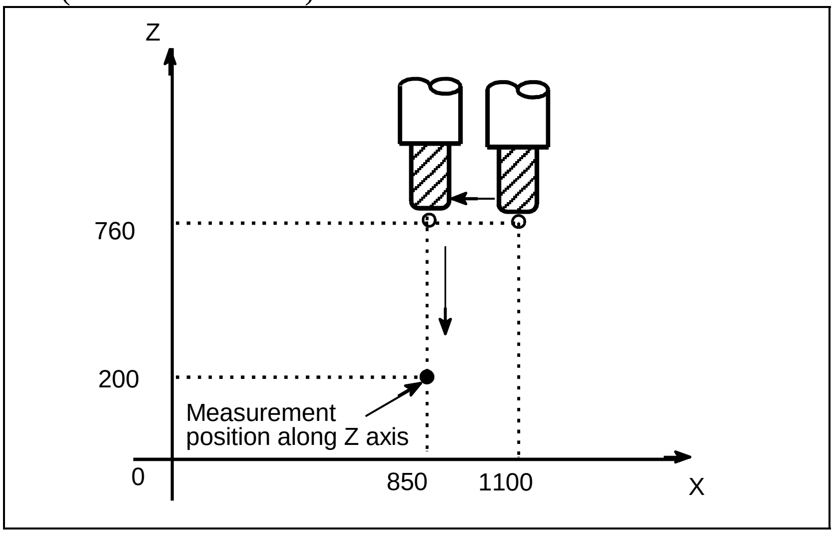
NOTE

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

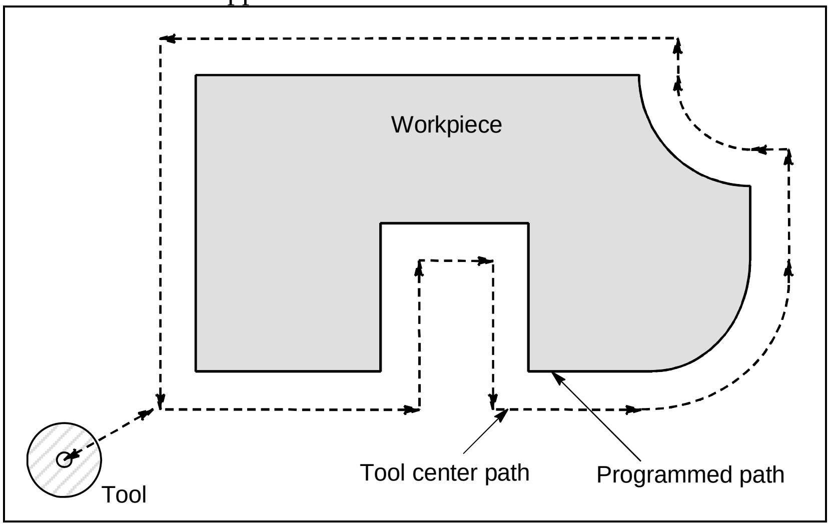
G92 Z760.0 X1100.0 ;	Sets a workpiece coordinate system with respect to the programmed absolute zero point.
G00 G90 X850.0 ;	Moves the tool to X850.0. That is the tool is moved to a position that is a specified distance from the measurement position along the Z-axis.
H01 ;	Specifies offset number 1.
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 TO G48)

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



Format

G45 IP_ D_ ; Increase the travel distance by the tool offset value
G46 IP_ D_ ; Decrease the travel distance by the tool offset value
G47 IP_ D_ ; Increase the travel distance by twice the tool offset value
G48 IP_ D_ ; Decrease the travel distance by twice the tool offset value

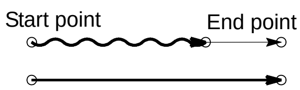
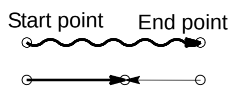
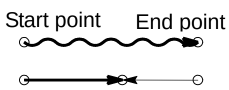
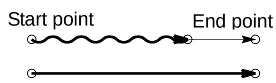
G45 to 48 : One-shot G code for increasing or decreasing the travel distance
IP_ : Command for moving the tool
D_ Code for specifying the tool offset value

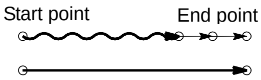
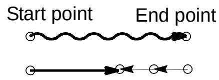
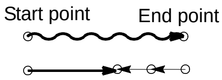
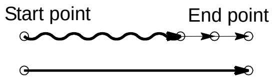
Explanation

- Increase and decrease

As shown in Table 6.4 (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		
G46		

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

Programmed movement distance
Tool offset value
Actual movement position

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

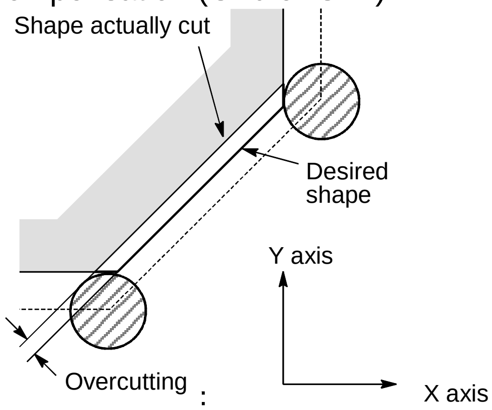
- Tool offset value

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

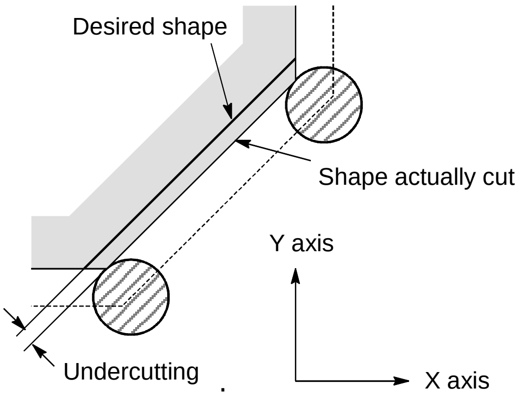
D0 always indicates a tool offset value of zero.

⚠ CAUTION

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




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

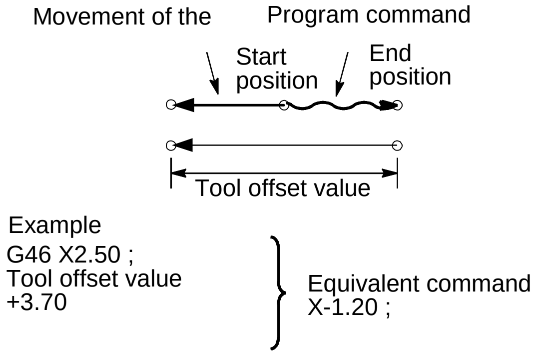



```
:
G01 G45 X_ F_ D_;
           X_ Y_ ;
G45 Y_ ;
:
```

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

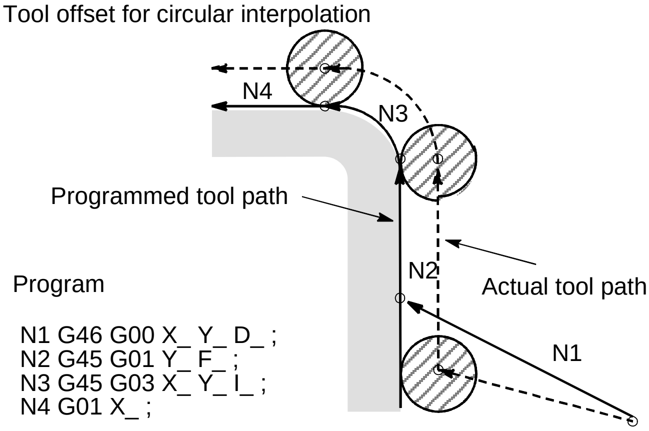
NOTE

1 When the specified direction is reversed by decrease, 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

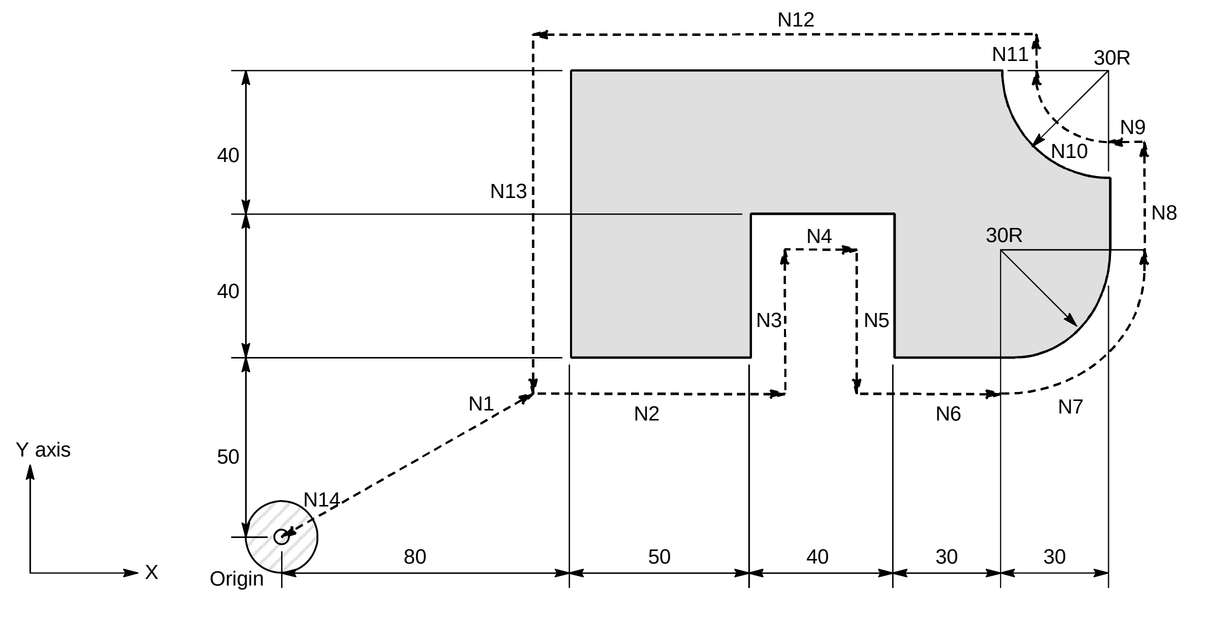


3 D code should be used in tool offset mode.

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.

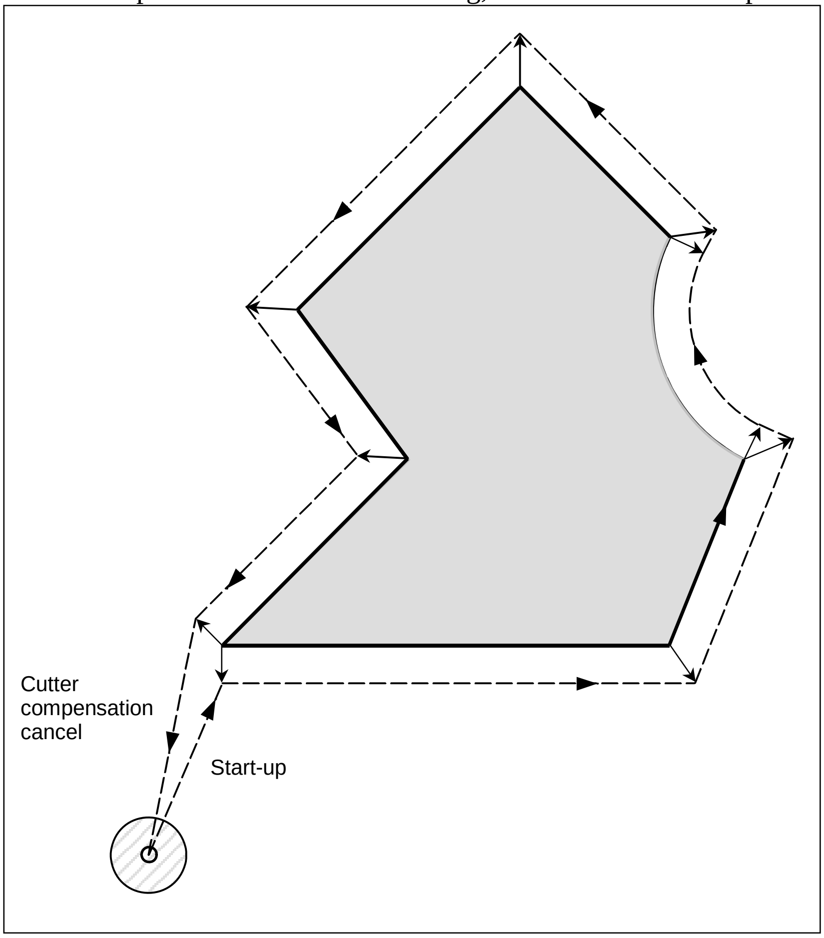


Fig. 6.5 (a) Outline of cutter compensation

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)

- Cutter compensation cancel (offset mode cancel)

G40 IP_;

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

- Selection of the offset plane

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

Explanation

- Offset cancel mode

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

- Start-up

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

- Offset mode

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

- Offset mode cancel

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

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

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

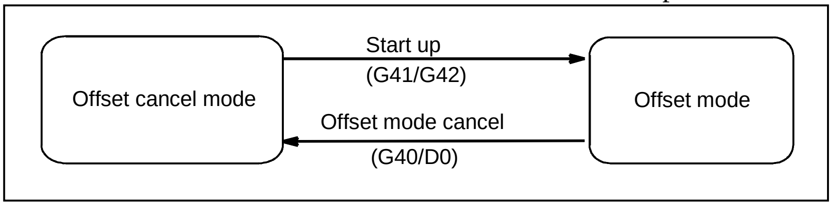


Fig. 6.5 (b) Changing the offset mode

- 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.

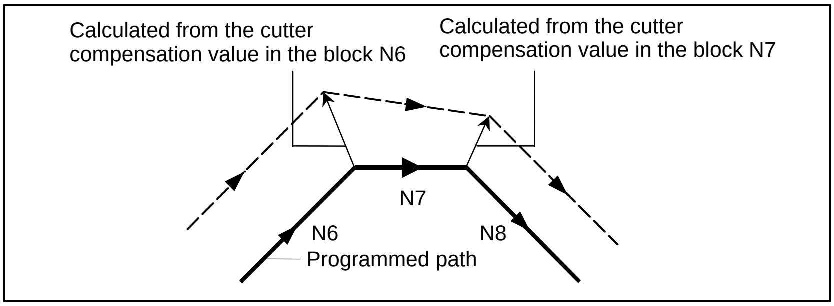


Fig. 6.5 (c) Changing the cutter compensation value

- 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.)

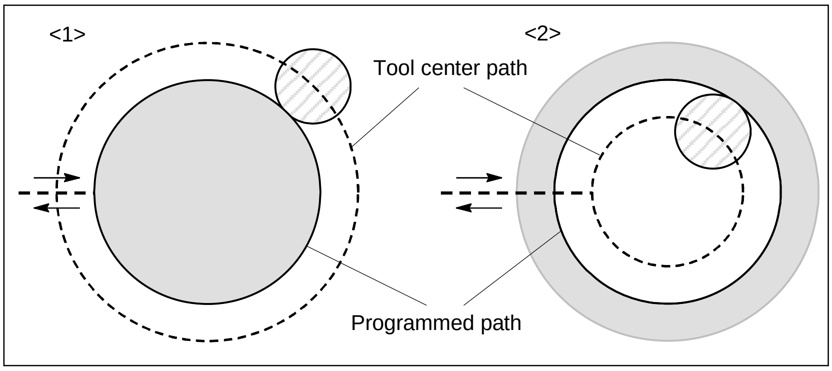


Fig. 6.5 (d) Tool center paths when positive and negative cutter compensation values are specified

- Cutter compensation value setting

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

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

- Valid compensation value range

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

Valid compensation range (metric input)

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

Valid compensation range (inch input)

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

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

- Offset vector

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

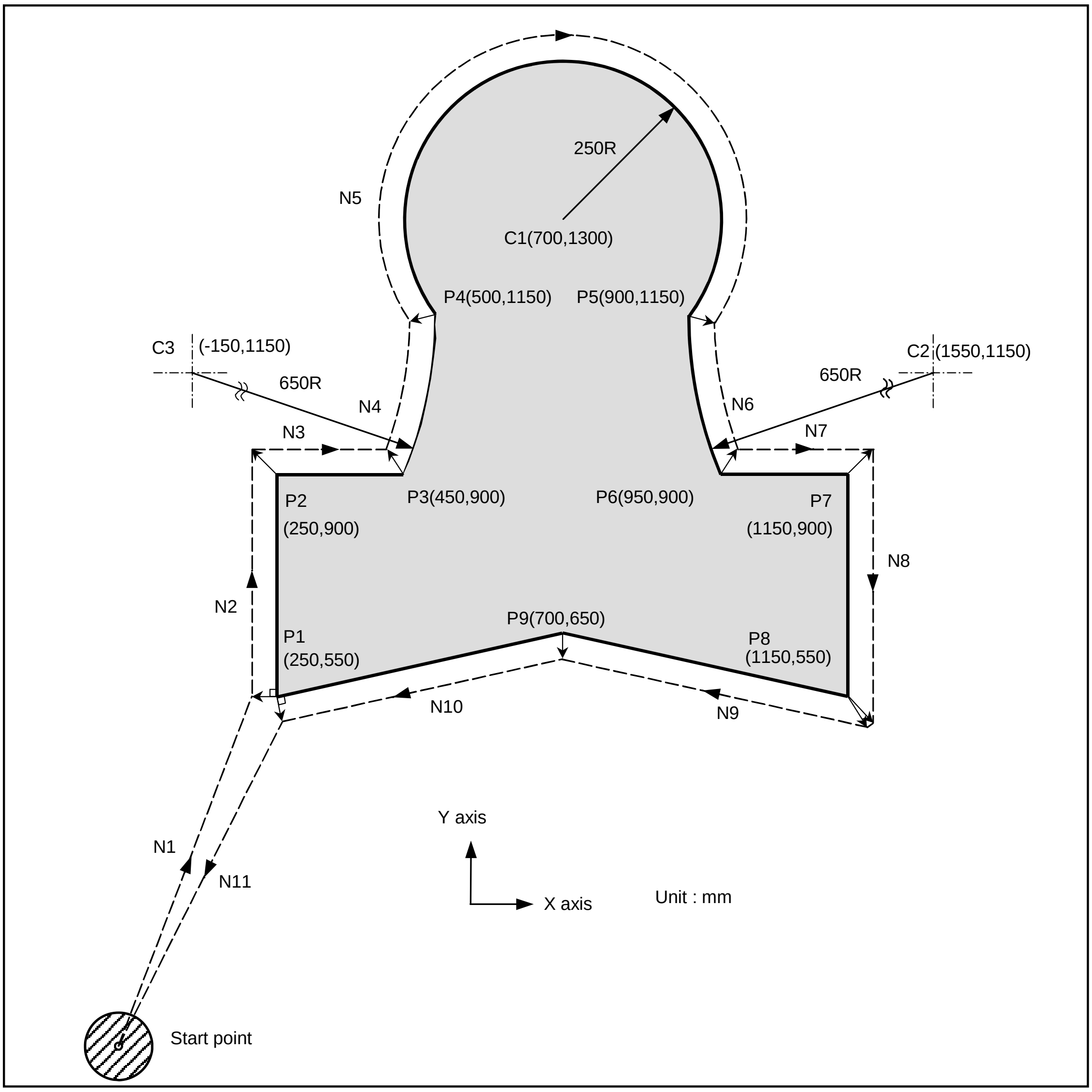
- Specifying a cutter compensation value

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

- Plane selection and vector

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

Example



	G17 G92 X0.0 Y0.0 Z0.0 ;	Specifies absolute coordinates. The tool is positioned at the start point (X0, Y0, Z0).
N1	G90 G00 G41 D07 X250.0 Y550.0 ;	Starts cutter compensation (start-up). The tool is shifted to the left of the programmed path by the distance specified in D07. In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15 mm).
N2	G01 Y900.0 F150 ;	Specifies machining from P1 to P2.
N3	X450.0 ;	Specifies machining from P2 to P3.
N4	G03 X500.0 Y1150.0 R650.0 ;	Specifies machining from P3 to P4.
N5	G02 X900.0 R-250.0 ;	Specifies machining from P4 to P5.
N6	G03 X950.0 Y900.0 R650.0 ;	Specifies machining from P5 to P6.
N7	G01 X1150.0 ;	Specifies machining from P6 to P7.
N8	Y550.0 ;	Specifies machining from P7 to P8.
N9	X700.0 Y650.0 ;	Specifies machining from P8 to P9.
N10	X250.0 Y550.0 ;	Specifies machining from P9 to P1.
N11	G00 G40 X0 Y0.0 ;	Cancels the offset mode. The tool is returned to the start point (X0.0, Y0.0, Z0.0).

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

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

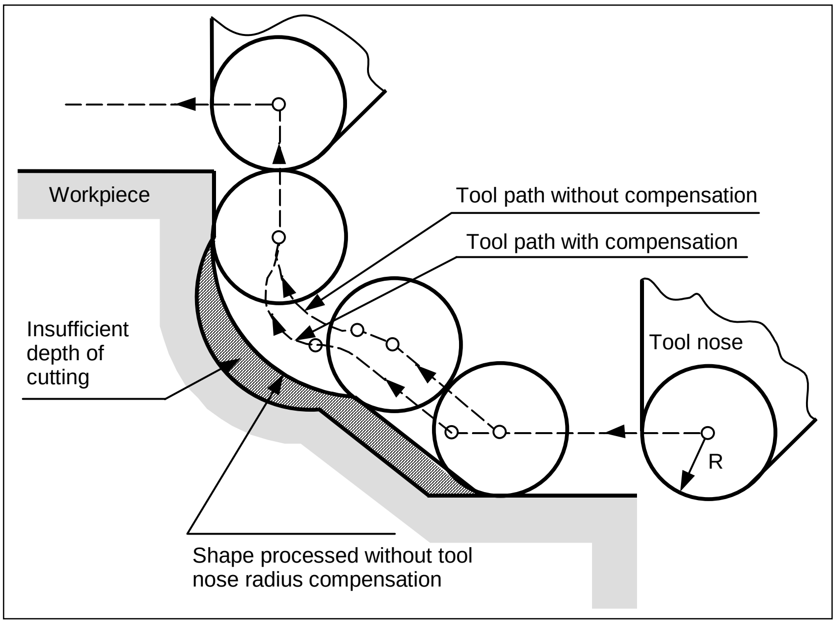


Fig. 6.6 (a) Tool path of tool nose radius compensation

6.6.1 Imaginary Tool Nose

The tool nose at position A in Fig. 6.6.1 (a) does not actually exist.
The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start point than the imaginary tool nose.
Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.
The position relationship when the tool is set to the start point is shown in Fig. 6.6.1 (a).

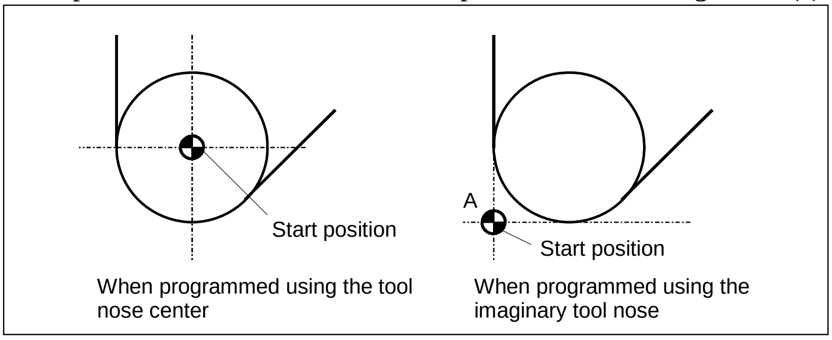


Fig. 6.6.1 (a) Tool nose radius center and imaginary tool nose

⚠ **CAUTION**

In a machine with reference positions, a standard position like the turret center can be placed over the start point. The distance from this standard position to the tool nose radius center or the imaginary tool nose is compensated by the tool length compensation function.

Setting the distance from the standard position to the tool nose radius center as the offset value is the same as placing the tool nose radius center over the start point, while setting the distance from the standard position to the imaginary tool nose is the same as placing the imaginary tool nose over the standard position.

To set the offset value, it is usually easier to measure the distance from the standard position to the imaginary tool nose than from the standard position to the tool nose radius center.

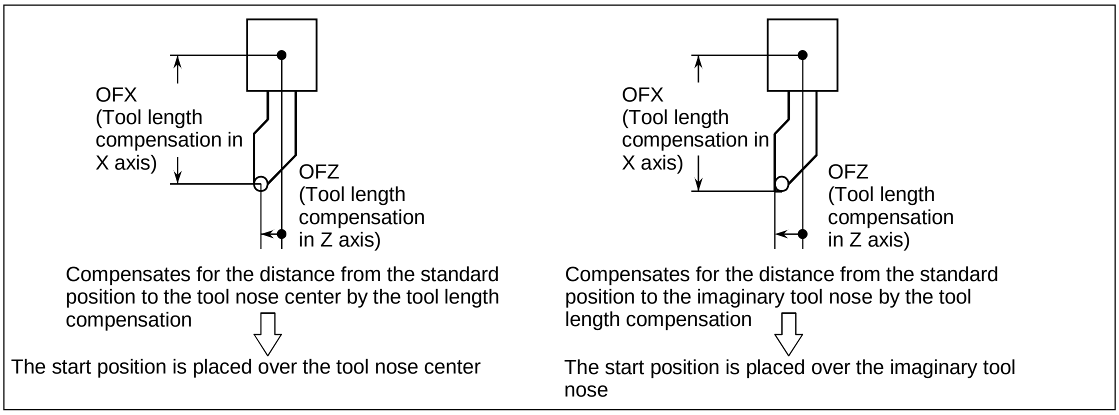


Fig. 6.6.1 (b) Tool length compensation when the turret center is placed over the start point

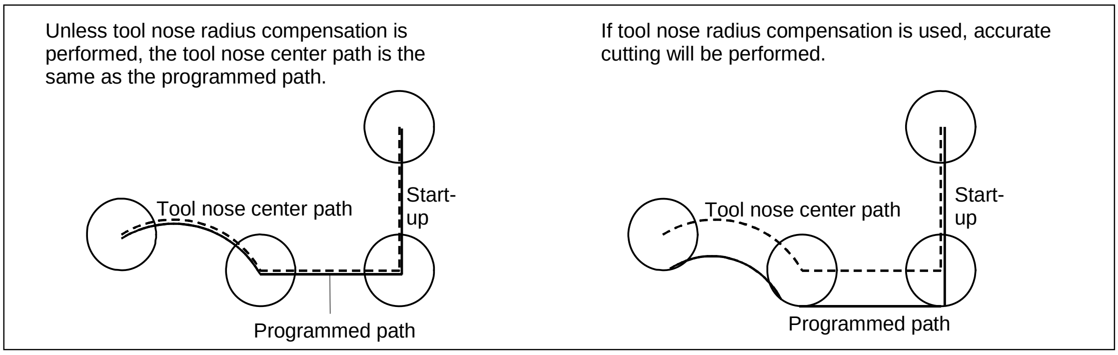


Fig. 6.6.1 (c) Tool path when programming using the tool nose center

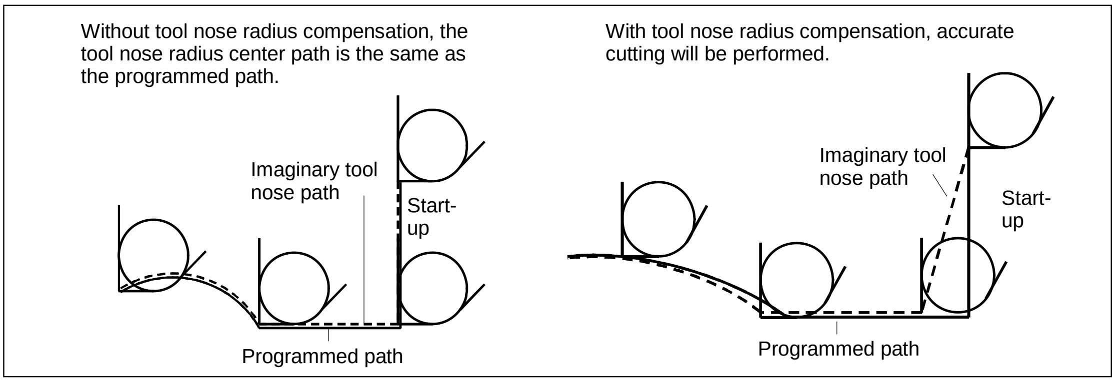


Fig. 6.6.1 (d) Tool path when programming using the imaginary tool nose

6.6.2 Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose center is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values.
The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig. 6.6.2 (a) below together with their corresponding codes. Fig. 6.6.2 (a) illustrates the relation between the tool and the start point. The following apply when the tool geometry offset and tool wear offset option are selected.

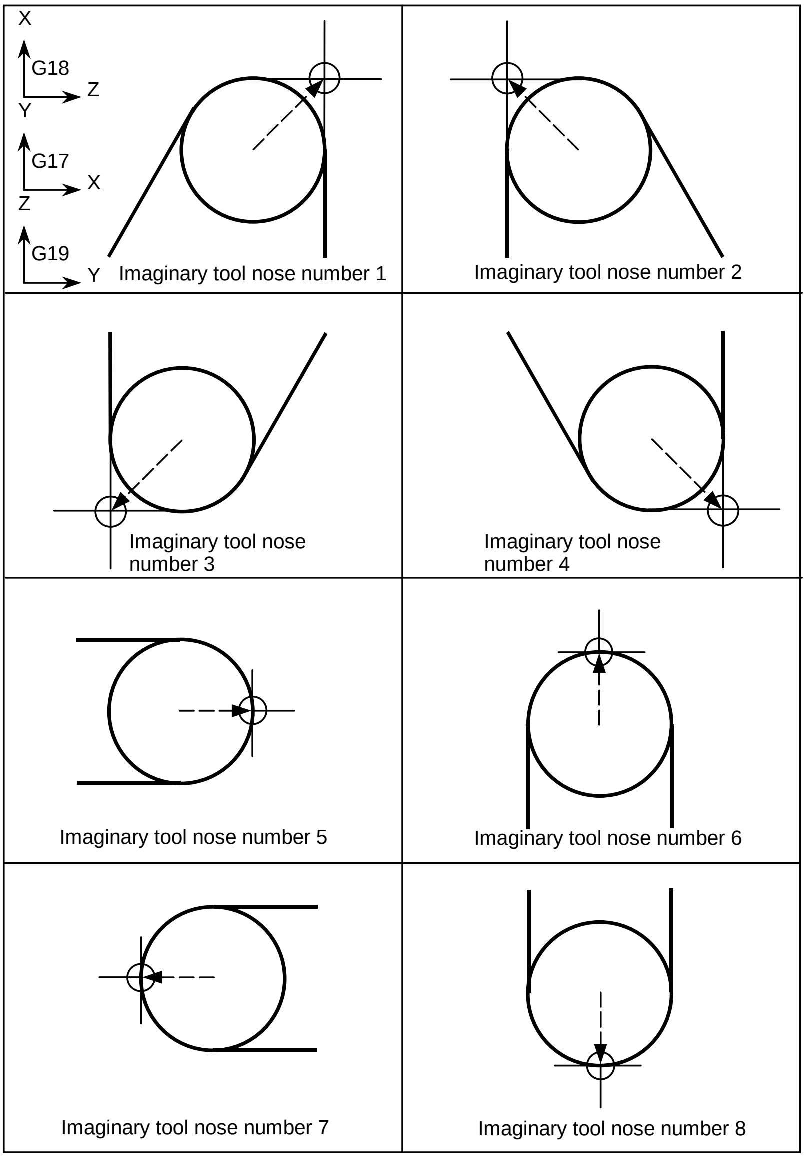
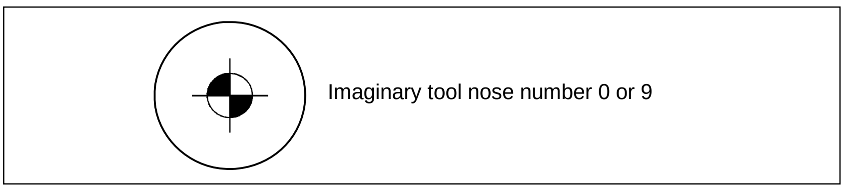


Fig. 6.6.2 (a) Direction of imaginary tool nose

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



6.6.3 Offset Number and Offset Value

Explanation

- Offset number and offset value

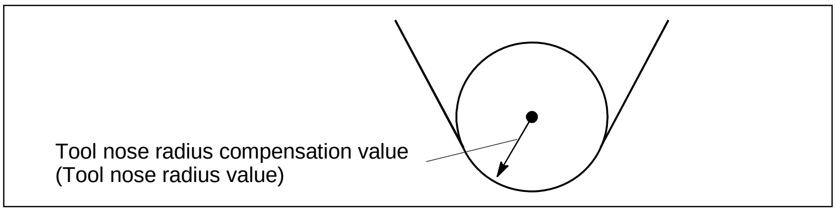


Table 6.6.3 (a) Offset number and offset value (example)

Offset number Up to 400 sets	(Tool compensation value)	(Direction of imaginary tool nose)
001	0.200	1
002	0.250	2
003	0.120	6
004	:	:
005	:	:
:	:	:

- Command of offset value

An offset number is specified with the D code.

- Setting range of offset value

The 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) of parameter No. 5042.

Table 6.6.3 (b) Valid compensation range (metric input)

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

Table 6.6.3 (c) Valid compensation range (inch input)

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

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

6.6.4 Workpiece Position and Move Command

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

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

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

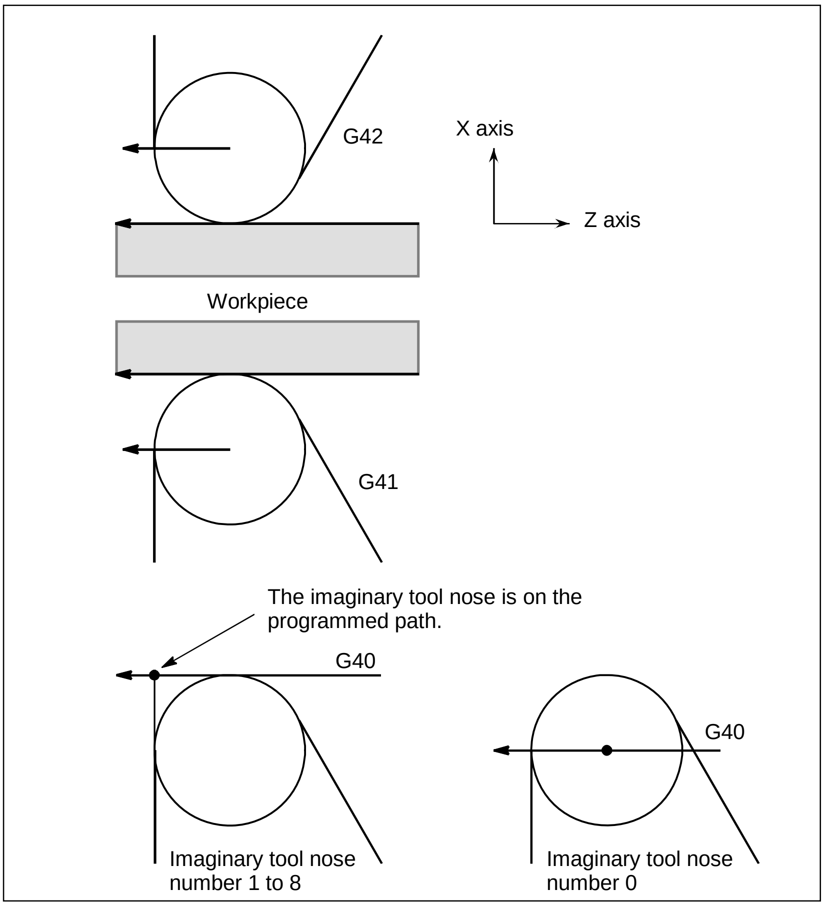


Fig. 6.6.4 (a) Workpiece position

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

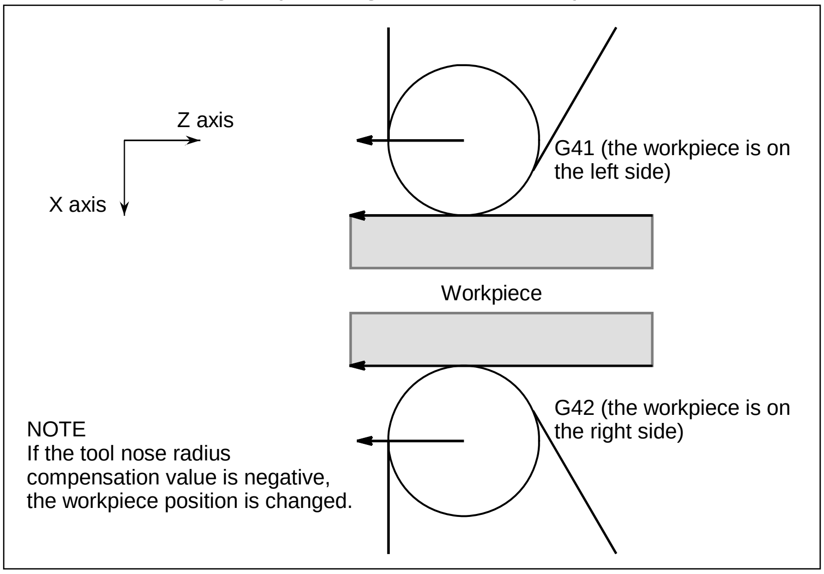


Fig. 6.6.4 (b) When the workpiece position is changed

G40, G41, and, G42 are modal.
Don't specify G41 while in the G41 mode. If you do, compensation will not work properly.
Don't specify G42 while in the G42 mode for the same reason.
G41 or G42 mode blocks in which G41 or G42 are not specified are expressed by (G41) or (G42) respectively.

> ⚠ **CAUTION**
> If the sign of the compensation value is changed from plus to minus and vice versa, the offset vector of tool nose radius compensation is reversed, but the direction of the imaginary tool nose does not change. For a use in which the imaginary tool nose is adjusted to the starting point, therefore, do not change the sign of the compensation value for the assumed program.

Explanation

- Tool movement when the workpiece position does not change

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

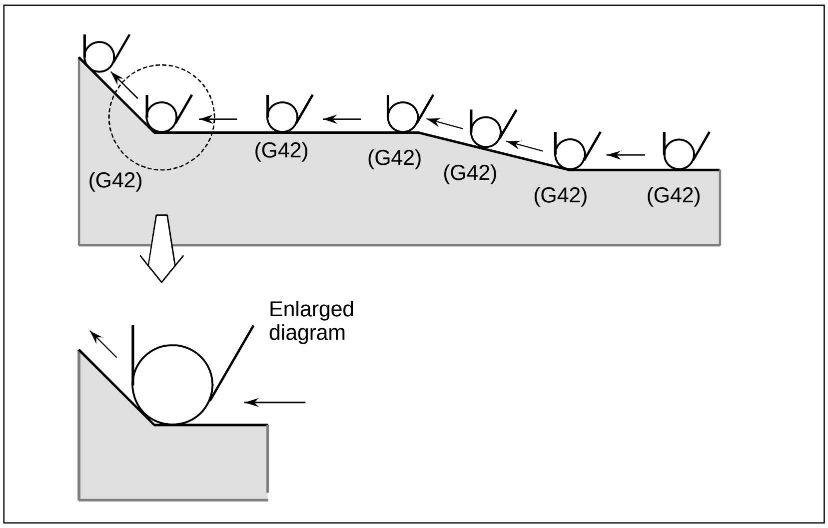


Fig. 6.6.4 (c) Tool movement when the workpiece position does not change

- Tool movement when the workpiece position changes

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

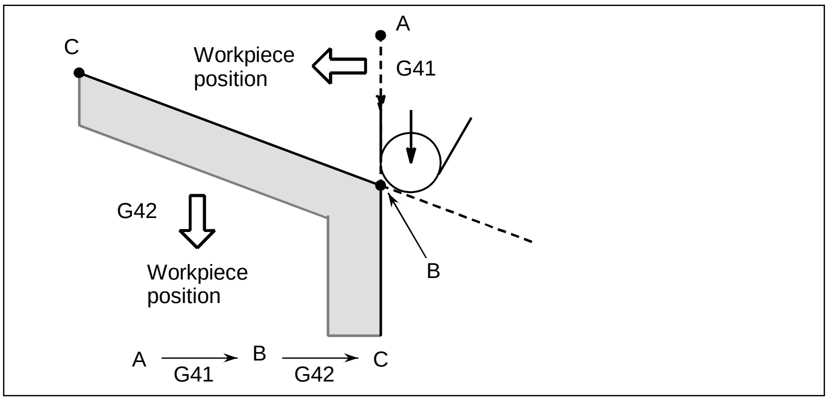


Fig. 6.6.4 (d) Tool movement when the workpiece position changes

Although the workpiece does not exist on the right side of the programmed path in the above case, the existence of the workpiece is assumed in the movement from A to B. The workpiece position must not be changed in the block next to the start-up block. In the example in the Fig. 6.6.4 (b), if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

- Start-up

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

G40 _ ;
G41 _ ; (Start-up block)

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

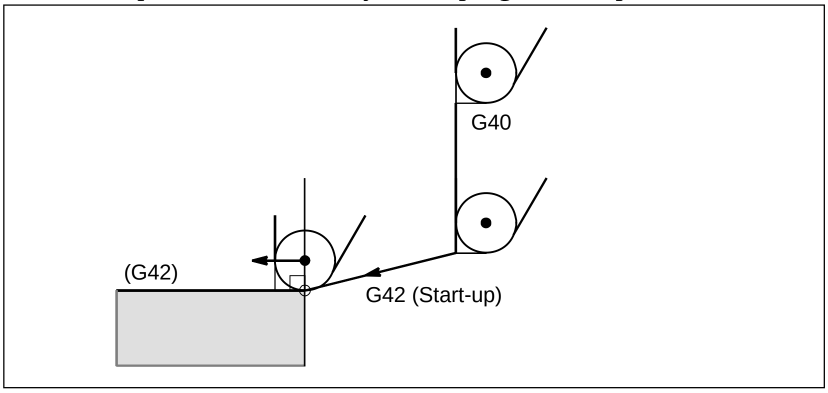


Fig. 6.6.4 (e) Start-up

- Offset cancel

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

G41 _ ;
G40 _ ; (Offset cancel block)

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

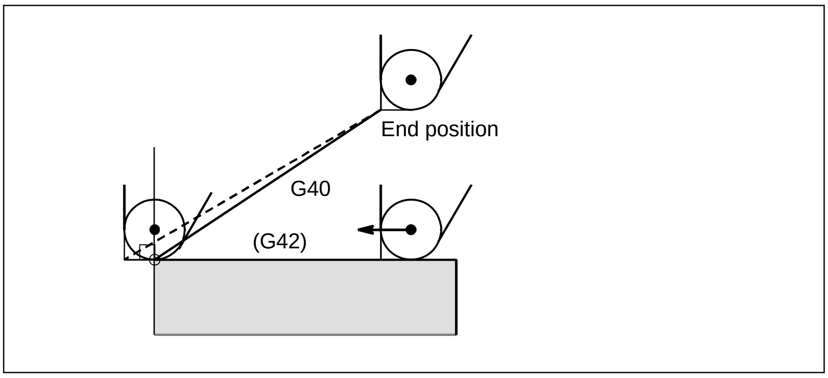


Fig. 6.6.4 (f) Offset cancel

- Changing the compensation value

In general, the compensation value is to be changed when the tool is changed in offset cancel mode. If the compensation value is changed in offset mode, however, the vector at the end point of the block is calculated using the compensation value specified in that same block.
The same applies if the imaginary tool nose direction and the tool offset value are changed.

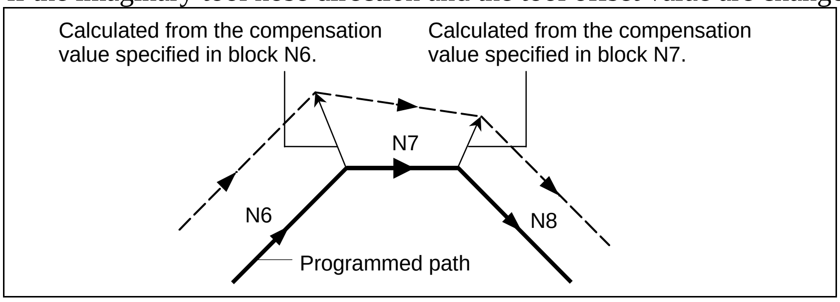


Fig. 6.6.4 (g) Changing the compensation value

- Specification of G41/G42 in G41/G42 mode

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

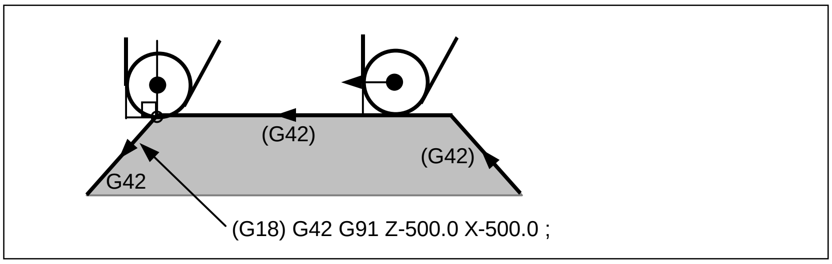


Fig. 6.6.4 (h) Specification of G41/G42 in G41/G42 mode

In the block that first changes from G40 to G41/G42, the above positioning of the tool nose center is not performed.

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

When you wish to retract the tool in the direction specified by X and Z canceling the tool nose radius compensation at the end of machining the first block in Fig. 6.6.4 (i), specify the following :

G40 X _ Z _ I _ K _ ;

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

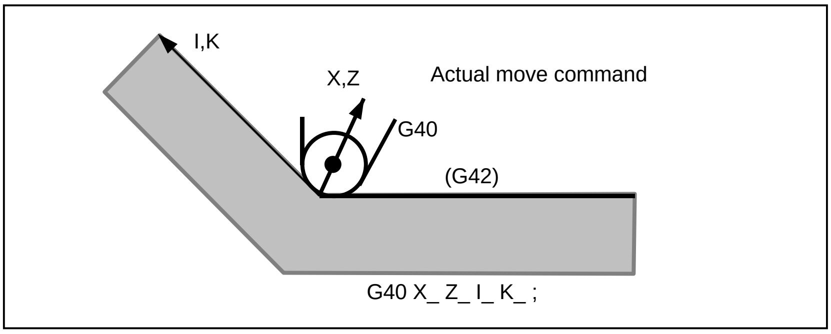


Fig. 6.6.4 (i) If I and K are specified in the same block as G40

Thus, this prevents the tool from overcutting, as shown in Fig. 6.6.4 (j).

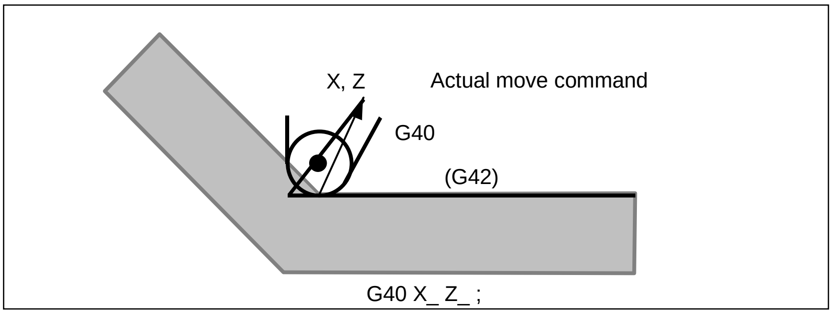


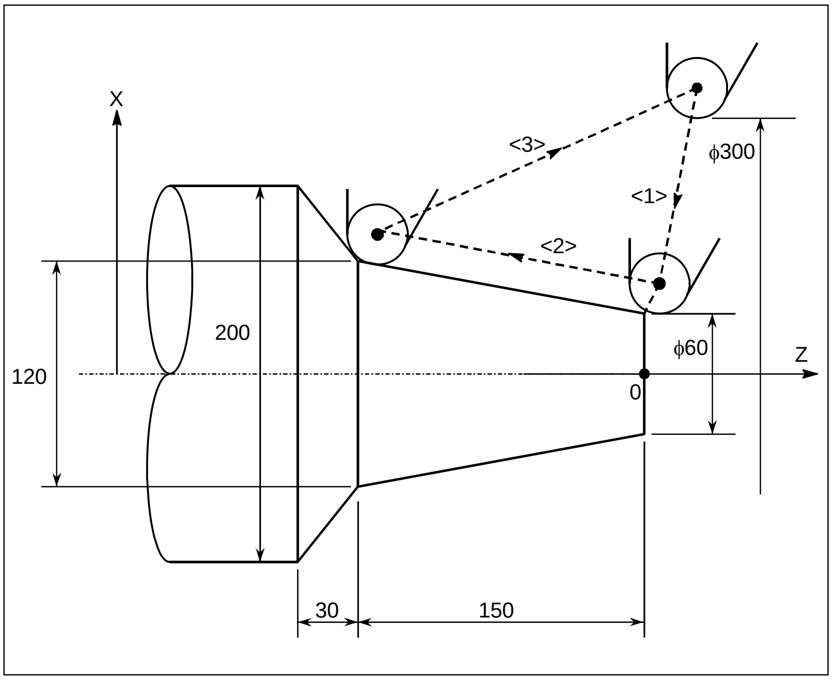
Fig. 6.6.4 (j) Case in which overcutting occurs in the same block as G40

The workpiece position specified by addresses I and K is the same as that in the preceding block.
Specify I_K_; in the same block as G40. If it is specified in the same block as G02 or G03, it is assumed to be the center of the arc.

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

If I and/or K is specified with G40 in the cancel mode, the I and/or K is ignored. The numeral is followed I and K should always be specified as radius values.
G40 G01 X_ Z_ ;
G40 G01 X_ Z_ I_ K_ ; Offset cancel mode (I and K are ineffective.)

Example



(G40 mode)
<1> **G42 G00 X60.0 ;**
<2> **G01 X120.0 Z-150.0 F10 ;**
<3> **G40 G00 X300.0 Z0 I40.0 K-30.0 ;**

6.6.5 Notes on Tool Nose Radius Compensation

Explanation

- Blocks without a move command that are specified in offset mode

<1> **M05 ;**	M code output
<2> **S210 ;**	S code output
<3> **G04 X10.0 ;**	Dwell
<4> **G22 X100000 ;**	Machining area setting
<5> **G91 G01 X0.0 ;**	Feed distance of zero
<6> **G90 ;**	G code only
<7> **G10 L11 P01 R10.0 ;**	Offset change

If the number of such blocks consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)), the tool arrives at the position vertical to this block at the end point of the previous block.
If the feed distance is 0 (<5>), this applies even if only one block is specified.

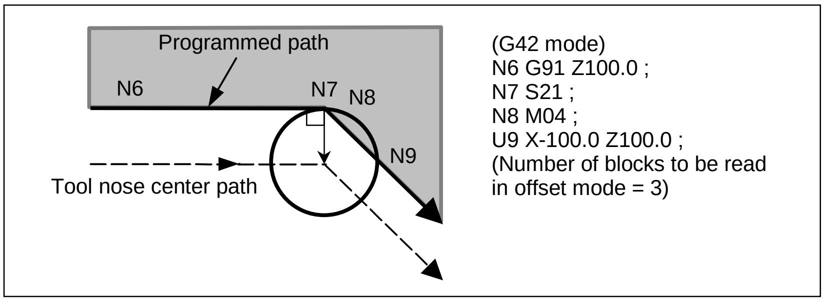


Fig. 6.6.5 (a)

Overcutting may, therefore, occur in the Fig. 6.6.5 (a).

- Tool nose radius compensation when chamfering is performed

Movement after compensation is shown Fig. 6.6.5 (b).

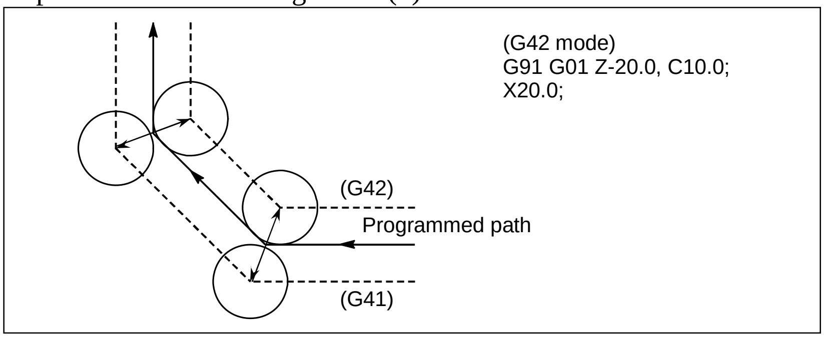


Fig. 6.6.5 (b)

- Tool nose radius compensation when a corner R is performed

Movement after compensation is shown Fig. 6.6.5 (c).

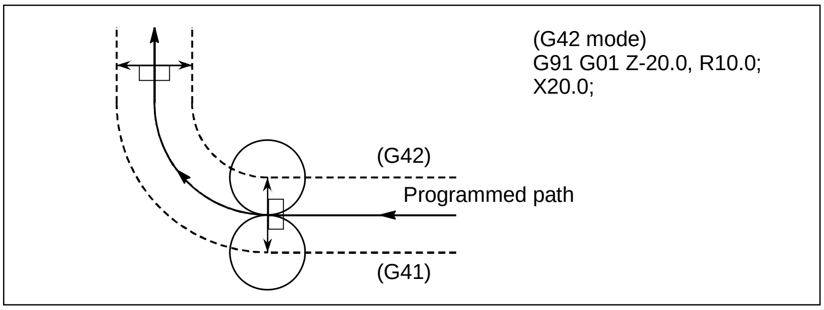


Fig. 6.6.5 (c)

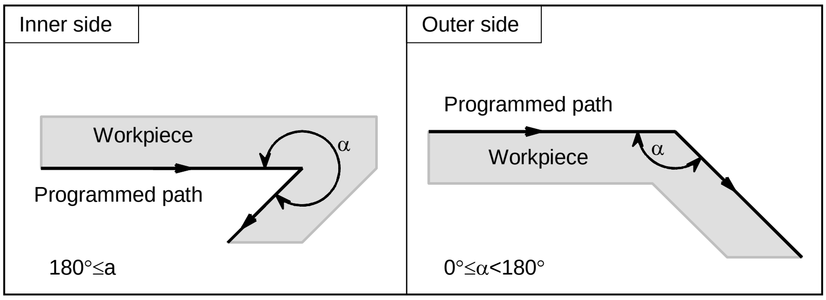
6.7 DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION

6.7.1 Overview

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

- Inner side and outer side

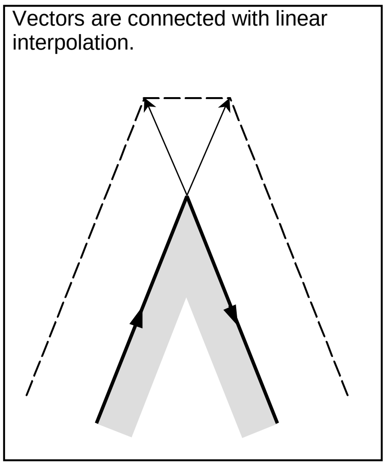
When an angle of intersection of the tool paths specified with move commands for two blocks on the workpiece side is over 180°, it is referred to as "inner side." When the angle is between 0° and 180°, it is referred to as "outer side."



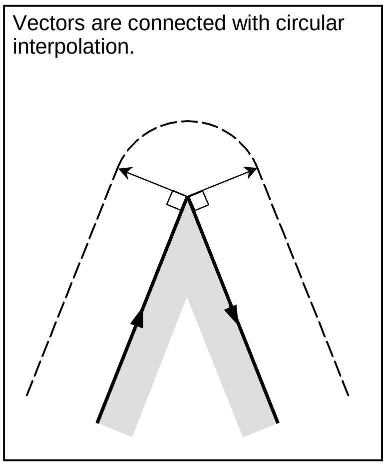
- Outer corner connection method

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

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



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



- Cancel mode

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

<1> Immediately after the power is turned on

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

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

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

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

NOTE

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

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

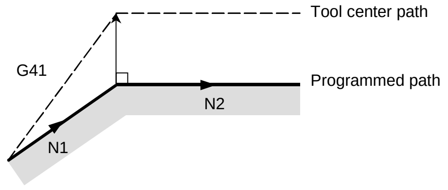
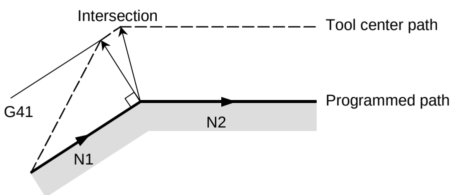
- Start-up

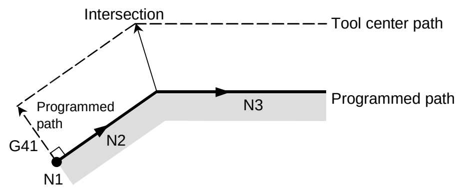
When a block which satisfies all the following conditions is executed in cancel mode, the CNC enters the cutter compensation mode. Control during this operation is called start-up.

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

If start-up is specified in circular interpolation (G02, G03) mode, alarm PS0034, “ONLY G00/G01 ALLOWED IN STUP/EXT BLK” will occur.
In a start-up block, workpiece coordinate system switching (G54 to G59) cannot be specified.
As a start-up operation, one of the three types A, B, and C can be selected by setting bit 0 (SUP) of parameter No. 5003 and bit 1 (SUV) of parameter No. 5003 appropriately. The operation to be performed if the tool moves around an inner side is of single type only.

Table 6.7.1 (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. 

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

- Reading input commands in cutter compensation mode

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

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

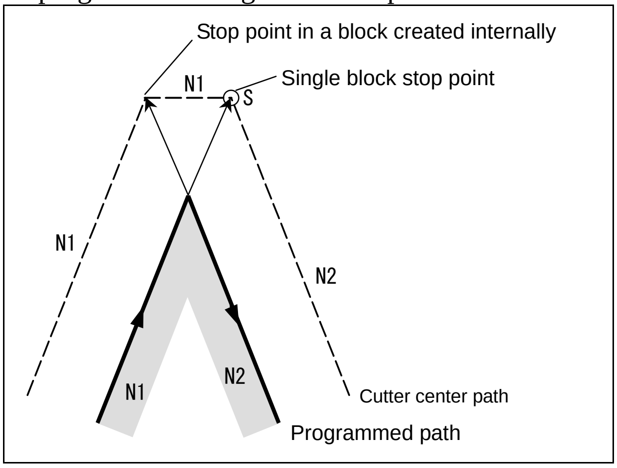
- Ending (canceling) cutter compensation

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

- Bit 0 (SBK) of parameter No. 5000

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

Use this parameter to check a program including cutter compensation.



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

- Meaning of symbols

The following symbols are used in subsequent figures:

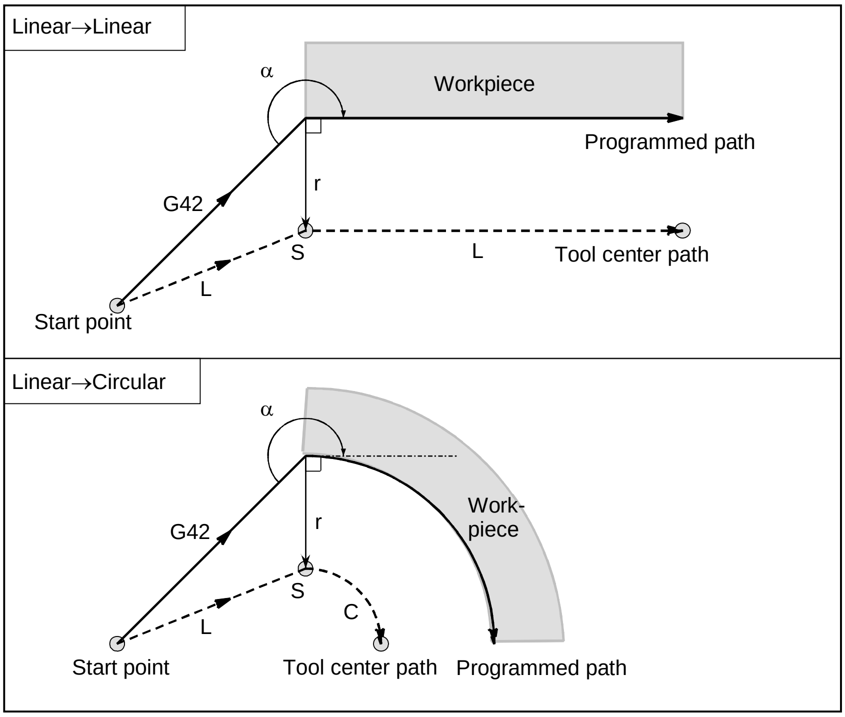
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves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

6.7.2 Tool Movement in Start-up

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

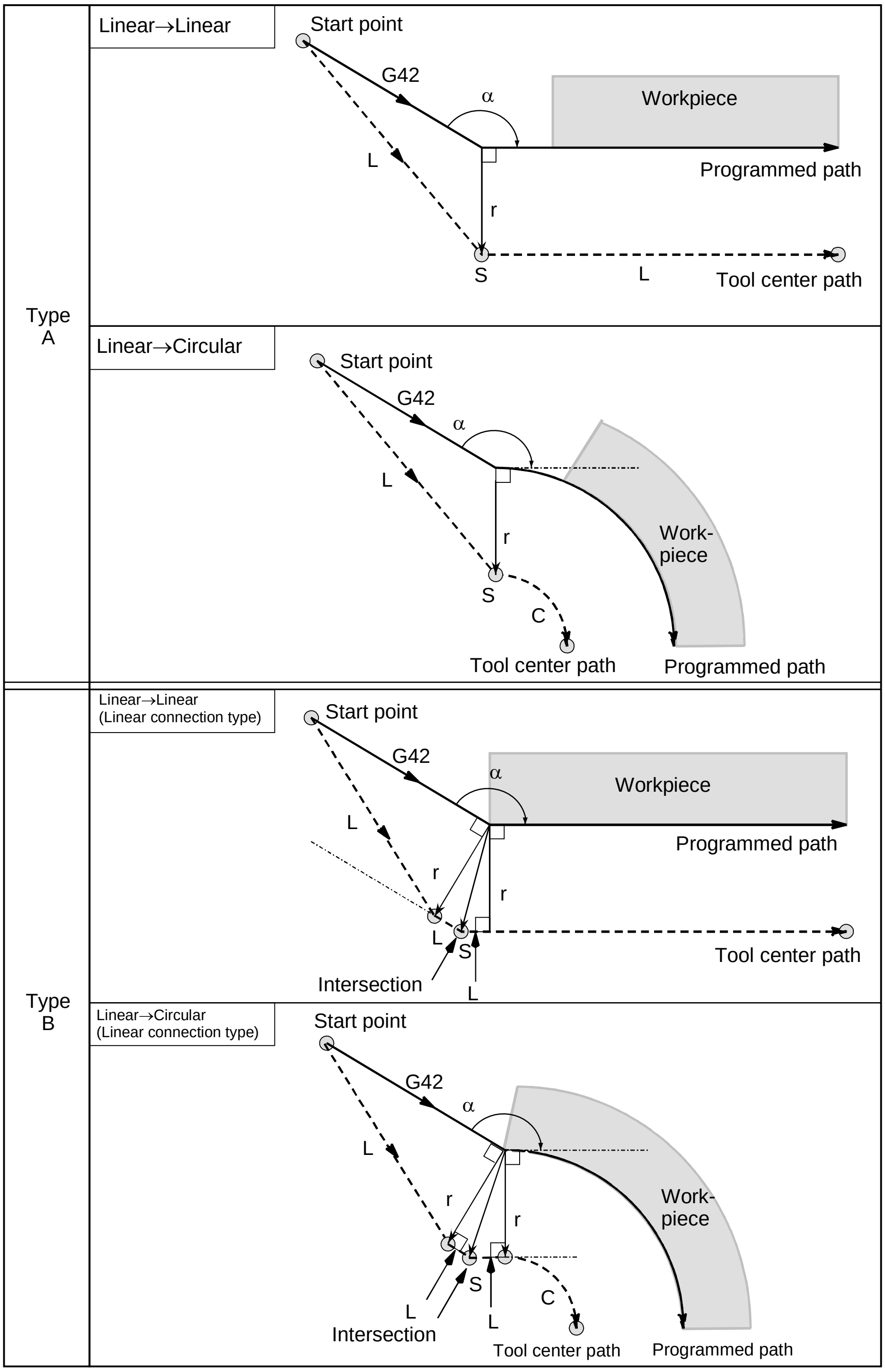
Explanation

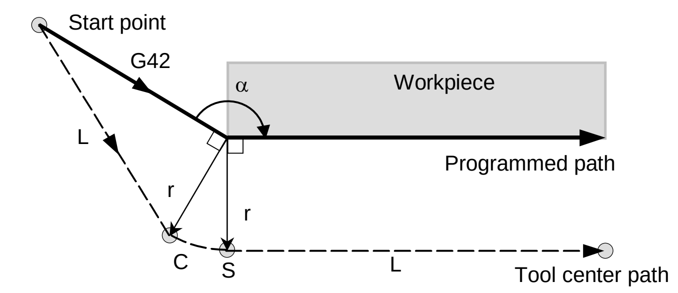
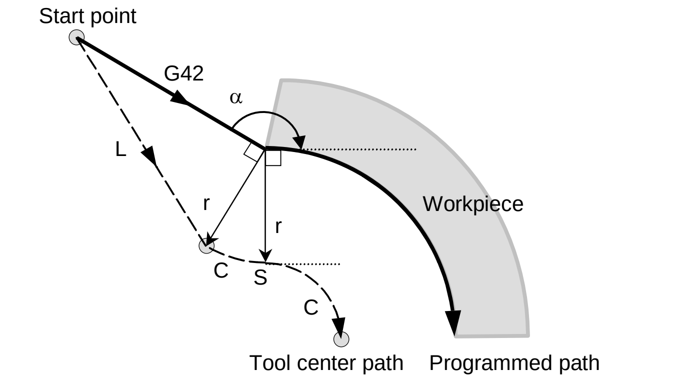
- Tool movement around an inner side of a corner (180°≤ α)



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

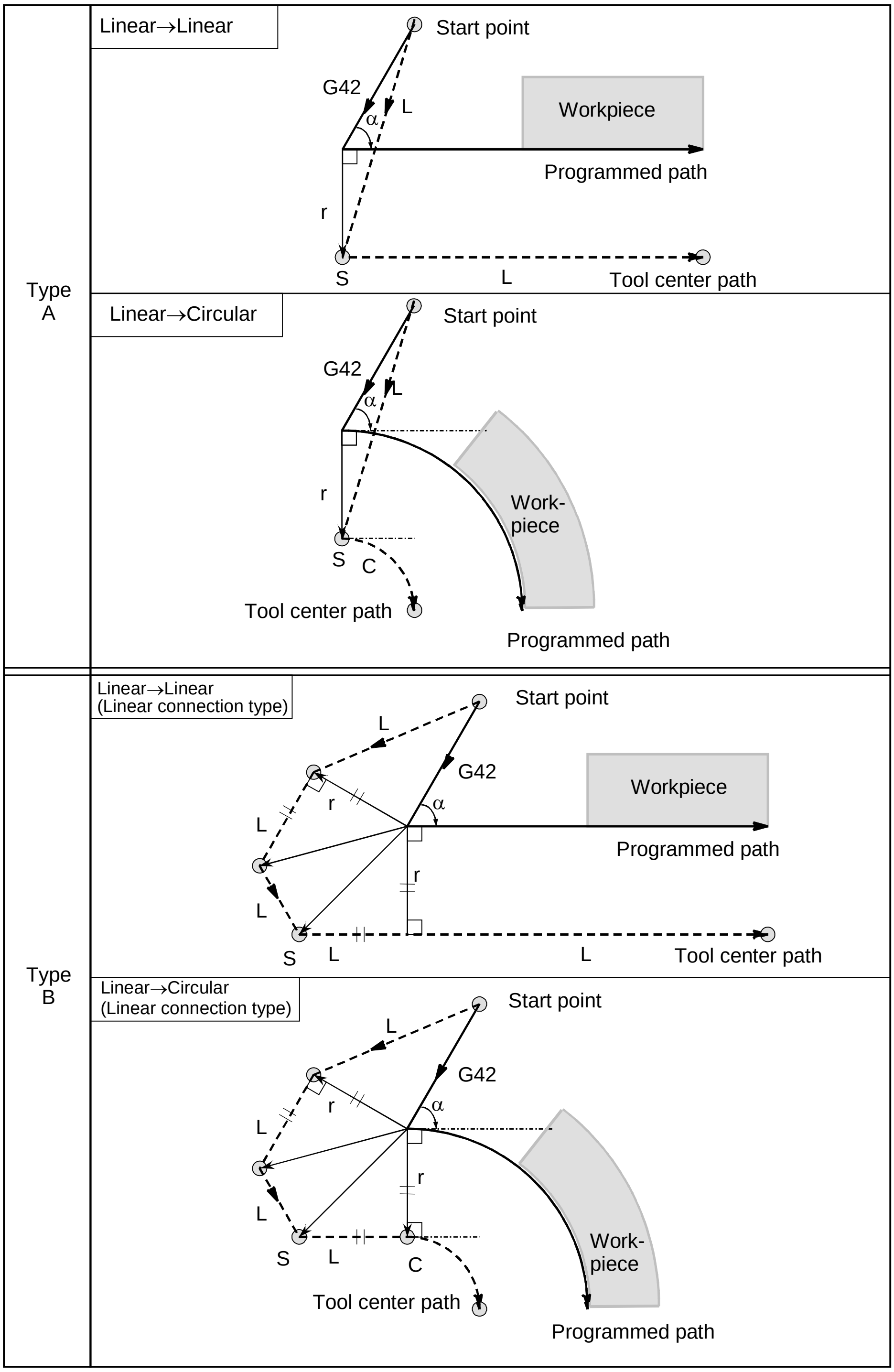
Tool path in start-up has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

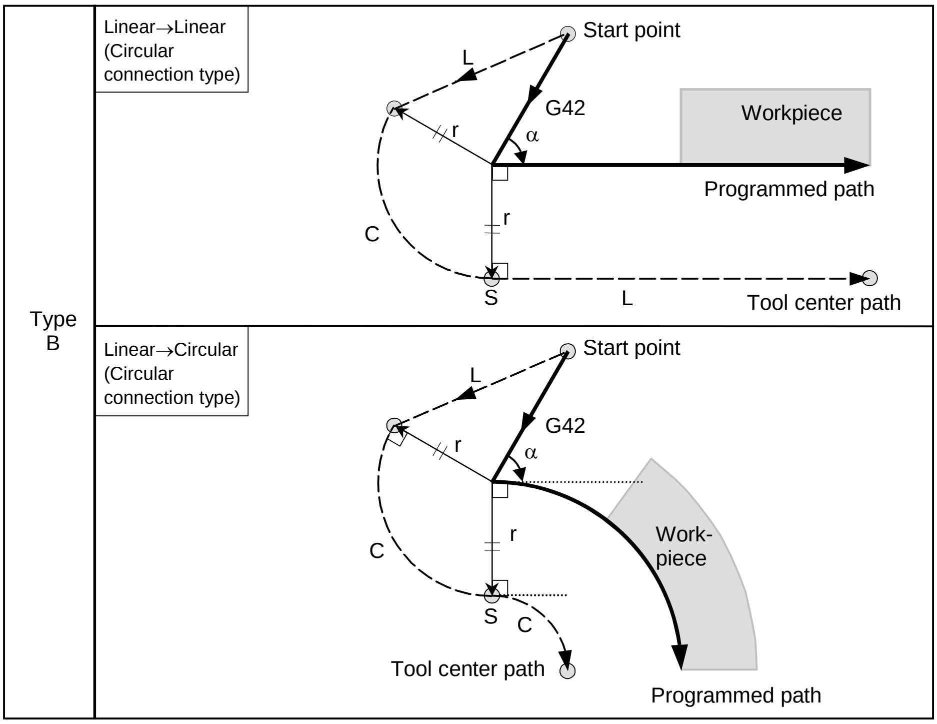


Type B	Linear→Linear (Circular connection type)	
	Linear→Circular (Circular connection type)	

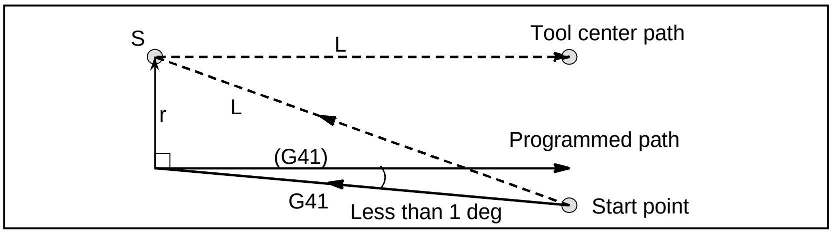
- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle (α<90°)

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





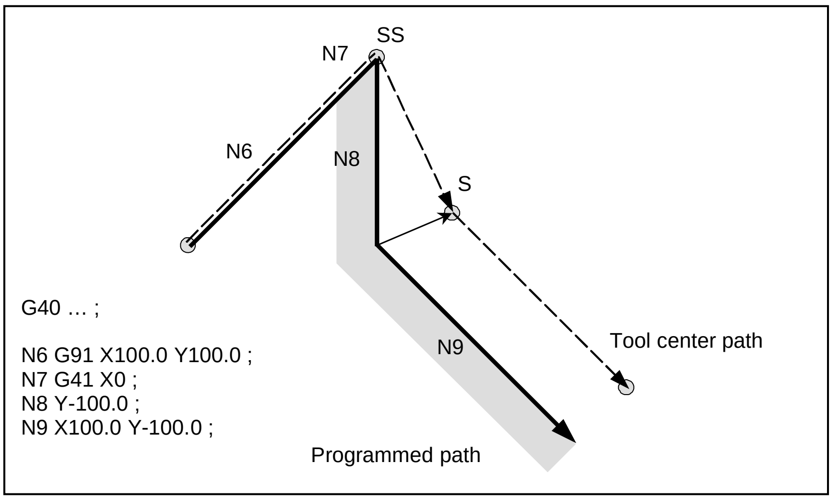
- Tool movement around the outside linear → linear at an acute angle less than 1 degree (α<1°)



- A block without tool movement specified at start-up

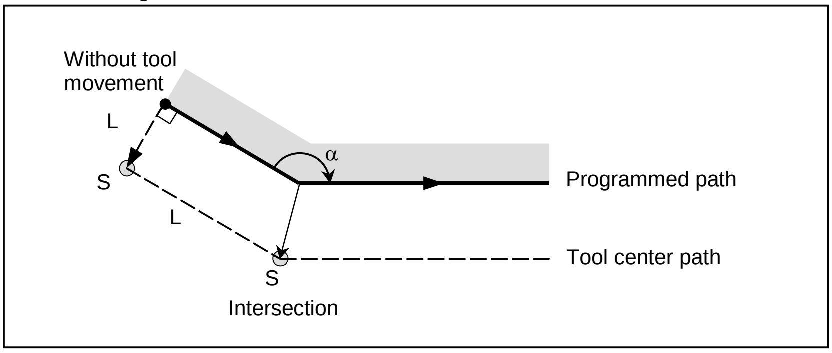
For type A and B

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.



6.7.3 Tool Movement in Offset Mode

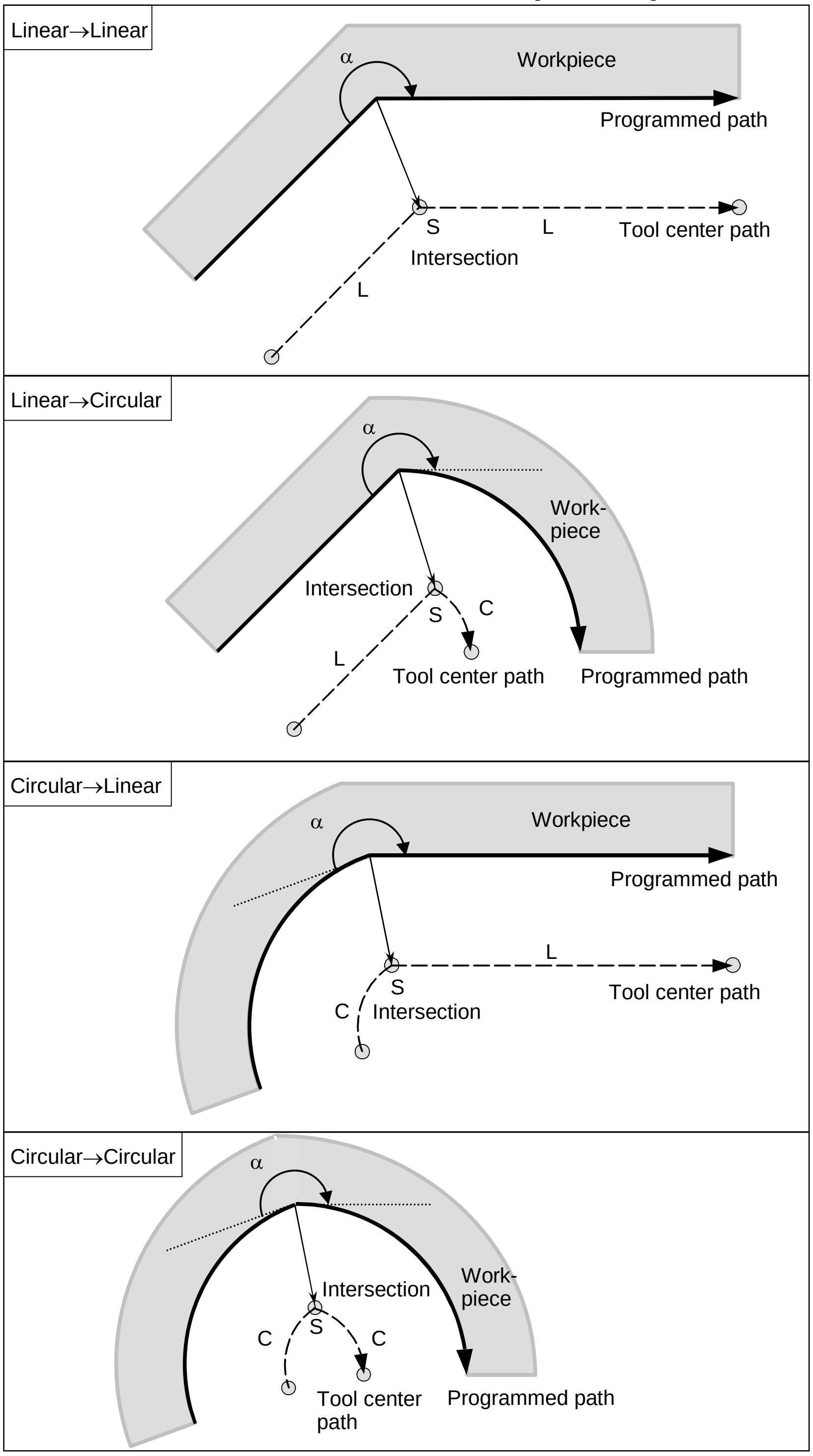
In offset mode, compensation is performed even for positioning commands, not to speak of linear and circular interpolations. To perform intersection calculation, it is necessary to read at least two blocks with tool movement. If, therefore, two or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as auxiliary function independent commands and dwell, are specified in succession, excessive or insufficient cutting may occur because intersection calculation fails. Assuming the number of blocks to read in offset mode, which is determined by parameter No. 19625, to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which intersection calculation is possible is $(N - 2) \geq M$. For example, if the maximum number of blocks to read in offset mode is 5, intersection calculation is possible even if up to three blocks without tool movement are specified.

NOTE

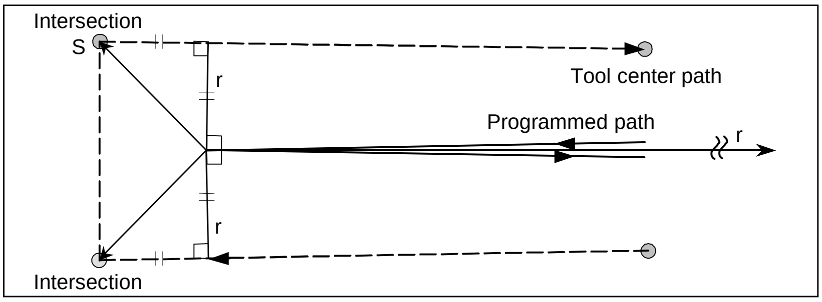
The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the interference check.

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

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

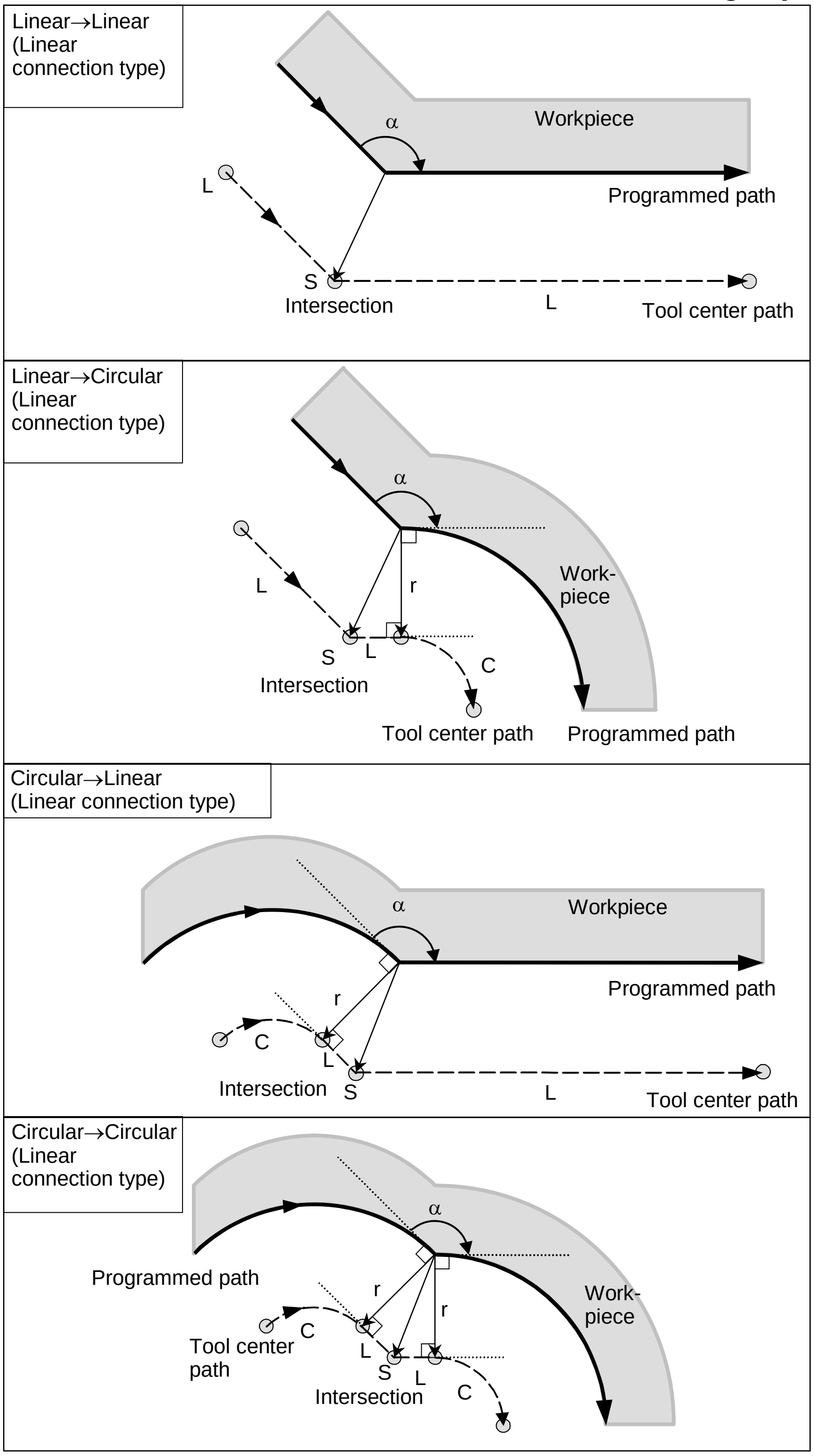


- Tool movement around the inside (α<1°) with an abnormally long vector, linear → linear

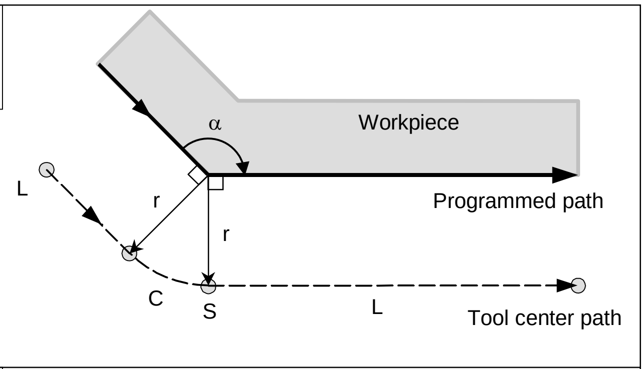


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

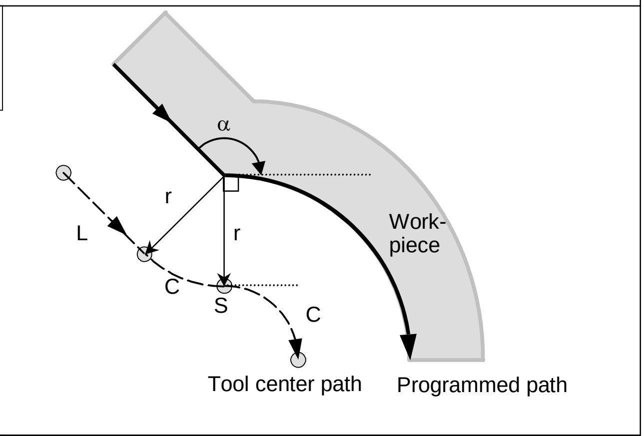
- Tool movement around the outside corner at an obtuse angle (90°≤α<180°)



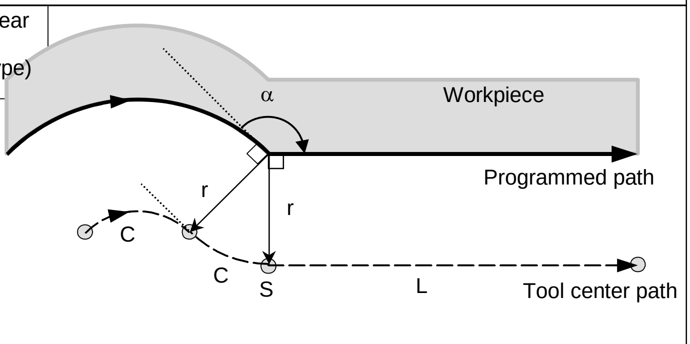
Linear→Linear (Circular connection type)



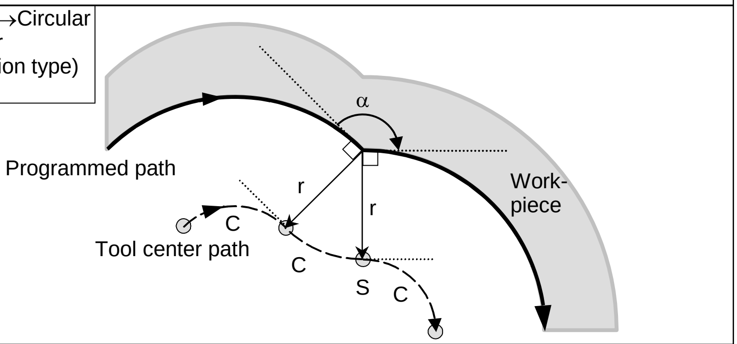
Linear→Circular (Circular connection type)



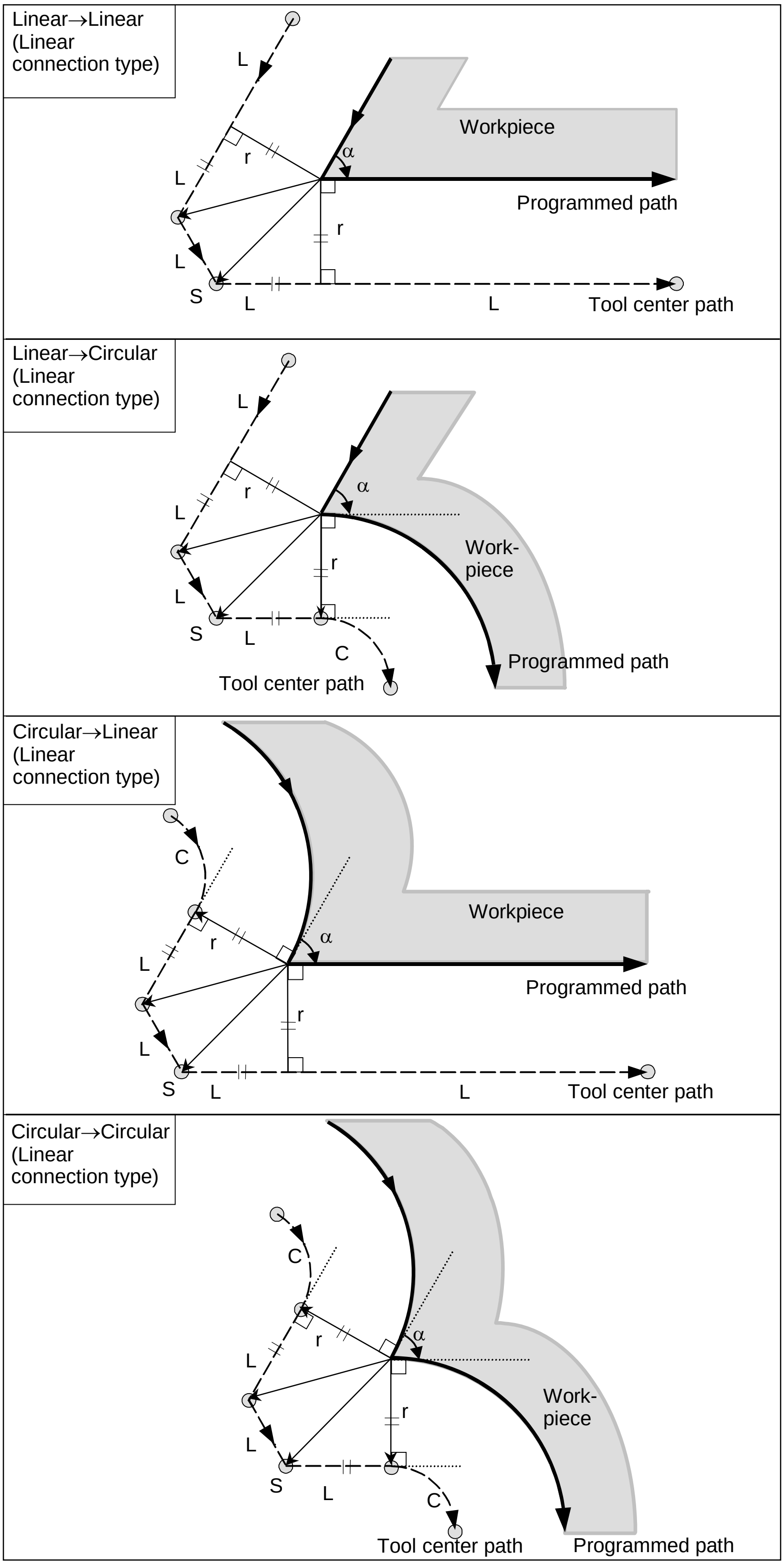
Circular→Linear (Circular connection type)



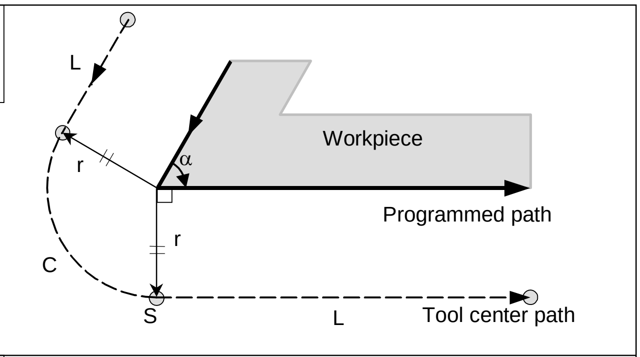
Circular→Circular (Circular connection type)



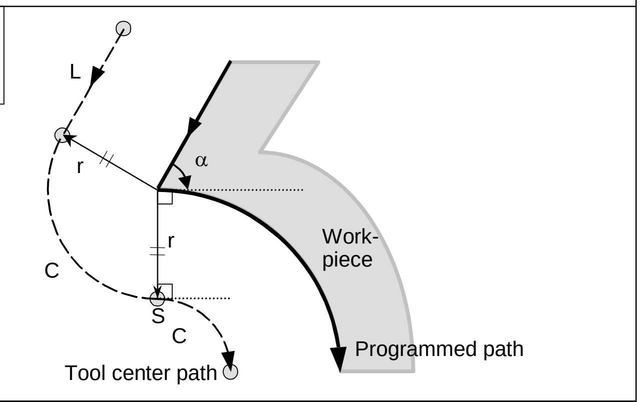
- Tool movement around the outside corner at an acute angle (α<90°)



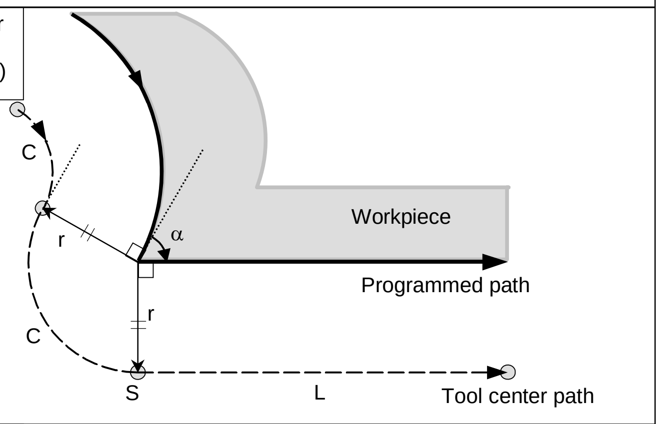
Linear→Linear
(Circular
connection type)



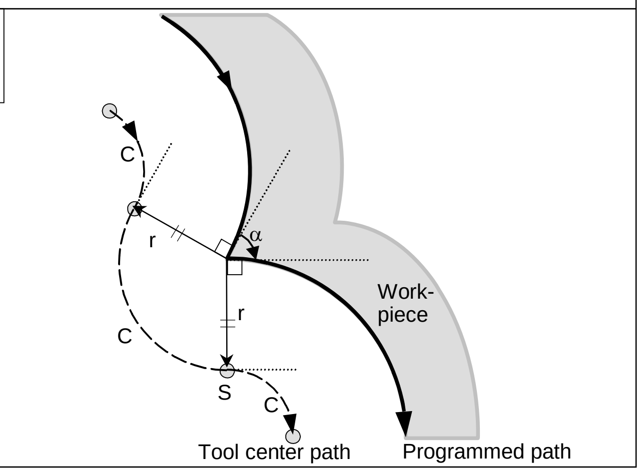
Linear→Circular
(Circular
connection type)



Circular→Linear
(Circular
connection type)

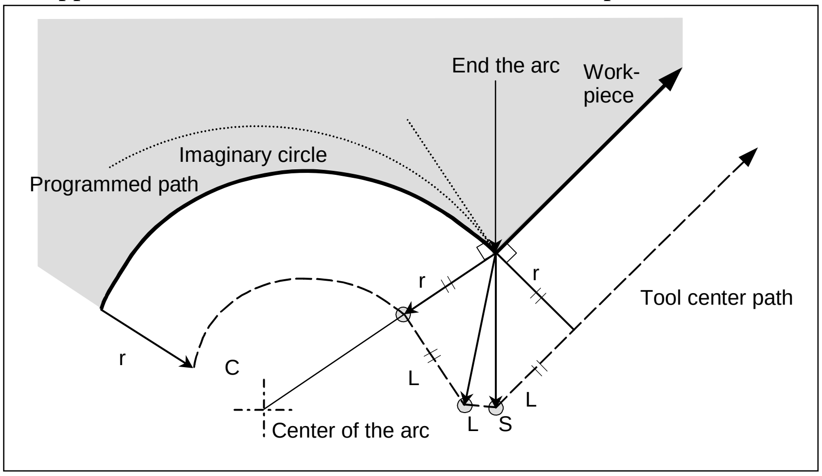


Circular→Circular
(Circular
connection type)



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

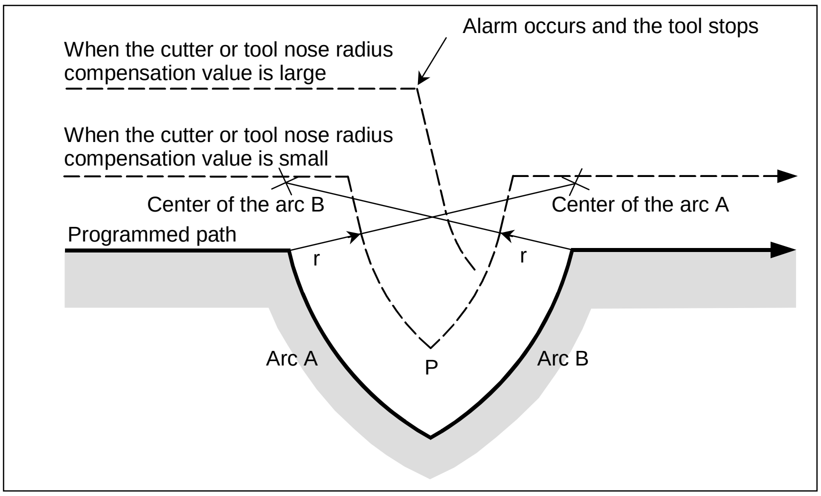
If the end of a line leading to an arc is not on the arc, the system assumes that the cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end point. Based on this assumption, the system creates a vector and carries out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



There is no inner intersection

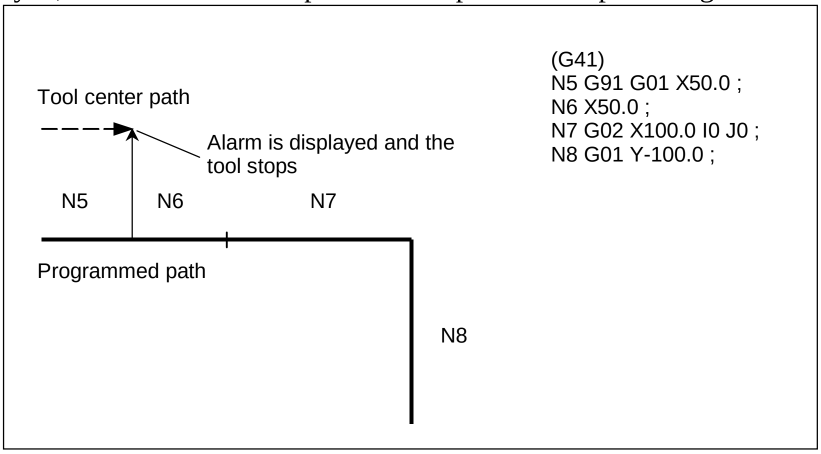
If the tool radius / tool nose radius compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool radius · tool nose radius compensation. When this is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

For example, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool radius · tool nose radius compensation. If an excessively large value is specified, this intersection does not occur.



- When the center of the arc is identical with the start point or the end point

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



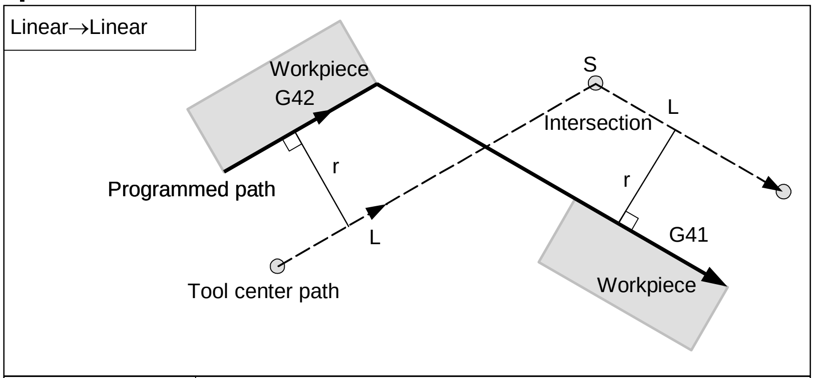
- Change in the offset direction in the offset mode

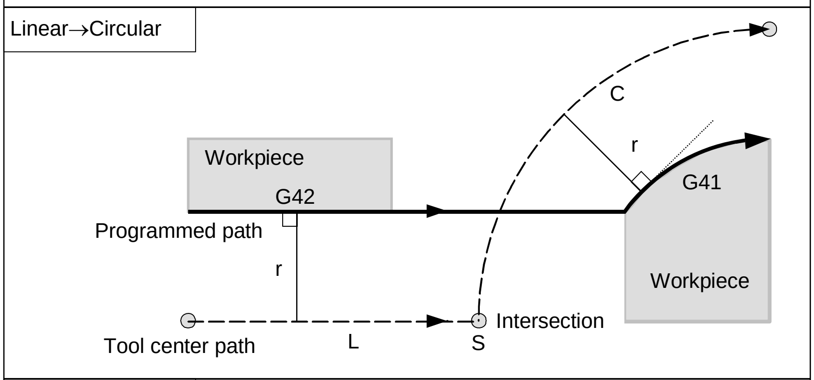
The offset direction is decided by G codes (G41 and G42) for tool radius · tool nose radius compensation and the sign of the compensation value as follows.

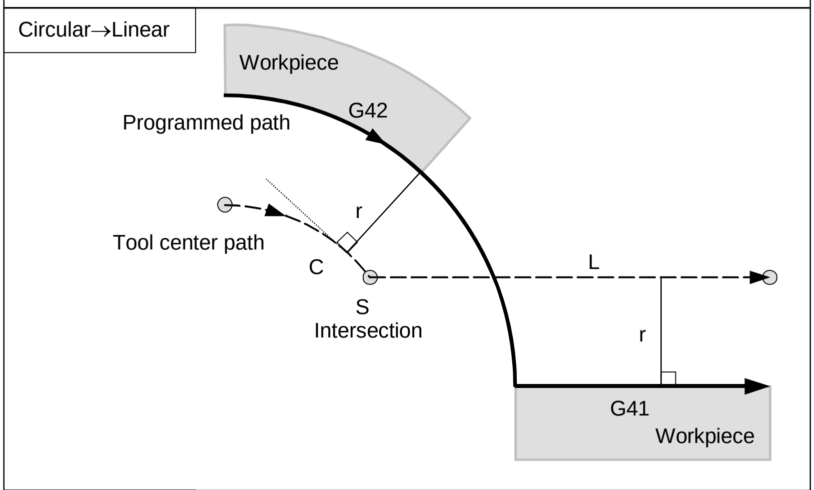
Sign of compensation / G code	+	–
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

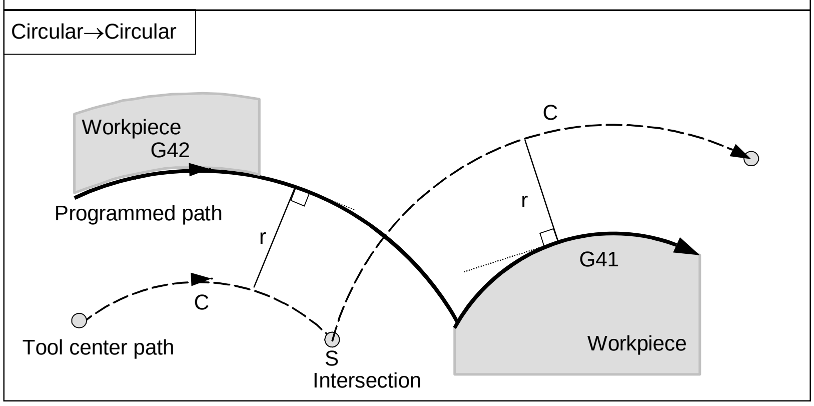
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block.
However, the change is not available in the start-up block and the block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

- Tool center path with an intersection



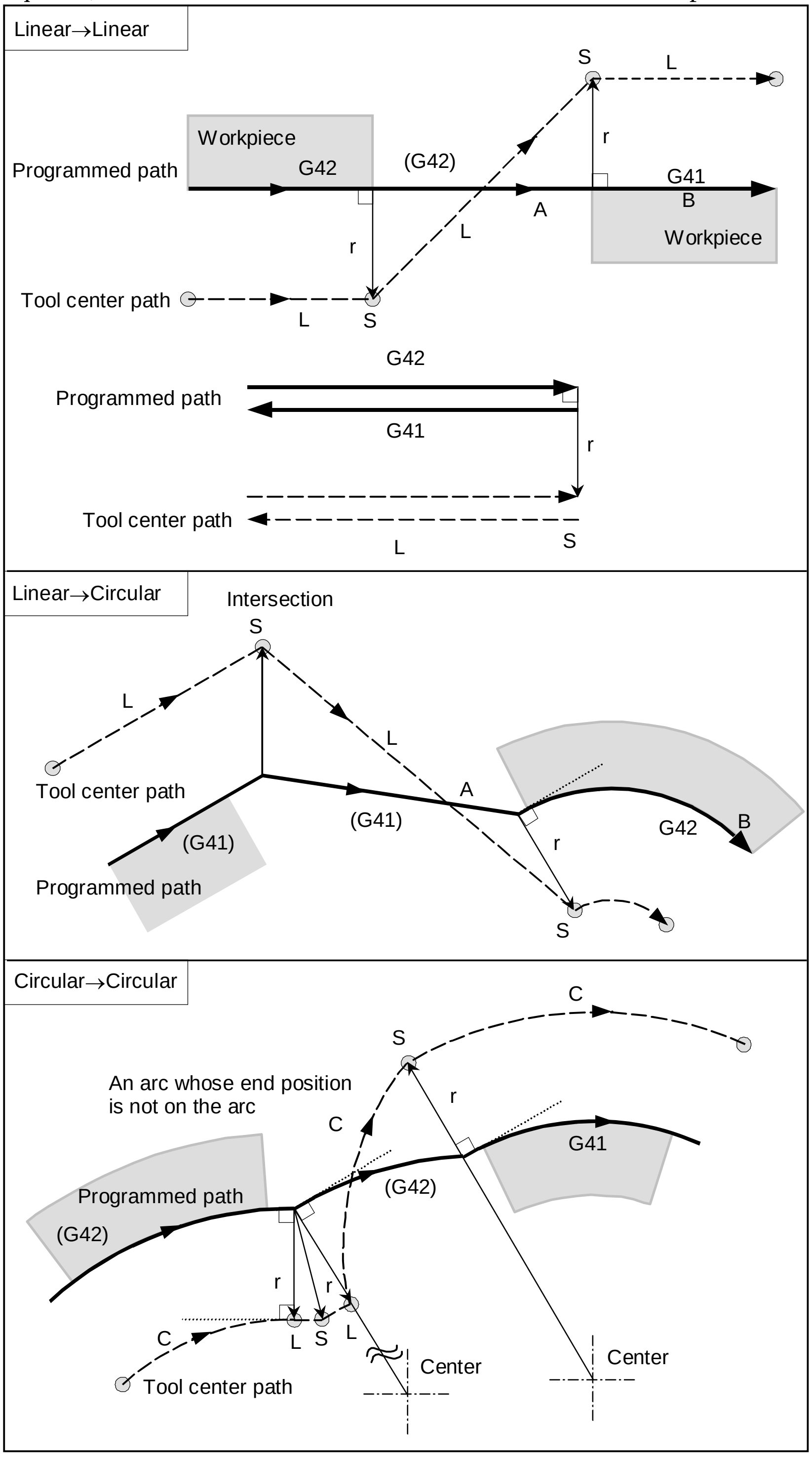






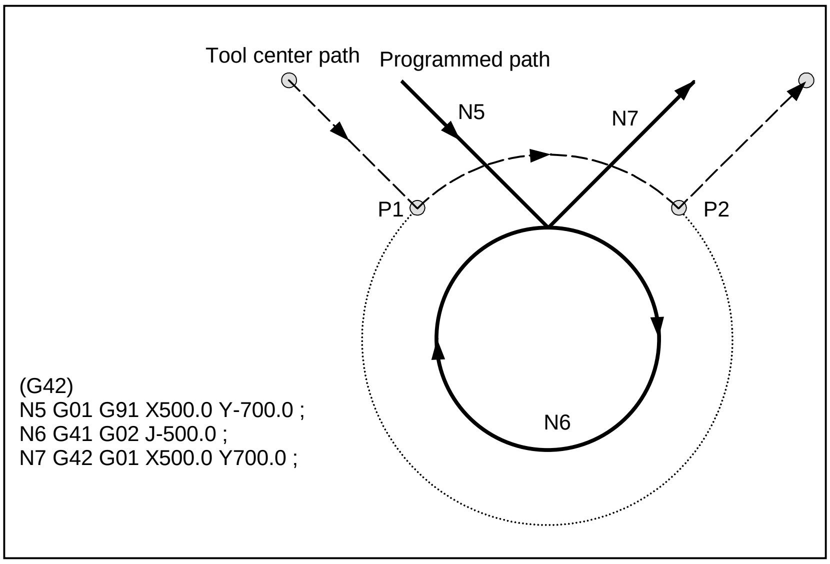
- Tool center path without an intersection

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.



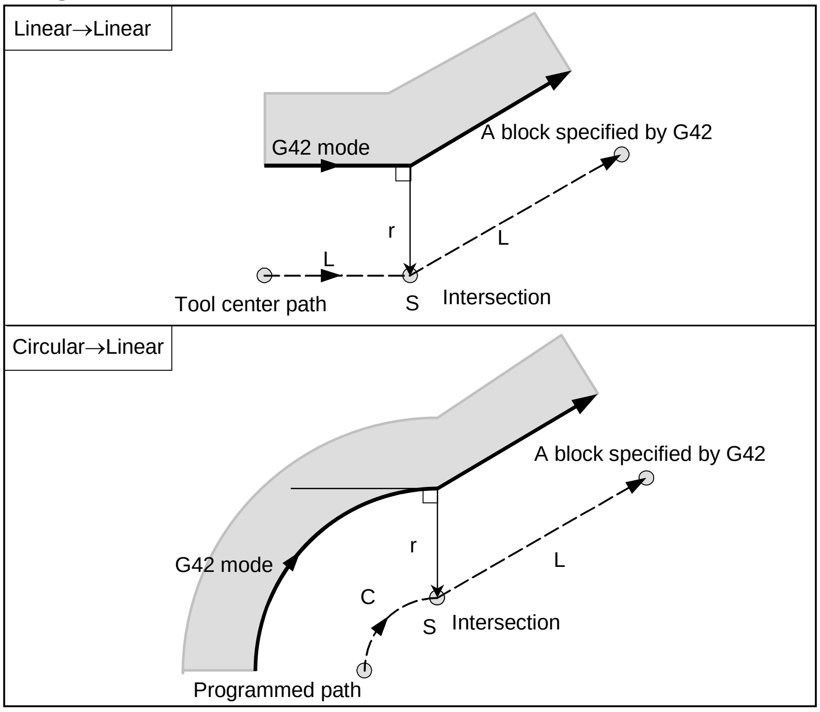
- The length of tool center path larger than the circumference of a circle

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.
The cutter compensation is not performed with more than one circle circumference: an arc is formed from P_1 to P_2 as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.



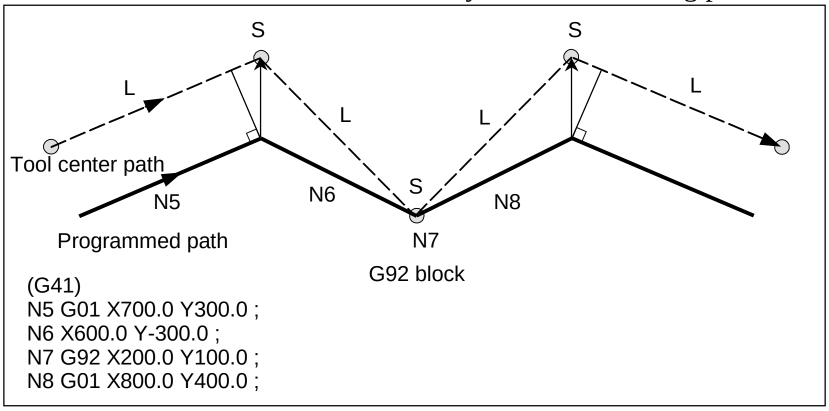
- Cutter compensation G code in the offset mode

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



- Command canceling the offset vector temporarily

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



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

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

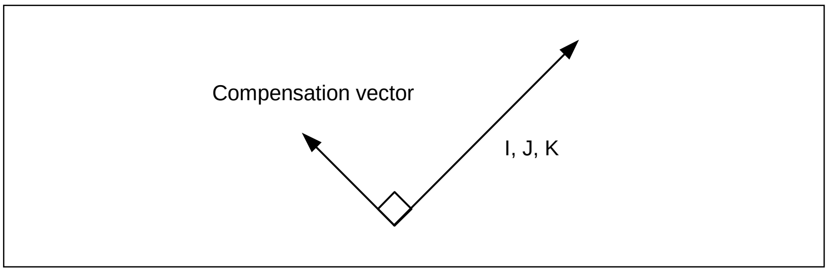
At the start of cutter compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This makes it possible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

IJ type vector (XY plane)

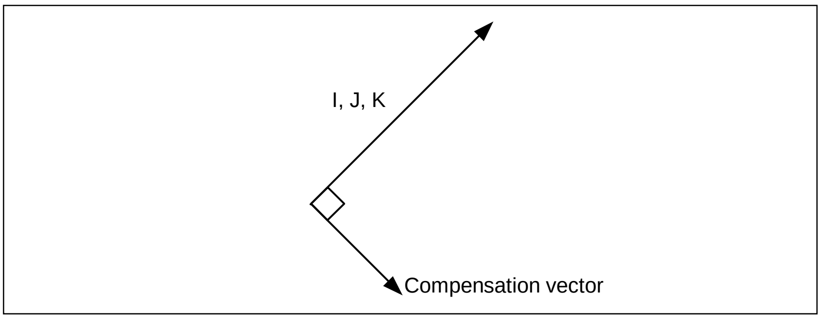
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) It is assumed that the compensation vector (IJ type vector) is the vector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

Offset vector direction

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



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

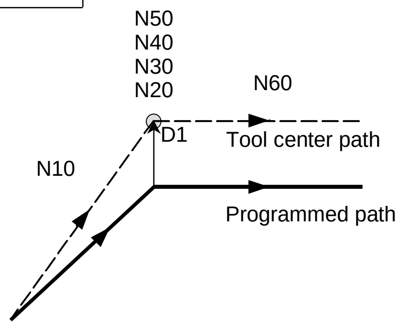


Example

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

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

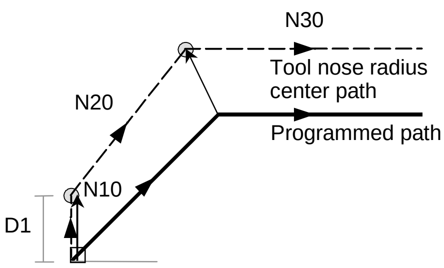
Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.



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

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

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

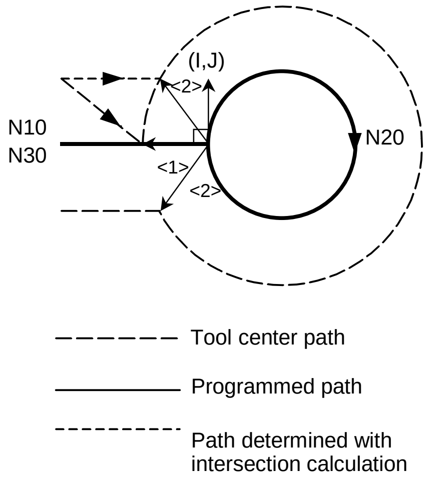


If I and J are specified at the start of compensation (with tool movement)

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

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

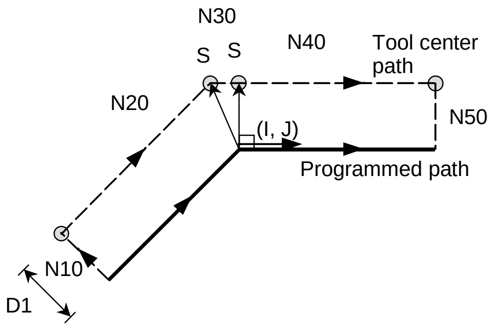
<1> IJ type vector
<2> Vector determined with intersection calculation



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 ;

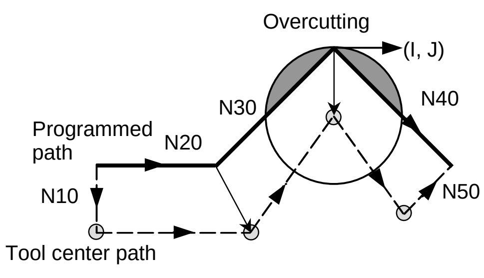


Limitation

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

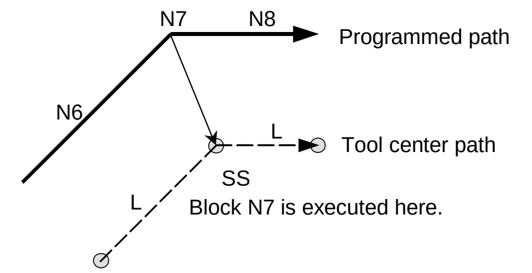
The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

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

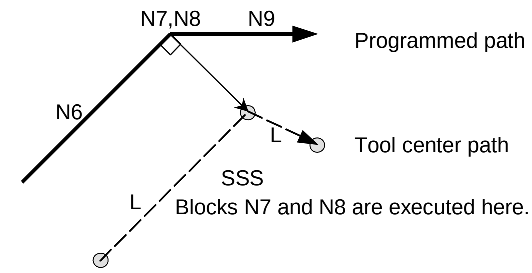
Unless the number of blocks without movement consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)) in offset mode, the vector and the tool center path will be as usual. This block is executed at the single block stop point.

```
N6 G91 X100.0 Y100.0 ;
N7 G04 X10.0 ;
N8 X100.0 ;
```



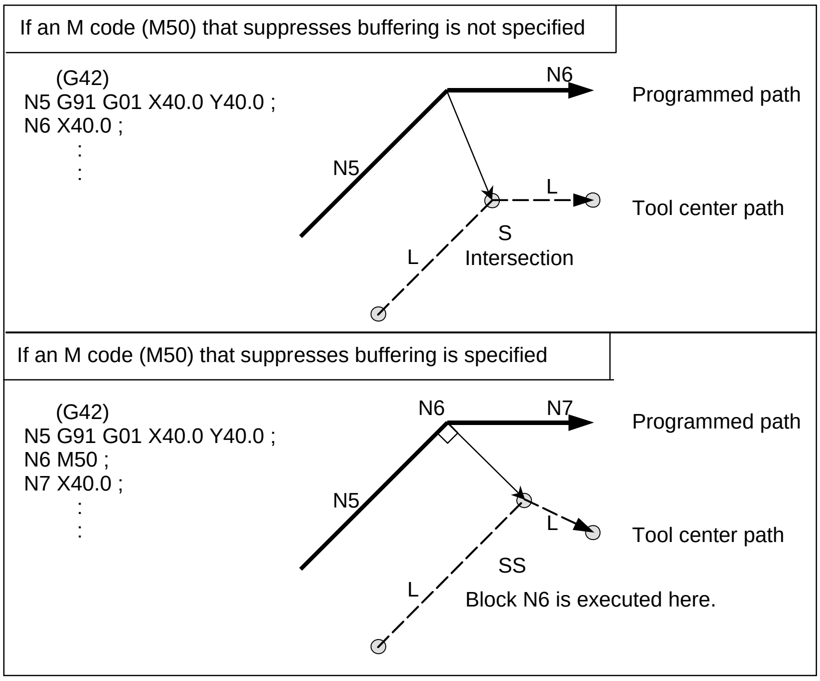
In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter No. 19625). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier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

```
N6 G91 X100.0 Y100.0 ;
N7 S21 ;
N8 G04 X10.0 ;
N9 X100.0 ;
(No. of blocks to read in
offset mode = 3)
```



- 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.

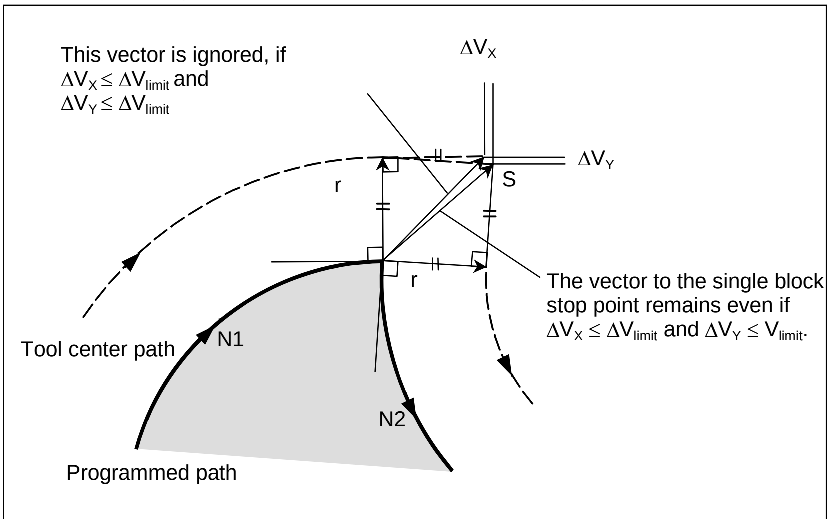


- Workpiece coordinate system or local coordinate system command in the offset mode

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.

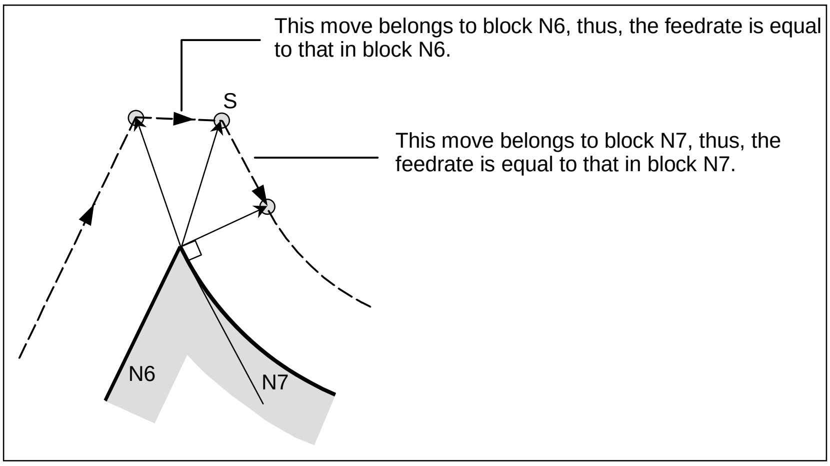
- Corner movement

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.

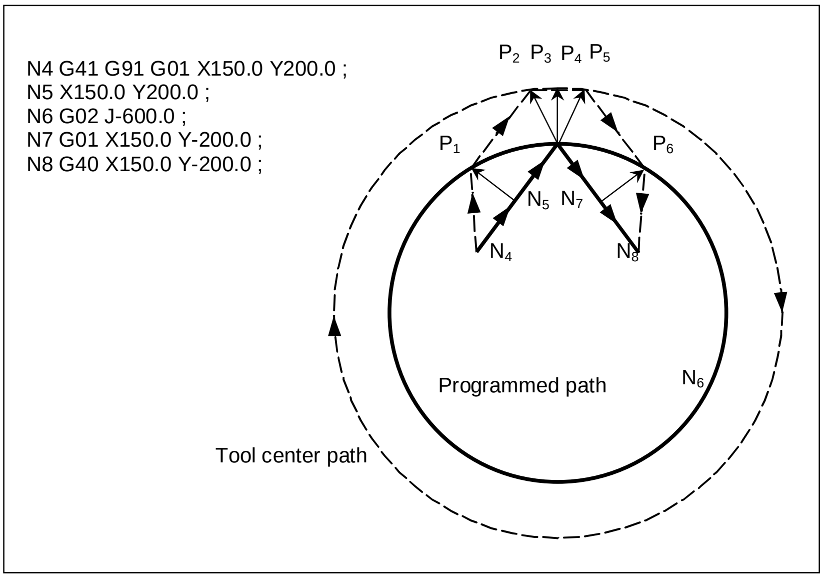


ΔV_{limit} is determined with the setting of parameter (No. 5010).

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.

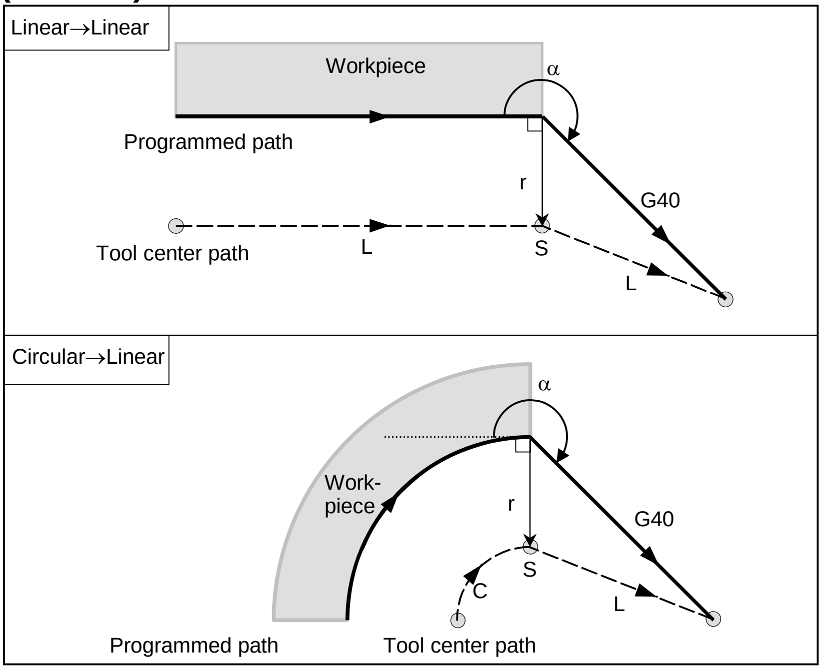
- Interruption of manual operation

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

6.7.4 Tool Movement in Offset Mode Cancel

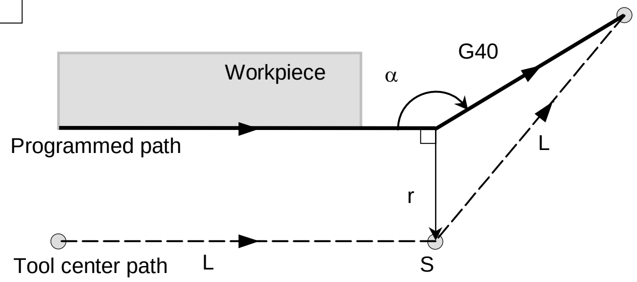
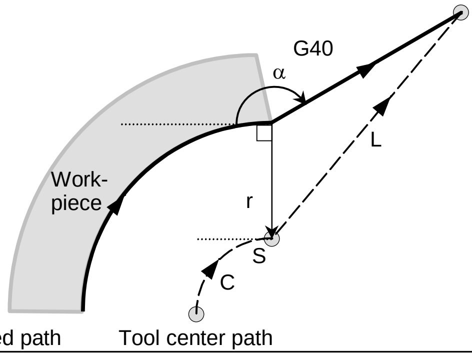
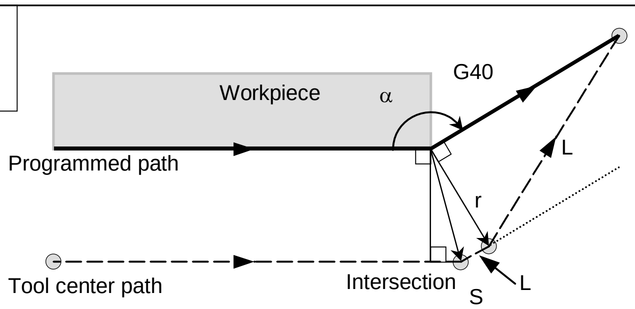
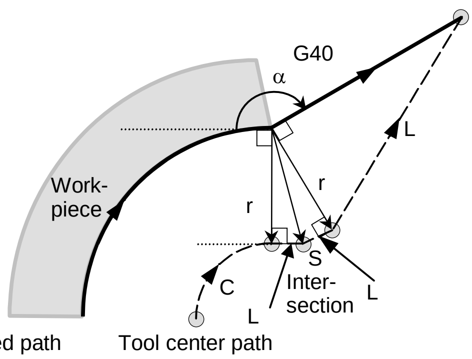
Explanation

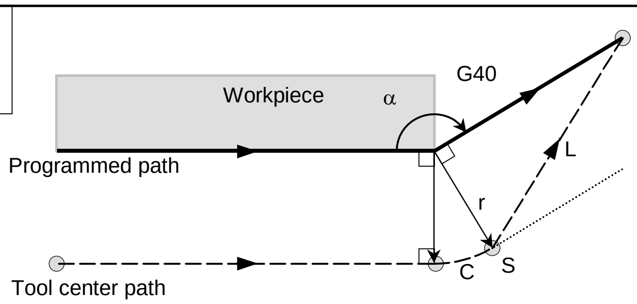
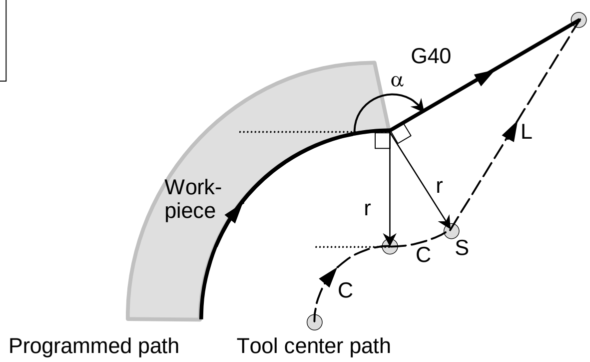
- **If the cancel block is a block with tool movement, and the tool moves around the inside (180° ≤ α)**



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

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

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

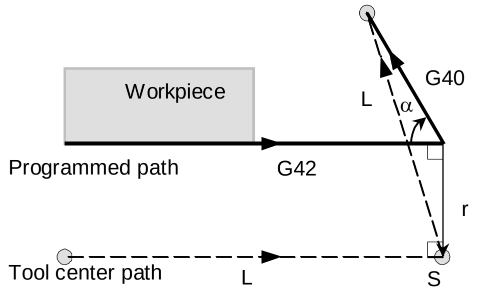
Type B	Linear→Linear (Circular connection type)	
	Circular→Linear (Circular connection type)	

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

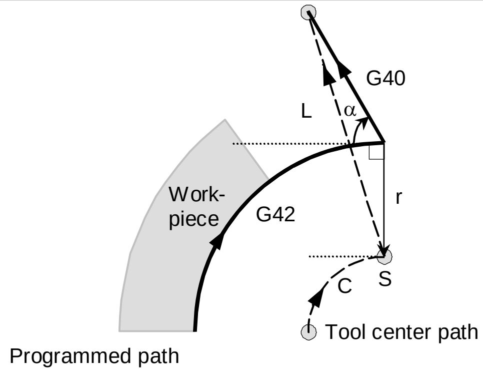
Tool path has two types A and B, and they are selected by bit 0 (SUP) of parameter No. 5003.

Type A

Linear→Linear

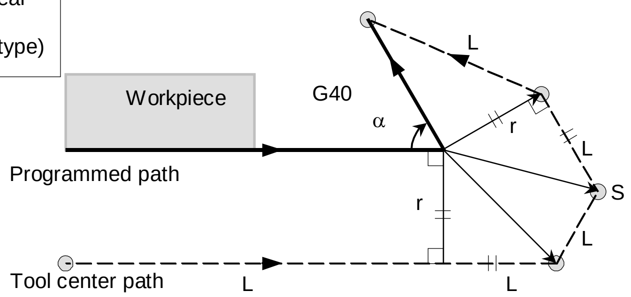


Circular→Linear

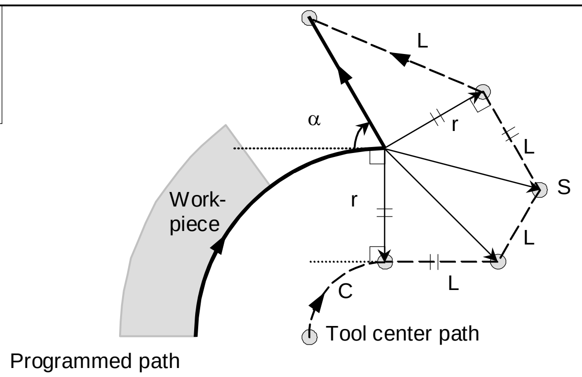


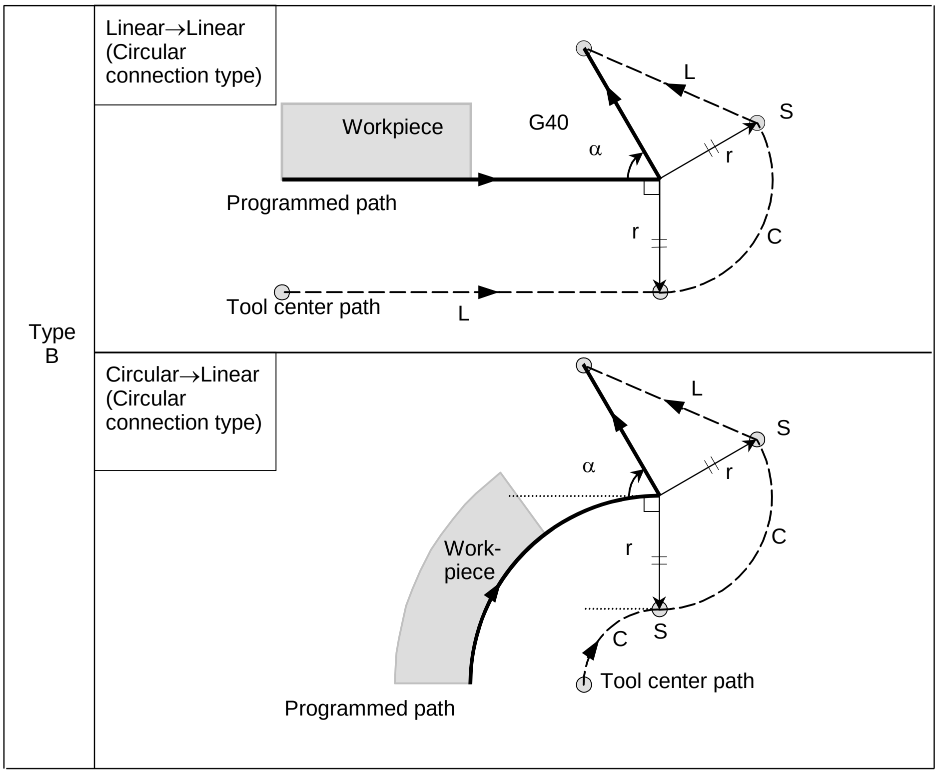
Type B

Linear→Linear
(Linear
connection type)

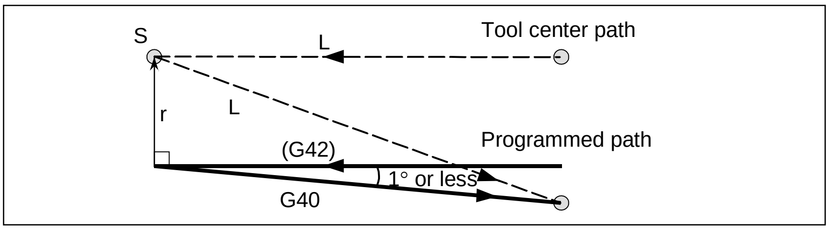


Circular→Linear
(Linear
connection type)





- 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°)



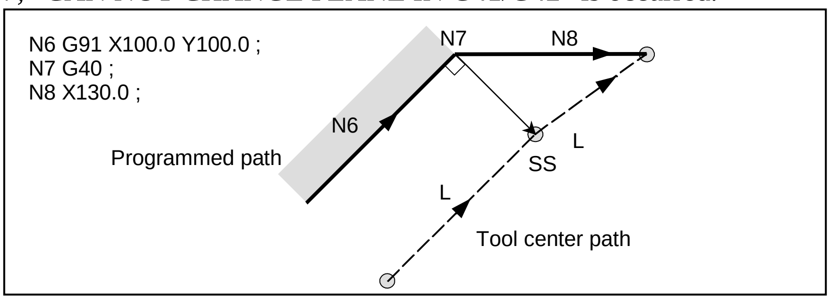
- A block without tool movement specified together with offset cancel

For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the tool radius · tool nose radius compensation value in the vertical direction. The tool does not operate in the cancel block. The remaining vectors are canceled with the next move command.

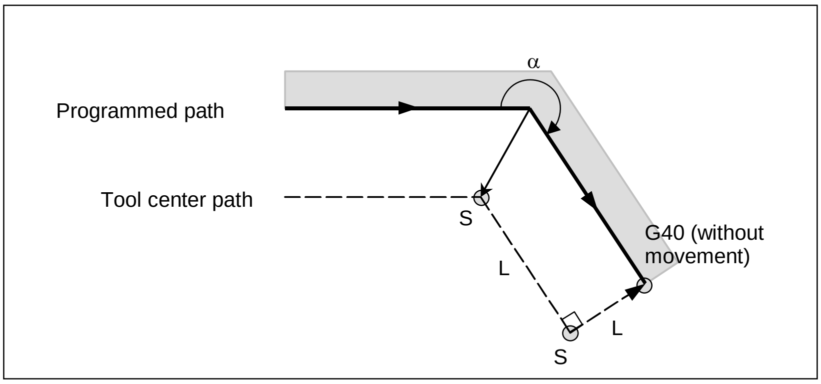
The compensation plane cannot be switched without canceling the remaining vector.

If the compensation plane (G17,G18,G19) is switched without canceling the remaining vector, the alarm PS0037, “CAN NOT CHANGE PLANE IN G41/G42” is occurred.



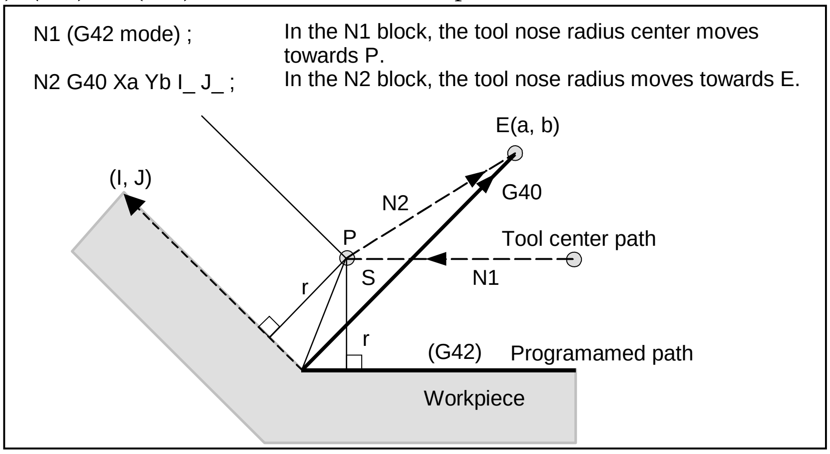
For type C

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

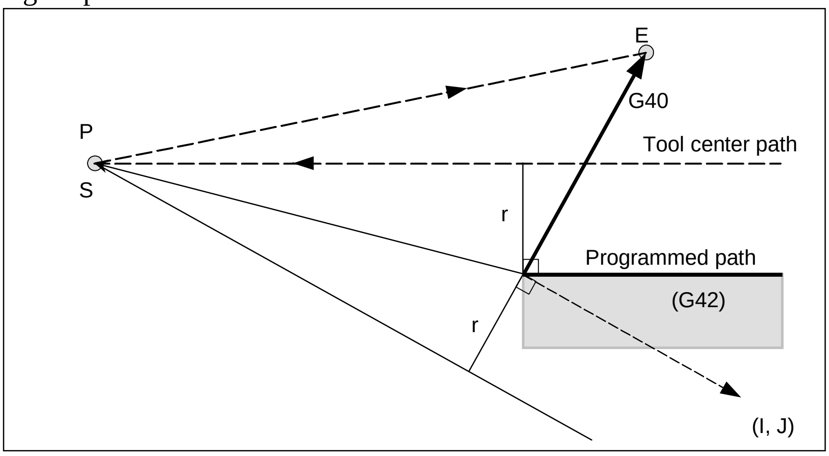


- Block containing G40 and I_J_K_
The previous block contains G41 or G42

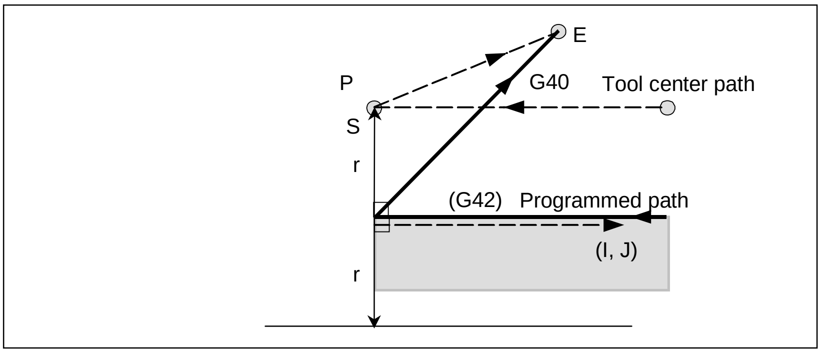
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end point determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



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

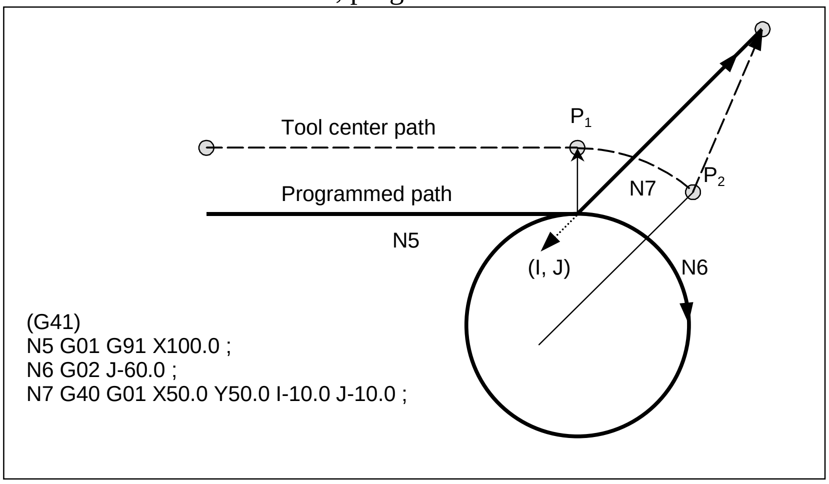


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



- **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.



6.7.5 Prevention of Overcutting Due to Tool Radius Compensation

Explanation

- **Machining a groove smaller than the diameter of the tool**

Since the cutter compensation forces the tool center path to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.

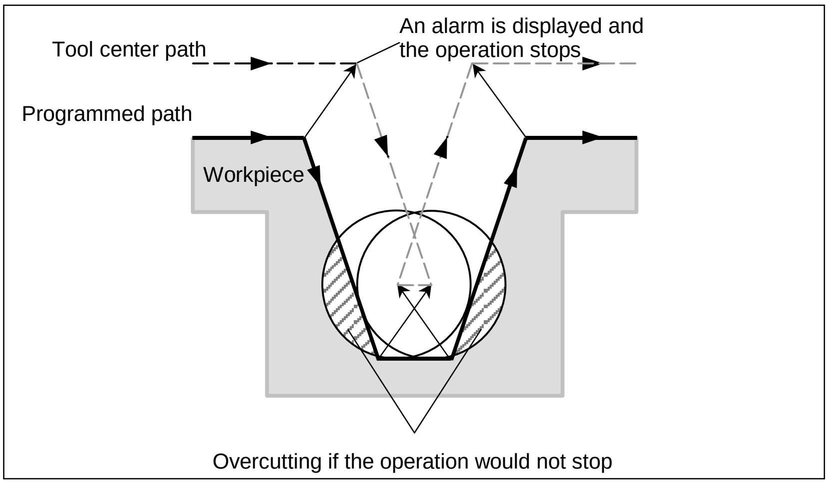


Fig. 6.7.5 (a) Machining a groove smaller than the diameter of the tool

- 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. 6.7.5 (b). If the step is smaller than the tool radius, the tool center path usually compensated as shown in Fig. 6.7.5 (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.

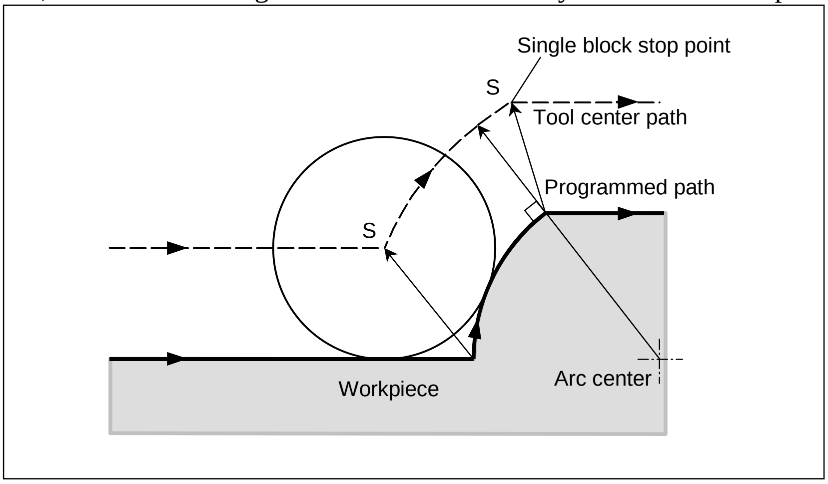


Fig. 6.7.5 (b) Machining a step larger than the tool radius

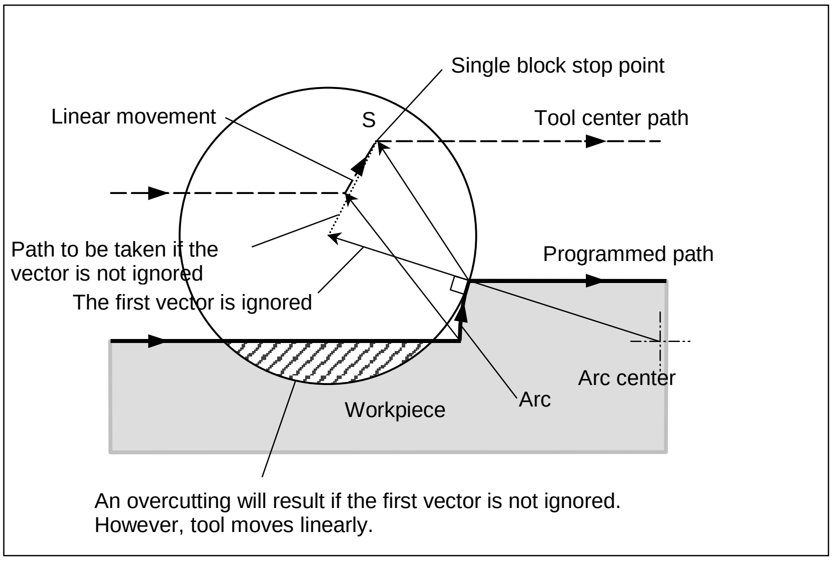


Fig. 6.7.5 (c) Machining a step smaller than the tool radius

- Starting compensation and cutting along the Z-axis

It is usually used such a method that the tool is moved along the Z axis after the cutter compensation (normally XY plane) is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.
Let us consider the program in the Fig. 6.7.5 (d), assuming the number of blocks to read in cutter compensation mode (parameter No. 19625) to be 3.

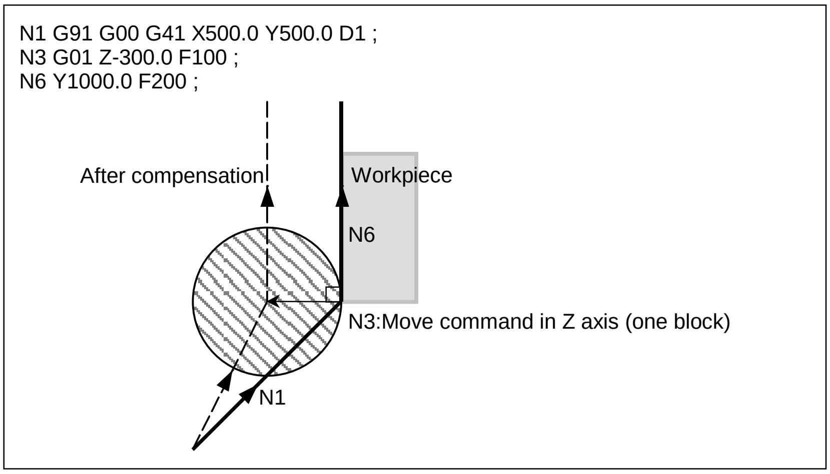


Fig. 6.7.5 (d)

In the program example in the Fig. 6.7.5 (d), when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the Fig. 6.7.5 (d).
Then, suppose that the block N3 (move command in Z axis) is divided into N3 and N5 in the Fig. 6.7.5 (e).

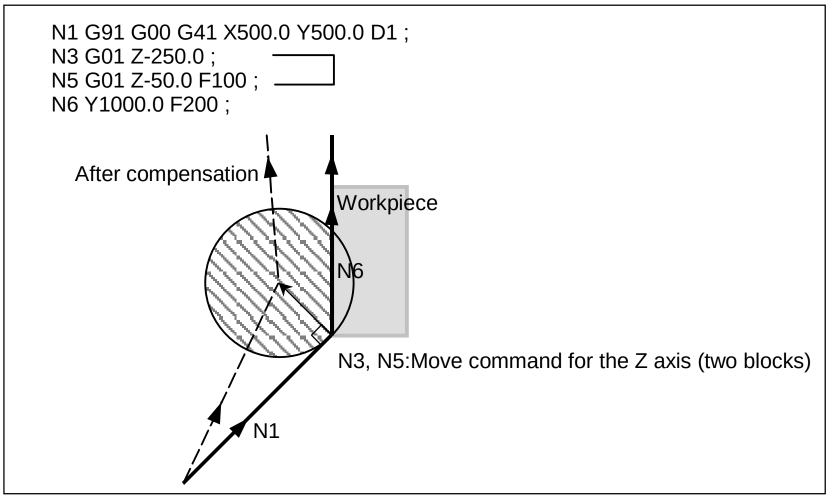


Fig. 6.7.5 (e)

At this time, because the number of blocks to read is 3, blocks up to N5 can be read at the start of N1 compensation, but block N6 cannot be read. As a result, compensation is performed only on the basis of the information in block N1, and a vertical vector is created at the end of the compensation start block. Usually, therefore, overcutting will result as shown in the Fig. 6.7.5 (e).
In such a case, it is possible to prevent overcutting by specifying a command with the exactly the same direction as the advance direction immediately before movement along the Z axis beforehand, after the tool is moved along the Z axis using the above rule.

```
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 ;
```

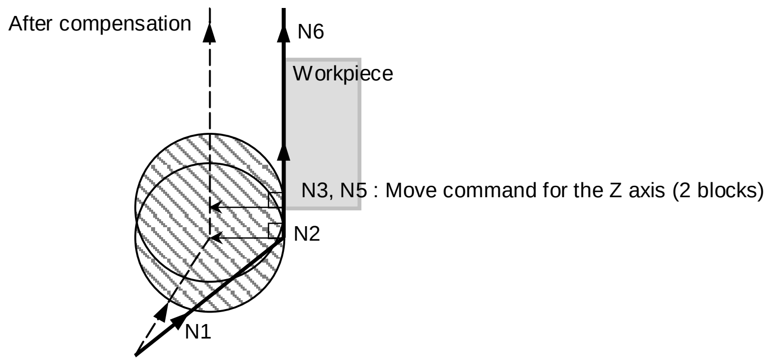


Fig. 6.7.5 (f)

As the block with sequence N2 has the move command in the same direction as that of the block with sequence N6, the correct compensation is performed.

Alternatively, it is possible to prevent overcutting in the same way by specifying an IJ type vector with the same direction as the advance direction in the start-up block, as in N1 G91 G00 G41 X500. Y500. I0 J1 D1;, after the tool has moved along the Z axis.

6.7.6 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

Explanation

- Condition under which an interference check is possible

To perform an interference check, it is necessary to read at least three blocks with tool movement. If, therefore, three or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as independent auxiliary function and dwell, are specified in succession, excessive or insufficient cutting may occur because an interference check fails. Assuming the number of blocks to read in offset mode, which is determined by parameter No. 19625, to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which an interference check is possible is

$(N - 3) \geq M$.

For example, if the maximum number of blocks to read in offset mode is 8, an interference check is possible even if up to five blocks without tool movement are specified. In this case, three adjacent blocks can be checked for interference, but any subsequent interference that may occur cannot be detected.

- Interference check method

Two interference check methods are available, direction check and circular angle check. Bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008 are used to specify whether to enable these methods.

CNV	CNC	Operation
0	0	An interference check is enabled, and a direction check and a circular angle check can be performed.
0	1	An interference check is enabled, and only a circular angle check is performed.
1	–	An interference check is disabled.

NOTE
There are no settings for performing a direction check only.

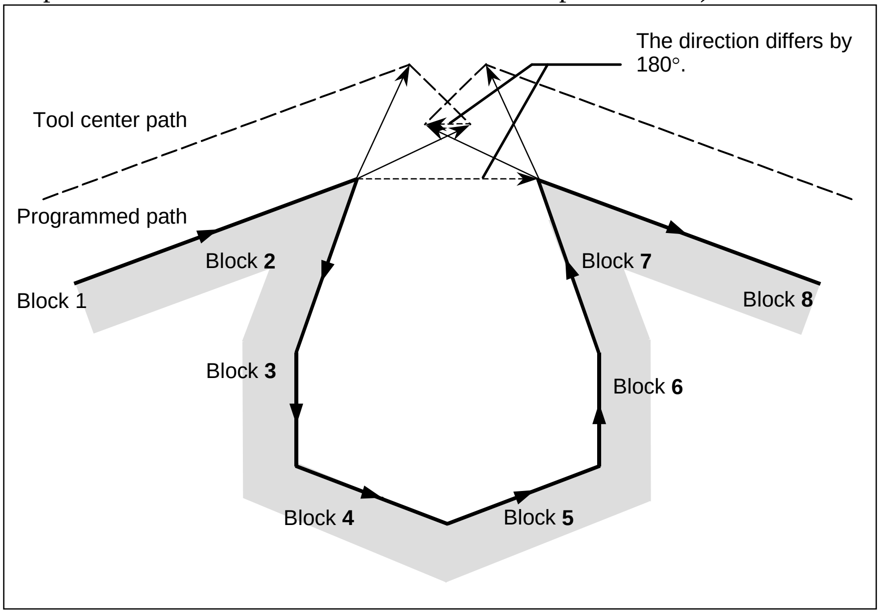
- Interference reference <1> (direction check)

Assuming the number of blocks to read during cutter compensation to be N, a check is first performed on the compensation vector group calculated in (block 1 - block 2) to be output this time and the compensation vector group calculated in (block N-1 - block N); if they intersect, they are judged to interfere. If no interference is found, a check is performed sequentially in the direction toward the compensation vector group to be output this time, as follows:

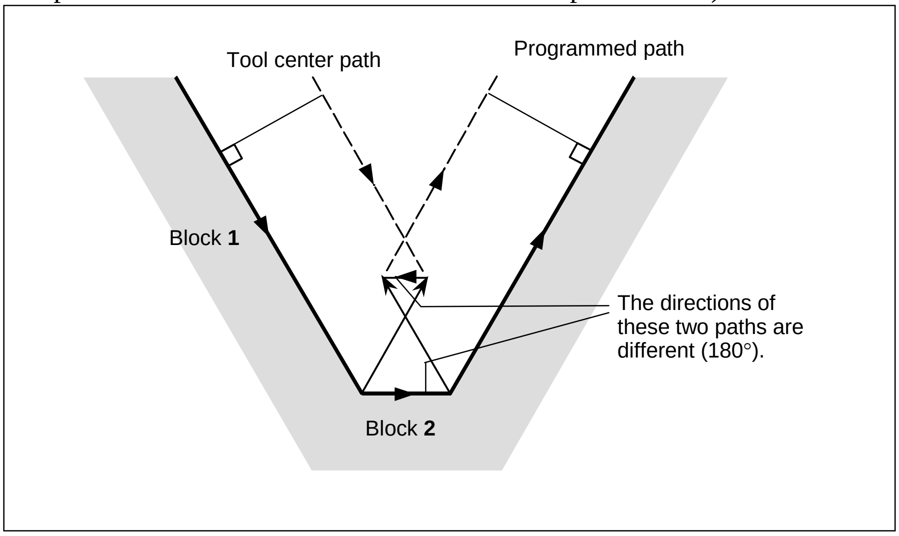
(Block 1 - block 2) and (block N-2 - block N-1)
(Block 1 - block 2) and (block N-3 - block N-2)
:
:
(Block 1 - block 2) and (block 2 - block 3)

Even if multiple number of compensation vector groups are generated, a check is performed on all pairs.
The judgment method is as follows: For a check on the compensation vector group in (block 1 - block 2) and those in (block N-1 - block N), the direction vector from the specified (end point of block 1) to the (end point of block N-1) is compared with the direction vector from the (point resulting from adding the compensation vector to be checked to the end of block 1) to the (point resulting from adding the compensation vector to be checked to the end of block N-1), and if the direction is 90^{o} or greater or 270^{o} or less, they are judged to intersect and interfere. This is called a direction check.

Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 7 end-point vector)



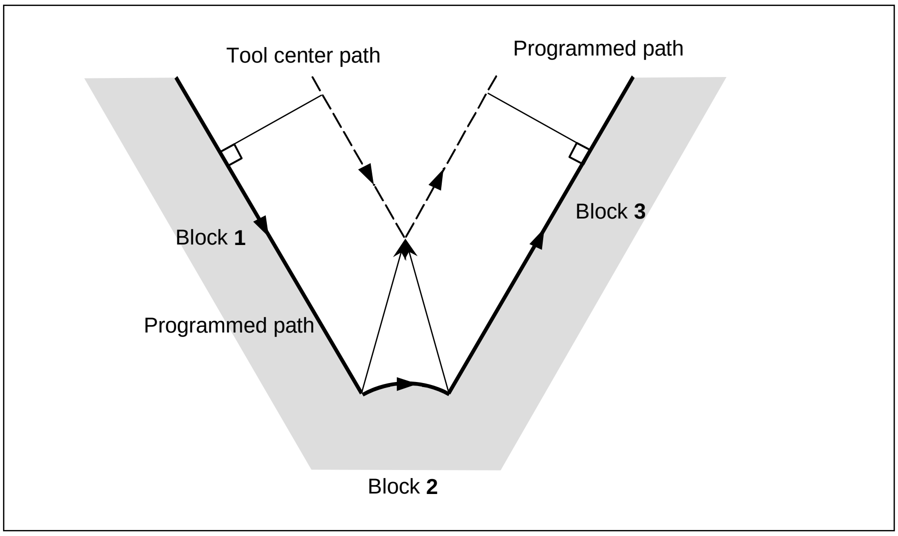
Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 2 end-point vector)



- Interference reference <2> (circular angle check)

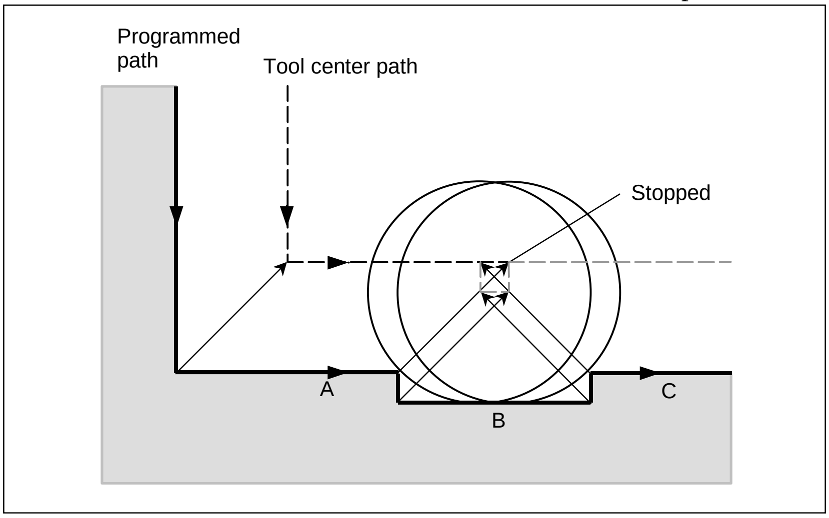
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed path and the circular angle of the start and end point of the post-compensation path, in addition to direction check <1>. If the difference is 180^{o} or greater, the blocks are judged to interfere. This is called a circular angle check.

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



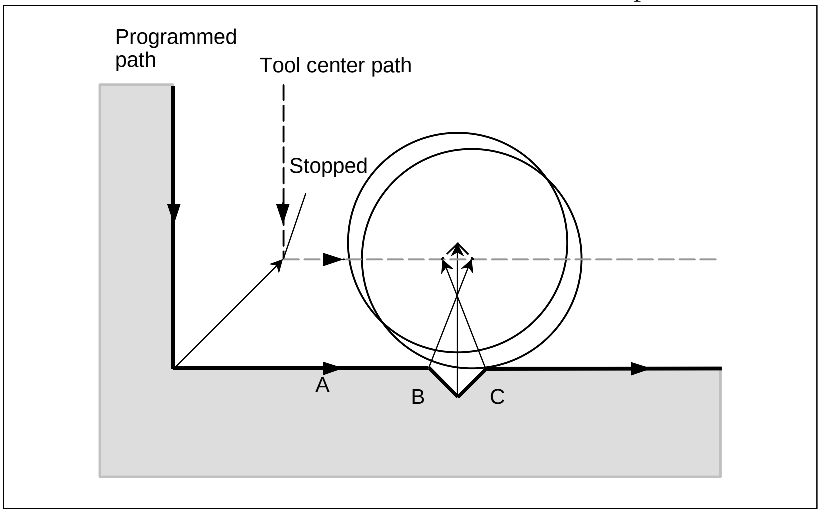
- When interference is assumed although actual interference does not occur

<1> Depression which is smaller than the tool radius · tool nose radius compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after the cutter compensation, the tool stops and an alarm is displayed.

<2> Groove which is smaller than the tool radius · tool nose radius compensation value



Like <1>, an alarm is displayed because of the interference as the direction is reverse in block B.

6.7.6.1 Operation to be performed if an interference is judged to occur

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

Parameter CAV	Function	Operation
0	Interference check alarm function	An alarm stop occurs before the execution of the block in which overcutting (interference) occurs.
1	Interference check avoidance function	The tool path is changed so that overcutting (interference) does not occur, and processing continues.

6.7.6.2 Interference check alarm function

- Interference other than those between adjacent three blocks

If the end-point vector of block 1 and the end-point vector of block 7 are judged to interfere as shown in the Fig. 6.7.6.2 (a), an alarm will occur before the execution of block 1 so that the tool stops. In this case, the vectors will not be erased.

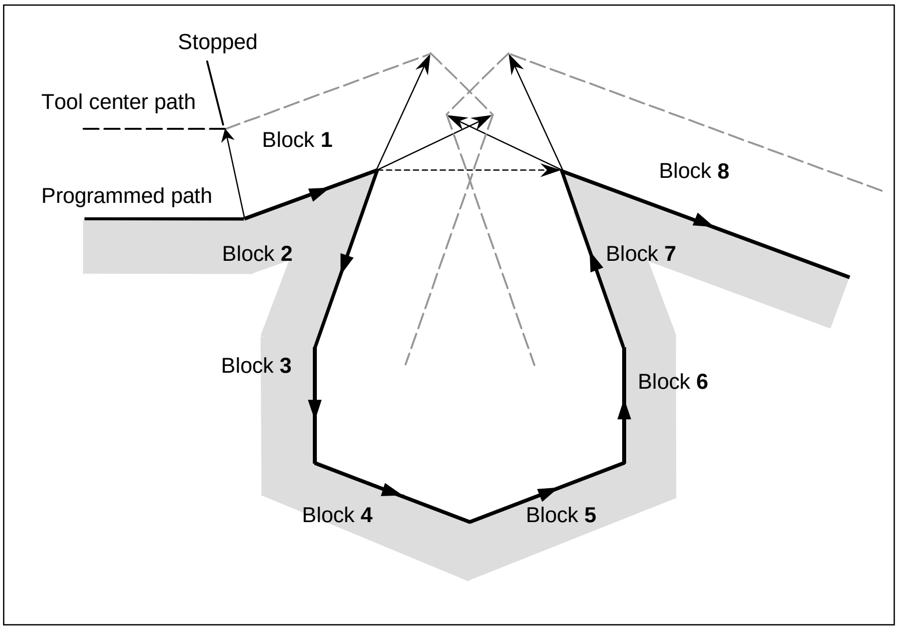


Fig. 6.7.6.2 (a)

- Interference between adjacent three blocks

If an interference is judged to occur between adjacent three blocks, the interfering vector, as well as any vectors existing inside of it, is erased, and a path is created to connect the remaining vectors. In the Example shown in the Fig. 6.7.6.2 (b), V_2 and V_5 interfere, so that V_2 and V_5 are erased, so are V_3 and V_4, which are inside of them, and V_1 is connected to V_6. The operation during this time is linear interpolation.

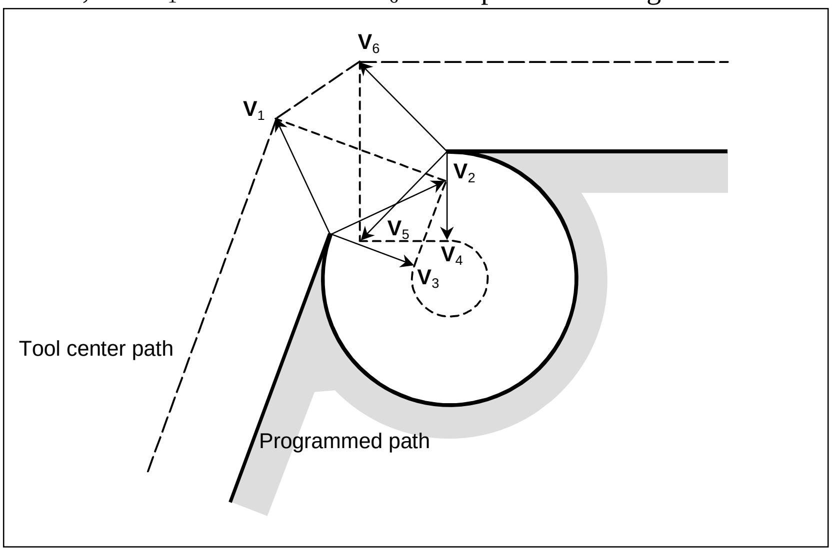


Fig. 6.7.6.2 (b)

If, after vector erasure, the last single vector still interferes, or if there is only one vector at the beginning and it interferes, an alarm will occur immediately after the start of the previous block (end point for a single block) and the tool stops. In the Example shown in the Fig. 6.7.6.2 (c), V_2 and V_3 interfere, but, even after erasure, an alarm will occur because the final vectors V_1 and V_4 interfere.

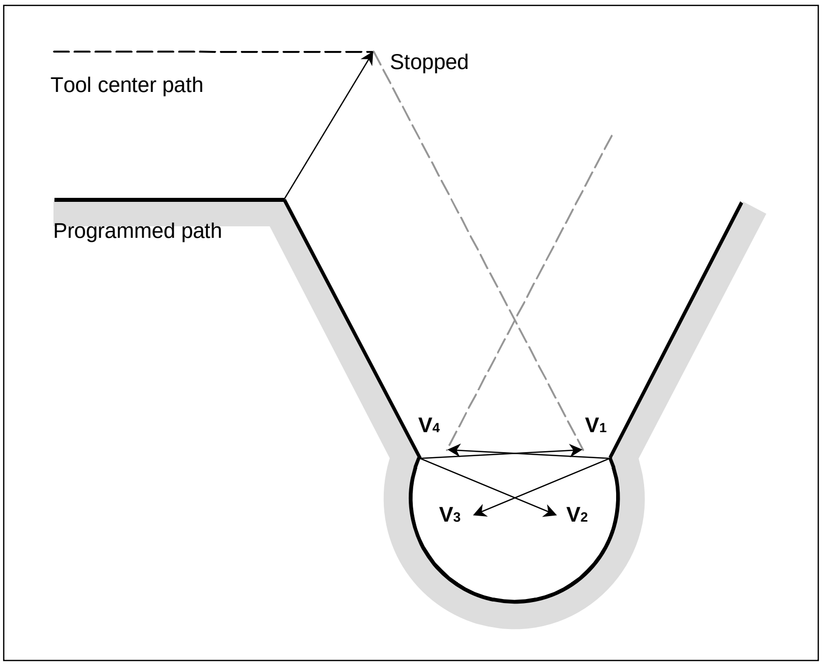


Fig. 6.7.6.2 (c)

6.7.6.3 Interference check avoidance function

Overview

If a command is specified which satisfies the condition under which the interference check alarm function generates an interference alarm, this function suppresses the generation of the interference alarm, but causes a new compensation vector to be calculated as a path for avoiding interference, thereby continuing machining. For the path for avoiding interference, insufficient cutting occurs in comparison with the programmed path. In addition, depending on the specified figure, no path for avoiding interference can be determined or the path for avoiding interference may be judged dangerous. In such a case, an alarm stop will occur. For this reason, it is not always possible to avoid interference for all commands.

- Interference avoidance method

Let us consider a case in which an interference occurs between the compensation vector between (block 1 - block 2) and the compensation vector between (block N-1 - block N). The direction vector from the end point of block 1 to the end point of block N-1 is called a gap vector. At this time, a post-compensation intersection vector between (block 1 - gap vector) and a post-compensation intersection vector between (gap vector - block N) is determined, and a path connecting them is created.

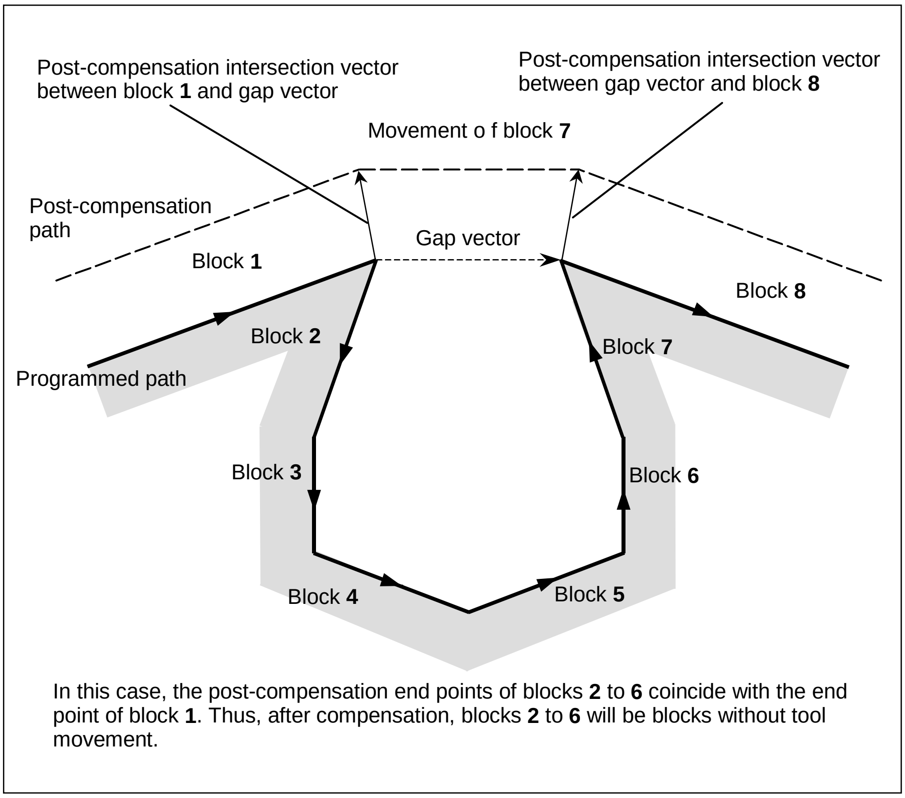


Fig. 6.7.6.3 (a)

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

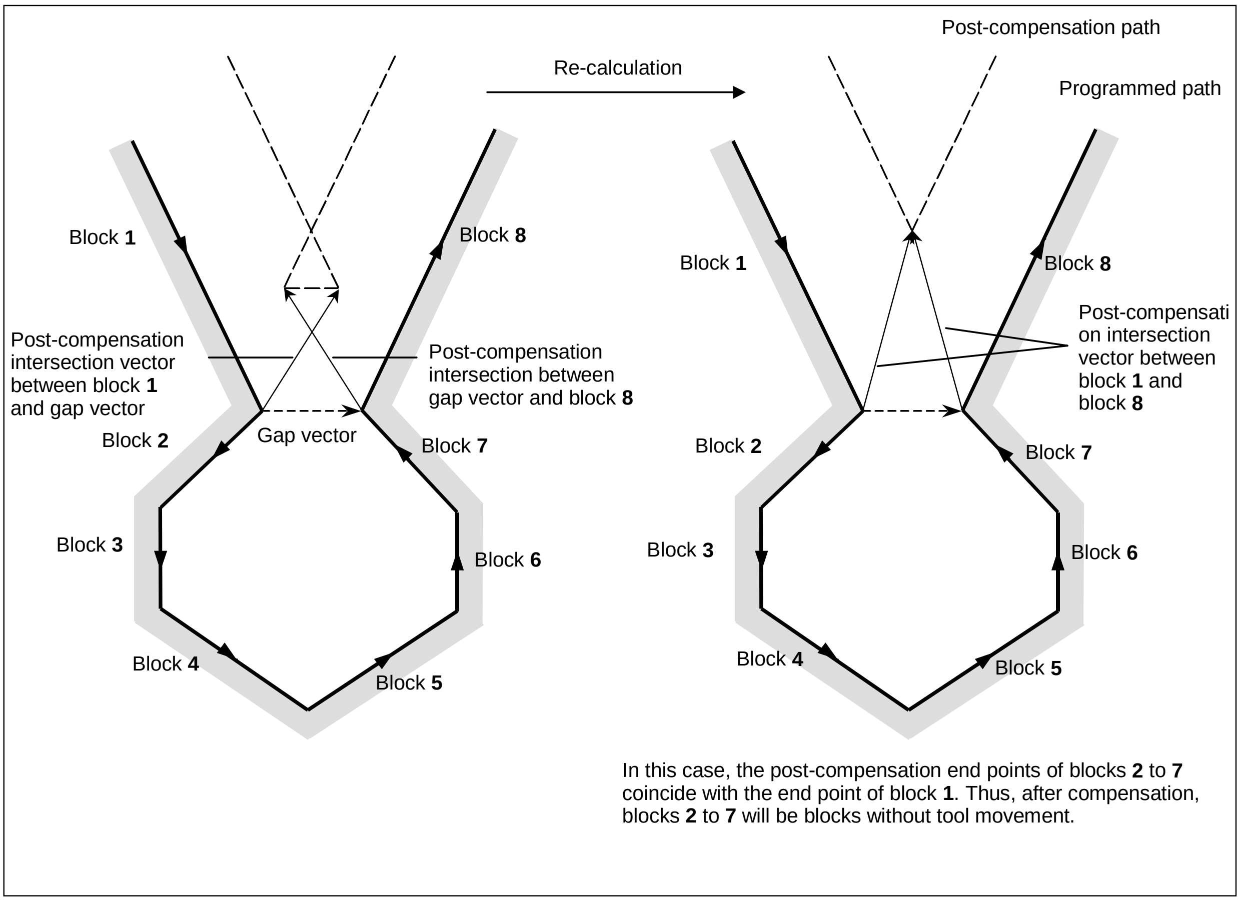


Fig. 6.7.6.3 (b)

If the tool radius/tool nose radius compensation value is greater than the radius of the specified arc as shown in the Fig. 6.7.6.3 (c), and a command is specified which results in compensation with respect to the inside of the arc, interference is avoided by performing intersection calculation with an arc command being assumed a linear one. In this case, avoided vectors are connected with linear interpolation.

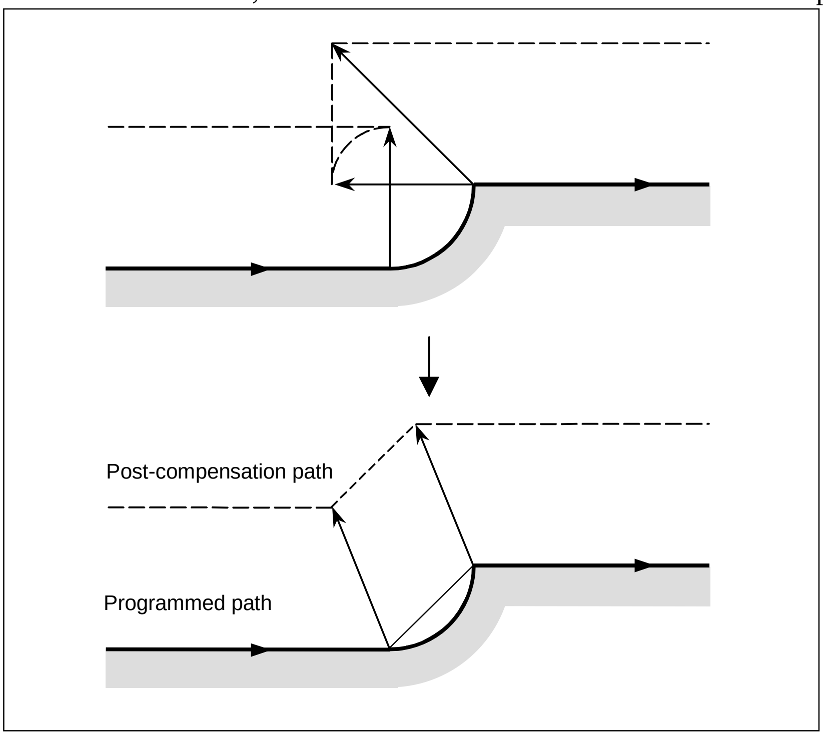


Fig. 6.7.6.3 (c)

- If no interference avoidance vector exists

If the parallel pocket shown in the Fig. 6.7.6.3 (d) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are parallel to each other, no intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop.

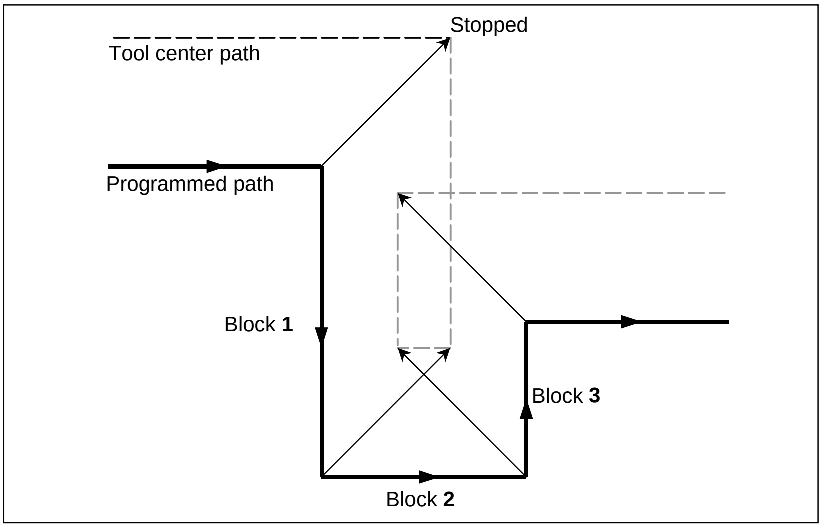


Fig. 6.7.6.3 (d)

If the circular pocket shown in the Fig. 6.7.6.3 (e) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are circular, no post-compensation intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop, as in the previous example.

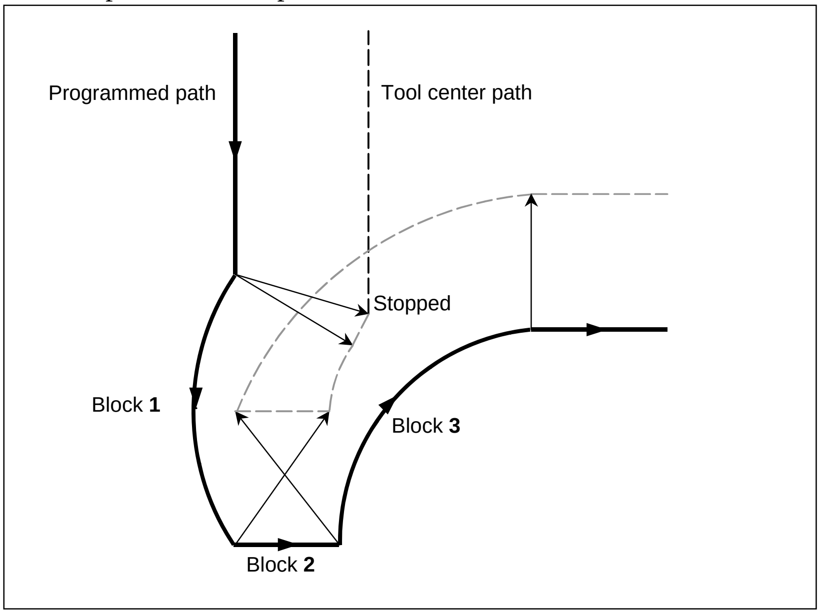


Fig. 6.7.6.3 (e)

- If it is judged dangerous to avoid interference

If the acute-angle pocket shown in the Fig. 6.7.6.3 (f) is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the movement direction of the post-avoidance path extremely differs from the previously specified direction. If the post-avoidance path extremely differs from that of the original command (90° or greater or 270° or less), interference avoidance operation is judged dangerous; an alarm will occur immediately before block 1 and the tool will stop.

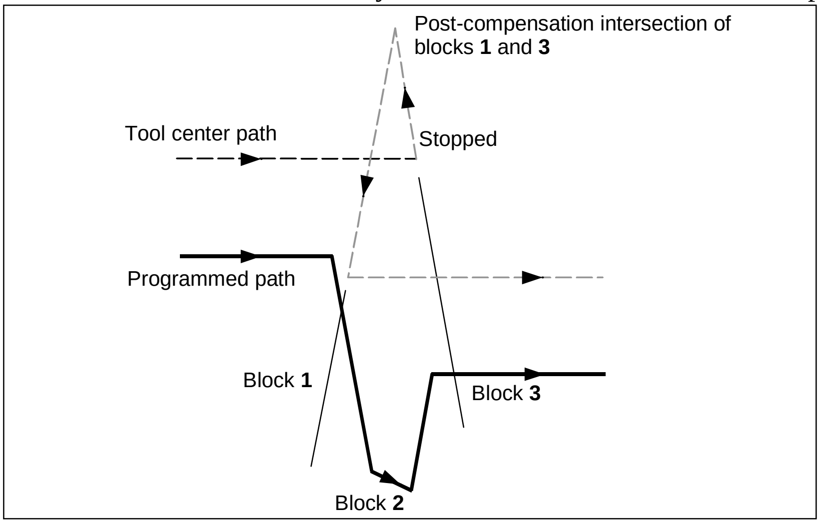


Fig. 6.7.6.3 (f)

If a pocket in which the bottom is wider than the top, such as that shown in the Fig. 6.7.6.3 (g), is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the relation between blocks 1 and 3 is judged an outer one, the post-avoidance path results in overcutting as compared with the original command. In such a case, interference avoidance operation is judge dangerous; an alarm will occur immediately before block 1 and the tool will stop.

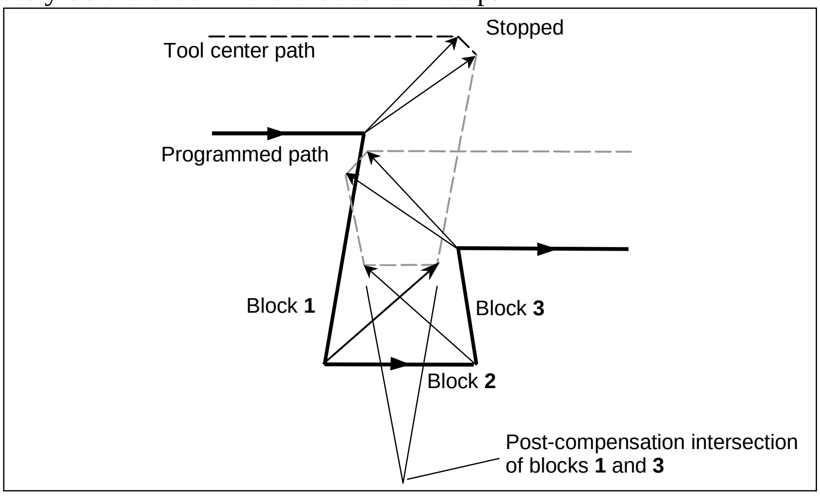


Fig. 6.7.6.3 (g)

- If further interference with an interference avoidance vector occurs

If the pocket shown in the Fig. 6.7.6.3 (h) is to be machined, if the number of blocks to read is 3, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, however, the end-point vector of block 3 that is to be calculated next further interferes with the previous interference avoidance vector.

If a further interference occurs to the interference avoidance vector once created and output, the movement in the block will not be performed; an alarm will occur immediately before the block and the tool will stop.

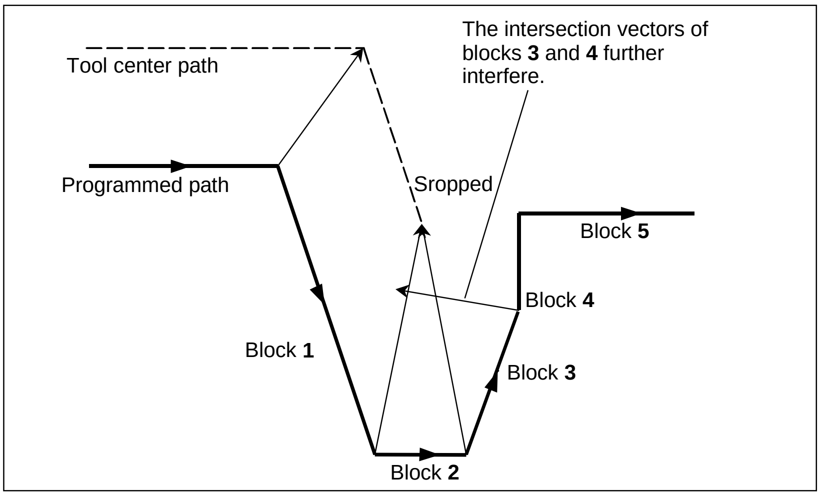


Fig. 6.7.6.3 (h)

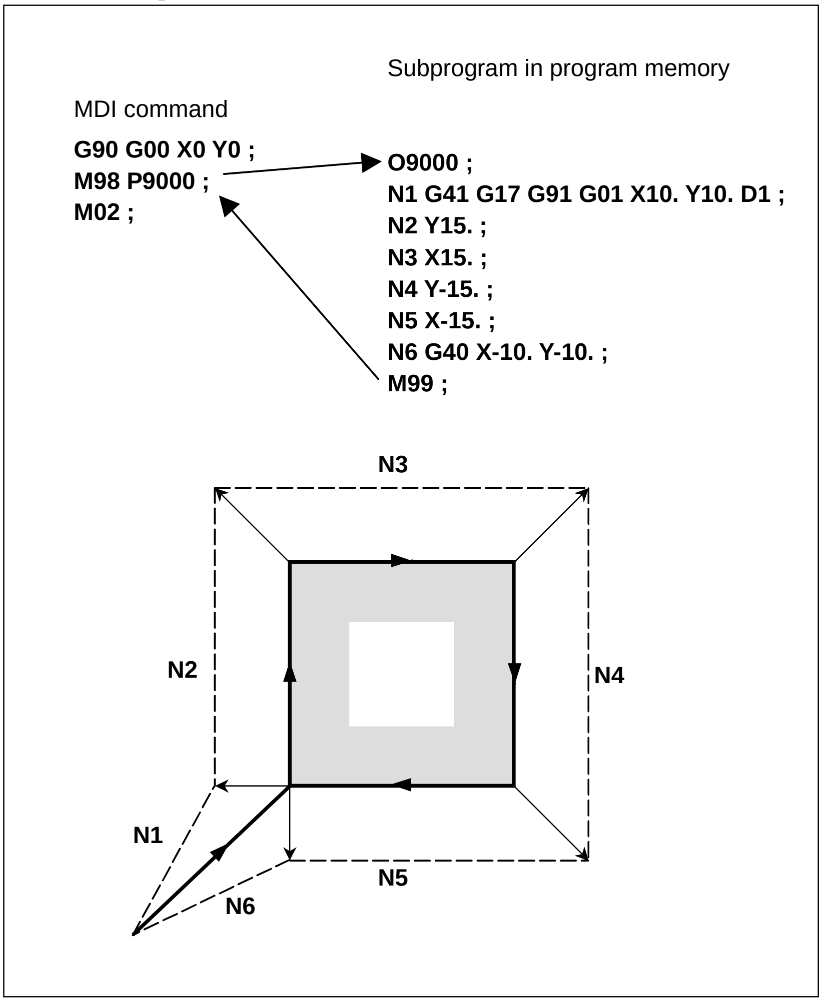
> **NOTE**
> 1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting bit 6 (NAA) of parameter No. 19607 appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", however, it is not possible to avoid an alarm regardless of the setting of this parameter.
> 2 If a single block stop occurs during interference avoidance operation, and an operation is performed which differs from the original movement, such as manual intervention, MDI intervention, tool radius / tool nose radius compensation value change, intersection calculation is performed with a new path. If such an operation is performed, therefore, an interference may occur again although interference avoidance has been performed once.

6.7.7 Tool Radius / Tool Nose Radius Compensation for Input from MDI

Explanation

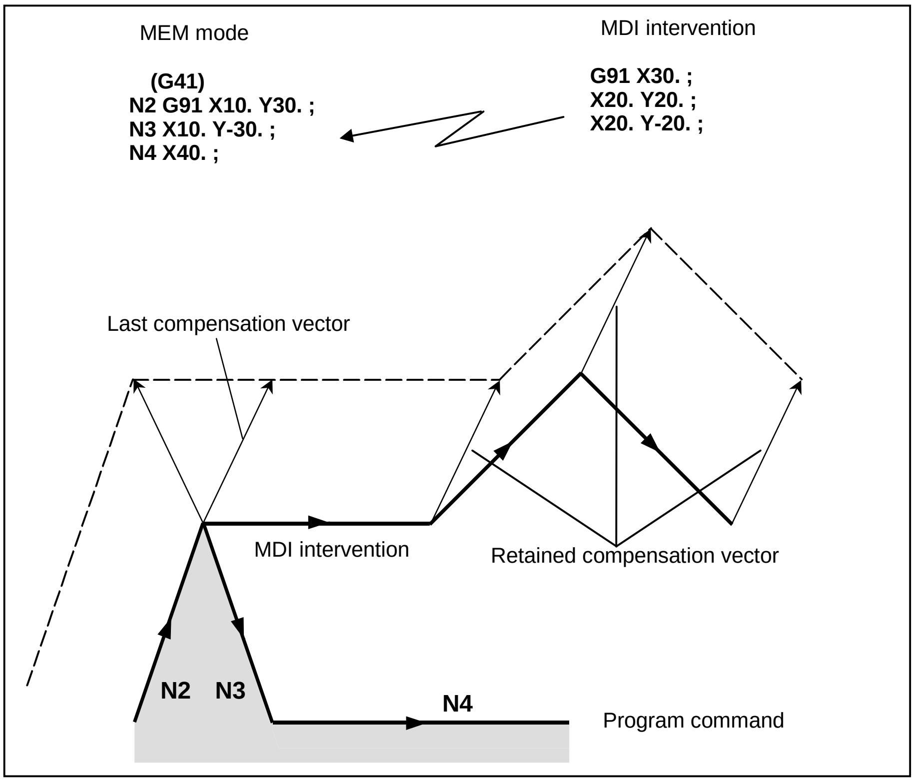
- MDI operation

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



- MDI intervention

If MDI intervention is performed, that is, if a single block stop is performed to enter the automatic operation stop state in the middle of memory operation, DNC operation, and the like, and a program command is specified in MDI mode to make a cycle start, cutter compensation does not perform intersection calculation, retaining the last compensation vector before the intervention.


MEM mode
(G41)
N2 G91 X10. Y30. ;
N3 X10. Y-30. ;
N4 X40. ;
MDI intervention
G91 X30. ;
X20. Y20. ;
X20. Y-20. ;
Last compensation vector
MDI intervention
Retained compensation vector
N2
N3
N4
Program command

6.8 VECTOR RETENTION (G38)

In tool radius / tool nose radius compensation, by specifying G38 in offset mode, it is possible to retain the compensation vector at the end point of the previous block, without performing intersection calculation.

Format

(In offset mode)
G38 IP_ ;

IP: Value specified for axial movement

Explanation

- Vector retention

By specifying the above command, a vector is created at the end point of the block immediately preceding the G38 block, vertical to that block. In the G38 block, the vertical vector output in the previous block is retained. G38 is a one-shot G code. With the next move command without a G38 command, the compensation vector is re-created.

Limitation

- Mode

Specify G38 in either G00 or G01 mode. If it is specified in G02 or G03 (circular interpolation) mode, a radial error may occur at the start and end points.

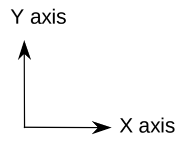
- Start-up/cancel

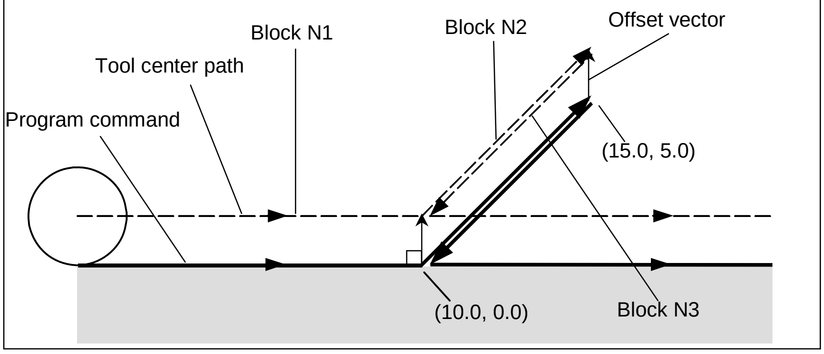
In start-up/cancel, the operation is as described in Subsections, “Tool Movement in Start-up” and “Tool Movement in Offset Mode Cancel”. Thus, G38 cannot be specified in the following blocks:

1) Start-up command (G41 or G42) block
2) Cancel command (G40) block
3) Block immediately preceding the cancel command (G40) block

Example

```
          :
          :       (In offset mode)
   (G90)
N1 G38 X10.0 Y0.0 ;
N2 G38 X15.0 Y5.0 ;
N3 G38 X10.0 Y0.0 ;
N4     X20.0 ;
          :
          :
```





6.9 CORNER CIRCULAR INTERPOLATION (G39)

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

Format

In offset mode
G39 ;

$$\mathbf{G39}\begin{Bmatrix}\mathbf{I\_\ J\_}\\ \mathbf{I\_\ K\_}\\ \mathbf{J\_\ K\_}\end{Bmatrix}\ \mathbf{;}$$

Explanation

- Corner circular interpolation

When the command indicated above is specified, corner circular interpolation in which the radius equals compensation value can be performed. G41 or G42 preceding the command determines whether the arc is clockwise or counterclockwise. G39 is a one-shot G code.

- G39 without I, J, or K

When G39 is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block.

- G39 with I, J, and K

When G39 is specified with I, J, and K, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the vector defined by the I, J, and K values.

Limitation

- Move command

In a block containing G39, no move command can be specified. Otherwise, an alarm will occur.

- Inner corner

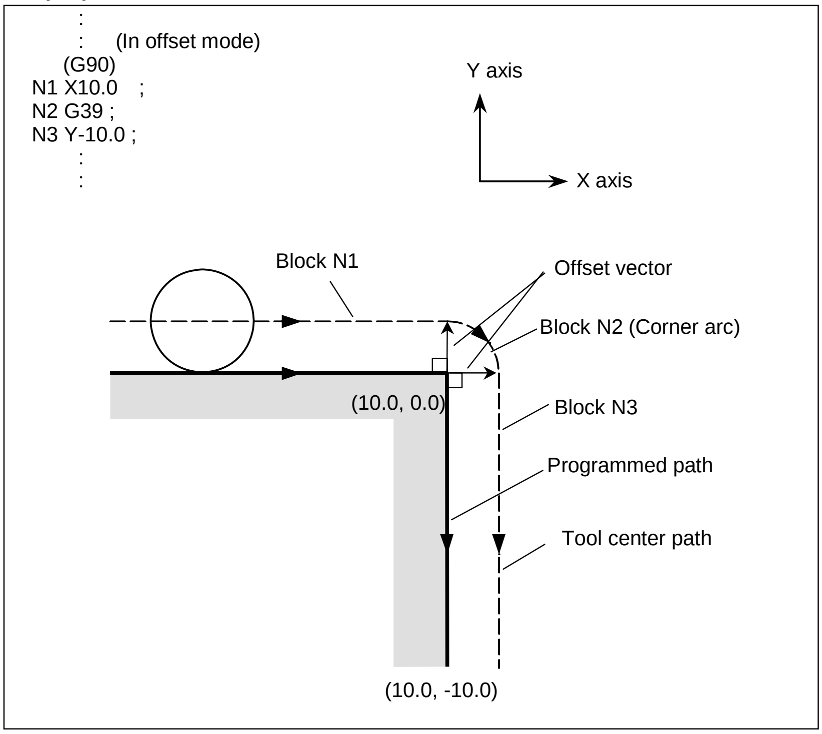
In an inner corner block, G39 cannot be specified. Otherwise, overcutting will occur.

- Corner arc velocity

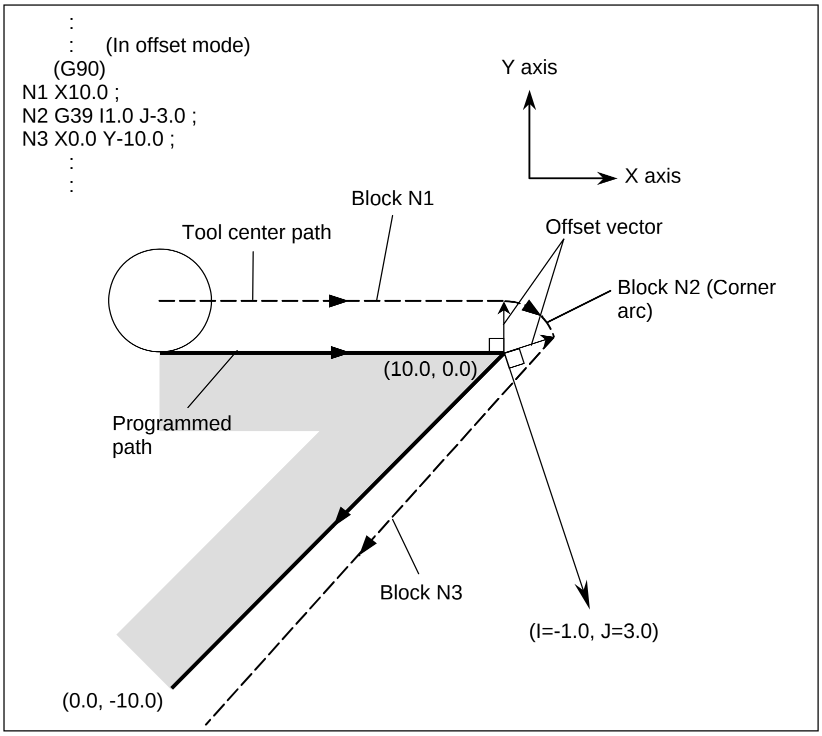
If a corner arc is specified with G39 in G00 mode, the corner arc block velocity will be that of the F command previously specified. If G39 is specified in a state in which no F command has never been specified, the velocity of the corner arc block will be that specified with parameter No. 1411.

Example

- G39 without I, J, or K



- G39 with I, J, and K



6.10 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Tool compensation values include tool geometry compensation values and tool wear compensation (Fig. 6.10 (a)).

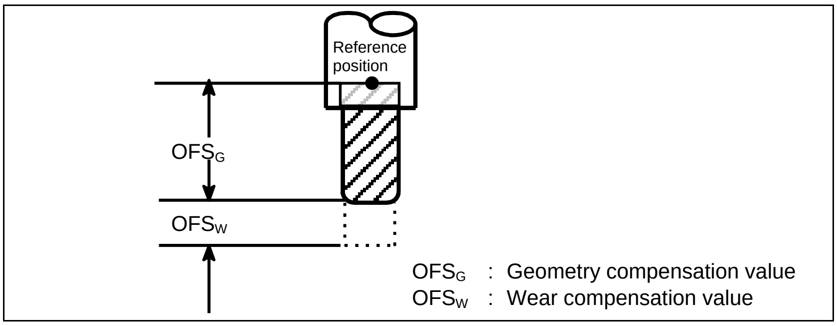


Fig. 6.10 (a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the MDI unit (see section III-11.1.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, cutter compensation, or the tool offset.
Two types of tool compensation memories are available according to the compensation value configuration: tool compensation memory A and C. One of the types can be selected.

Explanation

- Tool compensation memory A (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

The total number of items of tool compensation data is 400.

Format

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

For tool compensation memory A

G10 L11 P_ R_ Q_ ;

P_ : Tool compensation number
R_ : Tool compensation value
Q_ : Imaginary tool nose number

For tool compensation memory C

G10 L_ P_ R_ Q_ ;

L_ : Type of compensation memory
- L10 : Geometry compensation value corresponding to an H code
- L11 : Wear compensation value corresponding to an H code
- L12 : Geometry compensation value corresponding to a D code
- L13 : Wear compensation corresponding to a D code
- L110 : Geometry compensation value corresponding to a D code (for corner R offset)
- L111 : Wear compensation corresponding to a D code (for corner R offset)

P_ : Tool compensation number
R_ : Tool compensation value
Q_ : Imaginary tool nose number

By specifying G10, a tool compensation value can be set or modified.
When G10 is specified by absolute input (G90), the specified value is used as the new tool compensation value.
When incremental input (G91) is used, a specified value added to the tool compensation value currently set is used as the new tool compensation value.

NOTE
1 Address R follows the increment system for tool offset values.
2 If L is omitted for compatibility with the conventional CNC format, or L1 is specified, the same operation as when L11 is specified is performed.
3 Set a imaginary tool nose number when the cutter compensation function is specified and a imaginary tool nose direction is used.

6.11 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.

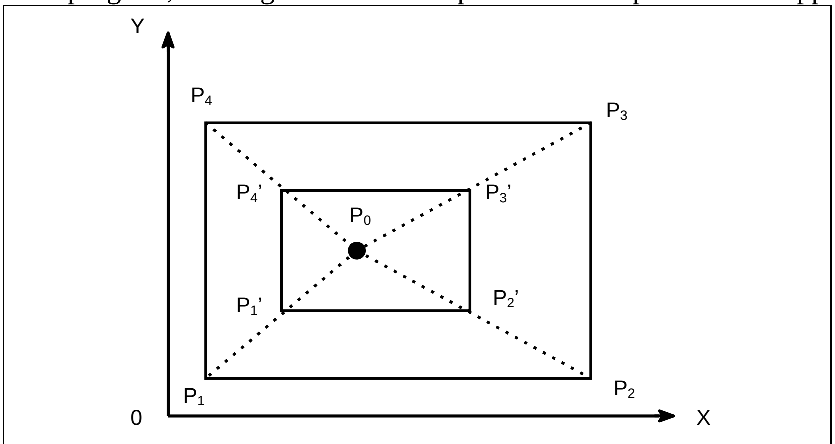


Fig. 6.11 (a) Scaling

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 bit 6 (XSC) of parameter No. 5400 = 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 bit 6 (XSC) of parameter No. 5400 = 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, "ILLEGAL SCALE RATE" 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, "ILLEGAL USE OF DECIMAL POINT" will occur.
A negative value cannot be specified as the magnification P. If a negative value is specified, alarm PS0006, "ILLEGAL USE OF MINUS SIGN" 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. 5400 to 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.

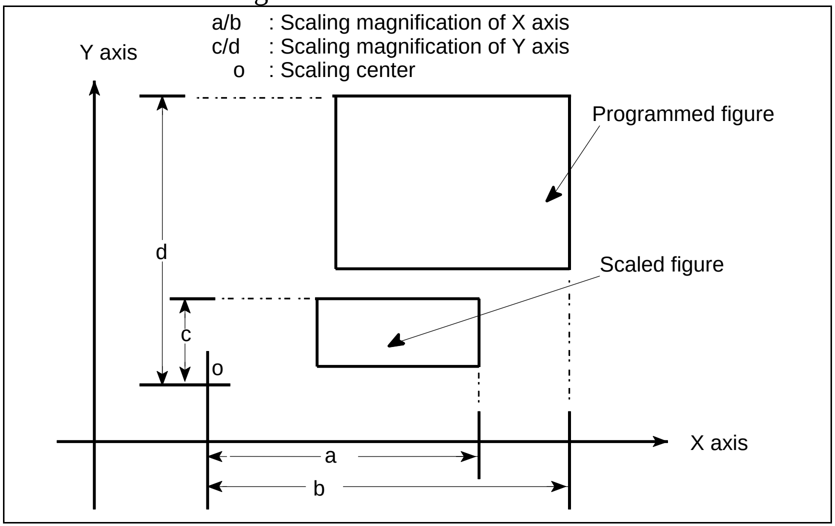


Fig. 6.11 (b) Scaling of each axis

> ⚠ **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.)

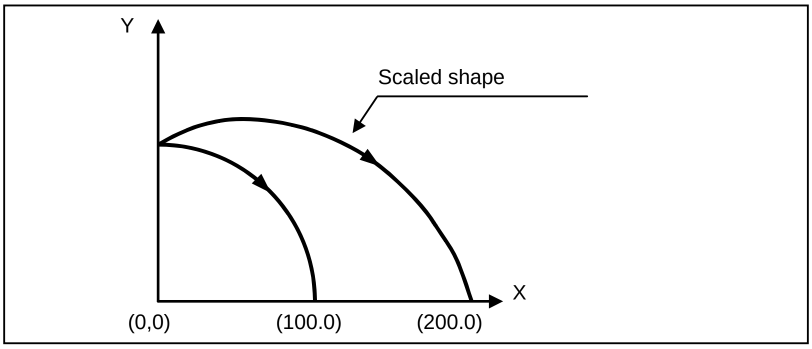


Fig. 6.11 (c) Scaling for circular interpolation

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 ;  (×3 in the X direction and ×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 ;
```

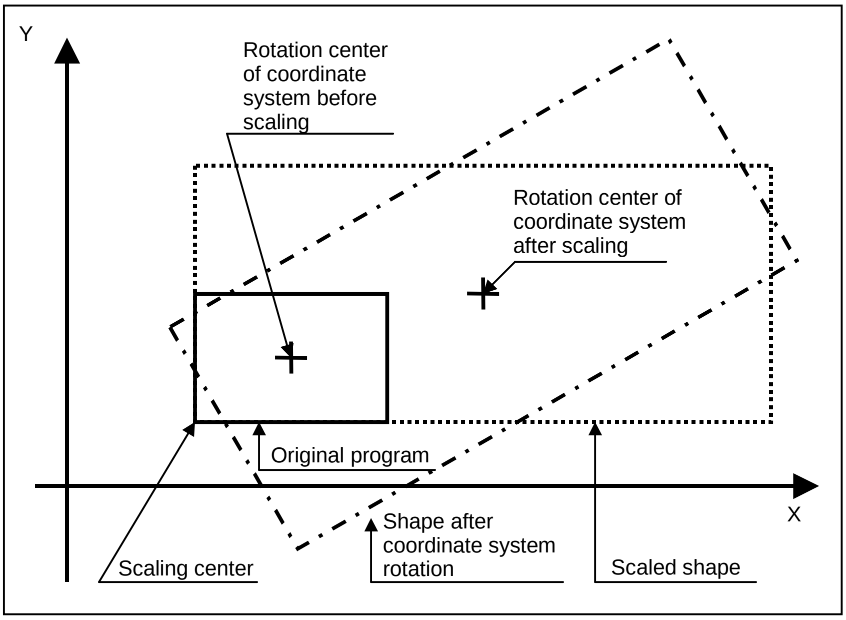


Fig. 6.11 (d) Scaling and coordinate system rotation

Limitation

- Tool compensation

This scaling is not applicable to tool radius · tool nose radius compensation values, tool length compensation values, and tool offset values (Fig. 6.11 (e)).

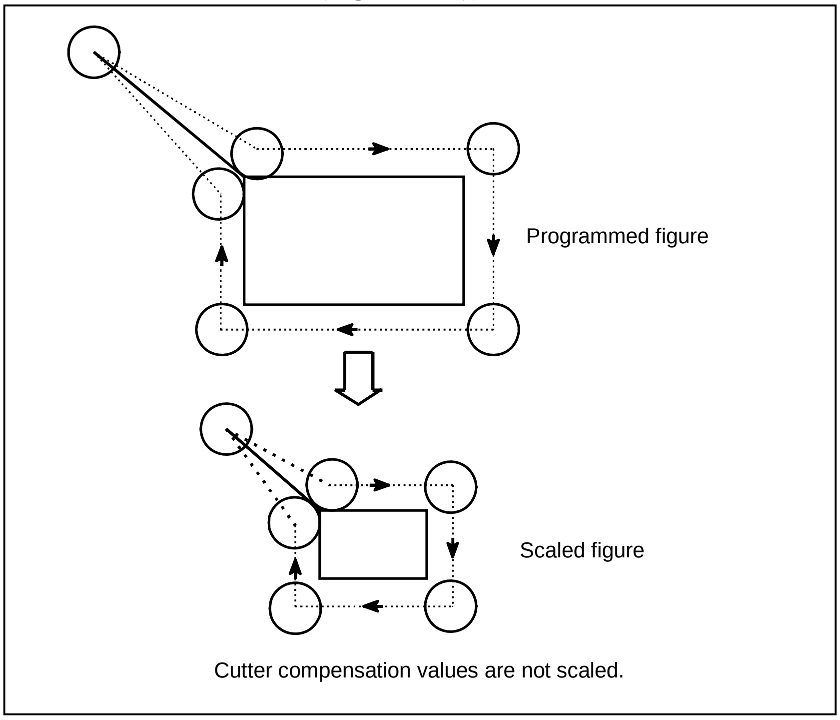


Fig. 6.11 (e) Scaling during cutter compensation

- 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.
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;
```

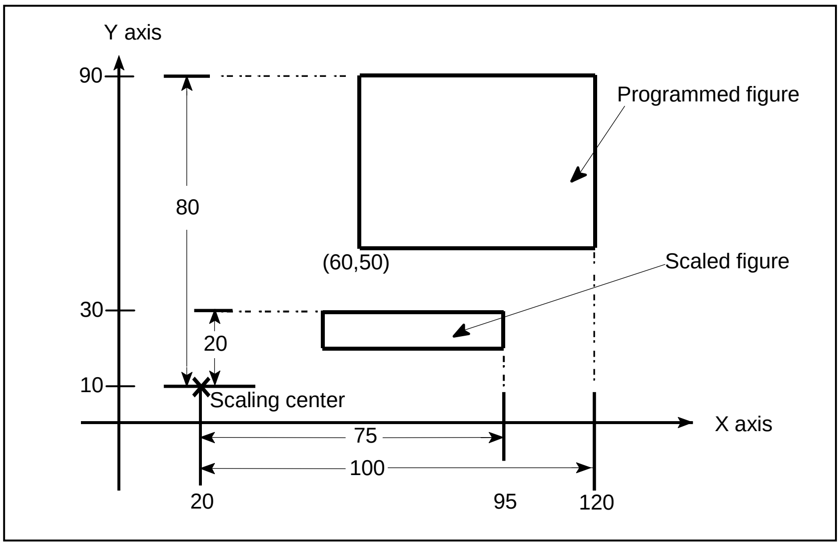


Fig. 6.11 (f) Program example of scaling in each axis

6.12 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.

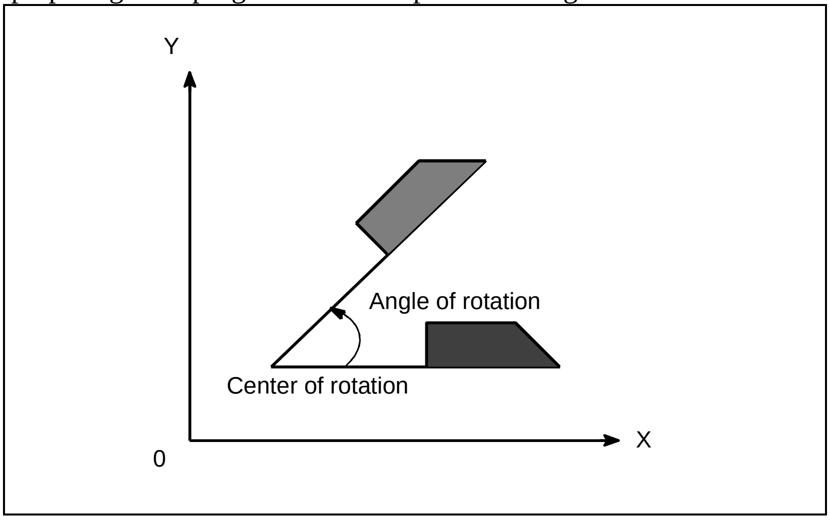


Fig. 6.12 (a) Coordinate system rotation

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**

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. Bit 0 (RIN) of parameter No. 5400 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).

Least input increment : 0.001 deg

Valid data range : -360,000 to 360,000

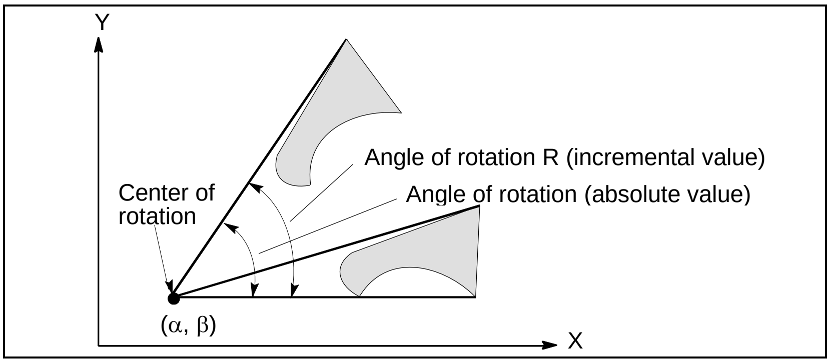


Fig. 6.12 (b) Coordinate system rotation

> **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. 6.12 (c)).

- Center of rotation

When α_β_ is not programmed, the tool position when G68 was programmed is assumed as the center of rotation.

- Angular displacement

When R_ is not specified, the value specified in parameter No. 5410 is assumed as the angular displacement.
To specify angular displacement (R_) in 0.00001 degrees (one hundred-thousandth), set bit 0 (FRD) of parameter No. 11630 to 1. In this case, angular displacement R is specified within the range of -36000000 to 36000000.

- Coordinate system rotation cancel command

The G code used to cancel coordinate system rotation (G69) may be specified in a block in which another command is specified.

- Tool compensation

Tool radius/tool nose radius compensation, tool length compensation, tool offset, and other compensation operations are executed after the coordinate system is rotated.

- Relationship with 3-dimensional coordinate conversion (G68, G69)

Both coordinate system rotation and 3-dimensional coordinate conversion use the same G codes: G68 and G69. The G code with I, J, and K is processed as a command for 3-dimensional coordinate conversion. The G code without I, J, and K is processed as a command for two-dimensional coordinate system rotation.

Limitation

- Commands related to reference position return and the coordinate system

In coordinate system rotation mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling coordinate system rotation mode.

- Incremental programming

The first move command after the coordinate system rotation cancel command (G69) must be specified with absolute values. If an incremental move command is specified, correct movement will not be performed.

- Note on the specification of one axis in coordinate system rotation

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

Bit 5 (AX1) of parameter No. 11600

If one axis is specified in the absolute mode when the coordinate system rotation mode is set:

0: The specified position is first calculated in the coordinate system before rotation then the coordinate system is rotated.

1: The coordinate system is first rotated then a movement is made to the specified position in the rotated coordinate system. (FS0*i*-C-compatible specification)

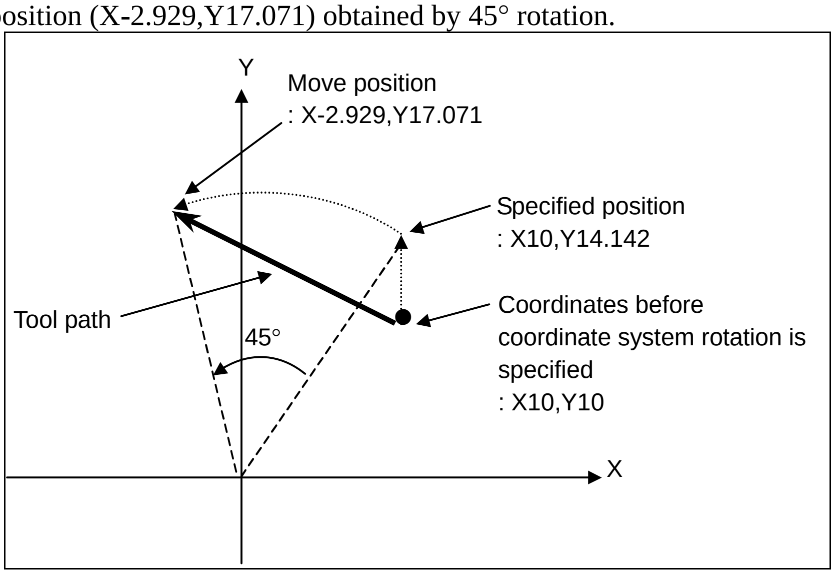
This parameter changes the handling of coordinates on axes not specified, so that a position to be reached by movement changes.

(Example)

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

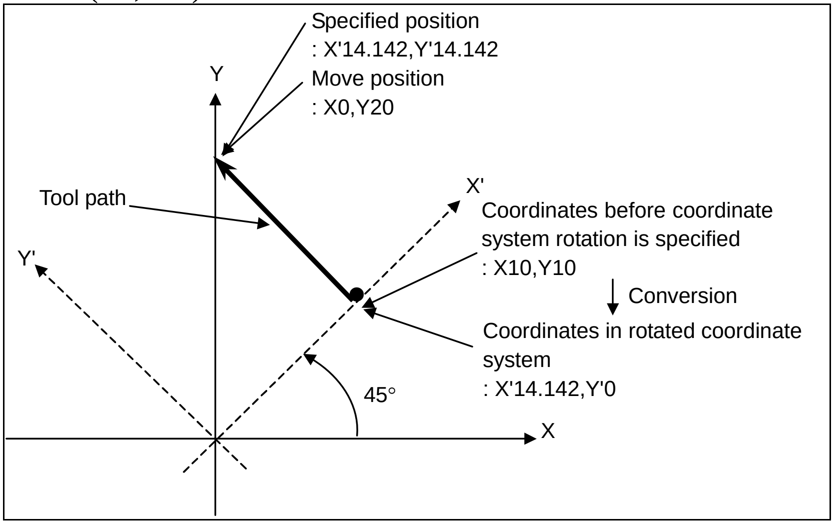
When bit 5 (AX1) of parameter No. 11600= 0:

The specified position is calculated in the coordinate system (XY) before rotation then the coordinate system is rotated. So, with the specification of (1), the position on the unspecified X axis is X10, and the specified position is (X10,Y14.142). Next, a movement is made to the move position (X-2.929,Y17.071) obtained by 45° rotation.



When bit 5 (AX1) of parameter No. 11600= 1:

With the specification of (1), coordinates (X10,Y10) before coordinate system rotation are converted to coordinates (X'14.142,Y'0) in the coordinate system (X'Y') obtained by 45° rotation. Next, a movement is made to the specified position (X'14.142,Y'14.142), that is, the move position (X0,Y20).



Explanation

- Absolute/Incremental position commands

```
N1 G92 X-500.0 Y-500.0 G69 G17 ;
N2 G68 X700.0 Y300.0 R60.0 ;
N3 G90 G01 X0 Y0 F200 ;
    (G91X500.0Y500.0)
N4 G91 X1000.0 ;
N5 G02 Y1000.0 R1000.0 ;
N6 G03 X-1000.0 I-500.0 J-500.0 ;
N7 G01 Y-1000.0 ;
N8 G69 G90 X-500.0 Y-500.0 M02 ;
```

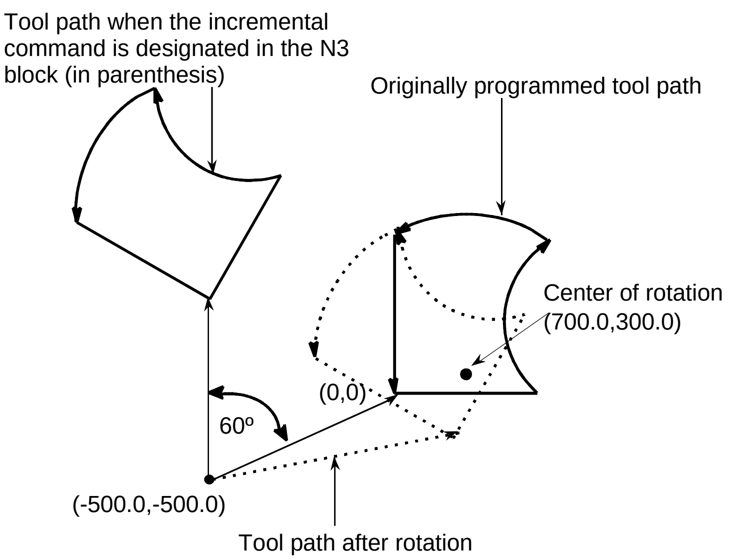


Fig. 6.12 (c) Absolute/incremental programming during coordinate system rotation

- 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 ;
```

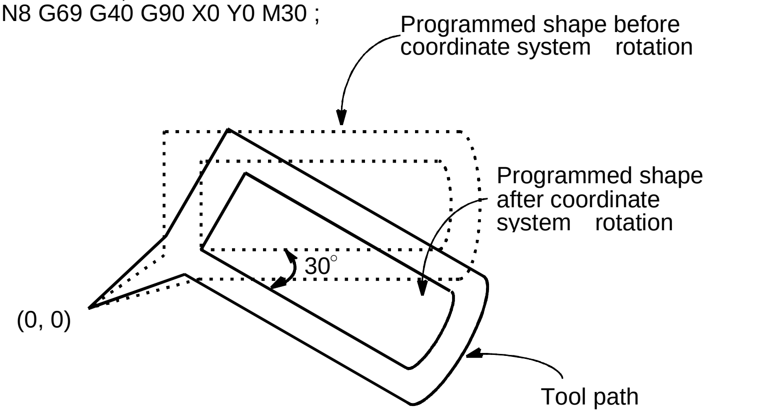


Fig. 6.12 (d) Cutter compensation and coordinate system rotation

- 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. 6.12 (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 ;
```

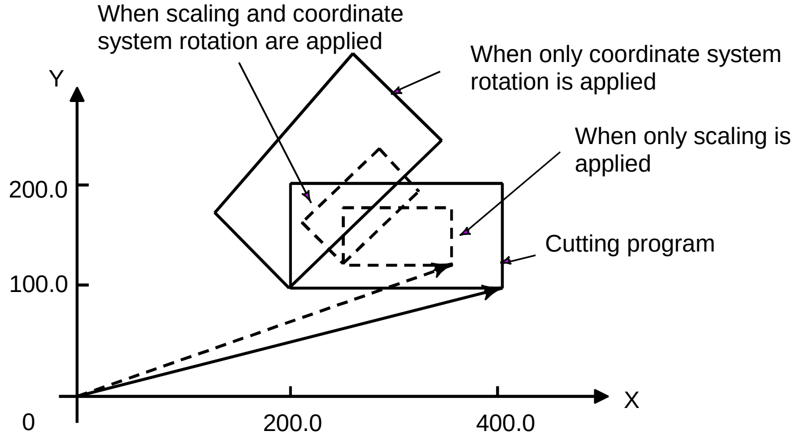


Fig. 6.12 (e) Scaling and coordinate system rotation in cutter compensation mode

- 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 ;
```

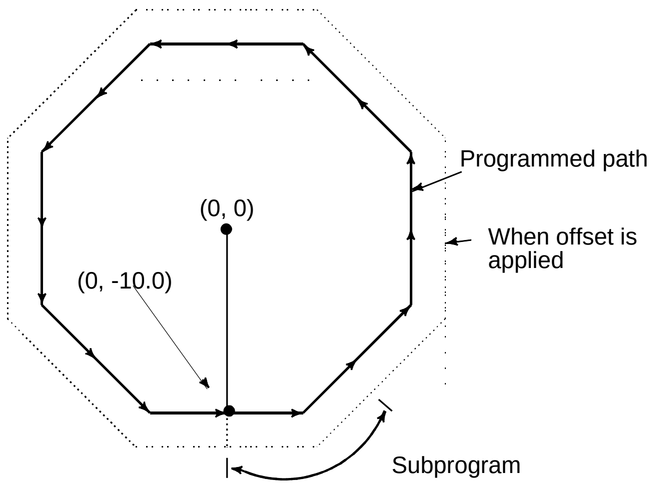


Fig. 6.12 (f) Coordinate system rotation command

6.13 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. 6.13 (a)).

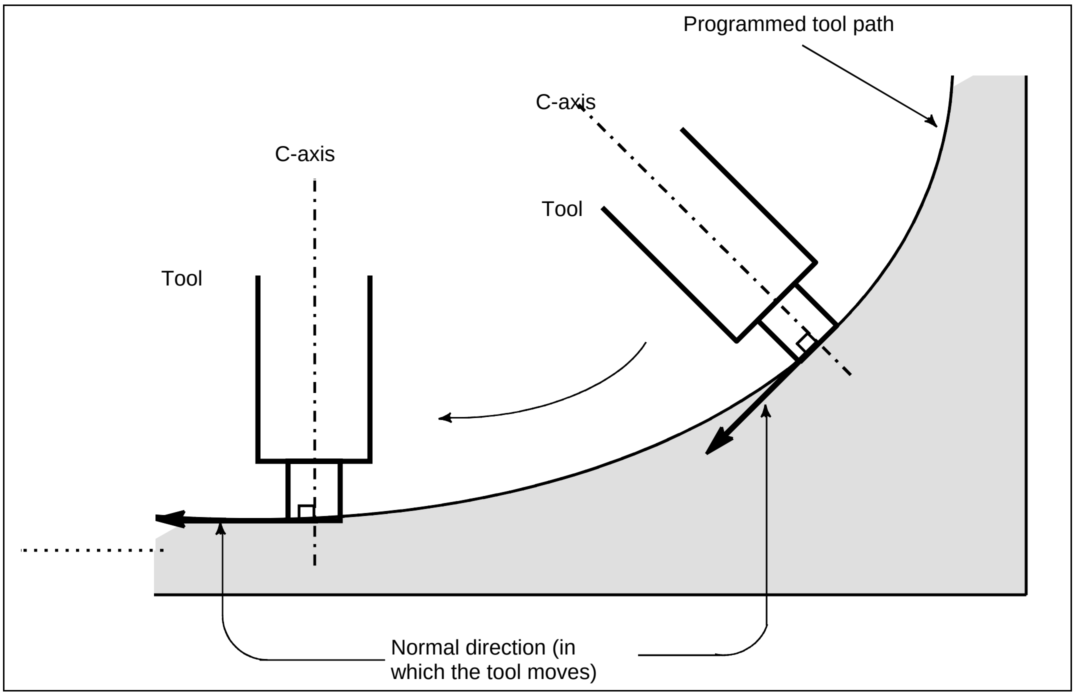


Fig. 6.13 (a) Sample Movement of the tool

Format

G41.1 ; Normal direction control, left

G42.1 ; Normal direction control, right

G40.1 ; Cancel normal direction control

The 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.

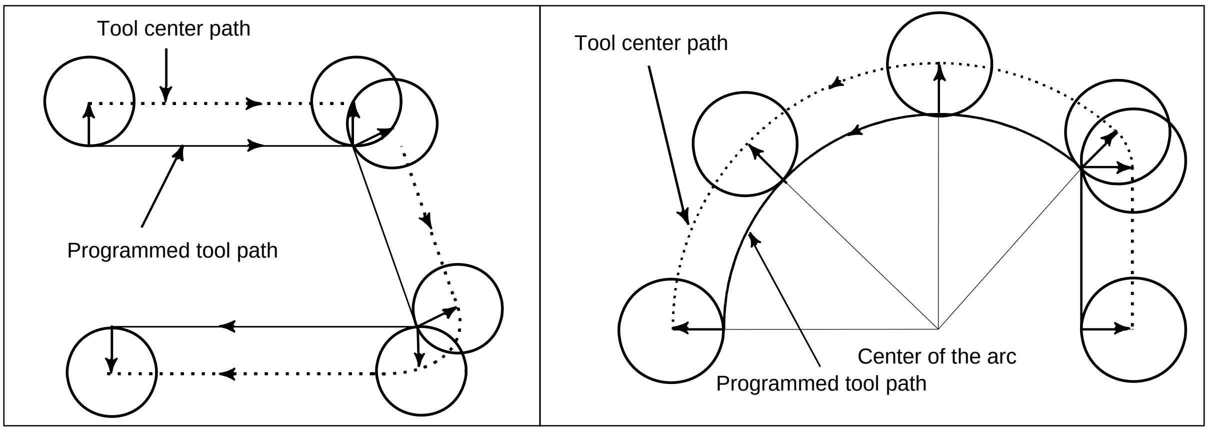


Fig. 6.13 (b) Normal direction control, left (G41.1) **Fig. 6.13 (c) Normal direction control, right (G42.1)**

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. 6.13 (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°.

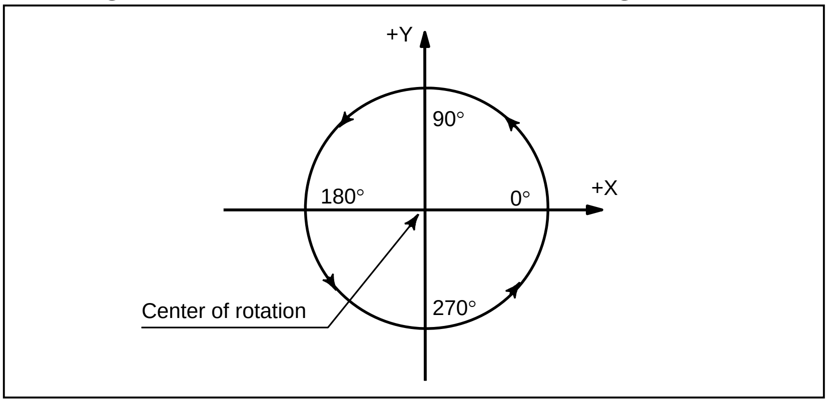


Fig. 6.13 (d) Angle of the C axis

- 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.

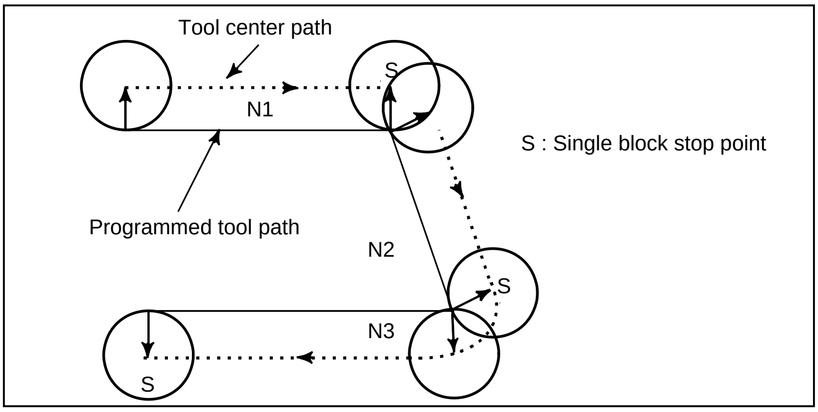


Fig. 6.13 (e) Point at which a single-block stop occurs in the normal direction control mode

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.

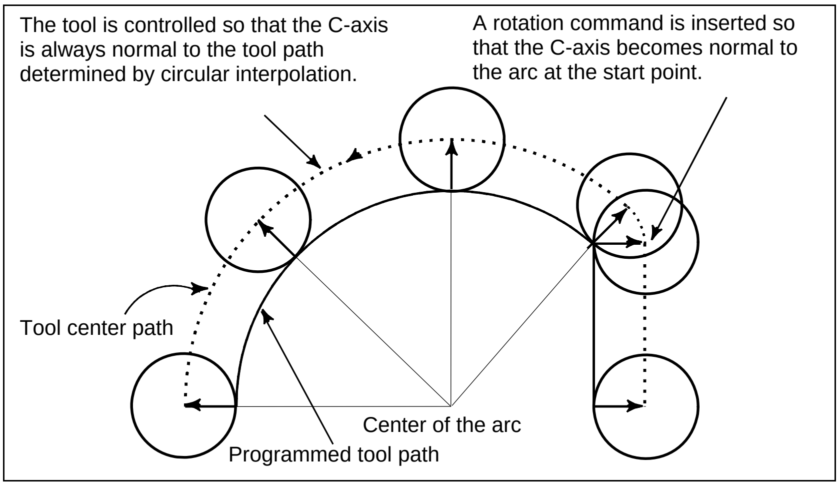


Fig. 6.13 (f) Normal direction control of the circular interpolation

> **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.

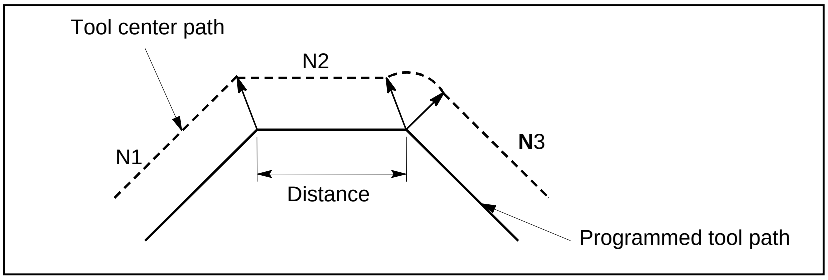
- Angle for which figure insertion is ignored

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.

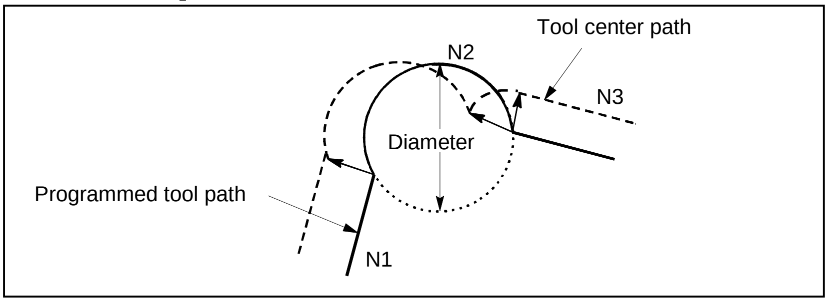
- Movement for which arc insertion is ignored

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.
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. The following functions must be commanded with normal direction control cancel mode.
   - Plane selection command
   - Spiral interpolation
   - Conical interpolation
   - Nano smoothing
   - AI Advanced Preview Control / AI contour control

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 bit 1 (FCV) of parameter No. 0001 to 1.

Explanation

Data formats for tool radius compensation, subprogram call, and canned cycles are different between the Series 0*i*-F and Series 10/11. The Series 15 program formats can be processed for memory operation.
Other data formats must comply with the Series 0*i*-F. When a value out of the specified range for the Series 0*i*-F is registered, an alarm occurs.

NOTE
1 Registration to memory and memory operation are possible only for the functions available in Series 0*i*-F.
2 Do not change the setting of this parameter (bit 1 of parameter No. 0001) during memory operation. Change the setting of this parameter in the reset state.

- 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*-F	**M98 P○○○ □□□□ ;** Repetition count Subprogram number (1 to 9999)

If, however, the custom macro is enabled (bit 5 (NMC) of parameter No. 8135 is 0), both formats can be used.

- Address for the canned cycle repetition count for drilling

The Series 10/11 and Series 0*i*-F 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*-F	K

8 AXIS CONTROL FUNCTIONS

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

8.1 ELECTRONIC GEAR BOX

8.1.1 Electronic Gear Box

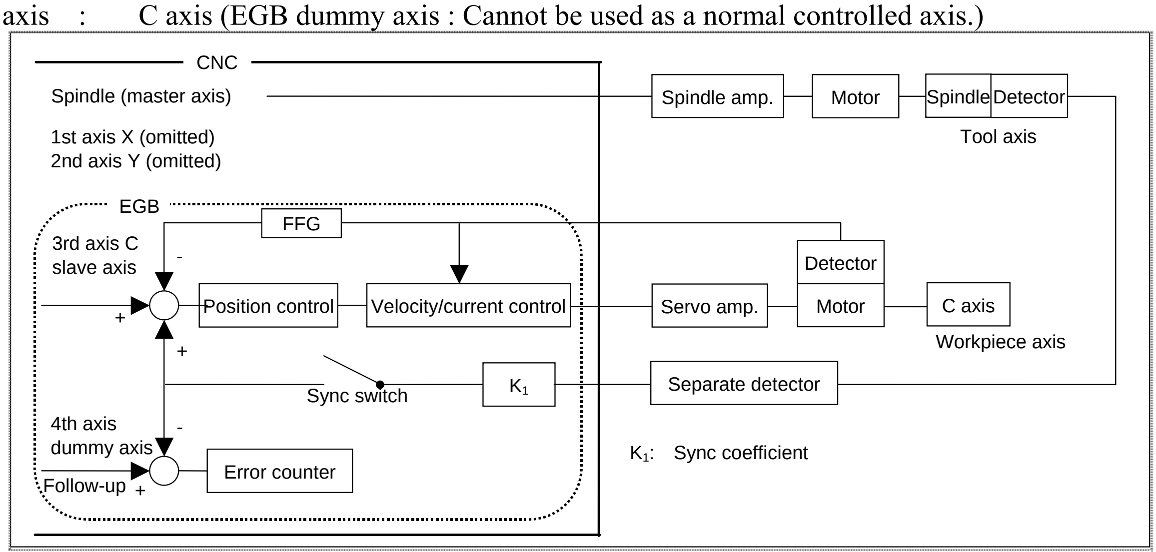
Overview

This function enables fabrication of high-precision gears, screws, and other components by rotating the workpiece in synchronization with a rotating tool or by moving the tool in synchronization with a rotating workpiece. The rate of synchronization can be specified with a program. The synchronization of tool and workpiece axes with this function adopts a system in which the synchronization is directly controlled by digital servo, so that the workpiece axis can follow up the speed fluctuations on the tool axis with no error, thereby allowing fabrication of high-precision cogwheels. In the subsequent explanation, the Electronic Gear Box is called the EGB.
There are several conditions in the setting and so on of the workpiece axis and the tool axis, and refer to the appropriate manual provided by the machine tool builder.

> **NOTE**
> This function is an 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

	Bit 0(EFX) of parameter No.7731=0	Bit 0(EFX) of parameter No.7731=1	
		Bit 5(HBR) of parameter No.7731=1	Bit 5(HBR) of parameter No.7731=0
Start of synchronization	**G81 T_ (L_) (Q_ P_) ;**	**G81.4 R_ (L_) (Q_ P_) ;**	**G81.4 T_ (L_) (Q_ P_) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to +250)
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 case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in 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)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 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 synchronization 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, ”EGB FORMAT ERROR” is generated.

If, during synchronization, G81 is issued again without synchronization cancelation, alarm PS1595, ”ILL-COMMAND IN EGB MODE” is generated if bit 3(ECN) of parameter No. 7731 is 0. If bit 3(ECN) 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

Synchronizing with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the G80 command 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 bit 0 (HBR) of parameter No. 7700 to 0, it is possible to cancel synchronization with a reset.

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.

⚠ **CAUTION**

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 500 (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 of bit 0 (ERV) 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 advanced preview control / AI contour control mode is temporarily canceled.
13 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

- 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

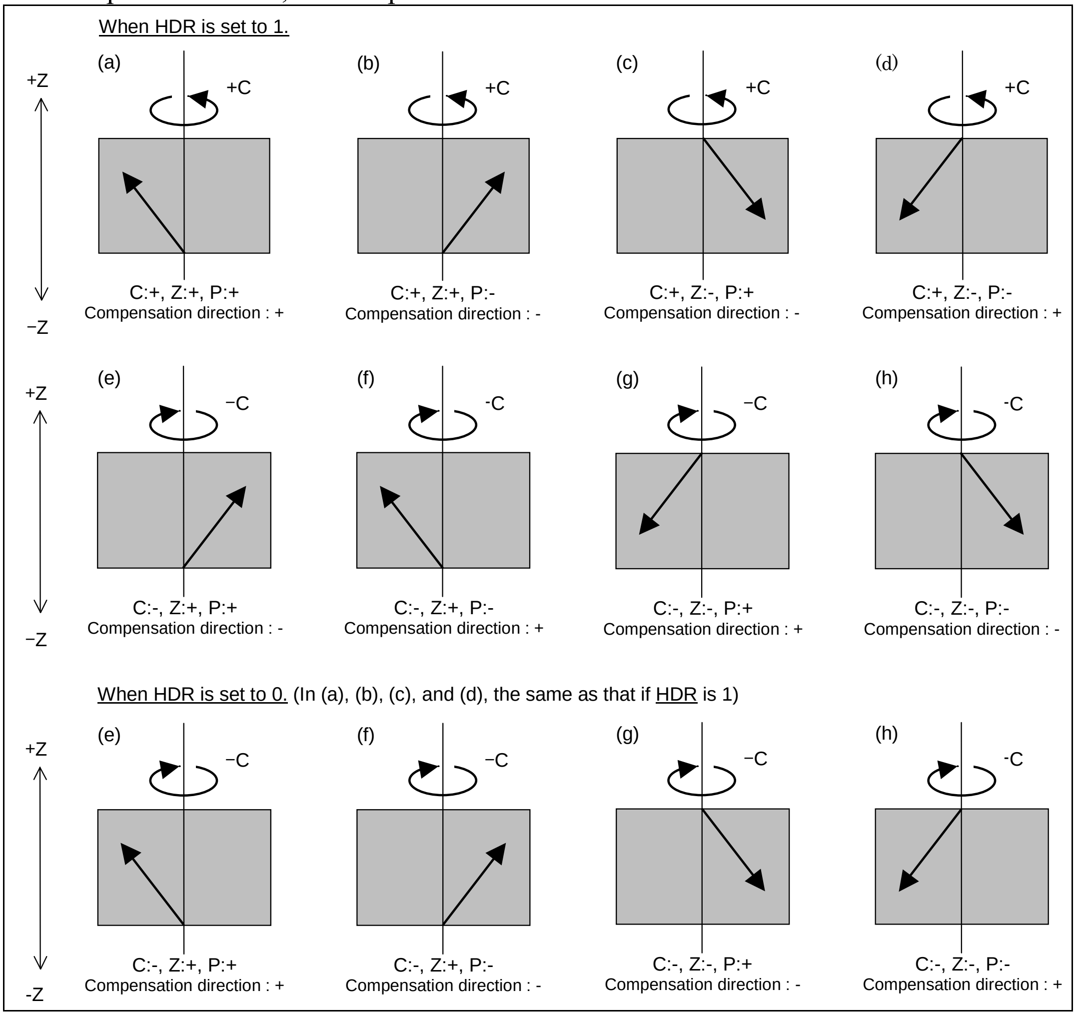
Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81

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

$$\text{Synchronization coefficient} = \frac{Kn}{Kd} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

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:

$-2147483648 \le Kn \le 2147483647$

$1 \le Kd \le 2147483647$

When this restriction is not satisfied, the alarm PS1596, "EGB OVERFLOW" 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.

<u>Conditions under the retract function with an alarm</u>

The conditions under which the retract function is performed by an alarm can be changed using the settings of bit 1(ARE) 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. (Mirror image is 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.
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 To enable the retract function with an alarm, bit 3 (ART) of parameter No. 7702, must be set.
10 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.
11 If a new alarm is issued during retraction with the retract function with an alarm, a retract operation is not performed.

8.1.2 Electronic Gear Box Automatic Phase Synchronization

Overview

In the electronic gear box (EGB), when synchronization start or cancellation is specified, the synchronizing state is changed to another state gradually by applying acceleration/deceleration. This is because if synchronization is started or canceled immediately, this sudden speed change shocks a machine. Therefore, synchronization can be started or canceled while the spindle is rotating. Also, synchronization ratio can be changed while the spindle is rotating.

At the start of synchronization, automatic phase synchronization is performed such that the position where the machine coordinate is 0 about the workpiece axis matches the position corresponding to the spindle one-rotation signal. With this synchronization, the same operation is performed as synchronization start caused by a one-rotation signal in hobbing synchronization when using the functions of a hobbing machine.

The spindle corresponds to the EGB master axis and the workpiece axis corresponds to an EGB slave axis (servo axis or Cs contouring axis).

NOTE

Electronic gear box Automatic Phase Synchronization is optional function.

Format

- Acceleration/deceleration type

G81 T _ L _ R1 ;	Synchronization start
G80 R1 ;	Synchronization cancellation

T : Number of teeth (range of valid settings: 1-5000)
L : Number of hob threads (range of valid settings: -250 to +250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

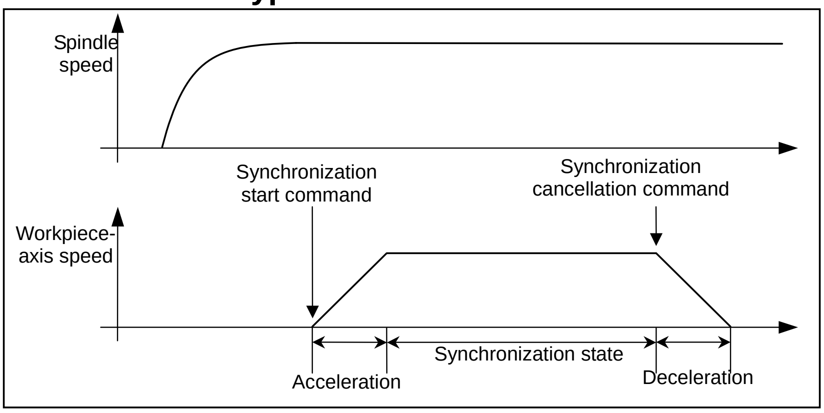
- Acceleration/deceleration plus automatic phase synchronization type

G81 T _ L _ R2 ;	Synchronization start
G80 R1 ;	Synchronization cancellation

T : Number of teeth (range of valid settings: 1-5000)
L : Number of hob threads (range of valid settings: -250 to +250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

Explanation

- Acceleration/deceleration type



1. Specify G81R1 to start synchronization.
   When G81R1 is specified, the workpiece axis (slave axis) is accelerated with the acceleration according to the acceleration rate set in the parameter No. 7778. When the speed reaches the synchronization speed, the G81R1 block is terminated.
2. For cancellation, specify G80R1 while the tool is moved away from the workpiece.
3. When G80R1 is specified, deceleration is started immediately at the acceleration rate set in parameter No. 7778.
   When the speed is reduced to 0, the G80R1 block is terminated.

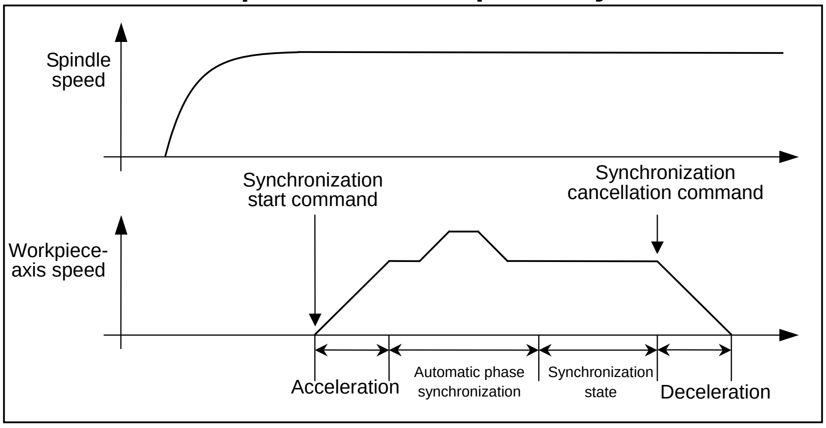
NOTE
1 During synchronization start/cancellation, acceleration/deceleration is linear type.

NOTE

2 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization is canceled:
<1> Reset
<2> Alarm PW0000, "POWER MUST BE OFF"
<3> IO alarm

3 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the 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.

- Acceleration/deceleration plus automatic phase synchronization type



1. Move the workpiece axis to the position that corresponds to that of the one-rotation signal of the spindle.
2. Specify G81R2 to start synchronization.
   When G81R2 is specified, the workpiece axis is accelerated with the acceleration according to the acceleration rate set in the parameter No. 7778. Upon completion of phase synchronization, the G81R2 block terminates.
3. For cancellation, specify G80R2 while the tool is moved away from the workpiece.
4. When G80R2 is specified, deceleration is started immediately at the acceleration rate set in parameter No. 7778. When the speed is reduced to 0, the G80R2 block is terminated.

⚠ CAUTION

1 In automatic phase synchronization, specify the speed in parameter No. 7776 and the movement direction in parameter PHD, bit 7 of No. 7702.

2 In phase synchronization, rapid–traverse linear acceleration/deceleration (with the time constant specified in parameter No. 1620) is performed.

3 The workpiece–axis speed is obtained by superimposing the speed in automatic phase synchronization onto the speed corresponding to spindle rotation. In consideration of this superimposition, specify a position deviation limit in parameter No. 1828.

NOTE

1 The one-rotation signal used for automatic phase synchronization is issued not by the spindle position coder but by the separate pulse coder attached to the spindle and used to collect EGB feedback information. This means that the orientation position based on the one-rotation signal issued by the spindle position coder does not match the position used as the reference for the workpiece axis when phase synchronization is performed with automatic phase synchronization based on G81R2.
Moreover, the one-rotation signal of the separate pulse coder must be turned on for each rotation of the spindle.

2 With the use of parameter No. 7777, the position at which the phase of the workpiece axis is matched can be shifted from the position corresponding to the one-rotation signal in automatic phase matching.

3 By setting bit 6 (EPA) of parameter No. 7731 to 1, automatic phase synchronization can be performed so that the workpiece axis position at the start of synchronization matches the position corresponding to the spindle one-rotation signal.

4 By setting bit 6 (EPA) of parameter No. 7731 to 1, in automatic phase synchronization, when a synchronization command is issued again in the synchronization state, movement about the workpiece axis is made such that the position corresponding to the one-rotation signal of the spindle matches the position about the workpiece axis specified in the G81R2 synchronization start command executed first.

5 In automatic phase synchronization, movement is performed about the workpiece axis from the current position to the nearest phase position in the phase synchronization movement direction specified by the parameter.

6 Linear acceleration/deceleration applies to synchronization start/cancellation.

7 The acceleration/deceleration plus automatic phase synchronization type can be executed by the bit 6 (PHS) of parameter No. 7702, without specifying an R2 command in a G81 or G80 block.

8 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization canceled:
<1> Reset
<2> Alarm PW0000, “POWER MUST BE OFF”
<3> IO alarm

9 When EGB is used, it is necessary to move the separate Pulsecoder attached to the spindle by one rotation or more before executing automatic phase synchronization.

10 The acceleration rate parameter No. 7778 must not be changed in the synchronization mode.

11 If parameter No. 7778 is 0, alarm PS1598, “EGB AUTO PHASE PARAMETER SETTING ERROR” is issued when G81 is issued.

12 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the 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.

Program example

- Acceleration/deceleration type

```
M03 ;                  Clockwise spindle rotation command
G81  T_  L_  R1 ;      Start of synchronization
```

```
G00  X_ ;              Positions the workpiece at the machining position.
```

Machining in the synchronous state

```
G00  X_ ;              Retract the workpiece from the tool.
G81  T_  L_  R1 ;      Synchronization ratio change.
G00  X_ ;              Positions the workpiece at the machining position.
```

Machining in the synchronous state

```
G00  X_ ;              Retract the workpiece from the tool.
G80  R1  ;             Cancel synchronization
```

- Acceleration/deceleration plus automatic phase synchronization type

```
M03  ;  Clockwise spindle rotation command
G00  G90  C_ ;         C-axis positioning
G81  T_  L_  R2 ;      Start of synchronization.
G00  X_ ;              Positions the workpiece at the machining position.
```

Machining in the synchronous state

```
G00  X_ ;              Retract the workpiece from the tool.
G81  T_  L_  R2 ;      Synchronization ratio change.
G00  X_ ;              Positions the workpiece at the machining position.
```

Machining in the synchronous state

```
G00  X_ ;              Retract the workpiece from the tool.
G80  R2  ;             Cancel synchronization
```

8.1.3 Skip Function for EGB Axis

Overview

This function enables the skip or high-speed skip signal (these signals are collectively called skip signals in the remainder of this manual) for the EGB slave axis in synchronization mode with the EGB (electronic gear box).
This function has features such as the following:

1. If a skip signal is input while an EGB axis skip command block is being executed, this block does not terminate until the specified number of skip signals have been input.
2. If a skip signal is input while an EGB axis skip command block is being executed, the tool remains in synchronous mode and the EGB slave axis is not stopped and keeps moving.
3. The machine coordinates recorded when skip signals are input and the number of input skip signals are stored in specified custom macro variables.

NOTE
This function is an optional function.

Format

G81 T_ L_ ; **EGB mode ON**

G31.8 G91 α0 P_ Q_ (R_) ; **EGB skip command**

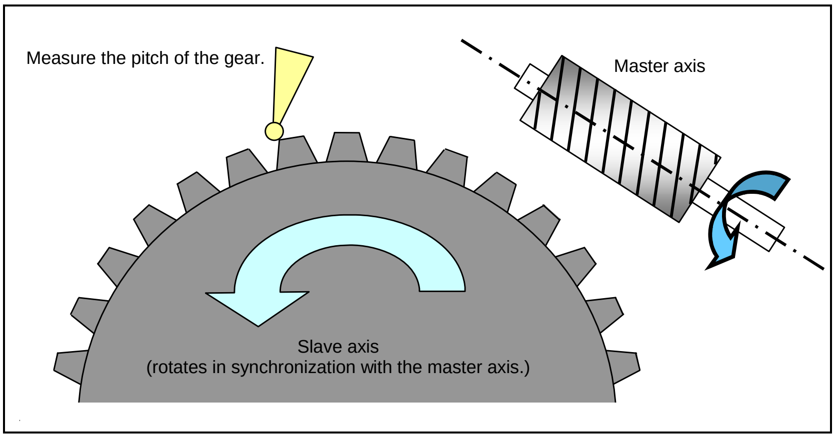
α: Specify an EGB slave axis. The specified value must always be "0".
P: Number of the first one of the custom macro variables used to store the machine coordinates recorded when skip signals are input.
Q: Number of skip signals that can be input during the execution of G31.8 (permissible range: 1 to 512).
R: Number of the custom macro variable used to store the number of input skip signals.
Specify it to check the number of input signals.

Explanation

G31.8 is a one-shot G code.
After the execution of G31.8, values of machine coordinate which is gotten at each time of skip signal input are set in custom macro variables. The numbers of variables are used from the top number commanded by P to the number added with the amount of times commanded by Q.
And the total times of skip signal input is set in the custom macro variable whose number is commanded by R.

Example

The pitch of a gear can be measured.



G81 T200 L2 ; ....EGB mode ON
X_ ;
Z_ ;
G31.8 G91 C0 P500 Q200 R1 ;...........EGB skip command

After 200 skip signals have been input, the 200 skip positions on the C-axis that correspond to the respective skip signals are stored in custom macro variables #500 to #699.
Also, the number of input skip signals is stored in custom macro variable #1.

NOTE
1 When specifying this function, specify only a single EGB slave axis. If no axis is specified for two or more axes are specified, alarm PS1152, "G31.9/G31.8 FORMAT ERROR" is generated.
2 If P is not specified, alarm PS1152 is generated.
3 If R is not specified, the number of input skip signals is not written to a custom macro variable.

NOTE

4 The custom macro variable numbers specified in P and R must be existing ones. If a non-existent variable number is specified, alarm PS0115, "VARIABLE NO. OUT OF RANGE" is generated.
If a variable shortage occurs, alarm PS0115 is generated.

5 Whether to use conventional skip signals or high-speed skip signals with this function can be specified with bit 4(HSS) of parameter No. 6200. If high-speed skip signals are used, specify which high-speed signals to enable with bits 0 to 3(9S1 to 9S4) of parameter No. 6208.

6 Skip positions are calculated from feedback pulses from the machine. Thus, they are free from errors due to delay in acceleration/deceleration and the servo system.

8.1.4 U-axis Control

Overview

Conventionally, the control of an axis on a spindle, such as the U-axis of a vertical lathe, from a motor mounted in a location other than the spindle has required a mechanism, consisting of a planetary gear box and differential gears, to prevent the axis from moving as the spindle is rotating.
The U-axis control function enables the U-axis to remain in a fixed position or to move at a programmed speed without using a mechanism such as a planetary gear box. This is done by causing the U-axis motor to rotate in such a way that U-axis movement, which would be caused by the rotation of the spindle, is canceled out. For details, refer to the manual supplied by the machine tool builder.

NOTE

This function is included in the option "Electronic gear box".
To use this function, the above option is required.

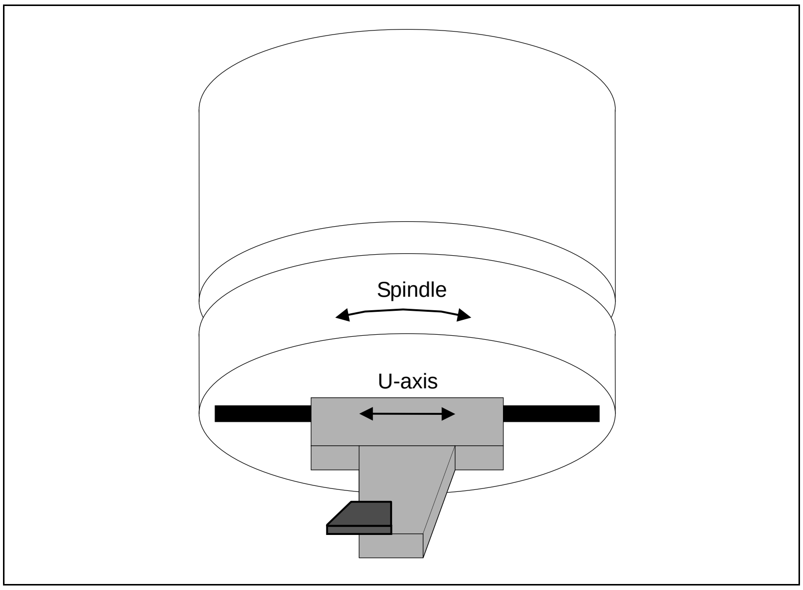


Fig. 8.1.4 (a) Example of a machine having the U-axis

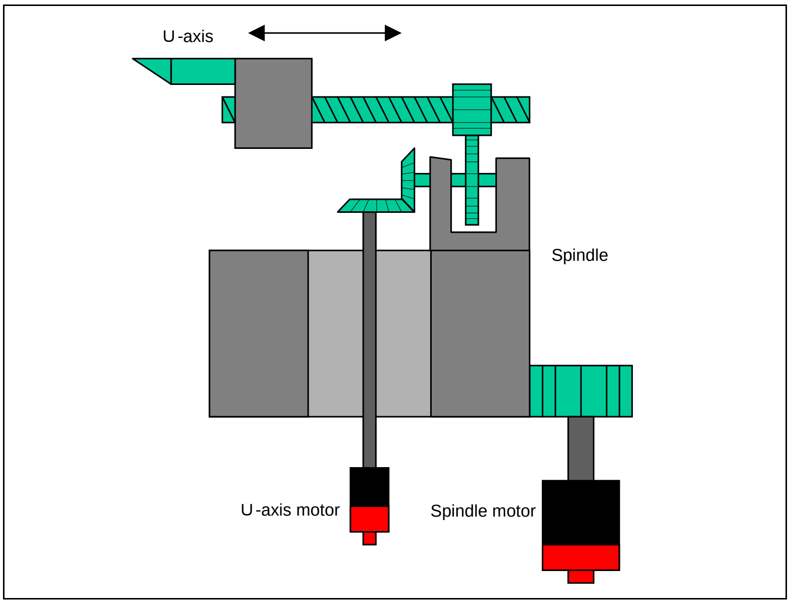


Fig. 8.1.4 (b) Example of the structure of a machine having the U-axis

In the example of the above structure, the tool moves along the U-axis when the spindle rotates. This movement is canceled out by rotating the U-axis motor.

III. OPERATION

1 MANUAL OPERATION

MANUAL OPERATION consists of the following operations:

1.1 3-DIMENSIONAL MANUAL FEED

This function enables the use of the following functions.

- 3-dimensional manual feed
  - Tool axis direction handle feed/tool axis direction JOG feed/tool axis direction incremental feed
  - Tool axis right-angle direction handle feed/tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed
  - Tool tip center rotation handle feed/tool tip center rotation JOG feed/tool tip center rotation incremental feed
  - Table vertical direction handle feed/table vertical direction JOG feed/table vertical direction incremental feed
  - Table horizontal direction handle feed/table horizontal direction JOG feed/table horizontal direction incremental feed

A handle interrupt can be generated for each handle feed. Handle interrupts work according to the corresponding handle feed specifications described hereinafter unless otherwise noted.

- Screen display functions
  - Display of the coordinate of the tool tip
  - Display of movement values(Tool axis reference, Tool tip center, Table reference)
  - Display of the amount of machine axes movement

NOTE
"3-Dimensional manual feed" is an optional function.

- Selecting a coordinate system when calculating the tool direction

If, in 3-dimensional manual feed, a workpiece coordinate system offset is set for a rotation axis, use bit 0 (CAC) of parameter No. 12319 to select whether to use values in the workpiece or machine coordinate system when calculating the tool direction.

- When the bit 0 (CAC) of parameter No. 12319 = 0:
  Values in the machine coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the machine coordinate on the rotation axis is 0.

- When the bit 0 (CAC) of parameter No. 12319 = 1:
  Values in the workpiece coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the workpiece coordinate on the rotation axis is 0.

For the functions below, values in the workpiece coordinate system of a rotation axis are used to calculate the tool direction and, therefore, if a workpiece coordinate system offset is set for a rotation axis, and 3-dimensional manual feed is to be used together with any of the functions below, set bit 0 (CAC) of parameter No. 12319 to 1.

- Tilted working plane indexing

(Example)

Related parameters:
No.19680=2 (Tool rotation type)
No.19682=3 (the master rotation axis (C-axis) is about the Z-axis)
No.19687=2 (the slave rotation axis (B-axis) is about the Y-axis)
No.19697=3 (the reference tool axis direction is the Z-axis direction)
No.19698=0 (angle RA when the reference tool axis direction is tilted)
No.19699=0 (angle RB when the reference tool axis direction is tilted)

Workpiece coordinate system offset:
B=10.0

Example 1:

The tool is assumed to face the Z-axis direction when it is in the following states.

Position in the workpiece coordinate system	B= -10.0
Position in the machine coordinate system	B=0.0

In this case, values in the machine coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 0.

Example 2:

The tool is assumed to face the Z-axis direction when it is in the following states.

Position in the workpiece coordinate system	B=0.0
Position in the machine coordinate system	B=10.0

In this case, values in the workpiece coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 1.

NOTE

1. To execute 3-dimensional handle feed, manual handle feed needs to be enabled by setting bit 0 (HPG) of parameter No. 8131 to 1.
2. A 3-dimensional handle interrupt must not be generated when a rotation axis command is being executed during automatic operation.
3. 3-dimensional manual feed is disabled when the manual reference position return mode is selected.
4. If per-axis interlock is enabled to at least one of 3-dimensional manual feed axes, movement with manual feed is not performed.

1.1.1 Tool Axis Direction Handle Feed / Tool Axis Direction JOG Feed / Tool Axis Direction Incremental Feed

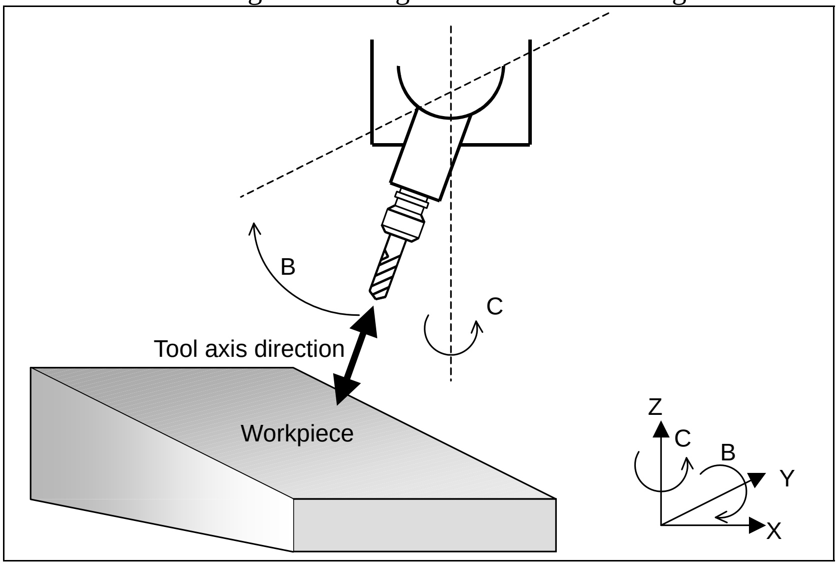
Overview

In the tool axis direction handle feed, tool axis direction JOG feed, and tool axis direction incremental feed, the tool or table is moved in the tool axis direction.

Explanation

- Tool axis direction

The tool axis direction that is taken when all the rotation axes for controlling the tool are at an angle of 0 degree is specified in parameters Nos.19697, 19698, and 19699. As the rotation axes for controlling the tool rotate, the tool axis direction changes according to the rotation axis angle.



- Tool axis direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis direction feed in the tilted working plane indexing mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane indexing.

- Tool axis direction handle feed

The tool axis direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1D) to make the tool axis direction handle feed mode effective is set in parameter No.12310.
<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1D).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool axis direction JOG feed/tool axis direction incremental feed

The tool axis direction JOG feed or tool axis direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis direction JOG feed or tool axis direction incremental feed is determined by parameter No.19697 only.)

Ex.) Parameter No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.

- +J3 : Tool axis direction +
- -J3 : Tool axis direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

1.1.2 Tool Axis Right-Angle Direction Handle Feed / Tool Axis Right-Angle Direction JOG Feed / Tool Axis Right-Angle Direction Incremental Feed

Overview

In the tool axis right-angle direction handle feed, tool axis direction JOG feed, or tool axis direction incremental feed, the tool or table is moved in the tool axis right-angle direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the tool axis direction vector.

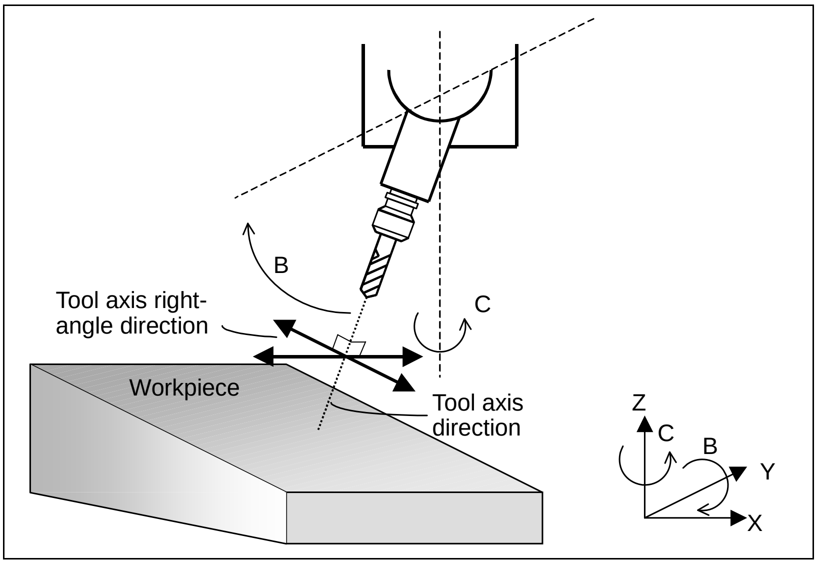
Explanation

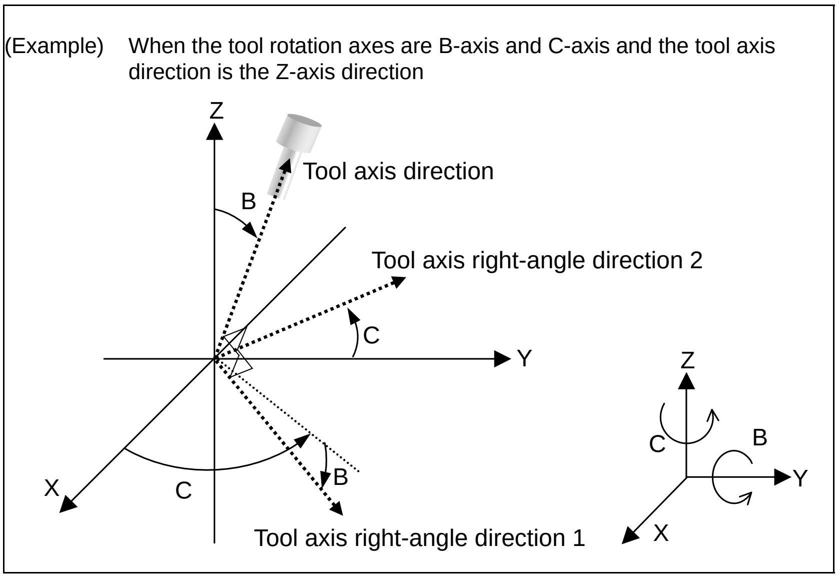
- Tool axis right-angle direction

There are two tool axis right-angle directions, which are perpendicular to the tool axis direction (see the previous section).

Parameter No.19697	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the tool axis right-angle directions that may be taken when the angles of all the rotation axes for controlling the tool are 0 degree and when parameters Nos.19698 and 19699 are both set to 0.
When the reference tool axis direction is inclined based on the settings of parameters Nos.19698 and 19699, the tool axis right-angle direction is also inclined as much.
As the rotation axes for controlling the tool rotate, the tool axis right-angle direction changes according to the rotation axis angle.





- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the tool axis direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the tool axis right-angle direction 1 (longitude direction) vector ($\vec{R}1$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R}1 = \vec{P} \times \vec{T}$

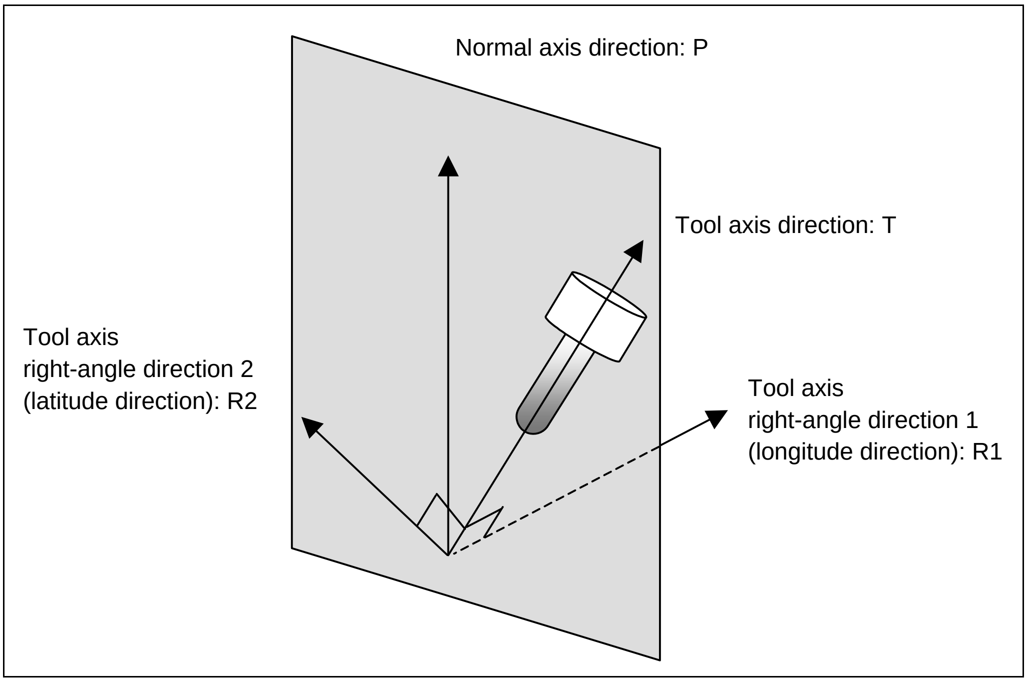
Let a vector perpendicular to the tool axis direction vector ($\vec{T}$) and tool axis right-angle direction 1 (longitude direction) vector ($\vec{R}1$) be the tool axis right-angle direction 2 (latitude direction) vector ($\vec{R}2$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

Equation: $\vec{R}2 = \vec{T} \times \vec{R}1$

When the tool axis direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No.12321	Normal axis direction	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the reference tool axis direction (parameter No. 19697).
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459, “MACHINE PARAMETER INCORRECT” is issued.



- Tool axis right-angle direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis right-angle direction feed in the tilted working plane indexing mode is defined as follows:
Tool axis right-angle direction 1:
X direction in the feature coordinate system of the tilted working plane indexing
Tool axis right-angle direction 2:
Y direction in the feature coordinate system of the tilted working plane indexing

- Tool axis right-angle direction handle feed

The tool axis right-angle direction handle feed is enabled when the following four conditions are satisfied:
<1> Handle mode is selected.
<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1D) to make the tool axis right-angle direction handle feed mode effective is set in parameter No.12311 or No.12312.
<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1D).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis right-angle direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed

The tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is determined by parameter No.19697 only.)

Ex.) Parameter No.19697=3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1 : Tool axis right-angle direction 1 +
- -J1 : Tool axis right-angle direction 1 -
- +J2 : Tool axis right-angle direction 2 +
- -J2 : Tool axis right-angle direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

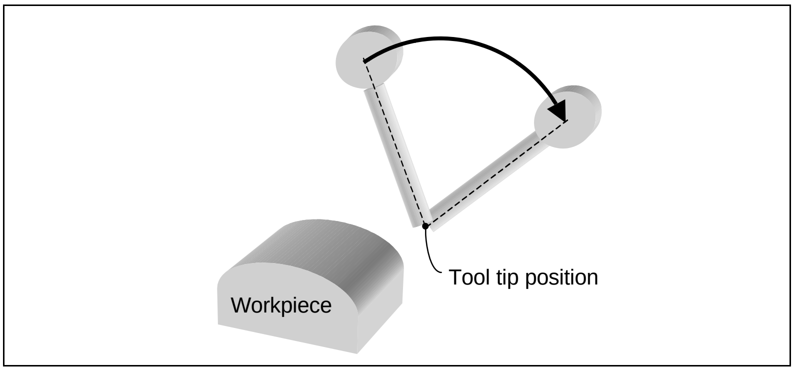
The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

1.1.3 Tool Tip Center Rotation Handle Feed / Tool Tip Center Rotation JOG Feed / Tool Tip Center Rotation Incremental Feed

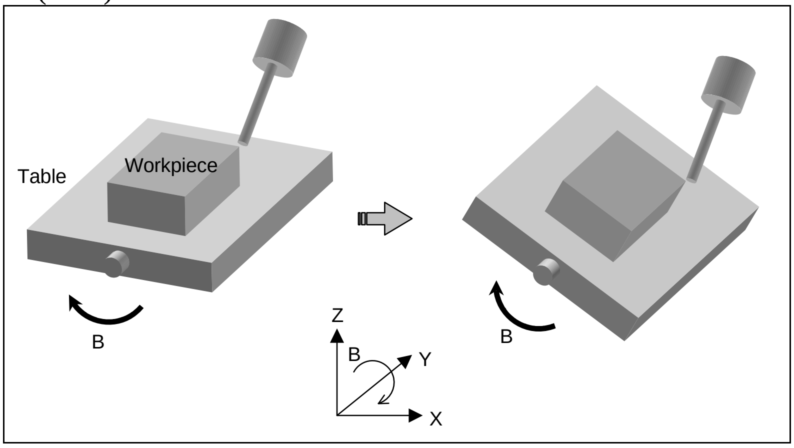
Overview

In the tool tip center rotation handle feed, tool tip center rotation JOG feed, and tool tip center rotation incremental feed, when a rotary axis is rotated by manual feed, the linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece (table).

- The following figure shows an example where the tool is rotated on the rotation axis. In this case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece.



- The following figure shows an example where the table is rotated on the rotation axis. As in the previous case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece (table).



- Tool tip center rotation handle feed

The tool tip center rotation handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1D) to make the tool tip center rotation handle feed mode effective is set in parameter No.12313 or No.12314.
<4> The value of parameter No.12313 or No.12314 matches the first manual handle feed axis selection signals (HS1A - HS1D).

Amount of movement

When the manual pulse generator is rotated, the rotation axis is moved by the amount of rotation. The linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool tip center rotation JOG feed/tool tip center rotation incremental feed

The tool tip center rotation JOG feed or tool tip center rotation incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the rotation axis to be rotated.
Ex.) When the B-axis (4th axis) is rotated
- +J4 : Tool tip center rotation feed +
- -J4 : Tool tip center rotation feed -

Feedrate

Control is exerted so that the synthetic speed of the linear axes (in the tangential direction) is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis).

- Selection of the tool length offset value

The tool length in 3-dimensional manual feed is determined as explained below. (Table 1.1.3 (a))
If bit 2 (LOD) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is performed, the offset data specified for the tool length offset is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is not performed, the tool length is determined as follows. If bit 3 (LOZ) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length in 3-dimensional manual feed; if LOZ is set to 1, the tool length is assumed to be 0.

Table 1.1.3 (a) Tool length offset value in 3-dimensional manual feed

<table>
<tr><th colspan="2" rowspan="3"></th><th colspan="3">Bit 2 (LOD) of parameter No. 19746</th></tr>
<tr><th rowspan="2">= 0</th><th colspan="2">= 1</th></tr>
<tr><th>Tool length offset enabled</th><th>Tool length offset canceled</th></tr>
<tr><td rowspan="2">Bit 3 (LOZ) of parameter No. 19746</td><td>= 0</td><td rowspan="2">Parameter No. 12318</td><td rowspan="2">Offset data</td><td>Parameter No. 12318</td></tr>
<tr><td>= 1</td><td>0</td></tr>
</table>

The tool length offset function is enabled when the following two conditions are both satisfied:
- The tool length offset function (G43/G44) is enabled (modal code of group 8 except G49)
- The H/D code is other than 0.

If bit 6 (CLR) of parameter No. 3402 is set to 0 not to clear the tool length offset vector, G codes of group 8, and H codes at the time of a reset, the tool length offset status is maintained when a reset is made in the tool length offset mode.

1.1.4 Table Vertical Direction Handle Feed / Table Vertical Direction JOG Feed / Table Vertical Direction Incremental Feed

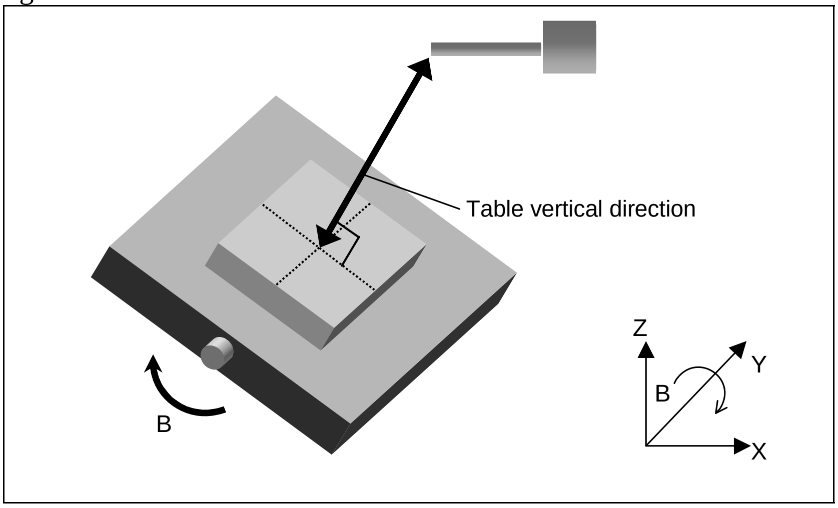
Overview

In the table vertical direction handle feed, table vertical direction JOG feed, and table vertical direction incremental feed, the tool is moved in the table vertical direction.

Explanation

- Table vertical direction

The table vertical direction is a direction vertical to the table. It is equal to the tool axis direction specified in parameter No.19697 when all of the rotation axes for controlling the table are at a an angle of 0 degree. When the rotation axes for controlling the table rotate, the table vertical direction changes according to the rotation axis angle.



- Table-based vertical direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based vertical direction feed in the tilted working plane indexing mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane indexing.

- Table vertical direction handle feed

The table vertical direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> Both the tool axis direction feed mode signal ALNGH and the table base signal TB_BASE are set to "1".
<3> The state of the first manual handle feed axis selection signals HS1A - HS1D to make the table vertical handle feed mode effective is set in parameter No.12310.
<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals HS1A - HS1D.

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table vertical direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Table vertical direction JOG feed/table vertical direction incremental feed

The table vertical direction JOG feed or table vertical direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> Both the tool axis direction feed mode signal ALNGH and the table base signal TB_BASE are set to "1".

<3> The feed axis direction selection signal [+Jn,-Jn (where n = 1 to the number of controlled axes)] is set to "1" for the axis corresponding to the direction specified by parameter No.19697.

Ex.) Parameter No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.

- +J3 : Table vertical direction +
- -J3 : Table vertical direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

1.1.5 Table Horizontal Direction Handle Feed / Table Horizontal Direction JOG Feed / Table Horizontal Direction Incremental Feed

Overview

In the table horizontal direction handle feed, table horizontal direction JOG feed, and table horizontal direction incremental feed, the tool is moved in the table horizontal direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the table-based vertical direction vector.

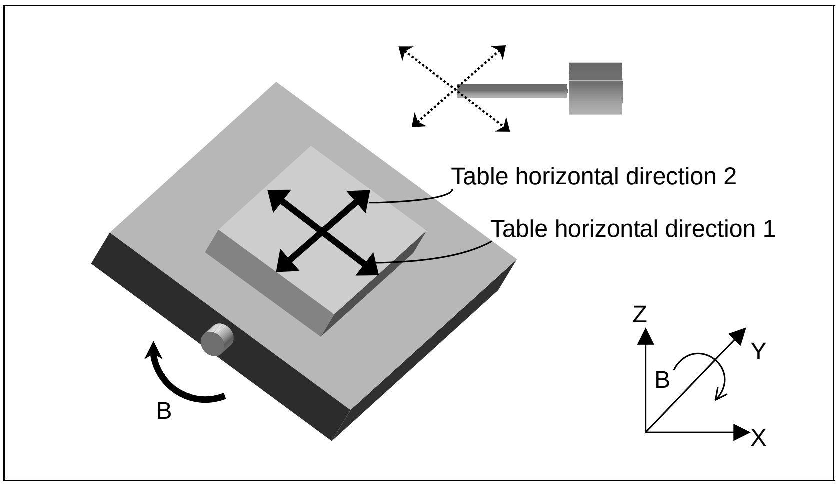
Explanation

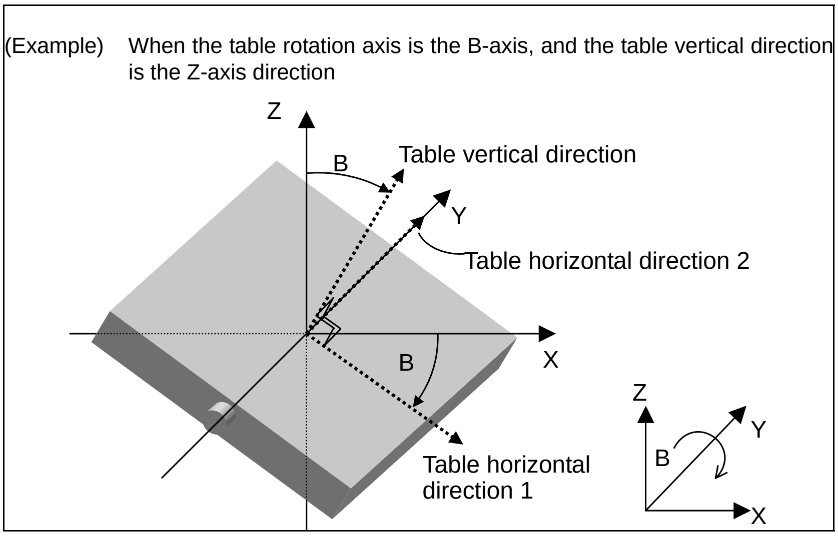
- Table horizontal direction

There are two table horizontal directions, which are perpendicular to the table vertical direction (see the previous section).

Parameter No.19697	Table horizontal direction 1	Table horizontal direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the table horizontal directions that may be taken when the angles of all the rotation axes for controlling the table are 0 degree.
As the rotation axes for controlling the table rotate, the table horizontal direction changes according to the rotation axis angle.





- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the table-based vertical direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the table-based horizontal direction 1 (longitude direction) vector ($\vec{R}1$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R}1 = \vec{P} \times \vec{T}$

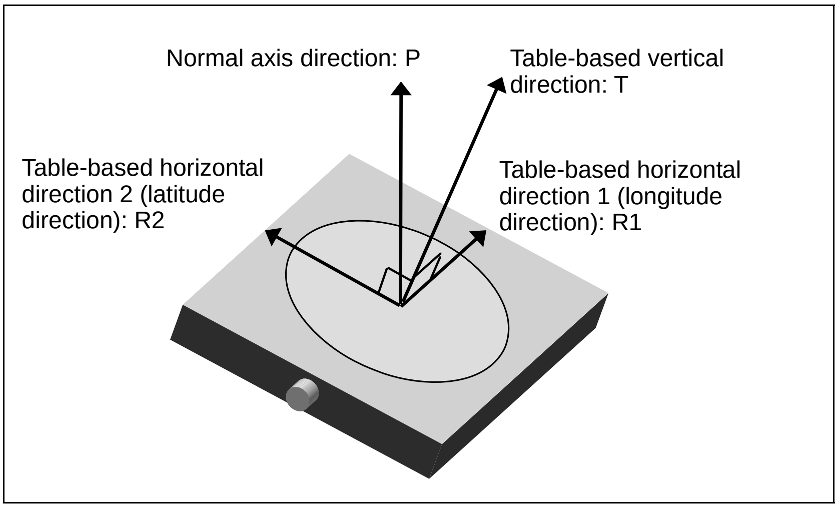
Let a vector perpendicular to the table-based vertical direction vector ($\vec{T}$) and table-based horizontal direction 1 (longitude direction) vector ($\vec{R}1$) be the table-based horizontal direction 2 (latitude direction) vector ($\vec{R}2$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

Equation: $\vec{R}2 = \vec{T} \times \vec{R}1$

When table-based vertical direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), table-based horizontal direction 1 and table-based horizontal direction 2 are assumed as follows:

Parameter No. 12321	Normal axis direction	Table-based horizontal direction 1	Table-based horizontal direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the tool axis direction.
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459, "MACHINE PARAMETER INCORRECT" is issued.



- Table-based horizontal direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based horizontal direction feed in the tilted working plane indexing mode is defined as follows:

Table-based horizontal direction 1:
X direction in the feature coordinate system of the tilted working plane indexing

Table-based horizontal direction 2:
Y direction in the feature coordinate system of the tilted working plane indexing

- Table horizontal direction handle feed

The table horizontal direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> Both the tool axis right-angle direction feed mode signal RGHTH and the table base signal TB_BASE are set to 1.
<3> The state of the first manual handle feed axis selection signals HS1A - HS1Dto make the table horizontal direction handle feed mode effective is set in parameter No.12311 or No.12312.
<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals HS1A - HS1D.

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table horizontal direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Table horizontal direction JOG feed/table horizontal direction incremental feed

The table horizontal direction JOG feed or table horizontal direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> Both the tool axis right-angle direction feed mode signal RGHTH and the table base signal TB_BASE are set to "1".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697.

Ex.) Parameter No.19697 = 3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1 : Table horizontal direction 1 +
- -J1 : Table horizontal direction 1 -
- +J2 : Table horizontal direction 2 +
- -J2 : Table horizontal direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) of a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

Note

1 To perform a 3-dimensional handle feed, the manual handle feed needs to be enabled by setting bit 0 (HPG) of parameter No. 8131 to 1.

2 When a 3-dimensional handle interrupt is performed, rotation axis command execution must not be in progress in automatic operation.

3 When the manual reference position return mode is selected, 3-dimensional manual feed is not enabled.

4 If per-axis interlock is enabled to at least one of 3-dimensional manual feed axes, movement with manual feed is not performed.

5 When the offset value specified for the tool length offset function is used for tool center point rotation feed (when bit 2 (LOD) of parameter No. 19746 is set to 1), the controlled point should generally be shifted. (Set bit 5 (SVC) of parameter No. 19665 to 1.)
In this case, specify the tool length with a radius value.

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.

> **NOTE**
> This function is an optional function.

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.

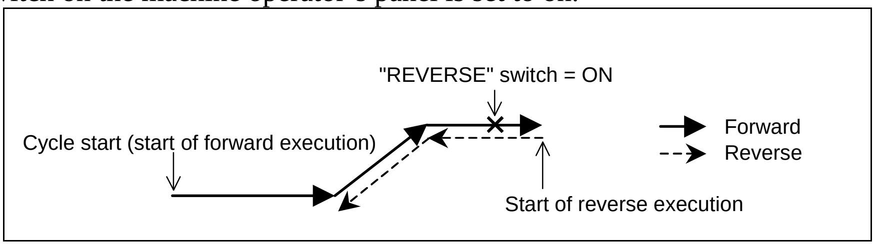


Fig. 2.1 (a)

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

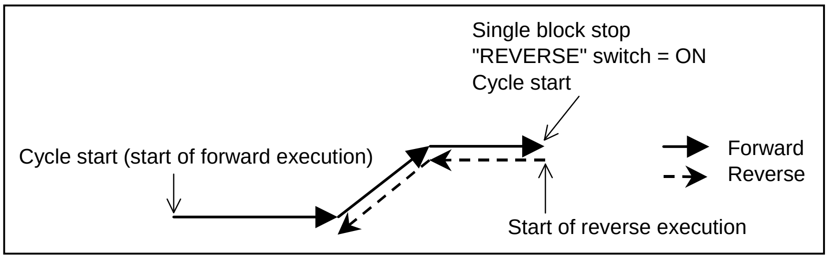


Fig. 2.1 (b)

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

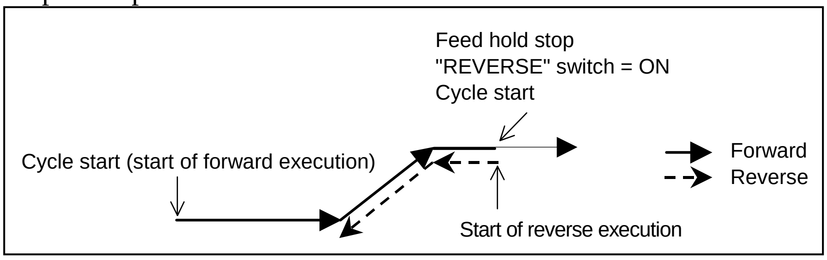


Fig. 2.1 (c)

- 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.

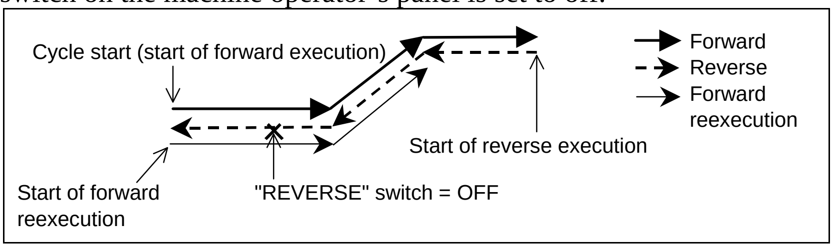


Fig. 2.1 (d)

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

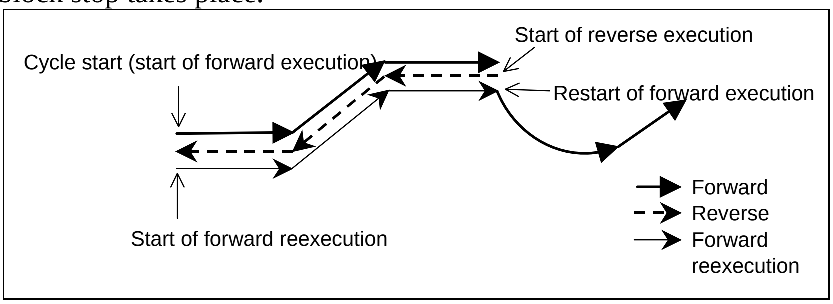


Fig. 2.1 (e)

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

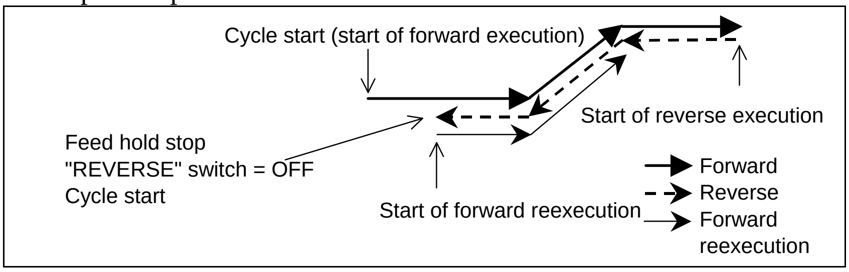


Fig. 2.1 (f)

- 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.

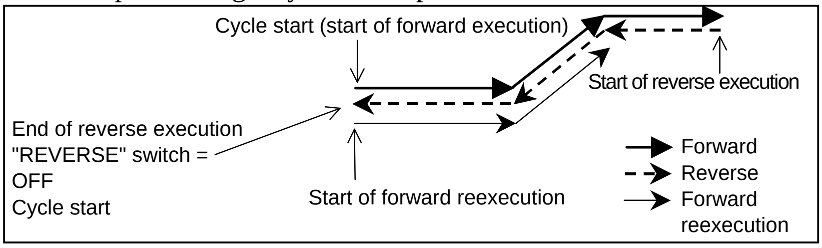


Fig. 2.1 (g)

- 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.

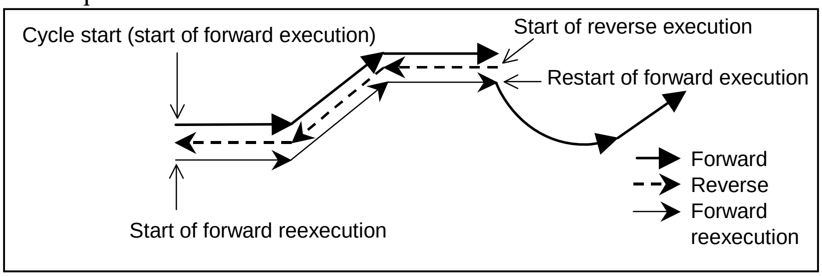


Fig. 2.1 (h)

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.

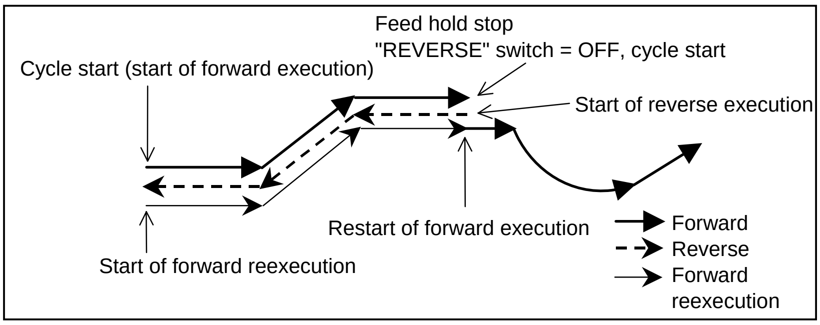


Fig. 2.1 (i)

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.

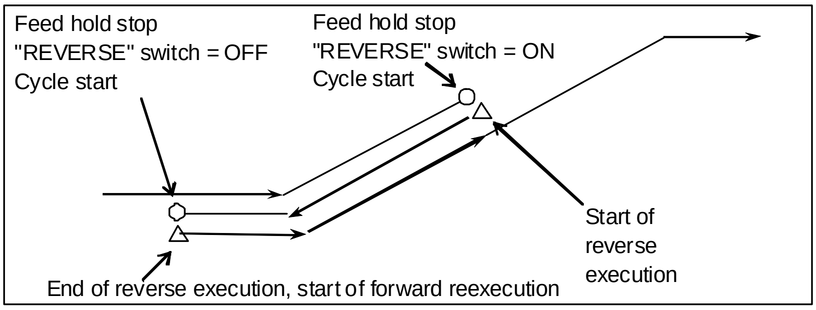


Fig. 2.1 (j)

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)
- Functions related thread cutting (G33,G34,G35,G36)
- 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 advanced preview control / AI contour control (G05.1,G08)
- 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)
- 3-dimensional coordinate system conversion (G68, G69)
- Feature coordinate system (G68.2)
- Figure copying (G72.1,G72.2)
- High precision oscillation function (G81.1)
- 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.

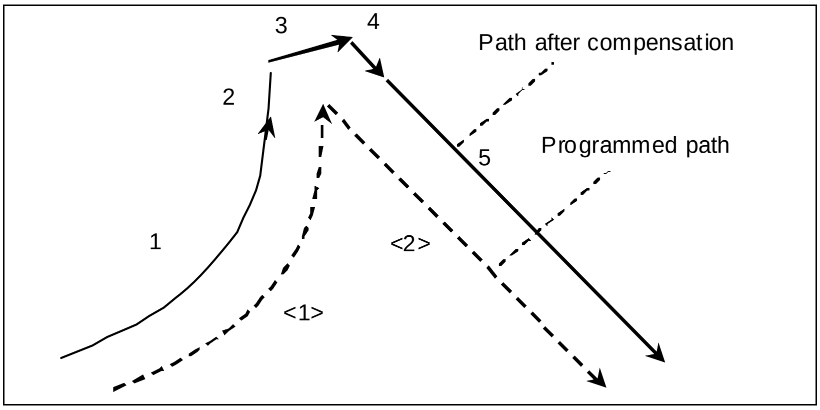


Fig. 2.1 (k) Path when cutter compensation is applied

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.

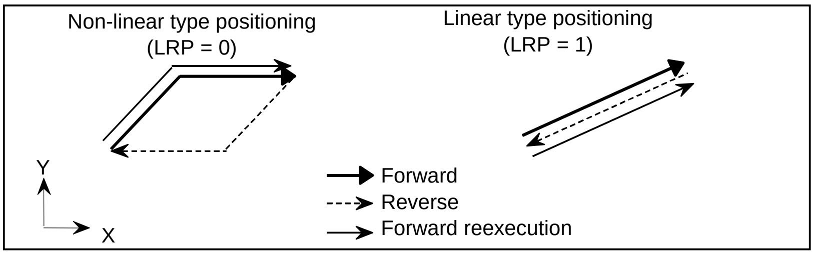


Fig. 2.1 (l)

- 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 measurement (G37)

The skip signal and the measurement position arrival 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.

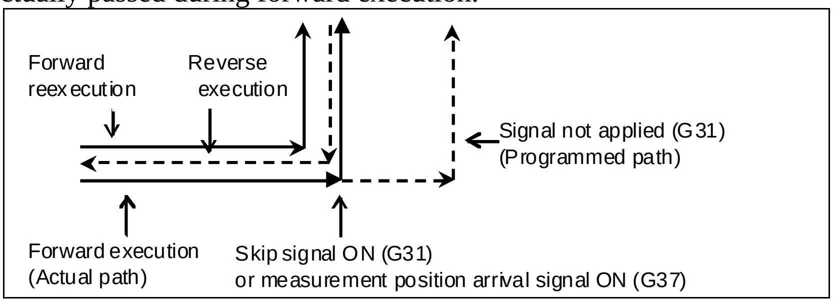


Fig. 2.1 (m)

- 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

Do not initiate any interrupt during reverse execution.
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

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

- Execution macro (macro executor)

Macro executor 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 advanced preview control / AI contour control

When the reverse execution is started in AI advanced preview control / AI contour control mode, a reverse execution ends immediately depending on a program then the backward movement is not possible.
During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI advanced preview control / AI contour control is disabled.

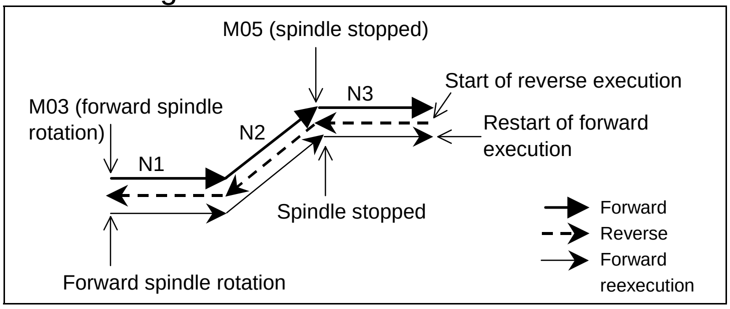
- Display

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**

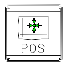
1 Auxiliary functions are output directly even during reverse execution and forward reexecution. Accordingly, the execution status of an auxiliary function during forward execution may be reversed during reverse execution.
Example:
When forward rotation of the spindle (M03) and stop (M05) are specified
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.

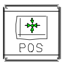


2 To perform reverse execution after a feed hold stop or single block stop operation, be sure to restore the original position if manual intervention has been performed after the stop , then set the "REVERSE" switch to on. Movements made by manual intervention are ignored during reverse execution and 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.

3 SETTING AND DISPLAYING DATA

Chapter 3, "SETTING AND DISPLAYING DATA", consists of the following sections:

3.1 SCREENS DISPLAYED BY FUNCTION KEY 

Press function key  to display or set the following data:

1. 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)

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.

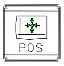
3.1.1 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)

The absolute coordinates of the tool tip, the number of pulses, and a machine axis move amount based on 3-dimensional manual feed are displayed.

NOTE
"3-Dimensional Manual Feed" is an optional function.

Displaying the screen for 3-dimensional manual feed

Procedure

1 Press function key .

2 Press the continuous menu key  several times to display the soft key [3-D MANUAL].

3 Press the soft key [3-D MANUAL] to display the screen of 3-dimensional manual feed.

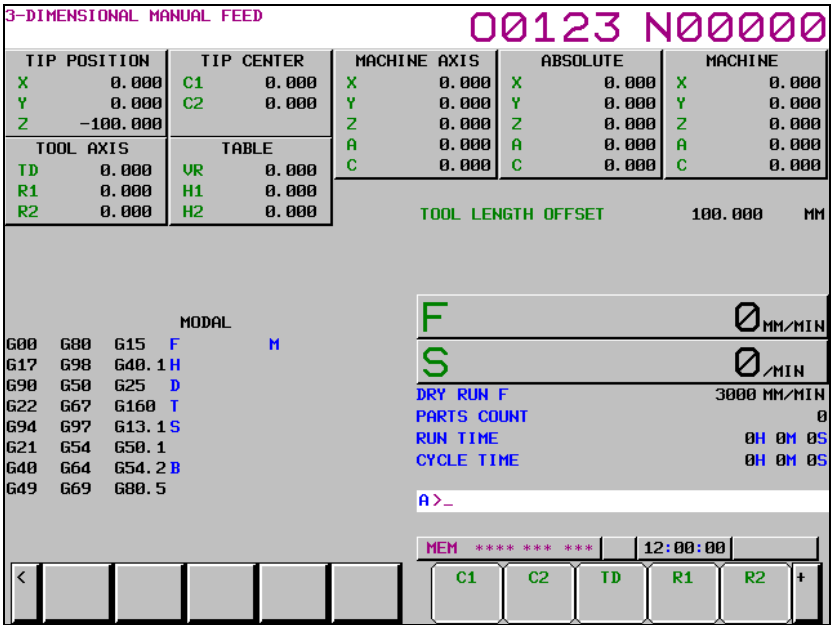


Fig. 3.1.1 (a) 3-dimensional manual feed screen(10.4-inch display unit)

Explanation

- Tool tip position

The addresses of the three basic machine configuration axes for performing 3-dimensional manual feed and the current position of the tool tip are displayed.

- Tool axis reference (number of pulses)

TD

The amount of travel in the tool axis direction in tool axis direction handle feed, tool axis direction jog feed, or tool axis direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

R1

The amount of travel in the first axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis right-angle direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

R2

The amount of travel in the second axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis normal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Tool tip center (number of pulses)

C1

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the first rotary axis is displayed. The unit is the least input increment of the first rotary axis.

C2

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the second rotary axis is displayed. The unit is the least input increment of the second rotary axis.

- Table reference (number of pulses)

VR

The amount of travel in the table reference vertical direction in table reference vertical direction handle feed, table reference vertical direction jog feed, or table reference vertical direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

H1

The amount of travel in the first axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

H2

The amount of travel in the second axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Amounts of machine axis travel

The addresses of machine configuration axes used for 3-dimensional manual feed and the sum of the amount of travel of each axis used for 3-dimensional manual feed are displayed.
The values of the basic three axes (X-axis, Y-axis, and Z-axis), the first rotary axis, and the second rotary axis are displayed in this order.
For the definition of the first rotary axis and second rotary axis, see the description of parameter No. 19680.
When bit 0 (CLR) of parameter No. 13113 is set to 1, the displayed data is cleared by a reset.

- Absolute coordinates, machine coordinates

The absolute coordinates and machine coordinates of all axes are displayed. If too many axes are involved for display on one screen, the remaining axes can be displayed by pressing the soft key [3-D MANUAL] for page feed.

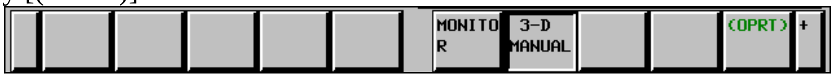
- F (feedrate)

- When bit 3 (CFD) of parameter No. 13113 is set to 0
  The composite feedrate at a control point on a linear axis or rotary axis is displayed.
- When bit 3 (CFD) of parameter No. 13113 is set to 1
  The feedrate of the tool tip is displayed.

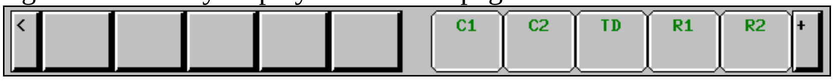
Operation

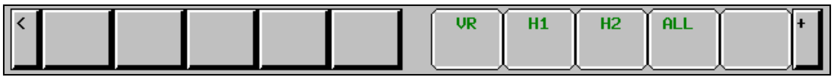
The display of the number of pulses can be cleared by soft key operations.

1 Press soft key [(OPRT)].

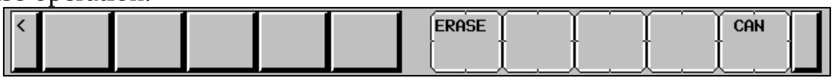


2 Select the soft key corresponding to a function subject to clearing of the amount of travel. Pressing the rightmost soft key displays the second page.

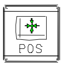




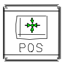
3 Press soft key [ERASE] to clear the amount of travel of the specified function. Press soft key [CAN] to cancel erase operation.

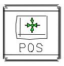


Screens of a 15-inch display unit

Press function key  to display the current overall position display screen that shows the current position of the tool.

The above screen can also display the feedrate, run time, and the number of parts. In addition, a floating reference position can be set on this screen.

Function key  can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key  can also be used to display the screen for displaying the distance moved by handle interruption. See III-4.6 for details on this screen.

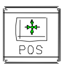
3.1.2 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15-inch Display Unit)

The absolute coordinates of the tool tip, the number of pulses, and a machine axis move amount based on 3-dimensional manual feed are displayed.

> **NOTE**
> "3-Dimensional Manual Feed" is an optional function.

Displaying the screen for 3-dimensional manual feed

Procedure

1 Press function key .

2 Press the vertical soft key [3-D MANUAL] to display the screen of 3-dimensional manual feed.

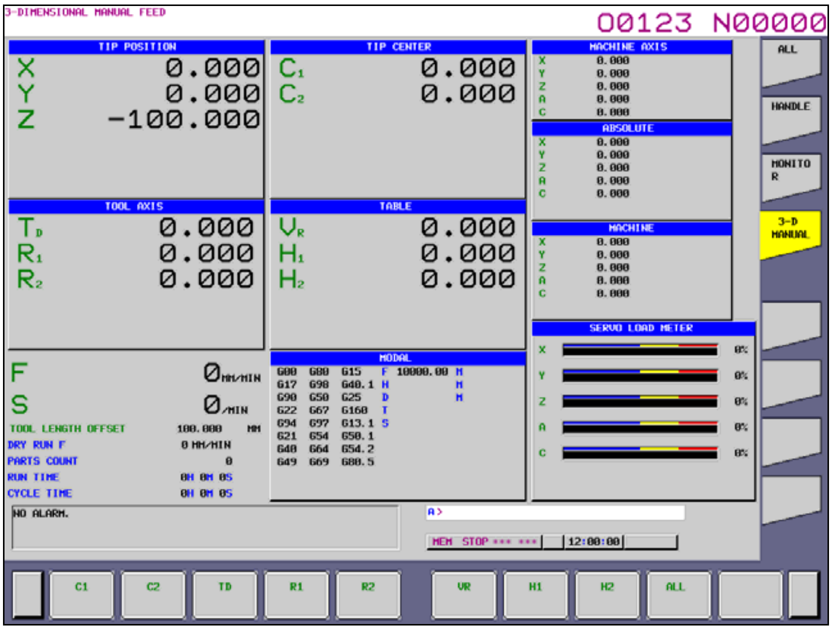


Fig. 3.1.2 (a) 3-dimensional manual feed screen(15-inch display unit)

Explanation

- Tool tip position

The addresses of the three basic machine configuration axes for performing 3-dimensional manual feed and the current position of the tool tip are displayed.

- Tool axis reference (number of pulses)

TD

The amount of travel in the tool axis direction in tool axis direction handle feed, tool axis direction jog feed, or tool axis direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

R1

The amount of travel in the first axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis right-angle direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

R2

The amount of travel in the second axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis normal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Tool tip center (number of pulses)

C1

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the first rotary axis is displayed. The unit is the least input increment of the first rotary axis. For the definition of the first rotary axis, see the description of parameter No. 19680.

C2

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the second rotary axis is displayed. The unit is the least input increment of the second rotary axis. For the definition of the second rotary axis, see the description of parameter No. 19680.

- Table reference (number of pulses)

VR

The amount of travel in the table reference vertical direction in table reference vertical direction handle feed, table reference vertical direction jog feed, or table reference vertical direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

H1

The amount of travel in the first axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

H2

The amount of travel in the second axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Amounts of machine axis travel

The addresses of machine configuration axes used for 3-dimensional manual feed and the sum of the amount of travel of each axis used for 3-dimensional manual feed are displayed.
The values of the basic three axes (X-axis, Y-axis, and Z-axis), the first rotary axis, and the second rotary axis are displayed in this order.
For the definition of the first rotary axis and second rotary axis, see the description of parameter No. 19680.
When bit 0 (CLR) of parameter No. 13113 is set to 1, the displayed data is cleared by a reset.

- Absolute coordinates, machine coordinates

The absolute coordinates and machine coordinates of all axes are displayed. If too many axes are involved for display on one screen, the remaining axes can be displayed by pressing the vertical soft key [3-D MANUAL] for page feed.

- F (feedrate)

- When bit 3 (CFD) of parameter No. 13113 is set to 0
  The composite feedrate at a control point on a linear axis or rotary axis is displayed.
- When bit 3 (CFD) of parameter No. 13113 is set to 1
  The feedrate of the tool tip is displayed.

Operation

The display of the number of pulses can be cleared to 0 with horizontal soft keys.

1 Select the horizontal soft key corresponding to a function for which the display of the amount of travel is to be cleared.

2 Press horizontal soft key [ERASE] to clear the display of the amount of travel of the specified function, or press horizontal soft key [CAN] to cancel the operation.

3.2 SCREENS DISPLAYED BY FUNCTION KEY PROG

Press function key PROG to display or set the following data:

1. Screen for Assistance in Entering Tilted Working Plane Indexing

Refer to the Operator's Manual (Common to Lathe System/Machining Center System) (B-64604EN) for explanations about how to display or specify the other types of data.

3.2.1 Screen for Assistance in Entering Tilted Working Plane Indexing

The screens for assistance in entering tilted working plane indexing (hereinafter referred to as guidance screens) include a command type selection screen and a tilted working plane data setting screen. The command type selection screen is used to select a tilted working plane indexing. The tilted working plane data setting screen is used to set specified tilted working plane data required for the selected command. By making settings and performing operations on these guidance screens, a tilted working plane indexing block can be created.
The created block is reflected as a new insertion to a program being edited or as a modification to an existing block.
This function can be enabled by setting bit 1 (GGD) of parameter No. 11304 to 1.

NOTE
"Tilted working plane indexing" is an optional function.

Creation of a new block

The following describes the procedure for creating a tilted working plane indexing block on guidance screens and for inserting the block to a program being edited on a program editing screen.

1 On a program editing screen, display a program to which you want to insert a tilted working plane indexing block.
The foreground editing screen, background editing screen, or MDI editing screen should be displayed.

- Displaying the foreground editing screen
<1> Select the EDIT mode.
<2> Press function key PROG.
<3> Press soft key [PROGRAM].

- Displaying the background editing screen
<1> Press function key PROG.
<2> Press soft key [FOLDER].
<3> Press any of the cursor keys to move the cursor to a program to be edited in the background.
<4> Press INPUT key.

- Displaying the MDI editing screen
<1> Select the MDI mode.
<2> Press function key PROG.

<3> Press soft key [PROGRAM].

The program editing screen is displayed.

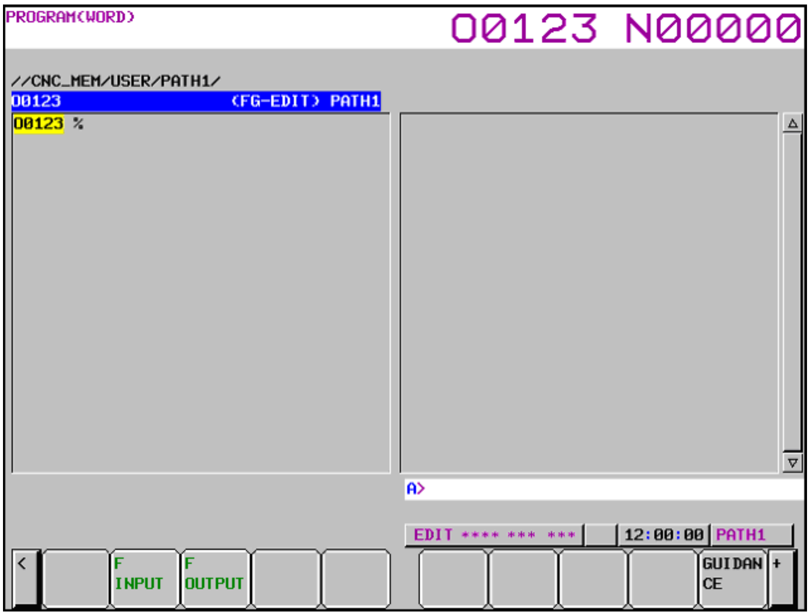


Fig.3.2.1 (a)

2 Press any of the cursor keys to move the cursor to a position where you want to insert a block. Note that a block created on the guidance screens is inserted after the block at the cursor position. (If the block at the cursor position includes a tilted working plane indexing, the existing block is modified. See "Modification to an existing block" below.)

3 Press soft key [(OPRT)].

4 Press continuous menu key ▷ several times, and then press soft key [GUIDANCE]. The command type selection screen is displayed.

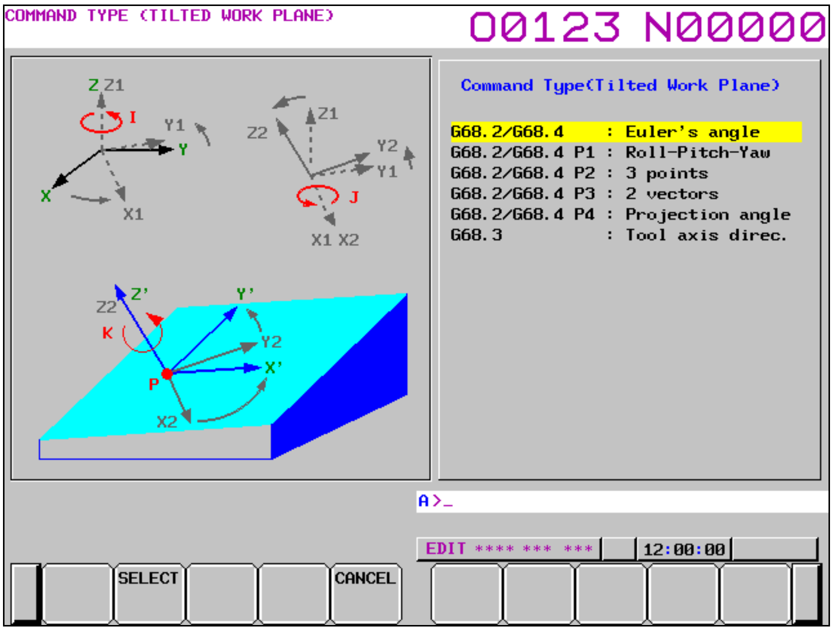


Fig.3.2.1 (b)

5 Select a command type with any of the cursor keys, and then press soft key [SELECT]. The tilted working plane data setting screen is displayed.

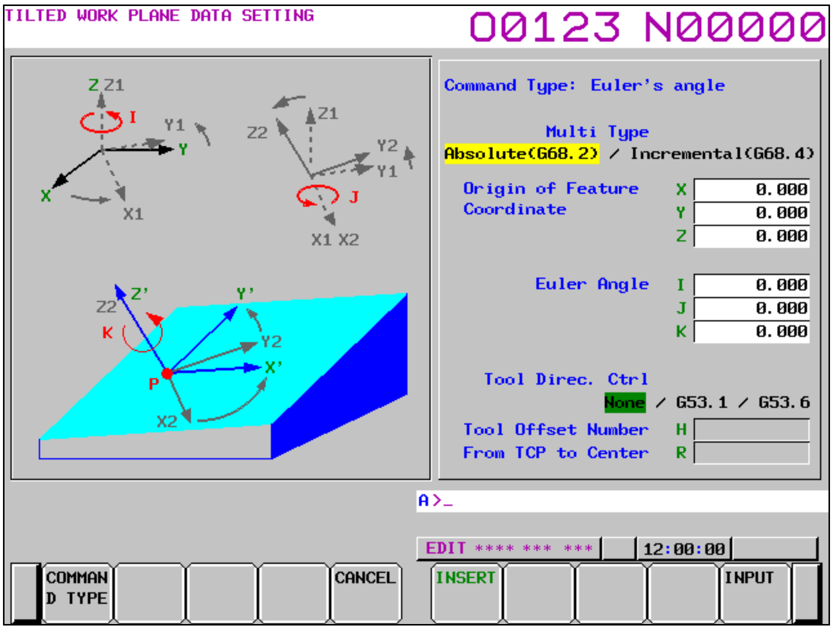


Fig.3.2.1 (c)

6 Enter command data for the setting items.
7 Press soft key [INSERT].

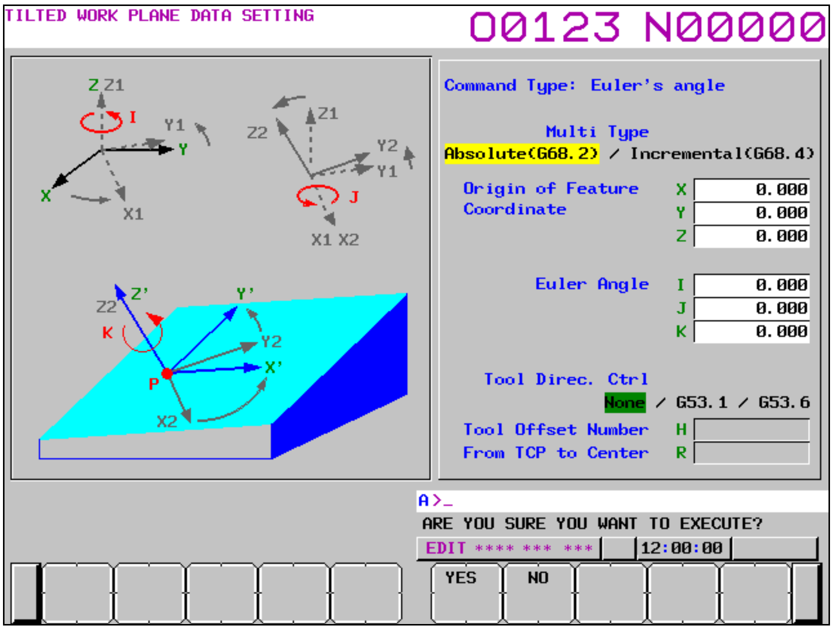


Fig.3.2.1 (d)

8 Press soft key [YES].
This takes you back to the program editing screen, where the new block is inserted after the block at the cursor position.

Modification to an existing block

The following describes the procedure for replacing a block in a program being edited on a program editing screen, with a tilted working plane indexing block created on a guidance screen.

1 On a program editing screen, display a program to be edited.
(For the procedure for displaying a program editing screen, see step 1 in "Creation of a new block".)

The program editing screen is displayed.

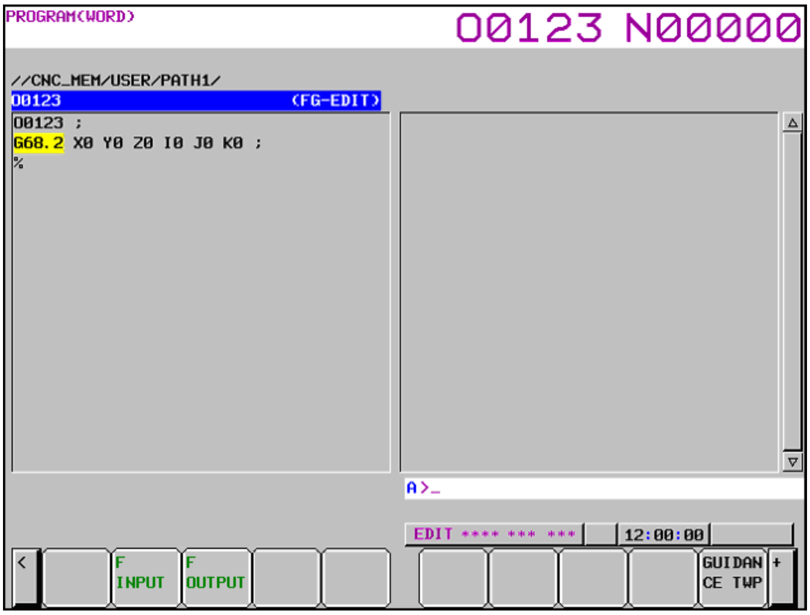


Fig.3.2.1 (e)

2 Press any of the cursor keys to move the cursor to a block to be modified.
For a command extending over more than one block, move the cursor to the first block.

3 Press soft key [(OPRT)].

4 Press continuous menu key ▷ several times, and then press soft key [GUIDANCE TWP]. The tilted working plane data setting screen is displayed.

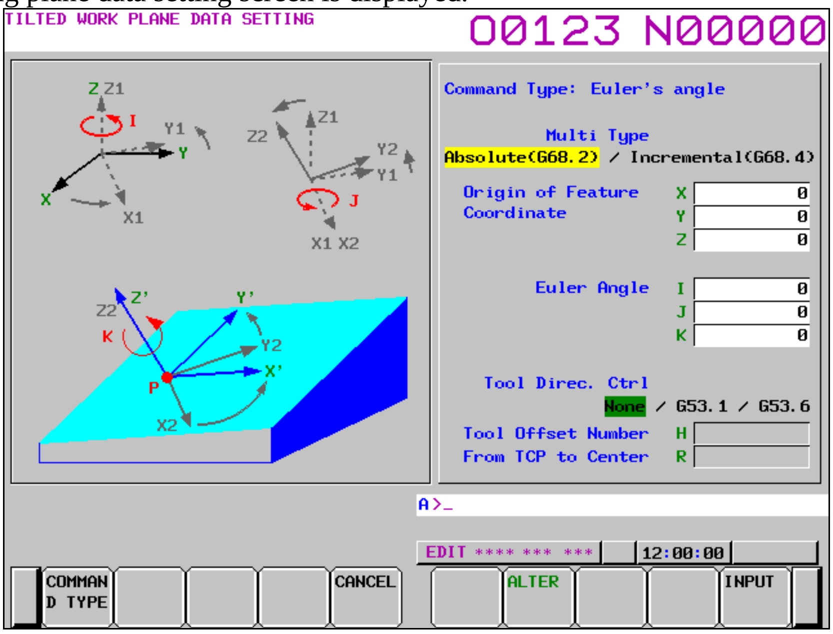


Fig.3.2.1 (f)

5 Enter command data for setting items to be modified.

6 Press soft key [ALTER].

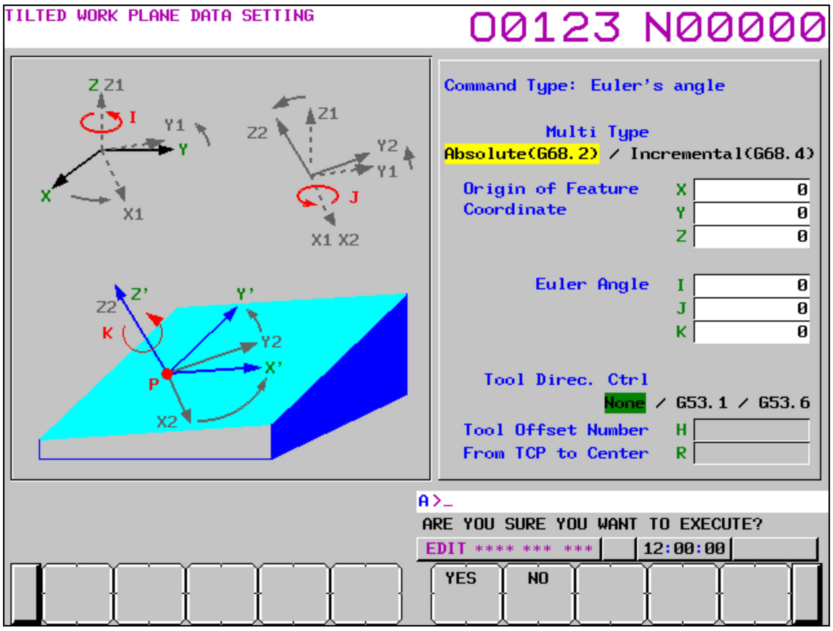


Fig.3.2.1 (g)

7 Press soft key [YES].
This takes you back to the program editing screen, where the block at the cursor position is replaced.

Guidance screen cancellation

Pressing soft key [CANCEL] on a guidance screen takes you back to the program editing screen. At this time, the data that has been set on the guidance screen is discarded.

NOTE

1 In addition to the above operation, the following operations also cancel a guidance screen. The data that has been set on the guidance screen is discarded.
   - When bit 7 (CPG) of parameter No. 11302 is 1 (setting for automatically switching between program-related screens according to CNC mode), the CNC mode is changed.
   - When a guidance screen is displayed from the foreground editing screen, the CNC mode is changed to a mode other than EDIT, TJOG, or THND.
   - When a guidance screen is displayed from the MDI editing screen, the CNC mode is changed to a mode other than MDI.
   - When a guidance screen is displayed on a 15-inch display unit, the screen is switched by a vertical soft key.
   - The screen is switched by an MDI key.
   - The screen is switched by a path select signal.
   - An event that causes screen switching has occurred, including occurrence of an alarm, display of an operator message, or signal-based display of a screen (such as a tool compensation screen, workpiece shift screen, workpiece coordinate system setting screen, or C Language Executor screen).

2 If MDI key 

is pressed after switching from a guidance screen to another screen, a program editing screen is displayed, instead of the guidance screen.

Notes

- Conditions under which soft key [GUIDANCE TWP] is displayed
  Soft key [GUIDANCE TWP] is displayed on a program editing screen under the following conditions:
  1 Foreground editing screen
     - The CNC mode is EDIT, TJOG, or THND.
     - The editing and display are not prohibited for a program to be edited.
  2 Background editing screen
     - A program to be edited is not in reference mode.
     - The editing and display are not prohibited for a program to be edited.
  3 MDI editing screen
     - The CNC mode is MDI.

- Screen displayed when soft key [GUIDANCE TWP] is pressed
  When soft key [GUIDANCE TWP] is pressed, which guidance screen is displayed is automatically determined according to the conditions described below.
  1 When a tilted working plane indexing is not included in a block at the cursor position on a program editing screen
     The command type selection screen is displayed. A block created on the guidance screens is inserted after the block at the cursor position on the program editing screen.
  2 When a tilted working plane indexing is included in a block at the cursor position on a program editing screen
     The tilted working plane data setting screen is displayed, which shows the data for the tilted working plane indexing in the block at the cursor position on the program editing screen.
     When a block of a tool axis direction control command exists immediately after the tilted working plane indexing, the command data for the block is also shown. The tilted working plane indexing in the block at the cursor position on the program editing screen is replaced with a block created on the guidance screen.

> **NOTE**
> If the CNC is in the reset state or emergency stop state when soft key [GUIDANCE TWP] is pressed on the foreground editing screen or MDI editing screen, the warning "PROGRAM READ FAILED" appears, and the operation cannot be continued. (Soft keys other than [CANCEL] are not displayed.) Press soft key [CANCEL] to return to the program editing screen, and then press soft key [GUIDANCE TWP] again.

3.2.1.1 Command type selection screen

The command type selection screen is used to select the type of a tilted working plane indexing you want to insert into a program to be edited. One of the following command types can be selected:

- G68.2 / G68.4 (Euler’s angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

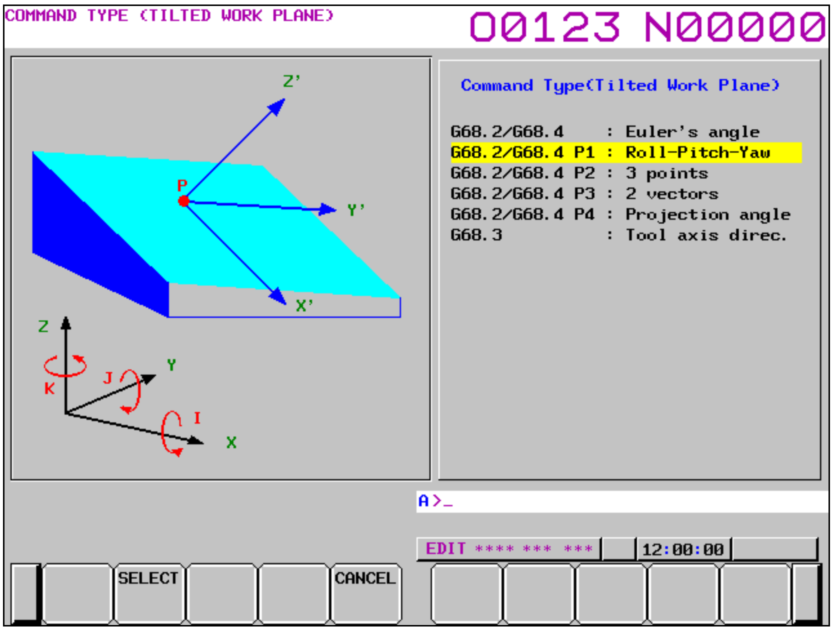


Fig. 3.2.1.1 (a) Command type selection screen (10.4-inch display unit)

Selection of a command type

1. Press cursor key ⬆ or ⬇ to move the cursor to a command type you want to select. As the cursor is moved, the figure corresponding to the command type at the cursor position is displayed.
2. When soft key [SELECT] is pressed, the command type at the cursor position is accepted, and the tilted working plane data setting screen is displayed.

> **NOTE**
> If the warning "PROGRAM READ FAILED" appears when the command type selection screen is displayed, the operation cannot be continued. (Soft keys other than [CANCEL] are not displayed.) Press soft key [CANCEL] to return to the program editing screen, and then press soft key [GUIDANCE TWP] again.

3.2.1.2 Tilted working plane data setting screen

The tilted working plane data setting screen is used to set specified tilted working plane data required for a tilted working plane indexing of the type that has been selected on the command type selection screen or has been selected when soft key [GUIDANCE TWP] is pressed.

Different types of tilted working plane data setting screen are provided for different command types. For details of each type of tilted working plane data setting screen, see "Details of the tilted working plane data setting screen".

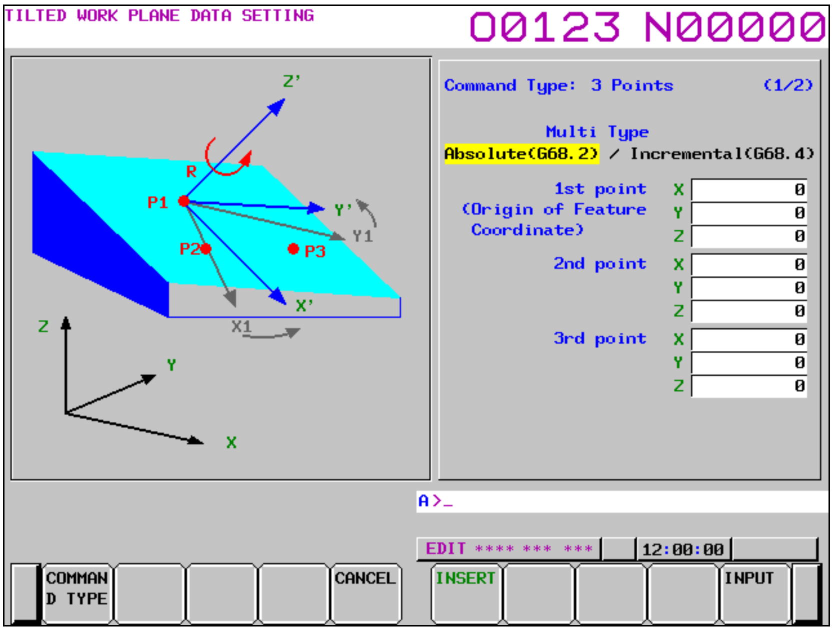


Fig. 3.2.1.2 (a) Tilted working plane data setting screen-3 points specification(10.4-inch display unit)

Display of the tilted working plane data setting screen

When a new block is created, the initial values are set in the setting and selection items.
When an existing block is modified, the command data for the block at the cursor position on the program editing screen is reflected in the setting and selection items.

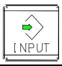
NOTE

1. For existing block modification, if the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the parameters for the block(s) before the cursor are not reflected in the setting and selection items, but only the parameters for the block(s) after the cursor are reflected. If no data is to be reflected in a setting or selection item, the initial value is set in the item.
2. For existing block modification, only the commands with parameter-specified axis names are reflected in the setting items on the screen. (The values specified in data for commands with incorrect axis names are not reflected in the setting items.)

Command data input

- Item for which to enter a value

Press cursor key  or  to move the cursor to an item you want to set.

Enter a value, and then press the  key or soft key [INPUT].

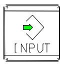
Example)

When the origin of a feature coordinate system is set as shown above, addresses X, Y, and Z are specified as follows:

G68.2 X0.001 Y0.01 Z1000 ...

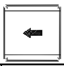
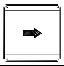
For two items "Tool Offset Number" and "From TCP to center", the state with no command can be set by deleting the specified values according to the following procedure.

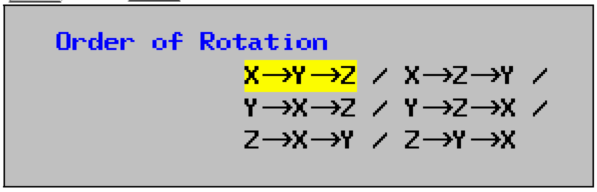
1 Press cursor key  or  to move the cursor to the item "Tool Offset Number" or "From TCP to Center".

2 Press  or soft key [INPUT] without entering anything.

- Item to be selected from a list

1 Press cursor key  or  to move the cursor to an item you want to set.

2 Press cursor key  or  to move the cursor to an item you want to select.



Example)
Order of rotation for Roll-Pitch Yaw angle

Insertion of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen does not include a tilted working plane indexing, soft key [INSERT] is displayed on the tilted working plane data setting screen. An operation for inserting a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block insertion, see "Limitation".

1 Press soft key [INSERT].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and soft keys [YES] and [NO] are displayed.

2 Press soft key [YES].
A block is created from the command type and command data and then is inserted after the block at the cursor position in the program to be edited. After block insertion, the guidance screen is closed and the program editing screen is displayed.
Pressing soft key [NO] takes you back to the tilted working plane data setting screen.

Example)
G00 X0.; When the guidance screen is displayed and the 3-point specification is selected as a
↓ command type for block insertion, a created block is inserted after the block at the cursor position.
G00 X0.;
G68.2 P2 Q0...
G68.2 P2 Q1...
G68.2 P2 Q2...
G68.2 P2 Q3...

Replacement of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen includes a tilted working plane indexing, soft key [ALTER] is displayed on the tilted working plane data setting screen. An operation for replacing a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block replacement, see "Limitation".

1 Press soft key [ALTER].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and soft keys [YES] and [NO] are displayed.
2 Press soft key [YES].
A block is created from the command type and command data and then replaces the tilted working plane indexing in the block at the cursor position on the program editing screen. After block replacement, the guidance screen is closed and the program editing screen is displayed.
Pressing soft key [NO] takes you back to the tilted working plane data setting screen.

NOTE
1 If a block to be replaced includes a command other than a tilted working plane indexing, the command is deleted during block replacement. However, only the sequence number at the beginning is preserved.
2 If the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the block(s) before the cursor are not replaced. These blocks remain unchanged after replacement.
3 If soft key [GUIDANCE TWP] is pressed when the cursor is placed on a block of a tool axis direction control command, the command type selection screen is displayed in new insertion mode. A block created on the guidance screens is inserted after the block of the tool axis direction control command.

Limitation

The following lists the warnings that may be issued at the time of block insertion or replacement.
If a warning is displayed, return to the program editing screen with soft key [CANCEL] and press soft key [GUIDANCE TWP] again, or eliminate the cause of the warning and retry the operation.

Warning	Description
“PROGRAM WRITE FAILED”	- The guidance screen was displayed from the foreground editing screen or MDI editing screen, and a block insertion or replacement operation was performed when the CNC was in the reset state or emergency stop state.
“PROGRAM CANNOT BE WRITTEN”	- A block insertion or replacement operation was performed after a program to be edited was updated due to program downloading by an external application while the guidance screen was displayed. - A block insertion or replacement operation was performed after the start of the main program was located by a reset while the guidance screen was displayed.
“WRITE PROTECT”	- A block insertion or replacement operation was performed when the editing or display was prohibited for a program to be edited. - A block insertion or replacement operation was performed when the memory protection signal (KEY signal) for program registration or editing was off. - A block insertion or replacement operation was performed when the operation level of the 8-level data protection function was lower than the protection level of a part program editing operation.

Display of the command type selection screen

When soft key [COMMAND TYPE] is pressed, the command type selection screen is displayed. When the command type is changed on the command type selection screen, the values set on the tilted working plane data setting screen are cleared.
When a tilted working plane indexing is included in the block at the cursor position on the program editing screen, you can display the guidance screen, change the command type on the command type selection screen, and then perform block replacement. In this case, the block at the cursor position on the program editing screen is replaced with a block created from the command type and command data that have been set.

3.2.1.3 Details of the tilted working plane data setting screen

The following six tilted working plane indexing are supported.
For details of the commands, see Section, "TILTED WORKING PLANE INDEXING".

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

G68.2 / G68.4(Euler's angle)

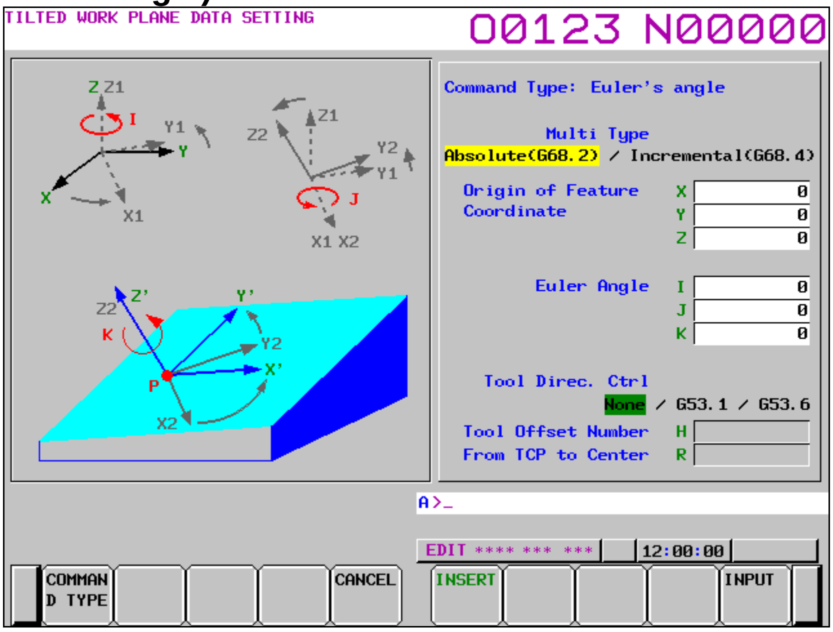


Fig. 3.2.1.3 (a) Tilted working plane data setting screen-Euler's angle(10.4-inch display unit)

- Multi Type

  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate

  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Euler's angle

  I : Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  This rotation determines coordinate system 1 (X1-Y1-Z1) from the workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type) (X-Y-Z).

  J : Specify an angle of rotation around the X-axis of coordinate system 1.

This rotation determines coordinate system 2 (X2-Y2-Z2) from coordinate system 1 (X1-Y1-Z1).

K : Specify an angle of rotation around the Z-axis of coordinate system 2.
After this rotation, a feature coordinate system is obtained by shifting the origin of the workpiece coordinate system by the coordinates specified in "Origin of Feature Coordinate".

G68.2 / G68.4(Roll-Pitch-Yaw angle)

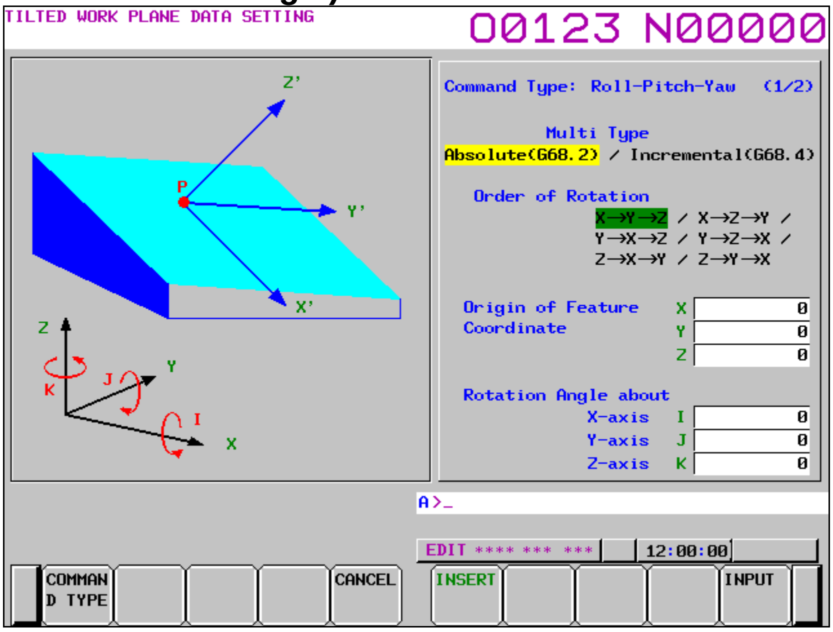


Fig. 3.2.1.3 (b) Tilted working plane data setting screen- Roll-Pitch-Yaw angle(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Order of Rotation
  Select an order in which the X-axis, Y-axis, and Z-axis are rotated in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The selectable rotation orders are as follows:

	1st rotation axis	2nd rotation axis	3rd rotation axis
X→Y→Z	X-axis	Y-axis	Z-axis
X→Z→Y	X-axis	Z-axis	Y-axis
Y→X→Z	Y-axis	X-axis	Z-axis
Y→Z→X	Y-axis	Z-axis	X-axis
Z→X→Y	Z-axis	X-axis	Y-axis
Z→Y→X	Z-axis	Y-axis	X-axis

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

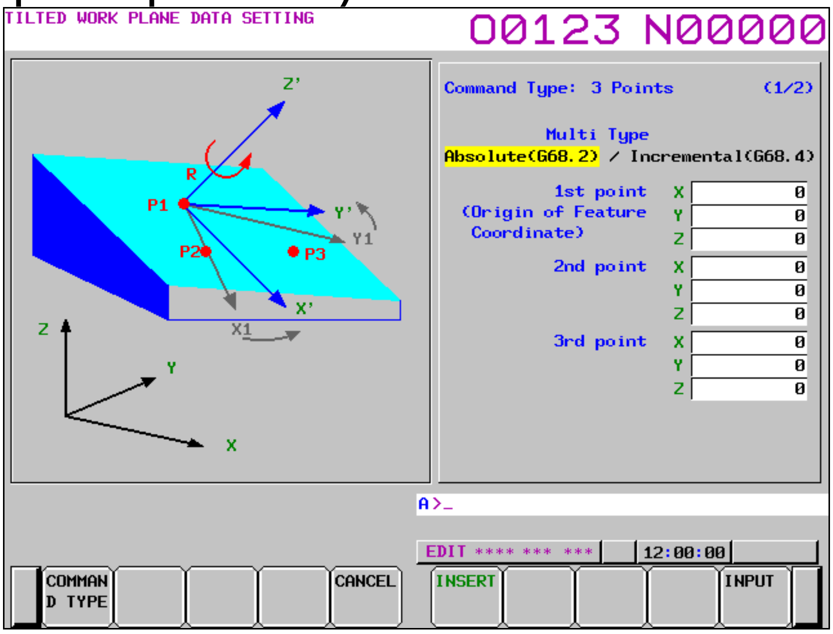
- Rotation Angle about X-axis

Specify an angle of rotation around the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Y-axis
  Specify an angle of rotation around the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Z-axis
  Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(3 points specification)



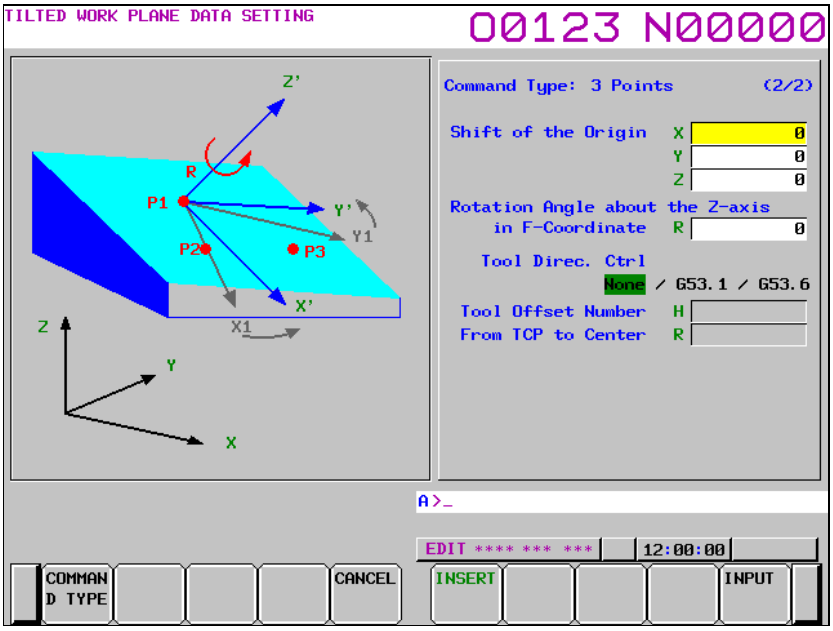


Fig. 3.2.1.3 (c) Tilted working plane data setting screen- 3 points specification(10.4-inch display unit)

- Multi Type
  Absolute:

It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

Incremental:

It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Shift of the Origin
  Specify, in a feature coordinate system, an amount of shift from the feature coordinate system origin specified for the 1st point (point P1).

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

- 1st point (Origin of Feature Coordinate)
  Specify the origin (X, Y, and Z of point P1) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- 2nd point
  Specify the coordinates (X, Y, and Z of point P2) of the 2nd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The 1st point and 2nd point determine the X-axis of the feature coordinate system.

- 3rd point
  Specify the coordinates (X, Y, and Z of point P3) of the 3rd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). Among the directions normal to the X-axis, a direction with a smaller angle relative to the P1 → P3 vector is the Y-axis of the feature coordinate system.

G68.2 / G68.4(2 vectors specification)

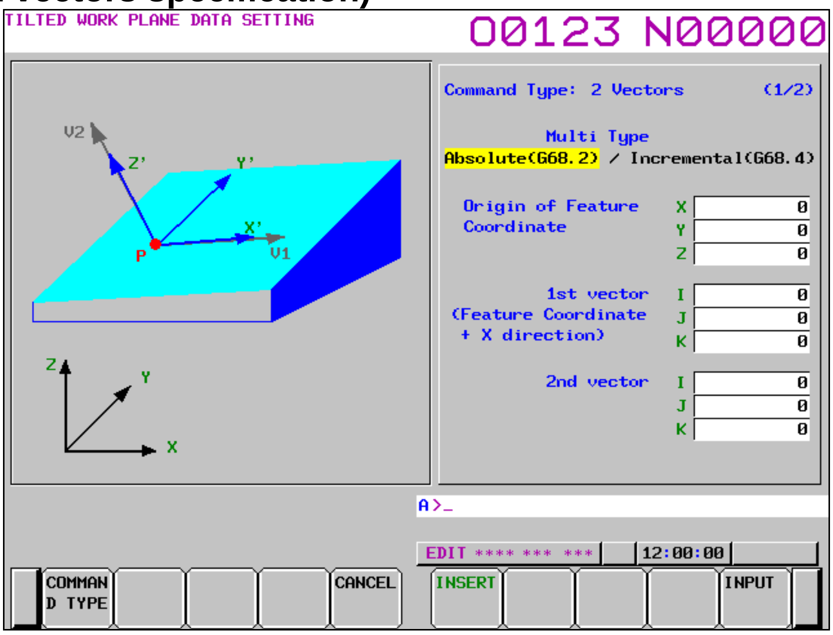


Fig. 3.2.1.3 (d) Tilted working plane data setting screen-2 vectors specification(10.4-inch display unit)

- Multi Type
  Absolute:

It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

Incremental:

It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V1 (X') vector
  Specify the X-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V2 (Z') vector
  Specify the Z-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(Projection angle)

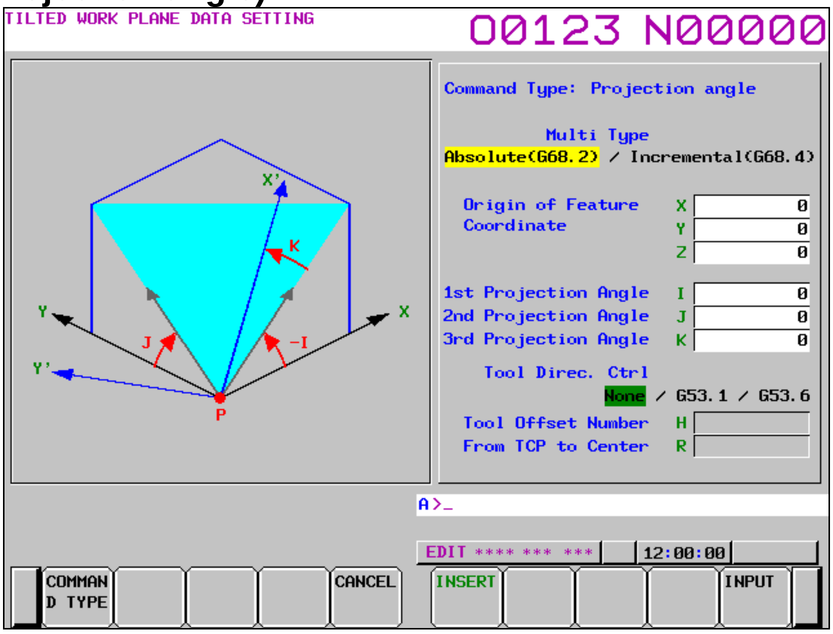


Fig. 3.2.1.3 (e) Tilted working plane data setting screen-Projection angle(10.4-inch display unit)

- Multi Type
  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Projection angle
  I : Specify a projection angle relative to the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  J : Specify a projection angle relative to the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  K : Specify an angle of rotation around the Z-axis of a feature coordinate system.

G68.3(Tool Axis Direction)

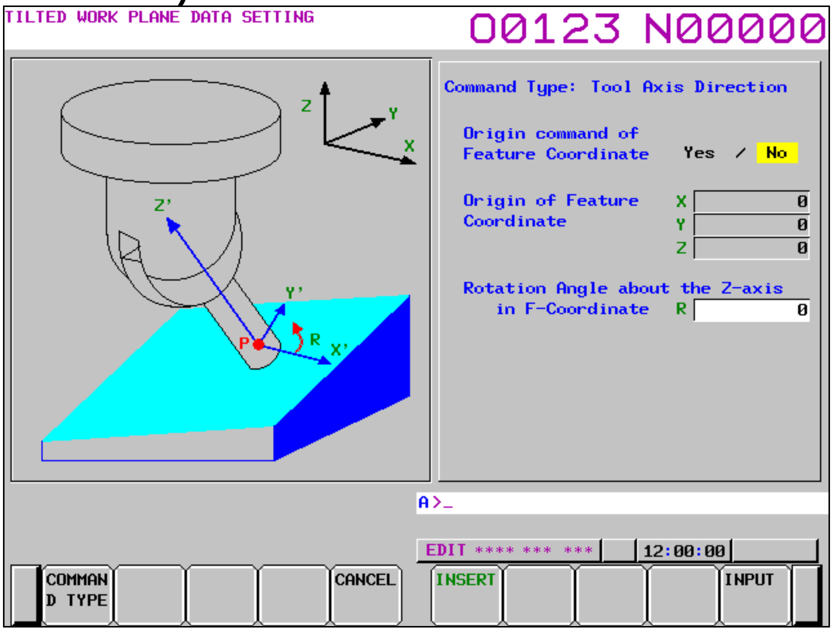


Fig. 3.2.1.3 (f) Tilted working plane data setting screen-Tool Axis Direction(10.4-inch display unit) (When "No" is selected in "Origin command of Feature Coordinate")

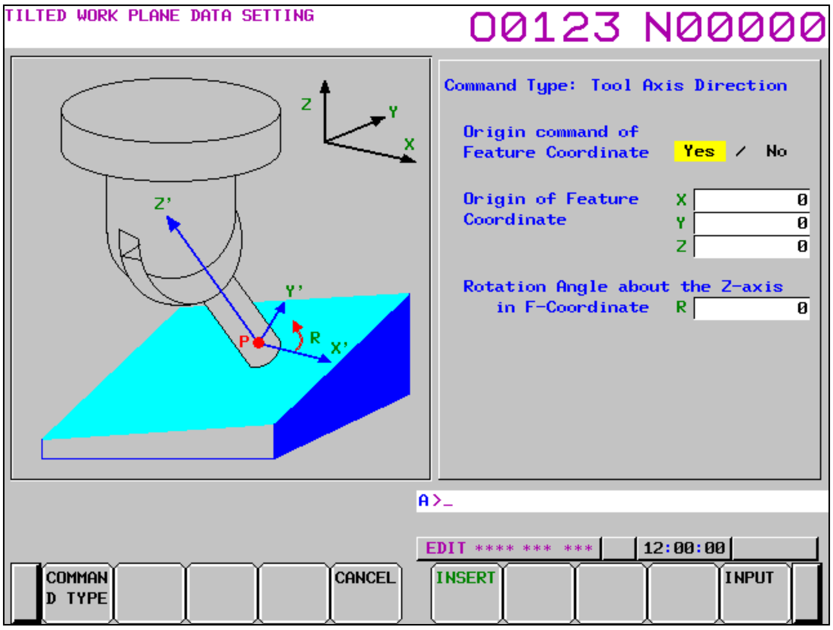


Fig. 3.2.1.3 (g) Tilted working plane data setting screen-Tool Axis Direction(10.4-inch display unit) (When "Yes" is selected in "Origin command of Feature Coordinate")

- Origin command of Feature Coordinate
  Select whether to specify a feature coordinate system origin.
  Yes : A feature coordinate system origin is specified.

No : A feature coordinate system origin is not specified.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system.
  This setting cannot be made if "No" is selected in "Origin command of Feature Coordinate".

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

3.2.1.4 Limitation

This function is supported on 10.4-inch and 15-inch display units. This function is not displayed on 8.4-inch display unit.

Screens of a 15-inch display unit

This section describes the screens displayed by pressing function key PROG. The screens include a program editing screen, program folder list display screen, and screens for displaying the command states of the program currently being executed.

1. Program screen
2. Program folder screen
3. Next block display screen
4. Program check screen

On the program screen, you edit the program that is currently selected, and display the block that is currently executed during program operation. In MDI mode, you also edit an MDI operation program, and display the block that is currently executed.

3.2.2 Screen for Assistance in Entering Tilted Working Plane Indexing (15-inch Display Unit)

The screens for assistance in entering tilted working plane indexing (hereinafter referred to as guidance screens) include a command type selection screen and a tilted working plane data setting screen. The command type selection screen is used to select a tilted working plane indexing. The tilted working plane data setting screen is used to set specified tilted working plane data required for the selected command. By making settings and performing operations on these guidance screens, a tilted working plane indexing block can be created.
The created block is reflected as a new insertion to a program being edited or as a modification to an existing block.
This function can be enabled by setting bit 1 (GGD) of parameter No. 11304 to 1.

> **NOTE**
> "Tilted working plane indexing" is an optional function.

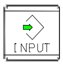
Creation of a new block

The following describes the procedure for creating a tilted working plane indexing block on guidance screens and for inserting the block to a program being edited on a program editing screen.

1 On a program editing screen, display a program to which you want to insert a tilted working plane indexing block.

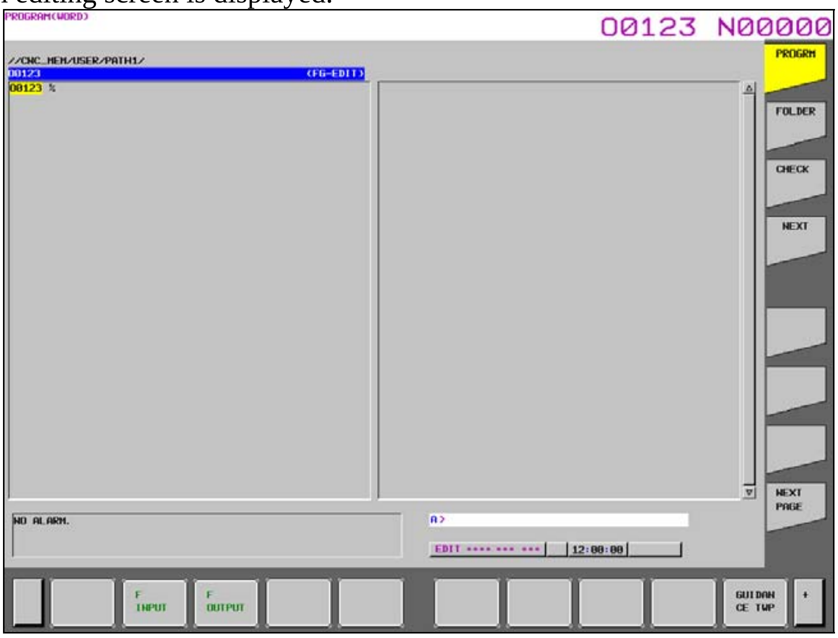
The foreground editing screen, background editing screen, or MDI editing screen should be displayed.

- Displaying the foreground editing screen
  <1> Select the EDIT mode.
  <2> Press function key .
  <3> Press vertical soft key [PROGRAM].

- Displaying the background editing screen
  <1> Press function key .
  <2> Press soft key [FOLDER].
  <3> Press any of the cursor keys to move the cursor to a program to be edited in the background.
  <4> Press  key.

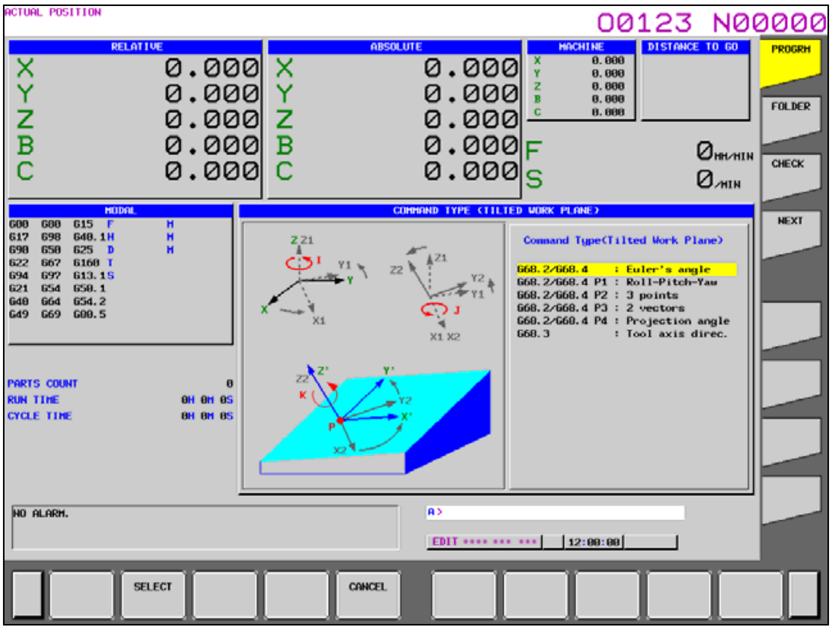
- Displaying the MDI editing screen
  <1> Select the MDI mode.
  <2> Press function key .
  <3> Press vertical soft key [PROGRAM].

The program editing screen is displayed.

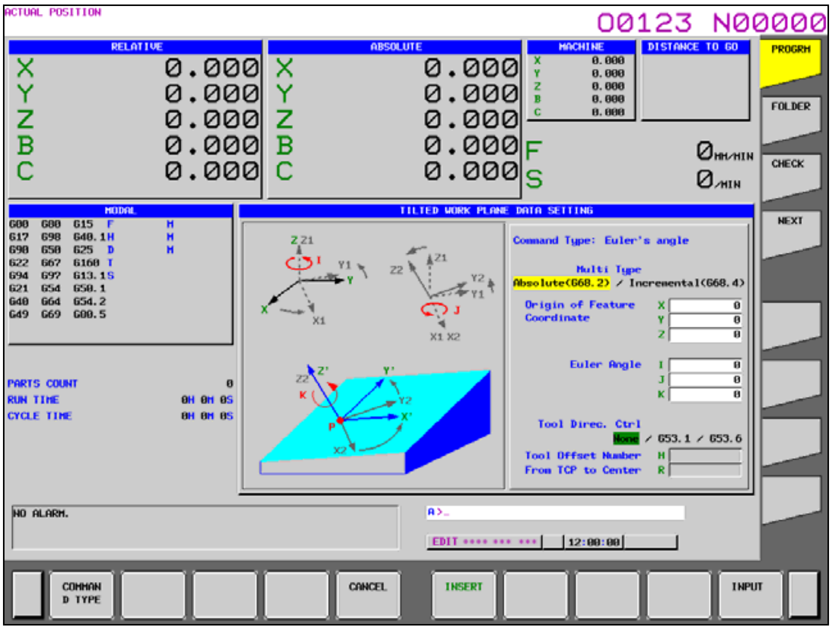


2 Press any of the cursor keys to move the cursor to a position where you want to insert a block.
Note that a block created on the guidance screens is inserted after the block at the cursor position. (If the block at the cursor position includes a tilted working plane indexing, the existing block is modified. See "Modification to an existing block" below.)

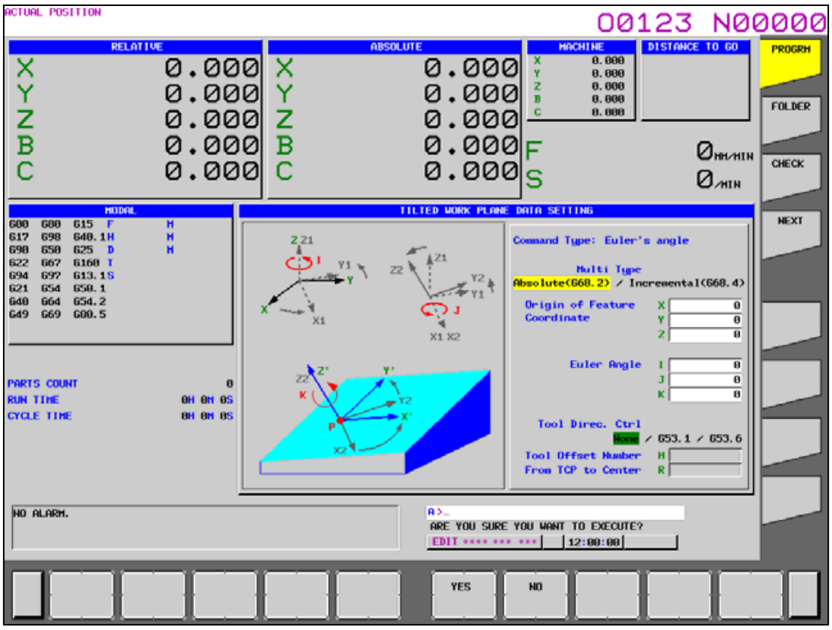
3 Press continuous menu key  several times, and then press horizontal soft key [GUIDANCE TWP].
The command type selection screen is displayed.



4 Select a command type with any of the cursor keys, and then press horizontal soft key [SELECT]. The tilted working plane data setting screen is displayed.



5 Enter command data for the setting items.
6 Press horizontal soft key [INSERT].

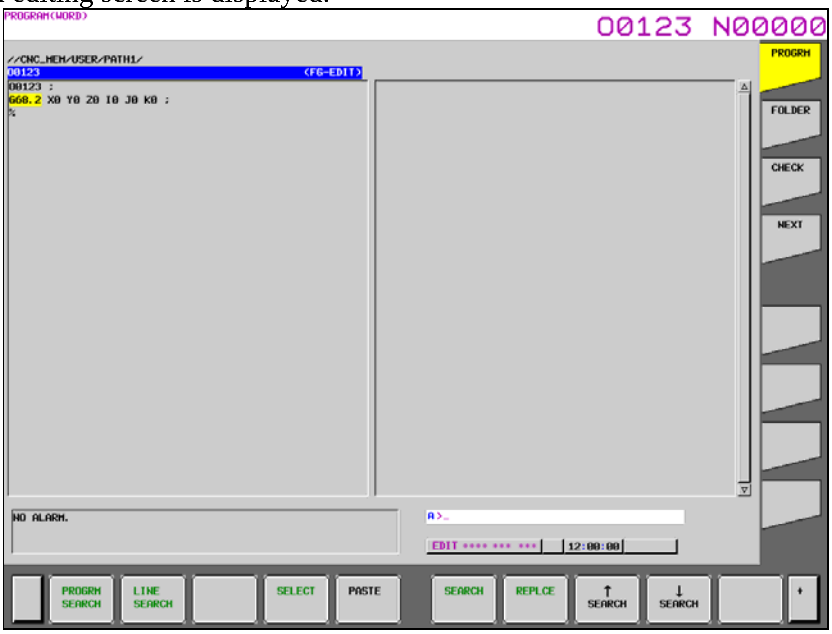


7 Press horizontal soft key [YES].
This takes you back to the program editing screen, where the new block is inserted after the block at the cursor position.

Modification to an existing block

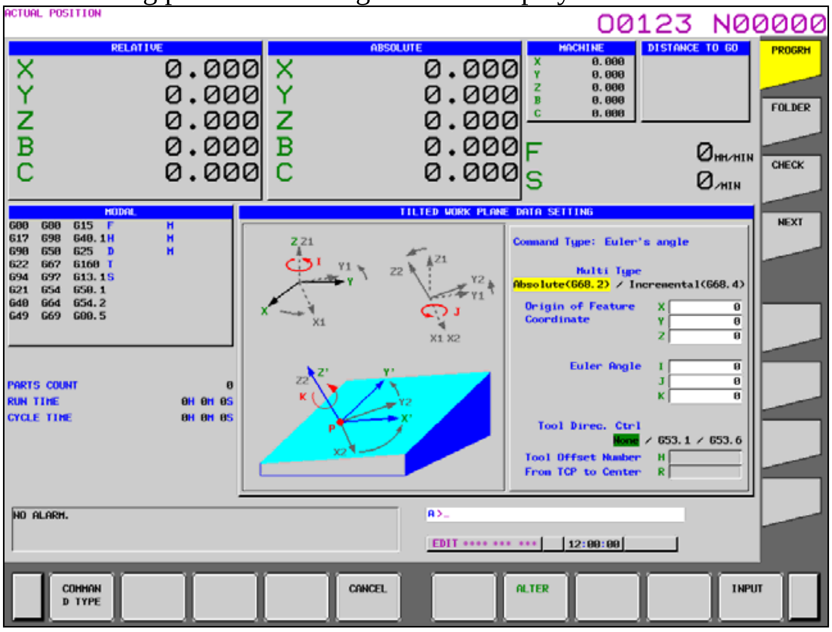
The following describes the procedure for replacing a block in a program being edited on a program editing screen, with a tilted working plane indexing block created on a guidance screen.

1 On a program editing screen, display a program to be edited.
(For the procedure for displaying a program editing screen, see step 1 in "Creation of a new block".)
The program editing screen is displayed.



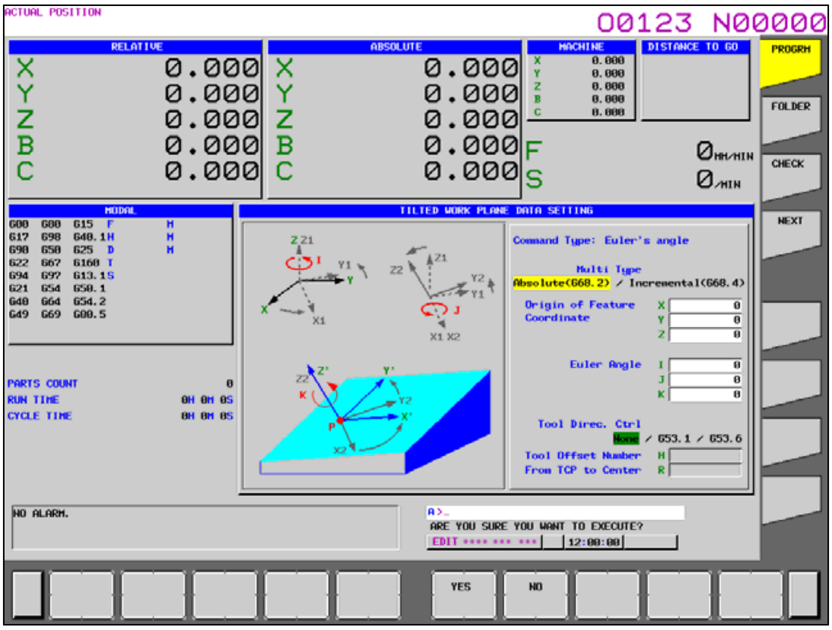
2 Press any of the cursor keys to move the cursor to a block to be modified.
For a command extending over more than one block, move the cursor to the first block.

3 Press continuous menu key several times, and then press horizontal soft key [GUIDANCE TWP]. The tilted working plane data setting screen is displayed.



4 Enter command data for setting items to be modified.

5 Press horizontal soft key [ALTER].



6 Press horizontal soft key [YES].
This takes you back to the program editing screen, where the block at the cursor position is replaced.

Guidance screen cancellation

Pressing horizontal soft key [CANCEL] on a guidance screen takes you back to the program editing screen. At this time, the data that has been set on the guidance screen is discarded.

NOTE

1 In addition to the above operation, the following operations also cancel a guidance screen. The data that has been set on the guidance screen is discarded.
- When bit 7 (CPG) of parameter No. 11302 is 1 (setting for automatically switching between program-related screens according to CNC mode), the CNC mode is changed.
- When a guidance screen is displayed from the foreground editing screen, the CNC mode is changed to a mode other than EDIT, TJOG, or THND.
- When a guidance screen is displayed from the MDI editing screen, the CNC mode is changed to a mode other than MDI.
- When a guidance screen is displayed on a 15-inch display unit, the screen is switched by a vertical soft key.
- The screen is switched by an MDI key.
- The screen is switched by a path select signal.
- An event that causes screen switching has occurred, including occurrence of an alarm, display of an operator message, or signal-based display of a screen (such as a tool compensation screen, workpiece shift screen, workpiece coordinate system setting screen, or C Language Executor screen).

2 If MDI key  is pressed after switching from a guidance screen to another screen, a program editing screen is displayed, instead of the guidance screen.

Notes

- Conditions under which horizontal soft key [GUIDANCE TWP] is displayed
  Horizontal soft key [GUIDANCE TWP] is displayed on a program editing screen under the following conditions:
  1 Foreground editing screen
     - The CNC mode is EDIT, TJOG, or THND.
     - The editing and display are not prohibited for a program to be edited.
  2 Background editing screen
     - A program to be edited is not in reference mode.
     - The editing and display are not prohibited for a program to be edited.
  3 MDI editing screen
     - The CNC mode is MDI.

- Screen displayed when horizontal soft key [GUIDANCE TWP] is pressed
  When horizontal soft key [GUIDANCE TWP] is pressed, which guidance screen is displayed is automatically determined according to the conditions described below.
  1 When a tilted working plane indexing is not included in a block at the cursor position on a program editing screen
     The command type selection screen is displayed. A block created on the guidance screens is inserted after the block at the cursor position on the program editing screen.
  2 When a tilted working plane indexing is included in a block at the cursor position on a program editing screen
     The tilted working plane data setting screen is displayed, which shows the data for the tilted working plane indexing in the block at the cursor position on the program editing screen. When a block of a tool axis direction control command exists immediately after the tilted working plane indexing, the command data for the block is also shown. The tilted working plane indexing in the block at the cursor position on the program editing screen is replaced with a block created on the guidance screen.

> **NOTE**
> If the CNC is in the reset state or emergency stop state when horizontal soft key [GUIDANCE TWP] is pressed on the foreground editing screen or MDI editing screen, the warning "PROGRAM READ FAILED" appears, and the operation cannot be continued. (Horizontal soft keys other than [CANCEL] are not displayed.) Press horizontal soft key [CANCEL] to return to the program editing screen, and then press horizontal soft key [GUIDANCE TWP] again.

3.2.2.1 Command type selection screen

The command type selection screen is used to select the type of a tilted working plane indexing you want to insert into a program to be edited. One of the following command types can be selected:

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

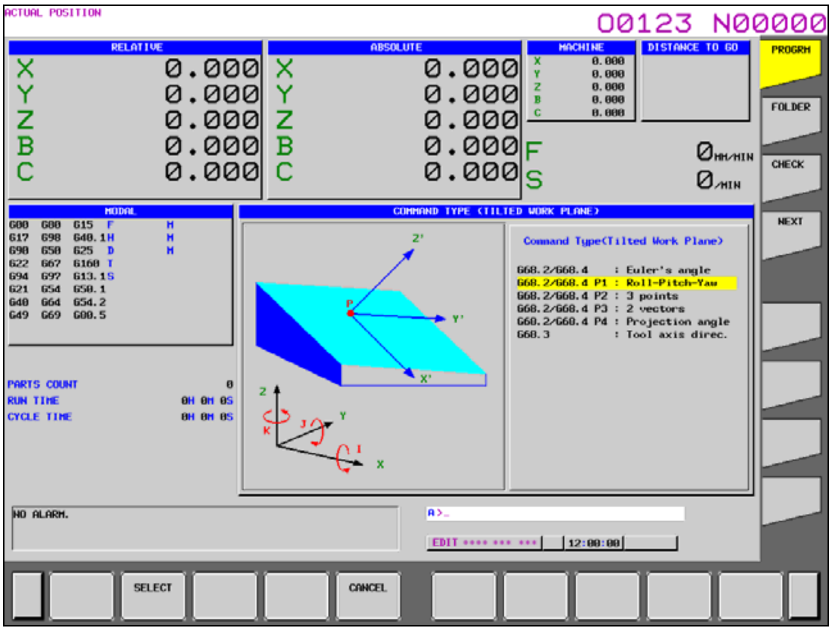


Fig. 3.2.2.1 (a) Command type selection screen (15-inch display unit)

Selection of a command type

1 Press cursor key [⬆] or [⬇] to move the cursor to a command type you want to select. As the cursor is moved, the figure corresponding to the command type at the cursor position is displayed.

2 When horizontal soft key [SELECT] is pressed, the command type at the cursor position is accepted, and the tilted working plane data setting screen is displayed.

NOTE

If the warning "PROGRAM READ FAILED" appears when the command type selection screen is displayed, the operation cannot be continued. (Horizontal soft keys other than [CANCEL] are not displayed.) Press horizontal soft key [CANCEL] to return to the program editing screen, and then press horizontal soft key [GUIDANCE TWP] again.

3.2.2.2 Tilted working plane data setting screen

The tilted working plane data setting screen is used to set specified tilted working plane data required for a tilted working plane indexing of the type that has been selected on the command type selection screen or has been selected when horizontal soft key [GUIDANCE TWP] is pressed.
Different types of tilted working plane data setting screen are provided for different command types. For details of each type of tilted working plane data setting screen, see "Details of the tilted working plane data setting screen".

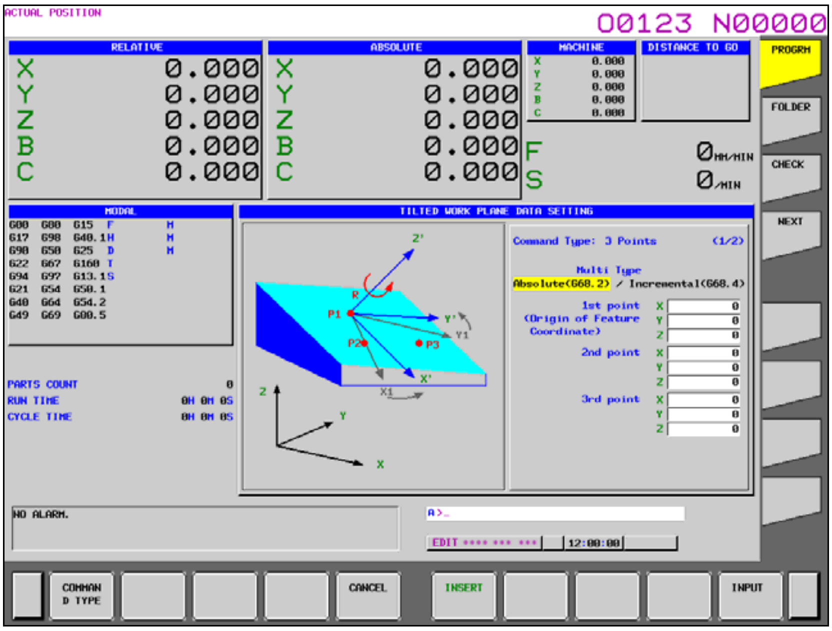


Fig. 3.2.2.2 (a) Tilted working plane data setting screen-3 points specification(15-inch display unit)

Display of the tilted working plane data setting screen

When a new block is created, the initial values are set in the setting and selection items.
When an existing block is modified, the command data for the block at the cursor position on the program editing screen is reflected in the setting and selection items.

NOTE

1 For existing block modification, if the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the parameters for the block(s) before the cursor are not reflected in the setting and selection items, but only the parameters for the block(s) after the cursor are reflected. If no data is to be reflected in a setting or selection item, the initial value is set in the item.

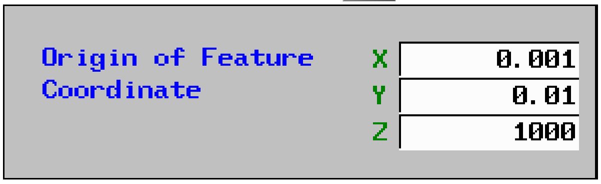
2 For existing block modification, only the commands with parameter-specified axis names are reflected in the setting items on the screen. (The values specified in data for commands with incorrect axis names are not reflected in the setting items.)

Command data input

- Item for which to enter a value

  Press cursor key ↑ or ↓ to move the cursor to an item you want to set.

  Enter a value, and then press the INPUT key or horizontal soft key [INPUT].



  Example)

  When the origin of a feature coordinate system is set as shown above, addresses X, Y, and Z are specified as follows:

  G68.2 X0.001 Y0.01 Z1000 ...

  For two items "Tool Offset Number" and "From TCP to center", the state with no command can be set by deleting the specified values according to the following procedure.

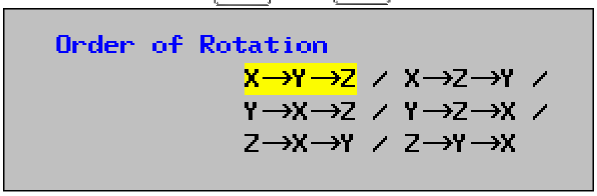
  1 Press cursor key ↑ or ↓ to move the cursor to the item "Tool Offset Number" or "From TCP to Center".

  2 Press INPUT or horizontal soft key [INPUT] without entering anything.

- Item to be selected from a list

  1 Press cursor key ↑ or ↓ to move the cursor to an item you want to set.

  2 Press cursor key ← or → to move the cursor to an item you want to select.



  Example)

  Order of rotation for Roll-Pitch Yaw angle

Insertion of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen does not include a tilted working plane indexing, horizontal soft key [INSERT] is displayed on the tilted working plane data setting screen. An operation for inserting a tilted working plane indexing block is described below.

For warnings that may be issued at the time of block insertion, see "Limitation".

1 Press horizontal soft key [INSERT].

The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and horizontal soft keys [YES] and [NO] are displayed.

2 Press horizontal soft key [YES].

A block is created from the command type and command data and then is inserted after the block at the cursor position in the program to be edited. After block insertion, the guidance screen is closed and the program editing screen is displayed.

Pressing horizontal soft key [NO] takes you back to the tilted working plane data setting screen.

Example)
G00 X0.;
↓
G00 X0.;
G68.2 P2 Q0...
G68.2 P2 Q1...
G68.2 P2 Q2...
G68.2 P2 Q3...

When the guidance screen is displayed and the 3-point specification is selected as a command type for block insertion, a created block is inserted after the block at the cursor position.

Replacement of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen includes a tilted working plane indexing, horizontal soft key [ALTER] is displayed on the tilted working plane data setting screen. An operation for replacing a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block replacement, see "Limitation".

1 Press horizontal soft key [ALTER].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and horizontal soft keys [YES] and [NO] are displayed.
2 Press horizontal soft key [YES].
A block is created from the command type and command data and then replaces the tilted working plane indexing in the block at the cursor position on the program editing screen. After block replacement, the guidance screen is closed and the program editing screen is displayed.
Pressing horizontal soft key [NO] takes you back to the tilted working plane data setting screen.

NOTE
1 If a block to be replaced includes a command other than a tilted working plane indexing, the command is deleted during block replacement. However, only the sequence number at the beginning is preserved.
2 If the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the block(s) before the cursor are not replaced. These blocks remain unchanged after replacement.
3 If horizontal soft key [GUIDANCE TWP] is pressed when the cursor is placed on a block of a tool axis direction control command, the command type selection screen is displayed in new insertion mode. A block created on the guidance screens is inserted after the block of the tool axis direction control command.

Limitation

The following lists the warnings that may be issued at the time of block insertion or replacement.
If a warning is displayed, return to the program editing screen with horizontal soft key [CANCEL] and press horizontal soft key [GUIDANCE TWP] again, or eliminate the cause of the warning and retry the operation.

Warning	Description
“PROGRAM WRITE FAILED”	• The guidance screen was displayed from the foreground editing screen or MDI editing screen, and a block insertion or replacement operation was performed when the CNC was in the reset state or emergency stop state.
“PROGRAM CANNOT BE WRITTEN”	• A block insertion or replacement operation was performed after a program to be edited was updated due to program downloading by an external application while the guidance screen was displayed. • A block insertion or replacement operation was performed after the start of the main program was located by a reset while the guidance screen was displayed.

Warning	Description
"WRITE PROTECT"	• A block insertion or replacement operation was performed when the editing or display was prohibited for a program to be edited. • A block insertion or replacement operation was performed when the memory protection signal (KEY signal) for program registration or editing was off. • A block insertion or replacement operation was performed when the operation level of the 8-level data protection function was lower than the protection level of a part program editing operation.

Display of the command type selection screen

When horizontal soft key [COMMAND TYPE] is pressed, the command type selection screen is displayed. When the command type is changed on the command type selection screen, the values set on the tilted working plane data setting screen are cleared.

When a tilted working plane indexing is included in the block at the cursor position on the program editing screen, you can display the guidance screen, change the command type on the command type selection screen, and then perform block replacement. In this case, the block at the cursor position on the program editing screen is replaced with a block created from the command type and command data that have been set.

3.2.2.3 Details of the tilted working plane data setting screen

The following six tilted working plane indexing are supported.
For details of the commands, see II-22.3, "TILTED WORKING PLANE INDEXING".

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

G68.2 / G68.4(Euler's angle)

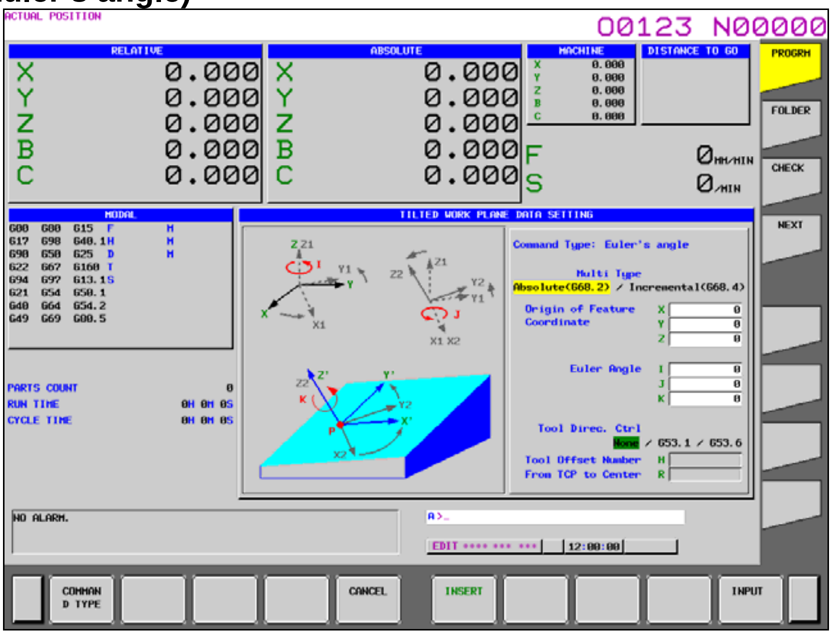


Fig. 3.2.2.3 (a) Tilted working plane data setting screen-Euler's angle(15-inch display unit)

- Multi Type

  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate

  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Euler's angle

  I : Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  This rotation determines coordinate system 1 (X1-Y1-Z1) from the workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type) (X-Y-Z).

  J : Specify an angle of rotation around the X-axis of coordinate system 1.
  This rotation determines coordinate system 2 (X2-Y2-Z2) from coordinate system 1 (X1-Y1-Z1).

  K : Specify an angle of rotation around the Z-axis of coordinate system 2.
  After this rotation, a feature coordinate system is obtained by shifting the origin of the workpiece coordinate system by the coordinates specified in "Origin of Feature Coordinate".

G68.2 / G68.4(Roll-Pitch-Yaw angle)

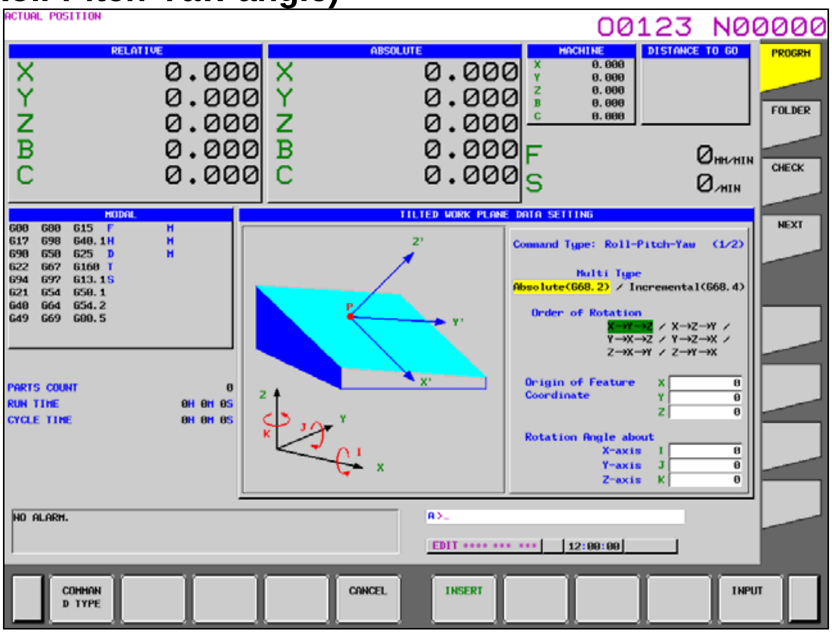


Fig. 3.2.2.3 (b) Tilted working plane data setting screen- Roll-Pitch-Yaw angle(15-inch display unit)

- Multi Type

  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Order of Rotation
  Select an order in which the X-axis, Y-axis, and Z-axis are rotated in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The selectable rotation orders are as follows:

	1st rotation axis	2nd rotation axis	3rd rotation axis
X→Y→Z	X-axis	Y-axis	Z-axis
X→Z→Y	X-axis	Z-axis	Y-axis
Y→X→Z	Y-axis	X-axis	Z-axis
Y→Z→X	Y-axis	Z-axis	X-axis
Z→X→Y	Z-axis	X-axis	Y-axis
Z→Y→X	Z-axis	Y-axis	X-axis

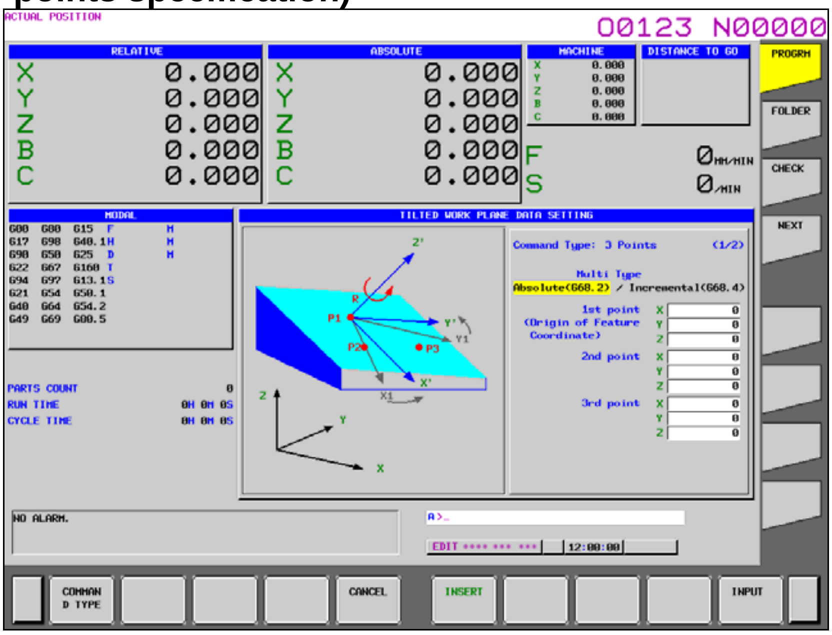
- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about X-axis
  Specify an angle of rotation around the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Y-axis
  Specify an angle of rotation around the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Z-axis
  Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(3 points specification)



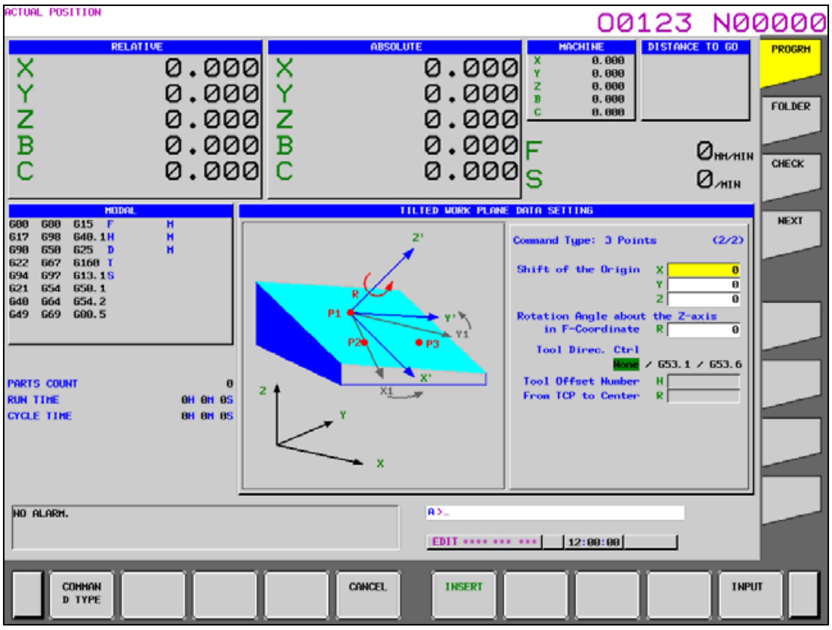


Fig. 3.2.2.3 (c) Tilted working plane data setting screen- 3 points specification(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Shift of the Origin
  Specify, in a feature coordinate system, an amount of shift from the feature coordinate system origin specified for the 1st point (point P1).

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

- 1st point (Origin of Feature Coordinate)
  Specify the origin (X, Y, and Z of point P1) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- 2nd point
  Specify the coordinates (X, Y, and Z of point P2) of the 2nd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The 1st point and 2nd point determine the X-axis of the feature coordinate system.

- 3rd point
  Specify the coordinates (X, Y, and Z of point P3) of the 3rd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). Among the directions normal to the X-axis, a direction with a smaller angle relative to the P1 → P3 vector is the Y-axis of the feature coordinate system.

G68.2 / G68.4(2 vectors specification)

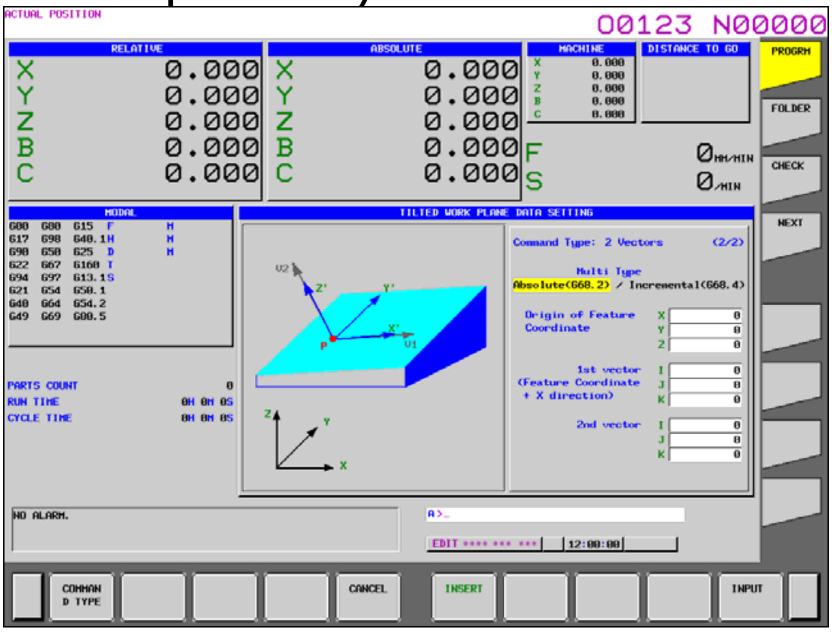


Fig. 3.2.2.3 (d) Tilted working plane data setting screen-2 vectors specification(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V1 (X') vector
  Specify the X-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V2 (Z') vector
  Specify the Z-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(Projection angle)

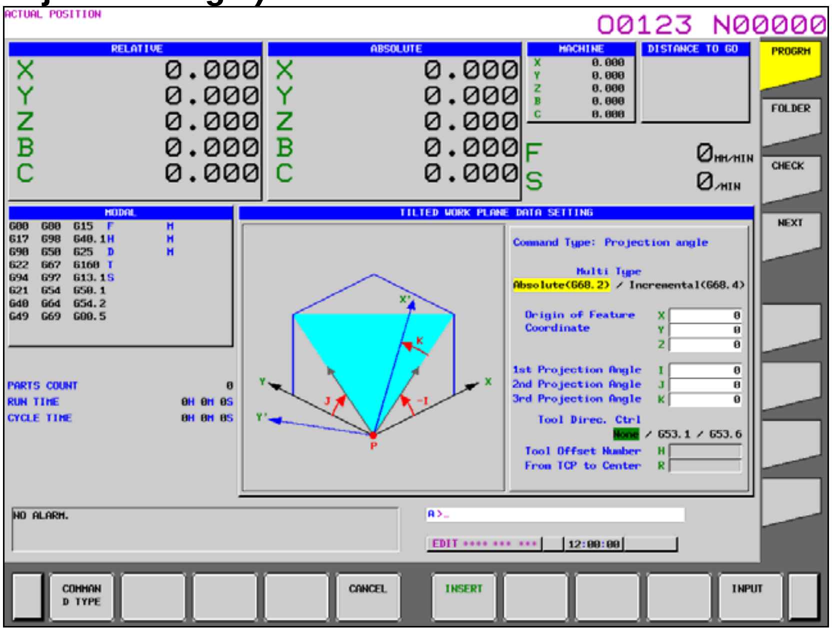


Fig. 3.2.2.3 (e) Tilted working plane data setting screen-Projection angle(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Projection angle
  I : Specify a projection angle relative to the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  J : Specify a projection angle relative to the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  K : Specify an angle of rotation around the Z-axis of a feature coordinate system.

G68.3(Tool Axis Direction)

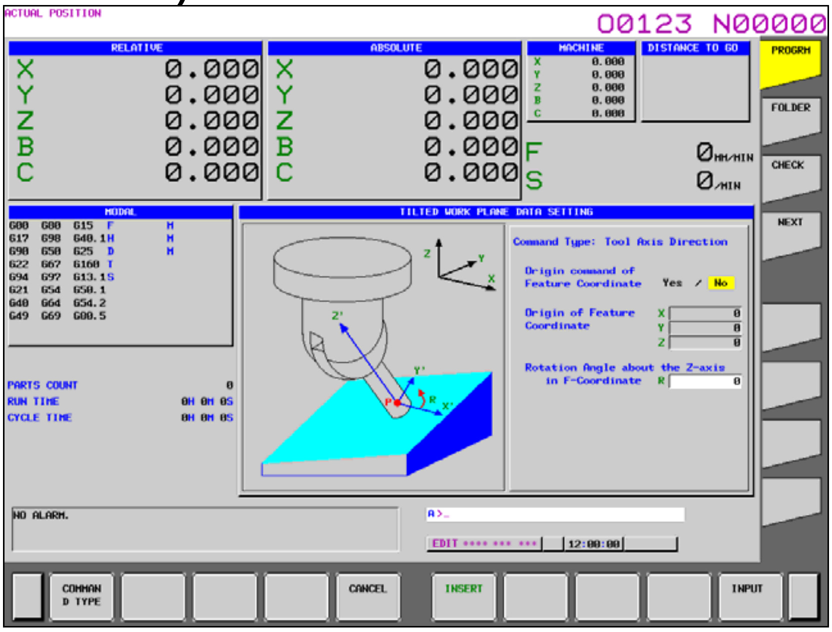


Fig. 3.2.2.3 (f) Tilted working plane data setting screen-Tool Axis Direction(15-inch display unit)
(When "No" is selected in "Origin command of Feature Coordinate")

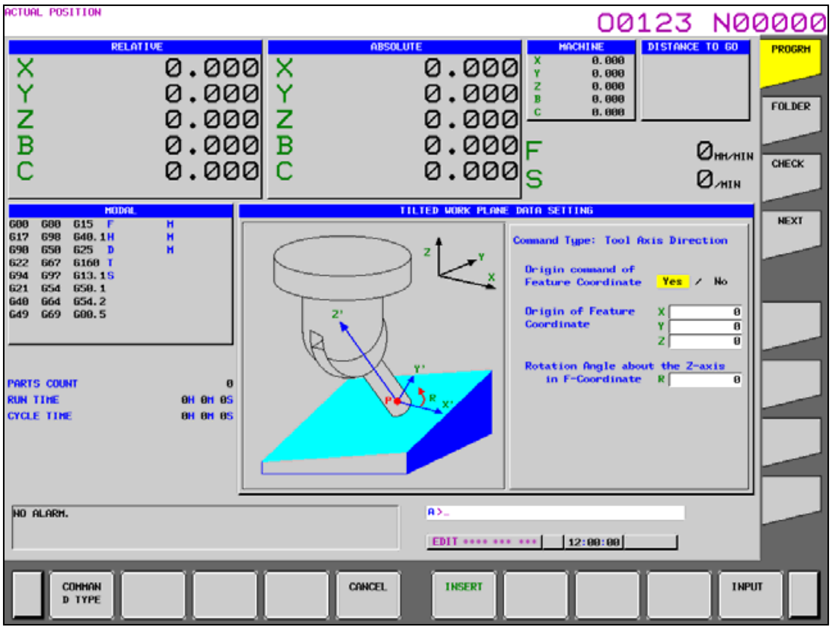


Fig. 3.2.2.3 (g) Tilted working plane data setting screen-Tool Axis Direction(15-inch display unit)
(When "Yes" is selected in "Origin command of Feature Coordinate")

- Origin command of Feature Coordinate
  Select whether to specify a feature coordinate system origin.
  Yes : A feature coordinate system origin is specified.
  No : A feature coordinate system origin is not specified.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system.
  This setting cannot be made if "No" is selected in "Origin command of Feature Coordinate".

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

3.2.2.4 Limitation

This function is supported on 10.4-inch and 15-inch display units. This function is not displayed on 8.4-inch display unit.

3.3 SCREENS DISPLAYED BY FUNCTION KEY

Press function key to display or set the following data:

1. Tool compensation value
2. Tool length measurement
3. Machining Level Selection
4. Machining Quality Level Selection

Refer to the Operator's Manual (Common to Lathe System/Machining Center System) (B-64604EN) for explanations about how to display or specify the other types of data.

3.3.1 Setting and Displaying the Tool Compensation Value

Tool offset values, tool length compensation values, and cutter compensation values are specified by D codes or H codes in a program. Compensation values corresponding to D codes or H codes are displayed or set on the screen.

Procedure for setting and displaying the tool compensation value (for 8.4/10.4-inch display unit)

Procedure

1. Press function key .

   For the two-path control, select the path for which tool compensation values are to be displayed with the tool post selection switch.

2. Press chapter selection soft key [OFFSET] or press function key several times until the tool compensation screen is displayed.

   The screen varies according to the type of tool compensation memory.

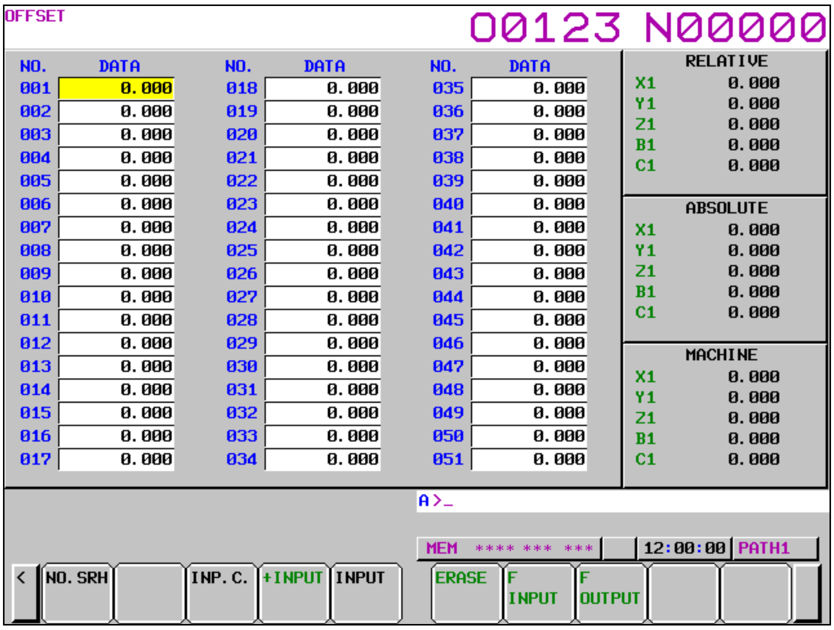


Fig. 3.3.1 (a) Tool compensation memory A (10.4-inch display unit)

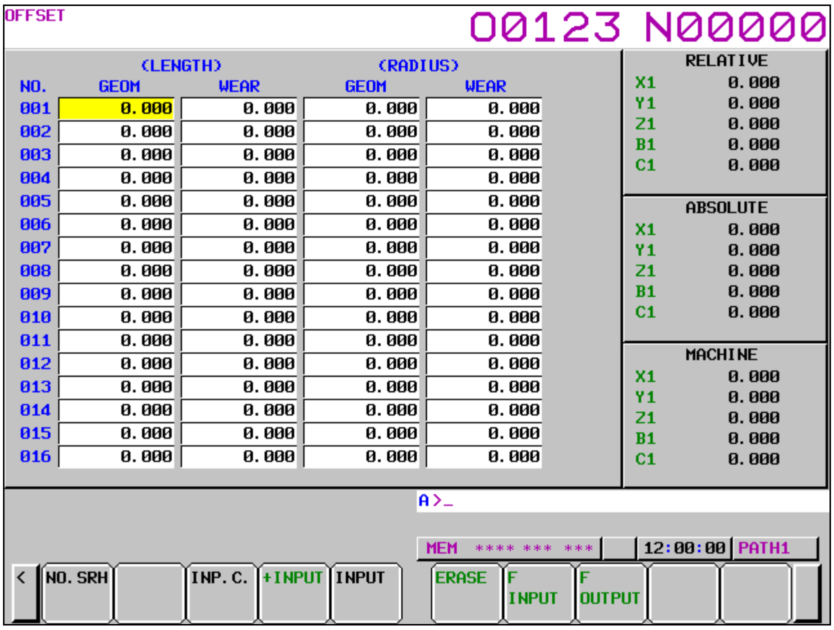
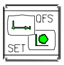
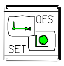


Fig. 3.3.1 (b) Tool compensation memory C (10.4-inch display unit)

3. Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].
4. To set a compensation value, enter a value and press soft key [INPUT].
   To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [+INPUT]. Or, enter a new value and press soft key [INPUT].

Procedure for setting and displaying the tool compensation value (for 15-inch display unit)

Procedure

1. Press function key  .
   For the two-path control, select the path for which tool compensation values are to be displayed with the tool post selection switch.
2. Press the chapter selection key, in Fig.3.3.1 (c), vertical soft key [OFFSET].
   Or, press function key  several times until the tool compensation screen is displayed.
   The screen varies according to the type of tool compensation memory.

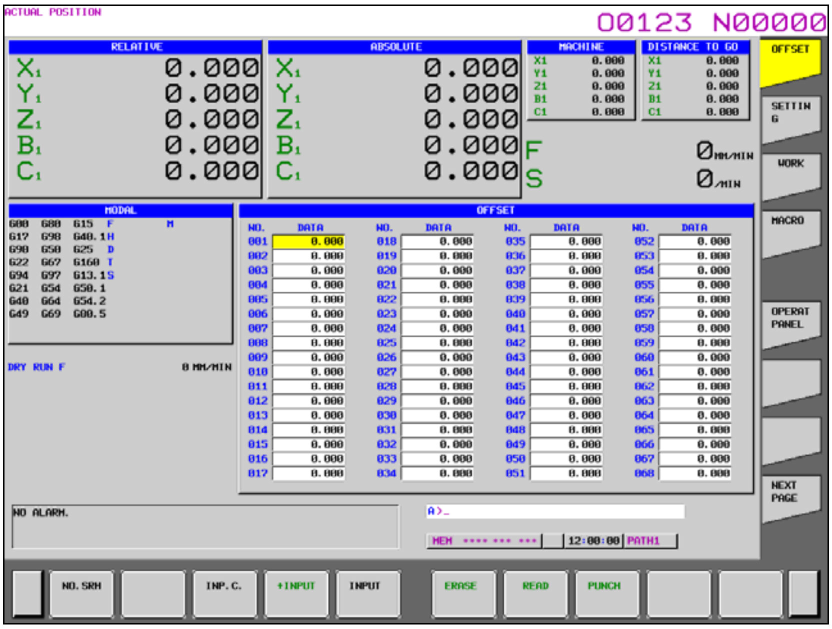


Fig.3.3.1 (c) Tool compensation memory A (15-inch display unit)

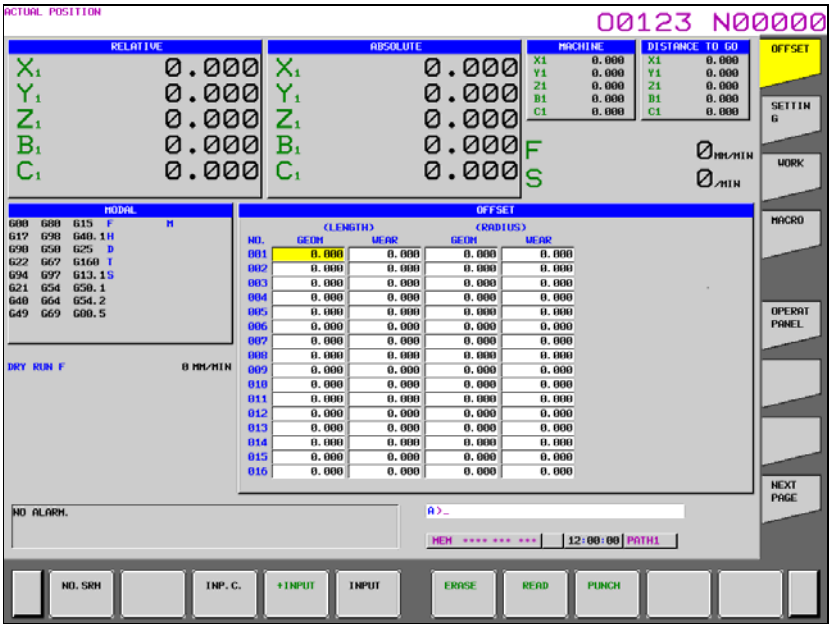


Fig. 3.3.1 (d) Tool compensation memory C (15-inch display unit)

3. Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press horizontal soft key [NO.SRH].
4. To set a compensation value, enter a value and press horizontal soft key [INPUT].
   To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press horizontal soft key [+INPUT]. Or, enter a new value and press horizontal soft key [INPUT].

Explanation

- Decimal point input

A decimal point can be used when entering a compensation value.

- Other setting method

An external input/output device can be used to input or output a tool offset value. See Chapter, “DATA INPUT/OUTPUT” in OPERATOR’S MANUAL (Common to 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.

- Disabling entry of compensation values

The entry of compensation values may be disabled by setting bit 0 (WOF) and bit 1 (GOF) of parameter No.3290 (not applied to tool compensation memory A).

And then, the input of tool compensation values from the MDI can be inhibited for a specified range of offset numbers. The first offset number for which the input of a value is inhibited is set in parameter No. 3294. The number of offset numbers, starting from the specified first number, for which the input of a value is inhibited is set in parameter No. 3295.

Consecutive input values are set as follows:

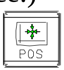
1) When compensation values are input consecutively from offset numbers for which the input of values is enabled to offset numbers for which the input of values is inhibited, a warning is issued, but the compensation values in the range of the offset numbers for which the input of values is enabled are set.
2) When compensation values are input consecutively from offset numbers for which the input of values is inhibited to offset numbers for which the input of values is enabled, a warning is issued and the compensation values are not set.

3.3.2 Tool Length Measurement

The length of the tool can be measured and registered as the tool length compensation value by moving the reference tool and the tool to be measured until they touch the specified position on the machine.

The tool length can be measured along the X-, Y-, or Z-axis.

Procedure for tool length measurement (for 8.4/10.4-inch display unit)

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.

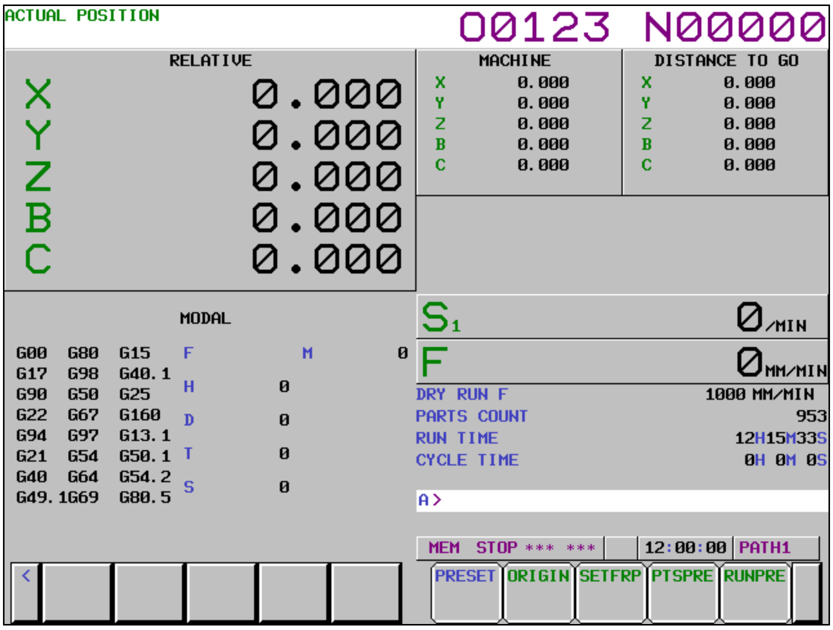
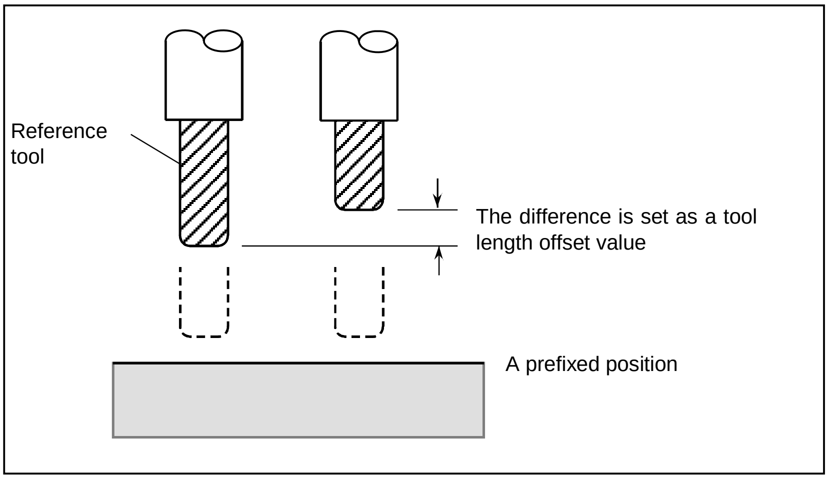


Fig. 3.3.2 (a) Current position display screen (10.4-inch display unit)

3. Reset the relative coordinate for the Z-axis to 0.
4. Press function key OFS SET several times until the tool compensation screen is displayed.
5. Use manual operation to move the tool to be measured until it touches the same specified position. The difference between the length of the reference tool and the tool to be measured is displayed in the relative coordinates on the screen.
6. Move the cursor to the compensation number for the target tool (the cursor can be moved in the same way as for setting tool compensation values).
7. Press the address key Z. If either X or Y key is depressed instead of Z key, the X or Y axis relative coordinate value is input as an tool length compensation value.
8. Press the soft key [INP.C.]. The Z axis relative coordinate value is input and displayed as an tool length compensation value.



Procedure for tool length measurement (for 15-inch display unit)

1. Use manual operation to move the reference tool until it touches the specified position on the machine (or workpiece.)
2. Press function key POS to display the overall position display screen.

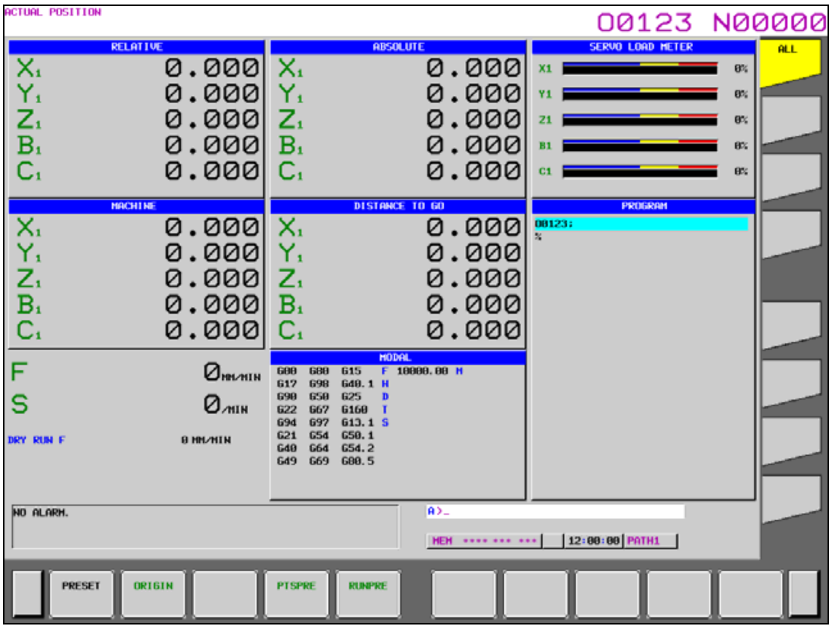
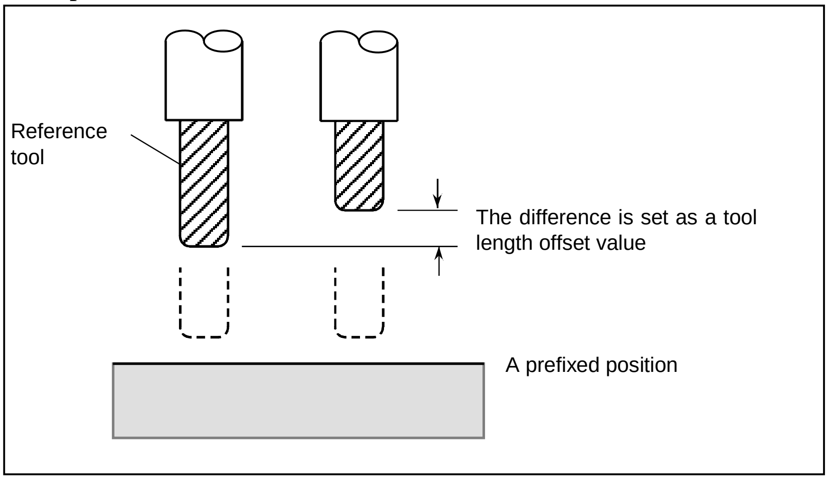


Fig. 3.3.2 (b) Current position display screen (15-inch display unit)

3. Reset the relative coordinate for the Z-axis to 0.
4. Press function key OFS SET several times until the tool compensation screen is displayed.
5. Use manual operation to move the tool to be measured until it touches the same specified position. The difference between the length of the reference tool and the tool to be measured is displayed in the relative coordinates on the screen.
6. Move the cursor to the compensation number for the target tool (the cursor can be moved in the same way as for setting tool compensation values).
7. Press the address key Z. If either X or Y key is depressed instead of Z key, the X or Y axis relative coordinate value is input as an tool length compensation value.
8. Press the horizontal soft key [INP.C.]. The Z axis relative coordinate value is input and displayed as an tool length compensation value.



3.3.3 Machining Level Selection

3.3.3.1 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. 3.3.3.1 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

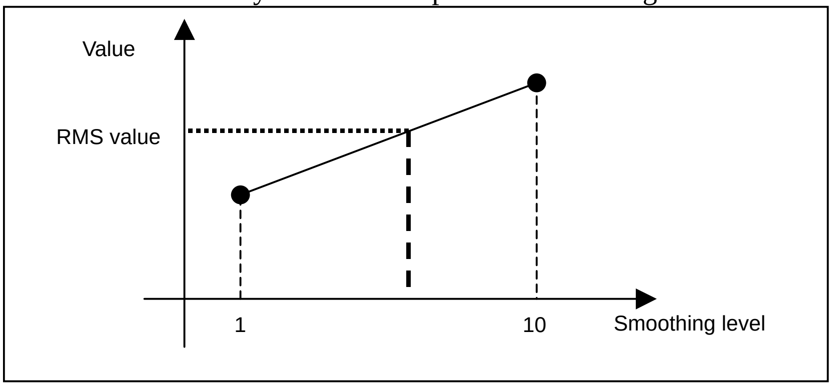
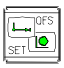


Fig. 3.3.3.1 (a) Image of “level”

> **NOTE**
> 1 To use the "Smoothing level selection screen", all following options are required.
> - AI contour control II
> - Machining condition selecting function
> - Nano smoothing
> - Machining quality level adjustment function
>
> 2 "Smoothing level selection screen" is displayed, when bit 6 (QLS) of parameter No.11350 is 0.
> "Machining quality level selection screen" is displayed, when parameter QLS is 1.

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].

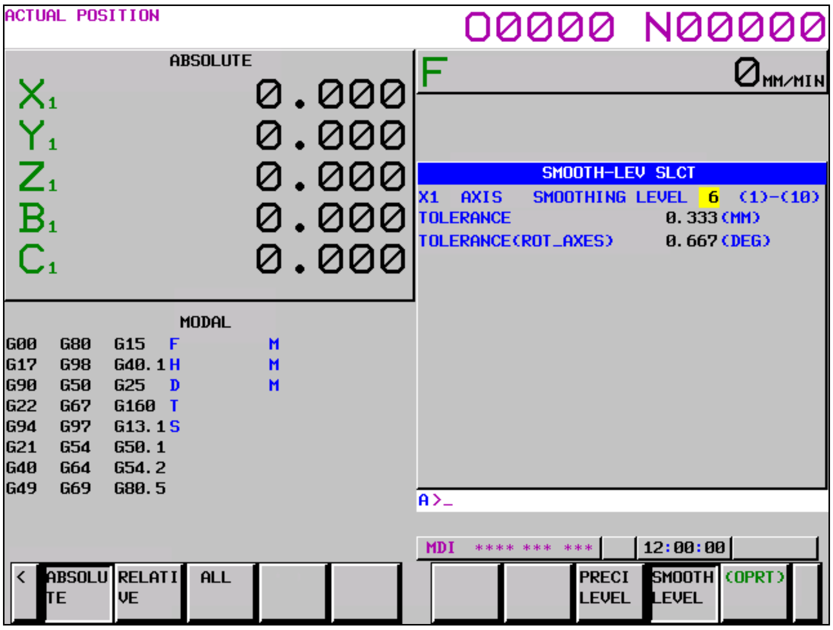


Fig. 3.3.3.1 (b) Smoothing level selection screen (10.4inch)

5. To change the smoothing level, key in a desired smoothing level (1 to 10), then press the INPUT key on the MDI unit.
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 PAGE or PAGE several times to display the screen for the axis.

3.3.3.2 Precision level selection

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen (AI contour) can be selected. As shown in the Fig. 3.3.3.2 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

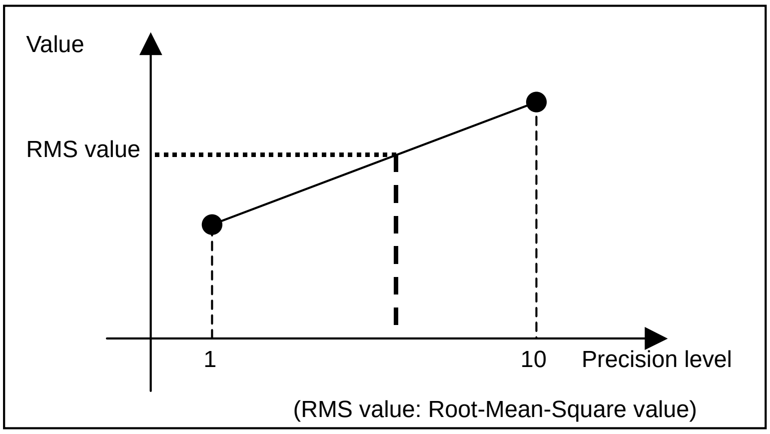
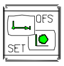


Fig. 3.3.3.2 (a) Image of "level"

NOTE
To use the "Precision level selection screen", the following options are required.
- "AI contour control I" or "AI contour control II"
- Machining condition selecting function

Procedure for precision level selection

1 Select the MDI mode.

2 Press function key .

3 Press soft key [PRECI LEVEL].

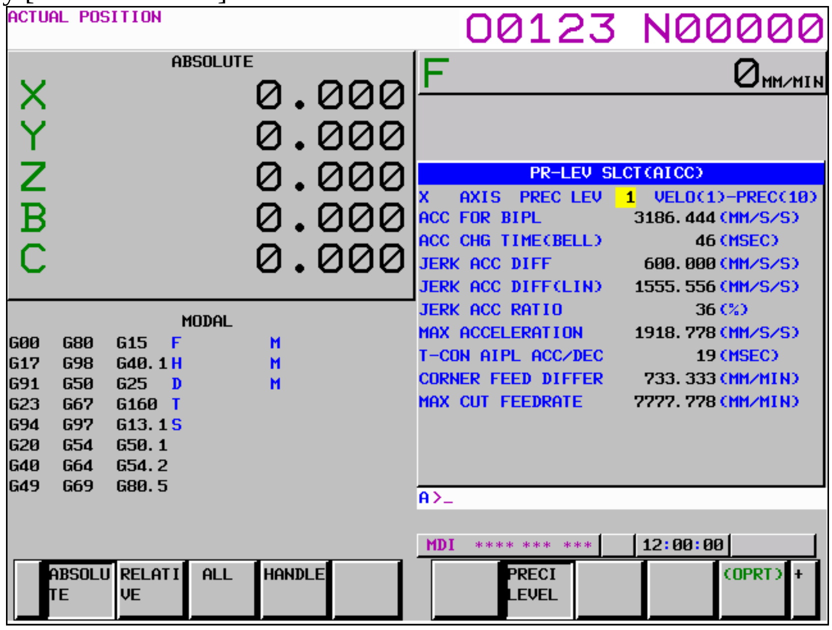
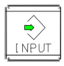


Fig. 3.3.3.2 (b) Precision level selection screen (10.4-inch display unit)

4 To change the precision level, key in a desired precision level (1 to 10), then press the  key on the MDI unit.

5 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

6 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.

3.3.4 Machining Quality Level Selection

Machining quality level selection allows the precision level and smoothing level to be adjusted intuitively and easily.

NOTE
1 The machining quality level selection screen cannot be displayed on the 8.4-inch display unit.
On these display units, "Machining Level Selection screen" ("Smoothing level selection screen" and "Precision level selection screen") can be displayed.

NOTE

2 To use the "Smoothing level selection screen", all following options are required.
   - AI contour control II
   - Machining condition selecting function
   - Nano smoothing
   - Machining quality level adjustment function

3 "Machining quality level selection screen" is displayed, when bit 6 (QLS) of parameter No.11350 is 1.
"Smoothing level selection screen" is displayed, when parameter QLS is 0.

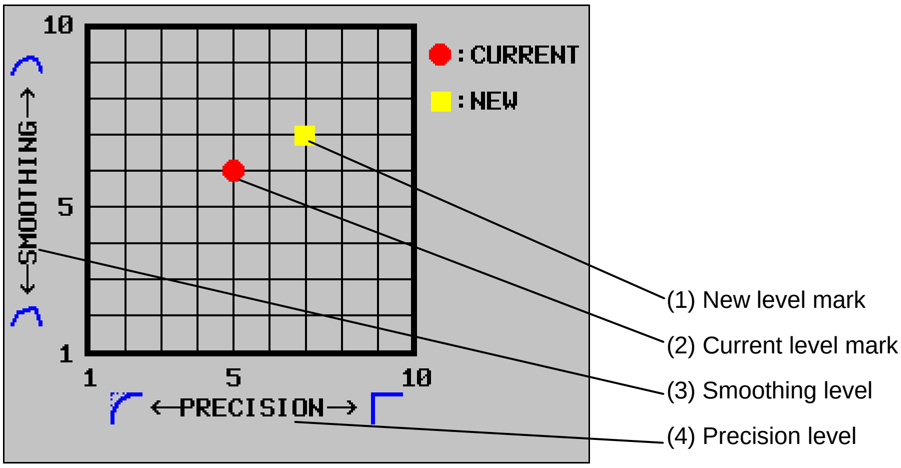


Fig. 3.3.4 (a) Quality level selection

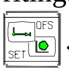
(1) New level mark :
Yellow square: Indicates the setting to be selected. (Cursor position)

(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].

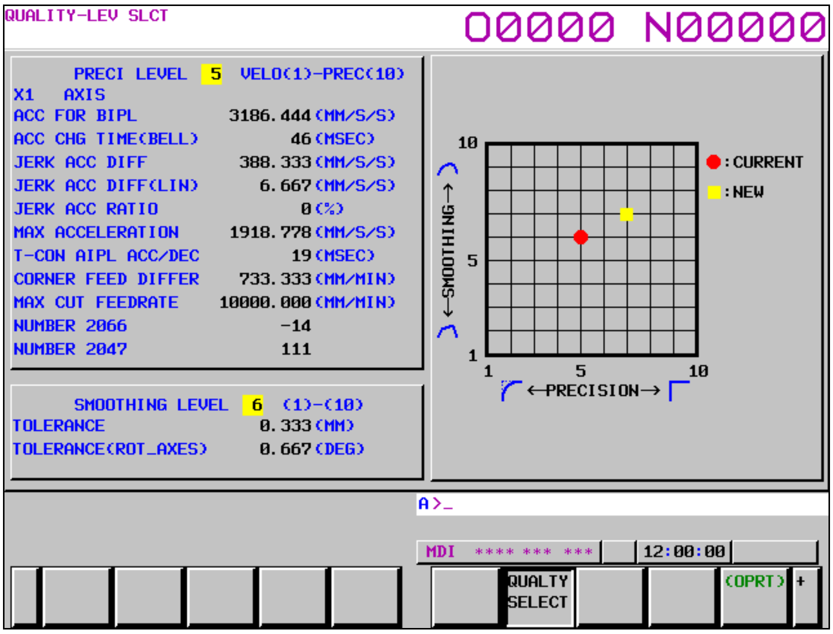


Fig. 3.3.4 (b) Machining quality level selection screen (10.4 inch)

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 INPUT 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 PAGE↑ or PAGE↓ several times to display the screen for the desired axis.

3.3.5 Machining Level Selection (15-inch Display Unit)

3.3.5.1 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. 3.3.5.1 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

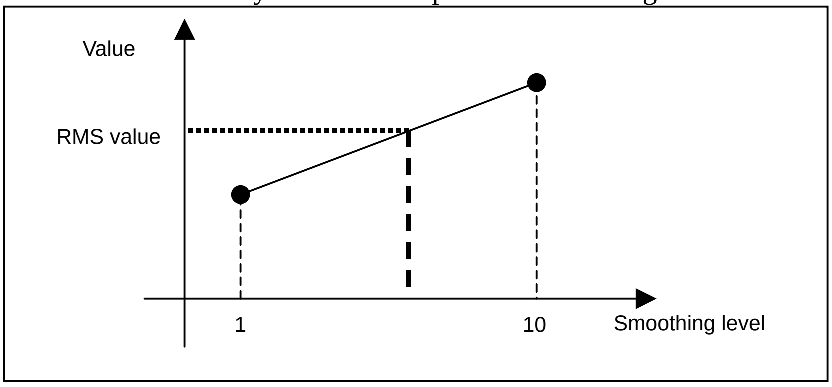
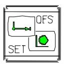


Fig. 3.3.5.1 (a) Image of “level”

> **NOTE**
> 1 To use the "Smoothing level selection screen", all following options are required.
> - AI contour control II
> - Machining condition selecting function
> - Nano smoothing
> - Machining quality level adjustment function
>
> 2 "Smoothing level selection screen" is displayed, when bit 6 (QLS) of parameter No.11350 is 0.
> "Machining quality level selection screen" is displayed, when parameter QLS is 1.

Procedure for smoothing level selection

1. Select the MDI mode.
2. Press function key 
.
3. Press vertical soft key [NEXT PAGE] several times to display vertical soft key [MACHIN LEVEL].
4. Press vertical soft key [MACHIN LEVEL] to display vertical soft key [SMOOTH LEVEL].

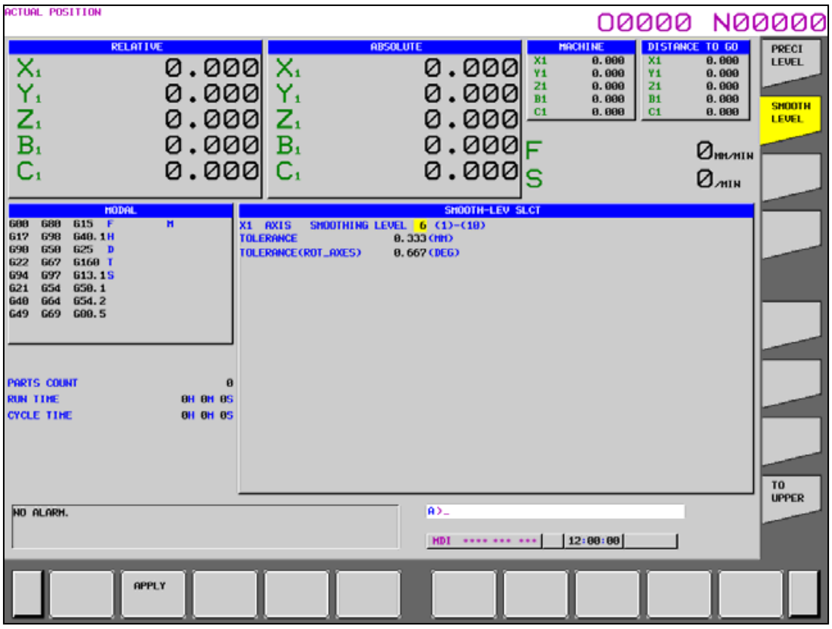
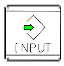


Fig. 3.3.5.1 (b) Smoothing level selection screen

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

3.3.5.2 Precision level selection

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen (AI contour) can be selected. As shown in the Fig. 3.3.5.2 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

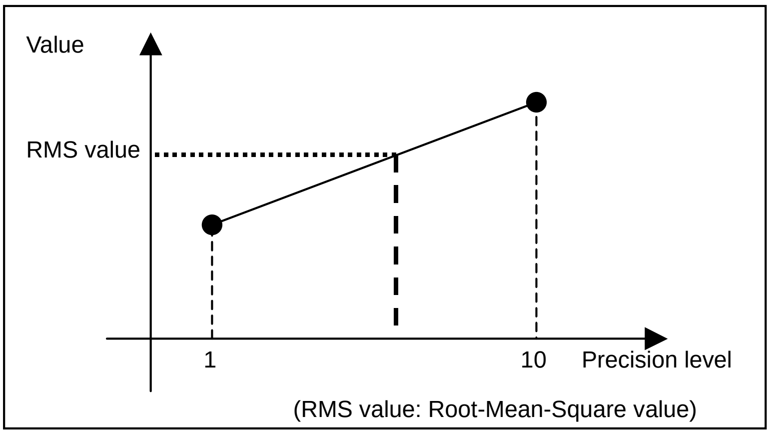


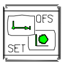
Fig. 3.3.5.2 (a) Image of "level"

NOTE

To use the "Precision level selection screen", the following options are required.
- "AI contour control I" or "AI contour control II"
- Machining condition selecting function

Procedure for precision level selection

1 Select the MDI mode.

2 Press function key 

.

3 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [PRECI LEVEL].

4 Press vertical soft key [PRECI LEVEL].

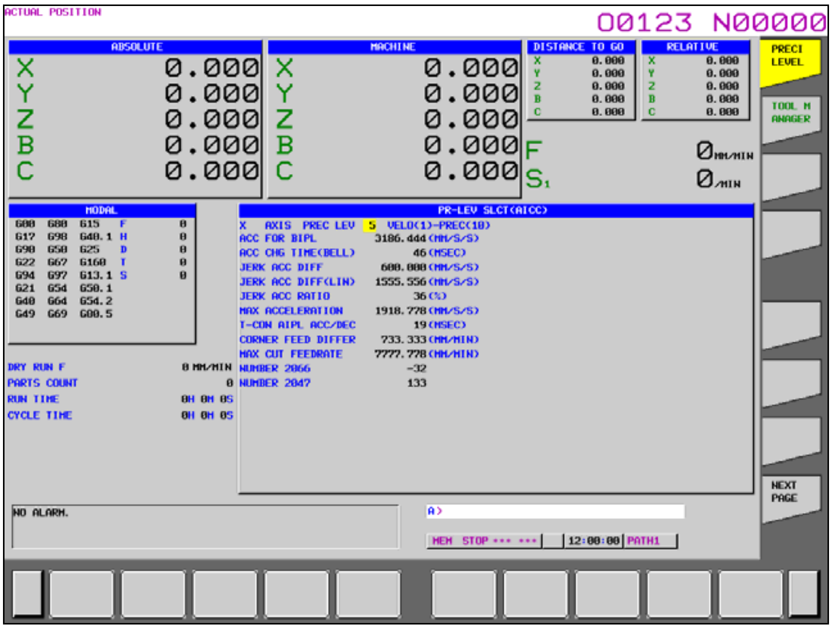
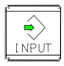


Fig. 3.3.5.2 (b) Precision level selection screen

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

key on the MDI unit.

6 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized 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.

3.3.6 Machining Quality Level Selection (15-inch Display Unit)

Machining quality level selection allows the precision level and smoothing level to be adjusted intuitively and easily.

NOTE

1 To use the "Smoothing level selection screen", all following options are required.
   - AI contour control II
   - Machining condition selecting function
   - Nano smoothing
   - Machining quality level adjustment function

2 "Machining quality level selection screen" is displayed, when bit 6 (QLS) of parameter No.11350 is 1.
"Smoothing level selection screen" is displayed, when parameter QLS is 0.

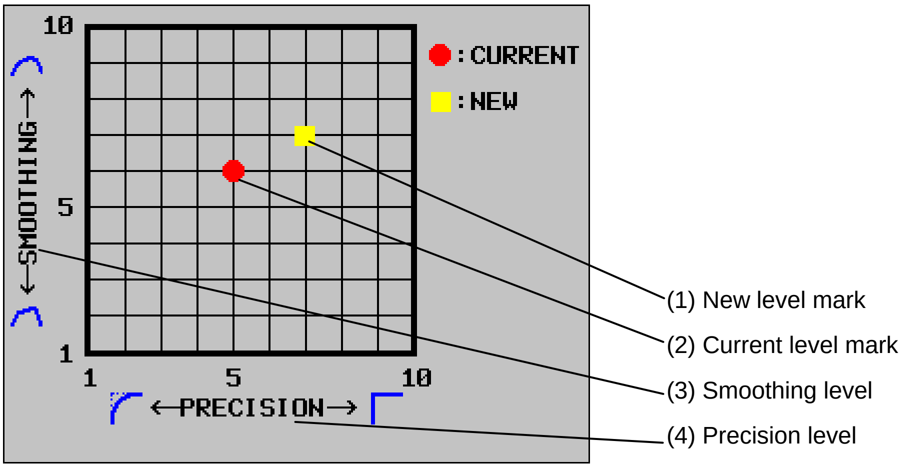


Fig. 3.3.6 (a) Quality level selection

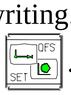
(1) New level mark :
Yellow square: Indicates the setting to be selected. (Cursor position)

(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].

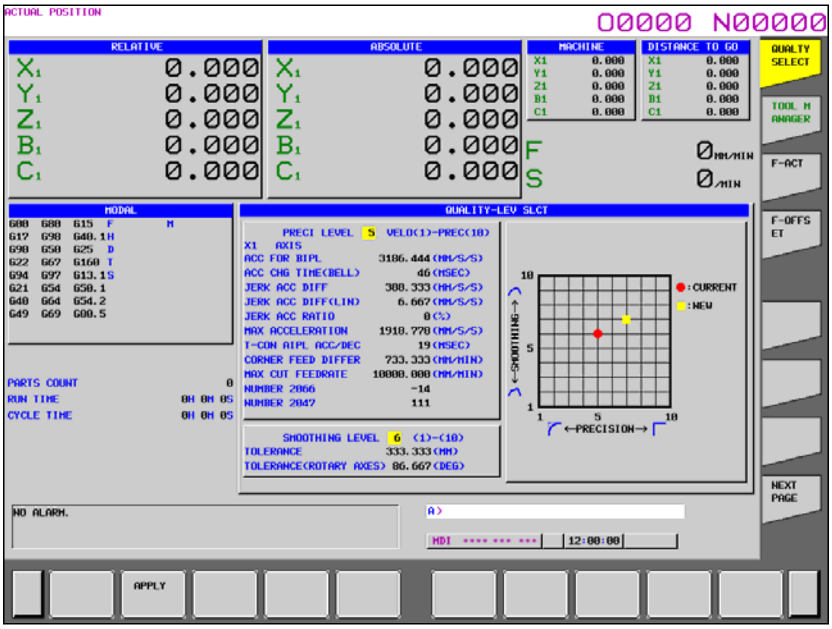
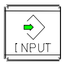


Fig. 3.3.6 (b) Machining quality level selection screen (15-inch display unit)

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.

3.4 SCREENS DISPLAYED BY FUNCTION KEY

Press function key to display or set the following data:

1. Machining Parameter Tuning

Refer to the Operator's Manual (Common to Lathe System/Machining Center System) (B-64604EN) for explanations about how to display or specify the other types of data.

3.4.1 Machining Parameter Tuning

3.4.1.1 Machining parameter tuning (nano smoothing)

In nano smoothing, by setting a parameter set and setting the smoothing level matching a machining condition on the smoothing level selection screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on precision (smoothing level 1) and emphasis on surface smoothing (smoothing level 10) can be set.
Set the following parameters:

- Tolerance

For details of each parameter, see the descriptions of nano smoothing.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a smoothing level, see the description of the smoothing level selection screen in Subsection, "Smoothing level selection".

NOTE

To use the "Smoothing level selection screen", all following options are required.

- AI contour control II
- Machining condition selection function
- Nano smoothing
- Machining quality level adjustment

Procedure for machining parameter tuning

1. Set the MDI mode.
2. Press function key .
3. Press soft key [MCHN TUNING].
4. Press soft key [NANO SMOOTH] to display the machining parameter tuning screen.

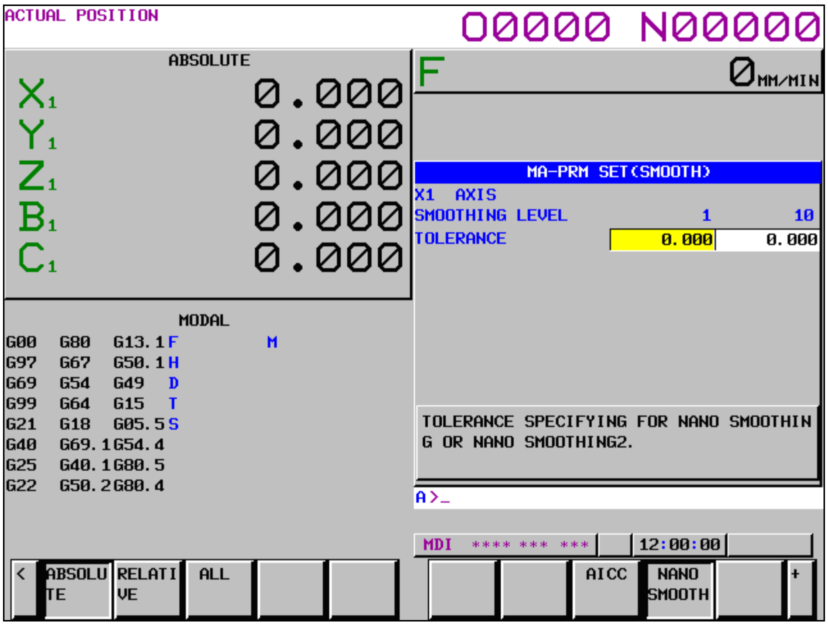


Fig. 3.4.1.1 (a) Machining parameter tuning screen (nano smoothing) (10.4-inch display unit)

5. Move the cursor to the position of a parameter to be set, as follows:
   Press page key or , and cursor keys , , or to move the cursor to the parameter.
6. Key in desired data then press the key on the MDI unit.
7. When data is input, a RMS value is found according to the smoothing level parameters. (The smoothing level can be changed on the smoothing level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.
8. Repeat steps 5 and 6 until all machining parameters are set.

Explanation

- Tolerance

Set the value specified for the tolerance for nano smoothing.
Unit of data: mm, inch, degree (input unit)

The parameter set on the machining parameter tuning screen (smoothing) is reflected in the following parameters:
Parameter No. 11682 : smoothing level 1
Parameter No. 11683 : smoothing level 10

Moreover, the following parameter is also set according to the smoothing level:
Parameter No. 19581 : Tolerance specified for nano smoothing

> ⚠ **CAUTION**
> Since the tolerance specified for nano smoothing is common to all axes, changing this item changes the setting for all axes.

3.4.2 Machining Parameter Tuning (15/19-inch Display Unit)

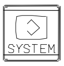
3.4.2.1 Machining parameter tuning (nano smoothing)

In nano smoothing, by setting a parameter set and setting the smoothing level matching a machining condition on the smoothing level selection screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on precision (smoothing level 1) and emphasis on surface smoothing (smoothing level 10) can be set.
Set the following parameters:
- Tolerance

For details of each parameter, see the descriptions of nano smoothing.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a smoothing level, see the description of the smoothing level selection screen in Subsection, "Smoothing level selection".

> **NOTE**
> To use the "Smoothing level selection screen", all following options are required.
> - AI contour control II
> - Machining condition selection function
> - Nano smoothing
> - Machining quality level adjustment

Procedure for machining parameter tuning

1. Set the MDI mode.
2. Press function key 

.
3. Press vertical soft key [MCHN TUNING].
4. Press vertical soft key [NANO SMOOTH] to display the machining parameter tuning screen.

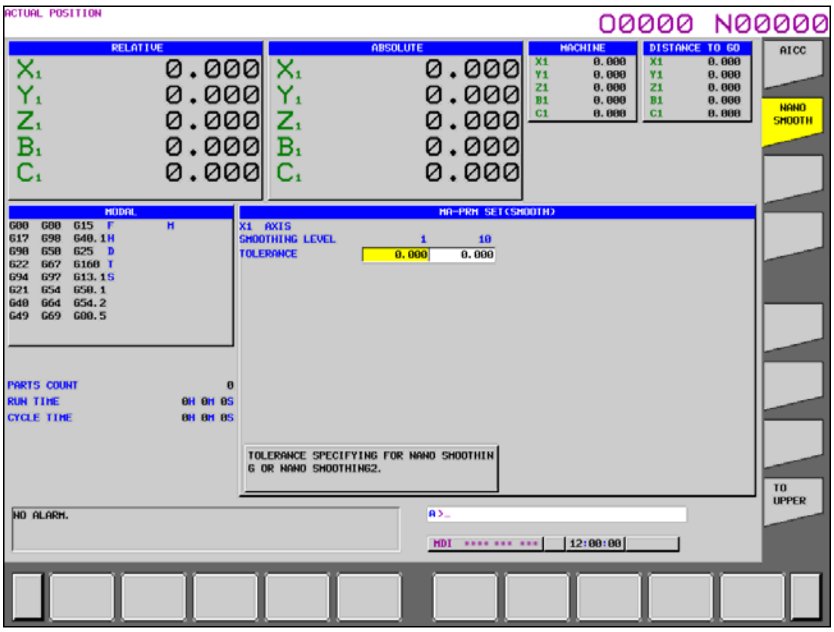


Fig. 3.4.2.1 (a) Machining parameter tuning screen (nano smoothing) (15-inch display unit)

5. Move the cursor to the position of a parameter to be set, as follows:
   Press page key [PAGE ↑] or [PAGE ↓], and cursor keys [↑], [↓], [←] or [→] to move the cursor to the parameter.
6. Key in desired data then press the [INPUT] key on the MDI unit.
7. When data is input, a RMS value is found according to the smoothing level parameters. (The smoothing level can be changed on the smoothing level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.
8. Repeat steps 5 and 6 until all machining parameters are set.

Explanation

- Tolerance

Set the value specified for the tolerance for nano smoothing.
Unit of data: mm, inch, degree (input unit)

The parameter set on the machining parameter tuning screen (smoothing) is reflected in the following parameters:
Parameter No. 11682 : smoothing level 1
Parameter No. 11683 : smoothing level 10

Moreover, the following parameter is also set according to the smoothing level:
Parameter No. 19581 : Tolerance specified for nano smoothing

> ⚠ **CAUTION**
> Since the tolerance specified for nano smoothing is common to all axes, changing this item changes the setting for all axes.

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
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
  2 Drilling canned cycle G73, G74, G76, G80 to G89
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 When a number with no decimal point is specified, the least input increment of each axis is:
0: Not 10 times greater than the least command increment
1: 10 times greater than the least command increment
When the increment system is IS-A, and bit 0 (DPI) of parameter No. 3401 is set to 1 (pocket calculator type decimal point programming), the least input increment cannot be 10 times greater than the least command increment.

	#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	Bit 1 (ISC)	Bit 0 (ISA)
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 (axis name 1: parameter No. 1020) can be arbitrarily selected from A, B, C, U, V, W, X, Y, and Z. (When G code system A is used with the lathe system, however, U, V, and W are not selectable.) When bit 0 (EEA) of parameter No. 1000 is set to 1, the length of an axis name can be extended to three characters by setting axis name 2 (parameter No. 1025) and axis name 3 (parameter No. 1026) (extended axis name).
For axis names 2 and 3, a character from 0 to 9 and A to Z of ASCII code can be arbitrarily selected. However, the setting of axis name 3 for each axis is invalid if axis name 2 is not set. Moreover, if a character from 0 to 9 is set as axis name 2, do not use a character from A to Z as axis name 3.

(Tip) ASCII code

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

When G code system A is used with the lathe system, and the character X, Y, Z, or C is used as axis name 1 of an axis, a command with U, V, W, or H specified for axis name 1 represents an incremental programming for the axis.

NOTE
1 When the custom macro function is enabled, the same extended axis name as a reserved word cannot be used. Such an extended axis name is regarded as a reserved word.
Because of reserved words of custom macros, extended axis names that start with the following two characters cannot be used:
AB, AC, AD, AN, AS, AT, AX, BC, BI, BP, CA, CL, CO, US, WH, WR, XO, ZD, ZE, ZO, ZW
2 In a macro call, no extended axis name can be used as an argument.

1022	Setting of each axis in the basic coordinate system

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

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) and a 3-dimensional tool compensation space (XpYpZp), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

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

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

1023	Number of the servo axis for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
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 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, 9, ...) 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, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

1031	Reference axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

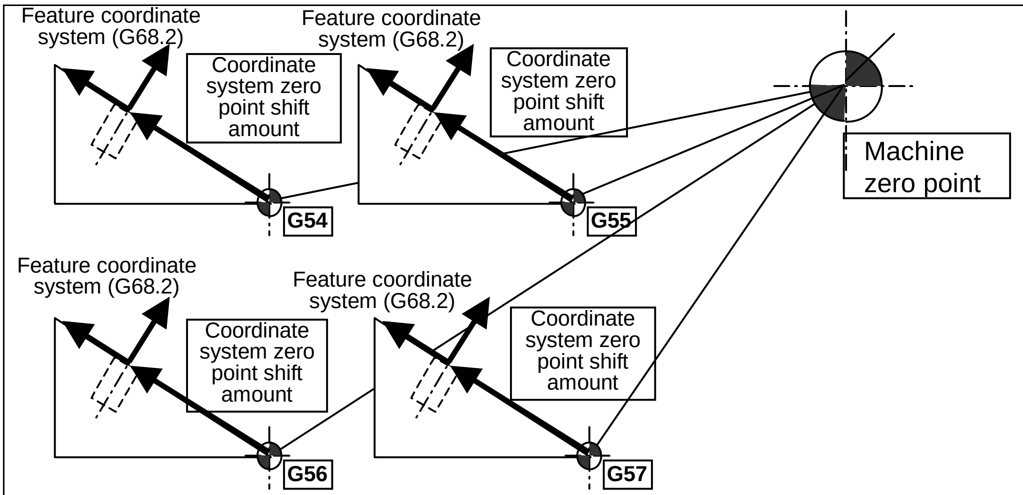
The unit of some parameters common to all axes such as those for dry run feedrate and single-digit F1 feedrate may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.
Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1205		3TW						

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

#6 3TW When workpiece coordinate system selection is specified with G code in tilted working plane indexing mode:
0: The alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)" is issued.
1: Workpiece coordinate system selection is executed.

> ⚠ **CAUTION**
> When this parameter is 1, only G54 to G59 or G54.1 can be specified. Specifying G52 or G92 causes alarm PS5462. Specifying G54 to G59 or G54.1 suppresses buffering.



Program example

```
G90 G17
G54
G68.2 X_Y_Z_ I_ J_ K_
G53.1
G43H_
G55 X_Y_Z_
G56 X_Y_Z_
G57 X_Y_Z_
G49
G69
```

X_Y_Z_: Coordinate system zero point shift amount

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

[Input type] Parameter input

[Data type] Bit path

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
When using 3-dimensional coordinate system conversion, set this parameter to 1.

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

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				

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

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

NOTE
Specify a feedrate in parameter No. 1423.

	#7	#6	#5	#4	#3	#2	#1	#0
1403				ROC				

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

#4 ROC In the threading cycle G76.7, rapid traverse override for retraction after threading is finished is:
0: Effective
1: Not effective (Override of 100%)

1410	Dry run rate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the dry run rate at the 100% position on the jog feedrate specification dial. The unit of data depends on the increment system of the reference axis. Setting this parameter to 0 results in alarm PS5009, "PARAMETER ZERO (DRY RUN)", being issued.

1411	Cutting feedrate

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

A cutting feedrate can be specified with this parameter for a machine which does not have to change the cutting feedrate frequently during machining. This eliminates the need to specify a cutting feedrate (F code) in the NC program.

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-64604EN).

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

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1423	Feedrate in manual continuous feed (jog feed) for each axis

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

(1) When bit 4 (JRV) of parameter No. 1402 is set to 0 (feed per minute), specify a jog feedrate (feed per minute) under an override of 100%.

(2) When bit 4 (JRV) of parameter No. 1402 is set to 1 (feed per revolution), specify a jog feedrate (feed per revolution) under an override of 100%.

NOTE
This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

1424	Manual rapid traverse rate for each axis

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

NOTE
1 If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2 When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

1430	Maximum cutting feedrate for each axis

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

1432	Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum cutting feedrate for each axis in the acceleration/deceleration before interpolation mode such as AI contour control. When the acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1601				RTO				

[Input type] Parameter input

[Data type] Bit path

#4 RTO Block overlap in rapid traverse
0: Blocks are not overlapped in rapid traverse.
1: Blocks are overlapped in rapid traverse.

	#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 during cutting feed
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

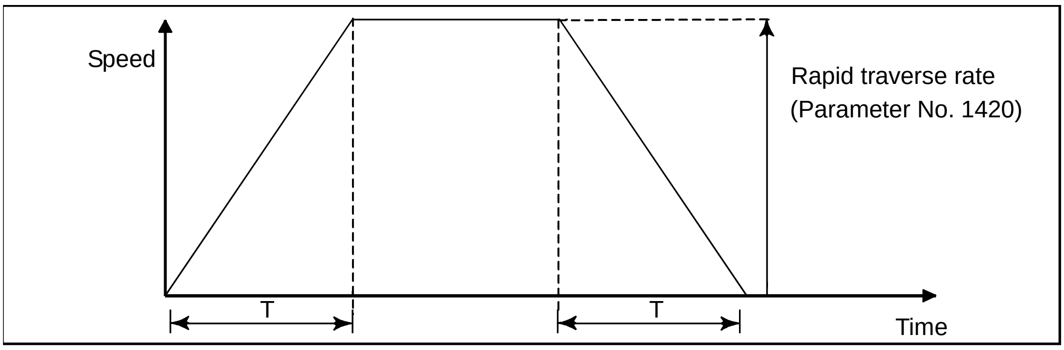
#1 CTBx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (depending on the setting in bit 0 (CTLx) of parameter No. 1610)
1: Bell-shaped acceleration/deceleration is applied.

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied. (Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)

1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

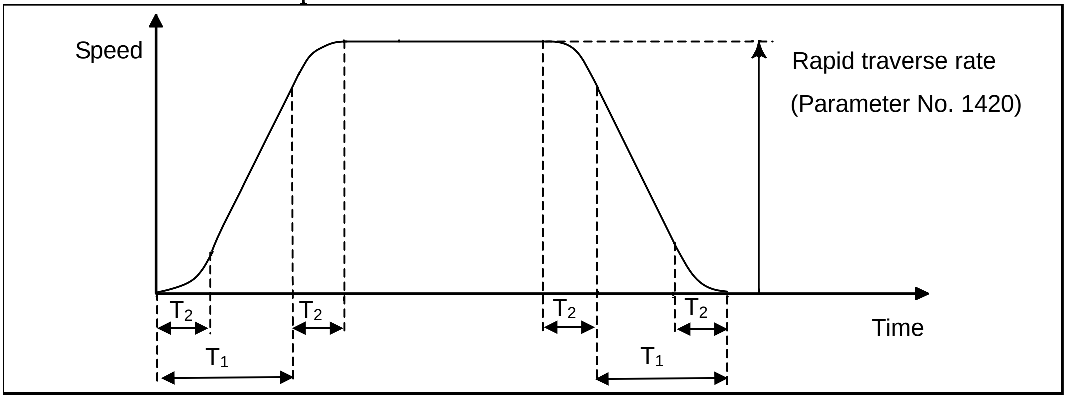
[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.
[Example]
For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621
(However, $T_1 \geq T_2$ must be satisfied.)
Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1621	Time constant T_2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

Specify time constant T_2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

1732	Minimum allowable feedrate for the deceleration function based on acceleration in circular interpolation

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

With the deceleration function based on acceleration in circular interpolation, an optimum feedrate is automatically calculated so that acceleration produced by changing the move direction in circular interpolation does not exceed the maximum allowable acceleration rate specified in parameter No. 1735.
If the radius of an arc is very small, a calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

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)
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation.
Feedrate is controlled so that acceleration produced by changing the move direction in circular interpolation does not exceed the value specified in this parameter.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled.
If a different value is set in this parameter for each axis, a feedrate is determined from the smaller of the acceleration rates specified for the two circular axes.

1737	Maximum allowable acceleration rate for the deceleration function based on acceleration in AI contour control 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)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate produced by changing the tool move direction.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled. If 0 is set for all axes, the deceleration function based on acceleration is not performed.
In circular interpolation, however, the deceleration function based on feedrate control using acceleration in circular interpolation (parameter No. 1735) is enabled.

1783	Maximum allowable feedrate difference for feedrate determination based on corner feedrate difference

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
If a feedrate component change for each axis exceeding the value set in this parameter occurs at the joint of blocks, the feedrate determination function based on corner feedrate difference finds a feedrate not exceeding the set value and performs deceleration by using acceleration/deceleration before interpolation. Thus, a shock to the machine and machining error at a corner can be reduced.

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.)

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.
If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm SV0411, "EXCESS ERROR (MOVING)" is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3106		DAK						

[Input type] Setting input
[Data type] Bit

#6 DAK Specifies whether to display coordinates in the program coordinate system or workpiece coordinate system as absolute coordinates when the 3-dimensional coordinate conversion mode, the tilted working plane indexing mode is set.
0: Display coordinates in the program coordinate system.
1: Display coordinates in the workpiece coordinate system.

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

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

#0 WOF Setting the tool offset value (tool wear offset) by MDI key input is:
0: Not disabled.
1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

> **NOTE**
> When tool offset memory A is selected, the tool offset set in the bit 0 (WOF) of parameter No.3290 is followed.

#1 GOF Setting the tool geometry offset value by MDI key input is:
0: Not disabled.
1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled

3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

When the modification of tool offset values by MDI key input is to be disabled using bits 0 (WOF) and 1 (GOF) of parameter No. 3290, parameters Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No. 3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No. 3295, set the number of such values. In the following cases, however, none of the tool offset values may be modified:

- When 0 or a negative value is set in parameter No. 3294
- When 0 or a negative value is set in parameter No. 3295
- When a value greater than the maximum tool offset number is set in parameter No. 3294

In the following case, a modification to the values ranging from the value set in parameter No. 3294 to the maximum tool offset number is disabled:

- When the value of parameter No. 3294 added to the value of parameter No. 3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI unit, the warning "WRITE PROTECT" is issued.

[Example] When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:

- Bit 1 (GOF) of parameter No. 3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No. 3290 = 1 (to disable tool wear offset value modification)
- Parameter No. 3294 = 51
- Parameter No. 3295 = 10

If the setting of bit 0 (WOF) of parameter No. 3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

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

#0 DPI When a decimal point is omitted in an address that can include a decimal point
0: The least input increment is assumed. (Normal decimal point input)
1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point input)

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR			G91	G19	G18	G01

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

#0 G01 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

#1 G18 Plane selected when power is turned on or when the control is cleared
0: G17 mode (plane XY)
1: G18 mode (plane ZX)

#2 G19 Plane selected when power is turned on or when the control is cleared
0: The setting of bit 1 (G18) of parameter No. 3402 is followed.
1: G19 mode (plane YZ)
When this bit is set to 1, set bit 1 (G18) of parameter No. 3402 to 0.

#3 G91 When the power is turned on or when the control is cleared
0: G90 mode (absolute programming)
1: G91 mode (incremental programming)

#6 CLR Reset button on the MDI unit, 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
3708								**SAR**

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

#0 SAR The spindle speed arrival signal SAR<Gn029.4> is:
0: Not checked
1: Checked

3740	Time elapsed prior to checking the spindle speed arrival signal

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
Set the time elapsed from the execution of the S function up to the checking of the spindle speed arrival signal SAR<Gn029.4>.

3770	Axis as the calculation reference in constant surface speed control

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis as the calculation reference in constant surface speed control.

NOTE
When 0 is set, constant surface speed control is always applied to the X-axis. In this case, specifying P in a G96 block has no effect on the constant surface speed control.

	#7	#6	#5	#4	#3	#2	#1	#0
5000							**MOF**	

[Input type] Setting input

[Data type] Bit path

#1 MOF When the tool length compensation shift type (bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400 is set to 1) is used, if the tool length compensation amount is changed(NOTE 3) in the tool length compensation mode (NOTE 1) when look-ahead blocks are present(NOTE 2):

0: Compensation is performed for the change in compensation amount as the movement type.

1: Compensation is not performed for the change until a tool length compensation command (offset number) and an absolute programming for the compensation axis are specified.

NOTE

1 The tool length compensation mode refers to the following state:
- Tool length offset (G43/G44)

2 "When look-ahead blocks are present" means as follows:
- The modal G code of the G codes (such as tool radius - tool nose radius compensation) of group 07 is other than G40.

One look-ahead block during automatic operation and multiple look-ahead blocks in the AI contour control mode are not included in the state "when look-ahead blocks are present".

3 Changes in tool length compensation amount are as follows:
- When the tool length compensation number is changed by H code (or D code for the extended tool selection function for lathe systems)
- When G43 or G44 is specified to change the direction of tool length compensation
- When the tool length compensation amount is changed using the offset screen, G10 command, system variable, PMC window, and so forth during automatic operation if bit 1 (EVO) of parameter No. 5001 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO		EVR	TAL		TLB	TLC

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

#0 TLC
#1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.

Tool length compensation A : Z-axis at all times
Tool length compensation B : Axis perpendicular to a specified plane (G17/G18/G19)
Tool length compensation C : Axis specified in a block that specifies G43/G44

#3 TAL Tool length compensation C

0: Generates an alarm when two or more axes are offset

1: Not generate an alarm even if two or more axes are offset

#4 EVR When a tool compensation value is changed in tool radius - tool nose radius compensation mode:

0: Enables the change, starting from that block where the next D or H code is specified.

1: Enables the change, starting from that block where buffering is next performed.

#6 EVO If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):

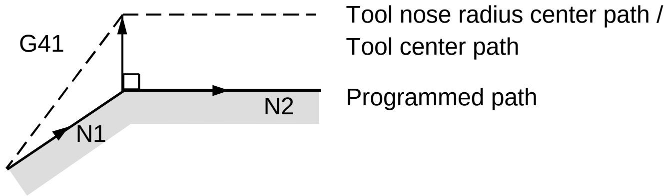
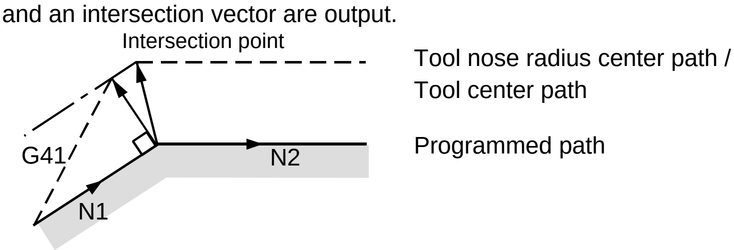
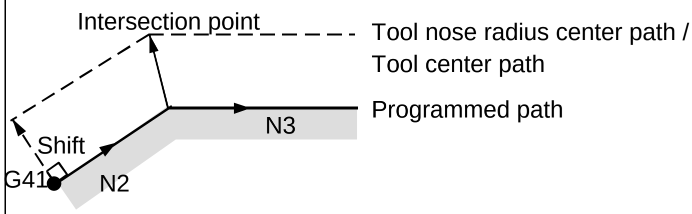
0: The new value becomes valid in a block where G43, G44, or an H code is specified next.

1: The new value becomes valid in a block where buffering is performed next.

	#7	#6	#5	#4	#3	#2	#1	#0
5003							SUV	SUP

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

#0 SUP
#1 SUV These bits are used to specify the type of startup/cancellation of tool radius - tool nose radius compensation.

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output. 
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 FS16*i*-T 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 unit 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 or tool offset operation.
0: Tool length compensation or tool offset operation is performed by an axis movement.
1: Tool length compensation or tool offset operation is performed by shifting the coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
5007				WMH	WMA	TMA	TC3	TC2

[Input type] Parameter input
[Data type] Bit path
#0 TC2
#1 TC3 If a tool length compensation value is set by pressing the [MEASURE] or [+MEASURE] soft key in tool length measurement, the tool automatically moves to the tool change position. Specify at which reference position the tool change position is located.

TC3	TC2	Meaning
0	0	The tool change position is at the first reference position.
0	1	The tool change position is at the second reference position.
1	0	The tool change position is at the third reference position.
1	1	The tool change position is at the fourth reference position.

#2 TMA 0: Tool length measurement is enabled along the Z-axis only.
1: Tool length measurement is enabled along each axis.

#3 WMA 0: Surface-based measurement of a workpiece zero point offset value is enabled along the Z-axis only.
1: Surface-based measurement of a workpiece zero point offset value is enabled along each axis.

#4 WMH 0: Hole-based measurement of a workpiece zero point offset value is disabled.
1: Hole-based measurement of a workpiece zero point offset value is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5008					CNV		CNC	

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

#1 CNC
#3 CNV These bits are used to select an interference check method in the tool radius - tool nose radius compensation mode.

CNV	CNC	Operation
0	0	Interference check is enabled. The direction and the angle of an arc are checked.
0	1	Interference check is enabled. Only the angle of an arc is checked.
1	-	Interference check is disabled.

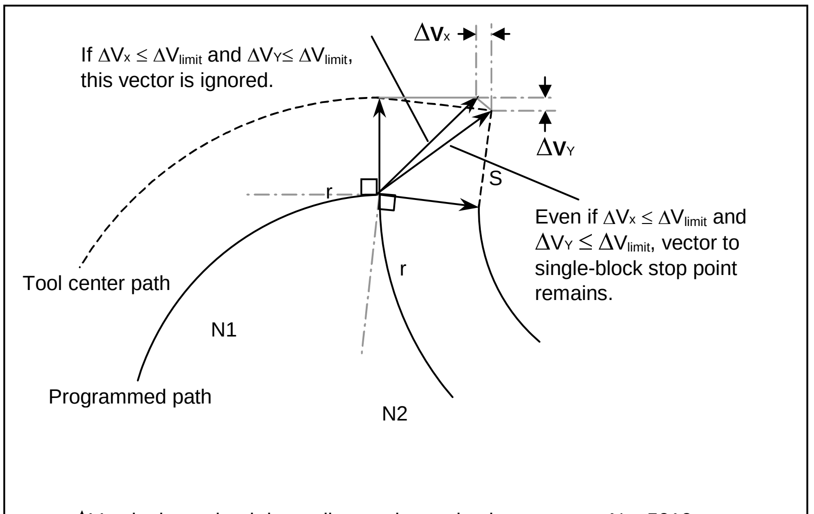
For the operation taken when the interference check shows the occurrence of an reference (overcutting) , see the description of bit 5 (CAV) of parameter No. 19607.

NOTE
Checking of only the direction cannot be set.

5010	Limit for ignoring the small movement resulting from tool radius - tool nose radius compensation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

When the tool moves around a corner in cutter compensation or tool nose radius compensation mode, the limit for ignoring the small travel amount resulting from compensation is set. This limit eliminates the interruption of buffering caused by the small travel amount generated at the corner and any change in feedrate due to the interruption.

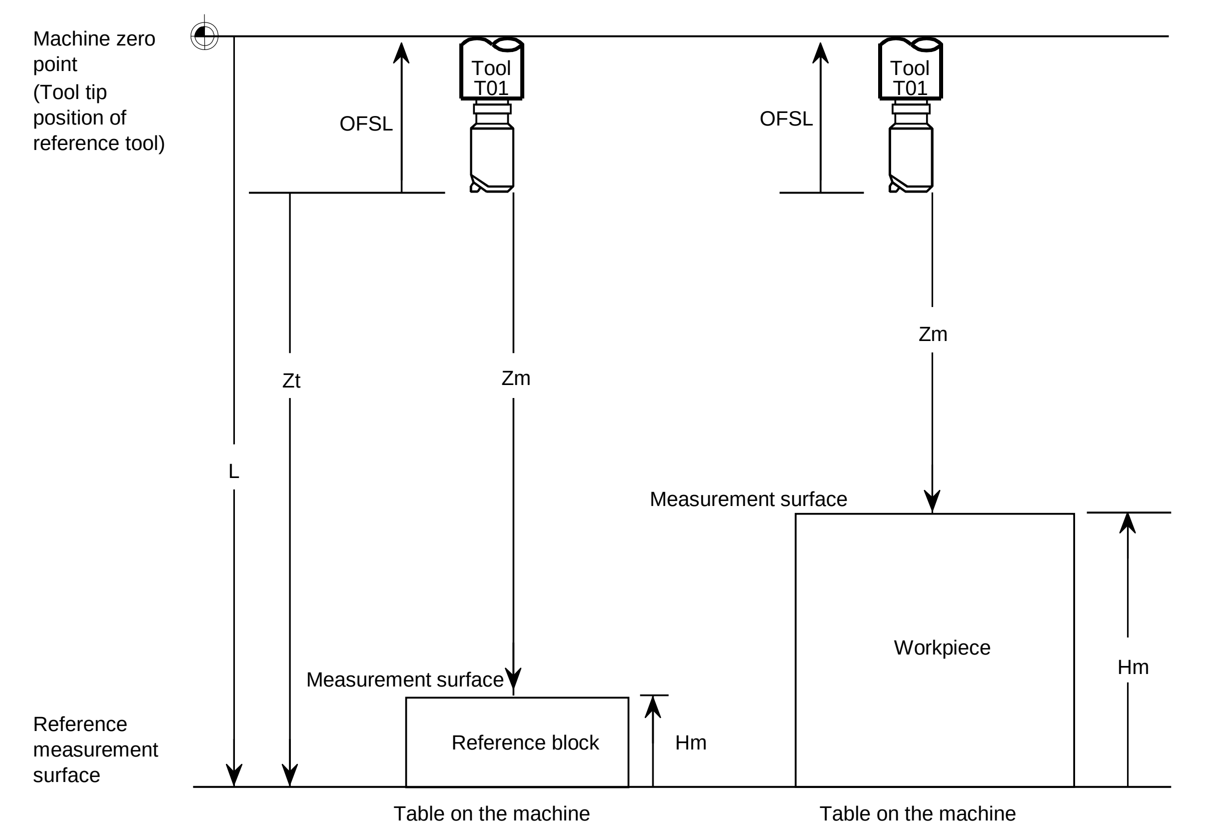


ΔV_{limit} is determined depending on the setting in parameter No. 5010.

5022	Distance (L) from reference tool tip position to the reference measurement surface

[Input type] Parameter input
[Data type] Real axis

[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
For each axis, this parameter sets the distance from the reference tool tip position to the reference measurement surface when the machine is at the machine zero point.



L: Distance from the reference tool tip to the reference measurement surface (machine coordinates of the reference measurement surface)
Hm: Distance from the reference measurement surface to actual measurement surface
Zm: Distance from the tool tip of the measured tool at the machine zero point to the measurement surface
Zt: Distance from the tool tip of the measured tool at the machine zero point to the reference measurement surface
OFSL: Tool length compensation (OFSL = Zm-Hm-L)

	#7	#6	#5	#4	#3	#2	#1	#0
5042					**OFE**	**OFD**	**OFC**	**OFA**

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

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OFA
#1 OFC
#2 OFD
#3 OFE These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.01mm	±9999.99mm
0	0	0	0	0.001mm	±9999.999mm
0	0	1	0	0.0001mm	±9999.9999mm
0	1	0	0	0.00001mm	±9999.99999mm
1	0	0	0	0.000001mm	±999.999999mm

For inch input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.001inch	±999.999inch
0	0	0	0	0.0001inch	±999.9999inch
0	0	1	0	0.00001inch	±999.99999inch
0	1	0	0	0.000001inch	±999.999999inch
1	0	0	0	0.0000001inch	±99.9999999inch

	#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
5104		PCT						

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

#6 PCT A Q command in a tapping cycle is:

0: Disabled.
1: Enabled.((High-speed) peck tapping cycle is assumed.)

When this parameter is set and the depth of cut for each time is specified with address Q in a tapping cycle command, a peck tapping cycle is assumed.
In a peck tapping cycle, either a high-speed peck tapping cycle or a peck tapping cycle can be selected by bit 5 (PCP) of parameter No. 5200.
Even when this parameter is set to 1, if Q is not specified or Q0 is specified, normal tapping is performed.

NOTE
1 Set also parameter No. 5213.
2 In rigid tapping, the Q command is valid regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5105								SBC

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

#0 SBC In a drilling canned cycle, chamfer cycle, or corner rounding cycle:
0: A single block stop is not performed.
1: A single block stop is performed.

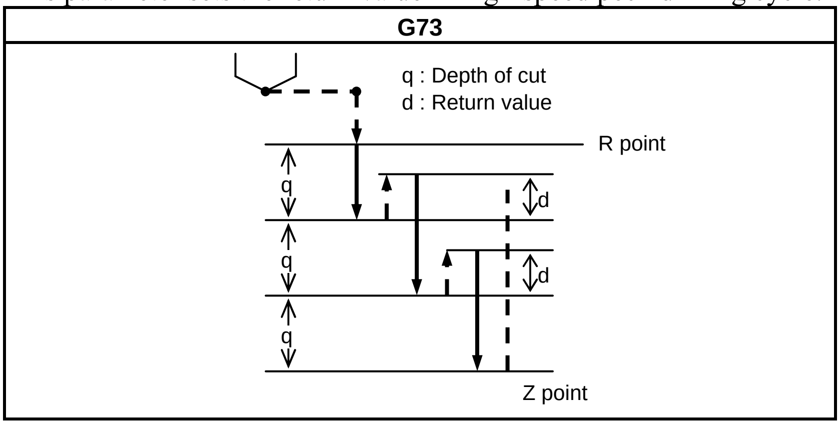
	#7	#6	#5	#4	#3	#2	#1	#0
5108		SPH						

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

#6 SPH When positioning the axes to hole position in Small-hole peck drilling cycle, the spindle is:
0: Stopped.
1: Not stopped.

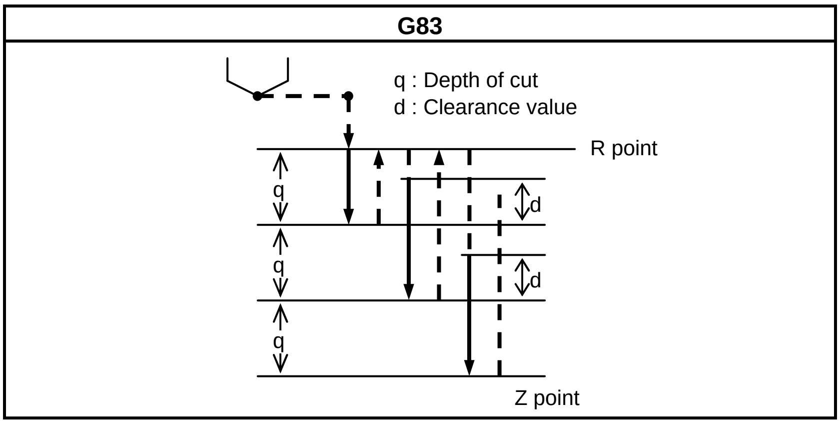
5114	Return value of high-speed peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the return value in high-speed peck drilling cycle.



5115	Clearance value in a peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a clearance value in a peck drilling cycle.



5148	Tool retraction direction after orientation in a fine boring cycle or back boring cycle

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] -24 to 24

This parameter sets an axis and direction for tool retraction after spindle orientation in a fine boring cycle or back boring cycle. For each boring axis, an axis and direction for tool retraction after orientation can be set. Set a signed axis number.

[Example] Suppose that:
When the boring axis is the X-axis, the tool retraction direction after orientation is -Y.
When the boring axis is the Y-axis, the tool retraction direction after orientation is +Z.
When the boring axis is the Z-axis, the tool retraction direction after orientation is -X.
Then, set the following (assuming that the first, second, and third axes are the X-axis, Y-axis, and Z-axis, respectively):
Set -2 in the parameter for the first axis. (The tool retraction direction is -Y.)
Set 3 in the parameter for the second axis. (The tool retraction direction is +Z.)
Set -1 in the parameter for the third axis. (The tool retraction direction is -X.)
Set 0 for other axes.

	#7	#6	#5	#4	#3	#2	#1	#0
5160						NOL	OLS	

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

#1 OLS When an overload torque detection signal is received in a peck drilling cycle of a small diameter, the feedrate and spindle speed are:
0: Not changed.
1: Changed.

#2 NOL When the depth of cut per action is satisfied although no overload torque detection signal is received in a peck drilling cycle of a small diameter, the feedrate and spindle speed are:
0: Not changed.
1: Changed.

5163	M code that specifies the peck drilling cycle mode of a small diameter

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999

This parameter sets an M code that specifies the peck drilling cycle mode of a small diameter.

5164	Percentage of the spindle speed to be changed at the start of the next advancing after an overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted because the overload torque detection signal is received.

$S2 = S1 \times d1 \div 100$

S1: Spindle speed to be changed
S2: Spindle speed changed

Set d1 as a percentage.

NOTE
When 0 is set, the spindle speed is not changed.

5165	Percentage of the spindle speed to be changed at the start of the next advancing when no overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted without the overload torque detection signal received.

$S2 = S1 \times d2 \div 100$

S1: Spindle speed to be changed
S2: Spindle speed changed

Set d2 as a percentage.

NOTE
When 0 is set, the spindle speed is not changed.

5166	Percentage of the cutting feedrate to be changed at the start of the next cutting after an overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances because the overload torque detection signal is received.

$F2 = F1 \times b1 \div 100$

F1: Cutting feedrate to be changed
F2: Cutting feedrate changed

Set b1 as a percentage.

> **NOTE**
> When 0 is set, the cutting feedrate is not changed.

5167	Percentage of the cutting feedrate to be changed at the start of the next cutting when no overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances without the overload torque detection signal received.

$F2 = F1 \times b2 \div 100$

F1: Cutting feedrate to be changed
F2: Cutting feedrate changed

Set b2 as a percentage.

> **NOTE**
> When 0 is set, the cutting feedrate is not changed.

5168	Lower limit of the percentage of the cutting feedrate in a peck drilling cycle of a small diameter

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the lower limit of the percentage of the cutting feedrate changed repeatedly to the specified cutting feedrate.

$FL = F \times b3 \div 100$

F: Specified cutting feedrate
FL: Changed cutting feedrate

Set b3 as a percentage.

5170	Number of the macro variable to which to output the total number of retractions during cutting

[Input type] Parameter input
[Data type] Word path
[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted during cutting. The total number cannot be output to common variables #500 to #599.

5171	Number of the macro variable to which to output the total number of retractions because of the reception of an overload torque detection signal

[Input type] Parameter input
[Data type] Word path
[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted after the overload torque detection signal is received during cutting. The total number cannot be output to common variables #500 to #599.

5172	Feedrate of retraction to point R when no address I is specified

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate of retraction to point R when no address I is specified.

5173	Feedrate of advancing to the position just before the bottom of a hole when no address I is specified

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate of advancing to the position just before the bottom of a previously machined hole when no address I is specified.

5174	Clearance in a peck drilling cycle of a small diameter

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the clearance in a peck drilling cycle of a small diameter.

5176	Grinding axis number in Plunge Grinding Cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Plunge Grinding Cycle(G75).

NOTE

The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5177	Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle (G77).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5178	Grinding axis number of Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Continuous feed surface grinding cycle(G78).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5179	Grinding axis number of Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the Grinding axis number of Intermittent feed surface grinding cycle(G79).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5180	Axis number of dressing axis in Plunge grinding cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Plunge grinding cycle(G75).

NOTE
The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5181	Axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5182	Axis number of dressing axis in Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Continuous feed surface grinding cycle(G78).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5183	Axis number of dressing axis in Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Intermittent feed surface grinding cycle(G79).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, "ILLEGAL PARAMETER IN GRINDING" is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5200		FHD	PCP	DOV				G84

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

#0 G84 Method for specifying rigid tapping:
0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No. 5210).
1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

#4 DOV Override during extraction in rigid tapping:
0: Invalidated
1: Validated (The override value is set in parameter No. 5211. However, set an override value for rigid tapping return in parameter No. 5381.)

#5 PCP 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
5203				OVS				

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

#4 OVS In rigid tapping, override by the feedrate override select signal and cancellation of override by the override cancel signal is:
0: Disabled.
1: Enabled.

When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

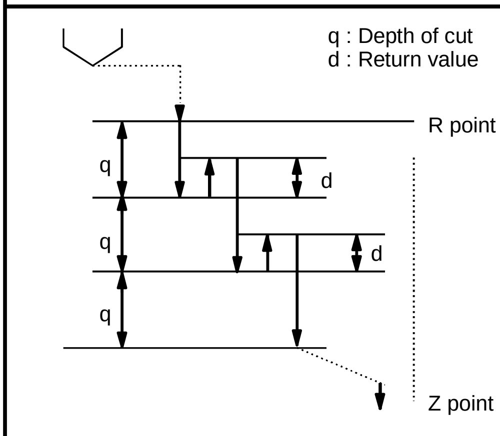
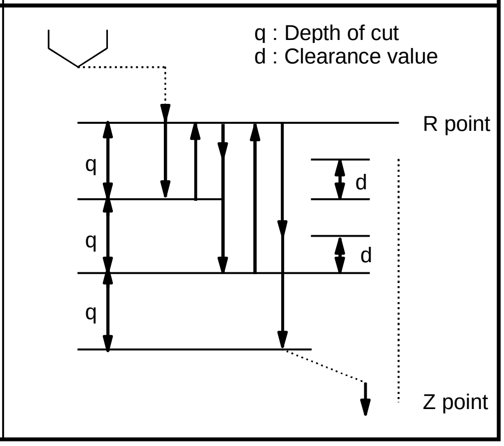
5211	Override value during rigid tapping extraction

[Input type] Parameter input
[Data type] Word path
[Unit of data] 1% or 10%
[Valid data range] 0 to 200
The parameter sets the override value during rigid tapping extraction.

> **NOTE**
> The override value is valid when bit 4 (DOV) of parameter No. 5200 is set to 1. When bit 3 (OVU) of parameter No. 5201 is set to 1, the unit of set data is 10%. An override of up to 200% can be applied to extraction.

5213	Return in peck rigid tapping cycle

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the drilling axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the 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, "ILLEGAL S CODE COMMAND" will be issued. This applies to the M series.

5321	Spindle backlash in rigid tapping (first-stage gear)
5322	Spindle backlash in rigid tapping (second-stage gear)
5323	Spindle backlash in rigid tapping (third-stage gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] -9999 to 9999
Each of these parameters is used to set a spindle backlash.

	#7	#6	#5	#4	#3	#2	#1	#0
5400	SCR	XSC	LV3					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)

#5 LV3 When system variables #100101 to #100132 (current position coordinates) and #100151 to #100182 (skip coordinates) are read in the 3-dimensional coordinate conversion mode or tilted working plane indexing mode:
0: Coordinates of the workpiece coordinate system can be read.
1: Coordinates of the program coordinate system after 3-dimensional coordinate conversion or tilted working plane indexing can be read.
This parameter applies also to system variables #5041 to #5060 (current position coordinates) and #5061 to #5080 (skip coordinates).

#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 bit 7 (SCR) 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 bit 7 (SCR) 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 this parameter is set, the power must be turned off before operation is continued.

#0 MDL The G60 code (single direction positioning) is:
0: One-shot G code (group 00).
1: Modal G code (group 01).

5480	Number of the axis for controlling the normal direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum controlled axis number

This parameter sets the controlled axis number of the axis for controlling the normal direction.

5481	Feedrate of rotation of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/min
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)

This parameter sets the feedrate of the movement along the normal direction controlled axis that is inserted at the start point of a block during normal direction control.

5482	Limit value used to determine whether to ignore the rotation insertion of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

The rotation block of the normal direction controlled axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting.

The ignored rotation angle is added to the next rotation insertion angle, and the block insertion is then judged.

> **NOTE**
> 1 No rotation block is inserted when 360 or more degrees are set.
> 2 If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation setting is 180 or more degrees.

	#7	#6	#5	#4	#3	#2	#1	#0
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.

> **NOTE**
> Be sure to set this parameter to 1.
> If an absolute programming is commanded to the index table indexing axis when this parameter is set to 0, the position display of the absolute coordinate system may be not corresponding to the absolute programming value like the following example.
>
> Example) When indexing by rotating in a positive direction :
> N10 G90 B20.;
> N20 B10.; ← It rotates in a positive direction by 350 degree. At this time, 370.0 is displayed to the position display of the absolute coordinate system.

#3 INC When the M code that specifies rotation in the negative direction (parameter No. 5511) is not set, rotation in the G90 mode is:
0: Not set to the shorter way around the circumference.
1: Set to the shorter way around the circumference.

#4 G90 A command for the index table indexing axis is:
0: Assumed to be an absolute or incremental programming according to the mode.
1: Always assumed to be an absolute programming.

#6 SIM When the same block includes a command for the index table indexing axis and a command for another controlled axis:
0: The setting of bit 0 (IXSx) of parameter No. 5502 is followed.
1: The commands are executed.

> **NOTE**
> Even when this parameter is set to 1, an alarm PS1564, "INDEX TABLE AXIS - OTHER AXIS SAME TIME" is issued if the block is neither G00, G28, nor G30 (or the G00 mode).

	#7	#6	#5	#4	#3	#2	#1	#0
5501								ITI

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

#0 ITI The index table indexing function is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5502								IXSx

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

#0 IXSx When a command is specified in a block that contains a command for the index table indexing axis:
0 : An alarm PS1564, "INDEX TABLE AXIS - OTHER AXIS SAME TIME" is issued.
1 : The command is executed.

If bit 6 (SIM) of parameter No. 5500 is set to 1, a simultaneous operation with all axes except the index table indexing axis can be performed regardless of the setting of this parameter.
To set an axis that allows simultaneous operation for each axis, set SIM to 0, and set this parameter.

> **NOTE**
> Even when this parameter is set to 1, an alarm PS1564, "INDEX TABLE AXIS - OTHER AXIS SAME TIME" is issued if the block is neither G00, G28, nor G30 (or the G00 mode).

5511	M code that specifies rotation in the negative direction for index table indexing

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999
0: The rotation direction for the index table indexing axis is determined according to the setting of bit 3 (INC) of parameter No. 5500 and a command.
1 to 99999999:
The rotation for the index table indexing axis is always performed in the positive direction. It is performed in the negative direction only when a move command is specified together with the M code set in this parameter.

> **NOTE**
> Be sure to set bit 2 (ABS) of parameter No. 5500 to 1.

5512	Minimum positioning angle for the index table indexing axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the minimum positioning angle (travel distance) for the index table indexing axis. The travel distance specified in the positioning command must always be an integer multiple of this setting. When 0 is set, the travel distance is not checked.

The minimum positioning angle is checked not only for the command, but also for the coordinate system setting and workpiece origin offset.

	#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 for specifying custom macro control is executed, a branch to 30 sequence numbers just before the GOTO statement or to up to 10 sequence numbers saved by a sequence number search previously made with a GOTO statement is:
0: Not made at high speed.
1: Made at high speed.

	#7	#6	#5	#4	#3	#2	#1	#0
6019				MSV				

[Input type] Parameter input
[Data type] Bit

#4 MSV When Tool length compensation shift type is used, the value in which Tool offset value, Tool length offset and Tool holder offset are :
#5041 - #5060, #100101 - #100151 (Current position)
#5061 - #5080, #100151 - #100200 (Skip position):
0: It is included in above-mentioned system value.
1: It is not included in above-mentioned system value.
Only in the machining center system, this parameter becomes effective.

	#7	#6	#5	#4	#3	#2	#1	#0
6200				HSS				

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

#4 HSS 0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

	#7	#6	#5	#4	#3	#2	#1	#0
6208					9S4	9S3	9S2	9S1

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

9S1 to 9S4 Specify which high-speed skip signal is enabled for the continuous high-speed skip command G31P90 or the EGB skip and the skip function for flexible synchronization control command G31.8.
The settings of each bit have the following meaning:
0: The high-speed skip signal corresponding to the bit is disabled.
1: The high-speed skip signal corresponding to the bit is enabled.
The bits correspond to signals as follows:

Parameter	High-speed skip signal
9S1	HDI0
9S2	HDI1
9S3	HDI2
9S4	HDI3

	#7	#6	#5	#4	#3	#2	#1	#0
6210	CCM	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.

#7 CCM The current offset amount of automatic tool length measurement is:
0: The offset amount set to the offset screen.
In case of C, the value for tool wear offset is selected.
1: The offset amount actually effected.

	#7	#6	#5	#4	#3	#2	#1	#0
6240	IGA							

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

#7 IGA Automatic tool length measurement (M series) or automatic tool compensation (T series) is:
0: Used.
1: Not used.

6241	Feedrate during measurement of automatic tool length measurement (for the XAE1 and GAE1 signals)

6242	Feedrate during measurement of automatic tool length measurement (for the XAE2 and GAE2 signals)

6243	Feedrate during measurement of automatic tool length measurement (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
These parameters set the relevant feedrate during measurement of automatic tool length measurement.

> **NOTE**
> When the setting of parameter No. 6242 or 6243 is 0, the setting of parameter No. 6241 is used.

6251	γ value during automatic tool length measurement (for the XAE1 and GAE1 signals)

6252	γ value during automatic tool length measurement (for the XAE2 and GAE2 signals)

6253	γ value during automatic tool length measurement (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the relevant γ value during automatic tool length measurement.

> **NOTE**
> When the Reference axis (parameter No.1031) is Diameter specification, specify the diameter value. When the Reference axis (parameter No.1031) is Radius specification, specify the radius value.

6254	ε value during automatic tool length measurement (for the XAE1 and GAE1 signals)

6255	ε value during automatic tool length measurement (for the XAE2 and GAE2 signals)

6256	ε value during automatic tool length measurement (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the relevant ε value during automatic tool length measurement.

> **NOTE**
> When the Reference axis (parameter No.1031) is Diameter specification, specify the diameter value. When the Reference axis (parameter No.1031) is Radius specification, specify the radius value.

		#7	#6	#5	#4	#3	#2	#1	#0
7001								ABS	

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

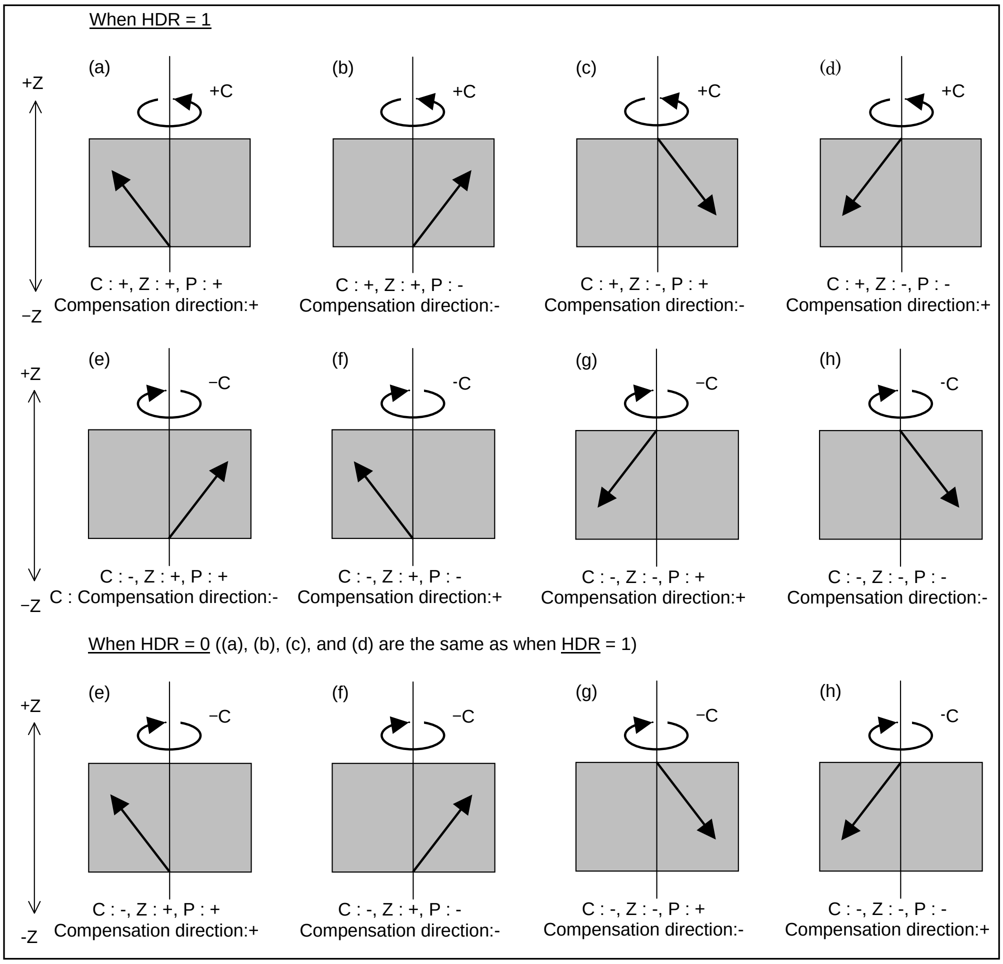
#1 ABS For the move command after manual intervention in the manual absolute on state:
0: Different paths are used in the absolute (G90) and incremental (G91) modes.
1: The same path (path in the absolute mode) is used in the absolute (G90) and incremental (G91) modes.

		#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 synchronization mode (G81).
1: Does not cancel the synchronization 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	**PHD**	**PHS**			**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 5000
1: 0.1 to 500 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 5000 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 (parameters 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.

#6 PHS When the G81/G80 block contains no R command:
0: Acceleration/deceleration is not performed at the start or cancellation of EGB synchronization.
1: Acceleration/deceleration is performed at the start or cancellation of EGB synchronization. After acceleration at the start of synchronization, phase synchronization is automatically performed.

#7 PHD The direction of movement for automatic phase synchronization is:
0: Positive (+).
1: Negative (-).

	#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 The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization or automatic operation (automatic operation signal OP <Fn000.7> = “1”).
1: EGB synchronization.

#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”).

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
1 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).
2 This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 1.

7710	Axis number of an axis to be synchronized using the method of command specification for a hobbing machine

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

When there are several groups of axes to be synchronized (the axes for which bit 0 (SYNMOD) of parameter No. 2011 is set to 1), an axis for which to start synchronization is specified using the following command (for a hobbing machine):

G81 T t L ± *l* ;

t: Spindle speed ($1 \leq t \leq 5000$)

l: Number of synchronized axis rotations ($-250 \leq l \leq 250$)

Synchronization between the spindle and a specified axis is established with the ratio of ±*l* rotations about the synchronized axis to t spindle rotations.

t and *l* correspond to the number of teeth and the number of threads on the hobbing machine, respectively.

Above command is issued without setting this parameter when there are several groups of axes to be synchronized, the alarm PS1593, “EGB PARAMETER SETTING ERROR” is issued.

NOTE
The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
7731	HAD	EPA			ECN			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 canned cycle for drilling can be used.

#3 ECN When the automatic phase synchronization function for the electronic gear box is disabled, during EGB synchronization, the G81 command:
0: Cannot be issued again. (The alarm PS1595, “ILL-COMMAND IN EGB MODE” is issued.)
1: Can be issued again.

#6 EPA Automatic phase synchronization for the electronic gear box is performed in such a way that:
0: The machine coordinate 0 of the slave axis is aligned to the position of the master axis one-rotation signal.
1: The position of the slave axis at synchronization start is aligned to the position of the master axis one-rotation signal. (Specification of Series 16*i*)

#7 HAD In electronic gear box, the timing for reflecting helical gear compensation and travel distance of automatic phase synchronization to absolute coordinates is:
0: When synchronization is canceled.
1: During helical gear compensation and automatic phase synchronization.

7740	Feedrate during retraction

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

7741	Retract amount

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the retract amount for each axis.

NOTE
The tool moves (is retracted) by the specified amount regardless of whether diameter or radius programming is specified.

7772	Number of position detector pulses per rotation about the tool axis

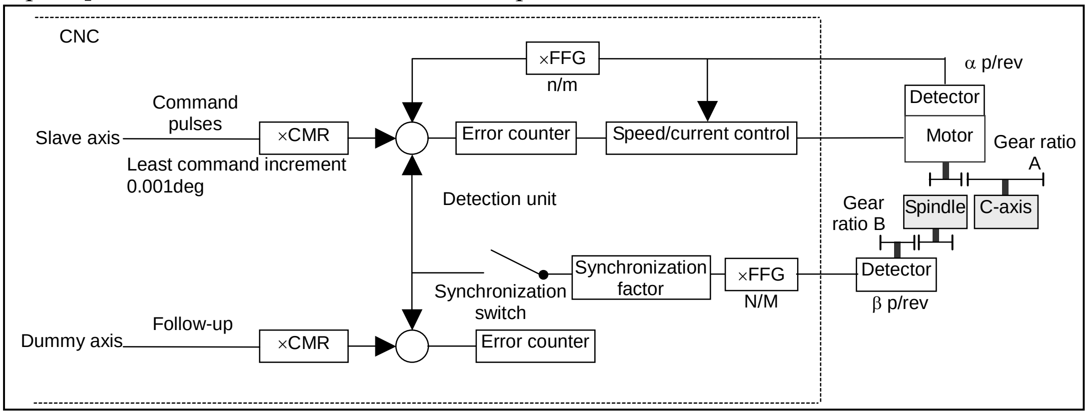
[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the tool axis (on the spindle side), for the position detector.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[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.

7776	Feedrate during automatic phase synchronization for the workpiece axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg/min
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets the feedrate during automatic phase synchronization for the workpiece axis.

When this parameter is set to 0, the rapid traverse rate (parameter No. 1420) is used as the feedrate during automatic phase synchronization.

7777	Angle shifted from the spindle position (one-rotation signal position) which the workpiece axis uses as the reference of phase synchronization

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0.000 to 360.000 (when the increment system is IS-B)

This parameter sets the angle shifted from the spindle position (one-rotation signal position) which the workpiece axis uses as the reference of phase synchronization.

7778	Acceleration for acceleration/deceleration for the workpiece axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/sec^2
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(For a millimeter machine, 0.0 to +100000.0, for an inch machine, 0.0 to +10000.0)

This parameter sets an acceleration for acceleration/deceleration for the workpiece axis.

NOTE

1 In the FS 16*i*, acceleration/deceleration for automatic phase matching is set by specifying a feedrate and a time constant in parameters Nos. 2135 and 2136 (Nos. 4384 and 4385 in the case of spindle EGB) separately; in the FS 0*i*-F, acceleration/deceleration is directly set in parameter No. 7778.

2 If this parameter is set to 0, specifying G81 results in an alarm PS1598, “EGB AUTO PHASE PARAMETER SETTING ERROR”.

	#7	#6	#5	#4	#3	#2	#1	#0
8131					AOV		F1D	HPG

NOTE

When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 HPG Manual handle feed is:
0: Not Used.
1: Used.

#1 F1D One-digit F code feed is:
0: Not Used.
1: Used.

#3 AOV Automatic corner override is:
0: Not Used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8132			SCL	SPK	IXC			

> **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

#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 diameter 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
8135			NMC		NRG			

> **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

#5 NMC Custom macro is:
0: Used.
1: Not Used.

#3 NRG Rigid tapping is:
0: Used.
1: Not Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8136		NGW				NWN		NWZ

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 NWZ Workpiece coordinate system is:
0: Used.
1: Not Used.

#2 NWN Addition of workpiece coordinate system pair (48 pairs) is:
0: Used.
1: Not Used.

> **NOTE**
> To use the addition of workpiece coordinate system pair (48 pairs), set 0 to bit 0 (NWZ) and bit 0 (NWN) of parameter No.8136

#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
8137				NCD				

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#4 NCD Canned cycles for drilling is:
0: Used.
1: Not Used.

8486	Maximum travel distance of a block where Nano smoothing is applied

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter specifies a block length used as a reference to decide whether to apply Nano smoothing. If the line specified in a block is longer than the value set in the parameter, Nano smoothing is not applied to that block.
When 0 is set in this parameter, this parameter setting value is regarded as 5.0mm.

8487	Angle at which Nano smoothing is turned off

[Input type] Setting input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90

This parameter sets the angle used to determine whether to apply Nano smoothing.
At a point having a difference in angle greater than this setting, Nano smoothing is turned off.
When 0 is set in this parameter, this parameter setting value is regarded as 20 degree.

8490	Minimum travel distance of a block where Nano smoothing is applied

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a block length used to determine whether to apply Nano smoothing.
If the line specified in a block is shorter than the value set in this parameter, Nano smoothing is not applied to that block.

	#7	#6	#5	#4	#3	#2	#1	#0
10351			PCC					

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#5 PCC In Polar coordinate command, the specification of which the address in the selected plane 1st axis (radius) or 2nd axis (angle) is omitted
0: is FS0*i*-F specification.
1: is FS0*i*-C compatible specification.

10360	Bias value for the offset number of a tool offset for each axis

[Input type] Setting input
[Data type] Word axis
[Valid data range] 0 to the number of tool offsets

When parallel operation is performed, this parameter specifies a bias value for the offset number of a tool offset for each axis. The offset data to be used as a tool offset for an axis has a number obtained by adding a value set in this parameter for the axis to a specified offset number.

10361	Bias for the offset number of tool length compensation for each axis

[Input type] Setting input
[Data type] Word axis
[Valid data range] 0 to the number of tool offsets

When parallel operation is performed, this parameter specifies a bias value for the offset number of tool length compensation for each axis. The offset data to be used as the tool length compensation amount for an axis has a number obtained by adding a value set in this parameter for the axis to a specified offset number.

	#7	#6	#5	#4	#3	#2	#1	#0
10370								RPC

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

#0 RPC When a return from the reference position (G29) is made, axis switching is:
0: Disabled.
1: Enabled.

10371	Axis switching number

[Input type] Setting input
[Data type] Byte path
[Valid data range] 0 to 5

One of six types of axis switching can be selected by setting its axis switching number in this parameter. Programmed addresses X, Y, and Z correspond to machine axes x, y, and z as follows:

Axis switching No.	Programmed address		
	X	Y	Z
0	x	y	z
1	x	z	y
2	y	x	z
3	y	z	x
4	z	x	y
5	z	y	x

Axis switching number 0 indicates that axis switching is not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
11302	CPG							

[Input type] Parameter input
[Data type] Bit

#7 CPG PROG function screen selection is:
0: Not changed according to the CNC mode.
1: Changed according to the CNC mode.

	#7	#6	#5	#4	#3	#2	#1	#0
11304							GGD	

[Input type] Parameter input

[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 GGD The G code guidance screen is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11350		QLS						

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#6 QLS The machining quality level adjustment screen is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11400						TOP		

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

#2 TOP Set a tool length compensation or tool offset operation.
0: Tool length compensation or tool offset operation is performed by an axis movement.
1: Tool length compensation or tool offset operation is performed by shifting the coordinate system.

> **NOTE**
> This parameter is an-individual path parameter having the same function as bit 6 (TOS) of parameter No. 5006.
> To use different compensation types for individual paths, set the bit 6 (TOS) of parameter No.5006 with 0 and specify a compensation type for each path separately, using the parameter TOP. If the bit 6 (TOS) of parameter No.5006 is 1, the bit 2 (TOP) of parameter No.11400 is assumed to be 1 even if it is 0.

	#7	#6	#5	#4	#3	#2	#1	#0
11507	SAC							

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#7 SAC When the spindle speed arrival SAR is checked in canned cycle for drilling,
0: It is waiting for elapsing time that is set parameter No.3740 at the starting of all drilling.
1: It is waiting for elapsing time that is set parameter No.3740 at the starting of only first drilling. It is available block overlap between rapid traverse to the initial lever and rapid traverse of positioning to a position of hole.

NOTE
1 This parameter is available when bit 0 (SAR) of parameter No.3708 is set 1.
2 Block overlap in rapid traverse is available when bit 4 (RTO) of parameter No.1601 is set 1.

	#7	#6	#5	#4	#3	#2	#1	#0
11600			AX1					

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

#5 AX1 If, in coordinate system rotation mode, a 1-axis command is issued in absolute mode,
0: First, the specified position is calculated in the coordinate system before rotation, and then the coordinate system is rotated.
1: First, the coordinate system is rotated, and then the tool moves to the specified position in the coordinate system.
(FS16*i*/18*i*/21*i* compatible specification)

	#7	#6	#5	#4	#3	#2	#1	#0
11630						TFR		FRD

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

#0 FRD The minimum command unit of the rotation angles of coordinate rotation and 3-dimensional coordinate system conversion is:
0: 0.001 degree.
1: 0.00001 degree. (1/100,000)

#2 TFR The minimum command unit of the rotation angles of the tilted working plane indexing command is:
0: 0.001 degree.
1: 0.00001 degree.

11682	Tolerance when nano smoothing is used (smoothing level 1)
11683	Tolerance when nano smoothing is used (smoothing level 10)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
Each of these parameters sets a tolerance value when nano smoothing is used.
It is necessary to set the value of both level1 and level10.

	#7	#6	#5	#4	#3	#2	#1	#0
11785								CAT

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

#0 CAT On start up of automatic operation, smart tolerance control is:
0: Ineffective on an axis.
1: Effective on an axis.

11786	Tolerance for linear axis in smart tolerance control mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
Set tolerance for linear axes for curves generated in smart tolerance control mode. If tolerance is not specified in tolerance mode, machining path is controlled so that the machining accuracy on curves represented by small segment is within the tolerance set to this parameter.
When this parameter is set to 0 or less, it operates as 10μm is set.

12310	States of the manual handle feed axis selection signals when tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt are performed

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes
This parameter sets the states of the manual handle feed axis selection signal (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signal (HS1IA to HS1IE for the first manual handle) to perform tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt.
The handle for which the signal states are set is determined by parameter No. 12323.

<Table of correspondence with the manual handle feed axis selection signals>
If parameter No. 12323 is set to 1, the states of the manual handle feed axis selection signals or manual handle interrupt axis selection signals for the first manual handle in the 3-dimensional manual feed (handle feed) mode and corresponding parameter settings are listed in the table below. When the first manual handle pulse generator is turned after setting the signals corresponding to the value set in the parameter, operation is performed in the specified mode.
If the value set in the parameter is larger than number of controlled axes, the movement is not generated.

HS1E (HS1IE)	HS1D (HS1ID)	HS1C (HS1IC)	HS1B (HS1IB)	HS1A (HS1IA)	Parameter (No. 12310)
0	0	0	0	1	1
0	0	0	1	0	2

HS1E (HS1IE)	HS1D (HS1ID)	HS1C (HS1IC)	HS1B (HS1IB)	HS1A (HS1IA)	Parameter (No. 12310)
0	0	0	1	1	3
0	0	1	0	0	4
0	0	1	0	1	5
0	0	1	1	0	6
0	0	1	1	1	7
0	1	0	0	0	8
0	1	0	0	1	9
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24

If parameter No. 12323 is set to 2 to 5, replace 1 in HS1A to HS1E and HS1IA to HS1IE above with 2 to 5.

12311	States of the manual handle feed axis selection signals when a movement is made in the first axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signal (HS1IA to HS1IE for the first manual handle) when a movement is made in the first axis direction. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

The table below indicates the relationships of tool axis directions, first axis directions, and second axis directions.

Parameter No. 19697	Tool axis directions	First axis directions	Second axis directions
1	X	Y	Z
2	Y	Z	X
3	Z	X	Y

Note, however, that the table above indicates the directions applicable when the angles of all rotation axes are set to 0.
In tool axis direction/tool axis normal direction feed (not table-based), the directions indicated above assume that 0 is set in parameter No. 19698 and No. 19699. When a rotation axis has made a turn or a nonzero value is set in these parameters in tool axis direction/tool axis normal direction feed, the relevant directions are inclined accordingly.

12312	States of the manual handle feed axis selection signals when a movement is made in the second axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE for the first manual handle) when a movement is made in the second axis direction. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)

The handle for which the signal states are set is determined by parameter No. 12323.

12313	States of the manual handle feed axis selection signals when the first rotation axis is turned in tool tip center rotation handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE for the first manual handle) when the first rotation axis is turned in tool tip center rotation handle feed or interrupt. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)

The handle for which the signal states are set is determined by parameter No. 12323.

12314	States of the manual handle feed axis selection signals when the second rotation axis is turned in tool tip center rotation handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE for the first manual handle) when the second rotation axis is turned in tool tip center rotation handle feed or interrupt. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)

The handle for which the signal states are set is determined by parameter No. 12323.

12318	Tool length in 3-dimensional machining manual feed

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a tool length when tool tip center rotation feed is performed with the 3-dimensional machining manual feed function and when the 3-dimensional machining manual feed screen is displayed.

> **NOTE**
> 1 Specify a radius value to set this parameter.
> 2 Don't change this parameter during 3-dimensional manual feed mode.

	#7	#6	#5	#4	#3	#2	#1	#0
12319								CAC

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

#0 CAC If a workpiece coordinate system offset is set for the rotation axis, the coordinate system of the rotation axis used to calculate the 3-dimensional manual feed is:
0: Machine coordinate system.
For those parameters Nos. 19680 to 19714 used to configure the machine that depend on the coordinates of the rotation axis, set the values assumed when the machine coordinates of the rotation axis are 0.
1: Workpiece coordinate system.
For those parameters Nos. 19680 to 19714 used to configure the machine that depend on the coordinates of the rotation axis, set the values assumed when the workpiece coordinates of the rotation axis are 0.

	#7	#6	#5	#4	#3	#2	#1	#0
12320						JFR	FLL	TWD

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

#0 TWD The directions of 3-dimensional machining manual feed (other than tool tip center rotation feed) when the tilted working plane indexing is issued are:
0: Same as those not in the tilted working plane indexing. That is, the directions are:
Tool axis normal direction 1 (table-based horizontal direction 1)
Tool axis normal direction 2 (table-based horizontal direction 2)
Tool axis direction (table-based vertical direction)
1: X, Y, and Z directions in the feature coordinate system.

> **NOTE**
> Don't change this parameter during 3-dimensional manual feed mode.

#1 FLL The directions of tool axis normal direction feed or table-based horizontal direction feed in the 3-dimensional machining manual feed mode are:
0: Tool axis normal direction 1 (table-based horizontal direction 1) and tool axis normal direction 2 (table-based horizontal direction 2).
1: Longitude direction and latitude direction.

Bit 1 (FLL) of parameter No. 12320	Bit 0 (TWD) of parameter No. 12320	Directions of 3-dimensional machining manual feed
0	0	Conventional directions
0	1	When the tilted working plane indexing is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Conventional directions
1	0	Longitude direction and latitude direction

Bit 1 (FLL) of parameter No. 12320	Bit 0 (TWD) of parameter No. 12320	Directions of 3-dimensional machining manual feed
1	1	When the tilted working plane indexing is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Longitude direction and latitude direction

NOTE
Don't change this parameter during 3-dimensional manual feed mode.

#2 JFR As the feeedrate of 3-dimensional machining manual feed (jog feed or incremental feed) :
0: The dry run rate (parameter No. 1410) is used.
1: The jog feedrate (parameter No. 1423) is used.

12321	Normal axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

When a tilted working plane indexing (G68.3) is issued to perform 3-dimensional machining manual feed in the latitude direction, longitude direction, and tool axis direction, this parameter sets an axis parallel to the normal direction.
1 : Positive (+) X-axis direction
2 : Positive (+) Y-axis direction
3 : Positive (+) Z-axis direction
0 : Reference tool axis direction (parameter No. 19697)

12322	Angle used to determine whether to assume the tool axis direction to be parallel to the normal direction (parameter No. 12321)

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90

When a tilted working plane indexing (G68.3) is issued to perform 3-dimensional machining manual feed in the latitude direction, longitude direction, and tool axis direction, if the angle between the tool axis direction and normal direction (parameter No. 12321) is too small, the tool axis direction is assumed to be parallel to the normal direction (parameter No. 12321). This parameter sets the maximum angle at which the tool axis direction is assumed to be parallel to the normal direction.
When this parameter is set to 0 or a value outside the valid range, it is set to 1 degree.

12323	Number of a manual handle used for 3-dimensional machining manual feed

[Input type] Setting input
[Data type] Byte path
[Valid data range] 0 to 5

When 3-dimensional machining manual feed (handle feed) is performed, set the number of the manual handle to be used.
When the fourth or fifth manual handle is used for 3-dimensional machining manual feed, the option for manual handle feed with 4/5 handles is required.

If 0 or the number of an unavailable handle is set, the first handle is assumed.

> **NOTE**
> Don't change this parameter during 3-dimensional manual feed mode.

	#7	#6	#5	#4	#3	#2	#1	#0
13113					CFD			CLR

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

#0 CLR Upon reset, the display of a travel distance by 3-dimensional machining manual feed is:
0: Not cleared.
1: Cleared.

#3 CFD As feedrate F, the 3-dimensional machining manual feed screen displays:
0: Composite feedrate at the linear axis/rotation axis control point.
1: Feedrate at the tool tip.

	#7	#6	#5	#4	#3	#2	#1	#0
13451							ATW	

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

#1 ATW When I, J, and K are all set to 0 in a block that specifies a feature coordinate system setup command (G68.2), which is a tilted working plane indexing:
0: An alarm PS5457, "G68.2 FORMAT ERROR" is issued.
1: A feature coordinate system with a tilted plane angle of 0 degrees is assumed for operation.

	#7	#6	#5	#4	#3	#2	#1	#0
13601								MPR

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 MPR The machining parameter adjustment screen is:
0: Displayed.
1: Not displayed.
Even if 1 is set in this parameter bit, the precision level selection screen for the machining condition selecting function and the precision level selection screens (machining quality level selection screen and the machining level selection screen) for the machining quality level adjustment function are displayed.

19581	Tolerance smoothing for nano smoothing

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

[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a tolerance value for a program created using miniature line segments in nano smoothing.
When is set in this parameter, this parameter setting value is regarded as 0.005mm.

19582	Minimum amount of travel of a block that makes a decision based on an angular difference between blocks for nano smoothing

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the minimum amount of travel of a block that makes a decision based on an angular difference between blocks for nano smoothing. A block that specifies an amount of travel less than the value set in this parameter makes no decision based on an angular difference.
When is set in this parameter, this parameter setting value is regarded as 0.5mm.
A value greater than the value set in parameter No. 8490 for making a decision based on the minimum travel distance of a block must be set.

	#7	#6	#5	#4	#3	#2	#1	#0
19594								ATC

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 ATC When G05.1 Q3 is specified:
0: Nano smoothing is effective.
1: Smart tolerance control is effective.

19595	Maximum block length for small line segments in smart tolerance control mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
Set maximum block length for regarding a linear block as a small line segment in smart tolerance control mode. If the block length is less than the value of this parameter, smoothing of small line segments is applied to the block.
When this parameter is set to 0 or less, it operates as 5mm is set.

19596	Tolerance for linear axis in smart tolerance control mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
Set tolerance for linear axes in smart tolerance control mode. If tolerance is not specified in tolerance mode, machining path is controlled so that the machining accuracy at corners is within the tolerance set to this parameter.
When this parameter is set to 0 or less, it operates as 10μm is set.

19599	Maximum allowable acceleration rate for the deceleration function based on acceleration in smart tolerance control mode for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec2, inch/sec2, degree/sec2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Feedrate is controlled so that acceleration produced by changing the move direction does not exceed the value specified to this parameter in tlerance control mode.
This parameter is valid for basic 3-axes (the axes that 1, 2, 3 are set in the parameter No.1022). This parameter of the other axes are invalid, and not necessary to be set.
For an axis with 0 set in this parameter, the value of No. 1737 is available for allowable acceleration rate. And for an axis with a negative value set in this parameter, the deceleration function based on acceleration in tolerance contorl mode is disabled.
If a different value is set to this parameter for each axis, a feedrate is determined from the smallest of the acceleration rates specified for the moving axes.

	#7	#6	#5	#4	#3	#2	#1	#0
19602			D3D					

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

#5 D3D Specifies whether to display the distance to go in the program coordinate system or workpiece coordinate system during the 3-dimensional coordinate system conversion or the tilted working plane indexing.
0: Display the distance to go in the program coordinate system.
1: Display the distance to go in the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
19607		NAA	CAV			CCC		

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

#2 CCC In the cutter compensation/tool nose radius compensation mode, the outer corner connection method is based on:
0: Linear connection type.
1: Circular connection type.

#5 CAV When an interference check finds that interference (overcutting) occurred:
0: Machining stops with the alarm PS0041, "INTERFERENCE IN CUTTER COMPENSATION". (Interference check alarm function)
1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)
For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

#6 NAA When the interference check avoidance function considers that an avoidance operation is dangerous or that a further interference to the interference avoidance vector occurs:
0: An alarm is issued.
When an avoidance operation is considered to be dangerous, the alarm PS5447, "DANGEROUS AVOIDANCE AT G41/G42" is issued.
When a further interference to the interference avoidance vector is considered to occur, the alarm PS5448, "INTERFERENCE TO AVD. AT G41/G42" is issued.
1: No alarm is issued, and the avoidance operation is continued.

> ⚠ **CAUTION**
> When this parameter is set to 1, the path may be shifted largely. Therefore, set this parameter to 0 unless special reasons are present.

19625	Number of blocks to be read in the cutter compensation/tool nose radius compensation mode

[Input type] Setting input
[Data type] Byte path
[Valid data range] 3 to 8

This parameter sets the number of blocks to be read in the cutter compensation/tool nose radius compensation mode. When a value less than 3 is set, the specification of 3 is assumed. When a value greater than 8 is set, the specification of 8 is assumed. As a greater number of blocks are read, an overcutting (interference) forecast can be made with a command farther ahead. However, the number of blocks read and analyzed increases, so that a longer block processing time becomes necessary.

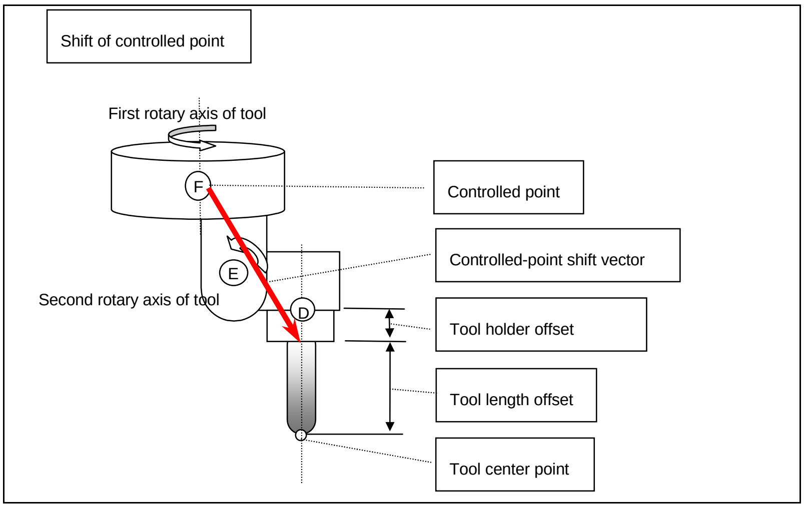
Even if the setting of this parameter is modified in the MDI mode by stopping in the cutter compensation/tool nose radius compensation mode, the setting does not become valid immediately. Before the new setting of this parameter can become valid, the cutter compensation/tool noise radius compensation mode must be canceled, then the mode must be entered again.

	#7	#6	#5	#4	#3	#2	#1	#0
19665			SVC	SPR				

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

#4 SPR The controlled point is shifted by:
0: Automatic calculation.
1: Using parameter No. 19667.

Bit 5 (SVC) of parameter No. 19665	Bit 4 (SPR) of parameter No. 19665	Shift of controlled point
0	-	Shift is not performed as not done conventionally.
1	0	The controlled point is shifted according to the result of the following automatic calculation: - (Intersection offset vector between the tool axis and the first rotation axis of the tool + intersection offset vector between the second and first rotation axes of the tool + tool holder offset (parameter No. 19666)) (See the figure next.)
1	1	The controlled point is shifted. As the shift vector, the vector set in parameter No. 19667 is used.



[Controlled-point shift vector when automatically calculated]

#5 SVC The controlled point is:
0: Not shifted.
1: Shifted.
The method of shifting is specified by bit 4 (SPR) of parameter No. 19665.

> **NOTE**
> When the machine has no rotation axis for rotating the tool (when parameter No. 19680 is set to 12 to specify the table rotation type), the controlled point is not shifted regardless of the setting of this parameter.

19666	Tool holder offset value

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set an offset value (tool holder offset value) specific to the machine from the control point to the tool attachment position in tool length compensation (after specification of G53.1 in the tilted working plane indexing mode), tool length compensation in tool axis direction, 3-dimensional manual feed. In tool length compensation (not in the tilted working plane indexing mode), however, tool holder offset can be enabled or disabled with bit 7 (ETH) of parameter No. 19665.

NOTE
Set a radius value.

19667	Controlled-point shift vector

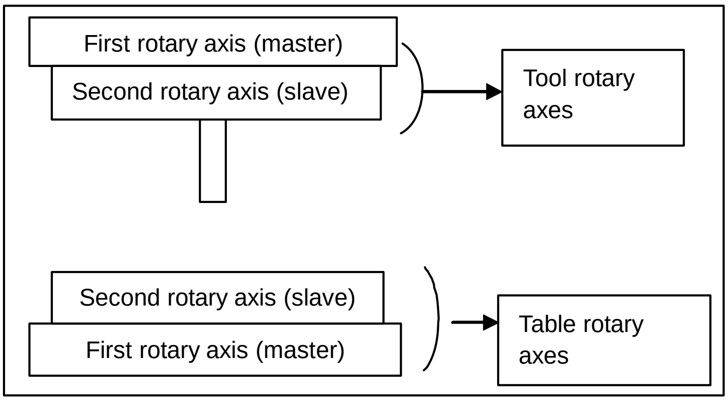
[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the shift vector for the controlled point. This value becomes valid when bit 5 (SVC) of parameter No. 19665 is set to 1, and bit 4 (SPR) of parameter No. 19665 is set to 1.

NOTE
Set a radius value.

19680	Mechanical unit type

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 21
Specify the type of the mechanical unit.

Parameter No. 19680	Mechanical unit type	Controlled rotation axis	Master and slave
0		Mechanism having no rotation axis	
2	Tool rotation type	Two rotation axes of the tool	The first rotation axis is the master, and the second rotation axis is the slave.
12	Table rotation type	Two rotation axes of the table	The first rotation axis is the master, and the second rotation axis is the slave.
21	Mixed type	One rotation axis of the tool + one rotation axis of the table	The first rotation axis is the tool rotation axis, and the second rotation axis is the table rotation axis.



> **NOTE**
> A hypothetical axis is also counted as a controlled rotary axis.
> <Hypothetical axis>
> In some cases, it is convenient to use an imaginary rotary axis whose angle is fixed to a certain value. For example, suppose that a tool is mounted in a tilted manner through an attachment. In such a case, the rotary axis considered hypothetically is a hypothetical axis. Bits 0 (IA1) and 1 (IA2) of parameter No. 19696 determine whether each rotary axis is an ordinary rotary axis or a hypothetical axis.

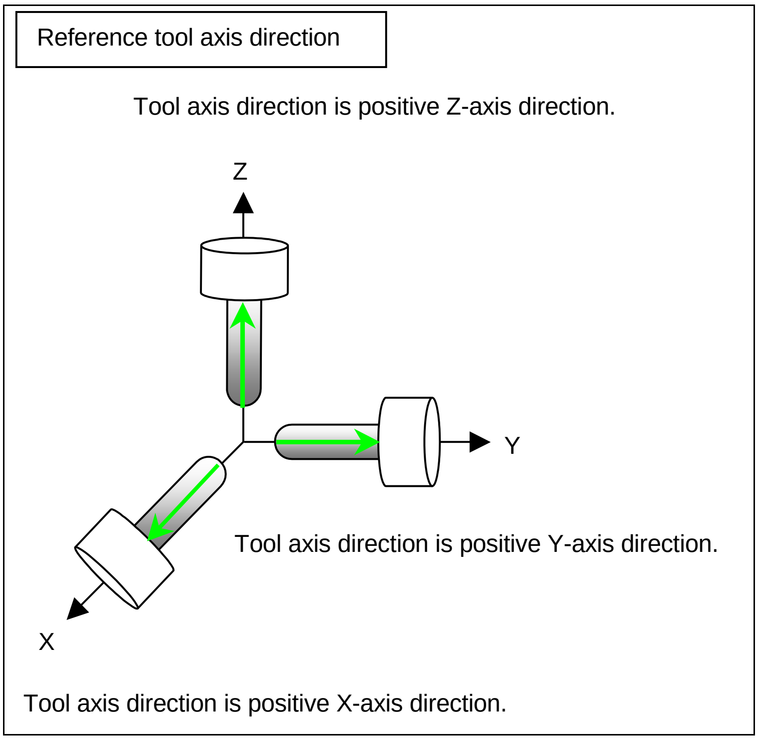
19697	Reference tool axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Set the tool axis direction in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees. Also, set the tool axis direction in the machine coordinate system in a mechanism in which only the rotation axes for controlling the table are present (there is no rotation axis for controlling the tool).

1: Positive X-axis direction
2: Positive Y-axis direction
3: Positive Z-axis direction

When the reference tool axis direction is neither the X-, Y-, nor Z-axis direction, set the reference direction in this parameter, then set appropriate angles as the reference angle RA and reference angle RB (parameter Nos. 19698 and 19699).



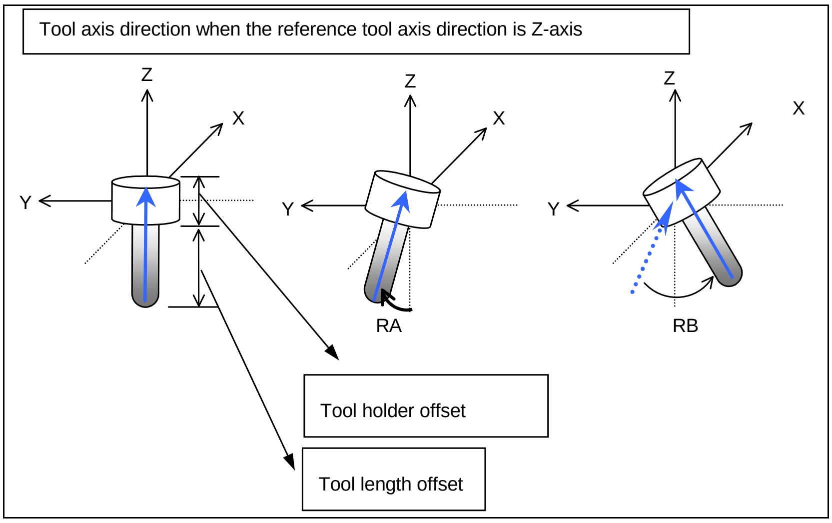
19698	Angle when the reference tool axis direction is tilted (reference angle RA)
19699	Angle when the reference tool axis direction is tilted (reference angle RB)

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)

When the reference tool axis direction (parameter No. 19697) is set to 1, the tool axis is tilted the RA degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction, then the tool axis is tilted the RB degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction.

When the reference tool axis direction (parameter No. 19697) is set to 2, the tool axis is tilted the RA degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction, then the tool axis is tilted the RB degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction.

When the reference tool axis direction (parameter No. 19697) is set to 3, the tool axis is tilted the RA degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction, then the tool axis is tilted the RB degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction.



	#7	#6	#5	#4	#3	#2	#1	#0
19746					LOZ	LOD		

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

#2 LOD As the tool length for 3-dimensional machining manual feed:
0: The value of parameter No. 12318 is used.
1: The tool length currently used for tool length compensation is used.

#3 LOZ When bit 2 (LOD) of parameter No. 19746 is set to 1 and tool length compensation is not applied, as the tool length for 3-dimensional machining manual feed:
0: The value of parameter No. 12318 is used.
1: 0 is used.

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.
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 (Common to Lathe System / Machining Center System.

(A) Length and angle parameters (type 1)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
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	0.00 to +999000.00
	IS-B	0.001	0.000 to +999000.000
	IS-C	0.0001	0.0000 to +99999.9999
inch/min	IS-A	0.001	0.000 to +96000.000
	IS-B	0.0001	0.0000 to +9600.0000
	IS-C	0.00001	0.00000 to +4000.00000

(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

B LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND

B.1 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE

The function including address P in the argument of G code is shown below.

Function name	Machining center system	Lathe system G code system A	Lathe system G code system B	Lathe system G code system C	Reference item
Dwell	G04	G04	G04	G04	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "FEED FUNCTIONS"
G code preventing buffering	G04.1	G04.1	G04.1	G04.1	- CONNECTION MANUAL (FUNCTION) "PROGRAM COMMAND" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "HIGH-SPEED CUTTING FUNCTIONS"
AI advanced preview control(M Series)/AI contour control (advanced preview control compatible command)	G08	G08	G08	G08	- CONNECTION MANUAL (FUNCTION) "FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "HIGH-SPEED CUTTING FUNCTIONS"
Programmable data input	G10	G10	G10	G10	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "PROGRAMMABLE DATA INPUT (G10)"
Programmable parameter input	G10	G10	G10	G10	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "PROGRAMMABLE DATA INPUT (G10)"
Spindle speed fluctuation detection	-	G26	G26	G26	- CONNECTION MANUAL (FUNCTION) "SPINDLE SPEED FUNCTION" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "SPINDLE SPEED FUNCTION"
Reference position return	G30	G30	G30	G30	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "REFERENCE POSITION"
Multi-step skip	G31	G31	G31	G31	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "INTERPOLATION FUNCTIONS"
Torque limit skip	G31	G31	G31	G31	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "INTERPOLATION FUNCTIONS"
Continuous high-speed skip	G31	G31	G31	G31	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "INTERPOLATION FUNCTIONS"
Skip function for EGB axis	G31.8	-	-	-	CONNECTION MANUAL (FUNCTION) "AXIS CONTROL"

Function name	Machining center system	Lathe system G code system A	Lathe system G code system B	Lathe system G code system C	Reference item
Scaling	G51	-	-	-	- CONNECTION MANUAL (FUNCTION) "PROGRAM COMMAND" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "COMPENSATION FUNCTION"
Polygon turning	-	G51.2 (G251)	G51.2 (G251)	G51.2 (G251)	- CONNECTION MANUAL (FUNCTION) "INTERPOLATION FUNCTION" - OPERATOR'S MANUAL (For T Series) II. PROGRAMMING, "AXIS CONTROL FUNCTIONS"
Synchronous, Composite and Superimposed control by program command	G51.4, G51.5, G51.6, G50.5	G51.4, G51.5, G51.6, G50.5	G51.4, G51.5, G51.6, G50.5	G51.4, G51.5, G51.6, G50.5	- CONNECTION MANUAL (FUNCTION) "MULTI-PATH CONTROL" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "MULTI-PATH CONTROL FUNCTION"
Flexible path axis assignment	G52.1, G52.2, G52.3	G52.1, G52.2, G52.3	G52.1, G52.2, G52.3	G52.1, G52.2, G52.3	CONNECTION MANUAL (FUNCTION) "AXIS CONTROL"
High-speed G53 function	G53	G53	G53	G53	- CONNECTION MANUAL (FUNCTION) "AXIS CONTROL" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "COORDINATE SYSTEM"
Workpiece coordinate system	G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	- CONNECTION MANUAL (FUNCTION) "AXIS CONTROL" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "COORDINATE SYSTEM"
Custom macro	G65, G66, G66.1	G65, G66, G66.1	G65, G66, G66.1	G65, G66, G66.1	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "CUSTOM MACRO"
Execution macro Note1)	G65, G66, G66.1 Note1)	G65, G66, G66.1 Note1)	G65, G66, G66.1 Note1)	G65, G66, G66.1 Note1)	Macro Executor PROGRAMMING MANUAL "EXECUTION MACRO FUNCTION"
Pattern Data Input	G65	G65	G65	G65	- CONNECTION MANUAL (FUNCTION) "PROGRAM COMMAND" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "PATTERN DATA INPUT"
Balanced cutting	-	G68	G68	G68	- CONNECTION MANUAL (FUNCTION) "MULTI-PATH CONTROL" - OPERATOR'S MANUAL (For T Series) II. PROGRAMMING, "BALANCE CUT"
Tilted working plane indexing	G68.2	-	-	-	- CONNECTION MANUAL (FUNCTION) "PROGRAM COMMAND" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"

Function name	Machining center system	Lathe system G code system A	Lathe system G code system B	Lathe system G code system C	Reference item
Tilted working plane indexing by tool axis direction	G68.3,	-	-	-	- CONNECTION MANUAL (FUNCTION) "PROGRAM COMMAND" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Tilted working plane indexing (incremental multi-command)	G68.4	-	-	-	- CONNECTION MANUAL (FUNCTION) "PROGRAM COMMAND" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Multiple repetitive cycles	-	G70 to G76	G70 to G76	G72 to G78	- CONNECTION MANUAL (FUNCTION) "PROGRAM COMMAND" - OPERATOR'S MANUAL (For T Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Figure copying	G72.1, G72.2	-	-	-	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Canned cycle	G74, G76	-	-	-	OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Canned cycle for grinding	G75, G77, G78, G79	G72, G74	G72, G74	G73, G75	- OPERATOR'S MANUAL (For T Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Canned cycle	G82 to G84, G87 to G89	G82 to G85, G87 to G89, G83.5, G83.6, G87.5, G87.6	G82 to G85, G87 to G89, G83.5, G83.6, G87.5, G87.6	G82 to G85, G87 to G89, G83.5, G83.6, G87.5, G87.6	- OPERATOR'S MANUAL (For T Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING" and "MEMORY OPERATION USING Series 10/11 FORMAT" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Electronic gear box	G81	-	-	-	CONNECTION MANUAL (FUNCTION) "AXIS CONTROL"
High-speed peck drilling cycle	-	G83.1	G83.1	G83.1	OPERATOR'S MANUAL (For T Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Rigid tapping cycle (FS10/11 format)	G84.2	G84.2	G84.2	G84.2	- CONNECTION MANUAL (FUNCTION) "SPINDLE SPEED FUNCTION" - OPERATOR'S MANUAL (For T Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"

Function name	Machining center system	Lathe system G code system A	B	C	Reference item
Left-handed rigid tapping cycle (FS10/11 format)	G84.3	-	-	-	- CONNECTION MANUAL (FUNCTION) "SPINDLE SPEED FUNCTION" - OPERATOR'S MANUAL (For M Series) II. PROGRAMMING, "FUNCTIONS TO SIMPLIFY PROGRAMMING"
Constant surface speed control	G96	G96	G96	G96	- CONNECTION MANUAL (FUNCTION) "SPINDLE SPEED FUNCTION" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "SPINDLE SPEED FUNCTION"
Spindle Indexing Function	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	- CONNECTION MANUAL (FUNCTION) "SPINDLE SPEED FUNCTION" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "SPINDLE SPEED FUNCTION"
Spindle control with servo motor	G96.4	G96.4	G96.4	G96.4	- CONNECTION MANUAL (FUNCTION) "SPINDLE SPEED FUNCTION" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "SPINDLE SPEED FUNCTION"

Note 1) : Arbitrary G code is optional with the following compilation parameter.
- No.9013 to No.9022, No.9034, No.9045 to No.9047, No.9129 to No.9137

B.2 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE

The function including address P in the argument of M and S code is shown below.

Function name	M code format	related parameters	Reference item
Waiting M codes	M_P_	No.8110, No.8111, MWP(No.8103#1)	- CONNECTION MANUAL (FUNCTION) "MULTI-PATH CONTROL" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "MULTI-PATH CONTROL FUNCTION"
Waiting M codes of high-speed Type	M_P_	No.8114, No.8115, MWP(No.8103#1)	- CONNECTION MANUAL (FUNCTION) "MULTI-PATH CONTROL" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "MULTI-PATH CONTROL FUNCTION"
Waiting function by specifying start point	M_P_L_IP	STW(No.8101#1), No.8110, No.8111, MWP(No.8103#1)	- CONNECTION MANUAL (FUNCTION) "MULTI-PATH CONTROL" - OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "MULTI-PATH CONTROL FUNCTION"
(Custom macro) Subprogram	M98P_		OPERATOR'S MANUAL (Common to T/M Series) III. PROGRAMMING, "PROGRAM CONFIGURATION"
Program end	M99P_		
External subprogram call	M198P_	SBP(No.3404#2)	OPERATOR'S MANUAL (Common to T/M Series) III. OPERATION, "AUTOMATIC OPERATION"
Custom macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_	MAA(No.6009#2)	OPERATOR'S MANUAL (Common to T/M Series) II. PROGRAMMING, "CUSTOM MACRO"
Execution macro Subprogram call	M98P_		Macro Executor PROGRAMMING MANUAL "EXECUTION MACRO FUNCTION"
Subprogram call for user program	M_P_		
Execution macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_		
Multi-spindle	S_P_	MPP(No.3703#3), No.3781	CONNECTION MANUAL (FUNCTION) "SPINDLE SPEED FUNCTION"

INDEX

<V>

<W>

REVISION RECORD

Edition	Date	Contents
01	Sep., 2014	

B-64604EN-2/01

* B - 6 4 6 0 4 E N - 2 / 0 1 *