FANUC Series 30i/300i/300is-MODEL A FANUC Series 31i/310i/310is-MODEL A5 FANUC Series 31i/310i/310is-MODEL A FANUC Series 32i/320i/320is-MODEL A

# For Lathe System USER'S MANUAL

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

The export of this product is subject to the authorization of the government of the country from where the product is exported.

In this manual we have tried as much as possible to describe all the various matters.

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

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

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

## **SAFETY PRECAUTIONS**

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

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

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

#### **CONTENTS**

1.1	DEFINITION OF WARNING, CAUTION, AND NOTE	s-2
1.2	GENERAL WARNINGS AND CAUTIONS	s-3
1.3	WARNINGS AND CAUTIONS RELATED TO	
	PROGRAMMING	s-6
1.4	WARNINGS AND CAUTIONS RELATED TO HANDLIN	IGs-9
1 5	WARNINGS RELATED TO DAILY MAINTENANCE	s-12

#### 1.1 **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.

## 1.2 GENERAL WARNINGS AND CAUTIONS

#### **⚠** WARNING

- Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 2 Before operating the machine, thoroughly check the entered data.
  - Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 3 Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate.
  - If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- 4 When using a tool compensation function, thoroughly check the direction and amount of compensation.
  - Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

#### **⚠** WARNING

- 5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
  - Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
- Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit.
  - Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.
- 7 The User's Manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.
- 8 Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.



#### **↑** CAUTION

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

#### NOTE

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

Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery. To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

# 1.3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

#### **⚠ WARNING**

#### 1 Coordinate system setting

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

#### 2 Positioning by nonlinear interpolation

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

#### 3 Function involving a rotation axis

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

#### 4 Inch/metric conversion

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

#### **. WARNING**

#### 5 Constant surface speed control

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

#### 6 Stroke check

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

#### 7 Tool post interference check

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

#### 8 Absolute/incremental mode

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

#### 9 Plane selection

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

#### 10 Torque limit skip

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

### **⚠ WARNING**

#### 11 Programmable mirror image

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

#### 12 Compensation function

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

# 1.4 WARNINGS AND CAUTIONS RELATED TO HANDLING

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

#### **⚠ WARNING**

#### 1 Manual operation

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

#### 2 Manual reference position return

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

#### 3 Manual 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.

#### **⚠** WARNING

#### 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.

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

#### 9 RESET kev

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

#### **. WARNING**

#### 10 Manual intervention

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

### 11 Feed hold, override, and single block

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

#### 12 Dry run

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

# 13 Cutter and tool nose radius compensation in MDI mode

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

#### 14 Program editing

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

#### 1.5 WARNINGS RELATED TO DAILY MAINTENANCE

#### **.↑** WARNING

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 \( \Delta \) and fitted with an insulating cover).

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

#### NOTE

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

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

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost. Refer to the Section "Method of replacing battery" in the User's Manual (Common to T/M series) for details of the battery replacement procedure.

#### **.** ₩ARNING

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 \( \Delta \) 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  $\alpha i$  series Maintenance Manual for details of the battery replacement procedure.

### **⚠ WARNING**

#### 3 Fuse replacement

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

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

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

## **TABLE OF CONTENTS**

SA	FETY	PREC	AUTIONS	s-1
I. (	GENE	RAL		
1	GEN	IERAL .		3
	1.1	NOTE	ES ON READING THIS MANUAL	7
	1.2	NOTE	ES ON VARIOUS KINDS OF DATA	7
II.	PROC	GRAM	MING	
1	GEN	IERAL .		11
	1.1	OFFS	SET	12
2	PRE	PARA	TORY FUNCTION (G FUNCTION)	13
3	INTE	RPOL	ATION FUNCTION	19
	3.1	CONS	STANT LEAD THREADING (G32)	20
	3.2		TINUOUS THREADING	
	3.3	MULT	ΓIPLE THREADING	25
	3.4	TORC	QUE LIMIT SKIP (G31 P99)	27
4	FUN	CTION	S TO SIMPLIFY PROGRAMMING	29
	4.1	CANN	NED CYCLE (G90, G92, G94)	30
		4.1.1	Outer Diameter/Internal Diameter Cutting Cycle (G90)	31
			4.1.1.1 Straight cutting cycle	
			4.1.1.2 Taper cutting cycle	
		4.1.2	Threading Cycle (G92)	
			4.1.2.1 Straight threading cycle	
		4.1.3	End Face Turning Cycle (G94)	
		7.1.3	4.1.3.1 Face cutting cycle	
			4.1.3.2 Taper cutting cycle	
		4.1.4	How to Use Canned Cycles (G90, G92, G94)	45
		4.1.5	Canned Cycle and Tool Nose Radius Compensation	47
		4.1.6	Restrictions on Canned Cycles	49
	4.2	MULT	TIPLE REPETITIVE CYCLE (G70-G76)	51
		4.2.1	Stock Removal in Turning (G71)	
		4.2.2	Stock Removal in Facing (G72)	65
		423	Pattern Reneating (G73)	70

		4.2.4	Finishing Cycle (G70)	73
		4.2.5	End Face Peck Drilling Cycle (G74)	77
		4.2.6	Outer Diameter / Internal Diameter Drilling Cycle (G75)	79
		4.2.7	Multiple Threading Cycle (G76)	81
		4.2.8	Restrictions on Multiple Repetitive Cycle (G70-G76)	88
	4.3	CANN	IED CYCLE FOR DRILLING	90
		4.3.1	Front Drilling Cycle (G83)/Side Drilling Cycle (G87)	94
		4.3.2	Front Tapping Cycle (G84) / Side Tapping Cycle (G88)	97
		4.3.3	Front Boring Cycle (G85) / Side Boring Cycle (G89)	99
		4.3.4	Canned Cycle for Drilling Cancel (G80)	100
		4.3.5	Precautions to Be Taken by Operator	101
	4.4	RIGID	) TAPPING	102
		4.4.1	FRONT FACE RIGID TAPPING CYCLE (G84) /	
			SIDE FACE RIGID TAPPING CYCLE (G88)	103
		4.4.2	Peck Rigid Tapping Cycle (G84 or G88)	109
		4.4.3	Canned Cycle Cancel (G80)	114
		4.4.4	Override during Rigid Tapping	115
			4.4.4.1 Extraction override	
			4.4.4.2 Override signal	
	4.5		MFERING AND CORNER R	
	4.6		OR IMAGE FOR DOUBLE TURRET (G68, G69)	
	4.7	DIRE	CT DRAWING DIMENSION PROGRAMMING	128
5	CON	<b>IPENS</b>	ATION FUNCTION	134
	5.1	TOOL	OFFSET	135
		5.1.1	Tool Geometry Offset and Tool Wear Offset	136
		5.1.2	T Code for Tool Offset	137
		5.1.3	Tool Selection	137
		5.1.4	Offset Number	137
		5.1.5	Offset	138
		5.1.6	Y Axis Offset	142
		5.1.7	Second Geometry Tool Offset	143
	5.2	OVER	RVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)	146
		5.2.1	Imaginary Tool Nose	147
		5.2.2	Direction of Imaginary Tool Nose	149
		5.2.3	Offset Number and Offset Value	151
		5.2.4	Workpiece Position and Move Command	154
		5.2.5	Notes on Tool Nose Radius Compensation	161

	5.3	OVER\	VIEW OF CUTTER COMPENSATION (G40-G42)	. 164
	5.4	DETAIL	LS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION	. 171
		5.4.1	Overview	171
		5.4.2	Tool Movement in Start-up	175
		5.4.3	Tool Movement in Offset Mode	181
		5.4.4	Tool Movement in Offset Mode Cancel	202
		5.4.5	Prevention of Overcutting Due to Cutter or Tool Nose Radius Compensation	209
		5.4.6	Interference Check	213
			5.4.6.1 Operation to be performed if an interference is judged to occur	217
			5.4.6.2 Interference check alarm function	
			5.4.6.3 Interference check avoidance function	219
		5.4.7	Cutter or Tool Nose Radius Compensation for Input from MDI	225
	5.5	VECTO	OR RETENTION (G38)	. 227
	5.6	CORNE	ER CIRCULAR INTERPOLATION (G39)	. 228
	5.7	EXTEN	IDED TOOL SELECTION	. 231
	5.8	AUTON	MATIC TOOL OFFSET (G36, G37)	. 235
	5.9		DINATE SYSTEM ROTATION (G68.1, G69.1)	
	5.10		E OFFSET VALUE CHANGE FUNCTION BASED ON MANUAL	
		FEED.		.243
2	MEM		DEDATION BY SEDIES 15 FORMAT	
6		ORY OI	PERATION BY SERIES 15 FORMAT	
6	<b>MEM</b> (6.1	ORY OI Addre	PERATION BY SERIES 15 FORMAT	247
6	6.1	ORY OI ADDRE PROGE	PERATION BY SERIES 15 FORMAT ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15 RAM FORMAT	<b>247</b> . 248
6	6.1	ORY OI ADDRE PROGE SUBPE	PERATION BY SERIES 15 FORMAT ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15 RAM FORMAT ROGRAM CALLING	. 248 . 249
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING	. 248 . 249 . 250
ô	6.1	ORY OI ADDRE PROGE SUBPE	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  ED CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)	. 248 . 249 . 250 251
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  D CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle	. 248 . 249 . 250 251
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE 6.3.1	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  ED CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle	.248 .249 .250 251 253
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  D CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)	. 248 . 249 . 250 251 253 255
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE 6.3.1	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  D CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle	.248 .249 .250 251 253 255
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE 6.3.1	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle	. 248 . 249 . 250 251 253 255 255
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE 6.3.1	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle  End Face Turning Cycle (G94)	. 248 . 249 . 250 251 253 255 255 259
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE 6.3.1	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  ED CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle  End Face Turning Cycle (G94)  6.3.3.1 Face cutting cycle	. 248 . 249 . 250 251 253 255 259 262
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE 6.3.1	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  D CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle  End Face Turning Cycle (G94)  6.3.3.1 Face cutting cycle  6.3.3.2 Taper cutting cycle  6.3.3.2 Taper cutting cycle	. 248 . 249 . 250 251 253 255 255 262 262
6	6.1	ORY OI ADDRE PROGE SUBPE CANNE 6.3.1 6.3.2	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  D CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle  End Face Turning Cycle (G94)  6.3.3.1 Face cutting cycle  6.3.3.2 Taper cutting cycle  How to Use Canned Cycles	. 248 . 249 . 250 251 253 255 255 262 262 264 266
6	6.1	ADDRE PROGE SUBPE CANNE 6.3.1 6.3.2 6.3.3 6.3.4 6.3.5	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  ED CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle  End Face Turning Cycle (G94)  6.3.3.1 Face cutting cycle  6.3.3.2 Taper cutting cycle  How to Use Canned Cycles  Canned Cycle and Tool Nose Radius Compensation	. 248 . 249 . 250 251 255 255 259 262 264 266
6	<ul><li>6.1</li><li>6.2</li><li>6.3</li></ul>	ORY OI  ADDRE PROGE SUBPE CANNE 6.3.1  6.3.2  6.3.3	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  COGRAM CALLING  D CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle  End Face Turning Cycle (G94)  6.3.3.1 Face cutting cycle  6.3.3.2 Taper cutting cycle  How to Use Canned Cycles  Canned Cycle and Tool Nose Radius Compensation  Restrictions on Canned Cycles	. 248 . 249 . 250 251 253 255 255 262 264 264 268 270
6	6.1	ORY OI  ADDRE PROGE SUBPE CANNE 6.3.1  6.3.2  6.3.3	PERATION BY SERIES 15 FORMAT  ESSES AND SPECIFIABLE VALUE RANGE FOR SERIES 15  RAM FORMAT  ROGRAM CALLING  ED CYCLE  Outer Diameter/Internal Diameter Cutting Cycle (G90)  6.3.1.1 Straight cutting cycle  6.3.1.2 Taper cutting cycle  Threading Cycle (G92)  6.3.2.1 Straight threading cycle  6.3.2.2 Taper threading cycle  End Face Turning Cycle (G94)  6.3.3.1 Face cutting cycle  6.3.3.2 Taper cutting cycle  How to Use Canned Cycles  Canned Cycle and Tool Nose Radius Compensation	. 248 . 249 . 250251253255259262264268270 . 272

		6.4.2	Stock Removal in Facing (G72)	288
		6.4.3	Pattern Repeating (G73)	294
		6.4.4	Finishing Cycle (G70)	297
		6.4.5	End Face Peck Drilling Cycle (G74)	301
		6.4.6	Outer Diameter / Internal Diameter Drilling Cycle (G75)	303
		6.4.7	Multiple Threading Cycle (G76 <g a="" b="" code="" system="">)</g>	
			(G78 <g c="" code="" system="">)</g>	305
		6.4.8	Restrictions on Multiple Repetitive Cycle	313
	6.5	CANN	NED CYCLE FOR DRILLING	315
		6.5.1	High-speed Peck Drilling Cycle (G83.1)	321
		6.5.2	Drilling Cycle, Spot Drilling Cycle (G81)	323
		6.5.3	Drilling Cycle, Counter Boring (G82)	325
		6.5.4	Peck Drilling Cycle (G83)	327
		6.5.5	Tapping Cycle (G84)	329
		6.5.6	Boring Cycle (G85)	331
		6.5.7	Boring Cycle (G85)	333
		6.5.8	Canned Cycle for Drilling Cancel (G80)	335
		6.5.9	Precautions to be Taken by Operator	335
7	MUIT	I-PATI	H CONTROL FUNCTION	336
	7.1		NCE CUT (G68, G69)	
	ODE	RATIC		
	_			- 4-
1			JT/OUTPUT	
	1.1	INPU	T/OUTPUT ON EACH SCREEN	
		1.1.1	Inputting and Outputting Y-axis Offset Data	
			1.1.1.1 Inputting Y-axis offset data	
		1 1 2	1.1.1.2 Outputting Y-axis Offset Data	
		1.1.2	Inputting and Outputting Tool Offset / 2nd Geometry Data	
			1.1.2.2 Outputting tool offset / 2nd geometry data	
	1.2	INPU <sup>-</sup>	T/OUTPUT ON THE ALL IO SCREEN	
		1.2.1	Inputting and Outputting Y-axis Offset Data	
		1.2.2	Inputting and Outputting Tool Offset / 2nd Geometry Data	
2	SET		ND DISPLAYING DATA	
_	2.1		EENS DISPLAYED BY FUNCTION KEY	
	۷.۱	2.1.1	Setting and Displaying the Tool Offset Value	
		2.1.2	Direct Input of Tool Offset Value	

		2.1.3	Input of Tool Offset Value Measured B	364
		2.1.4	Counter Input of Offset value	367
		2.1.5	Setting the Workpiece Coordinate System Shift Value	368
		2.1.6	Setting Tool Compensation/Second Geometry Offset Values	371
		2.1.7	Setting the Y-Axis Offset	374
		2.1.8	Chuck and Tail Stock Barriers	377
ΑP	PENI	DIX		
Α	PAR	AMETE	ERS	387
	A.1	DESC	CRIPTION OF PARAMETERS	388
	A.2	DATA	TYPE	431
	A 3	STAN	IDARD PARAMETER SETTING TABLES	432

# 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 lathe system path control type. For other functions not specific to the lathe system, refer to the User's Manual (Common to Lathe System/Machining Center System) (B-63944EN).
- 2 Some functions described in this manual may not be applied to some products. For detail, refer to the DESCRIPTIONS manual (B-63942EN).
- 3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the parameter manual (B-63950EN).
  - Parameters are used to set functions and operating conditions of a CNC machine tool, and frequently-used values in advance. Usually, the machine tool builder factory-sets parameters so that the user can use the machine tool easily.
- 4 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

#### Applicable models

The models covered by this manual, and their abbreviations are:

Model name	Abbreviation		
FANUC Series 30 <i>i</i> -MODEL A	30 <i>i</i> –A	Series 30i	
FANUC Series 300i-MODEL A	300 <i>i</i> –A	Series 300i	
FANUC Series 300is-MODEL A	300is-A	Series 300is	
FANUC Series 31 <i>i</i> -MODEL A	31 <i>i</i> –A	Series 31 <i>i</i>	
FANUC Series 31 <i>i</i> -MODEL A5	31 <i>i</i> –A5	Selles 311	
FANUC Series 310 <i>i</i> -MODEL A	310 <i>i</i> –A	Series 310i	
FANUC Series 310 <i>i</i> -MODEL A5	310 <i>i</i> –A5	Selles 3 lui	
FANUC Series 310 <i>i</i> s-MODEL A	310 <i>i</i> s–A	Series 310is	
FANUC Series 310 <i>i</i> s-MODEL A5	310 <i>i</i> s–A5	Selies 5 lois	
FANUC Series 32i-MODEL A	32 <i>i</i> –A	Series 32i	
FANUC Series 320 <i>i</i> -MODEL A	320 <i>i</i> –A	Series 320i	
FANUC Series 320is-MODEL A	320is-A	Series 320is	

#### NOTE

- 1 Unless otherwise noted, the model names 31*i*/310*i*/310*i*s-A, 31*i*/310*i*/310*i*s-A5, and 32*i*/320*i*/320*i*s-A are collectively referred to as 30*i*/300*i*/300*i*s. However, this convention is not necessarily observed when item 3 below is applicable.
- 2 Some functions described in this manual may not be applied to some products. For details, refer to the DESCRIPTIONS (B-63942EN).

### **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 30i/300i/300is- MODEL A

Series 31*i*/310*i*/310*i*s- MODEL A

Series 31i/310i/310is- MODEL A5

Series 32*i*/320*i*/320*i*s- MODEL A

The following table lists the manuals related to Series 30i/300i/300is-A, Series 31i/310i/310is-A, Series 31i/310i/310is-A5, Series 32i/320i/320is-A. This manual is indicated by an asterisk(\*).

Table 1 Related manuals

Manual name Specification					
	number				
DESCRIPTIONS	B-63942EN				
CONNECTION MANUAL (HARDWARE)	B-63943EN				
CONNECTION MANUAL (FUNCTION)	B-63943EN-1				
USER'S MANUAL	B-63944EN				
(Common to Lathe System/Machining Center System)					
USER'S MANUAL (For Lathe System)	B-63944EN-1	*			
USER'S MANUAL (For Lathe Machining Center System)	B-63944EN-2				
MAINTENANCE MANUAL	B-63945EN				
PARAMETER MANUAL	B-65950EN				
Programming					
Macro Compiler / Macro Executor PROGRAMMING	B-63943EN-2				
MANUAL					
Macro Compiler OPERATOR'S MANUAL	B-66264EN				
C Language Executor OPERATOR'S MANUAL	B-63944EN-3				
PMC					
PMC PROGRAMMING MANUAL	B-63983EN				
Network					
PROFIBUS-DP Board OPERATOR'S MANUAL	B-63994EN				
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN				
DeviceNet Board OPERATOR'S MANUAL	B-64044EN				
Operation guidance function					
MANUAL GUIDE i OPERATOR'S MANUAL	B-63874EN				
MANUAL GUIDE i Set-up Guidance	B-63874EN-1				
OPERATOR'S MANUAL					

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

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

**Table 2 Related manuals** 

Manual name	Specification number
FANUC AC SERVO MOTOR $lpha i$ s series	
FANUC AC SERVO MOTOR $lpha i$ series	B-65262EN
DESCRIPTIONS	
FANUC AC SPINDLE MOTOR $lpha i$ series	B-65272FN
DESCRIPTIONS	D-03272EN
FANUC AC SERVO MOTOR $\beta i$ s series	B-65302EN
DESCRIPTIONS	D-00302EIN
FANUC AC SPINDLE MOTOR $eta i$ series	B-65312EN
DESCRIPTIONS	D-05512EN
FANUC SERVO AMPLIFIER $lpha i$ series	B-65282EN
DESCRIPTIONS	D-03202EN
FANUC SERVO AMPLIFIER $\beta i$ series	B-65322EN
DESCRIPTIONS	D-05522EIN
FANUC SERVO MOTOR $lpha i$ s series	
FANUC SERVO MOTOR $lpha i$ series	
FANUC AC SPINDLE MOTOR $lpha i$ series	B-65285EN
FANUC SERVO AMPLIFIER $\alpha i$ series	
MAINTENANCE MANUAL	
FANUC SERVO MOTOR $\beta i$ s series	
FANUC AC SPINDLE MOTOR $eta i$ series	B-65325EN
FANUC SERVO AMPLIFIER $\beta i$ series	D 00020LIV
MAINTENANCE MANUAL	
FANUC AC SERVO MOTOR α <i>i</i> s series	
FANUC AC SERVO MOTOR $\alpha i$ series	B-65270EN
FANUC AC SERVO MOTOR $\beta i$ s series	5 5027 5211
PARAMETER MANUAL	
FANUC AC SPINDLE MOTOR $\alpha i$ series	
FANUC AC SPINDLE MOTOR $\beta i$ series	B-65280EN
PARAMETER MANUAL	

Any of the servo motors and spindles listed above can be connected to the CNC described in this manual. However,  $\alpha i$  series servo amplifiers can only be connected to  $\alpha i$  series SVMs (for 30i/31i/32i).

This manual mainly assumes that the FANUC SERVO MOTOR  $\alpha i$  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 NOTES ON READING THIS MANUAL

### **⚠** CAUTION

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

## 1.2 NOTES ON VARIOUS KINDS OF DATA

#### **♠** CAUTION

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

# II. PROGRAMMING

# 1

## **GENERAL**

## 1.1 OFFSET

#### **Explanation**

#### - Tool offset

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.

Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (see "Setting and Displaying Data" in the User's Manual (Common to T/M series)), machining can be performed without altering the program even when the tool is changed. This function is called tool offset.

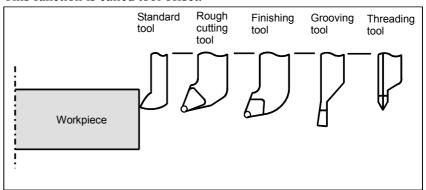


Fig. 1.1 (a) Tool offset

2

# PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.

G codes are divided into the following two types.

Туре	Meaning		
One-shot (+ 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 
$$X_{-}$$
;  $Z_{-}$ ;  $Z_{-}$ ; G01 is effective in this range.  $Z_{-}$ ;  $Z_{-}$ ; G00 is effective in this range.  $Z_{-}$ ;  $Z_{-}$ 

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

#### **Explanation**

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

	Table 2(a) G code list				
G code system			Group	Function	
Α	В	С	Отопр		
G00	G00	G00		Positioning (Rapid traverse)	
G01	G01	G01		Linear interpolation (Cutting feed)	
G02	G02	G02		Circular interpolation CW or helical interpolation CW	
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW	
G02.2	G02.2	G02.2	01	Involute interpolation CW	
G02.3	G02.3	G02.3	] "	Exponential interpolation CW	
G02.4	G02.4	G02.4		Three-dimensional coordinate conversion CW	
G03.2	G03.2	G03.2		Involute interpolation CCW	
G03.3	G03.3	G03.3		Exponential interpolation CCW	
G03.4	G03.4	G03.4		Three-dimensional coordinate conversion CCW	
G04	G04	G04		Dwell	
G05	G05	G05	00	Al contour control (command compatible with high precision contour control)	
G05.1	G05.1	G05.1		Al contour control / Nano smoothing / Smooth interpolation	
G05.4	G05.4	G05.4		HRV3,4 on/off	
G06.2	G06.2	G06.2	01	NURBS interpolation	
G07	G07	G07	<u> </u>	Hypothetical axis interpolation	
G07.1	G07.1	G07.1			
(G107)	(G107)	(G107)	00	Cylindrical interpolation	
G08	G08	G08		Advanced preview control	
G09	G09	G09		Exact stop	
G10	G10	G10		Programmable data input	
G10.6	G10.6	G10.6		Tool retract and recover	
G10.9	G10.9	G10.9		Programmable switching of diameter/radius specification	
G11	G11	G11		Programmable data input mode cancel	
G12.1	G12.1	G12.1	i		
(G112)	(G112)	(G112)		Polar coordinate interpolation mode	
G13.1	G13.1	G13.1	21	B	
(G113)	(G113)	(G113)		Polar coordinate interpolation cancel mode	
G15	G15	G15	6.1	Polar coordinate command cancel	
G16	G16	G16	24	Polar coordinate command	
G17	G17	G17	16	XpYp plane selection	
G18	G18	G18		ZpXp plane selection	
G19	G19	G19		YpZp plane selection	
G20	G20	G70	06	Input in inch	
G21	G21	G71	06	Input in mm	
G22	G22	G22	00	Stored stroke check function on	
G23	G23	G23	09	Stored stroke check function off	
G25	G25	G25	08	Spindle speed fluctuation detection off	
G26	G26	G26	00	Spindle speed fluctuation detection on	

Table 2(a) G code list

	Table 2(a) G code list				
	code syster		Group	Function	
Α	В	С			
G27	G27	G27	00	Reference position return check	
G28	G28	G28		Return to reference position	
G29	G29	G29		Movement from reference position	
G30	G30	G30		2nd, 3rd and 4th reference position return	
G30.1	G30.1	G30.1		Floating reference point return	
G31	G31	G31		Skip function	
G31.8	G31.8	G31.8		EGB-axis skip	
G32	G33	G33		Threading	
G34	G34	G34		Variable lead threading	
G35	G35	G35		Circular threading CW	
G36	G36	G36		Circular threading CCW (When the parameter G36 (No. 3405#3) is set to 1) or Automatic tool offset (X axis) (When the parameter G36 (No. 3405#3) is set to 0)	
G37	G37	G37	01	Automatic tool offset (Z axis) (When the parameter G36 (No. 3405#3) is set to 0)	
G37.1	G37.1	G37.1	01	Automatic tool offset (X axis) (When the parameter G36 (No. 3405#3) is set to 1)	
G37.2	G37.2	G37.2		Automatic tool offset (Z axis) (When the parameter G36 (No. 3405#3) is set to 1)	
G38	G38	G38		Cutter compensation of tool nose radius compensation: with vector held	
G39	G39	G39		Cutter compensation of tool nose radius compensation: corner rounding interpolation	
G40	G40	G40		Cutter compensation of tool nose radius compensation : cancel	
G41	G41	G41	07	Cutter compensation of tool nose radius compensation : left	
G42	G42	G42		Cutter compensation of tool nose radius compensation : right	
G41.2	G41.2	G41.2		Cutter compensation for 5-axis machining : left (type 1)	
				Cutter compensation for 5-axis machining :	
G41.3	G41.3	G41.3		(leading edge offset)	
G41.4	G41.4	G41.4		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)	
G41.5	G41.5	G41.5		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)	
G41.6	G41.6	G41.6		Cutter compensation for 5-axis machining : left (type 2)	
G42.2	G42.2	G42.2		Cutter compensation for 5-axis machining : right (type 1)	
G42.4	G42.4	G42.4		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)	
G42.5	G42.5	G42.5		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)	
G42.6	G42.6	G42.6		Cutter compensation for 5-axis machining : right (type 2)	
G43	G43	G43		Tool length compensation +	
	+ - · · ·			Tool length compensation -	
G44	G44	G44			
G44 G43.1	G44 G43.1	G44 G43.1		Tool length compensation in tool axis direction	
	•			-	
G43.1	G43.1	G43.1	23	Tool length compensation in tool axis direction	
G43.1 G43.4	G43.1 G43.4	G43.1 G43.4	23	Tool length compensation in tool axis direction Tool center point control (type 1) Tool center point control (type 2)	
G43.1 G43.4 G43.5	G43.1 G43.4 G43.5	G43.1 G43.4 G43.5	23	Tool length compensation in tool axis direction  Tool center point control (type 1)	
G43.1 G43.4 G43.5 G43.7	G43.1 G43.4 G43.5 G43.7	G43.1 G43.4 G43.5 G43.7	23	Tool length compensation in tool axis direction Tool center point control (type 1) Tool center point control (type 2)	

			Table 2(a) G code list				
G	G code system		Group	Function			
Α	В	С	Огоир	1 unction			
G50	G92	G92	00	Coordinate system setting or max. spindle speed clamp			
G50.3	G92.1	G92.1		Workpiece coordinate system preset			
-	G50	G50	18	Scaling cancel			
-	G51	G51		Scaling			
G50.1	G50.1	G50.1	22	Programmable mirror image cancel			
G51.1	G51.1	G51.1		Programmable mirror image			
G50.2	G50.2	G50.2		Polygon turning cancel			
(G250)	(G250)	(G250)	20	70 0			
G51.2	G51.2	G51.2		Polygon turning			
(G251)	(G251)	(G251)					
G52	G52	G52	00	Local coordinate system setting			
G53	G53	G53	00	Machine coordinate system setting			
G53.1	G53.1	G53.1		Tool axis direction control			
G54	G54	G54		Workpiece coordinate system 1 selection			
(G54.1)	(G54.1)	(G54.1)		Worknings coordinate system 2 colortion			
G55	G55	G55	4.4	Workpiece coordinate system 2 selection			
G56	G56	G56	14	Workpiece coordinate system 3 selection			
G57	G57	G57		Workpiece coordinate system 4 selection			
G58	G58	G58		Workpiece coordinate system 5 selection			
G59	G59	G59	00	Workpiece coordinate system 6 selection			
G60	G60	G60	00	Single direction positioning			
G61	G61	G61		Exact stop mode			
G62	G62	G62	15	Automatic corner override mode			
G63	G63	G63		Tapping mode			
G64	G64	G64	00	Cutting mode			
G65 G66	G65	G65 G66	00	Macro call			
	G66		12	Macro modal call A  Macro modal call B			
G66.1 G67	G66.1 G67	G66.1 G67					
			04	Macro modal call A/B cancel			
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode			
G68.1	G68.1	G68.1	17	Coordinate system rotation start or 3-dimensional coordinate conversion mode on			
G68.2	G68.2	G68.2		Feature coordinate system selection			
G69	G69	G69	04	Mirror image off for double turret or balance cutting mode cancel			
G69.1	G69.1	G69.1	17	Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off			
G70	G70	G72		Finishing cycle			
G71	G71	G73		Stock removal in turning			
G72	G72	G74		Stock removal in facing			
G73	G73	G75		Pattern repeating cycle			
G74	G74	G76	00	End face peck drilling cycle			
G75	G75	G77		Outer diameter/internal diameter drilling cycle			
G76	G76	G78		Multiple-thread cutting cycle			
G72.1	G72.1	G72.1		Figure copy (rotation copy)			
G72.2	G72.2	G72.2		Figure copy (parallel copy)			

Table 2(a) G code list

G code system			Group	Function
Α	В	С	Group	Fullction
G80	G80	G80	10	Canned cycle cancel for drilling
G80.5	G80.5	G80.5	27	Electronic gear box 2 pair: synchronization cancellation
G80.8	G80.8	G80.8	28	Electronic gear box: synchronization cancellation
G81	G81	G81	10	Spot drilling (FS15-T format)
G81.5	G81.5	G81.5	27	Electronic gear box 2 pair: synchronization start
G81.8	G81.81	G81.8	28	Electronic gear box: synchronization start
G82	G82	G82		Counter boring (FS15-T format)
G83	G83	G83		Cycle for face drilling
G83.1	G83.1	G83.1		High-speed peck drilling cycle (FS15-T format)
G83.5	G83.5	G83.5		High-speed peck drilling cycle
G83.6	G83.6	G83.6		Peck drilling cycle
G84	G84	G84		Cycle for face tapping
G84.2	G84.2	G84.2	10	Rigid tapping cycle (FS15-T format)
G85	G85	G85		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G87.5	G87.5	G87.5		High-speed peck drilling cycle
G87.6	G87.6	G87.6		Peck drilling cycle
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Threading cycle
G94	G79	G24		End face turning cycle
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G93	G93	G93		Inverse time feed
G98	G94	G94	05	Feed per minute
G99	G95	G95		Feed per revolution
-	G90	G90	03	Absolute programming
-	G91	G91	US	Incremental programming
-	G98	G98	11	Canned cycle : return to initial level
-	G99	G99	11	Canned cycle : return to R point level

# INTERPOLATION FUNCTION

# 3.1 CONSTANT LEAD THREADING (G32)

Tapered screws and scroll threads in addition to equal lead straight threads can be cut by using a G32 command.

The spindle speed is read from the position coder on the spindle in real time and converted to a cutting feedrate for feed-per minute mode, which is used to move the tool.

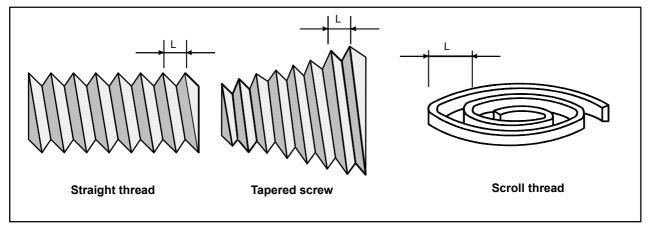


Fig. 3.1 (a) Thread types

#### **Format**

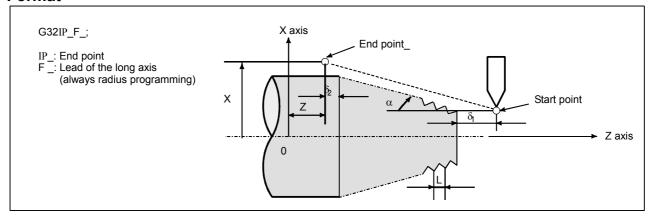


Fig. 3.1 (b) Example of threading

#### **Explanation**

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

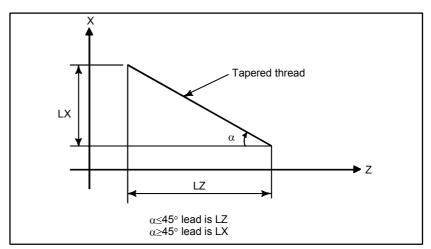


Fig. 3.1 (c) LZ and LX of a tapered thread

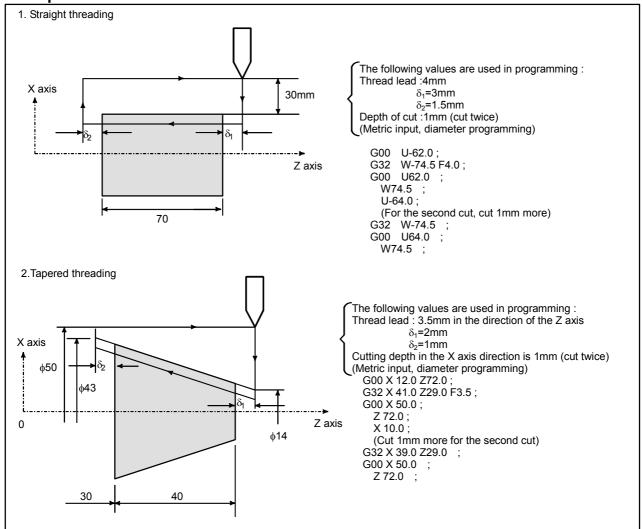
In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.

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

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

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

### **Example**



#### **↑** WARNING

- 1 Feedrate override is effective (fixed at 100%) during threading.
- 2 It is very dangerous to stop feeding the thread cutter without stopping the spindle. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while threading. If the feed hold button is pressed during threading, the tool will stop after a block not specifying threading is executed as if the SINGLE BLOCK button were pushed. However, the feed hold lamp (SPL lamp) lights when the FEED HOLD button on the machine control panel is pushed. Then, when the tool stops, the lamp is turned off (Single Block stop status).
- 3 When the FEED HOLD button is pressed again in the first block after threading mode that does not specify threading (or the button has been held down), the tool stops immediately at the block that does not specify threading.
- 4 When threading is executed in the single block status, the tool stops after execution of the first block not specifying threading.
- 5 When the mode was changed from automatic operation to manual operation during threading, the tool stops at the first block not specifying threading as when the feed hold button is pushed as mentioned in Warning 3. However, when the mode is changed from one automatic operation mode to another, the tool stops after execution of the block not specifying threading as for the single block mode in Note 4.
- 6 When the previous block was a threading block, cutting will start immediately without waiting for detection of the one-spindle-rotation signal even if the present block is a threading block.

G32Z F ;

(A 1-turn signal is not detected before this block.) Ζ;

(Regarded as threading block.) G32:

Z F; (One turn signal is also not detected.)

- 7 Because the constant surface speed control is effective during scroll thread or tapered screw cutting and the spindle speed changes, the correct thread lead may not be cut. Therefore, do not use the constant surface speed control during threading. Instead, use G97.
- 8 A movement block preceding the threading block must not specify chamfering or
- 9 A threading block must not specifying chamfering or corner R.
- 10 The spindle speed override function is disabled during threading. The spindle speed is fixed at 100%.
- 11 Thread cycle retract function is ineffective to G32.

# 3.2 CONTINUOUS THREADING

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

#### **Explanation**

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

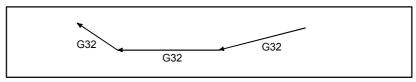


Fig. 3.2 (a) Continuous threading (Example of G32 in G code system A)

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

# 3.3 MULTIPLE THREADING

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

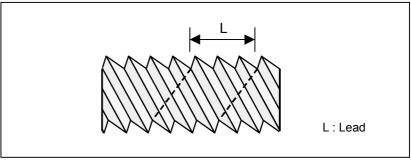


Fig. 3.3 (a) Multiple thread screws.

#### **Format**

#### (Constant lead threading)

**G32** IP \_ F\_ Q\_ ;

IP: End point

F\_: Lead in longitudinal direction

G32 IP \_ Q\_;

Q\_: Threading start angle

### **Explanation**

- Available threading commands

G32: Constant lead threading

G34: Variable lead threading

G76: Combined threading cycle

G92: Threading cycle

#### Limitation

- Start angle

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

#### - Start angle increment

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

Example:

For a shift angle of 180 degrees, specify Q180000.

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

#### - Specifiable start angle range

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

#### - Combined threading cycle (G76)

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

#### **Example**

# **3.4** TORQUE LIMIT SKIP (G31 P99)

With the motor torque limited (for example, by a torque limit command, issued through the PMC window), a move command following G31 P99 (or G31 P98) can cause the same type of cutting feed as with G01 (linear interpolation).

With the issue of a signal indicating a torque limit has been reached (because of pressure being applied or for some other reason), a skip occurs.

For details of how to use this function, refer to the manuals supplied by the machine tool builder.

#### **Format**

G31 P99 IP \_ F\_; G31 P98 IP \_ F\_;

G31 : One-shot G code (G code effective only in the block in which it is issued)

#### **Explanation**

- G31 P99

If the motor torque limit is reached, or a SKIP signal is received during execution of G31 P99, the current move command is aborted, and the next block is executed.

- G31 P98

If the motor torque limit is reached during execution of G31 P98, the current move command is aborted, and the next block is executed. The SKIP signal <X0004#7/Path 2 X0013#7> does not affect G31 P98. Entering a SKIP signal during the execution of G31 P98 does not cause a skip.

#### - Torque limit command

If a torque limit is not specified before the execution of G31 P99/98, the move command continues; no skip occurs even if a torque limit is reached.

#### - Custom macro system variable

When G31 P99/98 is specified, the custom macro variables hold the coordinates at the end of a skip.

If a SKIP signal causes a skip with G31 P99, the custom macro system variables hold the coordinates based on the machine coordinate system when it stops, rather than those when the SKIP signal is entered.

#### Limitation

#### - Axis command

Only one axis can be controlled in each block with G31 P98/99. If two or more axes are specified to be controlled in such blocks, or no axis command is issued, alarm PS0369 is generated.

#### - Simple synchronous control and angular axis control

G31 P99/98 cannot be used for axes subject to simple synchronous control or the X-axis or Z-axis when under angular axis control.

- Speed control

Parameter SKF (No. 6200#7) must be set to disable dry run, override, and automatic acceleration/deceleration for G31 skip commands.

#### - Consecutive commands

Do not use G31 P99/98 in consecutive blocks.



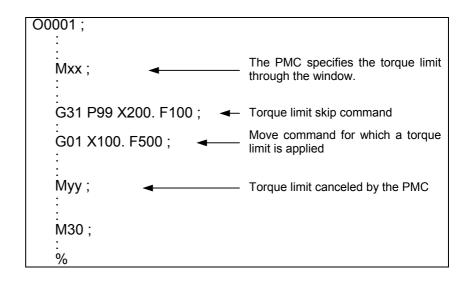
#### **⚠** WARNING

Always specify a torque limit before a G31 P99/98 command. Otherwise, G31 P99/98 allows move commands to be executed without causing a skip.

#### NOTE

If G31 is issued with cutter or tool nose radius compensation specified, alarm PS035 is generated. Therefore, before issuing G31, execute G40 to cancel cutter or tool nose radius compensation.

#### **Example**



4

# FUNCTIONS TO SIMPLIFY PROGRAMMING

This chapter explains the following items:

- 4.1 CANNED CYCLE (G90, G92, G94)
- 4.2 MULTIPLE REPETITIVE CYCLE (G70-G76)
- 4.3 CANNED CYCLE FOR DRILLING
- 4.4 RIGID TAPPING
- 4.5 CHAMFERING AND CORNER R
- 4.6 MIRROR IMAGE FOR DOUBLE TURRET (G68, G69)
- 4.7 DIRECT DRAWING DIMENSION PROGRAMMING

# **4.1** CANNED CYCLE (G90, G92, G94)

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

#### NOTE

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

ZX plane: Z-axis direction YZ plane: Y-axis direction XY plane: X-axis direction

4 The direction of the end face means the direction of the second axis on the plane as follows:

ZX plane: X-axis direction YZ plane: Z-axis direction XY plane: Y-axis direction

# 4.1.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

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

# 4.1.1.1 Straight cutting cycle

#### **Format**

#### G90X(U)\_Z(W)\_F\_;

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the length

F\_ : Cutting feedrate

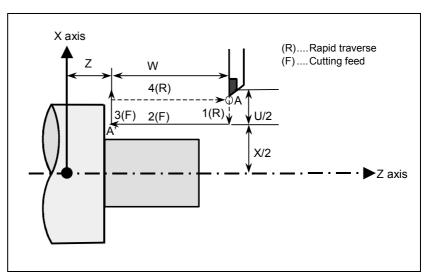


Fig. 4.1.1 (a) Straight cutting cycle

#### **Explanation**

#### - Operations

A straight cutting cycle performs four operations:

- (1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
- (2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the length.)
- (3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in cutting feed
- (4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

#### NOTE

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

# - Canceling the mode

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

### 4.1.1.2 Taper cutting cycle

#### **Format**

#### G90 X(U)\_Z(W)\_R\_F\_;

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the length

R\_ : Taper amount (R in the figure below)

F : Cutting feedrate

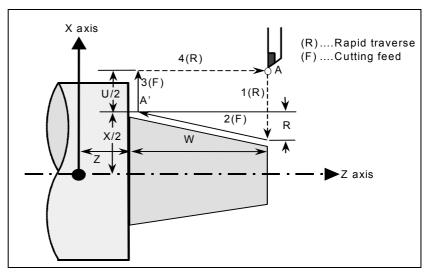


Fig. 4.1.1 (b) Taper cutting cycle

#### **Explanation**

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

#### **NOTE**

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

#### - Operations

A taper cutting cycle performs the same four operations as a straight cutting cycle.

However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

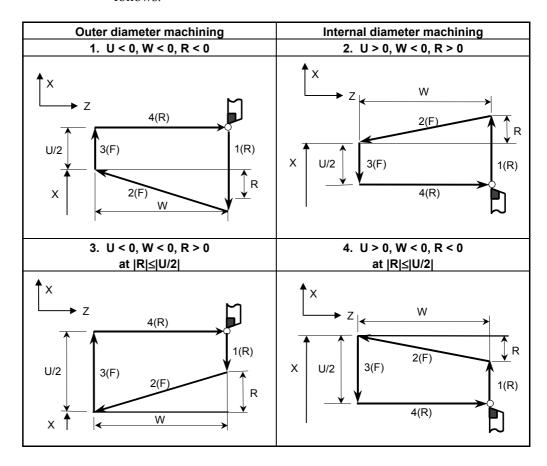
Operations 2, 3, and 4 after operation 1 are the same as for a straight cutting cycle.

#### **NOTE**

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

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

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



#### - Canceling the mode

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

# 4.1.2 Threading Cycle (G92)

# 4.1.2.1 Straight threading cycle

#### **Format**

#### G92 X(U)\_Z(W)\_F\_Q\_;

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the length

Q\_ : Angle for shifting the threading start angle

(Increment: 0.001 degrees,

Valid setting range: 0 to 360 degrees)

F\_ : Thread lead (L in the figure below)

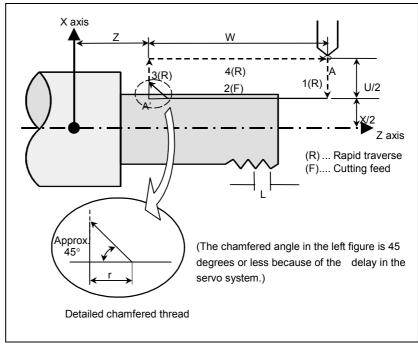


Fig. 4.1.2 (c) Straight threading

#### **Explanation**

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

#### - Operations

A straight threading cycle performs four operations:

- (1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
- (2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. At this time, thread chamfering is performed.

- (3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (Retraction after chamfering)
- (4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

#### ⚠ CAUTION

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

#### NOTE

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

#### - Canceling the mode

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

#### - Time constant and FL feedrate for threading

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

#### - Thread chamfering

Thread chamfering can be performed. A signal from the machine tool, initiates thread chamfering. The chamfering distance r is specified in a range from 0.1L to 12.7L in 0.1L increments by parameter No. 5130. (In the above expression, L is the thread lead.) A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.

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

#### NOTE

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

#### - Retraction after chamfering

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

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Performs an in-position check before retraction and uses the type of acceleration/deceleration after interpolation in rapid traverse, time constant for rapid traverse, FL feedrate, and rapid traverse rate specified in parameter No. 1420.

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

#### **NOTE**

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

#### - Shifting the start angle

Address Q can be used to shift the threading start angle.

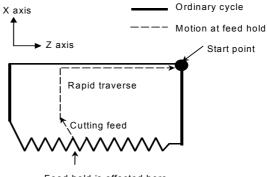
The start angle (Q) increment is 0.001 degrees and the valid setting range is between 0 and 360 degrees. No decimal point can be specified.

#### - Feed hold in a threading cycle

When the threading cycle retract function is not used, the machine stops at the end point of retraction after chamfering (end point of operation 3) by feed hold applied during threading.

#### - Threading cycle retract

When the "threading cycle retract" option function is used, feed hold may be applied during threading (operation 2). In this case, the tool immediately retracts with chamfering and returns to the start point on the second axis (X-axis), then the first axis (Z-axis) on the plane.



Feed hold is effected here.

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



Another feed hold cannot be made during retreat.

### - Inch threading

Inch threading specified with address E is not allowed.

# 4.1.2.2 Taper threading cycle

#### **Format**

# G92 X(U)\_Z(W)\_R\_F\_Q\_;

 $\mathsf{R}_{-}$ 

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the length

Q\_ : Angle for shifting the threading start angle (Increment: 0.001 degrees,

Valid setting range: 0 to 360 degrees)

: Taper amount (R in the figure below)

F\_ : Thread lead (L in the figure below)

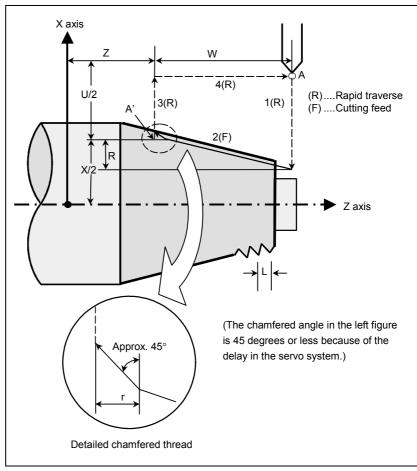


Fig. 4.1.2 (d) Taper threading cycle

#### **Explanation**

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

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

#### NOTE

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

#### - Operations

A taper threading cycle performs the same four operations as a straight threading cycle.

However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

Operations 2, 3, and 4 after operation 1 are the same as for a straight threading cycle.



#### **⚠** CAUTION

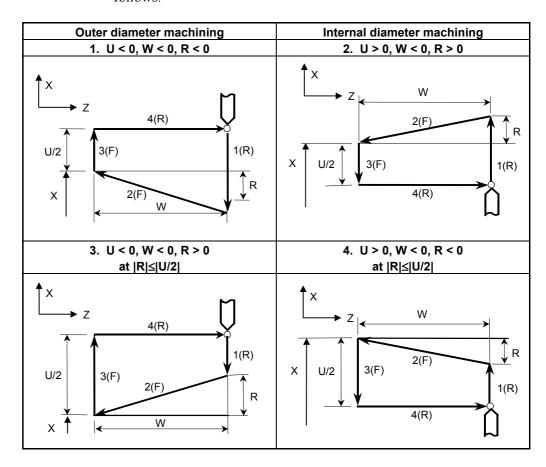
Notes on this threading are the same as in threading in G32. However, a stop by feed hold is as follows: Stop after completion of path 3 of threading cycle.

#### **NOTE**

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

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

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



#### - Canceling the mode

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

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

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

# **4.1.3** End Face Turning Cycle (G94)

# 4.1.3.1 Face cutting cycle

#### **Format**

#### G92 X(U)\_Z(W)\_F\_;

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the end face

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the end face

F\_ : Cutting feedrate

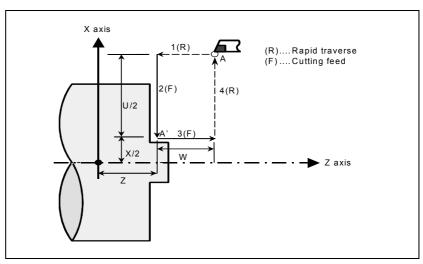


Fig. 4.1.3 (e) Face cutting cycle

#### **Explanation**

#### - Operations

A face cutting cycle performs four operations:

- (1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.
- (2) Operation 2 moves the tool to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the end face.)
- (3) Operation 3 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in cutting feed.
- (4) Operation 4 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

#### NOTE

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

#### - Canceling the mode

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

### 4.1.3.2 Taper cutting cycle

#### **Format**

#### G94 X(U)\_Z(W)\_R\_F\_;

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the end face

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the end face

R\_ : Taper amount (R in the figure below)

F\_ : Cutting feedrate

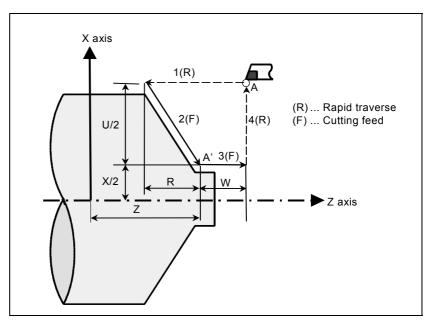


Fig. 4.1.3 (f) Taper cutting cycle

#### **Explanation**

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

#### **NOTE**

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

#### - Operations

A taper cutting cycle performs the same four operations as a face cutting cycle.

However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.

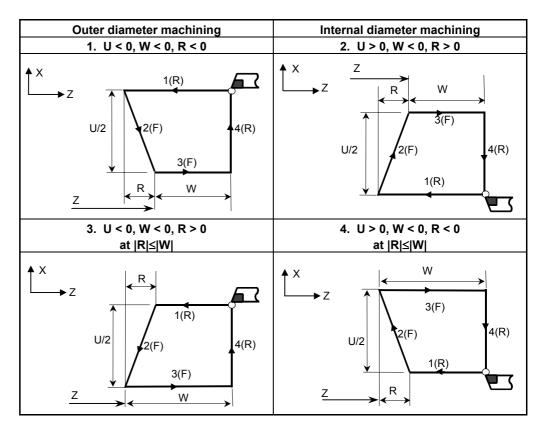
Operations 2, 3, and 4 after operation 1 are the same as for a face cutting cycle.

#### **NOTE**

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

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

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



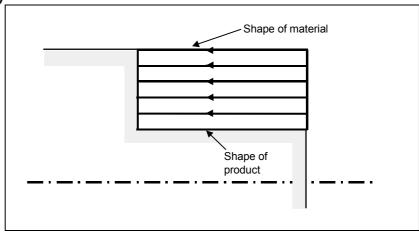
#### - Canceling the mode

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

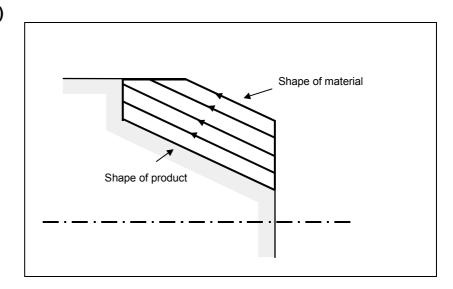
# **4.1.4** How to Use Canned Cycles (G90, G92, G94)

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

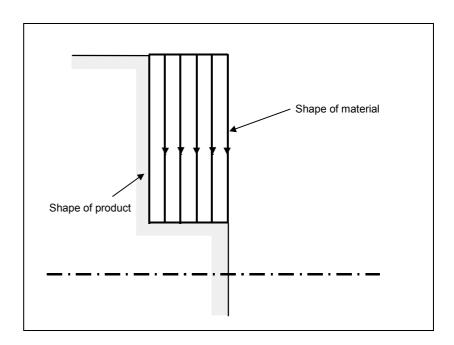
- Straight cutting cycle (G90)



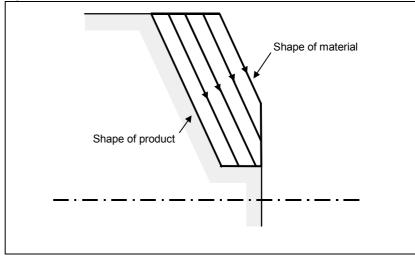
- Taper cutting cycle (G90)



# - Face cutting cycle (G94)



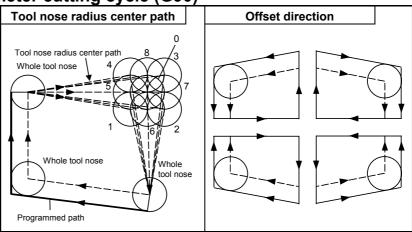
# - Face taper cutting cycle (G94)



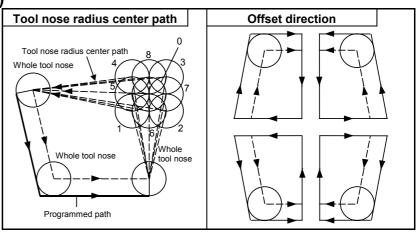
# 4.1.5 Canned Cycle and Tool Nose Radius Compensation

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

Outer diameter/internal diameter cutting cycle (G90)



End face cutting cycle (G94)



#### Threading cycle (G92)

Tool nose radius compensation cannot be applied.

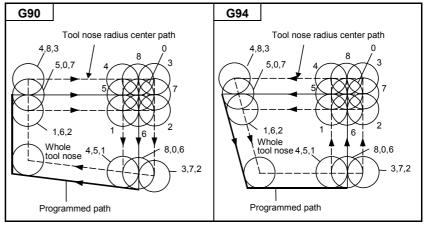
#### Differences between this CNC and the FANUC Series 16i/18i/21i

#### **NOTE**

This CNC is the same as the FANUC Series 16*i*/18*i*/21*i* in the offset direction, but differs from the series in the tool nose radius center path.

- For this CNC
   Cycle operations of a canned cycle are replaced
   with G00 or G01. In the first block to move the
   tool from the start point, start-up is performed.
   In the last block to return the tool to the start
   point, offset is canceled.
- For the FANUC Series 16i/18i/21i
   This series differs from this CNC in operations in the block to move the tool from the start point and the last block to return it to the start point.
   For details, refer to "FANUC Series 16i/18i/21i Operator's Manual."

### How compensation is applied for the FANUC Series 16i/18i/21i



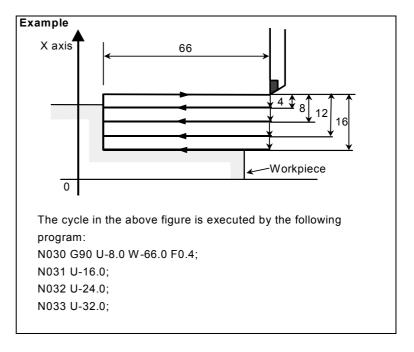
# **4.1.6** Restrictions on Canned Cycles

#### Limitation

#### - Modal

Since data items X (U), Z (W), and R in a canned cycle are modal values common to G90, G92, and G94. For this reason, if a new X (U), Z (W), or R value is not specified, the previously specified value is effective.

Thus, when the travel distance along the Z-axis does not vary as shown in the program example below, a canned cycle can be repeated only by specifying the travel distance along the X-axis.



The modal values common to canned cycles are cleared when a one-shot G code other than G04 is specified.

Since the canned cycle mode is not canceled by specifying a one-shot G code, a canned cycle can be performed again by specifying modal values. If no modal values are specified, no cycle operations are performed.

When G04 is specified, G04 is executed and no canned cycle is performed.

#### - Block in which no move command is specified

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

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

#### - Plane selection command

Specify a plane selection command (G17, G18, or G19) before setting a canned cycle or specify it in the block in which the first canned cycle is specified.

If a plane selection command is specified in the canned cycle mode, the command is executed, but the modal values common to canned cycles are cleared.

If an axis which is not on the selected plane is specified, alarm PS0330 is issued.

#### - Parallel axis

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

# **4.2** MULTIPLE REPETITIVE CYCLE (G70-G76)

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

### **NOTE**

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

# **4.2.1** Stock Removal in Turning (G71)

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

#### **Format**

```
ZpXp plane
 G71 U(∆d) R(e);
 G71 P(ns) Q(nf) U(\Deltau) W(\Deltaw) F(f) S(s) T(t);
             The move command between A and B is specified in the
             blocks from sequence number ns to nf.
 N (nf);
YpZp plane
 G71 W(∆d) R(e);
 G71 P(ns) Q(nf) V(\Deltaw) W(\Deltau) F(f) S(s) T(t);
 N (ns);
   ...
 N (nf);
XpYp plane
 G71 V(∆d) R(e);
 G71 P(ns) Q(nf) U(\Deltaw) V(\Deltau) F(f) S(s) T(t);
 N (ns);
   ...
 N (nf);
 \Delta d: Depth of cut
       The cutting direction depends on the direction AA'.
       This designation is modal and is not changed until the
       other value is designated. Also this value can be
       specified by the parameter (No. 5132), and the
       parameter is changed by the program command.
      : Escaping amount
       This designation is modal and is not changed until the
       other value is designated. Also this value can be
       specified by the parameter (No. 5133), and the
       parameter is changed by the program command.
 ns : Sequence number of the first block for the program of
       finishing shape.
     : Sequence number of the last block for the program of
       finishing shape.
 \Delta u: Distance of the finishing allowance in the direction of
       the second axis on the plane (X-axis for the ZX plane)
 \Delta w: Distance of the finishing allowance in the direction of
       the first axis on the plane (Z-axis for the ZX plane)
 f,s,t: Any F, S, or T function contained in blocks ns to nf in
       the cycle is ignored, and the F, S, or T function in this
       G71 block is effective.
```

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

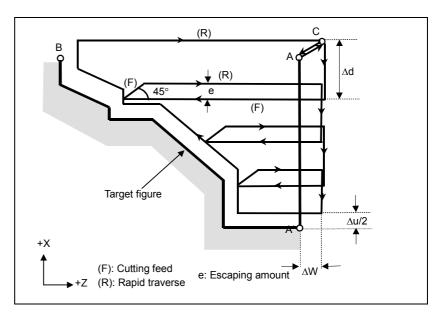


Fig. 4.2.1 (a) Cutting path in stock removal in turning (type I)

## **Explanation**

## - Operations

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

### NOTE

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

# - Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the first axis on the plane (Z-axis for the ZX plane). The signs of  $\Delta u$  and  $\Delta w$  are as follows:

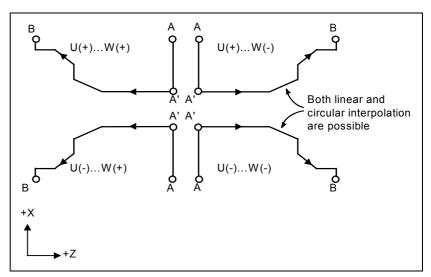


Fig. 4.2.1 (b) Four target figure patterns

#### Limitation

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

#### - Start block

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

When G00 is specified, cutting along path A-A' is performed in the positioning mode. When G01 is specified, cutting along path A-A' is performed in the linear interpolation mode.

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

#### - Check functions

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

#### **NOTE**

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

The following checks can also be made.

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

## - Types I and II Selection of type I or II

For G71, there are types I and II.

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

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

#### NOTE

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

## Selecting type I or II

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

- (1) When type I is selected Specify the second axis on the plane (X-axis for the ZX plane). Do not specify the first axis on the plane (Z-axis for the ZX plane).
- (2) When type II is selected
  Specify the second axis on the plane (X-axis for the ZX plane) and first axis on the plane (Z-axis for the ZX plane).
  When you want to use type II without moving the tool along the first axis on the plane (Z-axis for the ZX plane), specify the incremental programming with travel distance 0 (W0 for the ZX plane).

- Type I

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

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

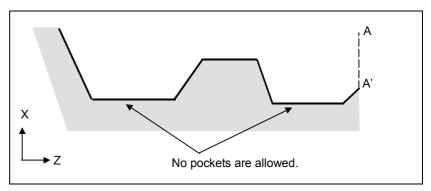


Fig. 4.2.1 (c) Figure which does not show monotone increase or decrease (type I)

### **⚠** CAUTION

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

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

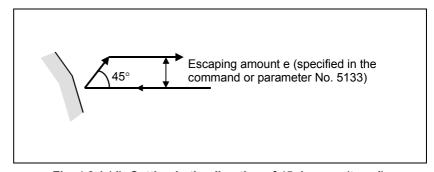


Fig. 4.2.1 (d) Cutting in the direction of 45 degrees (type I)

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

## - Type II

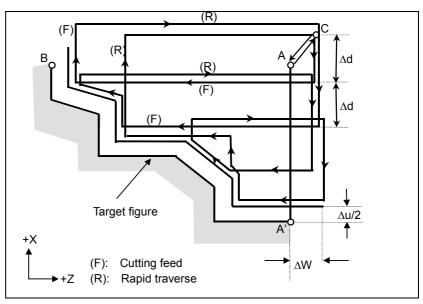


Fig. 4.2.1 (e) Cutting path in stock removal in turning (type II)

When a target figure passing through A, A', and B in this order is given by the program for a target figure as shown in the figure, the specified area is removed by  $\Delta d$  (depth of cut), with the finishing allowance specified by  $\Delta u/2$  and  $\Delta w$  left. Type II differs from type I in cutting the workpiece along the figure after rough cutting in the direction of the first axis on the plane (Z-axis for the ZX plane).

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

Type II differs from type I in the following points:

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

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

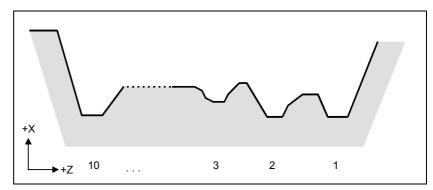


Fig. 4.2.1 (f) Figure having pockets (type II)

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

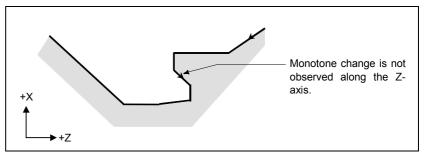


Fig. 4.2.1 (g) Figure which cannot be machined (type II)

## **⚠** CAUTION

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

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

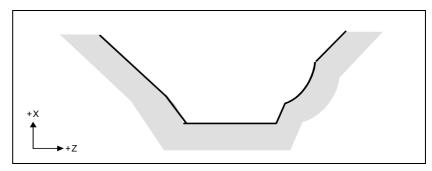


Fig. 4.2.1 (h) Figure which can be machined (type II)

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

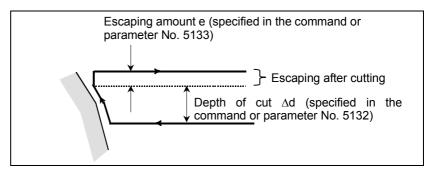


Fig. 4.2.1 (i) Cutting along the workpiece figure (type II)

The escaping amount after cutting (e) can be specified at address R or set in parameter No. 5133.

When moving from the bottom, however, the tool escapes to the direction of 45 degrees.

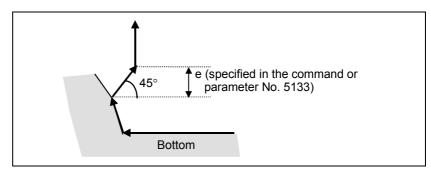


Fig. 4.2.1 (j) Escaping from the bottom to the direction of 45 degrees

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

(5) After all rough cutting terminates along the first axis on the plane (Z-axis for the ZX plane), the tool temporarily returns to the cycle start point. At this time, when there is a position whose height equals to that at the start point, the tool passes through the point in the position obtained by adding depth of cut Δd to the position of the figure and returns to the start point.

Then, rough cutting is performed as finishing along the target figure. At this time, the tool passes through the point in the obtained position (to which depth of cut  $\Delta d$  is added) when returning to the start point.

Bit 2 (RF2) of parameter No. 5105 can be set to 1 so that rough cutting as finishing is not performed.

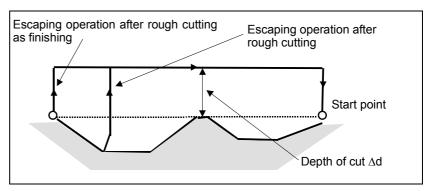


Fig. 4.2.1 (k) Escaping operation when the tool returns to the start point (type II)

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

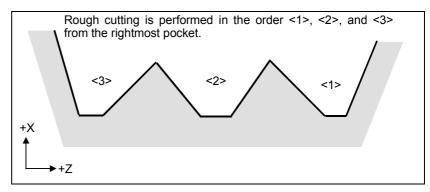


Fig. 4.2.1 (I) Rough cutting order in the case of monotone decrease (type II)

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

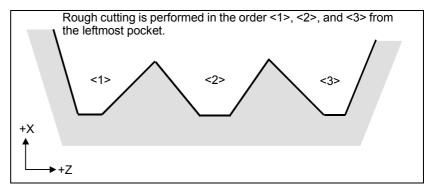


Fig. 4.2.1 (m) Rough cutting order in the case of monotone increase (type II)

The path in rough cutting is as shown below.

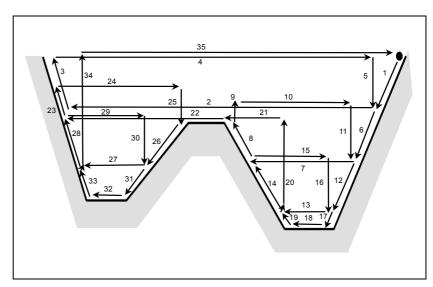


Fig. 4.2.1 (n) Cutting path for multiple pockets (type II)

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

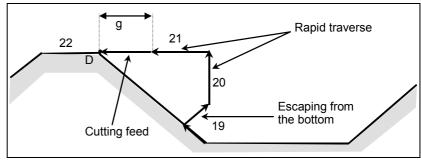


Fig. 4.2.1 (o) Details of motion after cutting for a pocket (type II)

Cuts the workpiece at the cutting feedrate and escapes to the direction of 45 degrees. (Operation 19)

Then, moves to the height of point D in rapid traverse. (Operation 20)

Then, moves to the position the amount of g before point D. (Operation 21)

Finally, moves to point D in cutting feed.

The clearance g to the cutting feed start position is set in parameter No. 5134

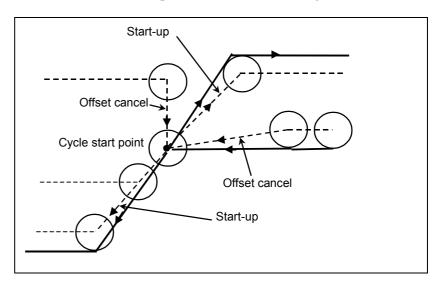
For the last pocket, after cutting the bottom, the tool escapes to the direction of 45 degrees and returns to the start point in rapid traverse. (Operations 34 and 35)

#### **!** CAUTION

- 1 This CNC differs from the FANUC Series 16*i*/18*i*/21*i* in cutting of a pocket.
  - The tool first cuts the nearest pocket to the start point. After cutting of the pocket terminates, the tool moves to the nearest but one pocket and starts cutting.
- 2 When the figure has a pocket, generally specify a value of 0 for  $\Delta w$  (finishing allowance). Otherwise, the tool may dig into the wall on one side.

### - Tool nose radius compensation

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



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

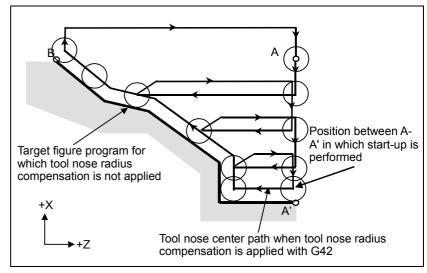
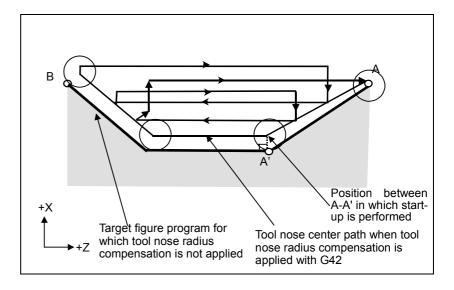


Fig. 4.2.1 (p) Path when tool nose radius compensation is applied



## NOTE

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

# 4.2.2 Stock Removal in Facing (G72)

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

#### **Format**

```
ZpXp plane
 G72 W(∆d) R(e);
 G72 P(ns) Q(nf) U(\Delta u) W(\Delta w) F(f) S(s) T(t);
             The move command between A and B is specified in the
             blocks from sequence number ns to nf.
 N (nf);
YpZp plane
 G72 V(∆d) R(e);
 G72 P(ns) Q(nf) V(\Deltaw) W(\Deltau) F(f) S(s) T(t);
 N (ns);
   ...
 N (nf);
XpYp plane
 G72 U(∆d) R(e);
 G72 P(ns) Q(nf) U(\Deltaw) W(\Deltau) F(f) S(s) T(t);
 N (ns);
   ...
 N (nf);
\Delta d: Depth of cut
      The cutting direction depends on the direction AA'. This
      designation is modal and is not changed until the other
      value is designated. Also this value can be specified by
      the parameter (No. 5132), and the parameter is changed
      by the program command.
    : Escaping amount
      This designation is modal and is not changed until the
      other value is designated. Also this value can be
      specified by the parameter (No. 5133), and the
      parameter is changed by the program command.
ns : Sequence number of the first block for the program of
      finishing shape.
nf : Sequence number of the last block for the program of
      finishing shape.
\Delta u: Distance of the finishing allowance in the direction of the
      second axis on the plane (X-axis for the ZX plane)
\Delta w: Distance of the finishing allowance in the direction of the
      first axis on the plane (Z-axis for the ZX plane)
f,s,t: Any F, S, or T function contained in blocks ns to nf in
      the cycle is ignored, and the F, S, or T function in this
      G72 block is effective.
```

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

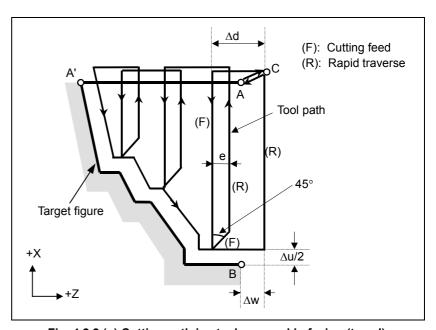


Fig. 4.2.2 (q) Cutting path in stock removal in facing (type I)

## **Explanation**

## - Operations

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

#### NOTE

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

# - Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the second axis on the plane (X-axis for the ZX plane). The signs of  $\Delta u$  and  $\Delta w$  are as follows:

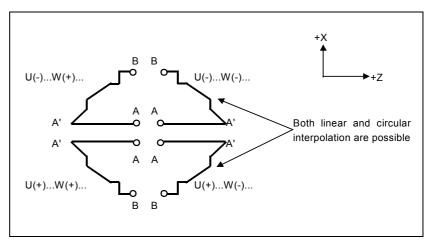


Fig. 4.2.2 (r) Signs of the values specified at U and W in stock removal in facing

#### Limitation

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

#### - Start block

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

When G00 is specified, cutting along path A-A' is performed in the positioning mode. When G01 is specified, cutting along path A-A' is performed in the linear interpolation mode.

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

#### - Check functions

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

#### **NOTE**

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

The following checks can also be made.

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

# Types I and II Selection of type I or II

For G72, there are types I and II.

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

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

## Selecting type I or II

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

- (1) When type I is selected
  Specify the first axis on the plane (Z-axis for the ZX plane). Do
  not specify the second axis on the plane (X-axis for the ZX
  plane).
- (2) When type II is selected
  Specify the second axis on the plane (X-axis for the ZX plane)
  and first axis on the plane (Z-axis for the ZX plane).
  When you want to use type II without moving the tool along the second axis on the plane (X-axis for the ZX plane), specify the incremental programming with travel distance 0 (U0 for the ZX plane).

## - Type I

G72 differs from G71 in the following points:

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

## Type II

G72 differs from G71 in the following points:

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

#### - Tool nose radius compensation

See the pages on which G71 is explained.

# 4.2.3 Pattern Repeating (G73)

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

#### **Format**

```
ZpXp plane
  G73 W(\Deltak) U(\Deltai) R(d);
  G73 P(ns) Q(nf) U(\Deltau) W(\Deltaw) F(f) S(s) T(t);
  N (ns);
             The move command between A and B is specified in the
             blocks from sequence number ns to nf.
YpZp plane
 G73 V(\Deltak) W(\Deltai) R(d);
 G73 P(ns) Q(nf) V(\Deltaw) W(\Deltau) F(f) S(s) T(t);
 N (ns);
   ...
 N (nf);
XpYp plane
 G73 U(\Deltak) V(\Deltai) R(d);
  G73 P(ns) Q(nf) U(\Deltaw) V(\Deltau) F(f) S(s) T(t);
 N (ns);
   ...
 N (nf);
      : Distance of escape in the direction of the second axis
        on the plane (X-axis for the ZX plane)
        This designation is modal and is not changed until the
       other value is designated. Also this value can be
        specified by the parameter No. 5135, and the
        parameter is changed by the program command.
      : Distance of escape in the direction of the first axis on
\Delta k
       the plane (Z-axis for the ZX plane)
        This designation is modal and is not changed until the
        other value is designated. Also this value can be
        specified by the parameter No. 5136, and the
        parameter is changed by the program command.
      : The number of division
d
        This value is the same as the repetitive count for rough
        cutting. This designation is modal and is not changed
       until the other value is designated. Also, this value can
        be specified by the parameter No. 5137, and the
        parameter is changed by the program command.
      : Sequence number of the first block for the program of
ns
        finishing shape.
      : Sequence number of the last block for the program of
nf
       finishing shape.
```

 $\Delta u$  : Distance of the finishing allowance in the direction of the second axis on the plane (X-axis for the ZX plane)

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

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

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

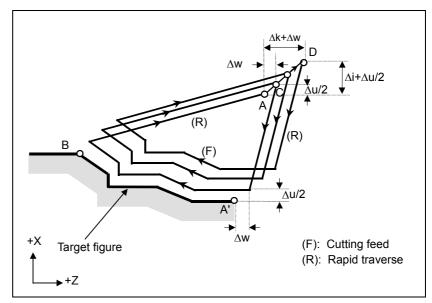


Fig. 4.2.3 (s) Cutting path in pattern repeating

## **Explanation**

## - Operations

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

#### **NOTE**

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

# - Target figure Patterns

As in the case of G71, there are four target figure patterns. Be careful about signs of  $\Delta u$ ,  $\Delta w$ ,  $\Delta i$ , and  $\Delta k$  when programming this cycle.

#### - Start block

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

When G00 is specified, cutting along path A-A' is performed in the positioning mode. When G01 is specified, cutting along path A-A' is performed in the linear interpolation mode.

## - Check function

The following check can be made.

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

### - Tool nose radius compensation

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

# **4.2.4** Finishing Cycle (G70)

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

#### **Format**

## G70 P(ns) Q(nf);

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

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

## **Explanation**

## - Operations

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

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

# - Target figure Check function

The following check can be made.

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

## - Storing P and Q blocks

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

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

#### NOTE

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

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

#### - Return to the cycle start point

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

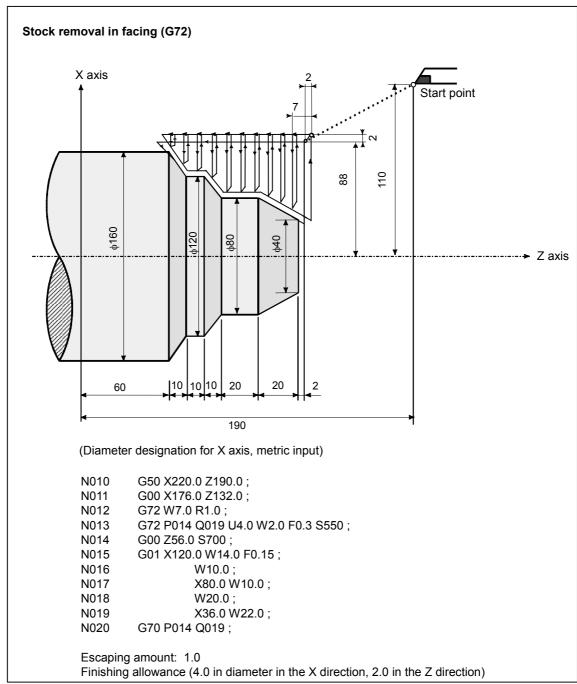
#### NOTE

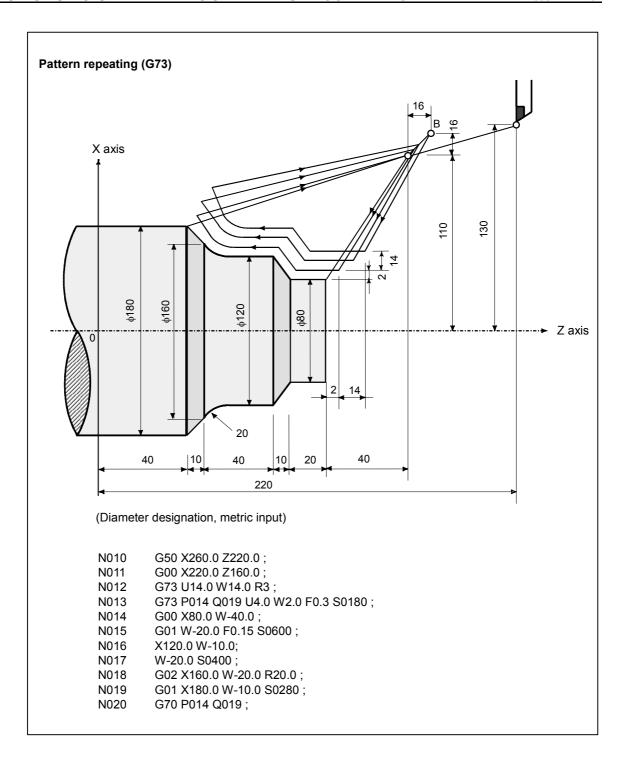
The tool returns to the cycle start point always in the nonlinear positioning mode regardless of the setting of bit 1 (LRP) of parameter No. 1401. Before executing a finishing cycle for a target figure with a pocket cut by G71 or G72, check that the tool does not interfere with the workpiece when returning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

## - Tool nose radius compensation

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

## Example





# **4.2.5** End Face Peck Drilling Cycle (G74)

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

#### **Format**

# G74R (e); G74X(U)\_ Z(W)\_ P( $\Delta$ i) Q( $\Delta$ k) R( $\Delta$ d) F (f);

e : Return amount

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

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

U\_,W\_: Travel distance along the second axis on the plane (U for the ZX plane) from point A to B

Travel distance along the first axis on the plane (W for the ZX plane) from point A to C

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

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

 $\Delta d$  : Relief amount of the tool at the cutting bottom

f : Feedrate

	Unit	Diameter/radius programming	Sign
е	Depends on the increment system for the reference axis.	Radius programming	Not required
Δί	Depends on the increment system for the reference axis.	Radius programming	Not required
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required
Δd	Depends on the increment system for the reference axis.	Radius programming	NOTE 1

### **NOTE**

- Normally, specify a positive value for ∆d. When X
   (U) and ∆i are omitted, specify a value with the sign indicating the direction in which the tool is to escape.
- 2 No decimal point can be input for  $P(\Delta i)$  and  $Q(\Delta k)$ .

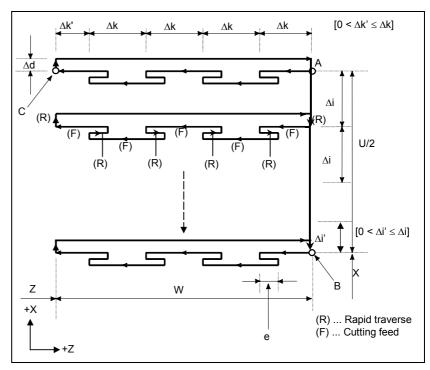


Fig. 4.2.5 (a) Cutting path in end face peek drilling cycle

## **Explanation**

- Operations

A cycle operation of cutting by  $\Delta k$  and return by e is repeated. When cutting reaches point C, the tool escapes by  $\Delta d$ . Then, the tool returns in rapid traverse, moves to the direction of point B by  $\Delta i$ , and performs cutting again.

## NOTE

- 1 While both e and  $\Delta d$  are specified by the same address, the meanings of them are determined by specifying the X, Y, or Z axis. When the axis is specified,  $\Delta d$  is used.
- 2 The cycle machining is performed by G74 command with specifying the axis.
- Tool nose radius compensation

Tool nose radius compensation cannot be applied.

# 4.2.6 Outer Diameter / Internal Diameter Drilling Cycle (G75)

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

#### **Format**

G75R (e);	
G75X(U)_ Z	$Z(W)_P(\Delta i) Q(\Delta k) R(\Delta d) F (f);$
е	: Return amount
	This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No.
	5139, and the parameter is changed by the
X_,Y_,Z_	program command.  : Coordinate of the second axis on the plane
^_, ' _,	(X-axis for the ZX plane) at point B and
	Coordinate of the first axis on the plane (Z-axis
	for the ZX plane) at point C
(U.V.W)	: Travel distance along the second axis on the
(/ _/ _/	plane (U for the ZX plane) from point A to B
	Travel distance along the first axis on the
	plane (W for the ZX plane) from point A to C
$\Delta i$	: Depth of cut in the direction of the second axis
	on the plane (X-axis for the ZX plane)
Δk	: Travel distance in the direction of the first axis
	on the plane (Z-axis for the ZX plane)
$\Delta d$	: Relief amount of the tool at the cutting bottom
f	: Feedrate

	Unit	Diameter/radius programming	Sign
е	Depends on the increment system for the reference axis.	Radius programming	Not required
Δί	Depends on the increment system for the reference axis.	Radius programming	Not required
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required
Δd	Depends on the increment system for the reference axis.	Radius programming	NOTE

### **NOTE**

Normally, specify a positive value for  $\Delta d$ . When Z (W) and  $\Delta k$  are omitted, specify a value with the sign indicating the direction in which the tool is to escape.

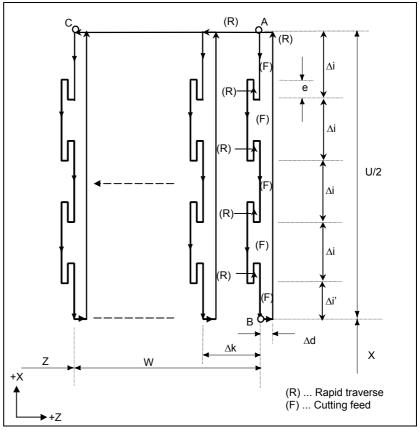


Fig. 4.2.6 (b) Outer diameter/internal diameter drilling cycle

## **Explanation**

## - Operations

A cycle operation of cutting by  $\Delta i$  and return by e is repeated. When cutting reaches point B, the tool escapes by  $\Delta d$ . Then, the tool returns in rapid traverse, moves to the direction of point C by  $\Delta k$ , and performs cutting again.

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

## - Tool nose radius compensation

Tool nose radius compensation cannot be applied.

# **4.2.7** Multiple Threading Cycle (G76)

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

#### **Format**

## G76 P(m) (r) (a) Q(∆dmin) R(d ) ; G76 X(U)\_ Z(W)\_ R(i ) P(k ) Q(∆d) F (L ) ;

m: Repetitive count in finishing (1 to 99)

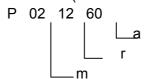
This value can be specified by the parameter No. 5142, and the parameter is changed by the program command.

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

a : Angle of tool nose

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

m, r, and a are specified by address P at the same time. (Example) When m=2, r=1.2L, a=60°, specify as shown below (L is lead of thread).



∆dmin : Minimum cutting depth

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

d : Finishing allowance

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

 $X_{-}, Z_{-}$ : Coordinates of the cutting end point (point D in the

figure) in the direction of the length

U\_, W\_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length

i : Taper amount

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

 $\begin{array}{lll} k & : \mbox{ Height of thread} \\ \Delta d & : \mbox{ Depth of cut in 1st cut} \end{array}$ 

L : Lead of thread

	Unit	Diameter/radius programming	Sign
∆dmin	Depends on the increment system for the reference axis.	Radius programming	Not required
d	Depends on the increment system for the reference axis.	Radius programming	Not required
i	Depends on the increment system for the reference axis.	Radius programming	Required
k	Depends on the increment system for the reference axis.	Radius programming	Not required
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required

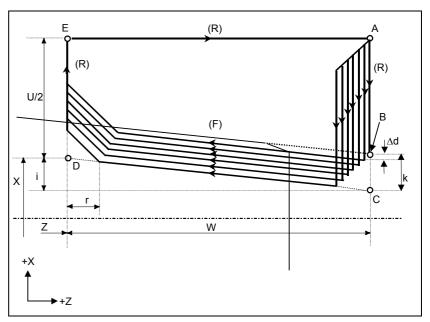


Fig. 4.2.7 (c) Cutting path in combined threading cycle cycle

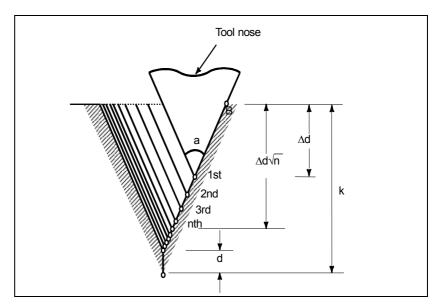
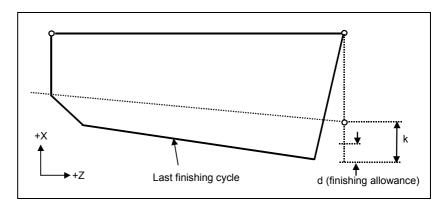


Fig. 4.2.7 (d) Detail of cutting

## - Repetitive count in finishing

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



## **Explanation**

## - Operations

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

The time constant for acceleration/deceleration after interpolation and FL feedrate for thread chamfering and the feedrate for retraction after chamfering are the same as for thread chamfering with G92 (canned cycle).

## **NOTE**

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

### **⚠** CAUTION

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

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

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

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

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

Taper amount (i):

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

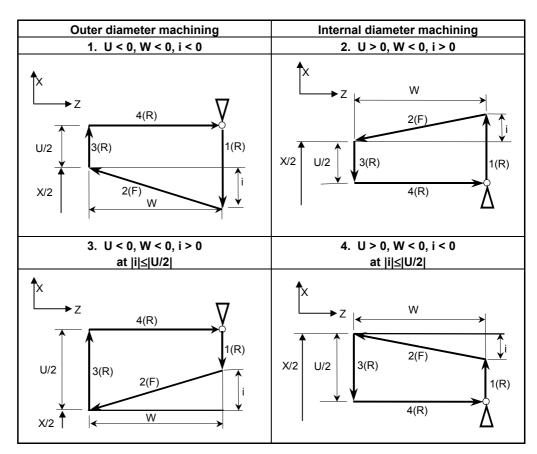
Height of thread (k):

Plus (always specified with a plus sign)

Depth of cut in the first cut ( $\Delta d$ ):

Plus (always specified with a plus sign)

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



## - Time constant and FL feedrate for threading

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

## - Thread chamfering

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

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

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

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

#### NOTE

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

## - Retraction after chamfering

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

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Performs an in-position check before retraction and uses the type of acceleration/deceleration after interpolation in rapid traverse, time constant for rapid traverse, FL feedrate, and rapid traverse rate specified in parameter No. 1420.

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

#### NOTE

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

## - Shifting the start angle

The threading start angle cannot be shifted.

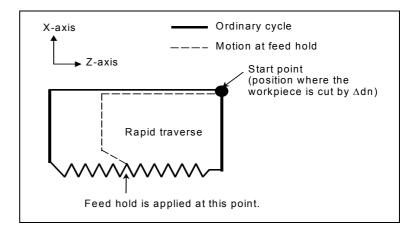
## - Feed hold in a threading cycle

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

## - Feed hold when the threading cycle retract function is used

When the "threading cycle retract" option function is used, feed hold may be applied during threading in a combined threading cycle cycle (G76). In this case, the tool quickly retracts in the same way as for the last chamfering in a threading cycle and returns to the start point in the current cycle (position where the workpiece is cut by  $\Delta$ dn).

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



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



#### ⚠ CAUTION

Another feed hold cannot be performed during retraction.

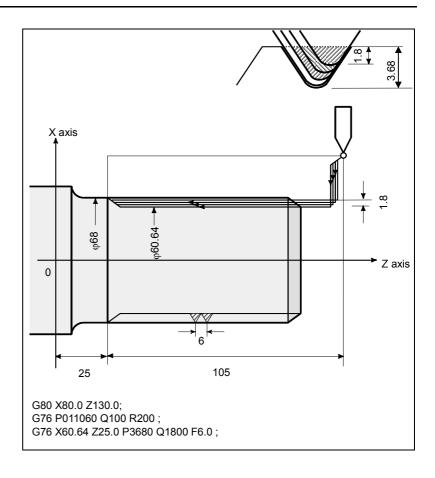
### - Inch threading

Inch threading specified with address E is not allowed.

### - Tool nose radius compensation

Tool nose radius compensation cannot be applied.

## Example



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

#### **Programmed commands**

- Program memory

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

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

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

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

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

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

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

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

- Dwell (G04)
- G00, G01, G02, and G03
- Custom macro branch and repeat command
  The branch destination must be between the sequence numbers specified at P and Q, however. High-speed branch specified by bits 1 and 4 of parameter No. 6000 is invalid. No custom macro call (simple, modal, or subprogram call) cannot be specified.
- Direct drawing dimension programming command and chamfering and corner R command

  Direct drawing dimension programming and chamfering and corner R require multiple blocks to be specified. The block with the last sequence number specified at Q must not be an intermediate block of these specified blocks.

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

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

#### Relation with other functions

#### - Manual intervention

While a multiple repetitive cycle (G70 to G76) is being executed, it is possible to stop the cycle and to perform manual intervention.

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

#### - Interruption type macro

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

#### - Program restart and tool retract and recover

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

#### - Axis name and second auxiliary functions

Even if address U, V, W, or A is used as an axis name or second auxiliary function, data specified at address U, V, W, or A in a G71 to G73 or G76 block is assumed to be that for the multiple repetitive cycle.

## 4.3 CANNED CYCLE FOR DRILLING

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

Table 4.3 (a) lists canned cycles for drilling.

Table 4.3 (a) Canned cycles for drilling

G code	Drilling axis	Hole machining operation	Operation in the bottom hole position	Retraction operation	Applications
G80	1	-	-	-	Cancel
G83	Z axis	Cutting feed / intermittent	Dwell	Rapid traverse	Front drilling cycle
G84	Z axis	Cutting feed	$\begin{array}{c} Dwell \to \\ spindle \; CCW \end{array}$	Cutting feed	Front tapping cycle
G85	Z axis	Cutting feed	Dwell	Cutting feed	Front boring cycle
G87	X axis	Cutting feed / intermittent	Dwell	Rapid traverse	Side drilling cycle
G88	X axis	Cutting feed	Dwell → Spindle CCW	Cutting feed	Side tapping cycle
G89	X axis	Cutting feed	Dwell	Cutting feed	Side boring cycle

#### **Explanation**

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

Operation 1 ....... Positioning of X (Z) and C axis Operation 2 ....... Rapid traverse up to point R level

Operation 3 ...... Hole machining

Operation 4 ...... Operation at the bottom of a hole

Operation 5 ...... Retraction to point R level

Operation 6 .......... Rapid traverse up to the initial level

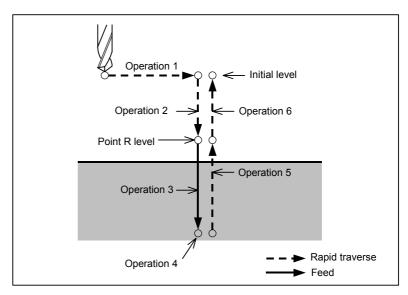


Fig. 4.3 (e) Operation sequence of canned cycle for drilling

#### - Positioning axis and drilling axis

The C-axis and X- or Z-axis are used as positioning axes. The X- or Z-axis, which is not used as a positioning axis, is used as a drilling axis. A drilling G code specifies positioning axes and a drilling axis as shown below.

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.

 G code
 Positioning axis
 Drilling axis

 G83, G84, G85
 X axis, C axis
 Z axis

 G87, G88, G89
 Z axis, C axis
 X axis

Table 4.3 (b) Positioning axis and drilling axis

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

#### - Drilling mode

G83 to G85/G87 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.

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

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

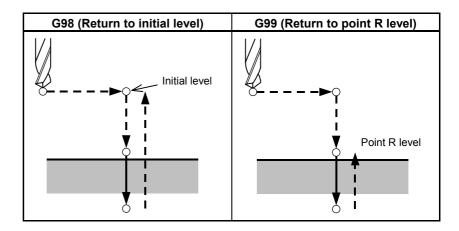
The feedrate specified at F is retained also after the drilling cycle is canceled. When Q data is required, it must be specified in each block. Once specified, the M code used for C-axis clamp/unclamp functions as a modal code. It is canceled by specifying G80.

#### - Return point level

In G code system A, the tool returns to the initial level from the bottom of a hole. In G code system B or C, specifying G98 returns the tool to the initial level from the bottom of a hole and specifying G99 returns the tool to the point R level from the bottom of a hole.

The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.

The initial level does not change even when drilling is performed in the G99 mode.



#### - Number of repeats

To repeat drilling for equally-spaced holes, specify the number of repeats in K .

K is effective only within the block where it is specified.

Specify the first hole position in incremental programming.

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

Number of repeats K The maximum command value = 9999

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

#### **NOTE**

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

#### - M code used for C-axis clamp/unclamp

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

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

#### - Cancel

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

#### Group 01 G codes (Example)

G00: Positioning (rapid traverse)

G01: Linear interpolation

G02: Circular interpolation (CW)G03: Circular interpolation (CCW)

#### - Symbols in figures

Subsequent subsections explain the individual canned cycles. Figures in these explanations use the following symbols:

	Positioning (rapid traverse G00)
<b>→</b>	Cutting feed (linear interpolation G01)
P1	Dwell specified in the program
P2	Dwell specified in parameter No.5111
$M \alpha$	Issuing the M code for C-axis clamp
	(The value of $\alpha$ is specified with parameter No. 5110.)
$M(\alpha + 1)$	Issuing the M code for C-axis unclamp

#### **⚠** CAUTION

1 In each canned cycle, addresses R, Z, and X are handled as follows:

R\_ : Always handled as a radius.

Z\_ or X\_: Depends on diameter/radius

programming.

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

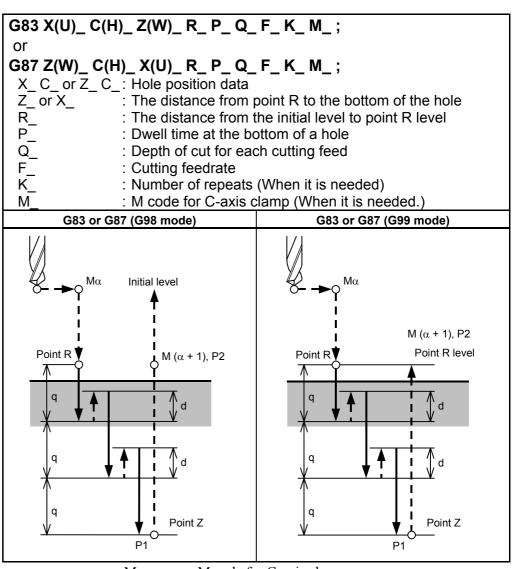
## 4.3.1 Front Drilling Cycle (G83)/Side Drilling Cycle (G87)

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

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

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

#### **Format**



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

d : Retraction distance specified in parameter No. 5114

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

#### **Format**

G83 X(U)\_ C(H)\_ Z(W)\_ R\_ P\_ Q\_ F\_ K\_ M\_; G87 Z(W)\_ C(H)\_ X(U)\_ R\_ P\_ Q\_ F\_ K\_ M\_ ; X\_ C\_ or Z\_ C\_: Hole position data **Z\_** or **X\_** : The distance from point R to the bottom of the hole R\_ The distance from the initial level to point R level : Dwell time at the bottom of a hole : Depth of cut for each cutting feed : Cutting feedrate : Number of repeats (When it is needed.) : M code for C-axis clamp (When it is needed.) G83 or G87 (G98 mode) G83 or G87 (G99 mode) Initial level M ( $\alpha$  + 1), P2 Point R level  $[M (\alpha + 1), P2]$ q Point Z Point Z P1 P1

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

d : Retraction distance specified in parameter No. 5115

#### **Example**

M51; Setting C-axis index mode ON

M3 S2000; Rotating the drill

G00 X50.0 C0.0; Positioning the drill along the X- and C-axes

G83 Z-40.0 R-5.0 Q5000 F5.0 M31; Drilling hole 1
C90.0 Q5000 M31; Drilling hole 2
C180.0 Q5000 M31; Drilling hole 3
C270.0 Q5000 M31; Drilling hole 4

G80 M05; Canceling the drilling cycle and stopping

drill rotation

M50; Setting C-axis index mode off

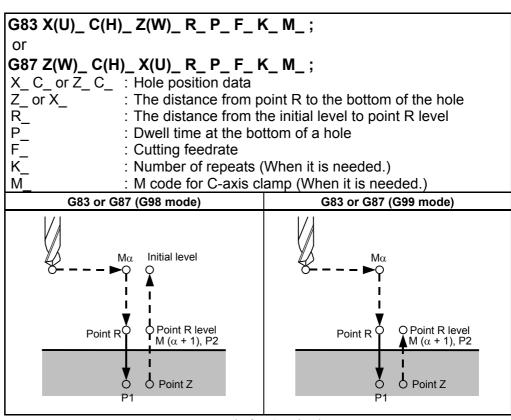
#### NOTE

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

#### - Drilling cycle (G83 or G87)

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

#### **Format**



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

#### **Example**

M51; Setting C-axis index mode ON

M3 S2000 ; Rotating the drill

G00 X50.0 C0.0; Positioning the drill along the X- and

C-axes

G83 Z-40.0 R-5.0 P500 F5.0 M31 ; Drilling hole 1
C90.0 M31 ; Drilling hole 2
C180.0 M31 ; Drilling hole 3
C270.0 M31 ; Drilling hole 4

G80 M05; Canceling the drilling cycle and

stopping drill rotation

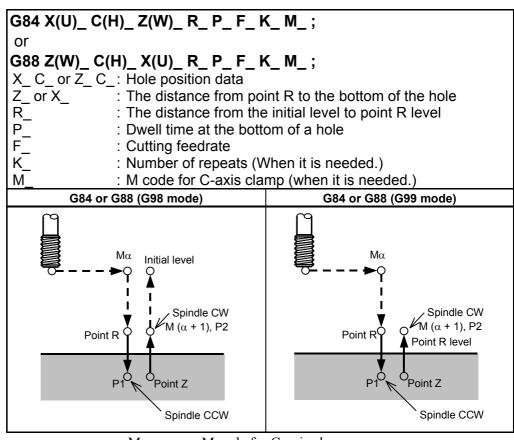
M50; Setting C-axis index mode off

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

This cycle performs tapping.

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

#### **Format**



 $\begin{array}{lll} M\alpha & : & M \ code \ for \ C\ -axis \ clamp \\ M \ (\alpha + 1): & M \ code \ for \ C\ -axis \ unclamp \\ P1 & : & Dwell \ specified \ in \ the \ program \\ P2 & : & Dwell \ specified \ in \ parameter \ No. \ 5111 \end{array}$ 

#### **Explanation**

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads. Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is completed.

#### **NOTE**

Bit 6 (M5T) of parameter No. 5101 specifies whether the spindle stop command (M05) is issued before the direction in which the spindle rotates is specified with M03 or M04. For details, refer to the operator's manual created by the machine tool builder.

#### **Example**

M51; Setting C-axis index mode ON

M3 S2000 ; Rotating the drill

G00 X50.0 C0.0 ; Positioning the drill along the X- and C-

axes

G84 Z-40.0 R-5.0 P500 F5.0 M31; Drilling hole 1
C90.0 M31; Drilling hole 2
C180.0 M31; Drilling hole 3
C270.0 M31; Drilling hole 4

G80 M05; Canceling the drilling cycle and

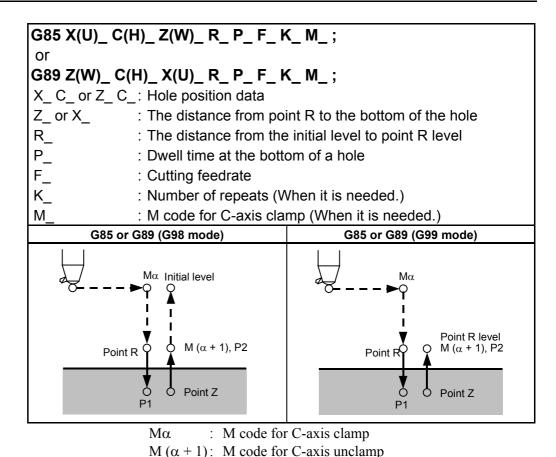
stopping drill rotation

M50 ; Setting C-axis index mode off

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

This cycle is used to bore a hole.

#### **Format**



**Explanation** 

After positioning, rapid traverse is performed to point R.

: Dwell specified in the program

Drilling is performed from point R to point Z.

After the tool reaches point Z, it returns to point R at a feedrate twice

: Dwell specified in parameter No. 5111

the cutting feedrate.

P1

P2

**Example** 

M51; Setting C-axis index mode ON

M3 S2000; Rotating the drill

G00 X50.0 C0.0; Positioning the drill along the X- and

C-axes

G85 Z-40.0 R-5.0 P500 F5.0 M31; Drilling hole 1 Drilling hole 2 C90.0 M31; C180.0 M31; Drilling hole 3 C270.0 M31; Drilling hole 4

G80 M05; Canceling the drilling cycle and

stopping drill rotation

M50; Setting C-axis index mode off

## 4.3.4 Canned Cycle for Drilling Cancel (G80)

G80 cancels canned cycle for drilling.

**Format** 

G80;

**Explanation** 

Canned cycle for drilling is canceled to perform normal operation.

Point R and point Z are cleared.

Other drilling data is also canceled (cleared).

**Example** 

M51; Setting C-axis index mode ON

M3 S2000; Rotating the drill

G00 X50.0 C0.0; Positioning the drill along the X- and

C-axes.

G83 Z-40.0 R-5.0 P500 F5.0 M31; Drilling hole 1
C90.0 M31; Drilling hole 2
C180.0 M31; Drilling hole 3
C270.0 M31; Drilling hole 4

G80 M05; Canceling the drilling cycle and

stopping drill rotation

M50 ; Setting C-axis index mode off

## **4.3.5** Precautions to Be Taken by Operator

#### - Reset and emergency stop

Even when the controller is stopped by resetting or emergency stop in the course of drilling cycle, the drilling mode and drilling data are saved; with this mind, therefore, restart operation.

#### - Single block

When drilling cycle is performed with a single block, the operation stops at the end points of operations 1, 2, 6 in Fig. 4.3 (a).

Consequently, it follows that operation is started up 3 times to drill one hole. The operation stops at the end points of operations 1, 2 with the feed hold lamp ON. The operation stops in the feed hold conditions at the end point of operation 6 if the repeat remains, and it stops in stop conditions in other cases.

#### - Feed hold

When "Feed Hold" is applied between operations 3 and 5 by G84/G88, the feed hold lamp lights up immediately if the feed hold is applied again to operation 6.

#### - Override

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

## **4.4** RIGID TAPPING

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

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

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

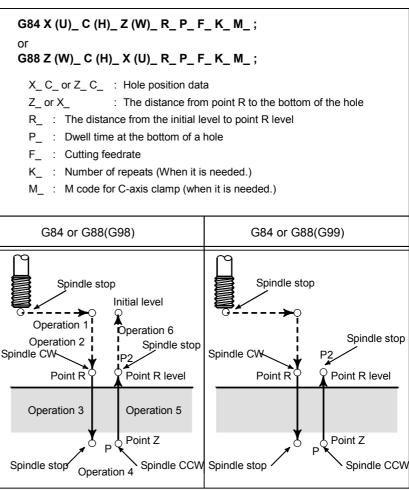
For rigid tapping, each turn of the spindle corresponds to a certain amount of feed (screw lead) along the spindle axis. This also applies to acceleration/deceleration. This means that rigid tapping does not demand the use of float tappers as in the case of conventional tapping, thus enabling high-speed,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

When the system is equipped with the optional multispindle control function, the second to fourth spindles can be used for rigid tapping.

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

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

#### **Format**



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

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

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

Xp: X axis or its parallel axis

Yp: Y axis or its parallel axis

Zp: Z axis or its parallel axis

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

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

Xp: X axis or its parallel axis

Yp: Y axis or its parallel axis

Zp: Z axis or its parallel axis

#### (Series 15 format)

#### G84.2 X (U)\_ C (H)\_ Z (W)\_ R\_ P\_ F\_ L\_ S\_;

X\_C\_ or Z\_C\_ : Hole position data

 $Z\_$  or  $X\_$  : The distance from point R to the bottom of the hole

R\_ : The distance from the initial level to point R level

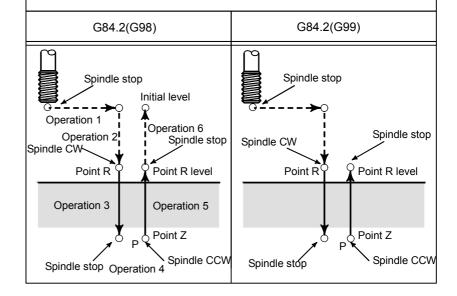
P\_ : Dwell time at the bottom of a hole

F\_ : Cutting feedrate

L\_ : Number of repeats (When it is needed.)

S\_ : Spindle speed

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



A G code cannot discriminate between front face tapping cycle and side face tapping cycle using Series 15 format commands. The drilling axis is determined by plane selection (G17/G18/G19). Specify the plane selection that becomes equivalent to front face tapping cycle or side face tapping cycle as appropriate. (When FXY (bit 0 of parameter No. 5101) is 0, the Z-axis is used as the drilling axis. When FXY is 1, plane selection is as follows.)

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

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

#### **Explanation**

Once positioning for the X-axis (G84) or Z-axis (G88) has been completed, the spindle is moved, by rapid traverse, to point R. Tapping is performed from point R to point Z, after which the spindle stops and observes a dwell time. Then, the spindle starts reverse rotation, retracts to point R, stops rotating, then moves to the initial level by rapid traverse.

During tapping, the feedrate override and spindle override are assumed to be 100%. For retraction (operation 5), however, a fixed override of up to 2000% can be applied by setting parameters DOV (No. 5200#4), OVU (No.5201#3), and No. 5211.

#### - Rigid mode

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

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

#### - Thread lead

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

#### - Series 15 format command

Rigid tapping can be performed using Series 15 format commands.

#### - Acceleration/deceleration after interpolation

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

#### - Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

#### - Override

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

- Extraction override
- Override signal

#### - Dry run

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

#### - Machine lock

Machine lock can be executed also in G84 (G88).

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

#### - Reset

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

#### - Interlock

Interlock can also be applied in G84 (G88).

#### - Feed hold and single block

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

#### - Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle."

With other manual operations, rigid tapping cannot be performed.

#### - Backlash compensation

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

Along the drilling axis, backlash compensation has been applied.

#### - C-axis clamp, C-axis unclamp

It is possible to specify an M code for mechanically fixing or releasing the C-axis during rigid tapping. Adding an M code for clamp to the G84 (G88) block outputs both M codes. Descriptions of timing are provided later.

An M code for clamp is set in parameter No. 5110. An M code for unclamp is assumed as follows depending on the setting of parameter No. 5110.

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

#### Limitation

#### - Axis switching

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

#### - S commands

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

#### <Example>

For a built-in motor equipped with a detector having a resolution of 4095 pulses per rotation, the maximum spindle speed during rigid tapping is as follows:

For a serial spindle

 $(32767 \times 1000 \div 8 \times 60) \div 4095 = 60012 \text{ (min}^{-1}) \text{ [Note: Ideal value]}$ 

#### - F commands

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

#### - Unit of F command

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

#### - M29

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

- 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 parameter MDL (No. 5431#0) 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.

- R

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

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P in an independent block.

#### **Example**

#### 4.4.2 Peck Rigid Tapping Cycle (G84 or G88)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.

In this cycle, cutting is performed several times until the bottom of the hole is reached. Two peck tapping cycles are available: High-speed peck tapping cycle and standard peck tapping cycle. These cycles are selected using the PCP bit (bit 5) of parameter 5200.

#### **Format**

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

G84 X(U)\_ C(H)\_Z(W)\_ R\_ P\_ Q\_ F\_ K\_ M\_; G88 Z(W)\_ C(H)\_X(U)\_ R\_ P\_ Q\_ F\_ K\_ M\_; X\_ C\_ or Z\_ C\_ : Hole position data Z or X: The distance from point R to the bottom of the hole R\_ : The distance from the initial level to point R level : Dwell time at the bottom of the hole Q : Depth of cut for each cutting feed : The cutting feedrate : Number of repeats (if required) : M code for C-axis clamp (if required) G84 or G88(G98 mode) G84 or G88(G99 mode) Spindle stop d = Amount of Spindle stop return Initial leve Spindle stop Spindle CW Spindle stop M(α+1) P2 Point R M(α+1) level Point R q q a q

Spindle CCW

Spindle stop

Spindle CCW

- High-speed peck rigid tapping cycle

In the first cutting from point R, perform cutting by depth "q" specified by address Q while rotating the spindle in the forward direction (operation <1>).

Then, perform returning by the amount specified by parameter No. 5213 while rotating the spindle in the reverse direction (operation <2>).

Then, perform tapping by (d+q) while rotating the spindle in the forward direction (operation <3>).

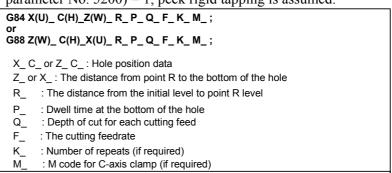
Repeat operations <2> and <3> until the bottom of the hole is reached.

The cutting speed and rigid tapping time constant are used for operations <1> and

For operation <2> and travel from the bottom of the hole (point Z) to point R, rigid tapping extract override is enabled and the rigid tapping extract time constant is used.

Spindle stop

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



G84 or G88(G99 mode)

- Peck rigid tapping cycle

In the first cutting from point R, perform cutting by depth "q" specified by address Q while rotating the spindle in the forward direction (operation <1>).

Then, return to point R by rotating the spindle in the reverse direction (operation <2>).

Then, rotate the spindle in the forward direction and perform cutting to the position indicated by [(Position to which cutting was performed previously) - (Cutting start distance set in parameter No. 5213)] as movement to the cutting start point (operation <3>).

Continue cutting by (d+q) (operation <4>).

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

The cutting speed and rigid tapping time constant are used for operations <1> and

For operations <2>, <3>, and travel from the bottom of the hole (point Z) to point R, rigid tapping extract override is enabled and

the rigid tapping extract time constant is The symbols in the figure above indicate the following operations.

used.

d = Cutting start distance Spindle stop Spindle stop Spindle CW Spindle stop Spindle stop Spindle CW oindle CW Spindle CW Spindle stop M(α+1) P2 Spindle stor M(α+1) P2 Point R Point R ₽d ₽d q q ∯d ∯d Spindle stop q q Spindle CCW Spindle CCW **⊉**d ∯d Point Z Spindle CCW Spindle stop Spindle CCW

- → : Positioning (Rapid traverse G00) Cutting feed (Linear interpolation G01)

G84 or G88(G98 mode)

Р1 : Dwell programmed by the address P command

: Output of the M code for C-axis clamp (The  $\alpha$  value is set in parameter No. 5110.)

 $M(\alpha+1)$ : Output of the M code for C-axis unclamp : Dwell set by parameter No.5111

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

#### **Explanation**

#### - Cutting start distance

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

#### - Amount of return

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

#### - Return speed

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

#### - Speed during cutting into the cutting start point

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

#### - Acceleration/deceleration after interpolation

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

#### - Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

#### - Override

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

- Extraction override
- Override signal

Details are given later.

#### - Dry run

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

#### - Machine lock

Machine lock can be executed also in G84 (G88).

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

#### - Reset

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

#### - Interlock

Interlock can also be applied in G84 (G88).

#### - Feed hold and single block

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

#### - Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle."

With other manual operations, rigid tapping cannot be performed.

#### - Backlash compensation

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

Along the drilling axis, backlash compensation has been applied.

#### Limitation

#### - Axis switching

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

#### - S commands

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

#### - Distribution amount for the spindle

The maximum distribution amount is as follows (displayed on diagnostic screen 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 larger than the upper limit for cutting feed will cause alarm PS0011 to be issued.

#### - Unit of F command

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

- M29

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

- P/Q

Specify P and Q in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data.

When Q0 is specified, the peck rigid tapping cycle is not performed.

- Cancel

Do not specify a G code of the 01 group (G00 to G03 or G60 (when the parameter MDL (No. 5431#0) 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.

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P\_ in an independent block.

## 4.4.3 Canned Cycle Cancel (G80)

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

#### NOTE

When the rigid tapping canned cycle is cancelled, the S value used for rigid tapping is also cleared (as if S0 is specified).

Accordingly, the S command specified for rigid tapping cannot be used in a subsequent part of the program after the cancellation of the rigid tapping canned cycle.

After canceling the rigid tapping canned cycle, specify a new S command as required.

## 4.4.4 Override during Rigid Tapping

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

- Extraction override
- Override signal

#### 4.4.4.1 Extraction override

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

#### **Explanation**

- Specifying the override in the parameter

Set bit 4 (DOV) of parameter No. 5200 to 1 and set the override in parameter No. 5211.

An override from 0% to 200% in 1% steps can be set. Bit 3 (OVU) of parameter No. 5201 can be set to 1 to set an override from 0% to 2000% in 10% steps.

#### - Specifying the override in a program

Set bit 4 (DOV) of parameter No. 5200 and bit 4 (OV3) of parameter No. 5201 to 1. The spindle speed at extraction can be specified in the program.

Specify the spindle speed at extraction using address "J" in the block in which rigid tapping is specified.

Example)

To specify 1000 min<sup>-1</sup> for S at cutting and 2000 min<sup>-1</sup> 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%.

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

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

	Parameter setting	DOV	= 1	DOV - 0
Command		OV3 = 1	OV3 = 0	DOV = 0
Spindle speed at extraction specified at address "J"	Within the range between 100% to 200%  Outside the range between 100% to 200%	Command in the program	Parameter No. 5211	100%
No spindle speed at extraction specified at address "J"		Parameter No. 5211	NO. 3211	

#### 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<sup>-1</sup>.
- 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<sup>-1</sup>.
- 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.

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

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

### 4.4.4.2 Override signal

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

- Applying override using the feedrate override signal (When the second feedrate override signal is enabled, 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.

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

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

## 4.5 CHAMFERING AND CORNER R

#### Overview

A chamfering or corner R block can automatically be inserted between linear interpolation (G01) along a single axis and that along a single axis normal to that single axis.

Chamfering or corner R is inserted for a command to move the tool along two axes on the plane determined by the plane selection (G17, G18, or G19) command.

#### **Format**

- Chamfering

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

	Format					
G17 plane: G18 plane: G19 plane:	G01 Z <sub>P</sub> (W)_ I(C) <u>±i</u> ;					
	Explanation	Tool movement				
X <sub>P</sub> (U) Y <sub>P</sub> (V) Z <sub>P</sub> (W) I(C)±i J(C)±k	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.  X <sub>P</sub> is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y <sub>P</sub> is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z <sub>P</sub> is the address of the Y-axis. Z <sub>P</sub> is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis. Specify the distance between points b and c in the figure shown at right with a sign following address I, J, K, or C. (Use I, J, or K when bit 4 (CCR) of parameter No. 3405 is set to 0 or C when the bit is set to 1.)	Positive direction along the second axis on the selected plane  a  Start point  Negative direction along the second axis on the selected plane  Moves from a to d and c.  (Positive direction along the second axis on the selected plane when a plus sign is specified at I, J, K, or C or negative direction when a minus sign is specified at I, J, K, or C)				

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

	Format					
G17 plane: G18 plane: G19 plane:	G01 X <sub>P</sub> (U)_ K(C) <u>±k</u> ;					
	Explanation	Tool movement				
X <sub>P</sub> (U) Y <sub>P</sub> (V) Z <sub>P</sub> (W) I(C) <u>世</u> J(C) <u>世</u> K(C) <u>±k</u>	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.  X <sub>P</sub> is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y <sub>P</sub> is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z <sub>P</sub> is the address of the Y-axis. Z <sub>P</sub> is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis. Specify the distance between points b and c in the figure shown at right with a sign following address I, J, K, or C. (Use I, J, or K when bit 4 (CCR) of parameter No. 3405 is set to 0 or C when the bit is set to 1.)	Moves from a to d and c.  (Positive direction along the first axis on the selected plane when a plus sign is specified at I, J, K, or C or negative direction when a minus sign is specified at I, J, K, or C)   a Start point  First axis on the selected plane  First axis on the selected plane				

#### - Corner R

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

	Forn	nat	
-	: G01 X <sub>P</sub> (U)_ R <u>±r;</u> : G01 Z <sub>P</sub> (W)_ R <u>±r;</u> : G01 Y <sub>P</sub> (V)_ R <u>±r;</u>		
	Explanation	Tool movement	
X <sub>P</sub> (U) Y <sub>P</sub> (V) Z <sub>P</sub> (W)	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.  X <sub>P</sub> is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y <sub>P</sub> is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z <sub>P</sub> is the address of the Y-axis. Z <sub>P</sub> is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis. Specify the radius of the arc connecting points d and c in the figure shown at right with a sign following address R.	Positive direction along the second axis on the selected plane  Negative direction along the second axis on the selected plane  Moves from a to d and c.  (Positive direction along the second axis on the selected plane when +r is specified at R or negative direction when -r is specified at R)	

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

Zp(VV) absolute of incremental	Tool movement loves from a to d and c.
X <sub>P</sub> (U) Specifies movement from Y <sub>P</sub> (V) point a to point b with an Z <sub>P</sub> (W) absolute or incremental	loves from a to d and c.
Y <sub>P</sub> (V) point a to point b with an Z <sub>P</sub> (W) absolute or incremental	
on the right.  X <sub>P</sub> is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y <sub>P</sub> is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z <sub>P</sub> is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.  R±r Specify the radius of the arc connecting points d	(Positive direction along the first axis on the selected plane when +r is specified at R or negative direction when -r is specified at R)  a Start point  First axis on the selected plane  Start point

#### **Explanation**

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

#### Example:

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

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

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

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

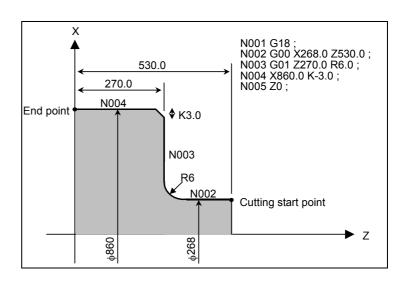
The following program also causes an alarm. (Because the block next to the chamfering command moves the tool along the X-axis, which is not on the selected plane)

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

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

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

#### **Example**



#### Limitation

#### - Alarms

In the following cases, an alarm is issued:

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

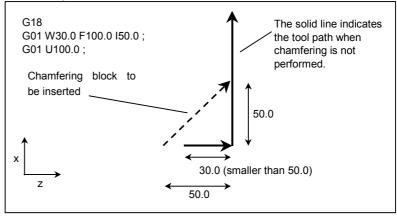


Fig. 4.5 (a) Example of machining which causes alarm PS0055

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

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

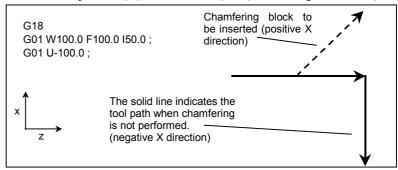


Fig. 4.5 (b) Example of machining which causes alarm PS0051

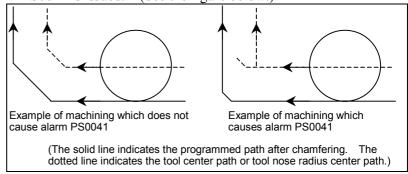
#### - Single block operation

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

#### - Cutter compensation or tool nose radius compensation

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

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



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

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

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

Chamfering or corner R can also be specified at address C or R.

# **4.6** MIRROR IMAGE FOR DOUBLE TURRET (G68, G69)

#### Overview

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

#### **Format**

G68: Double turret mirror image on

G69: Mirror image cancel

#### **Explanation**

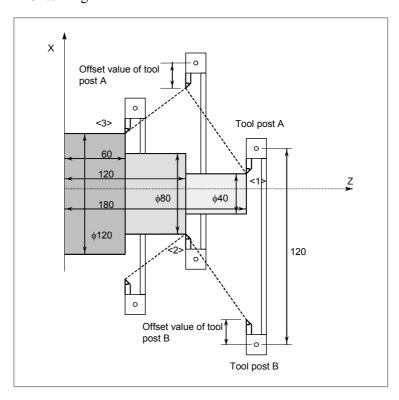
Mirror image can be applied to the X-axis of the three basic axes that is set by parameter No. 1022 with the G code command.

When G68 is designated, the coordinate system is shifted to the double turret side, and the X-axis sign is reversed from the programmed command to perform symmetrical cutting. This function is called the mirror image for double turret.

To use this function, set the distance between the two tool posts to a parameter (No. 1290).

#### **Example**

#### For turning



X40.0 Z180.0 T0101; Position tool post A at <1>

G68; Shift the coordinate system by the distance A to B

(120mm), and turn mirror image on.

X80.0 Z120.0 T0202; Position tool post B at <2>

G69; Shift the coordinate system by the distance B to A,

and cancel mirror image.

X120.0 Z60.0 T0101; Position tool post A at <3>

#### NOTE

A diameter value is specified for the X-axis.

#### Limitation

#### **NOTE**

This function cannot be used together with the balance cut function. If both option functions are specified, the operation depends on the combination of the systems as follows:

#### System in which this function is used

- One-path system
- One-path and loader-path system

#### System in which this function does not operate

System with two or more paths

# 4.7 DIRECT DRAWING DIMENSION PROGRAMMING

#### Overview

Angles of straight lines, chamfering value, corner R values, and other dimensional values on machining drawings can be programmed by directly inputting these values. In addition, the chamfering and corner R can be inserted between straight lines having an optional angle.

This programming is only valid in memory operation mode.

#### **Format**

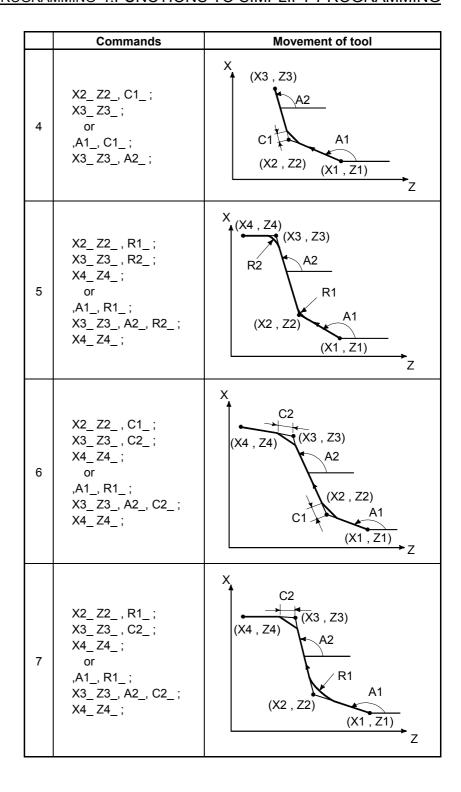
Examples of command formats for the G18 plane (ZX plane) are shown. This function can be specified in the following formats also for the G17 plane (XY plane) and G19 plane (YZ plane).

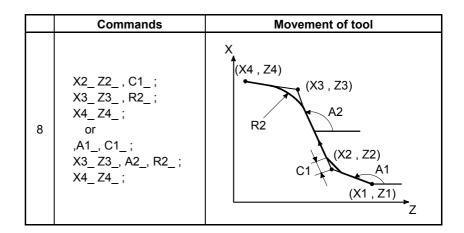
The following formats are changed as follows:

For the G17 plane:  $Z \rightarrow X$ ,  $X \rightarrow Y$ For the G19 plane:  $Z \rightarrow Y$ ,  $X \rightarrow Z$ 

Table 4.7 (a) Commands table

	Commands	Movement of tool
1	X2_ (Z2_), A_ ;	X (X2, Z2) A (X1, Z1) Z
2	,A1_; X3_Z3_, A2_;	X (X3, Z3) A2 (X1, Z1) Z
3	X2_Z2_, R1_; X3_Z3_; or ,A1_, R1_; X3_Z3_, A2_;	X (X3, Z3) R1 (X2, Z2) (X1, Z1) Z





#### **Explanation**

A program for machining along the curve shown in Fig. 4.7 (a) is as follows:

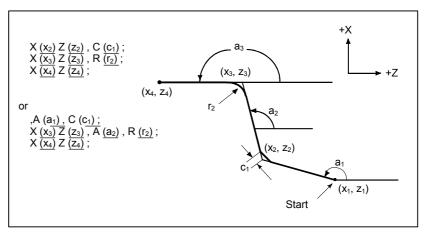


Fig. 4.7 (a) Machining Drawing (example)

For command a straight line, specify one or two out of X, Z, and A. If only one is specified, the straight line must be primarily defined by a command in the next block.

To command the degree of a straight line or the value of chamfering or corner R, command with a comma (,) as follows:



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

A\_ C\_ R

#### - Command using a supplement

When bit 5 (DDP) of parameter No. 3405 is set to 1, an angle can be specified using a supplement.

There is the following relationship, assuming that the supplement is A' and the actual specified angle is A:

$$A = 180 - A'$$

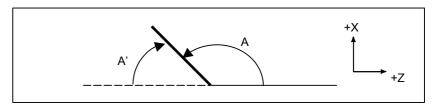


Fig. 4.7 (b) Supplement

#### Limitation

#### NOTE

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

#### NOTE

- The angle allowance in calculating the point of intersection in the program below is ±1°.
   (Because the travel distance to be obtained in this calculation is too large.)
  - (a) X\_,A\_; (If a value within 0°±1° or 180°±1° is specified for the angle instruction A, the alarm PS0057 occurs.)
  - (b) Z\_ ,A\_ ; (If a value within 90°±1° or 270°±1° is specified for the angle instruction A, the alarm PS0057 occurs.)
- 7 An alarm PS0058 occurs if the angle made by the 2 lines is within  $\pm 1^{\circ}$  when calculating the point of intersection.
- 8 Chamfering or corner R is ignored if the angle made by the 2 lines is within  $\pm 1^{\circ}$ .
- 9 Both a dimensional command (absolute programming) and angle instruction must be specified in the block following a block in which only the angle instruction is specified.

(Example)

In addition to the dimensional command, angle command must be specified in block No. 3. If the angle command is not specified, alarm PS0056 is issued. If the coordinates are not specified with an absolute programming, alarm PS0312 is issued.

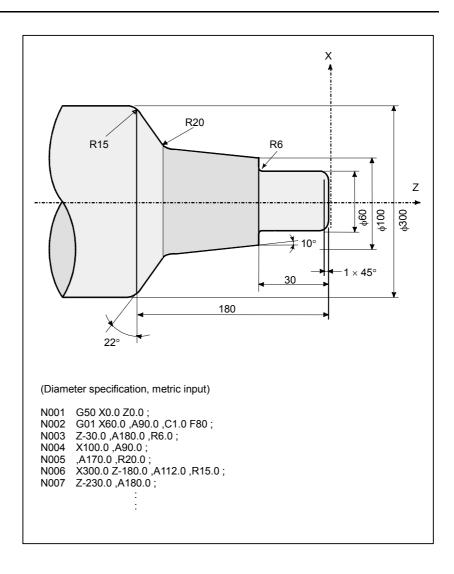
- 10 In the tool nose radius compensation mode, a block in which only the angle command is specified in direct drawing dimension programming is assumed to be a block with no move command. For details of compensation when sequential blocks with no move command are specified, see the explanation of tool nose radius compensation.
- 11 If two or more blocks with no move command are specified between sequential commands of direct drawing dimension programming, alarm PS0312 is issued.
- 12 When bit 4 (CCR) of parameter No. 3405 is set to 1, address A in the G76 (combined threading cycle cycle) block specifies the tool nose angle.

  When A or C is used as an axis name, it cannot be used in the angle or chamfering command in direct drawing dimension programming. Use ,A\_ or ,C\_ (when bit 4 (CCR) of parameter No. 3405 is set to 0).

#### NOTE

13 In a multiple repetitive cycle, in blocks with sequence numbers between those specified at P and Q, a program using direct drawing dimension programming can be used. The block with the last sequence number specified at Q must not be an intermediate block of these specified blocks.

#### **Example**



# **COMPENSATION FUNCTION**

This chapter describes the following compensation functions:

- 5.1 TOOL OFFSET
- 5.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)
- 5.3 OVERVIEW OF CUTTER COMPENSATION (G40-G42)
- 5.4 DETAILS OF CUTTER OR TOOL NOSE RADIUS **COMPENSATION**
- 5.5 VECTOR RETENTION (G38)
- 5.6 CORNER CIRCULAR INTERPOLATION (G39)
- 5.7 EXTENDED TOOL SELECTION
- 5.8 AUTOMATIC TOOL OFFSET (G36, G37)
- 5.9 COORDINATE SYSTEM ROTATION (G68.1, G69.1)
- 5.10 ACTIVE OFFSET VALUE CHANGE FUNCTION BASED ON MANUAL FEED

# 5.1 TOOL OFFSET

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool).

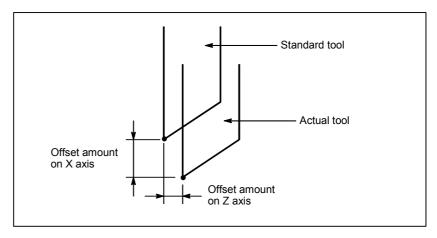


Fig. 5.1 (a) Tool offset

# **5.1.1** Tool Geometry Offset and Tool Wear Offset

Tool geometry offset and tool wear offset are possible to divide the tool offset to the tool geometry offset for compensating the tool shape or tool mounting position and the tool wear offset for compensating the tool nose wear. The tool geometry offset value and tool wear offset value can be set individually. When these values are not distinguished from each other, the total of the values is set as the tool offset value.

#### **NOTE**

Tool geometry offset and tool wear offset are optioned.

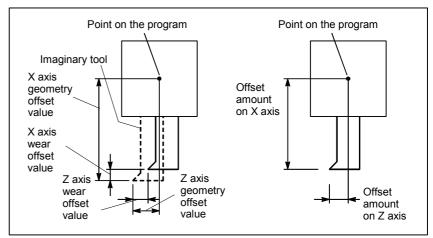


Fig. 5.1.1 (a) If tool geometry offset and tool wear offset are distinguished from each other (left) and if they are not (right)

#### 5.1.2 T Code for Tool Offset

#### **Format**

Select a tool with a numeric value after a T code. A part of the numeric value is used as a tool offset number for specifying data such The following selections can be made as a tool offset value. according to the specification method and parameter setting:

Meaning o	of T code (*1)	Parameter setting for specifying of
LGN(No.5002#1)=0	LGN(No.5002#1)=1	offset No. (*2)
T x x x x x x x y xxxxxxxx : Tool selection	T x x x x x x x y xxxxxxxx : Tool selection and tool	A tool wear offset number is specified using the lower one digit of a T code.
y : Tool wear and tool geometry offset	geometry offset y : Tool wear offset	When parameter No. 5028 is set to 1
T <u>xxxxxx</u> <u>yy</u>	T <u>xxxxxx</u> <u>yy</u>	A tool wear offset number is specified
xxxxxx : Tool selection	xxxxxx : Tool selection and tool	using the lower two digits of a T code.
yy : Tool wear and tool geometry offset	geometry offset yy : Tool wear offset	When parameter No. 5028 is set to 2
T <u>xxxxx</u> <u>yyy</u>	Txxxxx yyy	A tool wear offset number is specified
xxxxx : Tool selection	xxxxx : Tool selection and tool	using the lower three digits of a T code.
yyy : Tool wear and tool geometry offset	geometry offset yyy : Tool wear offset	When parameter No. 5028 is set to 3

- The maximum number of digits of a T code can be specified
- using parameter No. 3032. (1 to 8 digits)
  When parameter No. 5028 is set to 0, the number of digits of a T code used for offset number specification depends on the number of tool offsets.

Example)

When the number of tool offsets is 1 to 9: Lower one digit When the number of tool offsets is 10 to 99: Lower two digits When the number of tool offsets is 100 to 999: Lower three digits

#### 5.1.3 **Tool Selection**

Tool selection is made by specifying the T code corresponding to the tool number. Refer to the machine tool builder's manual for the relationship between the tool selection number and the tool.

#### 5.1.4 **Offset Number**

Tool offset number has two meanings. It is specifies the offset distance corresponding to the number that is selected to begin the offset function. A tool offset number of 0 indicates that the offset amount is 0 and the offset is cancelled.

#### **5.1.5** Offset

#### **Explanation**

#### - Offset methods

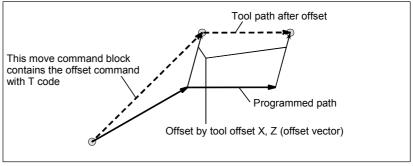
Two methods are available to geometry offset and wear compensation, offset with tool movement and compensation with coordinate shift. Which offset method to select can be specified with parameters LWT (No. 5002#2=0) and LGT (No. 5002#4=0). If, however, the tool geometry and wear offset options are not provided, offsetwith tool movement is assumed unconditionally.

Tool geometry	Compensation		Para	meter	
and wear offset options	element	LWT=0 LGT=0	LWT=1 LGT=0	LWT=0 LGT=1	LWT=1 LGT=1
Not provided	Wear and geometry not distinguished		Tool mo	ovement	
Provided	Wear compensation	Tool movement	Coordinate shift	Tool movement	Coordinate
	Geometry compensation	Coordinate shift	Coordinate shift	Tool movement	Tool movement

#### - Offset with tool movement

The tool path is offset by the X, Y, and Z tool offset values for the programmed path. The tool offset distance corresponding to the number specified by the T code is added to or subtracted from the end position of each programmed block.

The vector with tool offset X, Y, and Z is called the offset vector. Offset is the same as the offset vector.



Offset operation with tool movement

#### **NOTE**

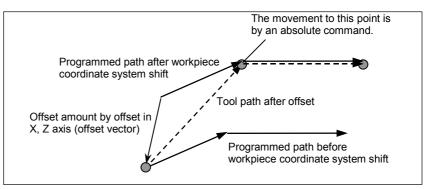
1 When G50 X\_Z\_T\_; is specified, the tool is not moved.

The coordinate system in which the coordinate value of the tool position is (X,Z) is set. The tool position is obtained by subtracting the offset value corresponding to the tool offset number specified in the T code

2 The G codes in the 00 group other than G50 must not be specified in the same block as that containing a T code.

#### - Offset with coordinate shift

The workpiece coordinate system is shifted by the X, Y, and Z tool offset amounts. Namely, the offset amount corresponding to the number designated with the T code is added to or subtracted from the absolute coordinates.



Offset operation with coordinate shift

#### - Starting and canceling offset by specifying a T code

Specifying an tool offset number with a T code means to select the tool offset value corresponding to it and to start offset. Specifying 0 as a tool offset number means to cancel offset.

For offset with tool movement, whether to start or cancel the offset can be specified with parameter LWN (No. 5002#6). For compensation with coordinate shift, the offset is started and canceled when a T code is specified. For the cancellation of geometry compensation, its operation can be selected with LGC (No. 5002#5).

Offset method	LWM (No.5002#6)=0 LWM (No.5002#6)=1				
Tool movement	When a T code is appoified	When an axial movement			
Tool movement	When a T code is specified	is specified			
	When a T code is specified				
Coordinate shift	Coordinate shift (Note that geometry offset can be canceled on				
	(No. 5002#5) = 1.)				

#### - Canceling offset with reset

Tool offset is canceled under one of the following conditions:

- <1> The power to the CNC is turned off and turned back on
- <2> The reset button on the MDI unit is pressed
- <3> A reset signal is input from the machine to the CNC

In cases <2> and <3> above, it is possible to select a cancel operation using parameters LVC (No. 5006#3) and TGC (No. 5003#7).

			Parai	meter	
Offset method		LVC=0 TGC=0	LVC=1 TGC=0	LVC=0 TGC=1	LVC=1 TGC=1
	Wear offset		0		0
Tool movement	Geometry offset	x	(When axial movement is specified)	x	(When axial movement is specified)
Coordinate	Wear offset	Х	0	Х	0
Coordinate shift	Geometry offset	х	х	0	О

o: Canceled.

x: Not canceled.

### **Example**

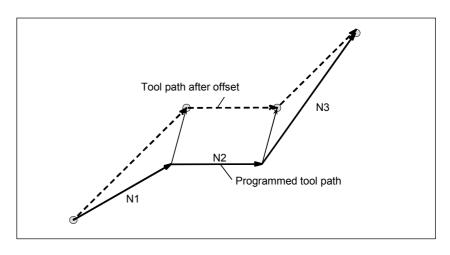
N1 X60.0 Z50.0 T0202; Creates the offset vector corresponding to

tool offset number 02.

N2 Z100.0;

N3 X200.0 Z150.0 T0200; Cancels the offset vector with offset number

0.



#### Limitation

- Helical interpolation (G02, G03)

Tool offset cannot be specified in a block in which helical interpolation is used.

- Coordinate system rotation (G68.1)

Coordinate system rotation is executed on the command program first, followed by tool offset.

- Three-dimensional coordinate conversion (G68.1)

If tool offset is used, tool offset with coordinate shift cannot be used. Offset with tool movement must be specified inside a nest of three-dimensional coordinate conversion.

```
Example) G68.1 ...;
T0101;
:
T0100;
G69.1 ...;
```

- Workpiece coordinate system preset (G50.3)

Performing workpiece coordinate system preset causes tool offset with tool movement to be canceled; this does not cause tool offset with coordinate shift to be canceled.

- Machine coordinate system setting (G53), reference position return (G28), second, third, and fourth reference position return (G30), floating reference position return (G30.1), and manual reference position return

Basically, before performing these commands or operations, cancel tool offset. These operations do not cause tool offset to be canceled. The following actions take place:

	When the command or operation is specified	When the next axial movement command is specified
Tool movement	The tool offset value is temporarily canceled.	The tool offset value is reflected.
Coordinate shift	Coordinates with the tool offset value reflected are assumed.	Coordinates with the tool offset value reflected are assumed.

## 5.1.6 Y Axis Offset

#### Overview

When the Y axis, one of the basic three axes, is used with a lathe system, this function performs Y axis offset.

If the tool geometry and wear offset options are provided, both tool geometry offset and tool wear offset are effective to Y axis offset.

#### **Explanation**

Y axis offset results in the same operation as tool offset. For an explanation of the operation, related parameters, and the like, refer to the item "Tool Offset."

# 5.1.7 Second Geometry Tool Offset

#### Overview

To compensate for a difference in tool attachment position or selection position, this function adds second geometry tool offset to the X axis, Y axis, and Z axis with all paths.

In contrast to this offset, the ordinary tool geometry offset is called the first geometry tool offset.

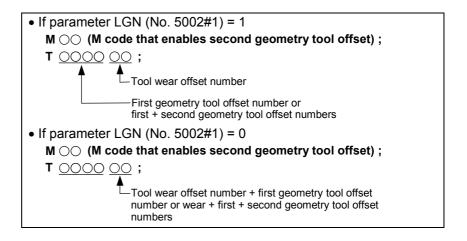
It is possible to apply a tool offset value (tool wear offset + tool geometry offset) in the reverse direction, using an appropriate signal. This function may be used if the offset value differs even with a single

tool for a mechanical reason depending on the mounting position (inside/outside) or the selected position (right/left).

#### **NOTE**

- 1. To use the second geometry tool offset, the geometry and wear offset options are required.
- 2. To use the second geometry tool offset for the Y axis, the Y axis offset option is required.

#### **Format**



#### **Explanation**

- Specification method

Offset using the second geometry tool offset is performed with a program command.

Specify the second geometry tool offset with a T code command, and using the second geometry tool offset signal G2SLC, specify whether to set the offset value of the specified tool geometry offset number to the first geometry tool offset only or to the first geometry tool offset plus the second geometry tool offset. If using the first geometry tool offset plus the second geometry tool offset, specify the axis to which the second geometry tool offset is to be applied, using the appropriate one of the second geometry tool offset axis select signal G2X, G2Z, and G2Y.

Generally, before a T code command, specify the M code to enable the second geometry tool offset. For details, refer to the manual supplied by the machine tool builder. The same number as the first geometry tool offset number is always selected as the second geometry tool offset number.

During execution, the tool offset value for each axis is either of the following:

- First geometry tool offset value + tool wear offset value
- First geometry tool offset value + second geometry tool offset value + tool wear offset value

#### Example)

- The code is a 4-digit code. (The number of digits of a T code is specified with parameter No. 3032.)
- The offset type is tool movement (parameter LGT (No. 5002#4) = 1).
- The lower two digits of the T code is the tool geometry offset number (parameter No. 5028 = 2).
- Offset is performed when the T code block is executed (parameter LWM (No. 5002#6) = 0).
- The X axis data for first geometry tool offset number 1 is 1.000.
- The X axis data for second geometry tool offset number 1 is 10.000.
- The tool geometry offset number is specified with the tool selection number (parameter LGN (No. 5002#1) = 1).
- Signals G2SLC = '1', G2X = '1', and G2Z = G2Y = '0'

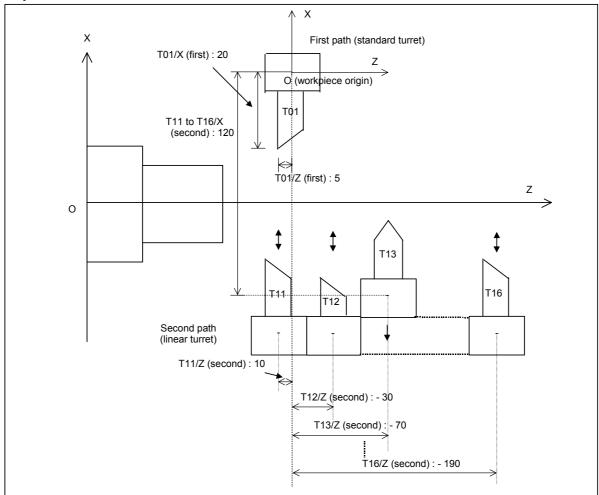
If T0102 is specified under the above conditions, the upper two digits 01 of the T code cause the first and second geometry tool offset numbers 1 to be selected, so that the absolute and machine coordinates will be 11.000 on the X axis only.

#### - Offset data

The data for the second geometry tool offset can be set for each path. The number of items of the data can be set with parameter No. 5024. The data remains stored even after the power is turned off.

If the data is to be made common to paths, use the memory common with both 2 path.

#### **Example**



In the machine configuration shown in the above figure, for the offset data for the tool mounted to the first path (standard turret), set the offset data of the tool itself as the first geometry tool offset data. (The second geometry tool offset data is 0.) For the offset data for the tool mounted to the second path (linear turret), set the offset data of the tool itself as the first geometry tool offset data and the offset data from the workpiece origin at the mounting position as the second geometry tool offset data. Usually, the offset data of the tool itself is measured separately from the offset data at the mounting position, these data can be set separately by using the second geometry tool offset.

First	First geometry tool offset			nd geometry t	ool offset
No.	X axis	Z axis	No.	X axis	Z axis
01	20.000	5.000	01	0.000	0.000
:	:	:	:	:	:
10	25.000	8.000	10	0.000	0.000
11	-20.000	5.000	11	120.000	10.000
12	-10.000	3.000	12	120.000	-30.000
13	-15.000	0.000	13	120.000	-70.000
:	:		:	:	:
16	-18.000	7.000	16	120.000	-190.000

# **5.2** OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

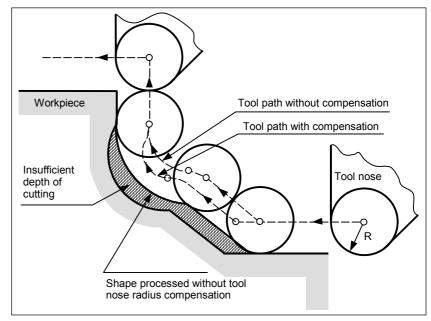


Fig 5.2 (a) Tool path of tool nose radius compensation

# **5.2.1** Imaginary Tool Nose

The tool nose at position A in Fig. 5.2.1 (a) does not actually exist.

The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start point than the imaginary tool nose.

Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.

The position relationship when the tool is set to the start point is shown in Fig. 5.2.1 (a).

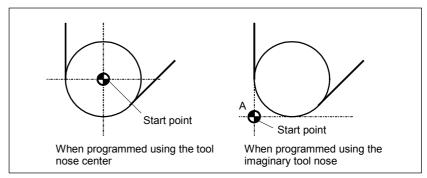


Fig. 5.2.1 (a) Tool nose radius center and imaginary tool nose

#### ⚠ CAUTION

In a machine with reference positions, a standard position like the turret center can be placed over the start point. The distance from this standard position to the nose radius center or the imaginary tool nose is set as the tool offset value.

Setting the distance from the standard position to the tool nose radius center as the offset value is the same as placing the tool nose radius center over the start point, while setting the distance from the standard position to the imaginary tool nose is the same as placing the imaginary tool nose over the standard position. To set the offset value, it is usually easier to measure the distance from the standard position to the imaginary tool nose than from the standard position to the tool nose radius center.

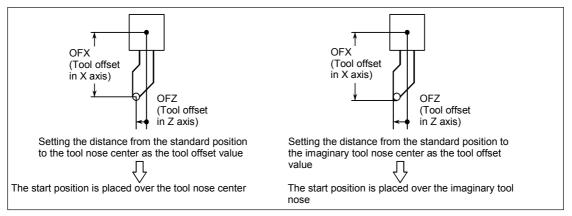


Fig. 5.2.1 (b) Tool offset value when the turret center is placed over the start point

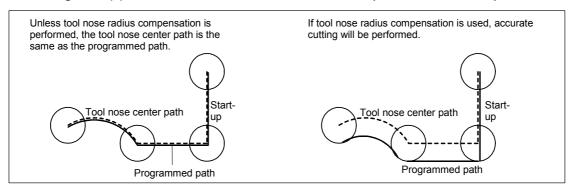


Fig. 5.2.1 (c) Tool path when programming using the tool nose center

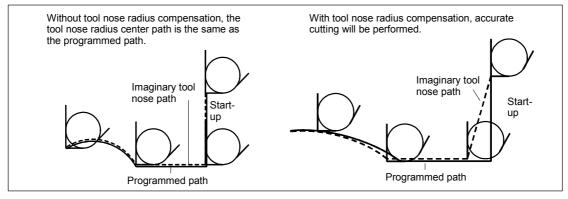


Fig. 5.2.1 (d) Tool path when programming using the imaginary tool nose

# **5.2.2** Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose center is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values.

The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig. 5.2.2 (a) below together with their corresponding codes. This Fig 5.2.2 (a) illustrates the relation between the tool and the start point. The following apply when the tool geometry offset and tool wear offset option are selected.

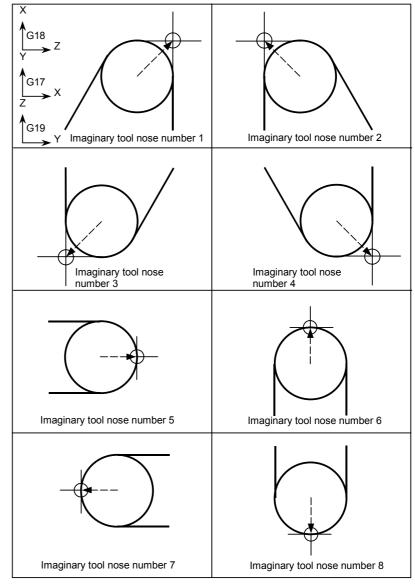
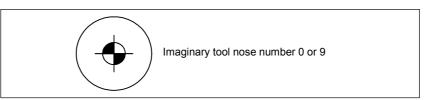


Fig. 5.2.2 (a) Direction of imaginary tool nose

Imaginary tool nose numbers 0 and 9 are used when the tool nose center coincides with the start point. Set imaginary tool nose number to address OFT for each offset number.

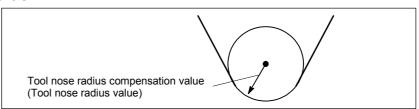
Bit 7 (WNP) of parameter No. 5002 is used to determine whether the tool geometry offset number or the tool wear offset number specifies the direction of the virtual tool nose for tool nose radius compensation.



## **5.2.3** Offset Number and Offset Value

#### **Explanation**

- Offset number and offset value



When the tool geometry compensation and tool wear compensation are not provided, offset values become as follows:

Table 5.2.3 (a) Offset number and offset value (example)

Offset number Up to 999 sets	OFX (Offset value on X axis)	OFZ (Offset value on Z axis)	OFR (Tool nose radius compensa- tion value)	OFT (Direction of imaginary tool nose)	OFY (Offset value on Y axis)
001	0.040	0.020	0.200	1	0.030
002	0.060	0.030	0.250	2	0.040
003	0.050	0.015	0.120	6	0.025
004	:	:	:	:	:
005	:	:	:	:	:
:	:	:	:	:	:

When the tool geometry compensation and tool wear compensation are provided, the offset values become as follows:

Table 5.2.3 (b) Tool geometry offset (example)

Geometry offset number	OFGX (X-axis geometry offset amount)	OFGZ (Z-axis geometry offset amount)	OFGR (Tool nose radius geometry offset value)	OFT (Imaginary tool nose direction)	OFGY (Y-axis geometry offset amount)
G001	10.040	50.020	0	1	70.020
G002	20.060	30.030	0	2	90.030
G003	0	0	0.200	6	0
G004	:	:	:	•	:
G005	:	:	:	:	:
:	:	:		•	:

Table 5.2.3 (c) Tool	gcomeny	Onset	(example)
	_		

Wear offset number	OFWX (X-axis wear offset amount)	OFWZ (Z-axis wear offset amount)	OFWR (Tool nose radius wear offset value)	OFT (Imaginary tool nose direction)	OFWY (Y-axis wear offset amount)
W001	0.040	0.020	0	1	0.010
W002	0.060	0.030	0	2	0.020
W003	0	0	0.200	6	0
W004	:	:	:	:	:
W005	:	:	:	:	:
:	:	:	•	:	:

#### - Tool nose radius compensation

In this case, the tool nose radius compensation value is the sum of the geometry and wear offset values.

OFR=OFGR+OFWR

#### - Imaginary tool nose direction

The imaginary tool nose direction is common to geometry and wear offsets.

#### - Command of offset value

A offset number is specified with the same T code as that used for tool offset.

#### **NOTE**

When the geometry offset number is made common to the tool selection by the parameter LGN (No.5002#1) setting and a T code for which the geometry offset and wear offset number differ from each other is designated, the imaginary tool nose direction specified by the geometry offset number is valid.

Example) T0102 OFR=OFGR $_{01}$ +OFWR $_{02}$  OFT=OFT $_{01}$ 

By setting parameter WNP (No. 5002#7) appropriately, the imaginary tool nose direction specified with the wear offset number can be made valid.

#### - Setting range of offset value

The range of values that can be set as a compensation value is either of the following, depending on the parameters OFE, OFD, OFC, and OFA (No. 5042#3 to No. 5042#0).

Valid compensation range (metric input)

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

Valid compensation range (inch input)

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

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

# **5.2.4** Workpiece Position and Move Command

In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified.

G code	Workpiece position	Tool path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side the programmed path
G42	Left side	Moving on the right side the programmed path

The tool is offset to the opposite side of the workpiece.

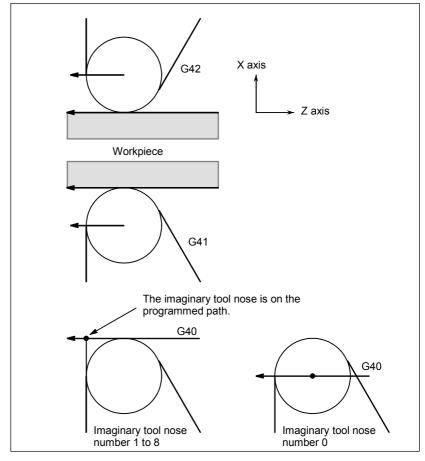


Fig. 5.2.4 (a) Workpiece position

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

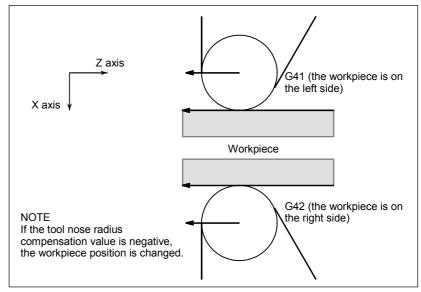


Fig. 5.2.4 (b) When the workpiece position is changed

G40, G41, and, G42 are modal.

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

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

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

#### **⚠** CAUTION

If the sign of the compensation value is changed from plus to minus and vice versa, the offset vector of tool nose radius compensation is reversed, but the direction of the imaginary tool tip does not change. For a use in which the imaginary tool tip is adjusted to the starting point, therefore, do not change the sign of the compensation value for the assumed program.

#### **Explanation**

#### - Tool movement when the workpiece position does not change

When the tool is moving, the tool nose maintains contact with the workpiece.

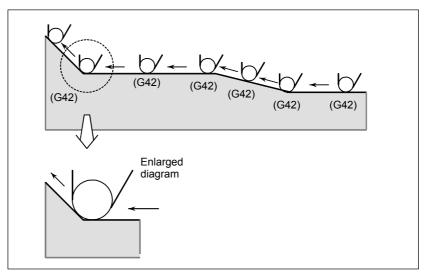


Fig. 5.2.4 (c) Tool movement when the workpiece position does not change

#### - Tool movement when the workpiece position changes

The workpiece position against the tool changes at the corner of the programmed path as shown in the following figure.

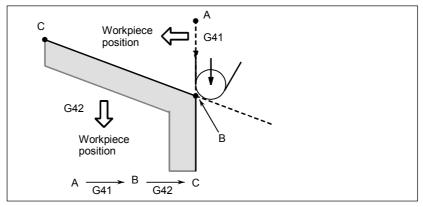


Fig. 5.2.4 (d) Tool movement when the workpiece position changes

Although the workpiece does not exist on the right side of the programmed path in the above case, the existence of the workpiece is assumed in the movement from A to B. The workpiece position must not be changed in the block next to the start-up block. In the above example, if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

#### - Start-up

The block in which the mode changes to G41 or G42 from G40 is called the start-up block.

G40\_;

G41 \_; (Start-up block)

Transient tool movements for offset are performed in the start-up block. In the block after the start-up block, the tool nose center is positioned Vertically to the programmed path of that block at the start point.

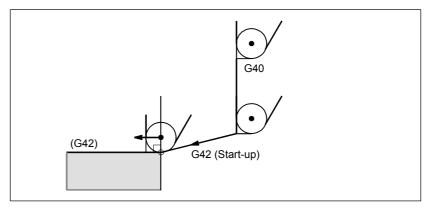


Fig. 5.2.4 (e) Start-up

#### - Offset cancel

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block.

G41\_;

G40 ; (Offset cancel block)

The tool nose center moves to a position vertical to the programmed path in the block before the cancel block.

The tool is positioned at the end position in the offset cancel block (G40) as shown below.

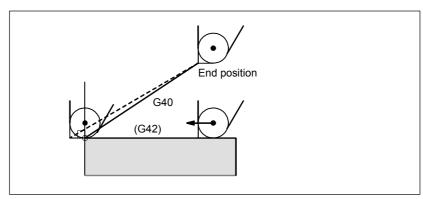


Fig. 5.2.4 (f) Offset cancel

#### - Changing the compensation value

In general, the compensation value is to be changed when the tool is changed in offset cancel mode. If the compensation value is changed in offset mode, however, the vector at the end point of the block is calculated using the compensation value specified in that same block. The same applies if the imaginary tool nose direction and the tool offset value are changed.

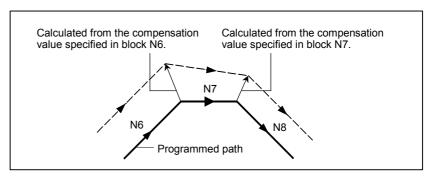


Fig. 5.2.4 (g) Changing the compensation value

#### - Specification of G41/G42 in G41/G42 mode

When a G41 or G42 code is specified again in G41/G42 mode, the tool nose center is positioned vertical to the programmed path of the preceding block at the end position of the preceding block.

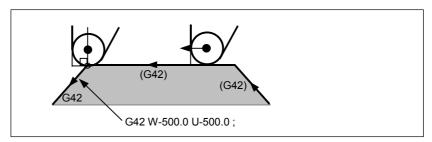


Fig. 5.2.4 (h) Specification of G41/G42 in G41/G42 mode

In the block that first changes from G40 to G41/G42, the above positioning of the tool nose center is not performed.

 Tool movement when the moving direction of the tool in a block which includes a G40 (offset cancel) command is different from the direction of the workpiece

When you wish to retract the tool in the direction specified by X(U) and Z(W) canceling the tool nose radius compensation at the end of machining the first block in the figure below, specify the following:

$$G40 X(U) _Z(W) _I _K _;$$

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

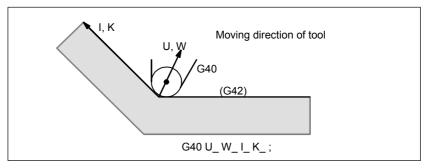


Fig. 5.2.4 (i) If I and K are specified in the same block as G40

Thus, this prevents the tool from overcutting, as shown in Fig. 5.2.4 (j).

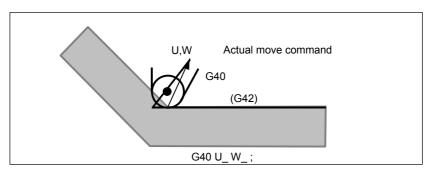


Fig. 5.2.4 (j) Case in which overcutting occurs in the same block as G40

The workpiece position specified by addresses I and K is the same as that in the preceding block.

Specify I\_K\_; in the same block as G40. If it is specified in the same block as G02 or G03, it is assumed to be the center of the arc.

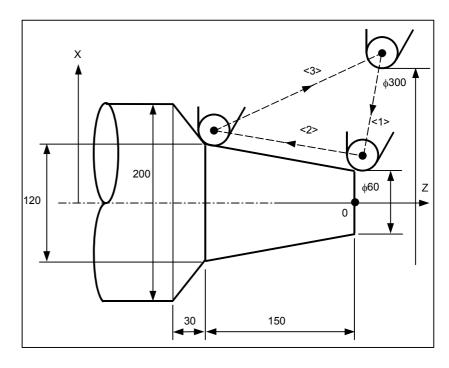
G40 X_ Z_ I_ K_ ;	Tool nose radius compensation
G02 X_ Z_ I_ K_ ;	Circular interpolation

If I and/or K is specified with G40 in the cancel mode, the I and/or K is ignored. The numeral is followed I and K should always be specified as radius values.

G40 G01 X Z ;

G40 G01 X Z I K ; Offset cancel mode (I and K are ineffective.)

# Example



### (G40 mode)

- <1> G42 G00 X60.0;
- <2> G01 X120.0 W-150.0 F10;
- <3> G40 G00 X300.0 W150.0 I40.0 K-30.0;

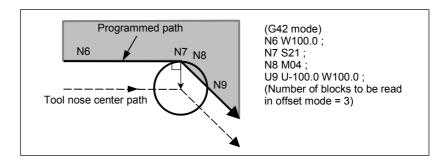
# **5.2.5** Notes on Tool Nose Radius Compensation

### **Explanation**

• Blocks without a move command that are specified in offset mode

If the number of such blocks consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)), the tool arrives at the position vertical to this block at the end point of the previous block.

If the feed distance is 0 (<5>), this applies even if only one block is specified.

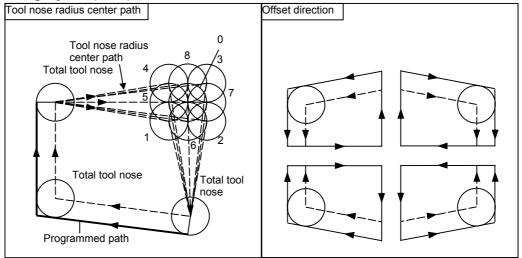


Overcutting may, therefore, occur in the above figure.

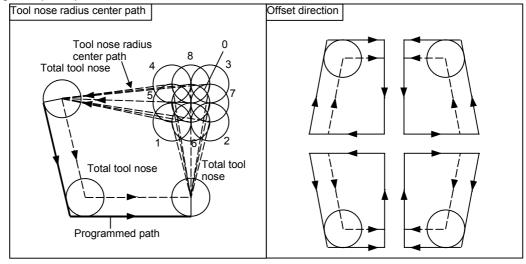
### - Tool nose radius compensation with G90 or G94

The tool nose center path and the offset direction are as shown below if tool nose radius compensation is applied. At the cycle start point, the offset vector disappears, and offset starts up with tool movement from the cycle start point. In addition, during a return to the cycle start point, the offset vector disappears temporarily, and offset is applied again with the next move command. The offset direction is determined by the cutting pattern, regardless of G41 or G42.

### - Outer/inner turning cycle (G90)



### - End cutting cycle (G94)



### - Difference from Series 16i/18i/21i

### NOTE

The offset direction is the same as that of Series 16*i*/18*i*/21*i*, but the tool nose radius center path is different.

For this CNC

The operation is the same as that performed if the canned cycle operation is replaced with G00 or G01, start-up is performed in the first block for movement from the start point, and offset cancel is performed in the last block for returning to the start point.

• For Series 16*i*/18*i*/21*i* 

The operation with the block for movement from the start point and the last block for returning to the start point differs from that of this CNC. For details, refer to the Series 16i/18i/21i Operator's Manual.

### - Tool nose radius compensation with G71 to G73

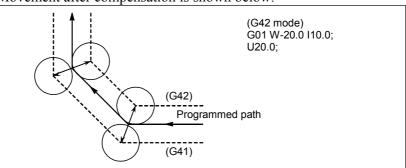
Tool nose radius compensation performed with G71 (outer surface rough cutting cycle or traverse grinding cycle), G72 (end rough cutting cycle or traverse direct constant-size grinding cycle), and G73 (closed loop cutting cycle or oscillation direct constant-size grinding cycle), see the explanations of the respective cycles.

### - Tool nose radius compensation with G74 to G76 and G92

With G74 (end cutting off cycle), G75 (outer/inner surface cutting off cycle), G76 (combined threading cycle cycle), and G92 (threading cycle), tool nose radius compensation cannot be applied.

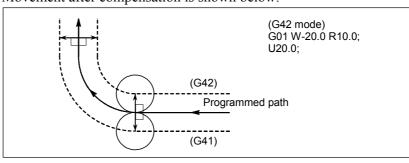
### - Tool nose radius compensation when chamfering is performed

Movement after compensation is shown below.



### - Tool nose radius compensation when a corner arc is inserted

Movement after compensation is shown below.



# **5.3** OVERVIEW OF CUTTER COMPENSATION (G40-G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 5.3 (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.

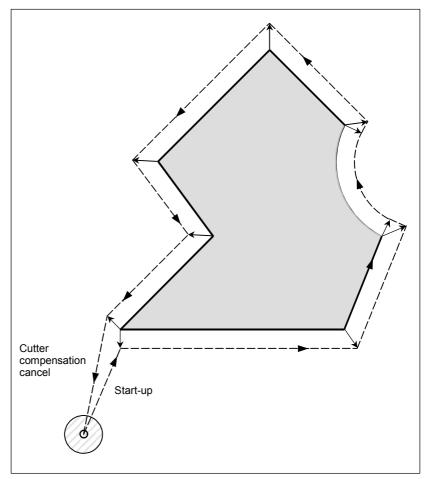


Fig. 5.3 (a) Outline of cutter compensation

### **Format**

### - Start up (tool compensation start)

### G00(or G01)G41(or G42) IP\_T\_;

G41 : Cutter compensation left (Group 07)G42 : Cutter compensation right (Group 07)

IP\_ : Command for axis movementT : Same T code as that of tool offset

### - 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) is specified in the offset cancel mode, the CNC enters the offset mode.

Moving the tool with this command is called start-up.

Specify positioning (G00) or linear interpolation (G01) for start-up. If circular interpolation (G02, G03) or involute interpolation (G02.2, G03.2) is specified, alarm PS0034 occurs.

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

### - Offset mode

In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03). If three or more blocks that move the tool cannot be read in offset mode, the tool may make either an excessive or insufficient cut.

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

### - Offset mode cancel

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

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

When performing offset cancel, circular arc commands (G02 and G03) and involute commands (G02.2 and G03.2) are not available. If these commands are specified, an 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.

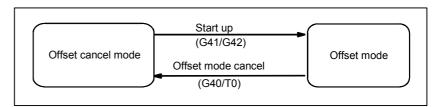


Fig. 5.3 (b) Changing the offset mode

### - 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.

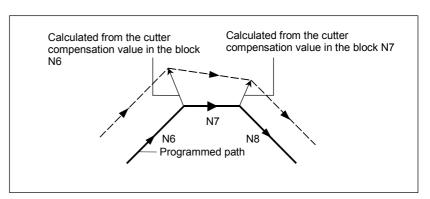


Fig. 5.3 (c) Changing the cutter compensation value

### - 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. 5.3 (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.)

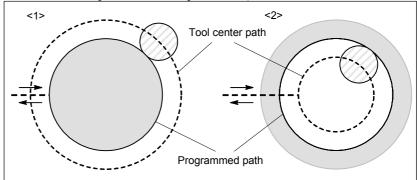


Fig. 5.3 (d) Tool center paths when positive and negative cutter compensation values are specified

### - Cutter compensation value setting

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

### **NOTE**

The cutter compensation value for which the T code corresponds to 0 always means 0.

It is not possible to set the cutter compensation value corresponding to T0.

### - 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 parameters OFE, OFD, OFC, and OFA (No. 5042 #3 to #0).

Valid compensation range (metric input)

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

Valid compensation range (inch input)

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

The compensation value corresponding to offset No. 0 always means 0

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

### - Offset vector

The offset vector is the two dimensional vector that is equal to the cutter compensation value assigned by T 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

Specifying a cutter compensation value is accomplished by specifying an offset number, using the same T code as that for specifying tool offset.

### - Plane selection and vector

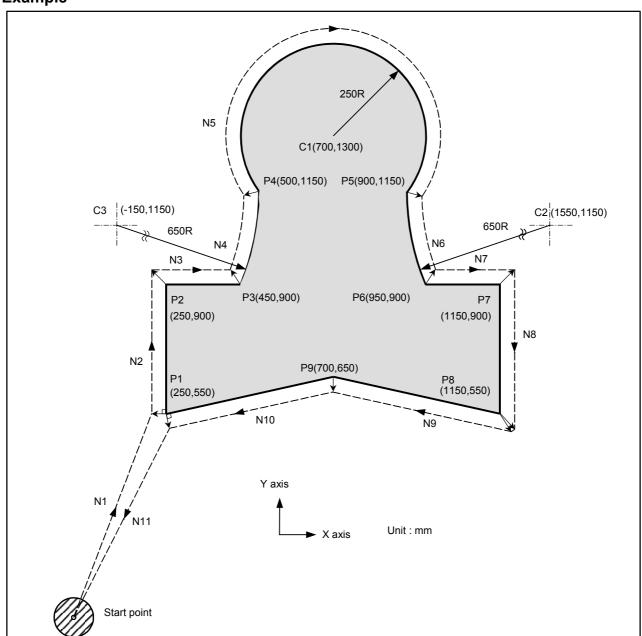
Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane.

Compensation is not executed for the coordinate of a position which is not in the specified plane. The programmed values are used as they are.

In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.

The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, an alarm PS0037 is displayed and the machine is stopped.

# Example



	G50 X0 Y0 Z0 ;	. Specifies absolute coordinates.
		The tool is positioned at the start point (X0, Y0, Z0).
N1	G17 G00 G41 T0707 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 T07.
		In other words the tool path is shifted by the radius of
		the tool (offset mode) because T07 is set to 15
		beforehand (the radius of the tool is 15 mm).
N2	G01 Y900.0 F150 ;	Specifies machining from P1 to P2.
N3	X450.0 ;	Specifies machining from P2 to P3.
N4	G03 X500.0 Y1150.0 R650.0 ;	Specifies machining from P3 to P4.
N5	G02 X900.0 R-250.0 ;	Specifies machining from P4 to P5.
N6	G03 X950.0 Y900.0 R650.0 ;	Specifies machining from P5 to P6.
N7	G01 X1150.0 ;	Specifies machining from P6 to P7.
N8	Y550.0 ;	Specifies machining from P7 to P8.
N9	X700.0 Y650.0 ;	Specifies machining from P8 to P9.
N10	X250.0 Y550.0 ;	Specifies machining from P9 to P1.
N11	G00 G40 X0 Y0 ;	. Cancels the offset mode.
		The tool is returned to the start point (X0, Y0, Z0).

# **5.4** DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION

### **5.4.1** Overview

The following explanation focuses on tool nose radius compensation, but applies to cutter compensation as well. Examples in which XY planes are used, however, apply to cutter compensation only.

### - Tool nose radius center offset vector

The tool nose radius center offset vector is a two dimensional vector equal to the offset value specified in a T code, and the vector is calculated in the CNC. Its dimension changes block by block according to tool movement.

This offset vector (simply called vector herein after) is internally crated by the control unit as required for proper offsetting and to calculate a tool path with exact offset (by tool nose radius) from the programmed path.

This vector is deleted by resetting.

The vector always accompanies the tool as the tool advances.

Proper understanding of vector is essential to accurate programming. Read the description below on how vectors are created carefully.

- G40, G41, G42

G40, G41 or G42 is used to delete or generate vectors.

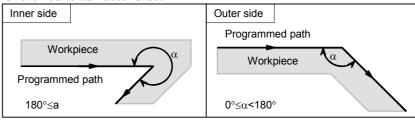
These codes are used together with G00, G01, G02, or G32 to specify a mode for tool motion (Offsetting).

G code	Workpiece position	Function
G40	Neither	Tool nose radius compensation cancel
G41	Right	Left offset along tool path
G42	Left	Right offset along tool path

G41 and G42 specify an offset mode, while G40 specifies cancellation of the offset.

### - Inner side and outer side

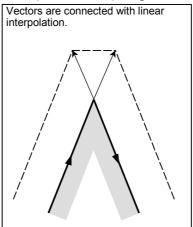
When an angle of intersection of the tool paths specified with move commands for two blocks on the workpiece side is over 180°, it is referred to as "inner side." When the angle is between 0° and 180°, it is referred to as "outer side."



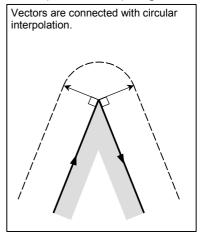
### - Outer corner connection method

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

<1> Linear connection type [Parameter CCC (No.19607#2) = 0]



<2> Circular connection type [Parameter CCC (No.19607#2) = 1]



### - Cancel mode

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

- <1> Immediately after the power is turned on
- <2> When the <RESET> button on the MDI panel is pushed
- <3> After a program is forced to end by executing M02 or M30
- <4> After the tool nose radius compensation cancel command (G40) is exercised

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

### - Start-up

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

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

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

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

Table 5.4.1 (a) Start-up/cancel operation

SUV	SUP	Type	Operation
0	0	Туре А	A compensation vector is output, which is vertical to the block subsequent to the start-up block and the block preceding the cancel block.  Tool nose radius center path  N2  Programmed path
0	1	Туре В	A compensation vector is output, which is vertical to the start-up block and the cancel block. An intersection vector is also output.  Intersection  Tool nose radius center path  N2  Programmed path
1	0 1	Type C	When the start-up block and the cancel block are blocks without tool movement, the tool moves by the cutter or 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.  Intersection  Programmed path  Programmed path  N3  Programmed path  SuP setting: If it is 0, type A is assumed and if 1, type B is assumed.

### - Reading input commands in tool nose radius compensation mode

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

To perform intersection calculation, it is necessary to read at least two blocks with tool movement. To perform an interference check, it is necessary to read at least three blocks with tool movement.

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

### - Meaning of symbols

The following symbols are used in subsequent figures:

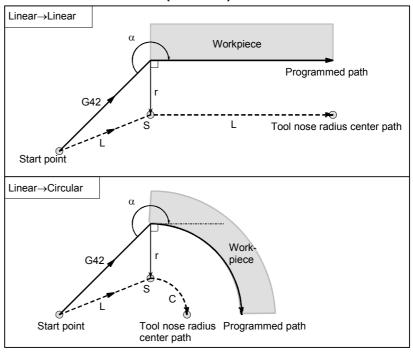
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves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

# **5.4.2** Tool Movement in Start-up

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

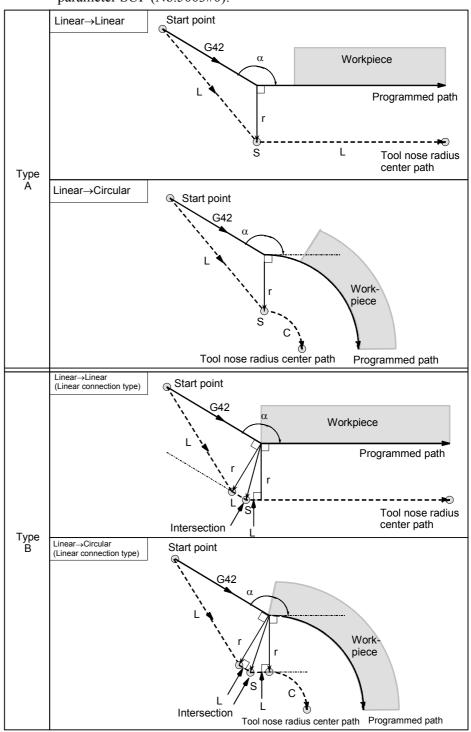
# **Explanation**

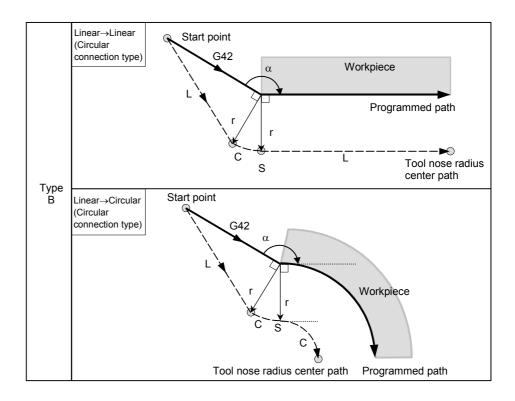
- Tool movement around an inner side of a corner (180°  $\leq \alpha$ )



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

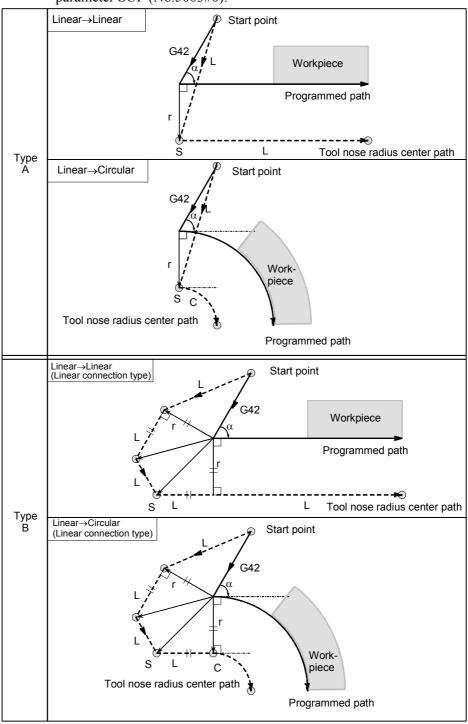
Tool path in start-up has two types A and B, and they are selected by parameter SUP (No.5003#0).

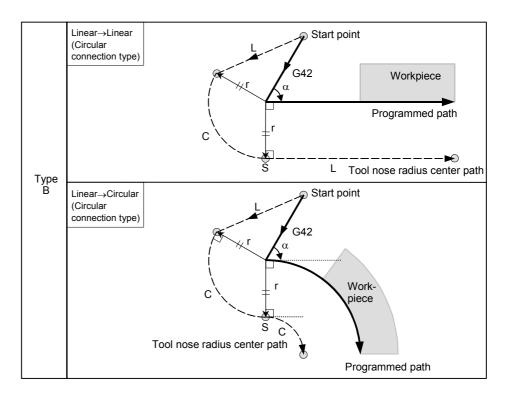




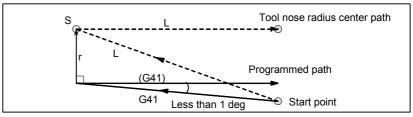
# - Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle ( $\alpha$ <90°)

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





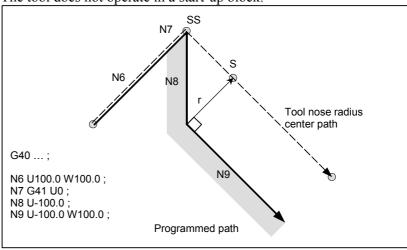
- Tool movement around the outside linear  $\rightarrow$  linear at an acute angle less than 1 degree ( $\alpha$ <1°)



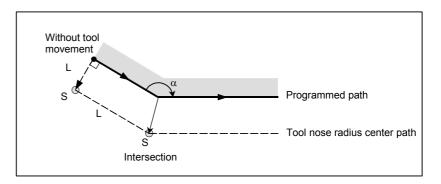
- A block without tool movement specified at start-up

For type A and B

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



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



## **5.4.3** Tool Movement in Offset Mode

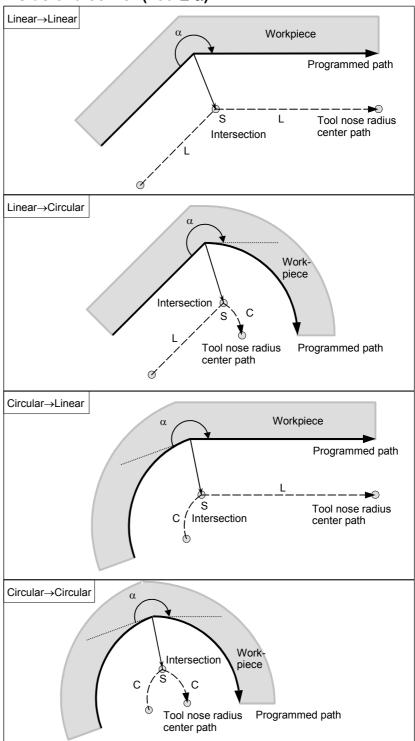
In offset mode, compensation is performed even for positioning commands, not to speak of linear and circular interpolations. To perform intersection calculation, it is necessary to read at least two blocks with tool movement. If, therefore, two or more blocks with tool movement cannot be read in offset 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  $(N-2) \ge M$ . 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**

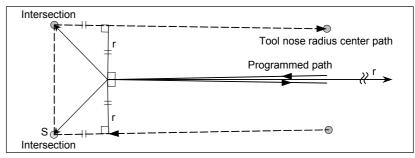
The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the interference check.

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° $\leq \alpha$ )

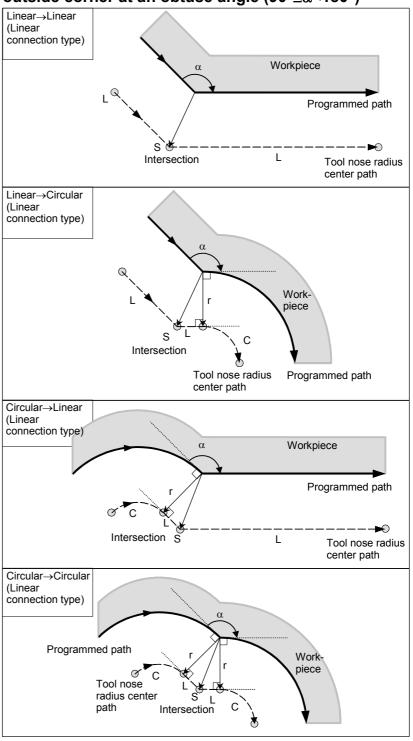


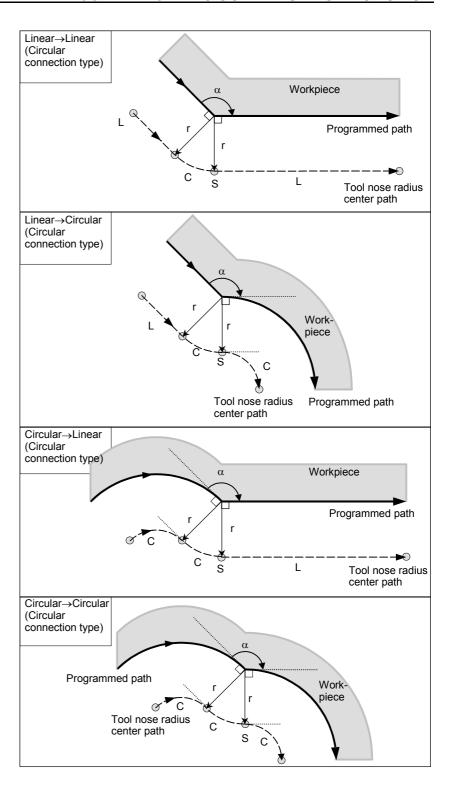
- Tool movement around the inside ( $\alpha$ <1°) with an abnormally long vector, linear  $\rightarrow$  linear



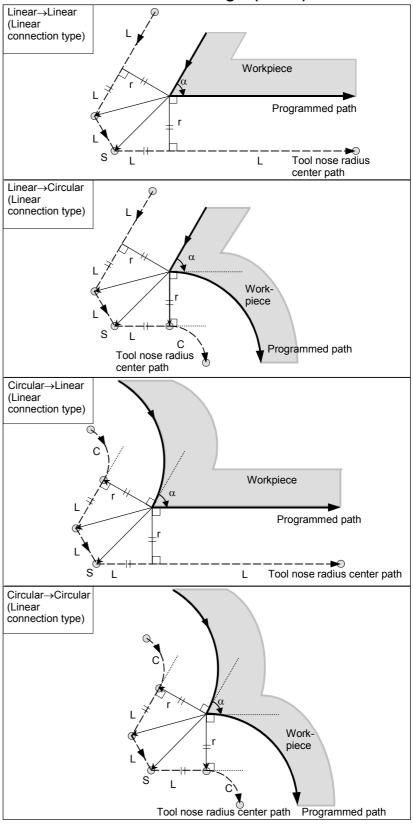
Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

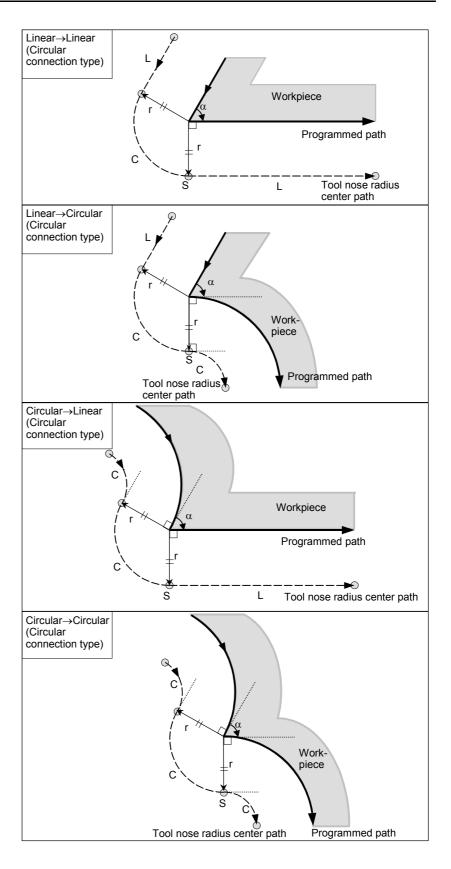
# - Tool movement around the outside corner at an obtuse angle (90°≤α<180°)





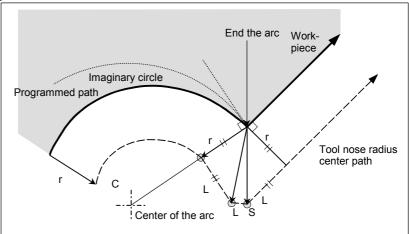
# - Tool movement around the outside corner at an acute angle ( $\alpha$ <90°)





# When it is exceptional End position for the arc is not on the arc

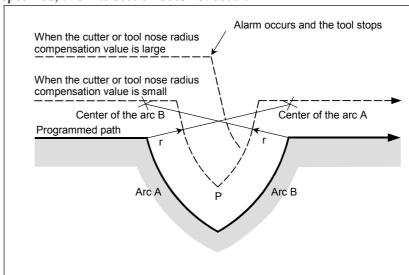
If the end of a line leading to an arc is not on the arc as illustrated below, the system assumes that the tool nose radius compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system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



### There is no inner intersection

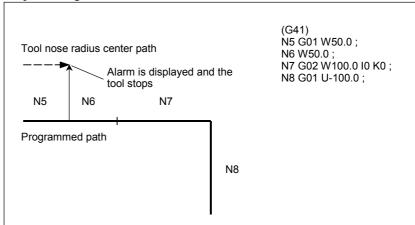
If the cutter or tool nose radius compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for cutter or tool nose radius compensation. When this is predicted,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

In the example shown below, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for cutter or tool nose radius compensation. If an excessively large value is specified, this intersection does not occur.



### - When the center of the arc is identical with the start point or the end position

If the center of the arc is identical with the start point or end point, alarm PS0041 is displayed, and the tool will stop at the start point of the preceding block of the arc.



### - Change in the offset direction in the offset mode

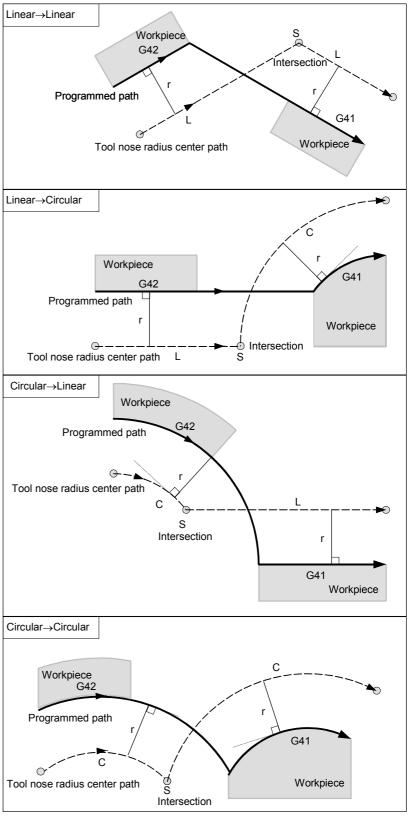
The offset direction is decided by G codes (G41 and G42) for cutter or tool nose radius compensation and the sign of the compensation value as follows.

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

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

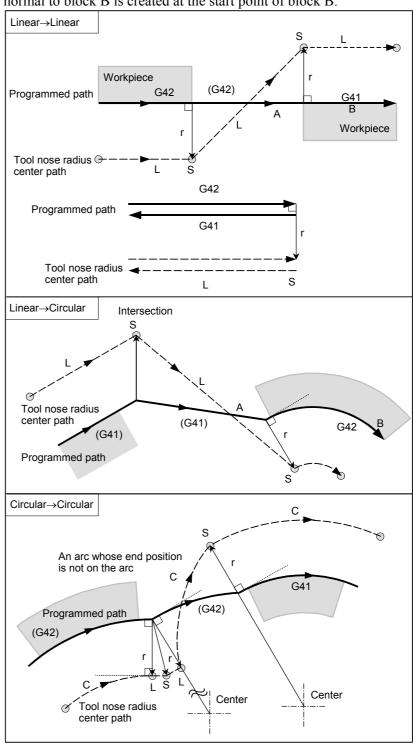
However, the change is not available in the start-up block and the block following it.

# - Tool nose radius center path with an intersection



### - Tool nose radius center path without an intersection

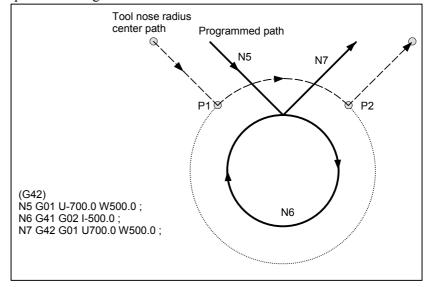
When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.



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

Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I, J, and K this situation can occur.

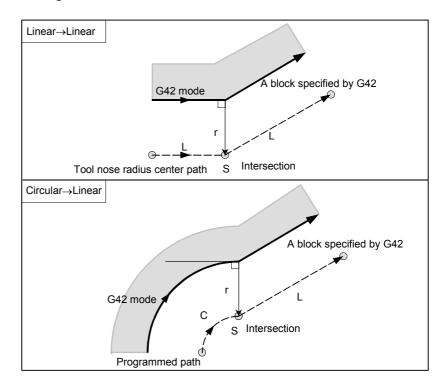
In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from  $P_1$  to  $P_2$  as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.



### - Tool nose radius compensation G code in the offset mode

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

When the direction of offset is expected to be changed by the command of tool nose radius compensation G code (G41, G42), see "Change in the offset direction in the offset mode".

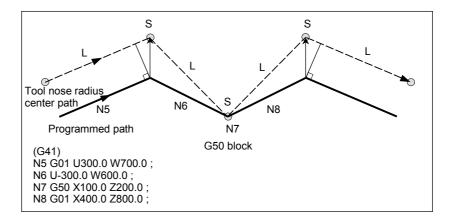


### - Command canceling the offset vector temporarily

During offset mode, if G50 (workpiece coordinate system setting) or G52 (local coordinate system setting) is commanded, the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.

In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled.

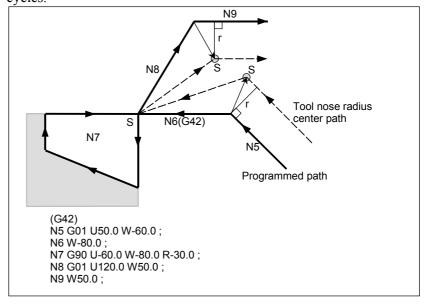
Also when restored to offset mode, the tool moves directly to the intersecting point.



Before specifying G28 (reference position return), G29 (return from reference position), G30 (second, third, and fourth reference position returns), G30.1 (floating reference position return), and G53 (machine coordinate system selection) commands, cancel offset mode, using G40. If an attempt is made to specify any of the commands in offset mode, the offset vector temporarily disappears.

### - Canned cycles (G90, G92, G94) and multiple repetitive cycles (G71 to G76)

See the cautions for the tool nose radius compensation related canned cycles.



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

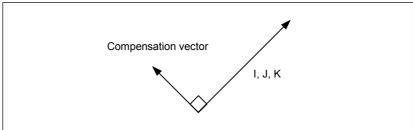
At the start of tool nose radius compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This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

## IJ type vector (XY plane)

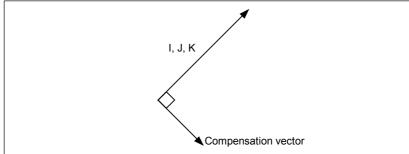
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) As shown in the figure below, it is assumed that the compensation vector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

#### Offset vector direction

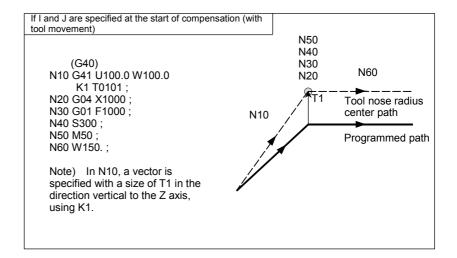
In G41 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the left side.

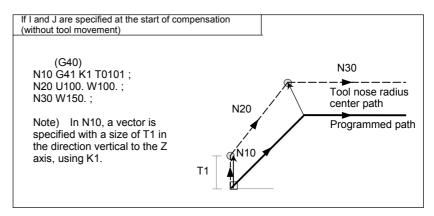


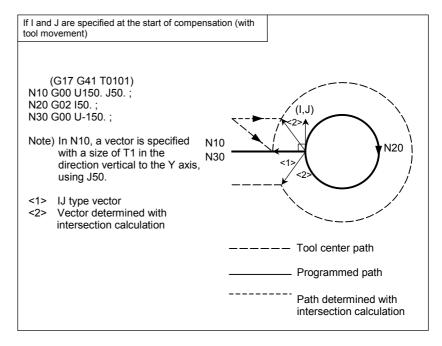
In G42 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the right side.

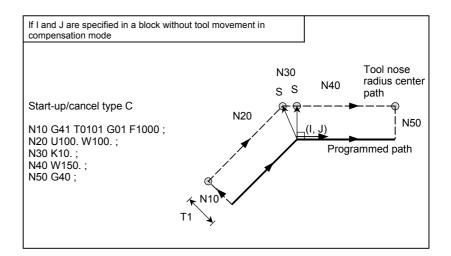


## **Example**



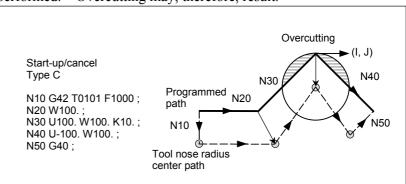






#### Limitation

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



#### - A block without tool movement

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

M05; : M code output : S code output

**G04 X10.0**; : Dwell

G22 X100000; : Machining area setting

G10 P01 X10 Z20 R10.0;

: Tool nose radius compensation

value setting/changing

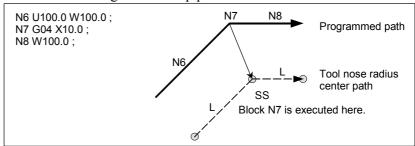
(G18) Y200.0; : Move command not included in the

offset plane.

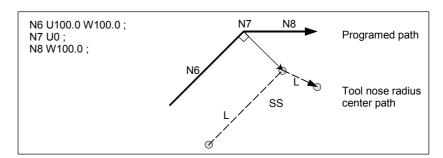
**G98**;, **O10**;, **N20**; : G, O, and N codes only **U0**: : Move distance is zero.

#### - A block without tool movement specified in offset mode

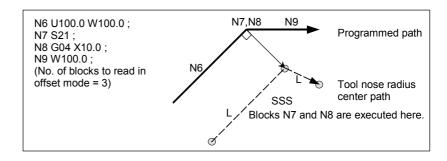
Unless the number of blocks without movement consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)) in offset mode, the vector and the tool nose radius center path will be as usual. This block is executed at the single block stop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



For an axis command for which the move distance is zero, however, a vector with a size equal to the compensation value will be created vertical to the movement direction in the previous block, even if the number of block is 1. Note that specifying such a command may result in overcutting.

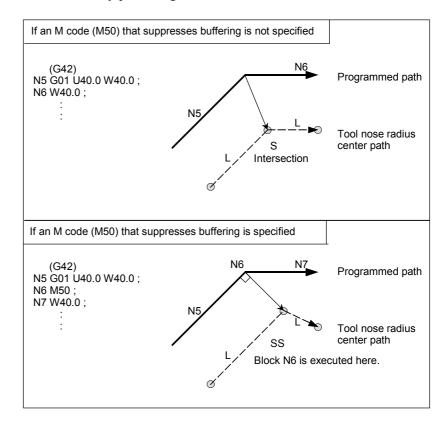


In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter (No. 19625)). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier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



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

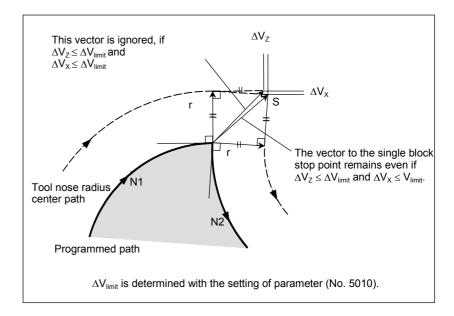
If an M/G code that suppresses buffering is specified in offset mode, it is no longer possible to read and analyze subsequent blocks regardless of the number of blocks to read in offset mode, which is determined by parameter (No. 19625). Then, intersection calculation and a interference check, described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



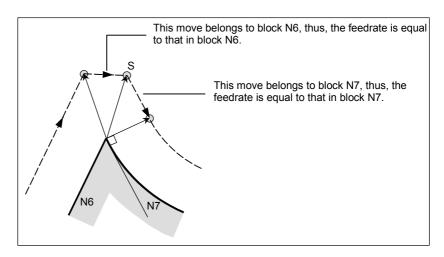
#### - Corner movement

When two or more offset vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.

If these vectors almost coincide with each other (the distance of corner movement between the vectors is judged short due to the setting of parameter (No. 5010)), corner movement is not performed. In this case, the vector to the single block stop point takes precedence and remains, while other vectors are ignored. This makes it possible to ignore the very small movements arising from performing tool nose radius compensation, thereby preventing velocity changes due to interruption of buffering.

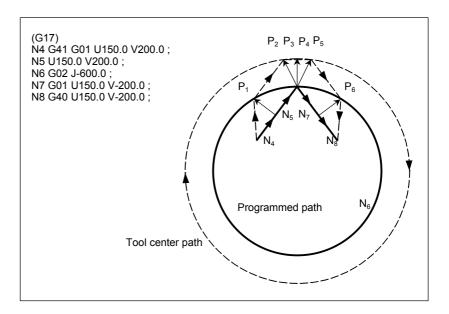


If the vectors are not judged to almost coincide (therefore, are not erased), movement to turn around the corner is performed. The corner movement that precedes the single block stop point belongs to the previous block, while the corner movement that succeeds the single block stop point belongs to the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



However, if the path of the next block is semicircular or more, the above function is not performed.

The reason for this is as follows:



If the vector is not ignored, the tool path is as follows:

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

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

$$P_2 \rightarrow P_4$$

Namely, circle cutting by the block N6 is ignored.

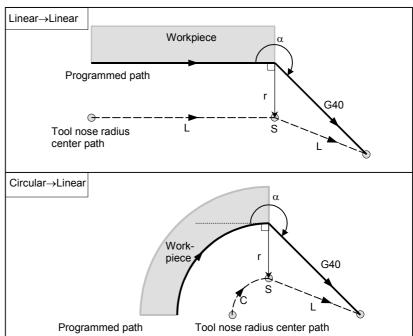
#### - Interruption of manual operation

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

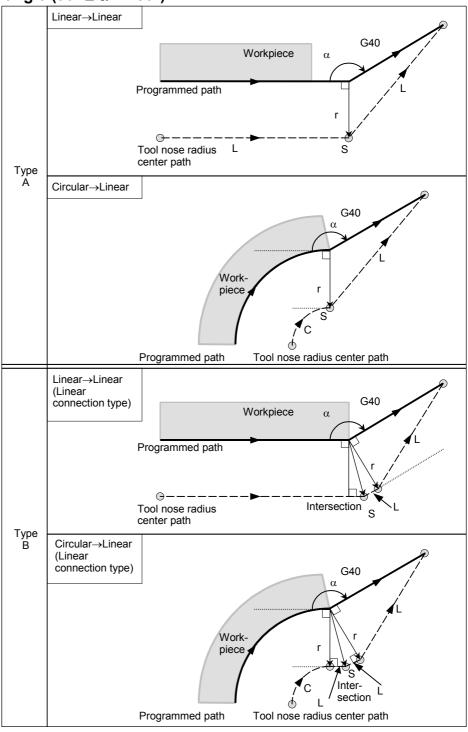
## **5.4.4** Tool Movement in Offset Mode Cancel

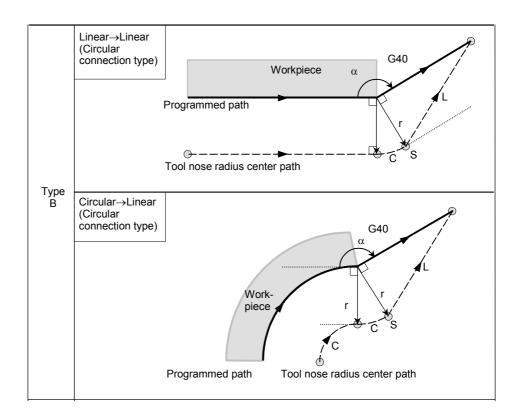
#### **Explanation**

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

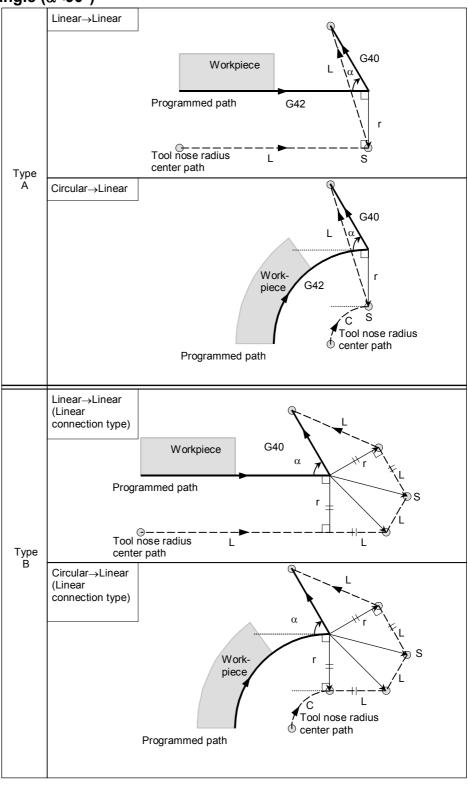


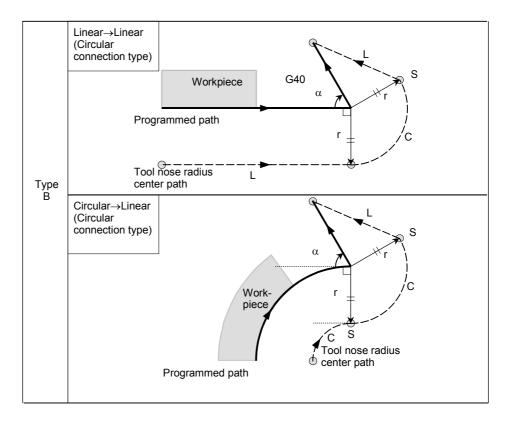
- If the cancel block is a block with tool movement, and the tool moves around the outside at an obtuse angle ( $90^{\circ} \le \alpha < 180^{\circ}$ )



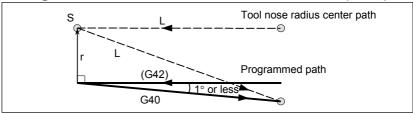


- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle ( $\alpha$ <90°)





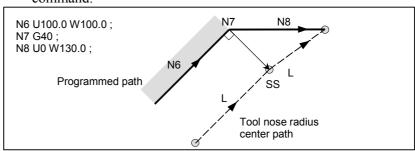
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle of 1 degree or less in a linear  $\rightarrow$  linear manner ( $\alpha \le 1^\circ$ )



- A block without tool movement specified together with offset cancel

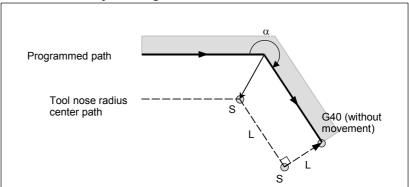
For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the cutter or tool nose radius compensation value in the vertical direction. The tool does not operate in the cancel block. The remaining vectors are canceled with the next move command.



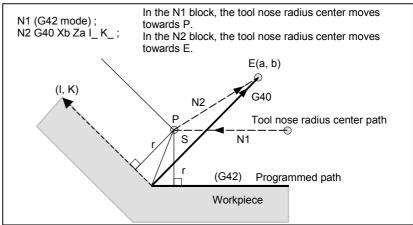
## For type C

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

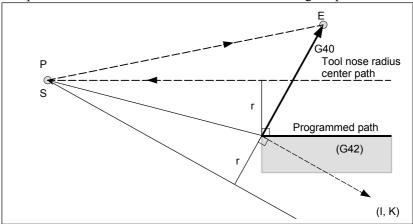


# Block containing G40 and I\_J\_K\_ The previous block contains G41 or G42

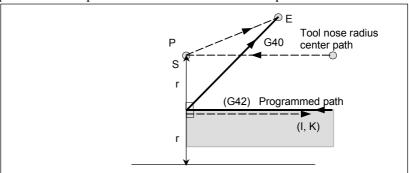
If a G41 or G42 block precedes a block in which G40 and  $I_{,}$   $J_{,}$   $K_{,}$  are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former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



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



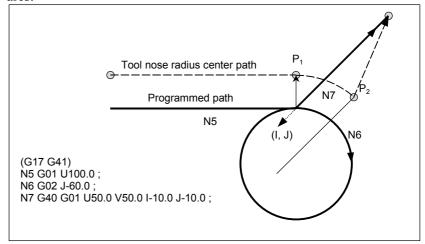
When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



## - Length of the tool center path larger than the circumference of a circle

In the example shown below, the tool does not trace the circle more than once. It moves along the arc from  $P_1$  to  $P_2$ . The interference check function described below may raise an alarm.

To make the tool trace a circle more than once, program two or more arcs.



# **5.4.5** Prevention of Overcutting Due to Cutter or Tool Nose Radius Compensation

## **Explanation**

- Machining a groove smaller than the diameter of the tool nose

Since the tool nose radius compensation forces the path of the center of the tool nose radius to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.

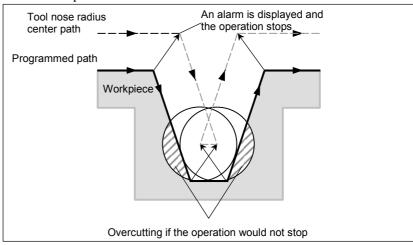


Fig. 5.4.5 (a) Machining a groove smaller than the diameter of the tool

#### - Machining a step smaller than the tool nose radius

For a figure in which a workpiece step is specified with an arc, the tool nose radius center path will be as shown in Fig. 5.4.5 (b). If the step is smaller than the tool nose radius, the tool nose radius center path usually compensated as shown in Fig. 5.4.5 (c) may be in the direction opposite to the programmed path. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued.

If the step is of linear, no alarm will be generated and cut correctly. However uncut part will remain.

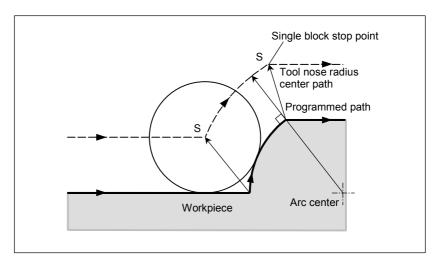


Fig. 5.4.5 (b) Machining a step larger than the tool nose radius

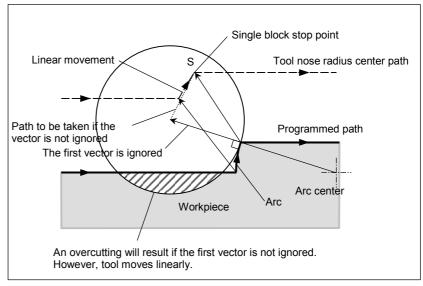
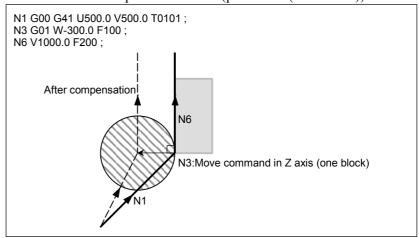


Fig. 5.4.5 (c) Machining a step smaller than the tool nose radius

## - Starting compensation and cutting along the Z-axis

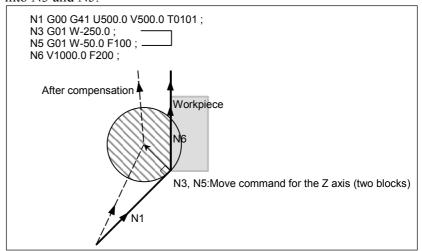
It is usually used such a method that the tool is moved along the Z axis after the cutter compensation (normally XY plane) is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.

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



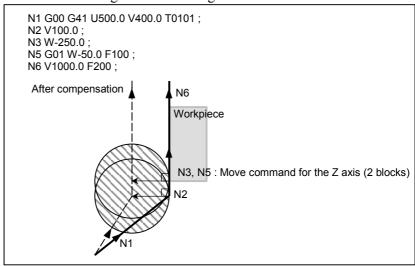
In the program example above, when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the figure above.

Then, suppose that the block N3 (move command in Z axis) is divided into N3 and N5.



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

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



As the block with sequence No. N2 has the move command in the same direction as that of the block with sequence No. N6, the correct compensation is performed.

Alternatively, it is possible to prevent overcutting in the same way by specifying an IJ type vector with the same direction as the advance direction in the start-up block, as in N1 G00 G41 U500. V500. I0 J1 T0101;, after the tool has moved along the Z axis.

## 5.4.6 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

$$(N-3) \ge M$$
.

For example, if the maximum number of blocks to read in offset mode is 8, an interference check is possible even if up to five blocks without tool movement are specified. In this case, three adjacent blocks can be checked for interference, but any subsequent interference that may occur cannot be detected.

#### - Interference check method

Two interference check methods are available, direction check and circular angle check. Parameter CNC (No. 5008#1) and parameter CNV (No. 5008#3) are used to specify whether to enable these methods.

CNV	CNC	Operation	
0	0	An interference check is enabled, and a direction check and a circular angle check can be performed.	
0	1	An interference check is enabled, and only a circular angle check is performed.	
1	-	An interference check is disabled.	

#### NOTE

There are no settings for performing a direction check only.

## - Interference reference <1> (direction check)

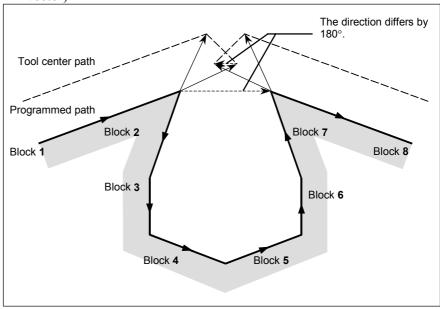
Assuming the number of blocks to read during tool nose radius compensation to be N, a check is first performed on the compensation vector group calculated in (block 1 - block 2) to be output this time and the compensation vector group calculated in (block N-1 - block N); if they intersect, they are judged to interfere. If no interference is found, a check is performed sequentially in the direction toward the compensation vector group to be output this time, as follows:

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

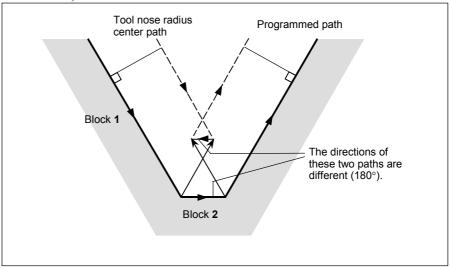
Even if multiple number of compensation vector groups are generated, a check is performed on all pairs.

The judgment method is as follows: For a check on the compensation vector group in (block 1 - block 2) and those in (block N-1 - block N), the direction vector from the specified (end point of block 1) to the (end point of block N-1) is compared with the direction vector from the (point resulting from adding the compensation vector to be checked to the end of block 1) to the (point resulting from adding the compensation vector to be checked to the end of block N-1), and if the direction is 90° or greater or 270° or less, they are judged to intersect and interfere. This is called a direction check.

Example of interference standard <1> (If the block 1 end-point vector intersects with the block 7 end-point vector)



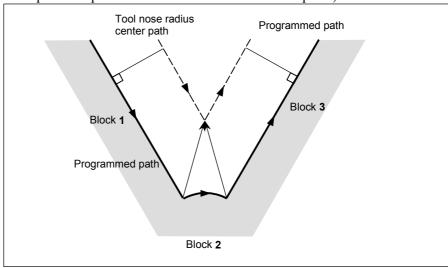
Example of interference standard <1> (If the block 1 end-point vector intersects with the block 2 end-point vector)



## - Interference reference <2> (circular angle check)

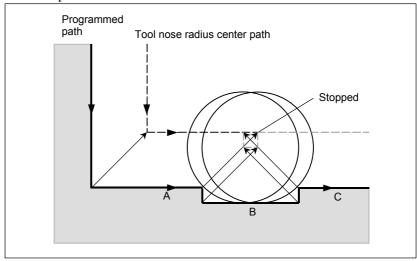
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed path and the circular angle of the start and end point of the post-compensation path, in addition to direction check <1>. If the difference is 180° or greater, the blocks are judged to interfere. This is called a circular angle check.

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



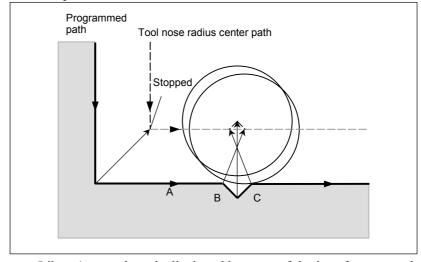
## - When interference is assumed although actual interference does not occur

<1> Depression which is smaller than the cutter or tool nose radius compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after the tool nose radius compensation, the tool stops and an alarm is displayed.

<2> Groove which is smaller than the cutter or tool nose radius compensation value



Like <1>, an alarm is displayed because of the interference as the direction is reverse in block B.

# **5.4.6.1** Operation to be performed if an interference is judged to occur

## **Explanation**

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

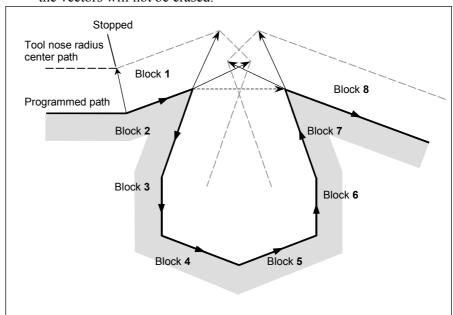
CAV	Function	Operation
0	Interference check alarm function	An alarm stop occurs before the execution of the block in which overcutting (interference) occurs.
1	Interference check avoidance function	The tool path is changed so that overcutting (interference) does not occur, and processing continues.

## 5.4.6.2 Interference check alarm function

## **Explanation**

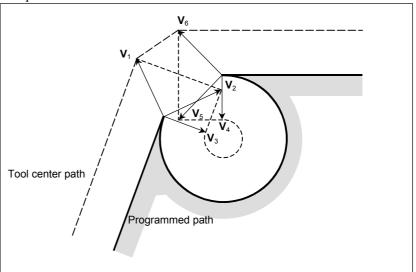
- Interference other than those between adjacent three blocks

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

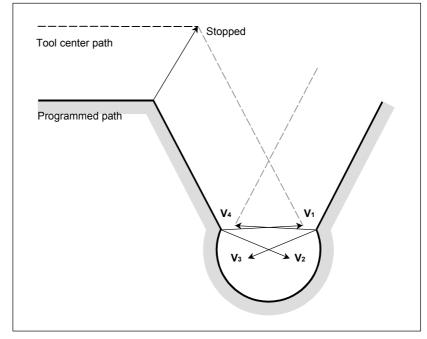


## - Interference between adjacent three blocks

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



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



## 5.4.6.3 Interference check avoidance function

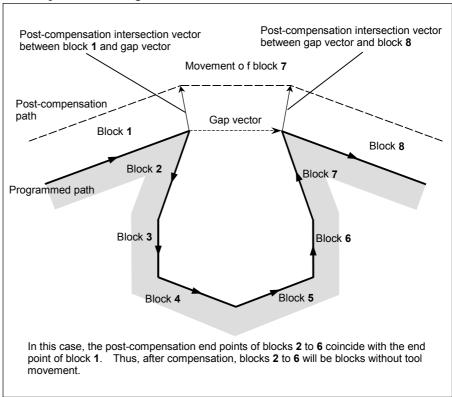
#### Overview

If a command is specified which satisfies the condition under which the interference check alarm function generates an interference alarm, this function suppresses the generation of the interference alarm, but causes a new compensation vector to be calculated as a path for avoiding interference, thereby continuing machining. For the path for avoiding interference, insufficient cutting occurs in comparison with the programmed path. In addition, depending on the specified figure, no path for avoiding interference can be determined or the path for avoiding interference may be judged dangerous. In such a case, an alarm stop will occur. For this reason, it is not always possible to avoid interference for all commands.

#### **Explanation**

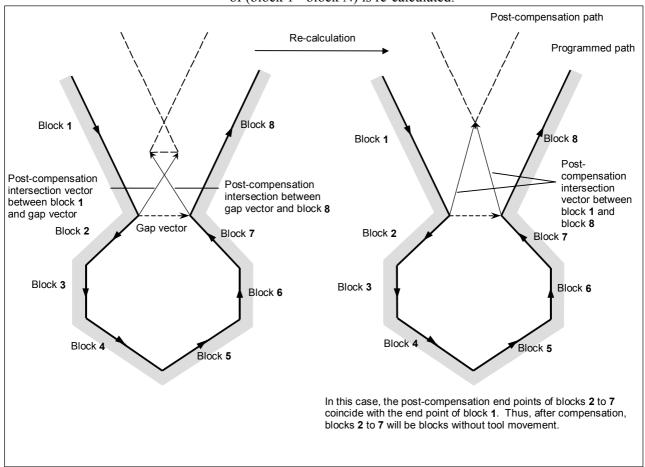
#### - Interference avoidance method

Let us consider a case in which an interference occurs between the compensation vector between (block 1 - block 2) and the compensation vector between (block N-1 - block N). The direction vector from the end point of block 1 to the end point of block N-1 is called a gap vector. At this time, a post-compensation intersection vector between (block 1 - gap vector) and a post-compensation intersection vector between (gap vector - block N) is determined, and a path connecting them is created.

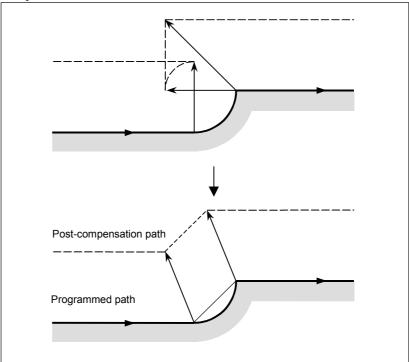


If the post-compensation intersection vector of (block 1 - gap vector) and the post-compensation intersection vector of (gap vector - block

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

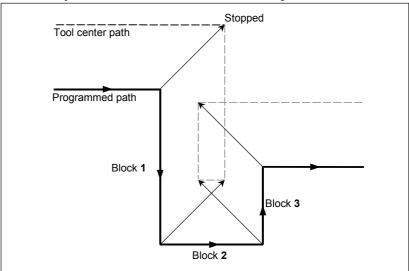


If the cutter or tool nose radius compensation value is greater than the radius of the specified arc as shown in the figure below, and a command is specified which results in compensation with respect to the inside of the arc, interference is avoided by performing intersection calculation with an arc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

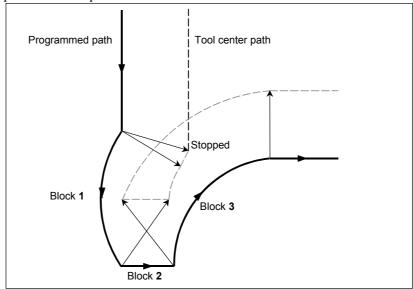


#### - If no interference avoidance vector exists

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

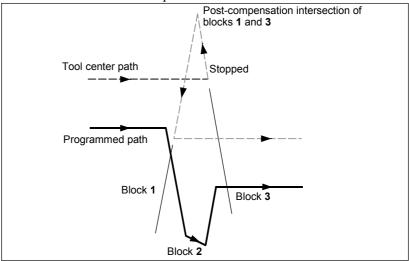


If the circular pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the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

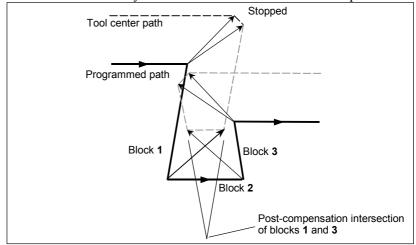


#### - If it is judged dangerous to avoid interference

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

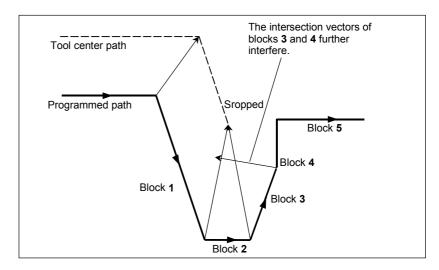


If a pocket in which the bottom is wider than the top, such as that shown in the figure, is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of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



#### - If further interference with an interference avoidance vector occurs

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



#### NOTE

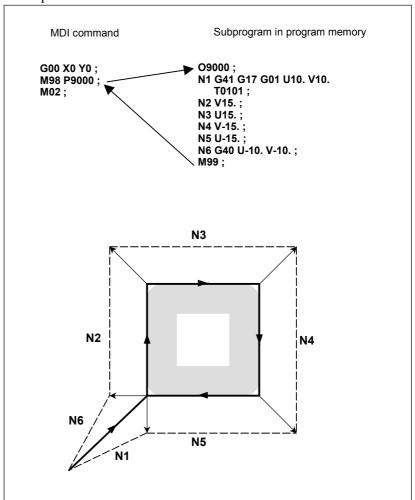
- 1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting parameter NAA (No. 19607#6) appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", however, it is not possible to avoid an alarm regardless of the setting of this parameter.
- 2 If a single block stop occurs during interference avoidance operation, and an operation is performed which differs from the original movement, such as manual intervention, MDI intervention, cutter or tool nose radius compensation value change, intersection calculation is performed with a new path. If such an operation is performed, therefore, an interference may occur again although interference avoidance has been performed once.

## **5.4.7** Cutter or Tool Nose Radius Compensation for Input from MDI

## **Explanation**

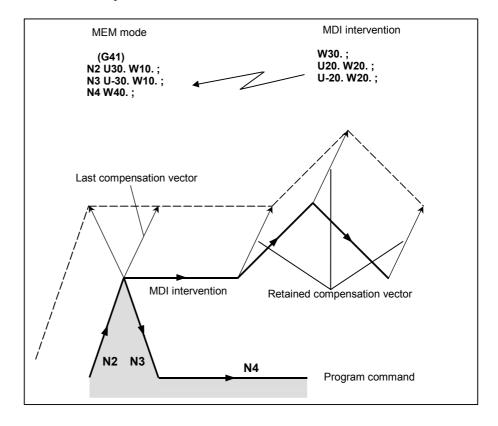
- MDI operation

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.



# **5.5** VECTOR RETENTION (G38)

In cutter or tool nose radius compensation, by specifying G38 in offset mode, it is possible to retain the compensation vector at the end position of the previous block, without performing intersection calculation.

#### **Format**

(In offset mode) **G38 IP\_**;

IP: Value specified for axial movement

## **Explanation**

- Vector retention

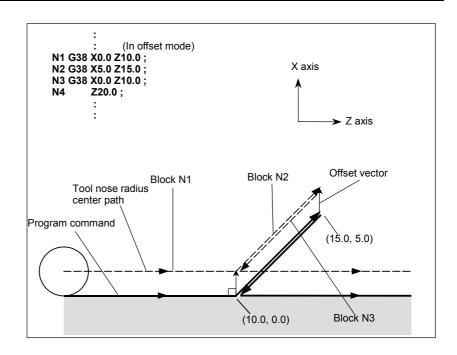
By specifying the above command, a vector is created at the end point of the block immediately preceding the G38 block, vertical to that block. In the G38 block, the vertical vector output in the previous block is retained. G38 is a one-shot G code. With the next move command without a G38 command, the 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.

#### **Example**



# **5.6** CORNER CIRCULAR INTERPOLATION (G39)

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

#### **Format**

```
In offset mode

G39;

or

G39 {I_J_
I_K_
J_K_
};
```

## **Explanation**

- Corner circular interpolation

When the command indicated above is specified, corner circular interpolation in which the radius equals compensation value can be performed. G41 or G42 preceding the command determines whether the arc is clockwise or counterclockwise. G39 is a one-shot G code.

- G39 without I, J, or K

When G39; is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block.

- G39 with I, J, and K

When G39 is specified with I, J, and K, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the vector defined by the I, J, and K values.

#### Limitation

- Move command

In a block containing G39, no move command can be specified. Otherwise, an alarm will occur.

- Inner corner

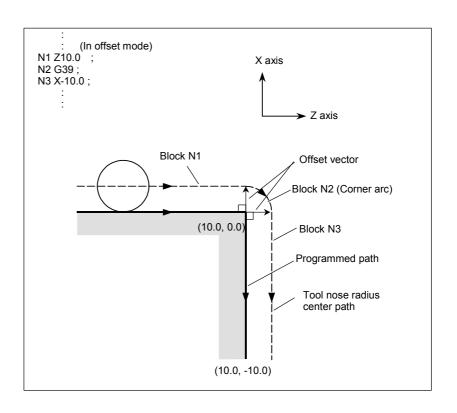
In an inner corner block, G39 cannot be specified. Otherwise, overcutting will occur.

- Corner arc velocity

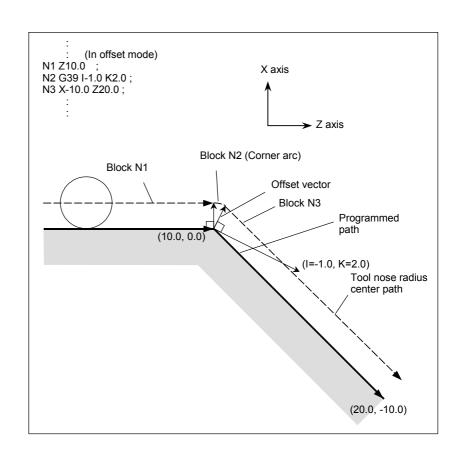
If a corner arc is specified with G39 in G00 mode, the corner arc block velocity will be that of the F command previously specified.

## Example

- G39 without I, J, or K



## - G39 with I, J, and K



### 5.7 EXTENDED TOOL SELECTION

### Overview

In lathe system machines, tools are changed mainly with the following two methods:

- (1) With a turret holding multiple tools, tools are changed by turning the turret (T command).
- (2) With an automatic tool changer (ATC), tools are changed by using both cartridge indexing (T command) and tool change (such as M06) commands.

To support the tool change method explained in (2) above, the following tool selection specifications apply to this function:

- <1> Tool compensation by a T command is disabled. This means that the T command performs auxiliary functions only.
- <2> Tool compensation is enabled by using a G code instead of the T command. In this case, the following types of tool compensation are enabled:
  - Tool length compensation
  - Tool length compensation in tool axis direction (\*1)
  - Tool center point control (\*1)
  - Tool offset (compensation equivalent to that of a T command in the case of turret rotation type)

#### **Format**

M06 T ; Change tool Gxx D: Tool compensation start G49: Tool compensation cancel Gxx : Type of tool compensation G43/G44: Tool length compensation G43.1 : Tool length compensation in tool axis direction G43.4 : Tool center point control type 1 G43.5 : Tool center point control type 2 G43.7 : Tool offset D : Tool compensation number

<sup>\*1</sup> Options are required separately.

### **Explanation**

### - Selecting a tool change method

Bit 3 (TCT) of parameter No. 5040 selects a tool change method. This changes the way for specifying tool compensation. This parameter setting has influence on the following range:

	Bit 3 (TCT) of parameter	Bit 3 (TCT) of parameter
	No. 5040 = 0	No. 5040 = 1
	(Turret type)	(ATC type)
Operation of T	Auxiliary function (tool	Auxiliary function (tool
command	change) and tool offset	indexing) only
Tool compensation	Specified with T code	Specified with G43.7 D_
Compensation No. of	Charified with T and	Specified with D code
tool compensation	Specified with T code	
Compensation No. of		
tool nose radius	Specified with T code	Specified with D code
compensation		
Command such as	Disabled (alarm)	Enabled
G43	Disabled (alarm)	Enabled
Compensation No. of		Specified with D code
G43, etc.	<del>-</del>	Specified with D code

### - T command operation

### (1) When bit 3 (TCT) of parameter No.5040 is 0

The T command performs an auxiliary function and tool offset. The code signal issued to the machine is the T command value except the last 1 to 3 digits.

For example, suppose that the following command is issued when 2 is set in parameter No. 5028 (the number of digits comprising the offset number in the T code command):

T0313;

The T code signal sent to the machine indicates 03.

The tool compensation number, indicated by the lowest 2 digits, is 13.

### (2) When bit 3 (TCT) of parameter No. 5040 is 1

The T command performs an auxiliary function only.

In this case, the code signal issued to the machine is the T command value.

For example, suppose that the following is specified:

T0313;

The T code signal issued to the machine is 0313. The T code signal is not affected by the setting of parameter No. 5028 (the number of digits comprising the offset number in the T code command).

The tool compensation number is not affected by this command.

#### - Tool offset

- (1) When bit 3 (TCT) of parameter No.5040 is 0 The T command performs tool offset.
- (2) When bit 3 (TCT) of parameter No.5040 is 1 The T command does not perform tool offset. To provide tool offset, specify G43.7 D\_. Tool offset is performed in the same way as with the T command when bit 3 of parameter No. 5040 is 0.

### - Compensation number of tool offset

(1) When bit 3 (TCT) of parameter No.5040 is 0

The lowest 1 to 3 digits of the T command are used. The number of digits is set in parameter No. 5028.

When the tool geometry/wear offset option is provided, tool geometry offset and wear offset may also be assigned different compensation values by setting bit 1 (LGN) of parameter No. 5002.

For details, see "TOOL SELECTION FUNCTION".

(2) When bit 3 (TCT) of parameter No.5040 is 1
Except that a compensation number is specified with a D command, the same operation as (1) above is performed.

### - Compensation number of tool nose radius compensation

Same as for tool offset described above.

### - Specification of G43, etc.

- (1) When bit 3 (TCT) of parameter No.5040 is 0 G codes of group 23 such as G43 cannot be specified. Specifying such a G code results in an alarm PS0366.
- (2) When bit 3 (TCT) of parameter No.5040 is 1
  G codes of group 23 such as G43 can be specified. The following G codes can be specified:

G43/G44: Tool length compensation

G43.1: Tool length compensation in tool axis direction

G43.4: Tool center point control type 1

G43.5: Tool center point control type 2

G43.7: Tool offset

Specify a compensation number with D\_. The D code is specified in the same way as for tool offset and tool nose radius compensation.

The reverse meanings can be set for G43/G44 and G43.7 by setting bit 4 (TLG) of parameter No. 5040.

### - Tool compensation memory

As the compensation amount used by the functions listed below, only the value set in Z on the tool compensation memory screen is used, Values such as values set for the X, R, and Y-axis offsets (optional function) are ignored.

G43/G44: Tool length compensation

G43.1: Tool length compensation in tool axis direction

G43.4: Tool center point control type 1

G43.5: Tool center point control type 2

As the compensation amount used in the following function, the values set for the X, Z, and Y-axis offsets (optional function) on the tool compensation memory screen are used:

G43.7: Tool offset

### Limitation

### - Switching between tool offset mode and another compensation mode

In the tool offset (G43.7) mode, a command such as a tool length compensation command (G43/G44, G43.1, G43.4, or G43.5) cannot be specified. Similarly, in a mode such as the tool length compensation mode (G43/G44, G43.1, G43.4, or G43.5), tool offset (G43.7) cannot be specified.

Specifying such a command results in the alarm PS368.

To switch between compensation types, specify G49 to cancel the current compensation mode before setting another compensation mode.

### - Multiple repetitive canned cycle

When a multiple repetitive canned cycle is executed with bit 3 (TCT) of parameter No. 5040 set to 1, note the following:

### **!** CAUTION

1 When a G71 to G76 command is specified in the Series 15 program format, a value such as the depth of cut is specified with a D code. In this case, a D command specified after G71 to G76 is assumed to be the depth of cut. For example, suppose that the command shown below is specified. In this case, the D command <1> is assumed to be the tool compensation number, and the D command <2> is assumed to be the depth of cut.

Example:

2 In the operation of G71 to G73, a G code such as G43 and a D command specified in the finish figure blocks (the portion enclosed by the sequence numbers specified with P\_ and Q\_) are ignored, and the compensation amount set when the G71 to G73 block is specified becomes valid.

### **5.8** AUTOMATIC TOOL OFFSET (G36, G37)

When a tool is moved to the measurement position by execution of a command given to the CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the command measurement position and uses it as the offset value for the tool. When the tool has been already offset, it is moved to the measurement position with that offset value. If the CNC judges that further offset is needed after calculating the difference between the coordinate values of the measurement position and the commanded coordinate values, the current offset value is further offset

Refer to the instruction manuals of the machine tool builder for details.

### **Explanation**

### - Coordinate system

When moving the tool to a position for measurement, the coordinate system must be set in advance. (The workpiece coordinate system for programming is used in common.)

### - Movement to measurement position

A movement to a measurement position is performed by specifying as follows in the MDI, or MEM mode:

G36 Xxa; or G37 Zza;

In this case, the measurement position should be xa or za (absolute programming).

Execution of this command moves the tool at the rapid traverse rate toward the measurement position, lowers the feedrate halfway, then continues to move it until the approach end signal from the measuring instrument is issued.

When the tool tip reaches the measurement position, the measuring instrument outputs the measurement position reach signal to the CNC which stops the tool.

### - Offset

The current tool offset value is further offset by the difference between the coordinate value ( $\alpha$  or  $\beta$ ) when the tool has reached the measurement position and the value of xa or za specified in G36Xxa or G37Zza.

Offset value x = Current offset value  $x+(\alpha-xa)$ 

Offset value z = Current offset value  $z+(\beta-za)$ 

xa: Programmed X-axis measurement point

za: Programmed Z-axis measurement point

These offset values can also be altered from the MDI keyboard.

#### - Feedrate and alarm

The tool, when moving from the stating position toward the measurement position predicted by xa or za in G36 or G37, is fed at the rapid traverse rate across area A. Then the tool stops at point T (xa- $\gamma$  or za- $\gamma$ ) and moves at the measurement feedrate set by parameter (No. 6241) across areas B, C, and D. If the approach end signal turns on during movement across area B, alarm is generated. If the approach end signal does not turn on before point V, and tool stops at point V and alarm PS0080 is generated.

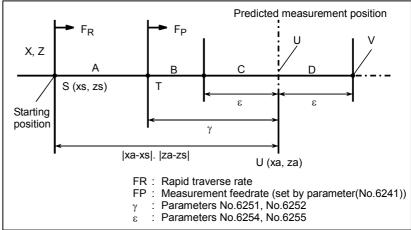
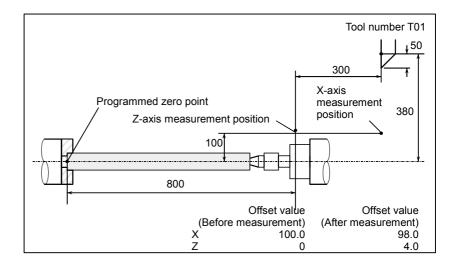


Fig. 5.8 (a) Feedrate and alarm

### - G code

If parameter G36 (No. 3405#3) has been set to 1, G37.1 and G37.2 are used as the G codes for automatic tool compensation for the X- and Z-axes, respectively.

### **Example**



G50 X760.0 Z1100.0; Programming of absolute zero point

(Coordinate system setting)

S01 M03 T0101; Specifies tool T1, offset number 1, and

spindle revolution

G36 X200.0; Moves to the measurement position

If the tool has reached the measurement position at X198.0; since the correct measurement position is 200 mm, the offset

value is altered by 198.0-200.0=-2.0mm.

G00 X204.0; Refracts a little along the X axis.

G37 Z800.0; Moves to the Z-axis measurement position.

> If the tool has reached the measurement position at X804.0, the offset value is altered

by 804.0-800.0=4.0mm.

T0101; Further offsets by the difference.

The new offset value becomes valid when the

T code is specified again.

### **⚠** WARNING

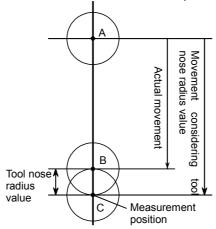
- 1 Measurement speed(Fp),  $\gamma$ , and  $\epsilon$  are set as parameters (Fp : No.6241,  $\gamma$  : No.6251,  $\epsilon$  : No.6254) by machine tool builder.  $\epsilon$  must be positive numbers so that  $\gamma > \epsilon$ .
- 2 Cancel the tool nose radius compensation before G36, G37.
- 3 A delay or variation in detection of the measurement position arrival signal is 0 to 2 msec on the CNC side excluding the PMC side (0.1 msec or less for high-speed measurement position arrival signal input (optional)). Therefore, the measurement error is the sum of 2 msec and a delay or variation (including a delay or variation on the receiver side) in propagation of the measurement position arrival signal on the PMC side, multiplied by the feedrate set in parameter No. 6241.
- 4 A delay or variation in time after detection of the measurement position arrival signal until a feed stops is 0 to 8 msec. To calculate the amount of overrun, further consider a delay in acceleration/deceleration, servo delay, and delay on the PMC side.
- 5 When a manual movement is inserted into a movement at a measurement feedrate, return the tool to the position before the inserted manual movement for restart.
- 6 When using the optional tool nose radius compensation function, the tool offset amount is determined considering the value of tool nose R. Make sure that tool nose radius value is set correctly.

(Condition under which the tool-nose radius compensation is considered)

For the X-axis (first axis of the basic three axes) : TIP=0/5/7

For the Z-axis (third axis of the basic three axes): TIP=0/6/8

For the Y-axis (second axis of the basic three axes): TIP=0



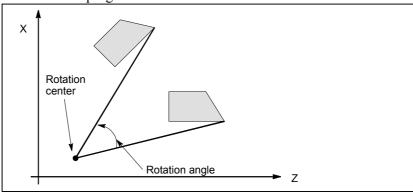
The tool actually moves from point A to point B, but the tool offset value is determined assuming that the tool moves to point C considering the tool nose radius value.

### NOTE

- 1 When there is no T code command before G36 or G37, alarm PS0081 is generated.
- When a T code is specified in the same block as G36 or G37, alarm PS0082 is generated.

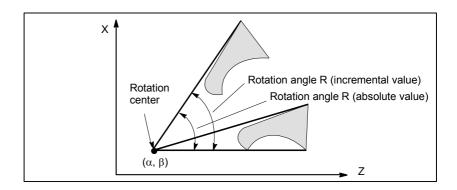
### **5.9** COORDINATE SYSTEM ROTATION (G68.1, G69.1)

With the coordinate system rotation function, it is possible to rotate a figure specified in a program. For example, a program that produces patterns of a figure rotated at increasingly larger angles can be created as a pair of subprograms, one of which defines a figure, the other of which calls the figure definition subprogram by specifying rotation. This method is useful for reducing the program development time and the size of the program.



#### **Format**

**G17 G18** G68.1  $\alpha$ \_  $\beta$ \_ R\_; Starts rotating the coordinate system G19 Coordinate system rotation mode (the coordinates are rotated) G69.1 Cancels coordinate system rotation G17 (G18 or G19): Selects a plane where the figure to be rotated is  $\alpha$ ,  $\beta$ : Specify two coordinates (from among X, Y, and Z) of the rotation center that match G17, G18, and G19. The values specified as the coordinates of the rotation center must always be absolute values. R: Specifies the rotation angle as an absolute value. Counterclockwise rotation is assumed to be positive. However, setting bit 0 (RIN) of parameter No. 5400 enables the use of an incremental value. Incremental units of the angle: 0.001 degrees Specifiable range: -360,000 to +360,000



### **Explanation**

### - Plane selection G code, G17, G18, or G19

Plane selection G code (G17, G18, or G19) can be specified in a block ahead of the coordinate system rotation G code (G68.1). Do not specify G17, G18, or G19 in coordinate system rotation mode.

### - Rotation center

If the rotation center  $(\alpha_{-}, \beta_{-})$  is not specified, the location of the tool when G68.1 is issued is assumed as the rotation center.

### - Rotation angle command

If the rotation angle command (R\_) is not specified, the value specified in parameter No. 5410 is used as the rotation angle.

### - Coordinate rotation cancel

The coordinate rotation cancel G code (G69.1) can be specified in the same block as other commands.

### - Tool compensation

Tool compensation, such as tool offset or tool nose radius compensation, is processed after coordinate system rotation is performed for a program defining a figure.

G68.1 can be used in either G00 or G01 mode.

### Limitation

### - Reference position return

A reference position return command G27, G28, G29, or G30 can be issued only in G69.1 mode.

#### - Changes to coordinates

Do not attempt to change coordinates in G68.1 mode (commands such as G50, G54 to G59, and the tool offset command).

### - Canned cycles

Coordinate system rotation cannot be used in simple canned cycles, multiple repetitive canned cycles, or canned drilling cycles.

#### - Incremental command

Always use absolute values in a move command that immediately follows the coordinate system rotation command (G68.1) or coordinate rotation cancel command (G69.1). pacifying an incremental value results in the move command failing to operate normally.

### **Example**

### - Tool nose radius and coordinate system rotation

G68.1 and G69.1 can be specified during tool nose radius compensation, provided that the coordinate system rotation plane coincides with the tool nose radius compensation plane.

```
N1 G50 X0 Z0 G69.1 G01;

N2 G42 X1000 Z1000 F1000 T0101;

N3 G68 R-30000;

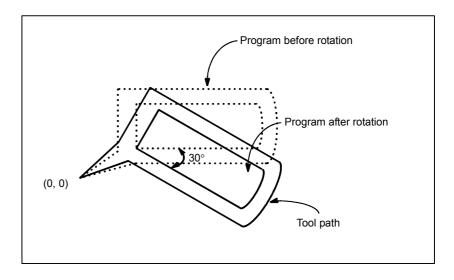
N4 Z3000;

N5 G03 U1000 R1000;

N6 G01 Z1000;

N7 U-1000;

N8 G69.1 G40 X0 Z0;
```



### - Repetitive coordinate system rotation

Coordinate system rotation can be repeated by calling a registered subprogram more than once, but with increasingly greater rotation angles.

Set bit 0 (RIN) of parameter No. 5400 to 1 to specify the rotation angle as being incremental. (G code A, radius programming along the X-axis) G50 X0 Z0 G18; G01 F200 T0101; M98 P2100; M98 P2200 L7; G00 X0 Z0 M30; O2200: G68.1 X0 Z0 R45.0; G90 M98 P2100; M99; O2100 ; G01 G42 X-10.0 Z0; X-10.0 Z4.142; X-7.071 Z7.071 ; G40 M99; Programmed tool path (0, 0)Tool path with an offset (0, -10.0)Subprogram

## 5.10 ACTIVE OFFSET VALUE CHANGE FUNCTION BASED ON MANUAL FEED

#### Overview

When rough machining/semifinish machining is to be performed using a single tool, you may make a fine adjustment of a tool offset value. Moreover, at setup time, you may want to make a fine adjustment of a workpiece origin offset once set. With this function, a travel distance moved on an axis by manual feed is automatically added to the workpiece coordinate system or the currently valid offset number among the specified offset values (tool offset value/workpiece origin offset) to make a offset value change.

### **Explanation**

### - Active offset value change mode

The active offset value change mode is set using the active offset value change mode signal. In this mode, a travel distance moved on an axis by manual feed is automatically added to the workpiece coordinate system or the currently valid offset number among the specified offset values (tool offset value/workpiece origin offset). The types of manual feed usable to make an offset value change in this mode are manual handle feed, incremental feed, and jog feed.

### **↑** CAUTION

- 1 When a movement is being made on an axis for which an offset value is to be changed, do not set the active offset value change mode.
- 2 In the active offset value change mode, do not reset the relative coordinate to 0 or preset the relative coordinate to a specified value.

### - Specifying an offset value to be changed

Specify a tool offset value/workpiece origin offset with the active offset selection signal. A method of axis selection for tool offset value change operation can be selected with bit 1 (ATP) of parameter No. 5041.

In the active offset value change mode, an offset value selected is indicated by blinking display in the state display area on the screen as follows:

- When changing the tool offset value for an axis on which a movement is made (parameter ATP = 0)

Offset value selected	State display
Tool offset value	TOFS
Workpiece origin offset	WZR

- When changing the tool offset value for an axis selected by the signal (parameter ATP = 1)

Offset value selected	State display
X-axis tool offset value	OFSX
Z-axis tool offset value	OFSZ
Y-axis tool offset value	OFSY
Workpiece origin offset	WZR

### **!** CAUTION

When a movement is being made on an axis for which an offset value is to be changed in the active offset value change mode, do not change the specification of the offset value to be changed.

### NOTE

To change the tool offset value for the Y-axis, the option for Y-axis offset is required.

### - Changing a tool offset value

The tool offset value with the offset number corresponding to an T code specified in automatic operation is changed. If there is no currently valid tool offset value as in a case where no T code is specified after a cycle start, tool offset value change is made even when a movement is made on an axis by manual feed.

By making a movement on a basic axis (X, Z, or Y), a tool offset value change can be made. With a movement on a rotation axis, no tool offset value change can be made. While a tool offset value is being changed, a movement by manual feed can be made on one axis only.

### Example

- Specified T code: T0110
- Value set with offset number 10: 54.700 mm
- Travel distance on the Z-axis by manual feed: -2.583 mm

In this example, the Z-axis value for tool offset value number 10 is:

54.700 + (-2.583) = 52.117 mm



### **⚠** CAUTION

When bit 1 (ATP) of parameter No. 5041 is set to 0, a tool offset value change can be made by making a movement on any of the basic axes. When an offset value change for an axis is undesirable, interlock the axis.

#### NOTE

A changed tool offset value is handled according to bit 6 (EVO) of parameter No. 5001 and bit 6 (AON) of parameter No. 5041.

### - Changing a workpiece origin offset value

The workpiece origin offset of the workpiece coordinate system corresponding to a G code from G54 to G59 or from G54.1 P1 to P48 (300) specified during automatic operation is changed on an axis-by-axis basis. A valid workpiece coordinate system exists at all times. So, when a movement is made on an axis by manual feed, the workpiece origin offset of the workpiece coordinate system is changed without fail. This change can be made by a movement on an arbitrary axis, which may be a linear axis or a rotation axis. While a workpiece origin offset change is being made, movements can be made on multiple axes by manual feed.

### **Example**

- Specified workpiece coordinate system : G56
- Workpiece origin offset of G56 (X axis): 50.000
- Workpiece origin offset of G56 (Z axis): 5.000
- Workpiece origin offset of G56 (C axis): 180.000
- Workpiece origin offset of G56 (Y axis): -60.000
- Travel distance on the X axis by manual feed : -10.000mm
- Travel distance on the Z axis by manual feed : 10.000mm
- Travel distance on the C axis by manual feed : 90.000deg
- Travel distance on the Y axis by manual feed :
   -5.000mm

In this example, the workpiece origin offsets of G56 are as follows:

- X axis: 50.000+(-10.000) = 40.000 - Z axis: 5.000+10.000 = 15.000 - C axis: 180.000+90.000 = 270.000

- Y axis : -60.000 + (-5.000) = -65.000

### - Presetting the relative position indication

By setting bit 5 (APL) of parameter No. 3115 to 1, the relative position indication (counter) can be automatically preset to 0 when the active offset value change mode is selected. In this case, the changed offset value can be restored to the original value by returning the relative position indication (counter) to 0 by manual feed.

### - Emergency stop, servo alarm

If an emergency stop occurs, a servo alarm is issued, or servo excitation is turned off, an offset value change is made also for a travel distance on an axis moved by follow-up in the active offset value change mode.

### NOTE

If a tool offset value is selected as an offset value to be changed, no offset value change is made for a travel distance on a rotation axis moved by follow-up.

### Limitation

- Manual operation that cannot change an active offset value

In a mode other than the manual handle feed mode/incremental feed mode/jog feed mode, no active offset value can be changed.

Moreover, no active offset value can be changed in the manual reference position return mode.

Even in the modes mentioned above, do not change an active offset value in the following operations:

- Manual feed for 5-axis machining
- Manual numerical command
- PMC axis control
- Axis that disables an active offset value from being changed

With a rotation axis, no tool offset value can be changed using this function.

- Offset values that cannot be changed with the active offset value change function

With this function, a tool-noose radius compensation value, B-axis offset value, and second geometry offset value cannot be changed.

6

# **MEMORY OPERATION BY Series 15 FORMAT**

By setting the setting-related parameter (bit 1 of parameter No. 0001), a program created in the Series 15 program format can be registered in memory for memory operation. Registration to memory and memory operation are possible for the functions which use the same tape format as that for the Series 15 as well as for the following functions which use a different program format:

- Subprogram calling
- Canned cycle
- Multiple repetitive canned cycle
- Canned cycle for drilling

### **NOTE**

Registration to memory and memory operation are possible only for the functions available in this CNC.

# 6.1 ADDRESSES AND SPECIFIABLE VALUE RANGE FOR Series 15 PROGRAM FORMAT

Some addresses which cannot be used for the this CNC can be used in the Series 15 program format. The specifiable value range for the Series 15 program format is basically the same as that for the this CNC. Sections II-6.2 to II-6.5 describe the addresses with a different specifiable value range. If a value out of the specifiable value range is specified, an alarm is issued.

## 6.2 SUBPROGRAM CALLING

### **Format**

### M98 Pxxxx Lyyyy;

P : Subprogram number L : Repetition count

### **Explanation**

- Address

Address L cannot be used in this CNC tape format but can be used in the Series 15 format.

- Subprogram number

The specifiable value range is the same as that for this CNC (1 to 9999)

If a value of more than four digits is specified, the last four digits are assumed as the subprogram number.

- Repetition count

The repetition count L can be specified in the range from 1 to 9999. If no repetition count is specified, 1 is assumed.

### 6.3 CANNED CYCLE

### **Explanation**

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

#### NOTE

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

ZX plane: Z-axis direction YZ plane: Y-axis direction XY plane: X-axis direction

4 The direction of the end face means the direction of the second axis on the plane as follows:

ZX plane: X-axis direction YZ plane: Z-axis direction XY plane: Y-axis direction

### 6.3.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

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

### 6.3.1.1 Straight cutting cycle

### **Format**

### $G90X(U)_Z(W)_F_;$

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the length

F\_ : Cutting feedrate

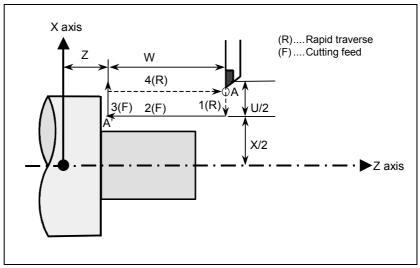


Fig. 6.3.1 (a) Straight cutting cycle

### **Explanation**

### - Operations

A straight cutting cycle performs four operations:

- (1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
- (2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the length.)
- (3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in cutting feed
- (4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

### NOTE

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

### - Canceling the mode

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

### 6.3.1.2 Taper cutting cycle

### **Format**

ZpXp-plane

G90 X(U)\_ Z(W)\_ I\_ F\_;

ZpXp-plane

G90 Y(V)\_ Z(W)\_ K\_ F\_;

ZpXp-plane

G90 X(U)\_ Y(V)\_ J\_ F\_;

X\_,Y\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length

U\_,V\_,W\_ : Travel distance to the cutting end point (point A' in the figure below) in the direction of the length

I\_J\_K\_ : Taper amount (I in the figure below)

F\_ : Cutting feedrate

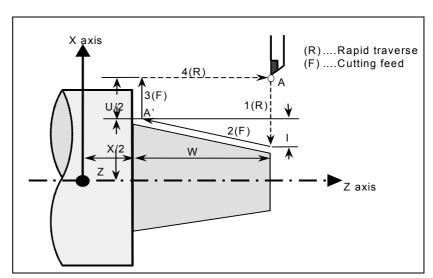


Fig. 6.3.1 (b) Taper cutting cycle

### **Explanation**

Address I, J, or K for specifying a taper varies with the plane selected. The figure of a taper is determined by the coordinates of the cutting end point (A') in the direction of the length and the sign of the taper amount (address I, J, or K). For the cycle in the figure above, a minus sign is added to the taper amount.

#### NOTE

The increment system of address I, J, or K for specifying a taper depends on the increment system for the reference axis. Specify a radius value at I, J, or K.

### - Operations

A taper cutting cycle performs the same four operations as a straight cutting cycle.

However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

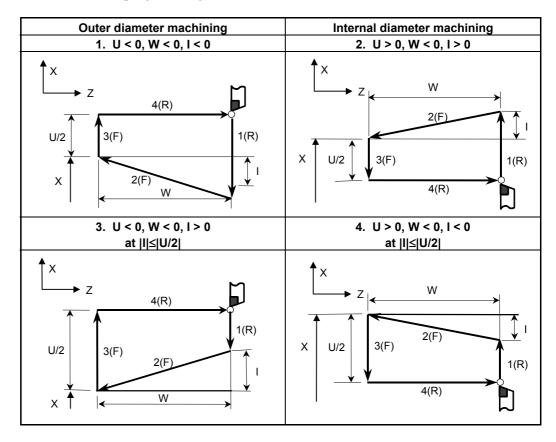
Operations 2, 3, and 4 after operation 1 are the same as for a straight cutting cycle.

### **NOTE**

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

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

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



### - Canceling the mode

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

### **6.3.2** Threading Cycle (G92)

### 6.3.2.1 Straight threading cycle

### **Format**

### G92 X(U)\_Z(W)\_F\_Q\_;

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the length

Q\_ : Angle for shifting the threading start angle

(Increment: 0.001 degrees,

Valid setting range: 0 to 360 degrees)

F : Thread lead (L in the figure below)

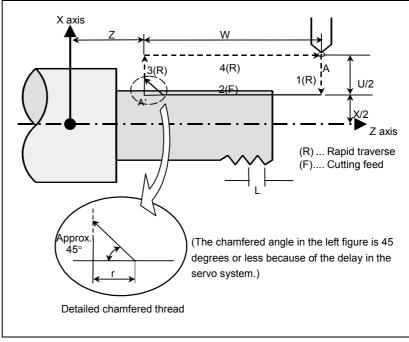


Fig. 6.3.2 (c) Straight threading

### **Explanation**

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

### - Operations

A straight threading cycle performs four operations:

- (1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.
- (2) Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. At this time, thread chamfering is performed.

- (3) Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (Retraction after chamfering)
- (4) Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

#### ⚠ CAUTION

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

#### NOTE

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

### - Canceling the mode

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

### - Time constant and FL feedrate for threading

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

#### - Thread chamfering

Thread chamfering can be performed. A signal from the machine tool, initiates thread chamfering. The chamfering distance r is specified in a range from 0.1L to 12.7L in 0.1L increments by parameter No. 5130. (In the above expression, L is the thread lead.) A thread chamfering angle between 1 to 89 degrees can be specified in parameter No. 5131. When a value of 0 is specified in the parameter, an angle of 45 degrees is assumed.

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

#### NOTE

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

### - Retraction after chamfering

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

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Performs an in-position check before retraction and uses the type of acceleration/deceleration after interpolation in rapid traverse, time constant for rapid traverse, FL feedrate, and rapid traverse rate specified in parameter No. 1420.

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

### **NOTE**

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

### - Shifting the start angle

Address Q can be used to shift the threading start angle.

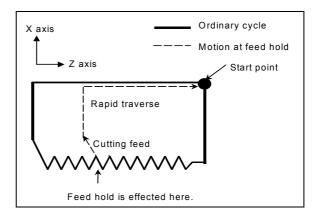
The start angle (Q) increment is 0.001 degrees and the valid setting range is between 0 and 360 degrees. No decimal point can be specified.

### - Feed hold in a threading cycle

When the threading cycle retract function is not used, the machine stops at the end point of retraction after chamfering (end point of operation 3) by feed hold applied during threading.

### - Threading cycle retract

When the "threading cycle retract" option function is used, feed hold may be applied during threading (operation 2). In this case, the tool immediately retracts with chamfering and returns to the start point on the second axis (X-axis), then the first axis (Z-axis) on the plane.



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



Another feed hold cannot be made during retreat.

### - Inch threading

Inch threading specified with address E is not allowed.

### 6.3.2.2 Taper threading cycle

### **Format**

ZpXp-plane G92 X(U)\_ Z(W)\_ I\_ F\_ Q\_ ; YpZp-plane G92 Y(V)\_ Z(W)\_ K\_ F\_ Q\_; XpYp-plane G92  $X(U)_Y(V)_J_F_Q;$ X\_,Y\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the length U ,V ,W : Travel distance to the cutting end point (point A' in the figure below) in the direction of the length  $Q_{\_}$ : Angle for shifting the threading start angle (Increment: 0.001 degrees, Valid setting range: 0 to 360 degrees) R\_ : Taper amount (R in the figure below) : Thread lead (L in the figure below)

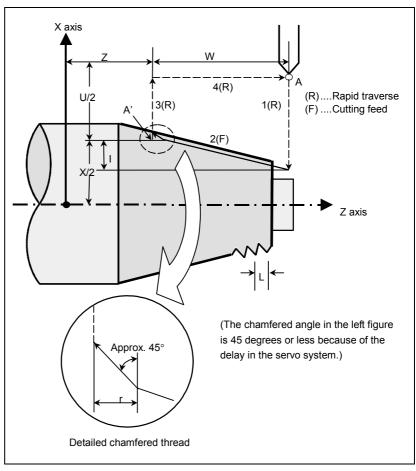


Fig. 6.3.2 (d) Taper threading cycle

### **Explanation**

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

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

#### NOTE

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

### - Operations

A taper threading cycle performs the same four operations as a straight threading cycle.

However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

Operations 2, 3, and 4 after operation 1 are the same as for a straight threading cycle.



### **⚠** CAUTION

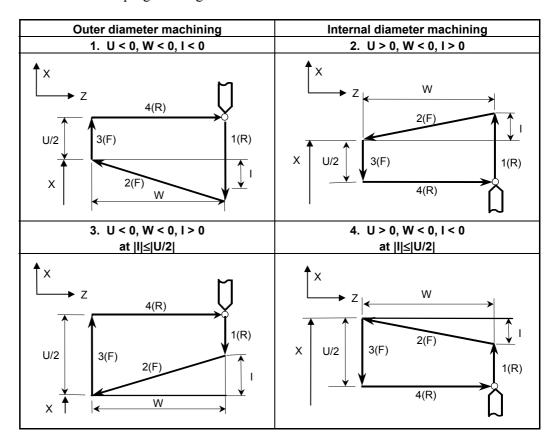
Notes on this threading are the same as in threading in G32. However, a stop by feed hold is as follows: Stop after completion of path 3 of threading cycle.

### **NOTE**

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

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

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



### - Canceling the mode

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

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

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

### **6.3.3** End Face Turning Cycle (G94)

### 6.3.3.1 Face cutting cycle

#### **Format**

### G92 X(U)\_Z(W)\_F\_;

X\_,Z\_ : Coordinates of the cutting end point (point A' in the figure below) in the direction of the end face

U\_,W\_: Travel distance to the cutting end point (point A' in the figure below) in the direction of the end face

F\_ : Cutting feedrate

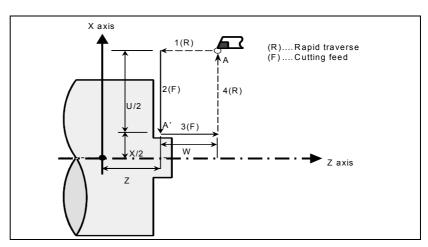


Fig. 6.3.3 (e) Face cutting cycle

### **Explanation**

### - Operations

A face cutting cycle performs four operations:

- (1) Operation 1 moves the tool from the start point (A) to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.
- (2) Operation 2 moves the tool to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the end face.)
- (3) Operation 3 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in cutting feed.
- (4) Operation 4 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

### NOTE

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

### - Canceling the mode

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

### 6.3.3.2 Taper cutting cycle

### **Format**

```
ZpXp-plane

G94 X(U)_ Z(W)_ K _ F_;

YpZp-plane

G94 Y(V)_ Z(W)_ J _ F_;

XpYp-plane

G94 X(U)_ Y(V)_ I _ F_;

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

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

R_ : Taper amount (R in the figure below)

F_ : Cutting feedrate
```

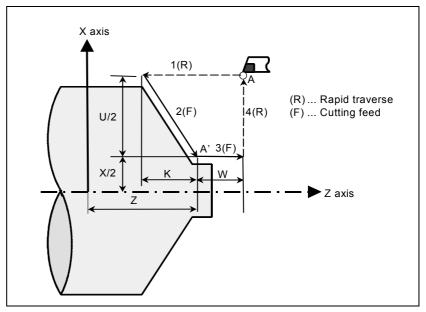


Fig. 6.3.3 (f) Taper cutting cycle

### **Explanation**

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

### **NOTE**

The increment system of address I, J, or K for specifying a taper depends on the increment system for the reference axis. Specify a radius value at I, J, or K.

### - Operations

A taper cutting cycle performs the same four operations as a face cutting cycle.

However, operation 1 moves the tool from the start point (A) to the position obtained by adding the taper amount to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in rapid traverse.

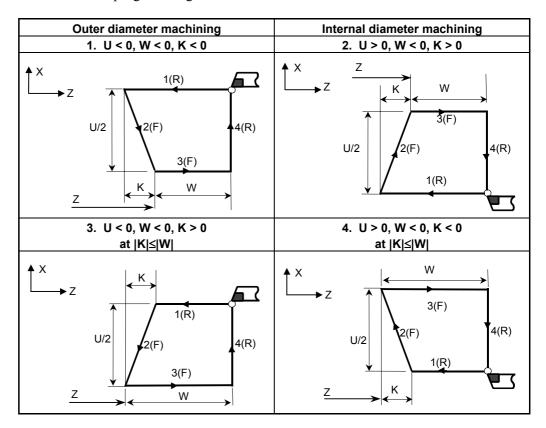
Operations 2, 3, and 4 after operation 1 are the same as for a face cutting cycle.

#### NOTE

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

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

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



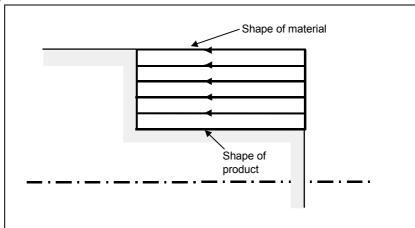
### - Canceling the mode

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

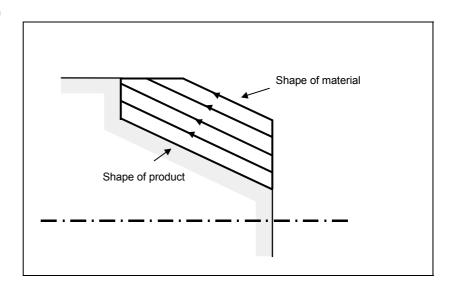
## 6.3.4 How to Use Canned Cycles

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

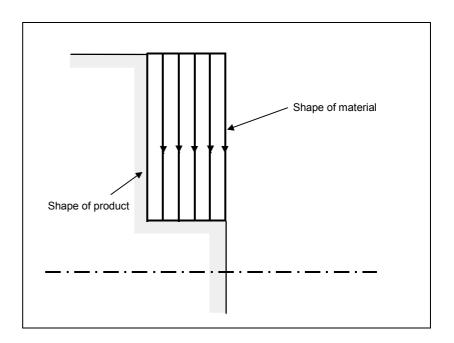
- Straight cutting cycle (G90)



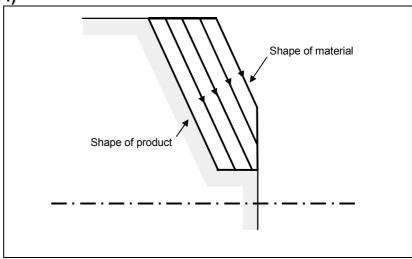
- Taper cutting cycle (G90)



## - Face cutting cycle (G94)



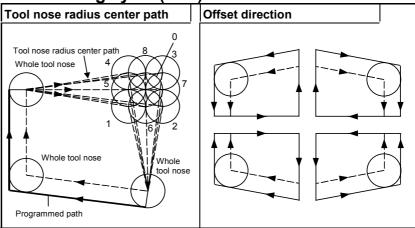
# - Face taper cutting cycle (G94)



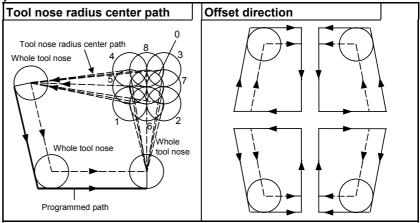
# **6.3.5** Canned Cycle and Tool Nose Radius Compensation

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

Outer diameter/internal diameter cutting cycle (G90)



End face cutting cycle (G94)



## Threading cycle (G92)

Tool nose radius compensation cannot be applied.

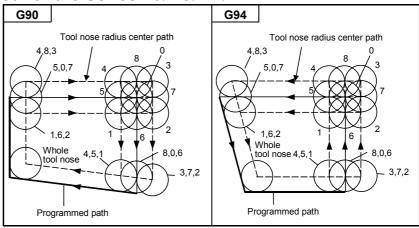
## Differences between this CNC and the Series 16i/18i/21i

### **NOTE**

This CNC is the same as the Series 16i/18i/21i in the offset direction, but differs from the series in the tool nose radius center path.

- For this CNC
   Cycle operations of a canned cycle are replaced
   with G00 or G01. In the first block to move the
   tool from the start point, start-up is performed.
   In the last block to return the tool to the start
   point, offset is canceled.
- For the Series 16i/18i/21i
   This series differs from this CNC in operations in the block to move the tool from the start point and the last block to return it to the start point.
   For details, refer to "Series 16i/18i/21i
   Operator's Manual."

## How compensation is applied for the Series 16i/18i/21i



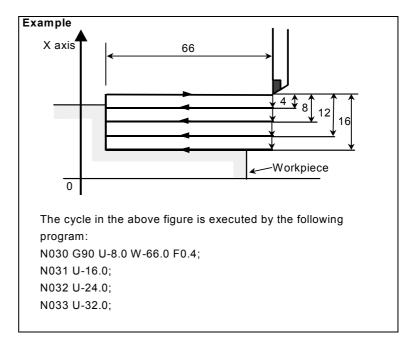
# **6.3.6** Restrictions on Canned Cycles

### Limitation

#### - Modal

Since data items X (U), Z (W), and R in a canned cycle are modal values common to G90, G92, and G94. For this reason, if a new X (U), Z (W), or R value is not specified, the previously specified value is effective.

Thus, when the travel distance along the Z-axis does not vary as shown in the program example below, a canned cycle can be repeated only by specifying the travel distance along the X-axis.



The modal values common to canned cycles are cleared when a one-shot G code other than G04 is specified.

Since the canned cycle mode is not canceled by specifying a one-shot G code, a canned cycle can be performed again by specifying modal values. If no modal values are specified, no cycle operations are performed.

When G04 is specified, G04 is executed and no canned cycle is performed.

## - Block in which no move command is specified

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

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

#### - Plane selection command

Specify a plane selection command (G17, G18, or G19) before setting a canned cycle or specify it in the block in which the first canned cycle is specified.

If a plane selection command is specified in the canned cycle mode, the command is executed, but the modal values common to canned cycles are cleared.

If an axis which is not on the selected plane is specified, alarm PS0330 is issued.

#### - Parallel axis

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

# **6.4** MULTIPLE REPETITIVE CYCLE

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

### **NOTE**

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

# **6.4.1** Stock Removal in Turning (G71)

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

#### **Format**

```
ZpXp plane
  G71 P(ns) Q(nf) U(\Deltau) W(\Deltaw) I(\Deltai) K(\Deltak) D(\Deltad) F(f ) S(s )
  T(t);
              The move command between A and B is specified in the
  N (ns);
             blocks from sequence number ns to nf.
 N (nf);
YpZp plane
  G71 P(ns) Q(nf) V(\Deltaw) W(\Deltau) J(\Deltak) K(\Deltai) D(\Deltad) F(f) S(s)
  T(t):
 N (ns);
   ...
 N (nf);
XpYp plane
  G71 P(ns) Q(nf) U(\Deltaw) V(\Deltau) I(\Deltak) J(\Deltai) D(\Deltad) F(f) S(s) T(t);
 N (ns);
  N (nf);
∆d : Depth of cut
      The cutting direction depends on the direction AA'.
ns: Sequence number of the first block for the program of finishing
      shape.
nf : Sequence number of the last block for the program of finishing
      shape.
\Delta u: Distance of the finishing allowance in the direction of the second
      axis on the plane (X-axis for the ZX plane)
\Delta w: Distance of the finishing allowance in the direction of the first axis
      on the plane (Z-axis for the ZX plane)
   : Distance of the finishing allowance of the roughing in the direction
      of the second axis on the plane (X-axis for the ZX plane)
\Delta k: Distance of the finishing allowance of the roughing in the direction
      of the first axis on the plane (Z-axis for the ZX plane)
f,s,t: Any F, S, or T function contained in blocks ns to nf in the cycle is
      ignored, and the F, S, or T function in this G71 block is effective.
```

#### **NOTE**

Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

	Unit	Diameter/radius programming	Sign
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required
Δί	Depends on the increment system for the reference axis.	Radius programming	Not required
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required

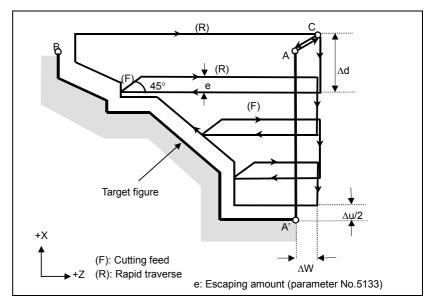


Fig. 6.4.1 (a) Cutting path of an outer surface rough cutting cycle without rough cutting finishing allowance (type I)

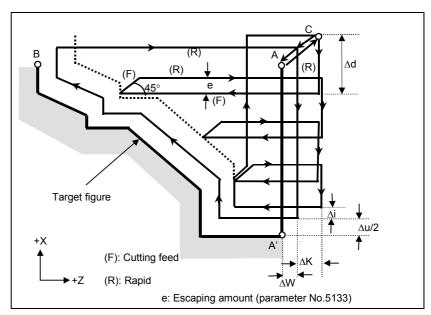


Fig. 6.4.1 (b) Cutting path of an outer surface rough cutting cycle with rough cutting finishing allowance (type I)

### **Explanation**

## - Operations

If a target figure passing through A, A', and B in this order is given by the program, a workpiece is cut away by depth of cut  $\Delta d$  at a time. The machining path varies as follows depending on whether the rough machining finishing allowance is specified.

- (1) When the rough cutting finishing allowance is not specified Cutting is performed by depth of cut  $\Delta d$  with finishing allowances  $\Delta u/2$  and  $\Delta w$  left, and rough cutting as finishing is performed according to the target figure program after the last machining.
- (2) When the rough cutting finishing allowance is specified Cutting is performed by depth of cut Δd with cutting allowances Δu/2+Δi and Δw+Δk left, and the tool returns to the start point (A) after the last cutting is performed. Then, rough machining as finishing is performed along the target figure to remove cutting allowances Δi and Δk.

Upon completion of rough machining as finishing, the block next to the sequence block specified by Q is executed.

#### NOTE

- 1 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
- When a constant surface speed control is privided, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71 block or the previous block is effective.

### - Escaping amount (e)

The escaping amount (e) is set in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5133	Depends on the increment	Radius programming	Not
	system for the reference axis.		required

## - Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the first axis on the plane (Z-axis for the ZX plane). The signs of  $\Delta u$  and  $\Delta w$  are as follows:

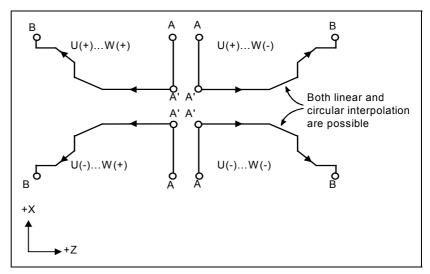


Fig. 6.4.1 (c) Four target figure patterns

### Limitation

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

#### Start block

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

When G00 is specified, cutting along path A-A' is performed in the positioning mode. When G01 is specified, cutting along path A-A' is performed in the linear interpolation mode.

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

#### **Check functions**

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

### **NOTE**

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

The following checks can also be made.

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

## - Types I and II Selection of type I or II

For G71, there are types I and II.

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

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

#### **NOTE**

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

### Selecting type I or II

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

- (1) When type I is selected Specify the second axis on the plane (X-axis for the ZX plane). Do not specify the first axis on the plane (Z-axis for the ZX plane).
- When type II is selected
  Specify the second axis on the plane (X-axis for the ZX plane)
  and first axis on the plane (Z-axis for the ZX plane).
  When you want to use type II without moving the tool along the
  first axis on the plane (Z-axis for the ZX plane), specify the
  incremental programming with travel distance 0 (W0 for the ZX
  plane).

## - Type I

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

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

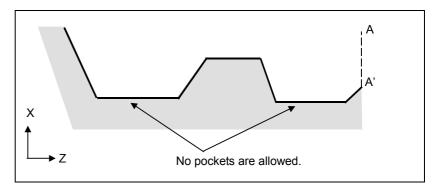


Fig. 6.4.1 (d) Figure which does not show monotone increase or decrease (type I)

## **⚠** CAUTION

If a figure does not show monotone change along the first or second axis on the plane, alarm PS0064 or 0329 is issued. If the movement does not show monotone change, but is very small, and it can be determined that the movement is not dangerous, however, the permissible amount can be specified in parameters Nos. 5145 and 5146 to specify that the alarm is not issued in this case.

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

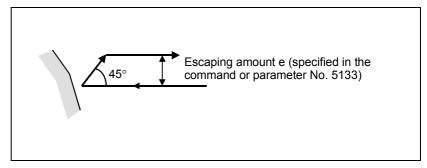


Fig. 6.4.1 (e) Cutting in the direction of 45 degrees (type I)

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

## - Type II

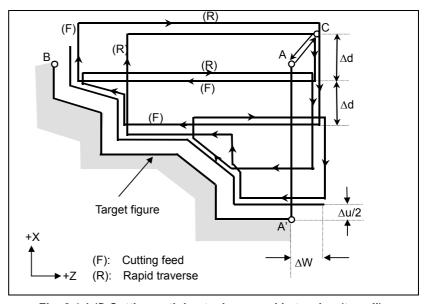


Fig. 6.4.1 (f) Cutting path in stock removal in turning (type II)

When the figure program for instructing a target figure passing through A, A', and B in this order is specified, a workpiece is cut away by depth of cut  $\Delta d$  at a time. In type II, cutting is performed along the figure after rough cutting in the direction of the plane first axis (z-axis for the ZX plane).

The machining path varies as follows depending on whether the rough machining finishing allowance is specified.

(1) When the rough cutting finishing allowance is not specified Cutting is performed by depth of cut Δd with finishing allowances Δu/2 and Δi left, and the tool returns to the start point (A) after the last cutting is performed (one pocket is assumed because Pn→Pm is parallel to the z-axis in the above figure, and

- the zone is cut). Then, rough machining as finishing is performed according to the finishing figure program with finishing allowances  $\Delta u/2$  and  $\Delta w$  left.
- (2) When the rough cutting finishing allowance is specified Cutting is performed by depth of cut  $\Delta d$  with cutting allowances  $\Delta u/2+\Delta i$  and  $\Delta w+\Delta k$  left, and the tool returns to the start point (A) after the last cutting is performed. Then, rough machining as finishing is performed along the target figure to remove cutting allowances  $\Delta i$  and  $\Delta k$ .

Upon completion of rough machining as finishing, the block next to the sequence block specified by Q is executed.

Type II differs from type I in the following points:

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

```
Example

ZX plane

G71 V10.0 R5.0;

G71 P100 Q200......;

N100 X(U)_ Z(W)_; (Specifies the two axes forming the plane.)

:;
:;
N200.....;
```

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

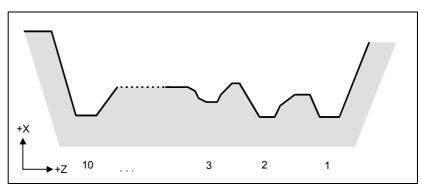


Fig. 6.4.1 (g) Figure having pockets (type II)

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

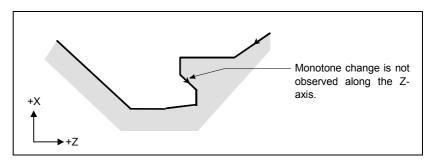


Fig. 6.4.1 (h) Figure which cannot be machined (type II)

## **⚠** CAUTION

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

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

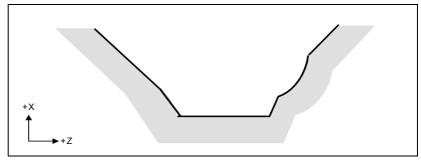


Fig. 6.4.1 (i) Figure which can be machined (type II)

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

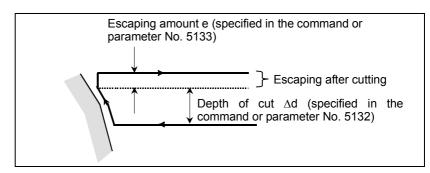


Fig. 6.4.1 (j) Cutting along the workpiece figure (type II)

The escaping amount e after cutting is set in parameter No. 5133. When moving from the bottom, however, the tool escapes to the direction of 45 degrees.

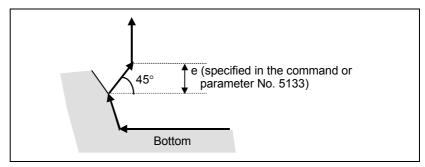


Fig. 6.4.1 (k) Escaping from the bottom to the direction of 45 degrees

- (4) When a position parallel to the first axis on the plane (Z-axis for the ZX plane) is specified in a block in the program for the target figure, it is assumed to be at the bottom of a pocket.
- (5) After all rough cutting terminates along the first axis on the plane (Z-axis for the ZX plane), the tool temporarily returns to the cycle start point. At this time, when there is a position whose height equals to that at the start point, the tool passes through the point in the position obtained by adding depth of cut  $\Delta d$  to the position of the figure and returns to the start point.

Then, rough cutting is performed as finishing along the target figure. At this time, the tool passes through the point in the obtained position (to which depth of cut  $\Delta d$  is added) when returning to the start point.

Bit 2 (RF2) of parameter No. 5105 can be set to 1 so that rough cutting as finishing is not performed.

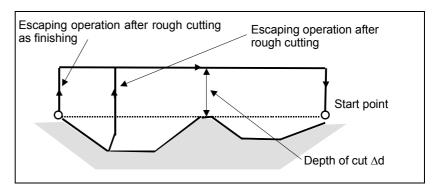


Fig. 6.4.1 (I) Escaping operation when the tool returns to the start point (type II)

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

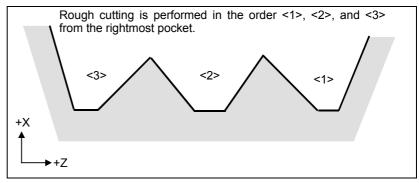


Fig. 6.4.1 (m) Rough cutting order in the case of monotone decrease (type II)

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

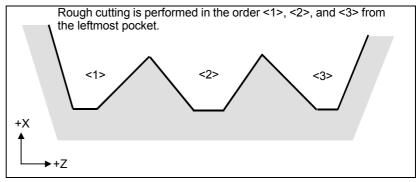


Fig. 6.4.1 (n) Rough cutting order in the case of monotone increase (type II)

35 3 34 24 25 2 9 10 28 30 26 8 15 7 20 16 12

The path in rough cutting is as shown below.

Fig. 6.4.1 (o) Cutting path for multiple pockets (type II)

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

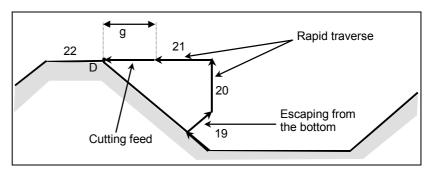


Fig. 6.4.1 (p) Details of motion after cutting for a pocket (type II)

Cuts the workpiece at the cutting feedrate and escapes to the direction of 45 degrees. (Operation 19)

Then, moves to the height of point D in rapid traverse. (Operation 20)

Then, moves to the position the amount of g before point D. (Operation 21)

Finally, moves to point D in cutting feed.

The clearance g to the cutting feed start position is set in parameter No. 5134.

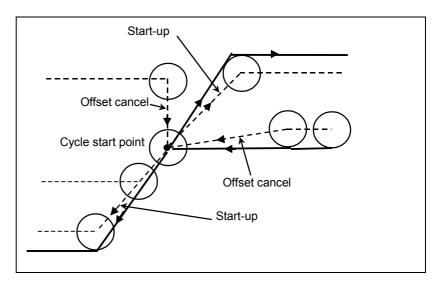
For the last pocket, after cutting the bottom, the tool escapes to the direction of 45 degrees and returns to the start point in rapid traverse. (Operations 34 and 35)

## **⚠** CAUTION

- 1 This CNC differs from the Series 16*i*/18*i*/21*i* in cutting of a pocket.
  - The tool first cuts the nearest pocket to the start point. After cutting of the pocket terminates, the tool moves to the nearest but one pocket and starts cutting.
- 2 When the figure has a pocket, generally specify a value of 0 for  $\Delta w$  (finishing allowance). Otherwise, the tool may dig into the wall on one side.

## - Tool nose radius compensation

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



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

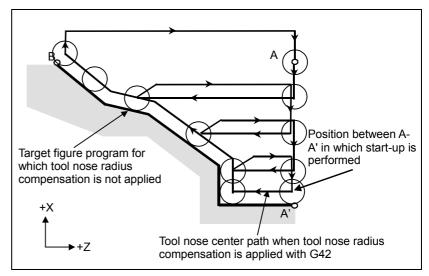
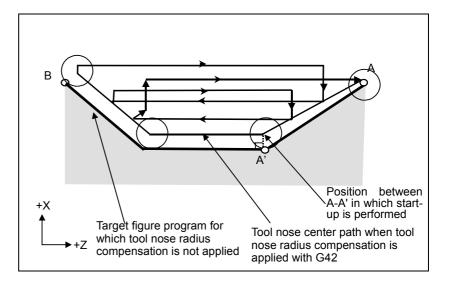


Fig. 6.4.1 (q) Path when tool nose radius compensation is applied



## **NOTE**

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

# 6.4.2 Stock Removal in Facing (G72)

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

#### **Format**

```
ZpXp plane
  G72 P(ns) Q(nf) U(\Deltau) W(\Deltaw) I(\Deltai) K(\Deltak) D(\Deltad) F(f ) S(s ) T(t );
  N (ns);
               The move command between A and B is specified in the blocks from
               sequence number ns to nf.
  N (nf);
YpZp plane
  G72 P(ns) Q(nf) V(\Deltaw) W(\Deltau) J(\Deltak) K(\Deltai) D(\Deltad) F(f ) S(s ) T(t );
  N (ns);
  N (nf);
XpYp plane
  G72 P(ns) Q(nf) U(\Deltaw) V(\Deltau) I(\Deltak) J(\Deltai) D(\Deltad) F(f) S(s) T(t);
  N (ns);
   ...
  N (nf);
∆d : Depth of cut
       The cutting direction depends on the direction AA'.
    : Sequence number of the first block for the program of finishing
     : Sequence number of the last block for the program of finishing
nf
      shape.
\Delta u: Distance of the finishing allowance in the direction of the second
      axis on the plane (X-axis for the ZX plane)
\Delta w: Distance of the finishing allowance in the direction of the first axis
      on the plane (Z-axis for the ZX plane)
     : Distance of the finishing allowance of the roughing in the direction
      of the second axis on the plane (X-axis for the ZX plane)
\Delta k: Distance of the finishing allowance of the roughing in the direction
      of the first axis on the plane (Z-axis for the ZX plane)
f,s,t: Any F, S, or T function contained in blocks ns to nf in the cycle is
      ignored, and the F, S, or T function in this G72 block is effective.
```

#### NOTE

Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

	Unit	Diameter/radius programming	Sign
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required
Δu	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the second axis on the plane.	Required
Δw	Depends on the increment system for the reference axis.	Depends on diameter/radius programming for the first axis on the plane.	Required
Δί	Depends on the increment system for the reference axis.	Radius programming	Not required
Δk	Depends on the increment system for the reference axis.	Radius programming	Not required

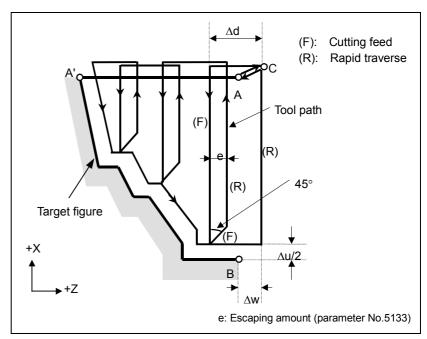


Fig. 6.4.2 (r) Cutting path in stock removal in facing (type I)

## **Explanation**

## - Operations

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

#### NOTE

- 1 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G72 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.
- 2 When a constant surface speed control is privided, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G72 block or the previous block is effective.

## - Escaping amount (e)

The escaping amount (e) is set in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5133	Depends on the increment system for the reference axis.	Radius programming	Not required

## - Target figure Patterns

The following four cutting patterns are considered. All of these cutting cycles cut the workpiece with moving the tool in parallel to the second axis on the plane (X-axis for the ZX plane). The signs of  $\Delta u$  and  $\Delta w$  are as follows:

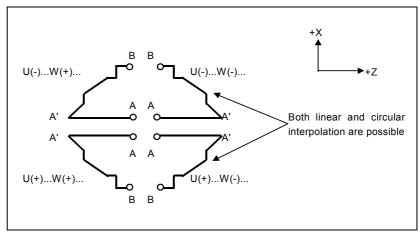


Fig. 6.4.2 (s) Signs of the values specified at U and W in stock removal in facing

#### Limitation

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

#### Start block

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

When G00 is specified, cutting along path A-A' is performed in the positioning mode. When G01 is specified, cutting along path A-A' is performed in the linear interpolation mode.

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

#### **Check functions**

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

#### NOTE

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

The following checks can also be made.

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

## - Types I and II Selection of type I or II

For G72, there are types I and II.

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

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

## Selecting type I or II

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

- (1) When type I is selected Specify the first axis on the plane (Z-axis for the ZX plane). Do not specify the second axis on the plane (X-axis for the ZX plane).
- (2) When type II is selected
  Specify the second axis on the plane (X-axis for the ZX plane)
  and first axis on the plane (Z-axis for the ZX plane).
  When you want to use type II without moving the tool along the second axis on the plane (X-axis for the ZX plane), specify the incremental programming with travel distance 0 (U0 for the ZX plane).

#### - Type I

G72 differs from G71 in the following points:

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

### - Type II

G72 differs from G71 in the following points:

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

## - Tool nose radius compensation

See the pages on which G71 is explained.

# 6.4.3 Pattern Repeating (G73)

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

#### **Format**

```
ZpXp plane
  G73 P(ns) Q(nf) U(\Deltau) W(\Deltaw) I(\Deltai) K(\Deltak) D(d) F(f ) S(s ) T(t ) ;
 N (ns);
              The move command between A and B is specified in the blocks from
              sequence number ns to nf.
  N (nf);
YpZp plane
  G73 P(ns) Q(nf) V(\Deltaw) W(\Deltau) J(\Deltak) K(\Deltai) D(d) F(f) S(s) T(t);
 N (ns);
   ...
  N (nf);
XpYp plane
  G73 P(ns) Q(nf) U(\Deltaw) V(\Deltau) I(\Deltak) J(\Deltai) D(d) F(f ) S(s ) T(t ) ;
 N (ns);
   ...
 N (nf);
      : Distance of escape in the direction of the second axis on the
        plane (X-axis for the ZX plane)
\Delta k
      : Distance of escape in the direction of the first axis on the plane
        (Z-axis for the ZX plane)
      : The number of division
d
        This value is the same as the repetitive count for rough cutting.
      : Sequence number of the first block for the program of finishing
ns
        shape.
nf
      : Sequence number of the last block for the program of finishing
        shape.
      : Distance of the finishing allowance in the direction of the second
Δu
        axis on the plane (X-axis for the ZX plane)
      : Distance of the finishing allowance in the direction of the first
        axis on the plane (Z-axis for the ZX plane)
f, s, t: Any F, S, and T function contained in the blocks between
        sequence number "ns" and "nf" are ignored, and the F, S, and T
        functions in this G73 block are effective.
```

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

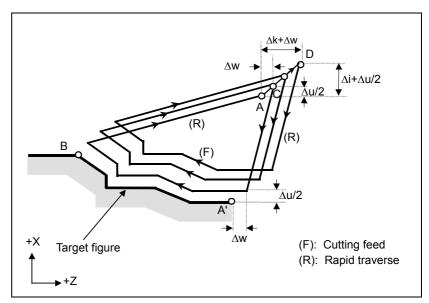


Fig. 6.4.3 (t) Cutting path in pattern repeating

## **Explanation**

- Operations

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

### **NOTE**

- 1 After cycle operation terminates, the tool returns to point A.
- 2 F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G73 block or the previous block are effective. M and second auxiliary functions are treated in the same way as F, S, and T functions.

## - Target figure patterns

As in the case of G71, there are four target figure patterns. Be careful about signs of  $\Delta u$ ,  $\Delta w$ ,  $\Delta i$ , and  $\Delta k$  when programming this cycle.

#### - Start block

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

When G00 is specified, cutting along path A-A' is performed in the positioning mode. When G01 is specified, cutting along path A-A' is performed in the linear interpolation mode.

### - Check function

The following check can be made.

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

## - Tool nose radius compensation

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

# **6.4.4** Finishing Cycle (G70)

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

#### **Format**

### G70 P(ns) Q(nf);

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

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

## **Explanation**

## - Operations

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

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

## - Target figure check function

The following check can be made.

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

## - Storing P and Q blocks

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

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

#### NOTE

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

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

#### - Return to the cycle start point

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

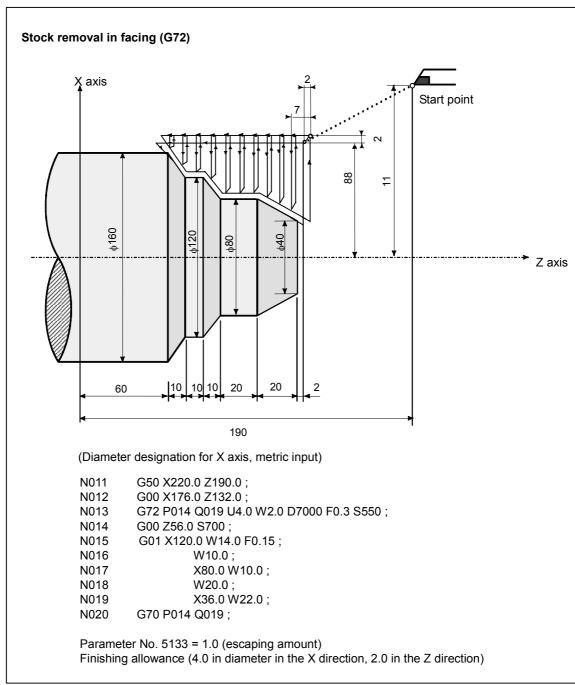
#### NOTE

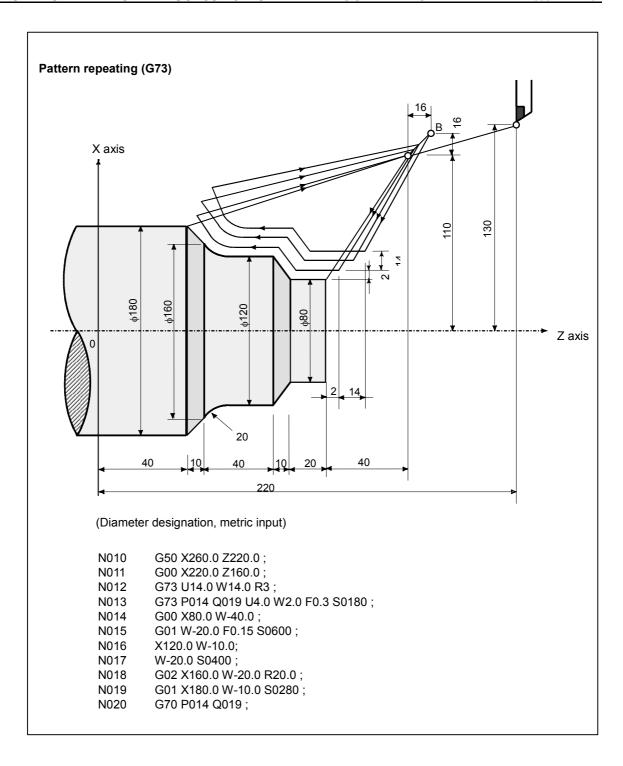
The tool returns to the cycle start point always in the nonlinear positioning mode regardless of the setting of bit 1 (LRP) of parameter No. 1401. Before executing a finishing cycle for a target figure with a pocket cut by G71 or G72, check that the tool does not interfere with the workpiece when returning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

### - Tool nose radius compensation

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

## Example





# 6.4.5 End Face Peck Drilling Cycle (G74)

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

#### **Format**

ZpXp-plan	ne		
$G74X(U)_Z(W)_I(\Delta i) K(\Delta k) D(\Delta d) F(f)$ ;			
YpZp-plar	ne		
G74Y(V)	)_ Z(W)_ J(∆k) K(∆i) D(∆d) F(f ) ;		
XpYp-plar	ne		
G74X(U	)_ Y(V)_ I(∆k) J(∆i) D(∆d) F(f ) ;		
X_,Y_,Z_	: Coordinate of the second axis on the plane		
	(X-axis for the ZX plane) at point B and		
	Coordinate of the first axis on the plane (Z-axis		
	for the ZX plane) at point C		
U_,V_,W_	: Travel distance along the second axis on the		
	plane (U for the ZX plane) from point A to B		
	Travel distance along the first axis on the plane		
	(W for the ZX plane) from point A to C		
$\Deltai$	: Travel distance in the direction of the second		
	axis on the plane (X-axis for the ZX plane)		
$\Delta k$	: Depth of cut in the direction of the first axis on		
	the plane (Z-axis for the ZX plane)		
$\Delta d$	: Relief amount of the tool at the cutting bottom		
f	: Feedrate		

	Unit	Diameter/radius programming	Sign
Δi	Depends on the increment	Radius programming	Not
ΔΙ	system for the reference axis.	Radius programming	required
Ale	Depends on the increment	Dadius programming	Not
Δk	system for the reference axis.	Radius programming	required
٨٨	Depends on the increment	Dadius programming	NOTE 1
∆d	system for the reference axis.	Radius programming	NOTET

### NOTE 1

- 1 Normally, specify a positive value for ∆d. When X (U) and ∆i are omitted, specify a value with the sign indicating the direction in which the tool is to escape.
- 2 Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

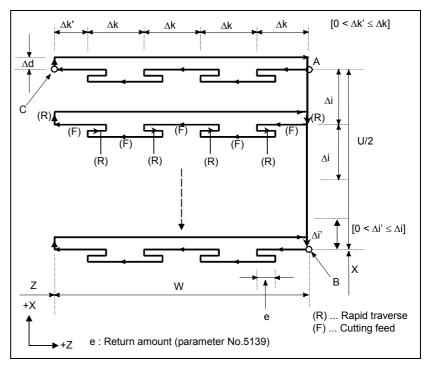


Fig. 6.4.5 (a) Cutting path in end face peek drilling cycle

## **Explanation**

## - Operations

A cycle operation of cutting by  $\Delta k$  and return by e is repeated. When cutting reaches point C, the tool escapes by  $\Delta d$ . Then, the tool returns in rapid traverse, moves to the direction of point B by  $\Delta i$ , and performs cutting again.

## - Return amount (e)

The escaping amount (e) is set in parameter No. 5139.

No.	Unit	Diameter/radius programming	Sign
5139	Depends on the increment system for the reference axis.	Radius programming	Not required

## - Tool nose radius compensation

Tool nose radius compensation cannot be applied.

# 6.4.6 Outer Diameter / Internal Diameter Drilling Cycle (G75)

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

#### **Format**

ZpXp-plane	
G75 X(U)	$_{\mathbf{Z}}(\mathbf{W})_{\mathbf{I}}(\Delta \mathbf{i}) \; \mathbf{K}(\Delta \mathbf{k}) \; \mathbf{D}(\Delta \mathbf{d}) \; \mathbf{F} \; (\mathbf{f} \; ) \; ;$
YpZp-plane	
G75 Y(V)	_ Z(W)_ J(∆k) K(∆i) D(∆d) F(f ) ;
XpYp-plane	
G75 X(U)	_ Y(V)_ I(Δk) J(Δi) D(Δd) F(f ) ;
X_,Y_,Z_	: Coordinate of the second axis on the plane
	(X-axis for the ZX plane) at point B and
	Coordinate of the first axis on the plane (Z-axis
	for the ZX plane) at point C
(U_,V_,W_)	: Travel distance along the second axis on the
	plane (U for the ZX plane) from point A to B
	Travel distance along the first axis on the
	plane (W for the ZX plane) from point A to C
Δi	: Depth of cut in the direction of the second axis
	on the plane (X-axis for the ZX plane)
Δk	: Travel distance in the direction of the first axis
	on the plane (Z-axis for the ZX plane)
Δd	: Relief amount of the tool at the cutting bottom
f	: Feedrate

	Unit	Diameter/radius programming	Sign
Δi	Depends on the increment	Radius programming	Not
ΔΙ	system for the reference axis.	radius programming	required
Δk	Depends on the increment	Radius programming	Not
ΔK	system for the reference axis.	Radius programming	required
٨٨	Depends on the increment	Dadius programming	NOTE
∆d	system for the reference axis.	Radius programming	NOTE

#### **NOTE**

- Normally, specify a positive value for Δd. When Z
   (W) and Δk are omitted, specify a value with the sign indicating the direction in which the tool is to escape.
- 2 Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.

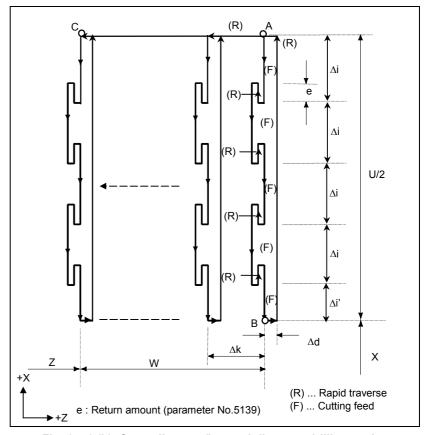


Fig. 6.4.6 (b) Outer diameter/internal diameter drilling cycle

# **Explanation**

#### - Operations

A cycle operation of cutting by  $\Delta i$  and return by e is repeated. When cutting reaches point B, the tool escapes by  $\Delta d$ . Then, the tool returns in rapid traverse, moves to the direction of point C by  $\Delta k$ , and performs cutting again.

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

#### - Return amount (e)

The escaping amount (e) is set in parameter No. 5133.

No.	Unit	Diameter/radius programming	Sign
5139	Depends on the increment	Radius	Not
5139	system for the reference axis.	programming	required

#### - Tool nose radius compensation

Tool nose radius compensation cannot be applied.

# **6.4.7** Multiple Threading Cycle

(G76 <G code system A/B>) (G78 <G code system C>)

The multiple threading cycle can select four cutting methods.

# **Format**

G76 X(U)_ Z(W)_ I(i) K(k) D(Δd) A(a) F(L) P(p) Q(q); YpZp-plane G76 Y(V)_ Z(W)_ J(k) K(i) D(Δd) A(a) F(L) P(p) Q(q); XpYp-plane G76 X(U)_ Y(V)_ I(k) J(i) D(Δd) A(a) F(L) P(p) Q(q); X_,Y_,Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length U_,V_,W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length a : Angle of tool nose	ZpXp-plane	<b>\</b>
<ul> <li>G76 Y(V)_ Z(W)_ J(k) K(i) D(Δd) A(a) F(L) P(p) Q(q);</li> <li>XpYp-plane</li> <li>G76 X(U)_ Y(V)_ I(k) J(i) D(Δd) A(a) F(L) P(p) Q(q);</li> <li>X_,Y_,Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length</li> <li>U_,V_,W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length</li> <li>a : Angle of tool nose</li></ul>		
<ul> <li>G76 Y(V)_ Z(W)_ J(k) K(i) D(Δd) A(a) F(L) P(p) Q(q);</li> <li>XpYp-plane</li> <li>G76 X(U)_ Y(V)_ I(k) J(i) D(Δd) A(a) F(L) P(p) Q(q);</li> <li>X_,Y_,Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length</li> <li>U_,V_,W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length</li> <li>a : Angle of tool nose</li></ul>	YpZp-plane	}
<ul> <li>G76 X(U)_ Y(V)_ I(k) J(i) D(Δd) A(a) F(L) P(p) Q(q);</li> <li>X_,Y_,Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length</li> <li>U_,V_,W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length</li> <li>a : Angle of tool nose</li></ul>		
<ul> <li>X_,Y_,Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length</li> <li>U_,V_,W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length</li> <li>a : Angle of tool nose</li></ul>	XpYp-plane	
<ul> <li>X_,Y_,Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length</li> <li>U_,V_,W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length</li> <li>a : Angle of tool nose</li></ul>	G76 X(U)	$Y(V) = I(k) J(i) D(\Delta d) A(a) F(L) P(p) Q(q)$ :
U_,V_,W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length  a : Angle of tool nose From 0 to 120 in steps of 1 degree (The default is 0.)  i : Taper amount If i = 0, ordinary straight threading can be made.  k : Height of thread  \( \Delta \) is the cut in 1st cut  L : Lead of thread  Depth of cut in 1st cut  Cutting method (one-edge threading with constant cutting amount by default or for P0) P1 : One-edge threading with constant cutting amount  P2 : Both-edge zigzag threading with constant cutting amount  P3 : One-edge threading with constant depth of cut  P4 : Both-edge zigzag threading with constant depth of cut  q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001)		: Coordinates of the cutting end point (point D in
<ul> <li>a : Angle of tool nose     From 0 to 120 in steps of 1 degree     (The default is 0.)</li> <li>i : Taper amount     If i = 0, ordinary straight threading can be made.</li> <li>k : Height of thread</li> <li>Δd : Depth of cut in 1st cut</li> <li>L : Lead of thread</li> <li>p : Cutting method (one-edge threading with constant cutting amount by default or for P0)     P1 : One-edge threading with constant cutting amount</li> <li>P2 : Both-edge zigzag threading with constant cutting amount</li> <li>P3 : One-edge threading with constant depth of cut</li> <li>P4 : Both-edge zigzag threading with constant depth of cut</li> <li>q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001</li> </ul>	U_,V_,W_ :	: Travel distance to the cutting end point (point D
From 0 to 120 in steps of 1 degree (The default is 0.)  i : Taper amount If i = 0, ordinary straight threading can be made.  k : Height of thread  \( \Delta \) : Depth of cut in 1st cut  L : Lead of thread  p : Cutting method (one-edge threading with constant cutting amount by default or for P0) P1 : One-edge threading with constant cutting amount  P2 : Both-edge zigzag threading with constant cutting amount  P3 : One-edge threading with constant depth of cut  P4 : Both-edge zigzag threading with constant depth of cut  q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001)		
<ul> <li>(The default is 0.)</li> <li>i Taper amount         If i = 0, ordinary straight threading can be made.</li> <li>k : Height of thread         Δd : Depth of cut in 1st cut         L : Lead of thread         p : Cutting method (one-edge threading with constant cutting amount by default or for P0)         P1 : One-edge threading with constant cutting amount         P2 : Both-edge zigzag threading with constant cutting amount         P3 : One-edge threading with constant depth of cut         P4 : Both-edge zigzag threading with constant depth of cut         q : Threading start angle shift         (From 0 to 360 degrees in steps of 0.001</li> </ul>		•
If i = 0, ordinary straight threading can be made.  k : Height of thread  Δd : Depth of cut in 1st cut  L : Lead of thread  p : Cutting method (one-edge threading with constant cutting amount by default or for P0)  P1 : One-edge threading with constant cutting amount  P2 : Both-edge zigzag threading with constant cutting amount  P3 : One-edge threading with constant depth of cut  P4 : Both-edge zigzag threading with constant depth of cut  9 : Threading start angle shift  (From 0 to 360 degrees in steps of 0.001		·
<ul> <li>k : Height of thread</li> <li>Δd : Depth of cut in 1st cut</li> <li>L : Lead of thread</li> <li>p : Cutting method (one-edge threading with constant cutting amount by default or for P0)</li> <li>P1 : One-edge threading with constant cutting amount</li> <li>P2 : Both-edge zigzag threading with constant cutting amount</li> <li>P3 : One-edge threading with constant depth of cut</li> <li>P4 : Both-edge zigzag threading with constant depth of cut</li> <li>q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001</li> </ul>	i :	Taper amount
<ul> <li>Δd : Depth of cut in 1st cut</li> <li>L : Lead of thread</li> <li>p : Cutting method (one-edge threading with constant cutting amount by default or for P0)</li> <li>P1 : One-edge threading with constant cutting amount</li> <li>P2 : Both-edge zigzag threading with constant cutting amount</li> <li>P3 : One-edge threading with constant depth of cut</li> <li>P4 : Both-edge zigzag threading with constant depth of cut</li> <li>q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001</li> </ul>		
L : Lead of thread p : Cutting method (one-edge threading with constant cutting amount by default or for P0) P1 : One-edge threading with constant cutting amount P2 : Both-edge zigzag threading with constant cutting amount P3 : One-edge threading with constant depth of cut P4 : Both-edge zigzag threading with constant depth of cut q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001		
p : Cutting method (one-edge threading with constant cutting amount by default or for P0) P1 : One-edge threading with constant cutting amount P2 : Both-edge zigzag threading with constant cutting amount P3 : One-edge threading with constant depth of cut P4 : Both-edge zigzag threading with constant depth of cut q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001		·
constant cutting amount by default or for P0) P1 : One-edge threading with constant cutting amount P2 : Both-edge zigzag threading with constant cutting amount P3 : One-edge threading with constant depth of cut P4 : Both-edge zigzag threading with constant depth of cut  q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001		
P1 : One-edge threading with constant cutting amount P2 : Both-edge zigzag threading with constant cutting amount P3 : One-edge threading with constant depth of cut P4 : Both-edge zigzag threading with constant depth of cut  q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001	p :	` ` `
amount P2: Both-edge zigzag threading with constant cutting amount P3: One-edge threading with constant depth of cut P4: Both-edge zigzag threading with constant depth of cut  q: Threading start angle shift (From 0 to 360 degrees in steps of 0.001		
cutting amount P3 : One-edge threading with constant depth of cut P4 : Both-edge zigzag threading with constant depth of cut q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001		
of cut P4 : Both-edge zigzag threading with constant depth of cut q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001		
P4 : Both-edge zigzag threading with constant depth of cut q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001		P3 : One-edge threading with constant depth
q : Threading start angle shift (From 0 to 360 degrees in steps of 0.001		P4 : Both-edge zigzag threading with constant
(From 0 to 360 degrees in steps of 0.001	q :	· · · · · · · · · · · · · · · · · · ·
40g1000 <i>)</i>		degrees)

#### NOTE

- 1 Even if pocket calculator type decimal point programming is specified (DPI (bit 0 of parameter No. 3401) = 1), the unit of address D is least input increment. In addition, when a decimal point is input in address D, the alarm (PS0007) is issued.
- 2 A decimal point included in address A has no meaning. That is, A120. is equivalent to A120 in specifying 120 degrees.
- 3 To use P2, P3, or P4 as a cutting method, the option for multiple repetitive lathe canned cycle II is required.

	Unit	Diameter/radius programming	Sign
i	Depends on the increment system for the reference axis.	Radius programming	Required
k	Depends on the increment system for the reference axis.	Radius programming	Not required
Δd	Depends on the increment system for the reference axis.	Radius programming	Not required

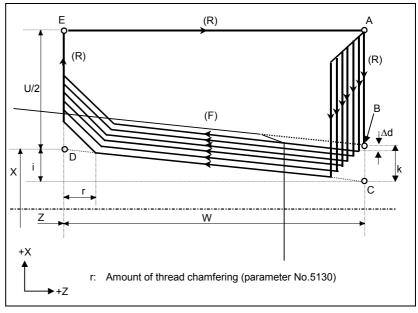


Fig. 6.4.7 (c) Cutting path in multiple threading cycle

### **Explanation**

# - Operations

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

The time constant for acceleration/deceleration after interpolation and FL feedrate for thread chamfering and the feedrate for retraction after chamfering are the same as for thread chamfering with canned cycle.

# **⚠** CAUTION

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

# - Cutting method

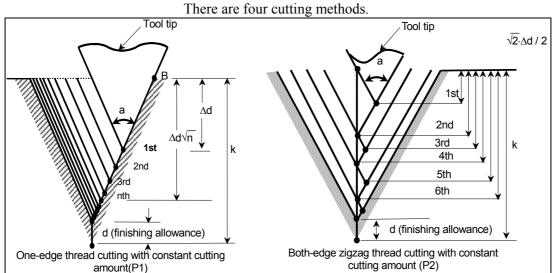


Fig. 6.4.7 (d) One-edge threading with constant cutting amount, both-edge zigzag threading with constant cutting amount (P1/2)

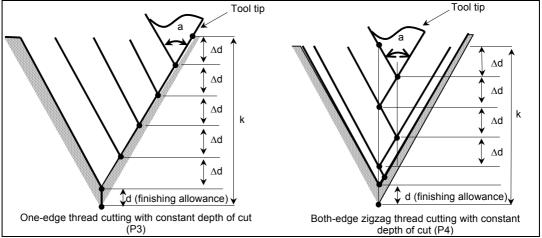


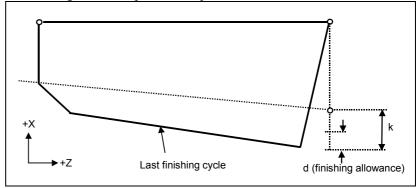
Fig. 6.4.7 (e) One-edge threading with constant depth of cut, both-edge zigzag threading with constant depth of cut (P3/4)

# - Repetitive count in finishing

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

The repetitive count is set in parameter No. 5142.

If the setting is 0, the operation is performed once.



# - Minimum depth of cut

When a cutting method with constant cutting amount is selected (P1 or P2), clamping can be performed with the minimum depth of cut to prevent the depth of cut from becoming too small.

The minimum depth of cut is set in parameter No. 5140.

No.	Unit	Diameter/radius programming	Sign
5140	Depends on the increment	Radius	Not
3140	system for the reference axis.	programming	required

# - Finishing allowance

The finishing allowance is set in parameter No. 5141.

No.	Unit	Diameter/radius programming	Sign
5141	Depends on the increment	Radius	Not
	system for the reference axis.	programming	required

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

The signs of incremental dimensions for the cycle shown in Fig. 6.4.7 (q) are as follows:

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

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

Taper amount (i):

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

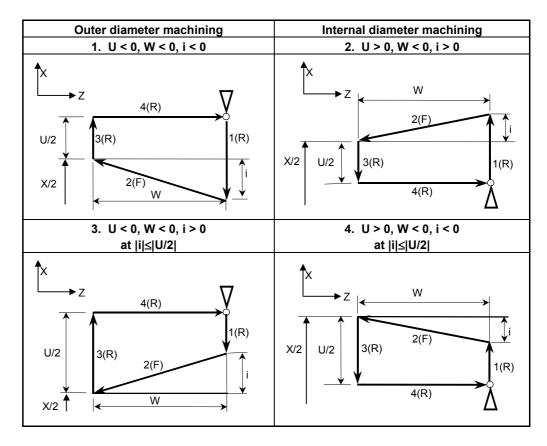
Height of thread (k):

Plus (always specified with a plus sign)

Depth of cut in the first cut ( $\Delta d$ ):

Plus (always specified with a plus sign)

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



#### - Time constant and FL feedrate for threading

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

### - Thread chamfering

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

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

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

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

#### NOTE

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

### - Retraction after chamfering

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

Parameter CFR (No. 1611#0)	Parameter No. 1466	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and retraction feedrate specified in parameter No. 1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No. 1626), FL feedrate (parameter No. 1627), and rapid traverse rate specified in parameter No. 1420.
1		Performs an in-position check before retraction and uses the type of acceleration/deceleration after interpolation in rapid traverse, time constant for rapid traverse, FL feedrate, and rapid traverse rate specified in parameter No. 1420.

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

#### NOTE

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

### - Shifting the start angle

Address Q can be used to shift the start angle of threading.

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

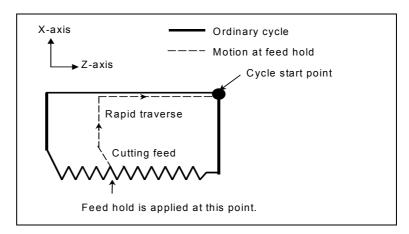
### - Feed hold in a threading cycle

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

#### - Feed hold when the threading cycle retract function is used

When the "threading cycle retract" option function is used, feed hold may be applied during threading in a multiple threading cycle (G76). In this case, the tool quickly retracts in the same way as for the last chamfering in a threading cycle and returns to the start point in the current cycle (position where the workpiece is cut by  $\Delta dn$ ).

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



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



#### 

Another feed hold cannot be performed during retraction.

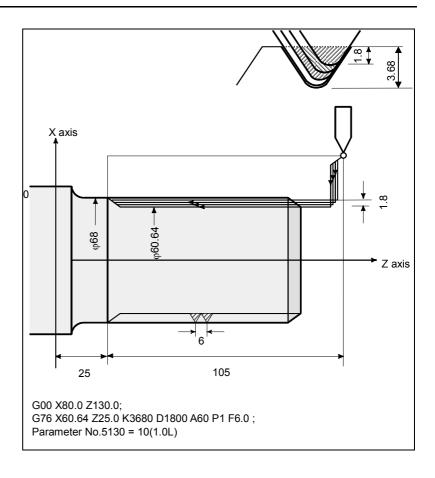
#### - Inch threading

Inch threading specified with address E is not allowed.

#### - Tool nose radius compensation

Tool nose radius compensation cannot be applied.

# Example



# **6.4.8** Restrictions on Multiple Repetitive Cycle

### **Programmed commands**

- Program memory

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

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

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

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

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

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

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

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

- Dwell (G04)
- G00, G01, G02, and G03
- Custom macro branch and repeat command
  The branch destination must be between the sequence numbers specified at P and Q, however. High-speed branch specified by bits 1 and 4 of parameter No. 6000 is invalid. No custom macro call (simple, modal, or subprogram call) cannot be specified.
- Direct drawing dimension programming command and chamfering and corner R command

Direct drawing dimension programming and chamfering and corner R require multiple blocks to be specified. The block with the last sequence number specified at Q must not be an intermediate block of these specified blocks.

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

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

#### Relation with other functions

#### - Manual intervention

While a multiple repetitive cycle (G70 to G76) is being executed, it is possible to stop the cycle and to perform manual intervention.

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

### - Interruption type macro

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

### - Program restart and tool retract and recover

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

# - Axis name and second auxiliary functions

Even if address U, V, W, or A is used as an axis name or second auxiliary function, data specified at address U, V, W, or A in a G71 to G73 or G76 block is assumed to be that for the multiple repetitive cycle.

# 6.5 CANNED CYCLE FOR DRILLING

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

Table 6.5 (a) lists canned cycles for drilling.

Table 6.5 (a) Canned cycles for drilling

G code	Drilling operation (-Z direction)	Operation in the bottom hole position	Retraction operation (-Z direction)	Applications
G83.1	Cutting feed/ intermittent		Rapid traverse	High-speed peck drilling cycle
G80				Canceling
G81	Cutting feed		Rapid traverse	Drilling, Spot drilling
G82	Cutting feed	Dwell	Rapid traverse	Drilling, Counter boring
G83	Cutting feed/intermittent		Rapid traverse	Peck drilling cycle
G84	Cutting feed	$\begin{array}{c} Dwell \to \\ Spindle \; CCW \end{array}$	Cutting feed	Tapping
G85	Cutting feed		Cutting feed	Boring
G89	Cutting feed	Dwell	Cutting feed	Boring

### **Explanation**

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

Operation 1 ....... Positioning of X and Z axis (Another axis may be targeted.)

Operation 2 ....... Rapid traverse up to point R level

Operation 3 ...... Hole machining

Operation 4 ...... Operation at the bottom of a hole

Operation 5 ...... Retraction to point R level

Operation 6 ......... Rapid traverse up to the initial level

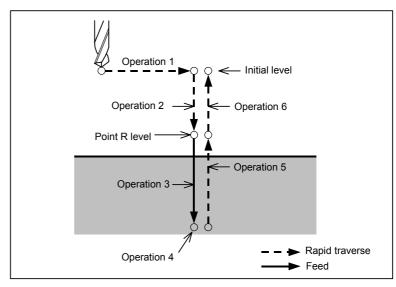


Fig. 6.5 (f) Operation sequence of canned cycle for drilling

# - Positioning plane

A positioning plane is determined by plane selection with G17, G18, and G19.

The axes other than the drilling axis are used as positioning axes.

# - Drilling axis

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.

The basic axis (X, Y, or Z) that does not exist on the positioning plane or its parallel axis is used as the drilling axis.

The axis address of the drilling axis specified in the same block as the G codes (G81 to G89) determines whether a basic axis or one of parallel axes is used as the drilling axis.

If the axis address of the drilling axis is not specified, the basic axis is used as the drilling axis.

Table 6.5 (b) Positioning plane and drilling axis

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

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

### - Example

Suppose parameter No. 1022 is set so that U, V, and W are the parallel axes of X, Y, and Z, respectively.

G17 G81 ..... Z \_ \_: Drilling axis is Z axis.

G17 G81 ...... W \_ : Drilling axis is W axis.
G18 G81 ...... Y \_ : Drilling axis is W axis.
G18 G81 ...... V \_ : Drilling axis is V axis.
G19 G81 ...... V \_ : Drilling axis is X axis.
G19 G81 ..... U \_ : Drilling axis is U axis.

G17, G18, and G19 may be specified in a block in which G73 to G89 are not present.

### **⚠** CAUTION

Before switching between drilling axes, cancel canned cycles.

#### NOTE

The Z-axis can always be used as the drilling axis by setting FXY (bit 0 of parameter No.5101). When FXY is 0, the Z-axis is always used as the drilling

### - Specification of point R

In the Series 16/18 command format, the distance from the initial level to point R is specified using an incremental value during specification of point R.

In the Series 15 command format, the specification method depends on RAB (bit 6 of parameter No. 5102). When RAB = 0, an incremental value is always used for specification. When RAB = 1 for G code system A, an absolute value is used for specification. When RAB = 1 for G code system B, C, an absolute value is used in G90 mode while an incremental value is used in G91 mode.

	Series 15 command format			
Parame	eter RAB (No.5	5102#6) = 1	RAB=0	
G code system A	G code system B,C		la ava ma antal	Incremental
A book its	G90	G91	Incremental	
Absolute	Absolute	Incremental		

#### - Diameter/radius programming

The diameter/radius specification of canned cycles for drilling R command in the series 15 command format can be matched with the diameter/radius specification of the drilling axis by setting RDI (bit 7 of parameter No.5102) to 1.

- P

In the following G codes, dwell operation differs between Series 15 and Series 15-T.

Operation of the Series 15 format

In G83, G83.1, G84, and G84.2, dwelling is performed only when address P is specified in a block.

Operation of Series 15-T

In G83 and G83.1, dwelling is not performed.

In G84 and G84.2, dwelling with address P can be performed by setting DWL (bit 1 of parameter No.6200). Address P is modal data.

- Q

Address Q is always specified by using an incremental value during specification of a radius.

#### - Feedrate for G85 and G89

In G85 and G89, the feedrate from point Z to point R is double the cutting feedrate. For Series 15-T, it is the same as the cutting feedrate.

# - Drilling mode

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

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

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

### - Return point level G98/G99

In G code system A, the tool returns to the initial level from the bottom of a hole. In G code system B or C, specifying G98 returns the tool to the initial level from the bottom of a hole and specifying G99 returns the tool to the point-R level from the bottom of a hole.

The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.

The initial level does not change even when drilling is performed in the G99 mode.

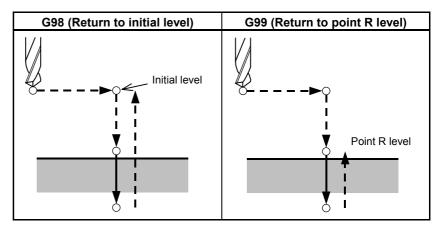


Fig. 6.5 (g) Initial level and point R level

# - Number of repeats

To repeat drilling for equally-spaced holes, specify the number of repeats in L .

L is effective only within the block where it is specified.

Specify the first hole position in incremental mode.

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

Number of repeats L The maximum command value = 9999

When L0 is specified, drilling data is just stored without drilling being performed.

#### NOTE

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

#### - C axis clamp

The M code for C axis clamp can be specified in the Series 16/18 command format, but it cannot be specified in the Series 15 command format.

### - Disabling the Series 15 format

The Series 15 command format can be disabled only during a canned cycle for drilling by setting F16 (bit 3 of parameter No.5102) to 1. However, the repetitive count must be specified by address L.

# **A** CAUTION

If F16 (bit 3 of parameter No.5102) is set to 1, the settings of RAB (bit 6 of No.5102) and RDI (bit 7 of No.5102) are disabled, and operation when RAB=0 and RDI=0 is performed.

#### - Cancel

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

#### **Group 01 G codes (Example)**

G00: Positioning (rapid traverse)

G01: Linear interpolation

G02: Circular interpolation (CW) or helical interpolation (CW) G03: Circular interpolation (CCW) or helical interpolation (CCW)

# - Symbols in figures

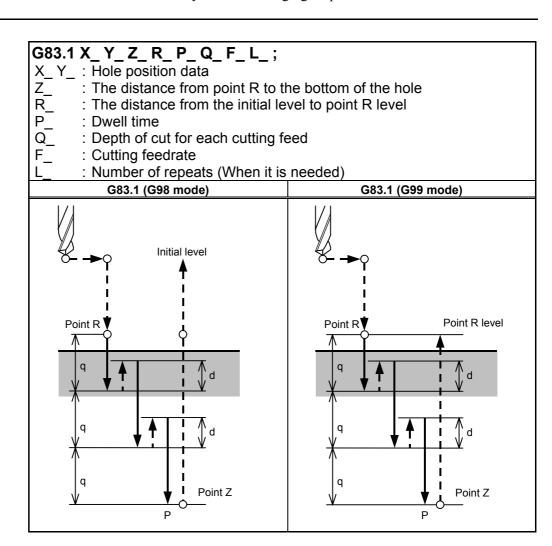
Subsequent subsections explain the individual canned cycles. Figures in these explanations use the following symbols:

	Positioning (rapid traverse G00)
<b>→</b>	Cutting feed (linear interpolation G01)
Р	Dwell

# 6.5.1 High-speed Peck Drilling Cycle (G83.1)

This cycle performs high-speed peck drilling. It performs cutting feed intermittently while discharging chips.

#### **Format**



# **Explanation**

#### - Operations

Since intermittent feed in the Z-axis direction makes discharge of chips easier and allows the fine setting of the escaping amount, efficient machining can be performed.

Escape amount d is set in parameter No. 5114.

Escaping moves in rapid traverse.

#### - Spindle rotation

Before specifying G83.1, use an auxiliary function (M code) to rotate the spindle.

# - Auxiliary function

If the G83.1 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

#### Limitation

### - Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

#### - Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

Dwelling is performed only when address P is specified in a block.

- Q

Q must be specified in a block in which drilling is instructed. Otherwise, data is not stored as modal data.

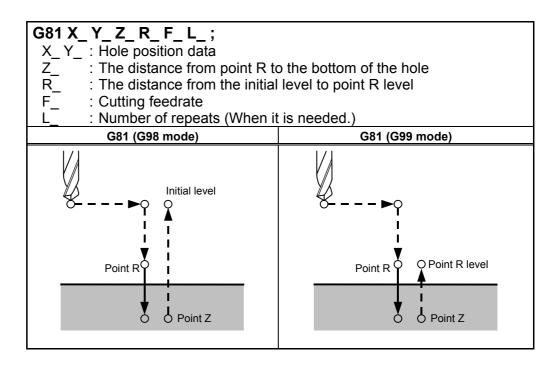
#### - Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G83.1 is specified. This cancels G83.1.

# **6.5.2** Drilling Cycle, Spot Drilling Cycle (G81)

The normal drilling cycle is used. The tool is then retracted from the bottom of the hole in rapid traverse.

#### **Format**



#### **Explanation**

- Operations

Rapid traverse to the point R level is performed after positioning of the X- and Y- axes.

Then, drilling is performed from point R level to point Z.

Escaping moves in rapid traverse.

- Spindle rotation

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

- Auxiliary function

If the G81 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

#### Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- Cancel

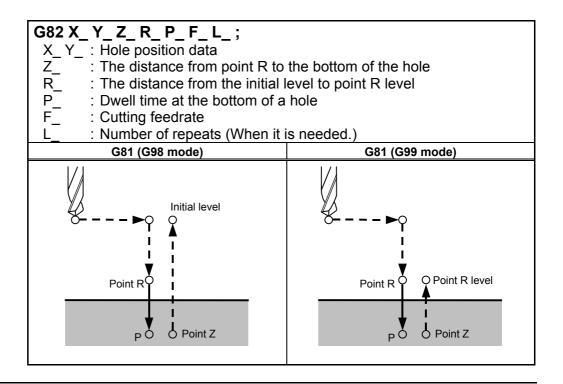
The G codes (G00 to G03) in group 01 must not be specified in the block in which G81 is specified. This cancels G81.

# 6.5.3 Drilling Cycle, Counter Boring (G82)

The normal drilling cycle is used. Cutting feed is performed to the bottom of the hole, dwelling is performed at the bottom, and then escaping from the bottom is performed in rapid traverse.

The accuracy of the hole depth is improved.

#### **Format**



# **Explanation**

# - Operations

Rapid traverse to the point R level is performed after positioning of the X- and Y- axes.

Then, drilling is performed from point R level to point Z.

Dwelling is performed at the bottom of the hole and then escaping is performed in rapid traverse.

# - Spindle rotation

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

### - Auxiliary function

If the G82 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

#### Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

P must be specified in a block in which drilling is instructed. Otherwise, data is not stored as modal data.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G82 is specified. This cancels G82.

#### 6.5.4 **Peck Drilling Cycle (G83)**

Peck drilling is performed.

Cutting feed is performed intermittently to the bottom of the hole while chips are discharged.

#### **Format**

# G83 X\_Y\_Z\_R\_P\_Q\_F\_L\_;

X\_Y\_: Hole position data

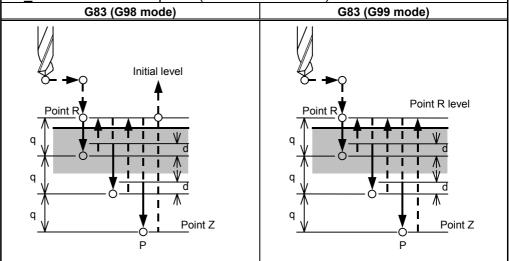
: The distance from point R to the bottom of the hole The distance from the initial level to point R level

: Dwell time

: Depth of cut for each cutting feed

: Cutting feedrate

: Number of repeats (When it is needed.)



### **Explanation**

### - Operations

Q indicates the depth of cut for each operation and is specified by an incremental value.

In the second and later cutting operations, rapid traverse is changed to cutting feed at the point distance "d" back from the previously drilled position. "d" is set in parameter No. 5115.

A positive value must be specified for Q. A negative value is ignored.

# - Spindle rotation

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

# - Auxiliary function

If the G83 command and M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

#### Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for

drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis,

drilling is not performed.

- P

Dwelling is performed only when address P is specified in a block.

- Q

Q must be specified in a block in which drilling is instructed.

Otherwise, data is not stored as modal data.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the

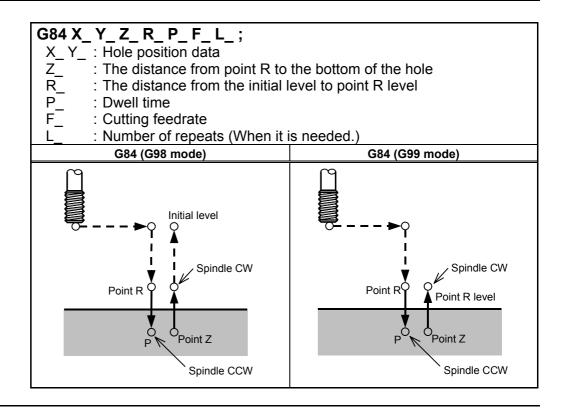
block in which G83 is specified. This cancels G83.

# **6.5.5** Tapping Cycle (G84)

This cycle performs tapping.

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

#### **Format**



# **Explanation**

#### - Operations

Tapping is performed by rotating the spindle clockwise.



Feedrate override is ignored during tapping. In addition, applying feed hold does not stop the machine until return operation is completed.

#### - Spindle rotation

Before specifying G84, use an auxiliary function (M code) to rotate the spindle.

When drilling in which the distance from the hole position and initial level to the point R level is short is continuously performed, the spindle may not reach the normal speed by the time cutting operation for the hole is ready to be performed. In this case, reserve a time by inserting dwelling by G04 before each drilling operation without specifying repetitive count L.

Since this consideration may not be required depending on the machine type, refer to the manual issued by the machine tool builder.

# - Auxiliary function

If the G84 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

#### Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

Dwelling is performed only when address P is specified in a block.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G84 is specified. This cancels G84.

#### NOTE

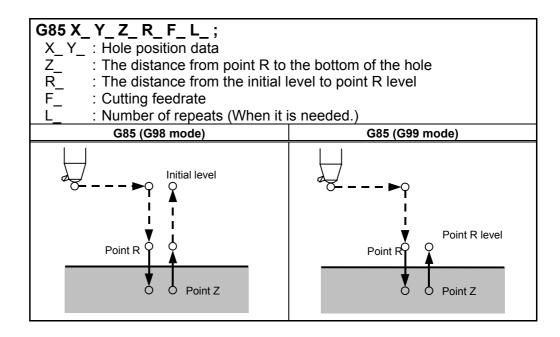
Set M5T (bit 6 of parameter No. 5101) to specify whether the spindle stop command (M05) is specified before the command for rotating the spindle in the forward or reverse direction (M03 or M04) is specified.

For details, refer to the manual issued by the machine tool builder.

# **6.5.6** Boring Cycle (G85)

This cycle is used to bore a hole.

#### **Format**



# **Explanation**

- Operations

Rapid traverse to the point R level is performed after positioning of the X- and Y- axes.

Then, drilling is performed from point R level to point Z. After reaching point Z, return to point R in cutting feed.

- Spindle rotation

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

- Auxiliary function

If the G85 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

#### Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

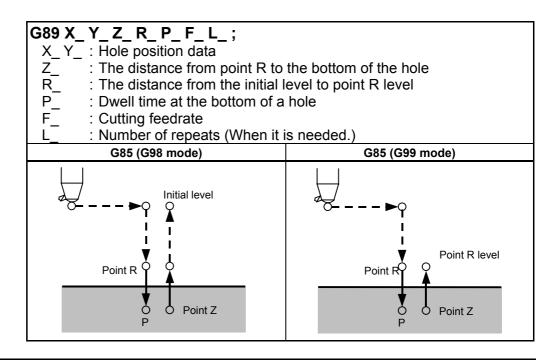
#### - Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G85 is specified. This cancels G85.

# **6.5.7** Boring Cycle (G85)

This cycle is used to bore a hole.

#### **Format**



# **Explanation**

- Operations

This is the same as G85, but dwelling is performed at the bottom of the hole.

- Spindle rotation

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

- Auxiliary function

If the G89 command and an M code are specified in the same block, the M code is executed at the first positioning. When repetitive count L is specified, the operation above is performed for the first time and the M code is not performed second and later times.

#### Limitation

- Axis switching

Before switching between drilling axes, cancel canned cycles for drilling.

- Drilling

In a block that does not include X, Y, Z, R, or an additional axis, drilling is not performed.

- P

P must be specified in a block in which drilling is instructed. Otherwise, data is not stored as modal data.

- Cancel

The G codes (G00 to G03) in group 01 must not be specified in the block in which G89 is specified. This cancels G89.

# 6.5.8 Canned Cycle for Drilling Cancel (G80)

G80 cancels canned cycle for drilling.

#### **Format**

G80;

#### **Explanation**

Canned cycle for drilling is canceled to perform normal operation. Point R and point Z are cleared.

Other drilling data is also canceled (cleared).

# 6.5.9 Precautions to be Taken by Operator

#### - Reset and emergency stop

Even when the controller is stopped by resetting or emergency stop in the course of drilling cycle, the drilling mode and drilling data are saved; with this mind, therefore, restart operation.

### - Single block

When drilling cycle is performed with a single block, the operation stops at the end points of operations 1, 2, 6 in Fig. 6.5 (a).

Consequently, it follows that operation is started up 3 times to drill one hole. The operation stops at the end points of operations 1, 2 with the feed hold lamp ON. The operation stops in the feed hold conditions at the end point of operation 6 if the repeat remains, and it stops in stop conditions in other cases.

#### - Feed hold

When "Feed Hold" is applied between operations 3 and 5 by G84/G88, the feed hold lamp lights up immediately if the feed hold is applied again to operation 6.

#### - Override

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

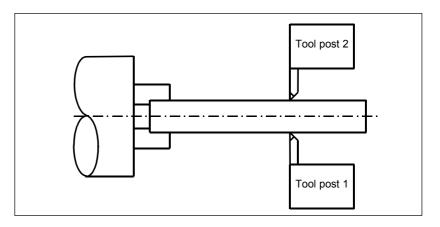
7

# **MUITI-PATH CONTROL FUNCTION**

# **7.1** BALANCE CUT (G68, G69)

#### Overview

When a thin workpiece is to be machined as shown below, a precision machining can be achieved by machining each side of the workpiece with a tool simultaneously; this function can prevent the workpiece from warpage that can result when only one side is machined at a time (see the figure below). When both sides are machined at the same time, the movement of one tool must be in phase with that of the other tool. Otherwise, the workpiece can vibrate, resulting in poor machining. With this function, the movement of one tool post can be easily synchronized with that of the other tool post.



#### **Format**

#### G68 (Pp); Balance cut mode on

- p: Number which specifies a balance cut combination
  - (1) In the binary value specification mode, specify the sum of the binary values corresponding to the numbers of paths which require balance cutting.
  - (2) In the path number specification mode, specify the numbers of all paths that require balance cutting in combination.

When address P is not specified, balance cutting is performed for paths 1 and 2.

G69; Balance cut mode cancel

#### **Explanation**

When G68 is specified in the programs for tool posts 1 and 2, the balance cut mode is set to on. When G69 is specified, the balance cut mode is canceled.

When G68 or G69 is specified for either tool post, the tool post waits until G68 or G69 is specified for the other tool post.

In the balance cut mode, balance cutting is performed when a move command in cutting feed is specified for both tool posts.

In balance cutting, the tool posts start moving simultaneously in each block in which a move command in cutting feed is specified.

Specify G68 or G69 in a single block. In the same block, only address P can be specified.

If G68 or G69 is specified incorrectly or the value specified at address P is invalid, alarm PS0163 occurs.

The following two methods for specifying a value at address P are available and can be selected using bit 1 (MWP) of parameter No. 8103.

Bit 1 (MWP) of parameter No. 8103	Address P specification method
0	Uses the method of specifying the sum of the binary values corresponding to the numbers of paths which require balance cutting (balance cut specified with binary values).
1	Uses the method of specifying the numbers of all paths that require balance cutting in combination (balance cut specified with path numbers).

#### NOTE

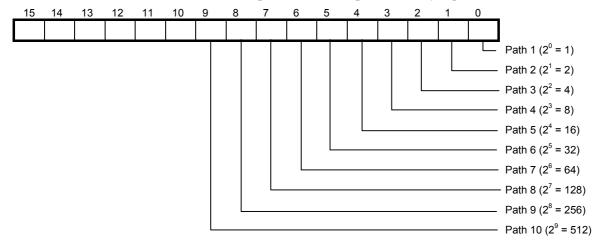
- 1 Balance cutting is not performed in dry run or machine lock state. G68 or G69 specified for one tool post is synchronized with G68 or G69 specified for the other tool post, however.
- 2 In the balance cut mode, G68 specified for one tool post is not synchronized with G68 specified for the other tool post. In the balance cut cancel mode, G69 specified for one tool post is not synchronized with G69 specified for the other tool post.
- 3 Balance cutting is not performed in a block in which 0 is specified for the travel distance.
- 4 Balance cutting is not performed when rapid traverse is specified.

#### - Balance cut specified with binary values

When bit 1 (MWP) of parameter No. 8103 is set to 0, the value specified at address P is assumed to be obtained using binary values. The following table lists the path numbers and corresponding binary values.

Path number	Binary value (decimal number)
1	1
2	2
3	4
4	8
5	16
6	32
7	64
8	128
9	256
10	512

The bit position of each path in binary representation is shown below.



To perform balance cutting for all of paths 1, 2, and 3, the P value is obtained as follows:

Binary value of path 1 1 (0000 0000 0000 0001) Binary value of path 2 2 (0000 0000 0000 0010) Binary value of path 3 4 (0000 0000 0000 0100) Sum 7 (0000 0000 0000 0111)

Balance cutting can be performed for all of the three paths by specifying P7 together with the balance cut G code (G68).

To perform balance cutting for all of paths 1, 3, 5, 7, and 9, the P value is obtained as follows:

Binary value of path 1 1 (0000 0000 0000 0001)

Binary value of path 3 4 (0000 0000 0000 0100)

Binary value of path 5 16 (0000 0000 0001 0000)

Binary value of path 7 64 (0000 0000 0100 0000)

Binary value of path 9 256 (0000 0001 0000 0000)

Sum 341 (0000 0001 0101 0101)

Balance cutting can be performed for all of the five paths by specifying P341 together with the balance cut G code (G68).

#### - Balance cut specified with a combination of path numbers

When bit 1 (MWP) of parameter No. 8103 is set to 1, the value specified at address P is assumed to be a combination of path numbers. The following table lists the path numbers and corresponding values.

Path number	Value (decimal number)
1	1
2	2
3	3
4	4
5	5
6	6
7	7
8	8
9	9
10	0

To perform balance cutting for all of paths 1, 2, and 3, the P value is a number consisting of 1, 2, and 3.

Example) P123

There are no restrictions on the order in which the numeric characters are specified, the following six possible values can be specified:

P123, P132, P213, P231, P312, P321

Path numbers specified in combination in different orders for different paths are effective as long as the numbers of the relevant paths are specified.

Example) The following are treated as the same P value and these paths can be made to wait for one another:

M200P123 for path 1, M200P231 for path 2, and M200P321 for path 3

To perform balance cutting for all of paths 1, 3, 5, 7, and 9, the P value is a number consisting of 1, 3, 5, 7, and 9.

Example) P13579

#### - Balance cutting with path 10

For balance cutting with path 10, specify a value of 0 for the combination.

If a number begins with 0, 0 cannot be recognized. Specify 0 in the second or subsequent digit from the left.

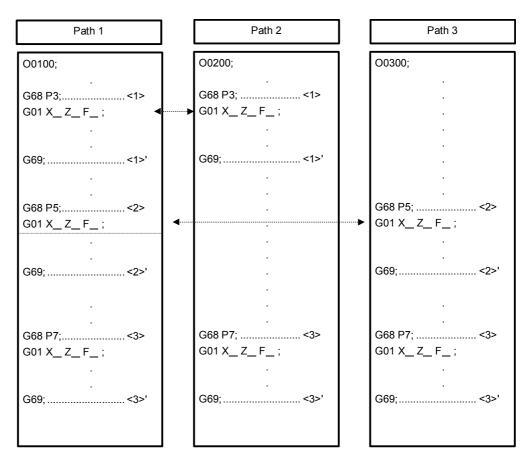
Incorrect example) P013579 Correct example) P103579

In the incorrect example, the P value is assumed to be the same as P13579 and balance cutting with path 10 cannot be performed.

#### **Example**

#### - When the value specified at P is obtained using binary values

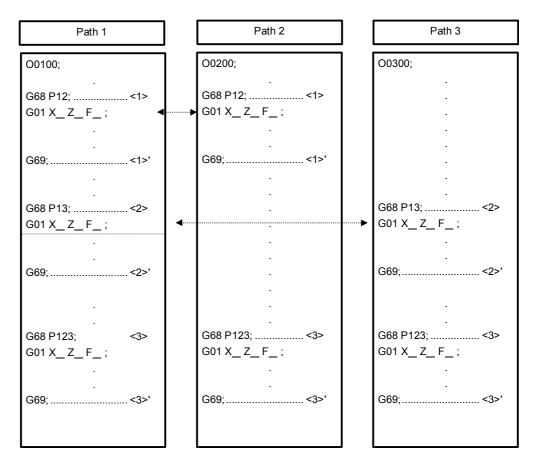
Programs O100, O200, and O300 for individual paths are executed as follows:



- <1> G68 P3; (balance cut for paths 1 and 2)
  Performs balance cutting for paths 1 and 2.
  Balance cutting is performed according to the cutting feed commands between <1> and <1>'.
- <2> G68 P5; (balance cut for paths 1 and 3)
  Performs balance cutting for paths 1 and 3.
  Balance cutting is performed according to the cutting feed commands between <2> and <2>'.
- <3> G68 P7; (balance cut for paths 1, 2, and 3)
  Performs balance cutting for paths 1, 2, and 3.
  Balance cutting is performed according to the cutting feed commands between <3> and <3>'.

#### - When the value specified at P is obtained using path numbers in combination

Programs O100, O200, and O300 for individual paths are executed as follows:



- <1> G68 P12; (balance cut for paths 1 and 2)
  Performs balance cutting for paths 1 and 2.
  Balance cutting is performed according to the cutting feed commands between <1> and <1>'.
- <2> G68 P13; (balance cut for paths 1 and 3)
  Performs balance cutting for paths 1 and 3.
  Balance cutting is performed according to the cutting feed commands in the section between <2> and <2>'.
- <3> G68 P123; (balance cut for paths 1, 2, and 3)
  Performs balance cutting for paths 1, 2, and 3.
  Balance cutting is performed according to the cutting feed commands in the section between <3> and <3>'.

#### **⚠** CAUTION

- 1 Balance cut only starts cutting feed on both tool posts at the same time; it does not maintain synchronization thereafter. To synchronize all the movements of both tool posts, the setting for both tool posts, such as the travel distance and feedrate, must be the same. Override and interlock can be applied independently to both tool posts. The settings for both tool posts that are related to override and interlock must also be the same to perform balance cutting.
- 2 After feed hold is applied during execution of balance cutting for both tool posts, balance cutting is not performed at the restart. Balance cutting is performed when the next move command is executed for both tool posts.

#### NOTE

- 1 Time delay before the pulse distribution of both tool posts is started is 2 msec or shorter.
- Overlap is invalid. In the balance cut mode, synchronization is established at the start of each move block in which cutting is specified, so movement can momentarily stop.
- 3 In the balance cut mode, continuous thread cutting overlap is also invalid. Perform continuous thread cutting in the balance cut cancel mode.
- 4 To establish synchronization at the start of pulse distribution in a block in which thread cutting is specified, the same PC must be selected.
- 5 The cancel mode (G69) is unconditionally set by a reset.
- 6 When the option "mirror image for double turret" is selected, the balance cut function cannot be used.

## **OPERATION**

1

## **DATA INPUT/OUTPUT**

By using the memory card interface on the left side of the display, information written in a memory card is read into the CNC and information is written from the CNC to a memory card.

The following types of data can be input and output:

- 1. Y-axis offset data
- $2.\ Tool\ offset\ /\ 2nd\ geometry\ data$

The above data can be input and output on the screens used for displaying and setting the data and on the ALL IO screen.

## 1.1 INPUT/OUTPUT ON EACH SCREEN

Data can be input to and output from the Y-axis offset and tool offset/second geometry operation screens.

## 1.1.1 Inputting and Outputting Y-axis Offset Data

### 1.1.1.1 Inputting Y-axis offset data

Y-axis offset data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. The Y-axis offset data that is registered in the memory and has a corresponding data number is replaced with data input by this operation.

#### Inputting Y-axis offset data

- 1 Make sure the input device is ready for reading.
- 2 Press the EDIT switch on the machine operator's panel.
- Press function key string, then the Y-axis offset data screen appears.
- 4 Press soft key [(OPRT)].
- 5 Press the rightmost soft key (continuous menu key).
- 6 Press soft key [READ].
- 7 Type the name of the file that you want to input.

  If the input file name is omitted, default input file name
  "TOOLOFST.TXTT" is assumed.
- Press soft key [EXEC].
  This starts reading the Y-axis offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
  To cancel the input of the program, press soft key [CANCEL].

## 1.1.1.2 Outputting Y-axis Offset Data

Y-axis offset data is output in a output format from the memory of the CNC to a memory card.

#### **Outputting Y-axis offset data**

- 1 Make sure the output in device is ready for outputting.
- 2 Press the EDIT switch on the machine operator's panel.
- Press function key , then the Y-axis offset data screen appears.
- 4 Press soft key [(OPRT)].
- 5 Press the rightmost soft key (continuous menu key).
- 6 Press soft key [PUNCH].
- 7 Type the file name that you want to output. If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
- 8 Press soft key [EXEC].
  This starts outputting the Y-axis offset data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press soft key [CANCEL].

#### 1.1.2 Inputting and Outputting Tool Offset / 2nd Geometry Data

#### 1.1.2.1 Inputting tool offset / 2nd geometry data

Tool offset / 2nd geometry data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. The tool offset / 2nd geometry data that is registered in the memory and has a corresponding data number is replaced with data input by this operation.

#### Inputting tool offset / 2nd geometry data

- 1 Make sure the input device is ready for reading.
- 2 Press the EDIT switch on the machine operator's panel.
- 3 screen appears.
- 4 Press soft key [(OPRT)].
- Press the rightmost soft key (continuous menu key). 5
- 6 Press soft key [READ].
- Type the name of the file that you want to input. If the input file name is omitted, default input file name "SEC\_GEOM.TXT" is assumed.
- Press soft key [EXEC]. 8 This starts reading the 2nd geometry data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears. To cancel the input of the program, press soft key [CANCEL].

#### 1.1.2.2 Outputting tool offset / 2nd geometry data

Tool offset / 2nd geometry data is output in a output format from the memory of the CNC to a memory card.

#### Outputting tool offset / 2nd geometry data

- 1 Make sure the output device is ready for outputting.
- 2 Press the EDIT switch on the machine operator's panel.
- Press function key ( ), then the offset / 2nd geometry data screen appears.
- 4 Press soft key [(OPRT)].
- 5 Press the rightmost soft key (continuous menu key).
- 6 Press soft key [PUNCH].
- 7 Type the file name that you want to output. If the file name is omitted, default file name "SEC\_GEOM.TXT" is assumed.
- 8 Press soft key [EXEC].
  This starts outputting the 2nd geometry data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press soft key [CANCEL].

## 1.2 INPUT/OUTPUT ON THE ALL IO SCREEN

Just by using the ALL IO screen, you can input and output Y-axis offset data and tool offset / 2nd geometry data.

The following explains how to display the ALL IO screen:

#### Displaying the ALL IO screen

#### **Procedure**

- 1 Press function key System .
- 2 Press the rightmost soft key (continuous menu key) several times.
- 3 Press soft key [ALL IO] to display the ALL IO screen.

The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

#### 1.2.1 Inputting and Outputting Y-axis Offset Data

With the lathe system, Y-axis offset data can be input and output using the ALL IO screen.

#### Inputting Y-axis offset data

#### **Procedure**

- On the ALL IO screen, press the rightmost soft key (continuous menu key) several times.
- Press soft key [Y OFFSET]. 2
- Select EDIT mode. 3
- 4 Press soft key [(OPRT)].
- Press soft key [N READ].
- Set the name of the file that you want to input. Type a file name, and press soft key [F NAME]. If the input file name is omitted, default file name "TOOLOFST.TXT" is assumed.
- Press soft key [EXEC]. This starts reading the Y-axis offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears. To cancel the input of the program, press soft key [CANCEL].

#### **Outputting Y-axis offset data**

- On the ALL IO screen, press the rightmost soft key (continuous menu key) several times.

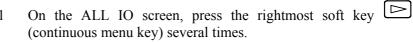
- Press soft key [Y OFFSET].
- 3 Select EDIT mode.
- Press soft key [(OPRT)].
- Press soft key [PUNCH].
  - Set the file name to be output. Type a file name, and press soft key [F NAME]. If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
- Press soft key [EXEC]. This starts outputting the Y-axis offset data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears. To cancel the output, press soft key [CANCEL].

#### 1.2.2 Inputting and Outputting Tool Offset / 2nd Geometry Data

With the lathe system, tool offset / 2nd geometry data can be input and output using the ALL IO screen.

#### Inputting tool offset / 2nd geometry data

#### **Procedure**



- Press soft key [GEOM.2]. 2
- 3 Select EDIT mode.
- Press soft key [(OPRT)].
- Press soft key [N READ].
- Set the name of the file that you want to input. Type a file name, and press soft key [F NAME]. If the input file name is omitted, default input file name "SEC GEOM.TXT" is assumed.
- 7 Press soft key [EXEC]. This starts reading the 2nd geometry data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears. To cancel the input of the program, press soft key [CANCEL].

#### Outputting tool offset / 2nd geometry data

#### **Procedure**

On the ALL IO screen, press the rightmost soft key (continuous menu key) several times.



- 2 Press soft key [GEOM.2].
- 3 Select EDIT mode.
- Press soft key [(OPRT)].
- Press soft key [PUNCH].
- Set the file name to be output. Type a file name, and press soft key [F NAME]. If the file name is omitted, default file name "SEC GEOM.TXT" is assumed.
- Press soft key [EXEC]. This starts outputting the 2nd geometry data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears. To cancel the output, press soft key [CANCEL].

# 2

## **SETTING AND DISPLAYING DATA**

## 2.1 SCREENS DISPLAYED BY FUNCTION KEY

sat tool compensation values

Press function key Gerset to display or set tool compensation values and other data.

This section describes how to display or set the following data:

- 1. Tool offset value
- 2. Workpiece coordinate system shift value
- 3. Tool offset / 2nd geometry data
- 4. Y-axis offset value
- 5. Chuck and tail stock barriers

## **2.1.1** Setting and Displaying the Tool Offset Value

Dedicated screens are provided for displaying and setting tool offset values and tool nose radius compensation values.

Setting and displaying the tool offset value and the tool nose radius compensation value

- Press function key FETTING .

  When using a multi-path system, select, in advance, a path for
  - when using a multi-path system, select, in advance, a path for which a tool offset value is to be set, by using the path selection switch.
- 2 Press chapter selection soft key [OFFSET] or press several times until the tool compensation screen is displayed.

  Different screens are displayed depending on whether tool geometry offset, wear offset, or neither is applied.

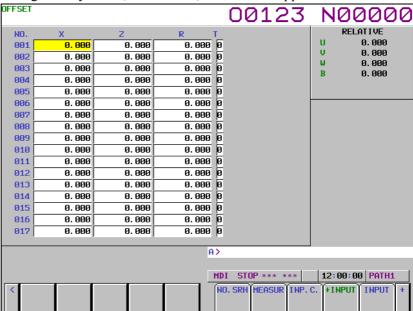


Fig. 2.1.1 (a) Without tool geometry/wear offset

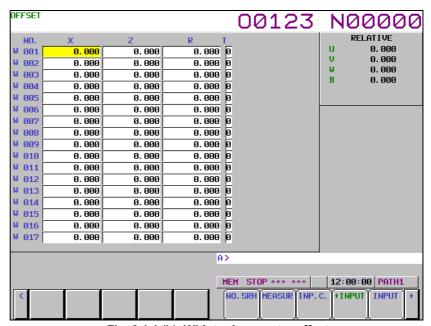


Fig. 2.1.1 (b) With tool geometry offset



Fig. 2.1.1 (c) With tool wear offset

- 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].
  - TIP is the number of the imaginary tool nose.

TIP may be specified on the geometry compensation screen or on the wear compensation screen.

#### **Explanation**

#### - Decimal point input

A decimal point can be used when entering a compensation value.

#### - Other method

An external input/output device can be used to input or output a cutter compensation value. See Chapter III-8 "Data Input/Output" in the User's Manual (Common to T/M Series).

Tool length compensation values can be set using the following functions described in subsequent subsections: direct input of tool offset value measured, direct input of tool offset value measured B, and counter input of offset value.

#### - Number of tool compensation values

16 groups are provided for tool compensation. The number of groups can be optionally extended to 32, 64, or 999. For the multi-path control, the above number of groups can be used for each path. Tool geometry compensation or wear compensation can be selected for each group.

#### - Disabling entry of compensation values

In some cases, tool wear compensation or tool geometry compensation values cannot be input because of the settings in bits 0 (WOF) and 1 (GOF) of parameter 3290. 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:

- When values are input for offset numbers, starting from one for which input is not inhibited to one for which input is inhibited, a warning is issued and values are set only for those offset numbers for which input is not inhibited.
- 2) When values are input for offset numbers, starting from one for which input is inhibited to one for which input is not inhibited, a warning is issued and no values are set.

#### - Displaying radius and TIP

The radius and TIP are not displayed if the tool nose radius compensation option is not displayed.

#### - Changing offset values during automatic operation

When offset values have been changed during automatic operation, bit 4 (LGT) and bit 6 (LWM) of parameter 5002 can be used for specifying whether new offset values become valid in the next move command or in the next T code command.

Fig. 2.1.1 (a)

	· ·9· =···· (+)			
LGT	LWM	When geometry compensation values and wear compensation values are separately specified	When geometry compensation values and wear compensation values are not separately specified	
0	0	Become valid in the next T code block	Become valid in the next T code block	
1	0	Become valid in the next T code block	Become valid in the next T code block	
0	1	Become valid in the next T code block	Become valid in the next move command	
1	1	Become valid in the next move command	Become valid in the next move command	

## **2.1.2** Direct Input of Tool Offset Value

To set the difference between the tool reference position used in programming (the nose of the standard tool, turret center, etc.) and the tool nose position of a tool actually used as an offset value.

#### Direct input of tool offset value

#### **Procedure**

- Setting of Z axis offset value

Cut surface A in manual mode with an actual tool.
Suppose that a workpiece coordinate system has been set.

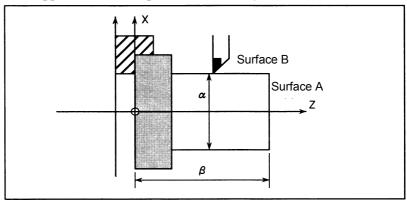


Fig. 2.1.2 (a)

- 2 Release the tool in X-axis direction only, without moving Z-axis and stop the spindle.
- 3 Measure distance  $\beta$  from the origin in the workpiece coordinate system to surface A.

Set this value as the measured value along the Z-axis for the desired offset number, using the following procedure:



Fig. 2.1.2 (b)

- 3-1 Press the function key setting or the soft key [OFFSET] to display the tool offset screen. If geometry compensation values and wear offset values are separately specified, display the screen for either of them.
- 3-2 Move the cursor to the set offset number using cursor keys.
- 3-3 Press the address key Z to be set.
- 3-4 Key in the measured value ( $\beta$ ).
- 3-5 Press the soft key [MESURE].

  The difference between measured value β and the coordinate is set as the offset value.

#### - Setting of X axis offset value

- 4 Cut surface B in manual mode.
- 5 Release the tool in the Z-axis direction without moving the X-axis and stop the spindle.
- 6 Measure the diameter  $\alpha$  of surface B. Set this value as the measured value along the X-axis for the desired offset number in the same way as when setting the value along the Z-axis.
- Repeat above procedure the same time as the number of the necessary tools.

The offset value is automatically calculated and set.

For example, in case  $\alpha$ =69.0 when the coordinate value of surface B in the diagram above is 70.0, set 69.0 [MEASURE] at offset No. 2. In this case, 1.0 is set as the X-axis offset value to offset No. 2.

#### **Explanation**

- Offset values for a program created in diameter programming

Enter diameter values for the offset values for axes for which diameter programming is used.

- Tool geometry offset value and tool wear offset value

If measured values are set on the tool geometry offset screen, all offset values become geometry offset values and all wear offset values are set to 0. If measured values are set on the tool wear offset screen, the differences between the measured offset values and the current wear offset values become the new offset values.

#### - Release of both axes

When the record button is provided on the machine side, the tool can be released in the directions of the two axes by setting bit 2 (PRC) of parameter No. 5005 or using the position record signal. For details on the position record signal, refer to the manual issued by the machine tool builder.

## **2.1.3** Input of Tool Offset Value Measured B

#### **Explanation**

#### Basic procedure to set tool offset value

To use the tool setter function for a one-turret/two-spindle lathe, first specify the spindle to be measured, using the S2TLS (G040.5) (spindle measurement select) signal.

- (1) Execute manual reference position return.
  - By executing manual reference position return, a machine coordinate system is stablished.
  - The tool offset value is computed on the machine coordinate system.
- (2) Select manual handle mode or manual continuous feed mode and set the tool compensation value write mode select signal GOQSM to "1". The LCD display is automatically changed to the tool offset screen (geometry), and the "OFST" indicator starts blinking in the status indication area at the bottom of the screen, which indicates that the tool compensation value writing mode is ready. When the tool setter function for a one–turret/two–spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes 1.

#### **⚠** CAUTION

After this, it is impossible to switch the S2TLS (spindle measurement selection) signal until the GOQSM (offset write mode) signal becomes 0.

- (3) Select a tool to be measured.
- (4) When the cursor does not coincide with the tool offset number desired to be set, move the cursor to the desired offset number using the page key and cursor key.
  - The cursor can also be coincided with the tool offset number desired to be set utomatically by the tool offset number input signals (when parameter QNI(No.5005#5)=1).
  - In this case, the position of the cursor cannot be changed on the tool compensation screen using page keys or cursor keys.
- (5) Near the tool to the sensor by manual operation.
- (6) Place the tool edge to a contacting surface of the sensor by manual handle feed.
  - Bring the tool edge in contact with the sensor. This causes the tool compensation value writing signals to input to be CNC.
  - The tool compensation value writing signal is set to "1", and the : The following tool compensation amount write signals are set up according to the setting of the TS1 parameter (bit 3 of No. 5004). When the parameter is 0: +MIT1, -MIT1, +MIT2, -MIT2

When the parameter is 1: +MIT1 only

i) The axis is interlocked in this direction and its feed is stopped.

- ii) The tool offset value extracted by the tool offset memory (tool geometry offset value) which corresponds to the offset number shown by the cursor is set up.
- (7) For both X-axis and Z-axis, their offset values are set by operations (5) and (6).
- (8) Repeat operations (3) to (7) for all necessary tools.
- (9) Set the tool compensation value writing mode signal GOQSM to "0".

The writing mode is canceled and the blinking "OFST" indicator light goes off.

When the tool setter function for a one-turret/two-spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal for the spindle being measured becomes 0.

#### Basic procedure to set workpiece coordinate shift value

To use the tool setter function for a one-turret/two-spindle lathe, first specify the spindle to be measured, using the S2TLS <G040.5> (spindle measurement select) signal.

- (1) Set the tool geometry offset values of each tool in advance.
- (2) Execute manual reference position return.

By executing manual reference position return, the machine coordinate system is established.

The workpiece coordinate system shift amount is computed based on the machine coordinate system of the tool.

(3) Set the workpiece coordinate system shift amount writing mode select signal WOOSM to "1".

The LCD display automatically switches to the workpiece shifting screen, the "WFST" indicator starts blinking at the status indicator area in the bottom of the screen, which indicates that the workpiece coordinate system shift amount writing mode is ready.

When the tool setter function for a one-turret/two-spindle lathe is in use, the workpiece coordinate system screen is selected, and the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes 1.

#### **⚠** CAUTION

After this, it is impossible to switch the S2TLS (spindle measurement selection) signal until the WOQSM (offset write mode) signal becomes 0.

- (4) Select a tool to be measured.
- (5) Check tool offset numbers.

The tool offset number corresponding to the tool required for measurement, shall be set in the parameter (No.5020) in advance. The tool offset number can also be set automatically by setting the tool offset number input signal (with parameter QNI(No.5005#5)=1).

(6) Manually approach the tool to an end face of the workpiece.

- (7) Place the tool edge to the end face (sensor) of the workpiece using manual handle feed.
  - When the tool edge contacts the end face of the workpiece, input the workpiece coordinate system shift amount signal WOSET.
  - The workpiece coordinate system shift amount on the Z-axis is automatically set.
- (8) Release the tool.
- (9) Set the workpiece coordinate system shift amount write mode select signal WOQSM to "0".

The writing mode is canceled and the blinking "WSFT" indicator light goes off.

When the tool setter function for a one-turret/two-spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes 0.

## **2.1.4** Counter Input of Offset value

By moving the tool until it reaches the desired reference position, the corresponding tool offset value can be set.

#### Counter input of offset value

#### **Procedure**

- 1 Manually move the reference tool to the reference position.
- 2 Reset the relative coordinates along the axes to 0.
- 3 Move the tool for which offset values are to be set to the reference position.
- 4 Select the tool offset screen. Move the cursor to the offset value to be set using cursor keys.

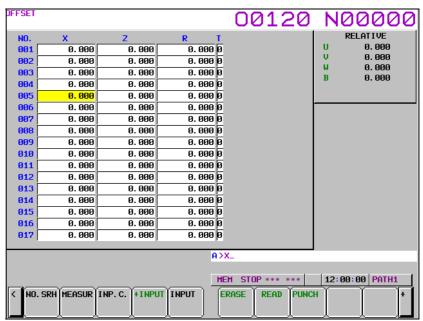


Fig. 2.1.4 (a) Tool offset screen

5 Press address key X (or Z) and the soft key [INP.C.].

#### **Explanation**

#### - Geometry offset and wear offset

When the above operations are performed on the tool geometry offset screen, tool geometry offset values are input and tool wear offset values do not change.

When the above operations are performed on the tool wear offset screen, tool wear offset values are input and tool geometry offset values do not change.

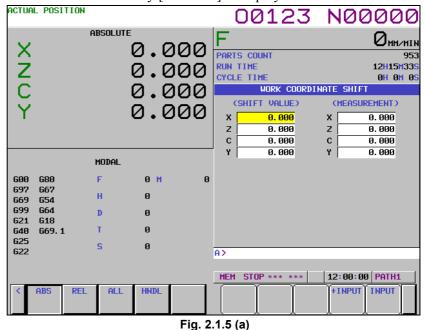
## 2.1.5 Setting the Workpiece Coordinate System Shift Value

The set coordinate system can be shifted when the coordinate system which has been set by a G50 command (or G92 command for G code system B or C) or automatic coordinate system setting is different from the workpiece coordinate system assumed at programming. When a T series system is used, the workpiece coordinate system shift screen is displayed.

#### Setting the workpiece coordinate system shifting amount

#### **Procedure**

- 1 Press function key OFFSET SETTING .
- 2 Press the continuous menu key several times until the screen with soft key [W.SHFT] is displayed.



Press soft key [W.SHFT].

3

- 4 Move the cursor using cursor keys to the axis along which the coordinate system is to be shifted.
- 5 Enter the shift value and press soft key [INPUT].

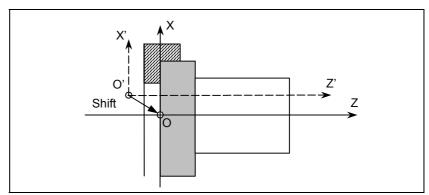


Fig. 2.1.5 (b)

#### **Explanation**

#### - When shift values become valid

Shift values become valid immediately after they are set.

#### - Shift values and coordinate system setting command

Setting a command (G50 or G92) for setting a coordinate system disables the set shift values.

Example)

When G50 X100.0 Z80.0; is specified, the coordinate system is set so that the current tool reference position is X+100.0, Z+80.0 regardless of the shift values.

#### - Shift values and coordinate system setting

If the automatic coordinate system setting is performed by manual reference position return after shift amount setting, the coordinate system is shifted instantly.

#### - Diameter or radius value

Whether the shift amount on the X-axis is diameter or radius value depends on that specified in program.

#### - Position record signal

When bit 2 (PRC) of parameter No. 5005 is 1, the absolute coordinates when the position record signal is ON are recorded for calculation of the shift amount.

#### **Example**

When the actual position of the reference point is X=121.0 (diameter), Z=69.0 with respect to the workpiece origin but it should be X=120.0, Z=70.0, set the following shift values:

X=1.0, Z=-1.0

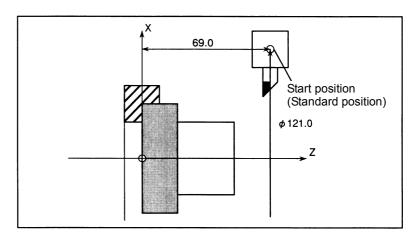


Fig. 2.1.5 (c)

## **2.1.6** Setting Tool Compensation/Second Geometry Offset Values

To compensate for a difference in tool attachment position or selection position, second geometry offset can be applied in addition to tool offset. As second geometry offset values, X-axis, Y-axis, and Z-axis offset values can be set.

#### Procedure for displaying and setting second geometry offset values

#### **Procedure**

- 1 Press function key OFFSET SETTING.
- 2 Press the continuous menu key several times until the screen with soft key [GEOM.2] is displayed.
- Press the soft key [GEOM.2]. The tool compensation/second geometry offset screen is displayed. When displaying data other than the data of the second geometry offset number currently displayed, press the page key or to

display the screen for a desired second geometry offset number. 00123 N00000 ABSOLUTE OHM/HIN 0.000 XZC PARTS COUNT RUN TIME 12H15M33 0.000 CYCLE TIME OH OM 09 0.000 0.000 0.000 01 0.000 02 0.000 0.000 G 03 0.000 0.000 0.000 0. 000 G 05 0.000 0.000 MODAL 0.000 0.000 06 G G50 0 M 07 0.000 0.000 G97 G80 **650.2** Й 08 0.000 0.000 Н G 690 G98 G13. 1 Ø9 0.000 0. 000 G G69 G67 G50.1 0.000 G95 G54 G 10 0.000 G21 G64 Й G 11 0. 000 0. 000 G40 G18 G69. 1 A> 12:00:00 PATH1 MEM STOP \*\*\* \*\*\* REL ALL HNDL +INPUT INPUT

Fig. 2.1.6 (a)

3-1 If one screen cannot fully display the second geometry offset values of the Y-axis, press the [SWITCH] soft key to switch the screen display.

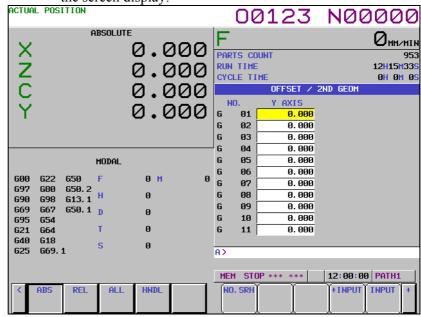


Fig. 2.1.6 (b)

- 4 By using one of the methods described below, move the cursor to the second geometry offset value of a desired second geometry offset number.
  - By using the page keys and cursor keys, move the cursor to the desired offset value position.
  - Key in the desired offset number then press the [NO.SRH] soft key.
- Input a desired second geometry offset value then press the key or the [INPUT] soft key. The input second geometry offset value is set. When an additional offset to the second geometry offset value currently displayed is input, press the [+INPUT] soft key. In this case, the result of addition is set as the second geometry offset value.
- 6 Repeat steps 4 and 5 to set all parameters to be modified.

#### **Explanation**

- Input of a second geometry offset value

The value input in the key input buffer is set as the second geometry offset value where the cursor is currently placed.

- +Input of a second geometry offset value

The value input in the key input buffer is added to the second geometry offset value where the cursor is currently placed.

- Number search for a second geometry offset value

The value input in the key input buffer is used as a second geometry offset number to move the cursor to the corresponding position.

#### Limitation

- Setting of a second geometry offset value

The setting of a second geometry offset value is valid only when bit 5 (GF2) of parameter No. 3290 is set to 1.

## 2.1.7 Setting the Y-Axis Offset

Tool position offset values along the Y-axis can be set. Counter input of offset values is also possible.

For the Y-axis, no tool offset value can be directly input.

#### Procedure for setting the tool offset value of the Y axis

- 1 Press function key OFFSET SETTING
- 2 Press the continuous menu key several times until the screen with soft key [OFST.2] is displayed.
- 3 Press soft key [OFST.2]. The Y-axis offset screen is displayed.



Fig. 2.1.7 (a)

3-1 When you press the [CHANGE] soft key, Y-axis tool geometry compensation data is displayed. When you press the [CHANGE] soft key again, the screen display switches to the display of tool wear compensation data.



Fig. 2.1.7 (b)

- 4 Position the cursor at the offset number to be changed by using either of the following methods:
  - Move the cursor to the offset number to be changed using page keys and cursor keys.
  - Type the offset number and press soft key [NO.SRH].
- 5 Enter the offset value.
- 6 Press soft key [INPUT]. The offset value is set and displayed.



Fig. 2.1.7 (c)

# Procedure for counter input of the offset value

#### **Procedure**

To set relative coordinates along the Y-axis as offset values:

- 1 Move the reference tool to the reference point.
- 2 Reset relative coordinate Y to 0.
- 3 Move the tool for which offset values are to be set to the reference point.
- 4 Move the cursor to the value for the offset number to be set, press Y, then press soft key [INP.C.].
  Relative coordinate Y (or V) is now set as the offset value.

# **2.1.8** Chuck and Tail Stock Barriers

The chuck and tail stock barrier function prevents damage to the machine by checking whether the tool nose fouls either the chuck or tail stock. Specify an area into which the tool may not enter (entry-inhibition area). This is done using the special setting screen, according to the shapes of the chuck and tail stock. If the tool nose should enter the set area during a machining operation, this function stops the tool and outputs an alarm message. The tool can be cleared from the area only by retracting it in the direction opposite to that in which the tool entered the area.

# Setting the chuck and tail stock barriers

#### **Procedure**

- Setting the shapes of the chuck and tail stock
  - Press the Setting function key.
  - 2 Press the continuous menu key. Then, press the [CHUCK TAIL] chapter selection soft key.

Pressing the page key toggles the display between the chuck barrier setting screen and tail stock barrier setting screen.

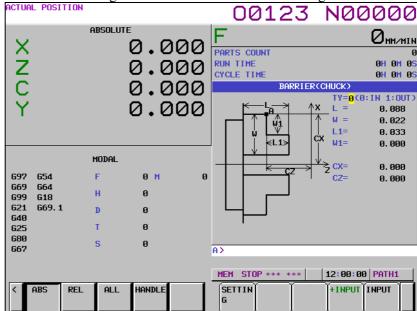


Fig. 2.1.8 (a) Chuck barrier setting screen

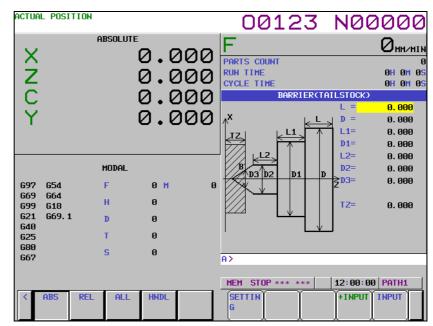


Fig. 2.1.8 (b) Tail stock barrier setting screen

Position the cursor to each item defining the shape of the chuck or tail stock, enter the corresponding value, then press the [INPUT] soft key. The value is set. Pressing the [+INPUT] soft key after a value has been entered adds the entered value to the current value, the new setting being the sum of the two values. Items CX and CZ, both on the chuck barrier setting screen, and item TZ on the tail stock barrier setting screen can also be set in another way. Manually move the tool to the desired position, then press the [SETTING] soft key to set the coordinate(s) of the tool in the workpiece coordinate system. If a tool having an offset other than 0 is manually moved to the desired position with no compensation applied, compensate for the tool offset in the set coordinate system. Items other than CX, CZ, and TZ cannot be set by using the [SETTING] soft key. Example

When an alarm is issued, the tool stops before the entry-inhibition area if bit 7 (BFA) of parameter No. 1300 is set to 1. If bit 7 (BFA) of parameter No. 1300 is set to 0, the tool stops at a more inside position than the specified figure because the CNC and machine system stop with some delay in time.

For safety, therefore, set an area a little larger than the determined area. The distance between the boundaries of these two areas, L, is calculated from the following equation, based on the rapid traverse rate.

 $L = (Rapid traverse rate) \times 7500$ 

When the rapid traverse rate is 15 m/min, for example, set an area having a boundary 2 mm outside that of the determined area.

The shapes of the chuck and tail stock can be set using parameters Nos. 1330 to 1348

#### **NOTE**

Set G23 mode before attempting to specify the shapes of the chuck and tail stock.

# - Reference position return

Return the tool to the reference position along the X- and Z-axes. The chuck-tail stock barrier function becomes effective only once reference position return has been completed after power on.

When an absolute position detector is provided, reference position return need not always be performed. The positional relationship between the machine and the absolute position detector, however, must be determined.

#### - G22/G23

When G22 (stored stroke limit on) is specified, the chuck and tail stock area becomes an entry-inhibition area. When G23 (stored stroke limit off) is specified, the entry-inhibition area is canceled. Even if G22 is specified, the entry-inhibition area for the tail stock can be disabled by issuing a tail stock barrier signal. When the tail stock is pushed up against a workpiece or separated from the workpiece by using the auxiliary functions, PMC signals are used to enable or disable the tail stock setting area.

Table 2.1.8 (a)

		1110 (41)		
G code	Tail stock barrier signal	Chuck barrier	Tail stock barrier	
G22	0	Valid	Valid	
GZZ	1	Invalid	Disable	
G23	Unrelated	Invalid	Invalid	

G22 is selected when the power is turned on. Using G23, bit 7 of parameter No. 3402, however, it can be changed to G23.

# **Explanation**

- Setting the shape of the chuck barrier

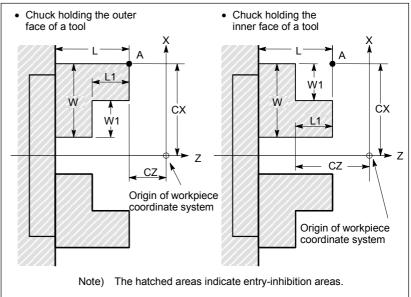


Fig. 2.1.8 (c)

Table2.1.8 (b)

Symbol	Description
TY	Chuck-shape selection (0: Holding the inner face of a tool,
	1: Holding the outer face of a tool)
CX	Chuck position (along X-axis)
CZ	Chuck position (along Z-axis)
L	Length of chuck jaws
W	Depth of chuck jaws (radius)
L1	Holding length of chuck jaws
W1	Holding depth of chuck jaws (radius)

TY: Selects a chuck type, based on its shape. Specifying 0 selects a chuck that holds the inner face of a tool. Chuck that holds the outer face of a tool. A chuck is assumed to be symmetrical about its Z-axis.

#### CX, CZ:

Specify the coordinates of a chuck position, point A, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. The unit of data is indicated in Table 2.1.7(c).

# **↑** CAUTION

Whether diameter programming or radius programming is used for the axis determines the programming system. When diameter programming is used for the axis, use diameter programming to enter data for the axis.

Table 2.1.8 (c)

Increment	Unit o	f data	Valid data rango		
system	IS-A	IS-B	Valid data range		
Metric input	0.001 mm	0.0001 mm	-99999999 to +99999999		
Inch input	0.0001 inch	0.00001 inch	-99999999 to +99999999		

L, L1, W, W1: Define the figure of a chuck. The unit of data is indicated in Table 2.1.7(c).

# **!** CAUTION

Always specify W and W1 in radius. When radius programming is used for the Z-axis, specify L and L1 in radius.

# - Setting the shape of a tail stock barrier

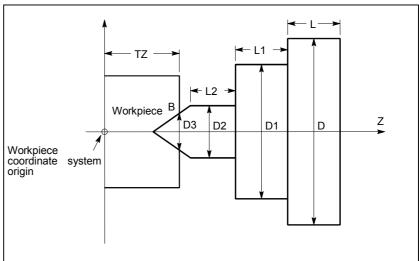


Table 2.1.8 (d)

Description		
Tail stock position (along the Z-axis)		
Tail stock length		
Tail stock diameter		
Tail stock length (1)		
Tail stock diameter (1)		
Tail stock length (2)		
Tail stock diameter (2)		
Tail stock hole diameter (3)		

TZ: Specifies the Z coordinate of the chuck position, point B, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. The unit of data is indicated in Table 2.1.7(c). A tail stock is assumed to be symmetrical about its Z-axis.

### **⚠** CAUTION

Whether diameter programming or radius programming is used for the Z-axis determines the programming system.

#### L, L1, L2, D, D1, D2, D3:

Define the figure of a tail stock. The valid data range is indicated in Table 2.1.7(c).

# **!** CAUTION

Always specify D, D1, D2, and D3 in diameter programming. When radius programming is used for the Z-axis, specify L, L1, and L2 in radius.

# - Setting the entry-inhibition area for the tail stock tip

The tip angle of the tail stock is 60 degrees. The entry-inhibition area is set around the tip, assuming the angle to be 90 degrees, as shown below.

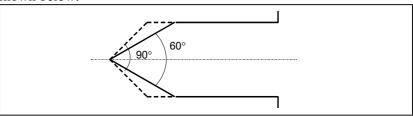


Fig. 2.1.8 (d)

#### Limitation

### - Correct setting of an entry-inhibition area

If an entry-inhibition area is incorrectly set, it may not be possible to make the area effective. Avoid making the following settings:

- $L \le L1$  or  $W \le W1$  in the chuck-shape settings.
- $D2 \le D3$  in the tail stock-shape settings.
- A chuck setting overlapping that of the tail stock.

#### - Retraction from the entry-inhibition area

If the tool enters the entry-inhibition area and an alarm is issued, switch to manual mode, retract the tool manually, then reset the system to release the alarm. In manual mode, the tool can be moved only in the opposite direction to that in which the tool entered the area

The tool cannot be moved in the same direction (further into the area) as it was travelling when the tool entered the area.

When the entry-inhibition areas for the chuck and tail stock are enabled, and the tool is already positioned within those areas, an alarm is issued when the tool moves.

When the tool cannot be retracted, change the setting of the entry-inhibition areas, such that the tool is outside the areas, reset the system to release the alarm, then retract the tool. Finally, reinstall the original settings.

### - Coordinate system

An entry-inhibition area is defined using the workpiece coordinate system. Note the following.

<1> When the workpiece coordinate system is shifted by means of a command or operation, the entry-inhibition area is also shifted by the same amount.

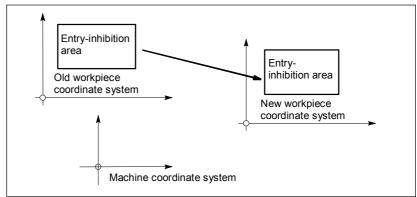


Fig. 2.1.8 (e)

Use of the following commands and operations will shift the workpiece coordinate system.

Commands:

G54 to G59, G52, G50 (G92 in G code system B or C) Operations:

Manual handle interruption, change in offset relative to the workpiece origin, change in tool offset (tool geometry offset), operation with machine lock, manual operation with machine absolute signal off

<2> When the tool enters an entry-inhibition area during automatic operation, set the manual absolute signal, \*ABSM, to 0 (on), then manually retract the tool from the area. If this signal is 1, the distance the tool moves in manual operation is not counted in the tool coordinates in the workpiece coordinate system. This results in the state where the tool can never be retracted from the entry-inhibition area.

#### - Stored stroke check 2/3

When both stored stroke check 2/3 and the chuck tail stock barrier function are provided, the barrier takes priority over the stored stroke check. Stored stroke check 2/3 is ignored.

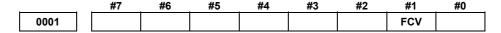
# **APPENDIX**



# **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.

# A.1 DESCRIPTION OF PARAMETERS



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

**# 1 FCV** Program format

0: Series 16 standard format

1: Series 15 format

#### **NOTE**

- 1 Programs created in the Series 15 program format can be used for operation on the following functions:
  - 1 Subprogram call M98
  - 2 Equal leads threading G32 (T series)
  - 3 Canned cycle G90, G92, G94 (T series)
  - 4 Multiple repetitive canned cycle G71 to G76 (T series)
  - 5 Canned cycle for drillingG83.1, G80 to G89 (T series)G73, G74, G76, G80 to G89(M series)
- When the program format used in the Series 15 is used for this CNC, some limits may add. Refer to the Part II.

#### Setting of each axis in the basic coordinate system

[Input type]
[Data type]

Parameter input

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 three-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 programming of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1290

Distance between two opposite tool posts in mirror image for double turret

**.** 

[Input type] Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Minimum unit of data]

[Valid data range]

Depend on the increment system of the reference axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B) )

(When the increment system is IS-B, 0.0 to +999999.999)

Set the distance between two opposite tool posts in mirror image for double turret.

#7	#6	#5	#4	#3	#2	#1	#0
BFA							

[Input type] [Data type]

Setting input Bit path

# 7 BFA

When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the interference check function between paths (T series), or a chuck tail stock barrier (T series) alarm is issued:

0: The tool stops after entering the prohibited area.

1: The tool stops before the prohibited area.

# Profile of a chuck

[Input type]
[Data type]

Parameter input

Data type] Byte path ata range] 0 to 1

[Valid data range]

Select a chuck figure.

0: Chuck which holds a workpiece on the inner surface1: Chuck which holds a workpiece on the outer surface

1331

#### Dimensions of the claw of a chuck (L)

[Input type] [Data type]

Parameter input

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range] Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the length (L) of the claw of the chuck.

#### **NOTE**

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter programming or radius programming.

#### Dimensions of the claw of a chuck (W)

[Input type]

Parameter input

[Data type] R

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the width (W) of the claw of the chuck.

#### NOTE

Specify this parameter by using a radius value at all times.

1333

Dimensions of the part of a claw at which a workpiece is held (L1)

Parameter input

[Input type] [Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the length (L1) of the claw of the chuck.

#### NOTE

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter programming or radius programming.

1334

Dimensions of the part of a claw at which a workpiece is held (W1)

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the width (W1) of the claw of the chuck.

#### NOTE

Specify this parameter by using a radius value at all times.

#### X coordinate of a chuck (CX)

[Input type]
[Data type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999) Set the chuck position (X coordinate) in the workpiece coordinate system.

# NOTE

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter programming or radius programming.

1336

#### Z coordinate of a chuck (CZ)

[Input type]
[Data type]

Parameter input

Real path

[Unit of data] mm, inch (input unit)

[Minimum unit of data]
[Valid data range]

Depend on the increment system of the applied axis

9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999) Set the chuck position (Z coordinate) in the workpiece coordinate system.

## NOTE

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter programming or radius programming.

#### Length of a tail stock (L)

[Input type]

Parameter input

[Data type] Real path

[Unit of data] mm, inch (input unit)

[Minimum unit of data]

Depend on the increment system of the applied axis

[Valid data range]

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the length (L) of the tail stock.

#### **NOTE**

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter programming or radius programming.

1342

#### Diameter of a tail stock (D)

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the diameter (D) of the tail stock.

#### NOTE

Specify this parameter by using a diameter value at all times.

1343

#### Length of a tail stock (L1)

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the length (L1) of the tail stock.

#### NOTE

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter programming or radius programming.

1344

#### Diameter of a tail stock (D1)

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B) )

(When the increment system is IS-B, 0.0 to +999999.999)

Set the diameter (D1) of the tail stock.

#### NOTE

Specify this parameter by using a diameter value at all times.

1345

#### Length of a tail stock (L2)

[Input type]
[Data type]
[Unit of data]

Parameter input

Real path mm, inch (input unit)

[Minimum unit of data] [Valid data range] Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the length (L2) of the tail stock.

#### **NOTE**

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter programming or radius programming.

#### Diameter of a tail stock (D2)

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the diameter (D2) of the tail stock.

#### NOTE

Specify this parameter by using a diameter value at all times.

1347

#### Diameter of the hole of a tail stock (D3)

Parameter input

[Input type] [Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

Set the diameter (D3) of the tail stock.

## NOTE

Specify this parameter by using a diameter value at all times.

1348

#### Z coordinate of a tail stock (TZ)

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data]

Depend on the increment system of the applied axis

[Valid data range]

9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999) Set the tail stock position (Z coordinate) in the workpiece coordinate system.

### **NOTE**

Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

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

[Input type]

Parameter input

[Data type] Bit path

# #1 LRP Positioning (G00)

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

When using three-dimensional coordinate 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
				ROC				
1403								

1403

[Input type]

Parameter input

[Data type] Bit path

**#4 ROC** In the threading cycles G92 and G76, rapid traverse override for retraction after threading is finished is:

0: Effective

1: Not effective (Override of 100%)

1411

Cutting feedrate

#### **NOTE**

When this parameter is set, the power must be turned off before operation is continued.

[Input type] [Data type]

Setting input

Real path

[Unit of data]
[Minimum unit of data]

[Valid data range]

mm/min, inch/min, degree/min (input unit)

Depend on the increment system of the reference axis Refer to the standard parameter setting table (C)

Kelei to the standard parameter setting table (C)

(When the increment system is IS-B, 0.0 to +240000.0)

When the machine requires little change in cutting feedrate during cutting, a cutting feedrate can be specified in the parameter. This eliminates the need to specify a cutting feedrate (F command) in the NC program.

#### Rapid traverse rate for each axis

[Input type] Par [Data type] Re

Parameter input Real axis

[Unit of data]

mm/min, inch/min, degree/min (machine unit)

[Minimum unit of data] [Valid data range] Depend on the increment system of the applied axis

Refer to the standard parameter setting table (C)

(When the increment system is IS-B, 0.0 to +240000.0)

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1466

#### Feedrate at retraction operation in threading cycle G92 or G76

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm/min, inch/min (machine unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

Refer to the standard parameter setting table (C)

(When the increment system is IS-B, 0.0 to +240000.0)

In the threading cycle G92 or G76, a retraction operation is performed after threading. Set a feedrate for this retraction operation.

#### NOTE

When bit 1 (CFR) of parameter No. 1611 is set to 1, or 0 is set in this parameter, the rapid traverse rate set in parameter No. 1420 is used.

1611
------

#7	#6	#5	#4	#3	#2	#1	#0
							CFR

[Input type]

Parameter input

[Data type]

Bit path

# 0 CFR

For retraction after threading in the threading cycles G92 and G76:

- 0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL feedrate (parameter No. 1627).
- 1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and FL feedrate.

#### NOTE

When this parameter is set to 1, an in-position check is made before retraction. For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse before look-ahead interpolation and optimum torque acceleration/deceleration are disabled.

1626

Acceleration/deceleration time constant in threading cycles for each axis

[Input type]

Parameter input

[Data type]

Word axis

[Unit of data]

msec

[Valid data range]

0 to 4000

Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627

FL rate for acceleration/deceleration in threading cycles for each axis

[Input type]

Parameter input

[Data type]

Real axis

[Unit of data]

mm/min, inch/min, degree/min (machine unit)

[Minimum unit of data]

Depend on the increment system of the applied axis

[Valid data range]

Refer to the standard parameter setting table (C)

(When the increment system is IS-B, 0.0 to +240000.0)

Set an FL feedrate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

3032

#### Allowable number of digits for the T code

[Input type] [Data type] Parameter input

Byte path

[Valid data range]

1 to 8

Set the allowable numbers of digits for the T code.

When 0 is set, the allowable number of digits is assumed to be 8.

	#7	#6	#5	#4	#3	#2	#1	#0
3115			APLx					

[Input type]

Parameter input

[Data type]

Bit axis

# 5 APLx

When the active offset value modification mode based on manual feed is selected, the relative position display is automatically:

- 0: Not preset.
- 1: Preset.

Use this parameter when returning a modified offset value to the original value before modification in the active offset value modification mode based on manual feed. The offset value can be returned to the original value by making a movement on the axis by manual feed so that the relative position display (counter) indicates the position 0.

_	
ı	
ı	3290
ı	

#7	#6	#5	#4	#3	#2	#1	#0
		GO2				GOF	WOF

[Input type]

Parameter input

[Data type]

Bit path

# 0 WOF

Setting the tool offset value (tool wear offset) by MDI key input is:

- 0: Not disabled
- 1: Disabled (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)

# **NOTE**

When tool offset memory A is selected with the M series, the tool offset set in the parameter WOF is followed even if geometry offset and wear offset are not specified with the T series.

- **#1 GOF** Setting the tool geometry offset value by MDI key input is:
  - 0: Not disabled
  - 1: Disabled (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)
- #5 GO2 Setting the second geometry tool offset value by MDI key input is:
  - 0: Disabled.
  - 1: Not disabled.

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]
[Data type]
[Valid data range]

Parameter input Word path

0 to 999

When the modification of tool offset values by MDI key input is to be disabled using bit 0 (WOF) of parameter No.3290 and bit 1 (GOF) of parameter No.3290, parameter Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No.3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No.3295, set the number of such values.

In the following cases, however, none of the tool offset values may be modified:

- When 0 or a negative value is set in parameter No. 3294
- When 0 or a negative value is set in parameter No. 3295
- When a value greater than the maximum tool offset number is set in parameter No. 3294

In the following case, a modification to the values ranging from the value set in parameter No. 3294 to the maximum tool offset number is disabled:

When the value of parameter No. 3294 added to the value of parameter No. 3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI panel, the warning "WRITE PROTECT" is issued.

#### [Example]

When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:

- Bit 1 (GOF) of parameter No. 3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No. 3290 = 1 (to disable tool wear offset value modification)
- Parameter No. 3294 = 51
- Parameter No. 3295 = 10

If the setting of bit 0 (WOF) of parameter No. 3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

	_	#7	#6	#5	#4	#3	#2	#1	#0
3401		GSC	GSB						DPI

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

- # 0 DPI When a decimal point is omitted in an address that can include a decimal point
  - The least input increment is assumed. (Decimal point programming)
  - 1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point programming)
- # 6 **GSB** The G code system is set.
- #7 **GSC**

GSC	GSB	G code		
0	0	G code system A		
0	1 G code system E			
1	0	G code system C		

#### NOTE

G code system B and G code system C are optional functions. When no option is selected, G code system A is used, regardless of the setting of these parameters.

	#/	#6
	G23	CLR
3402		

#3 #0 G91 G01

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

- # 0 **G01** G01 Mode entered when the power is turned on or when the control is cleared
  - G00 mode (positioning) 0:
  - G01 mode (linear interpolation)
- #3 G91 When the power is turned on or when the control is cleared
  - G90 mode (absolute programming)
  - G91 mode (incremental programming)
- Reset button on the MDI panel, external reset signal, reset and rewind # 6 **CLR** signal, and emergency stop signal
  - 0: Cause reset state.
  - Cause clear state.

For the reset and clear states, refer to Appendix in the User's Manual.

# 7 G23 When the power is turned on

0: G22 mode (stored stroke check on)

1: G23 mode (stored stroke check off)

	. 1	<b>#</b> 7	#6	#5	#4	#3	#2	#1	#0
3405				DDP	CCR	G36			
3403									

[Input type] Par

Parameter input

[Data type] Bit path

#3 G36 For a G code used with the automatic tool offset function:

0: G36/G37 is used.

1: G37.1/G37.2 is used.

#### NOTE

If it is necessary to perform circular threading (counterclockwise), set this parameter to 1.

# 4 CCR Addresses used for chamfering

0: Address is "I", "J", or "K".

In direct drawing dimension programming, addresses ",C", ",R", and ",A" (with comma) are used in stead of "C", "R", and "A".

1: Address is "C".

Addresses used for direct drawing dimension programming are "C", "R", and "A" without comma.

#### **NOTE**

If this bit (CCR) is set to 0, the function for changing the compensation direction by specifying I, J, or K in a G01 block in the cutter or tool nose radius compensation mode cannot be used. If this bit (CCR) is set to 1 when address C is used as an axis name, the chamfer function cannot be used.

# 5 DDP Angle commands by direct drawing dimension programming

0: Normal specification

1: A supplementary angle is given.

	_	#7	#6	#5	#4	#3	#2	#1	#0
3453									CRD

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

#0 CRD If the functions of chamfering or corner R and direct drawing dimension programming are both enabled,

0: Chamfering or corner R is enabled.

1: Direct drawing dimension programming is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO						
5001								

[Input type] Parameter input

[Data type] Bit path

5002

#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
WNP	LWM	LGC	LGT		LWT	LGN	

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

#1 LGN Geometry offset number of tool offset

- 0: Is the same as wear offset number
- 1: Specifies the geometry offset number by the tool selection number

# NOTE

This parameter is valid when the option for tool geometry offset or tool wear offset is specified.

- **#2 LWT** Tool wear offset is performed by:
  - 0: Moving the tool.
  - 1: Shifting the coordinate system.

#### NOTE

This parameter is valid when the option for tool geometry offset or tool wear offset is specified.

#4 LGT Tool geometry offset

0: Compensated by the shift of the coordinate system

1: Compensated by the tool movement

#### **NOTE**

This parameter is valid when the option for tool geometry offset or tool wear offset is specified.

#5 LGC When tool geometry offset is based on a coordinate shift, the specification of offset number 0:

0: Does not cancel tool geometry offset.

1: Cancels tool geometry offset.

#### **NOTE**

This parameter is valid when the option for tool geometry offset or tool wear offset is specified.

# 6 LWM Tool offset operation based on tool movement is performed:

0: In a block where a T code is specified.

1: Together with a command for movement along an axis.

#7 WNP Imaginary tool nose number used for tool nose radius compensation, when the geometry/wear offset function is equipped, is the number specified by:

0: Geometry offset number

1: Wear offset number

	 #7	#6	#5	#4	#3	#2	#1	#0
5003	TGC						SUV	SUP
5003								

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

# 0 SUP

**SUV** These bits are used to specify the type of startup/cancellation of cutter or tool nose radius compensation.

CSC	CSU	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.
			G41 / Programmed path
0	1	Туре В	A compensation vector perpendicular to the startup block or cancellation block and an
			Intersection vector are output.  Intersection point
			N2 Programmed path
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.  Intersection point  Shift
			When the block specifies movement operation, the type is set according to the SUB setting; if
			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 Series 16i-T is performed.

- #7 TGC A tool geometry offset based on a coordinate shift is:
  - 0: Not canceled by reset.
  - 1: Canceled by reset.

#### NOTE

This parameter is valid when the option for tool geometry offset or tool wear offset is specified.

5004

#7	#6	#5	#4	#3	#2	#1	#0
				TSI			

[Input type]

Parameter input

[Data type]

Bit path

- **TSI** For touch sensor contact detection with the function for direct input of offset value measured B:
  - 0: Four-contact input is used.
  - 1: One-contact input is used.

5005
------

#/	#6	#5	#4	#3	#2	#1	#0
		QNI			PRC		

[Input type]

Parameter input

[Data type]

Bit path

- #2 PRC For direct input of a tool offset value or workpiece coordinate system shift amount:
  - 0: The PRC signal is not used.
  - 1: The PRC signal is used.
- **QNI** With the tool length measurement function, a tool compensation number is selected by:
  - 0: Operation through the MDI panel by the operator (selection based on cursor operation).
  - 1: Signal input from the PMC.

5006

	#7	#6	#5	#4	#3	#2	#1	#0	
					LVC				

[Input type]

Parameter input

[Data type]

Bit

- #3 LVC A tool offset (geometry/wear) based on a tool movement and wear offset based on a coordinate shift are:
  - 0: Not canceled by reset.
  - 1: Canceled by reset.

	#7	#6	#5	#4	#3	#2	#1	#0
5008					CNV		CNC	

[Input type]

Parameter input

[Data type]

Bit path

# 1 CNC

# 3 CNV

These bits are used to select an interference check method in the cutter or tool nose radius compensation mode.

CNV	CNC	Operation
0	0	Interference check is enabled. The direction and the angle of an arc are checked.
0	1	Interference check is enabled. Only the angle of an arc is checked.
1	-	Interference check is disabled.

For the operation taken when the interference check shows the occurrence of an interference (overcutting), see the description of bit 5 (CAV) of parameter No. 19607.

#### **NOTE**

Checking of only the direction cannot be set.

5010

Limit for ignoring the small movement resulting from cutter or tool nose radius

[Input type] [Data type]

Setting input Real path

[Unit of data]

mm, inch (input unit)

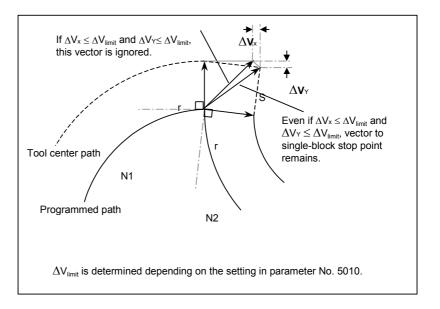
[Minimum unit of data]
[Valid data range]

Depend on the increment system of the reference axis

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 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.



Tool offset number used with the function for direct input of offset value measured B

[Input type]
[Data type]
[Valid data range]

Parameter input

Word path

0 to number of tool compensation values

Set a tool offset number used with the function for direct input of offset value measured B (when a workpiece coordinate system shift amount is set). (Set the tool offset number corresponding to a tool under measurement beforehand.) This parameter is valid when automatic tool offset number selection is not performed (when bit 5 (QNI) of parameter No. 5005 is set to 0).

Number of tool compensation values

#### NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type]
[Data type]
[Valid data range]

Parameter input Word path 0 to 999

Set the maximum allowable number of tool compensation values used for each path.

Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system. The number of compensation values usable in the entire system depends on the option configuration.

If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.

Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the number of compensation values usable for each path are specified, an alarm is issued.

For example, 64 tool compensation sets are used, 20 sets may be allocated to path 1, 30 sets to path 2, and 14 sets to path 3. All of 64 sets need not be used.

#### Number of digits of an offset number used with a T code command

[Input type]
[Data type]
[Valid data range]

Parameter input

Byte path

[Valid data range] 0 to 3

Specify the number of digits of a T code portion that is used for a tool offset number (wear offset number when the tool geometry/wear offset function is used).

When 0 is set, the number of digits is determined by the number of tool compensation values.

When the number of tool compensation values is 1 to 9: Lower 1 digit When the number of tool compensation values is 10 to 99: Lower 2 digits

When the number of tool compensation values is 100 to 999: Lower 3 digits

#### Example:

When an offset number is specified using the lower 2 digits of a T code, set 2 in parameter No. 5028.

Txxxxxx yy

xxxxxx : Tool selection yy : Tool offset number

#### NOTE

A value longer than the setting of parameter No. 3032 (allowable number of digits of a T code) cannot be set.

5040

#7	#6	#5	#4	#3	#2	#1	#0
			TLG	TCT			

[Input type]

Parameter input

[Data type] Bit path

#### # 3 TCT

The tool change method is based on:

- 0: Turret rotation. (Tool change operation is performed with a T command only.)
  - With a T command, an auxiliary function and tool offset operation are performed.
- 1: Automatic tool changer (ATC). (Tool change operation is performed with an M command (such as M06)).

With a T command, an auxiliary function only is performed.

This parameter is valid with the T series only.

#4 TLG When tool change operation is performed with the automatic tool changer (when bit 3 (TCT) of parameter No. 5040 is set to 1), tool offset operation is specified by:

0: G43.7.

At this time, G43 and G44 function as G codes for tool length compensation.

1: G43.

At this time, G43.7 and G44.7 function as G codes for tool length compensation.

_	
	5041

#7	#6	#5	#4	#3	#2	#1	#0
	AON					ATP	

[Input type]

Parameter input

[Data type] B

Bit path

### # 1 ATP

0: By moving the tool along the X-axis, Z-axis, and Y-axis, the compensation value for each axis can be changed.

Move axis	Selected offset value	State display
X-axis	X-axis compensation value	TOFS
Z-axis	Z-axis compensation value	TOFS
Y-axis	Y-axis compensation value	TOFS

1: By moving the tool along an arbitrary axis (other than rotation axes), the compensation value can be changed according to the selection of the output signals AOFS1 and AOFS2 (Gn297#5,#6).

Output	signal	Selected offset value	State display	
AOFS2	AOFS1	Selected offset value	State display	
0	0	X-axis compensation value	OFSX	
0	1	Z-axis compensation value	OFSZ	
1	1	Y-axis compensation value	OFSY	

### **NOTE**

Do not change the setting of this parameter in the active offset value modification mode.

- # 6 AON
- If a change is made to a tool compensation value (tool length compensation value used with tool length compensation A/B in the case of the M series):
- 0: In the case of the M series, the change becomes effective starting with the next block specifying G43, G44, or an H code.

  In the case of the T series, the change becomes effective starting with the next block specifying a T code.
- 1: The change becomes effective starting with the next block to be buffered.

### NOTE

- 1 This parameter is valid when bit 6 (EVO) of parameter No. 5001 is set to 0.
- 2 The operation of this parameter set to 1 is valid even if a new compensation value is further changed by MDI input or a G10 command before the new compensation value becomes effective.
- 3 The operation of this parameter set to 1 is invalid if a reset operation is performed before a new compensation value becomes effective.

	#7	#6	#5	#4	#3	#2	#1	#0
5042					OFE	OFD	OFC	OFA

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

### NOTE

When this parameter 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	±9999.9999mm					
1	0	0	0	0.00001mm	±999.99999mm					

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	0.000001inch	±99.9999999inch

	 #7	#6	#5	#4	#3	#2	#1	#0
5101						RTR		FXY
5101								

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

**# 0 FXY** The drilling axis in the drilling canned cycle is:

0: Always the Z-axis

1: The axis selected by the program

### **NOTE**

In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 15 format.

# **2 RTR** G83 and G87

0: Specify a high-speed peck drilling cycle

1: Specify a peck drilling cycle

	#7	′ #6	#5	#4	#3	#2	#1	#0
5102	RD	I RA	В		F16	QSR		
3102								

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

**QSR** Before a multiple repetitive turning canned cycle (G70 to G73) is started, a check to see if the program contains a block that has the sequence number specified in address Q is:

0: Not made.

1: Made.

When 1 is set in this parameter and the sequence number specified in address Q is not found, the alarm (PS0063) is issued and the canned cycle is not executed.

- #3 F16 When the Series 15 format is used (with bit 1 (FCV) of parameter No.0001 set to 1), a drilling canned cycle is specified using:
  - 0: Series 15 format
  - 1: Series 16 format. However, the number of repetitions is specified using address L.
- #6 RAB When a canned drilling cycle using the Series 15 format is specified (with bit 1 (FCV) of parameter No. 0001 set to 1 and bit 3 (F16) of parameter No. 5102 set to 0), address R specifies:
  - 0: Increment command.
  - 1: Absolute programming with G code system A. With G code system B or C, G90 and G91 are followed.

- #7 RDI When a canned drilling cycle using the Series 15 format is specified (with bit 1 (FCV) of parameter No. 0001 set to 1 and bit 3 (F16) of parameter No. 5102 set to 0), address R is based on:
  - Radius specification.
  - 1: Diameter/radius programming of the drilling axis.

	 #7	#6	#5	#4	#3	#2	#1	#0
5104						FCK		
5104								

[Input type]

Parameter input

[Data type]

Bit path

# 2 **FCK**  In a multiple repetitive turning canned cycle (G71/G72), the machining profile is:

- Not checked.
- Checked. 1:

The target figure specified by G71 or G72 is checked for the following before machining operation:

- If the start point of the canned cycle is less than the maximum value of the machining profile even when the plus sign is specified for a finishing allowance, the alarm (PS0322) is issued.
- If the start point of the canned cycle is greater than the minimum value of the machining profile even when the minus sign is specified for a finishing allowance, the alarm (PS0322) is issued.
- If an unmonotonous command of type I is specified for the axis in the cutting direction, the alarm (PS0064 or PS0329) is issued.
- If an unmonotonous command is specified for the axis in the roughing direction, the alarm (PS0064 or PS0329) is issued.
- If the program does not include a block that has a sequence number specified by address Q, the alarm (PS0063) is issued. This check is made, regardless of bit 2 (QSR) of parameter No.
- If a command (G41/G42) on the blank side in tool nose radius compensation is inadequate, the alarm (PS0328) is issued.

	 #7	#6	#5	#4	#3	#2	#1	#0
5105					M5T	RF2	RF1	SBC
3103								

[Input type] Parameter input

[Data type] Bit path

# 0 **SBC** 

In a drilling canned cycle, chamfering, or corner R cycle:

- A single block stop is not performed.
- A single block stop is performed.
- #1 RF1 In a multiple repetitive turning canned cycle (G71/G72) of type I, roughing is:
  - Performed. 0.
  - Not performed. 1:

### NOTE

When a roughing allowance  $(\Delta i/\Delta k)$  is specified using the Series 15 program format, roughing is performed, regardless of the setting of this parameter.

**RF2** In a multiple repetitive turning canned cycle (G71/G72) of type II, roughing is:

0: Performed.

1: Not performed.

### **NOTE**

When a roughing allowance  $(\Delta i/\Delta k)$  is specified using the Series 15 program format, roughing is performed, regardless of the setting of this parameter.

#3 M5T When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G88 with the T series, or G84/G74 with the M series):

0: M05 is output before output of M04 or M03.

1: M05 is not output before output of M04 or M03.

5110

### M code for C-axis clamping in a drilling canned cycle

[Input type]
[Data type]
[Valid data range]

Parameter input 2-word path

0 to 99999998

This parameter sets an M code for C-axis clamping in a drilling canned cycle.

Dwell time when C-axis unclamping is specified in drilling canned cycle

[Input type]
[Data type]
[Unit of data]

Parameter input

2-word path

Increment system	IS-A	IS-B	IS-C	IS-D	IS-E	Unit
Metric input	10	1	0.1	0.01	0.001	msec
Inch input	1	0.1	0.01	0.001	0.0001	msec

[Valid data range]

0 to 32767

This parameter sets the dwell time when C-axis unclamping is specified in a drilling canned cycle.

5114

### Return value of high-speed peck drilling cycle

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

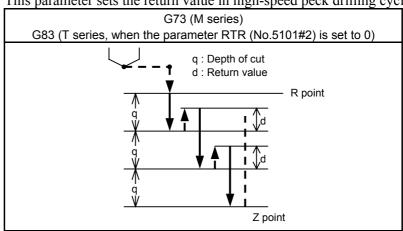
[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

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.



### Clearance value in a peck drilling cycle

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

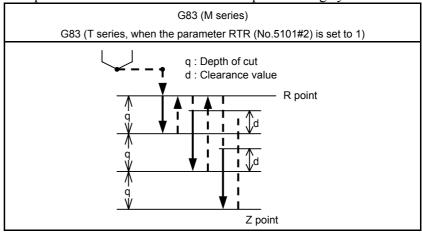
[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a clearance value in a peck drilling cycle.



5130

Cutting value (chamfering value) in thread cutting cycles G92 and G76

[Input type]

Parameter input

[Data type]

Byte path

[Unit of data]

0.1

[Valid data range]

0.1 0 to 127

This parameter sets a cutting value (chamfering value) in the thread cutting cycle (G76) of a multiple repetitive turning canned cycle and in the thread cutting cycle (G92) of a canned cycle.

Let L b a lead. Then, a cutting value range from 0.1L to 12.7L is allowed.

To specify a cutting value of 10.0L, for example, specify 100 in this parameter.

5131

### Cutting angle in thread cutting cycles G92 and G76

[Input type]

Parameter input

[Data type]

Byte path

[Unit of data]

Degree

[Valid data range]

1 to 89

This parameter sets a thread cutting angle in a thread cutting cycle (G92/G76).

When 0 is set, an angle of 45 degrees is specified.

Depth of cut in multiple repetitive turning canned cycles G71 and G72

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B) )

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the depth of cut in multiple repetitive turning canned cycles G71 and G72.

This parameter is not used with the Series 15 program format.

### NOTE

Specify a radius value at all times.

5133

Escape in multiple repetitive turning canned cycles G71 and G72

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range] Depend on the increment system of the reference axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the escape in multiple repetitive turning canned cycle G71 and G72.

### NOTE

Specify a radius value at all times.

5134

Clearance value in multiple repetitive turning canned cycles G71 and G72

.

[Input type] Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets a clearance value up to the cutting feed start point in a multiple repetitive turning canned cycle (G71/G72).

### NOTE

Specify a radius value at all times.

Retraction distance in the multiple repetitive turning canned cycle G73 (second axis on the plane)

[Input type]

Parameter input

[Data type] [Unit of data] Real path mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a retraction distance along the second axis on the plane in the multiple repetitive turning canned cycle G73. This parameter is not used with the Series 15 program format.

### NOTE

Specify a radius value at all times.

5136

Retraction distance in the multiple repetitive turning canned cycle G73 (first axis on the plane)

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

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range] Depend on the increment system of the reference axis

9 digit of minimum unit of data (refer to standard parameter setting table (A))

(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a retraction distance along the first axis on the plane in the multiple repetitive turning canned cycle G73. This parameter is not used with the Series 15 program format.

### NOTE

Specify a radius value at all times.

5137

Number of divisions in the multiple repetitive turning canned cycle G73

[Input type] [Data type] Parameter input

2-word path

[Unit of data] [Valid data range] Cycle 1 to 99999999

This parameter sets the number of divisions in the multiple repetitive turning canned cycle G73.

This parameter is not used with the Series 15 program format.

### Return in multiple repetitive turning canned cycles G74 and G75

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B) )

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the return in multiple repetitive turning canned cycles G74 and G75.

### NOTE

Specify a radius value at all times.

5140

Minimum depth of cut in the multiple repetitive turning canned cycle G76

Parameter input

[Input type] [Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets a minimum depth of cut in the multiple repetitive turning canned cycle G76 so that the depth of cut does not become too small when the depth of cut is constant.

### NOTE

Specify a radius value at all times.

5141

Finishing allowance in the multiple repetitive turning canned cycle G76

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the reference axis

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the finishing allowance in multiple repetitive turning canned cycle G76.

### NOTE

Specify a radius value at all times.

- 420 -

Repetition count of final finishing in multiple repetitive turning canned cycle

G76

[Input type]
[Data type]

Parameter input

[Unit of data]

2-word path Cycle

[Unit of data]
[Valid data range]

1 to 99999999

This parameter sets the number of final finishing cycle repeats in the multiple repetitive turning canned cycle G76.

When 0 is set, only one final finishing cycle is executed.

5143

### Tool nose angle in multiple repetitive turning canned cycle G76

[Input type]

Parameter input

[Data type]
[Unit of data]

Byte path

[Unit of data]
[Valid data range]

Degree 0, 29, 30, 55, 60, 80

This parameter sets the tool nose angle in multiple repetitive turning canned cycle G76.

This parameter is not used with the Series 15 program format.

5145

### Allowable value 1 in multiple repetitive turning canned cycles G71 and G72

[Input type]

Parameter input

[Data type]

Real path

[Unit of data]

mm, inch (input unit)

[Minimum unit of data]

Depend on the increment system of the reference axis

[Valid data range]

0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B) )

(When the increment system is IS-B, 0.0 to +999999.999)

If a monotonous command of type I or II is not specified for the axis in the roughing direction, the alarm (PS0064 or PS0329) is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71 and G72 cycles can be executed even in a program including an unmonotonous figure.

Example)

Suppose that a G71 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the plus direction along the Z-axis is specified in a target figure program, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

### NOTE

A check for a monotonous figure is made at all times during G71 and G72 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71 G72 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked.

Note that no alarm is issued when an allowable value is set.

Use a radius value to set this parameter at all times.

5146

Allowable value 2 in multiple repetitive turning canned cycles G71 and G72

[Input type]
[Data type]
[Unit of data]
[Minimum unit of data]
[Valid data range]

Parameter input

Real path

mm, inch (input unit)

Depend on the increment system of the reference axis

0 to cut of depth

If a monotonous command of type I is not specified for the axis in the cutting direction, the alarm (PS0064 or PS0329) is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71 and G72 cycles can be executed even in a program including an unmonotonous figure.

The allowable value is clamped to the depth of cut specified by a multiple repetitive turning canned cycle.

Example)

Suppose that a G71 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the minus direction along the X-axis is specified in a target figure program for moving from the bottom of cutting to the end point, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

### NOTE

A check for a monotonous figure is made at all times during G71 and G72 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71 G72 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked.

Note that no alarm is issued when an allowable value is set.

Use a radius value to set this parameter at all times.

5200	

#7	#6	#5	#4	#3	#2	#1	#0
	FHD	PCP	DOV				G84

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

### # 0 G84 Method for specifying rigid tapping

- 0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
- 1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

### # 4 DOV Override during extraction in rigid tapping

- 0: Invalidated
- 1: Validated (The override value is set in parameter No.5211. However, set an override value for rigid tapping return in parameter No. 5381.)

### **# 5 PCP** Rigid tapping

- 0: Used as a high-speed peck tapping cycle
- 1: Not used as a high-speed peck tapping cycle

### # 6 FHD Feed hold and single block in rigid tapping

- 0: Invalidated
- 1: Validated

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU			

[Input type]

Parameter input

[Data type]

Bit path

# 3 **OVU** 

The increment unit of the override parameter (No.5211) for tool rigid tapping extraction is:

0: 1%

1: 10%

# 4 OV3

A spindle speed for extraction is programmed, so override for extraction operation is:

0: Disabled.

1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				ovs				

[Input type]

Parameter input

[Data type]

Bit path

# 4 OVS

In rigid tapping, override by the feedrate override select signal and cancellation of override by the override cancel signal is:

0: Disabled.

1: Enabled.

When feedrate override is enabled, extraction override is disabled. The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

5211

### Override value during rigid tapping extraction

[Input type]

Parameter input

[Data type]

Word path

[Unit of data]

1% or 10%

[Valid data range] 0 to 200

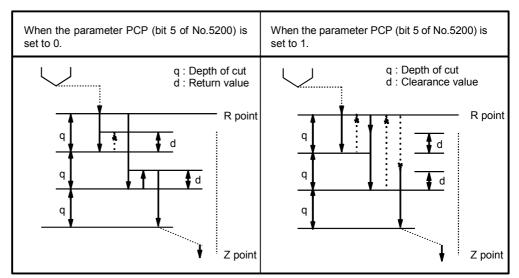
The parameter sets the override value during rigid tapping extraction.

### **NOTE**

The override value is valid when DOV in parameter No.5200 #4 is "1". When OVU (bit 3 of parameter No.5201) is 1, the unit of set data is 10%. An override of up to 200% can be applied to extraction.

# [Input type] Setting input [Data type] Real path [Unit of data] Menumumit of data Depend on the increment system of the drilling axis [Valid data range] O or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B)) (When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the return or clearance in the peck tapping cycle.



5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)
5244	Maximum spindle speed in rigid tapping (fourth gear)

[Input type]
[Data type]
[Unit of data]
[Valid data range]

Parameter input

2-word spindle

min<sup>-1</sup>

0 to 9999

Spindle position coder gear ratio

1:1 0 to 7400

1:2 0 to 9999

1:4 0 to 9999

1:8 0 to 9999

Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.

Set the same value for both parameter No.5241 and parameter No.5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200 will be issued. 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)										
5324	Spindle backlash in rigid tapping (fourth-stage gear)											
[Input type] [Data type] [Unit of data] [Valid data range]	Word s Detecti -9999 t	on unit o 9999		rs is used	l to set a	spindle	backlasł	1.				
	#7	#6	#5	#4	#3	#2	#1	#0				

5400 #1 #0 #3 #4 #3 #2 #1 #0 RIN

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

# 0 RIN

Coordinate system rotation angle command (R)

0: Specified by an absolute programming

1: Specified by an absolute programming (G90) or incremental programming (G91)

5410

Angular displacement used when no angular displacement is specified for coordinate system rotation

[Input type] Setting input [Data type] 2-word path [Unit of data] 0.001 degree

[Valid data range]

-360000 to 360000

Set an angular displacement for coordinate system rotation. When no angular displacement is specified with address T for coordinate system rotation in a block specifying G68, the setting of this parameter is used as the angular displacement for coordinate system rotation.

	#7	#6	#5	#4	#3	#2	#1	#0
5431								MDL

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

### **NOTE**

When this parameter is set, the power must be turned off before operation is continued.

A.PARAMETERS

# 0 MDL The G60 code (single direction positioning) is:

0: One-shot G code (group 00).

1: Modal G code (group 01).

	#7	#6	#5	#4	#3	#2	#1	#0
6000				HGO			MGO	
6000								

[Input type] Parameter input

[Data type] Bit path

- **#1 MGO** When a GOTO statement for specifying custom macro control is executed, a high-speed branch to 20 sequence numbers executed from the start of the program is:
  - 0: A high-speed branch is not caused to n sequence numbers from the start of the executed program.
  - 1: A high-speed branch is caused to n sequence numbers from the start of the program.
- **#4 HGO** When a GOTO statement in a custom macro control command is executed, a high-speed branch to the 30 sequence numbers immediately before the executed statement is:

0: Not made.

1: Made.

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF							

[Input type] Parameter input

[Data type] Bit path

**SKF** Dry run, override, and automatic acceleration/deceleration for G31 skip command

0: Disabled

1: Enabled

Feedrate during measurement of automatic tool compensation (T series) (for the XAE1 and GAE1 signals)

6242

Feedrate during measurement of automatic tool compensation (T series) (for the XAE2 and GAE2 signals)

[Input type]

Parameter input

[Data type] Real path

[Unit of data] mm/min

mm/min, inch/min, deg/min (machine unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis Refer to the standard parameter setting table (C)

(When the improved parameter in IC D. 0.0 to 1240000

(When the increment system is IS-B, 0.0 to +240000.0)

These parameters set the relevant feedrate during measurement of automatic tool compensation (T series) or automatic tool length measurement (M series).

### NOTE

When the setting of parameter No. 6242 or 6243 is 0, the setting of parameter No. 6241 is used.

6251

 $\gamma$  value on the X axis during automatic tool compensation (T series)

6252

[Input type]

 $\gamma$  value on the Z axis during automatic tool compensation (T series)

[Data type]
[Unit of data]
um unit of data]

Parameter input

2-word path

mm, inch, deg (machine unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

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  $\gamma$  value during automatic tool compensation (T series) or automatic tool length measurement (M series).

### NOTE

Set a radius value regardless of whether diameter or radius programming is specified.

ε value on the X axis during automatic tool compensation (T series)

6255

 $\epsilon$  value on the Z axis during automatic tool compensation (T series)

[Input type]

Parameter input

[Data type]

2-word path

[Unit of data]

mm, inch, deg (machine unit)

[Minimum unit of data] [Valid data range]

Depend on the increment system of the applied axis

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  $\varepsilon$  value during automatic tool compensation (T series) or automatic tool length measurement (M series).

### NOTE

Set a radius value regardless of whether diameter or radius programming is specified.

	#7	#6	#5	#4	#3	#2	#1	. 1
8103							MWP	

[Input type]

Parameter input

[Data type]

Bit

### NOTE

When this parameter is set, the power must be turned off before operation is continued.

### # 1 MWP

To specify a P command for the waiting M code/balance cut:

- 0: A binary value is used as conventionally done.
- 1: A path number combination is used.

19607
-------

#7	#6	#5	#4	#3	#2	#1	#0
	NAA	CAV			CCC		

[Input type]

Parameter input

[Data type]

Bit path

# 2 CCC

In the cutter compensation/tool nose radius compensation mode, the outer corner connection method is based on:

- 0: Linear connection type.
- 1: Circular connection type.

#5 CAV When an interference check finds that interference (overcutting) occurred:

- 0: Machining stops with the alarm (PS0041). (Interference check alarm function)
- 1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)

For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

**NAA** When the interference check avoidance function considers that an avoidance operation is dangerous or that a further interference to the interference avoidance vector occurs:

0: An alarm is issued.

When an avoidance operation is considered to be dangerous, the alarm (PS5447) is issued.

When a further interference to the interference avoidance vector is considered to occur, the alarm (PS5448) is issued.

1: No alarm is issued, and the avoidance operation is continued.

### NOTE

Usually, set this parameter to 0.

19625

Number of blocks to be read in the cutter or tool nose radius compensation mode

[Input type]
[Data type]
[Valid data range]

Setting input Byte path 3 to 8

This parameter sets the number of blocks to be read in the cutter or 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 or tool nose radius compensation mode, the setting does not become valid immediately. Before the new setting of this parameter can become valid, the cutter compensation/tool noise radius compensation mode must be canceled, then the mode must be entered again.

# A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks	
Bit			
Bit machine group			
Bit path	0 or 1		
Bit axis			
Bit spindle			
Byte			
Byte machine group	-128 to 127	Some parameters handle	
Byte path	0 to 255	these types of data as	
Byte axis		unsigned data.	
Byte spindle			
Word			
Word machine group	-32768 to 32767	Some parameters handle	
Word path	0 to 65535	these types of data as unsigned data.	
Word axis			
Word spindle			
2-word			
2-word machine group		Some parameters handle	
2-word path	0 to ±999999999	these types of data as	
2-word axis		unsigned data.	
2-word spindle			
Real			
Real machine group	See the Standard		
Real path	Parameter Setting		
Real axis	Tables.		
Real spindle			

### NOTE

- 1 Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
- 2 For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
- 3 For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
- 4 For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
- 5 For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
- 6 The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

# A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

### **NOTE**

- 1 Values are rounded up or down to the nearest multiples of the minimum data unit.
- 2 A valid data range means data input limits, and may differ from values representing actual performance.
- 3 For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value."

### (A) Length and angle parameters (type 1)

Unit of data	Increment	Minimum	Valid data range	
	IS-A	0.01	-999999.99 to +999999.99	
mm	IS-B	0.001	-999999.999 to +999999.999	
mm	IS-C	0.0001	-99999.9999 to +99999.9999	
deg.	IS-D	0.00001	-9999.99999 to +9999.99999	
	IS-E	0.000001	-999.999999 to +999.999999	
	IS-A	0.001	-99999.999 to +99999.999	
	IS-B	0.0001	-99999.9999 to +99999.9999	
inch	IS-C	0.00001	-9999.99999 to +9999.99999	
	IS-D	0.000001	-999.999999 to +999.999999	
	IS-E	0.0000001	-99.9999999 to +99.9999999	

### (B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range	
	IS-A	0.01	0.00 to +999999.99	
mm	IS-B	0.001	0.000 to +999999.999	
mm deg.	IS-C	0.0001	0.0000 to +99999.9999	
ueg.	IS-D	0.00001	0.00000 to +9999.99999	
	IS-E	0.000001	0.000000 to +999.999999	
	IS-A	0.001	0.000 to +99999.999	
	IS-B	0.0001	0.0000 to +99999.9999	
inch	IS-C	0.00001	0.00000 to +9999.99999	
	IS-D	0.000001	0.000000 to +999.99999	
	IS-E	0.0000001	0.0000000 to +99.9999999	

# (C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
	IS-A	0.01	0.00 to +999000.00
	IS-B	0.001	0.000 to +999000.000
mm/min	IS-C	0.0001	0.0000 to +99999.9999
degree/min	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
	IS-A	0.001	0.000 to +96000.000
	IS-B	0.0001	0.0000 to +9600.0000
inch/min	IS-C	0.00001	0.00000 to +4000.00000
	IS-D	0.000001	0.000000 to +400.000000
	IS-E	0.000001	0.0000000 to +40.0000000

# (D)Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
	IS-A	0.01	0.00 to +999999.99
mm/sec <sup>2</sup>	IS-B	0.001	0.000 to +999999.999
deg./sec <sup>2</sup>	IS-C	0.0001	0.0000 to +99999.9999
ueg./sec	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
inch/sec <sup>2</sup>	IS-C	0.00001	0.00000 to +9999.99999
	IS-D	0.000001	0.000000 to +999.999999
	IS-E	0.0000001	0.0000000 to +99.9999999

B-63944EN-1/02 INDEX

# **INDEX**

<a></a>	< <u>E</u> >
ACTIVE OFFSET VALUE CHANGE FUNCTION	End Face Peck Drilling Cycle (G74)77, 301
BASED ON MANUAL FEED243	End Face Turning Cycle (G94)42, 262
ADDRESSES AND SPECIFIABLE VALUE RANGE	EXTENDED TOOL SELECTION231
FOR SERIES 15 PROGRAM FORMAT248	Extraction override115
AUTOMATIC TOOL OFFSET (G36, G37)235	<f></f>
<b></b>	Face cutting cycle
BALANCE CUT (G68, G69)337	Finishing Cycle (G70)73, 297
Boring Cycle (G85)331, 333	Front Boring Cycle (G85) / Side Boring Cycle (G89)99
	Front Drilling Cycle (G83)/Side Drilling Cycle (G87) 94
<c></c>	FRONT FACE RIGID TAPPING CYCLE (G84) /
CANNED CYCLE	SIDE FACE RIGID TAPPING CYCLE (G88)103
CANNED CYCLE (G90, G92, G94)30	Front Tapping Cycle (G84) /
Canned Cycle and Tool Nose Radius Compensation	Side Tapping Cycle (G88)97
47, 268	FUNCTIONS TO SIMPLIFY PROGRAMMING29
Canned Cycle Cancel (G80)	<h></h>
Canned Cycle for Drilling Cancel (G80)	High-speed Peck Drilling Cycle (G83.1)
Chuck and Tail Stock Barriers	How to Use Canned Cycles (G90, G92, G94)45
COMPENSATION FUNCTION	riow to use callifed cycles (090, 092, 094)43
CONSTANT LEAD THREADING (G32)20	
CONTINUOUS THREADING (G32)24	Imaginary Tool Nose
COORDINATE SYSTEM ROTATION	Input of Tool Offset Value Measured B
(G68.1, G69.1)	INPUT/OUTPUT ON EACH SCREEN348
CORNER CIRCULAR INTERPOLATION (G39)228	INPUT/OUTPUT ON THE ALL IO SCREEN353
Counter Input of Offset value	Inputting and Outputting Tool Offset /
Cutter or Tool Nose Radius Compensation for Input	2nd Geometry Data351, 355
from MDI	Inputting and Outputting Y-axis Offset Data349, 354
	Inputting tool offset / 2nd geometry data351
<d></d>	Inputting Y-axis offset data349
DATA INPUT/OUTPUT347	Interference Check
DATA TYPE431	Interference check alarm function
DESCRIPTION OF PARAMETERS388	Interference check avoidance function
DETAILS OF CUTTER OR TOOL NOSE RADIUS	INTERPOLATION FUNCTION
COMPENSATION171	<m></m>
DIRECT DRAWING DIMENSION	MEMORY OPERATION BY SERIES 15 FORMAT 247
PROGRAMMING128	MIRROR IMAGE FOR DOUBLE TURRET
Direct Input of Tool Offset Value	(G68, G69)126
Direction of Imaginary Tool Nose	MUITI-PATH CONTROL FUNCTION336
Drilling Cycle, Counter Boring (G82)325	MULTIPLE REPETITIVE CYCLE272
Drilling Cycle, Spot Drilling Cycle (G81)323	MULTIPLE REPETITIVE CYCLE (G70-G76)51

MULTIPLE THREADING25	<\$>
Multiple Threading Cycle (G76 <g a="" b="" code="" system="">)</g>	SCREENS DISPLAYED BY FUNCTION KEY
(G78 < G code system C>)305	<offset setting="">357</offset>
Multiple Threading Cycle (G76)81	Second Geometry Tool Offset143
	SETTING AND DISPLAYING DATA356
<n></n>	Setting and Displaying the Tool Offset Value358
NOTES ON READING THIS MANUAL7	Setting the Workpiece Coordinate System Shift Value. 368
Notes on Tool Nose Radius Compensation161	Setting the Y-Axis Offset
NOTES ON VARIOUS KINDS OF DATA7	Setting Tool Compensation/Second Geometry Offset
<0>	Values371
OFFSET12	STANDARD PARAMETER SETTING TABLES432
Offset	Stock Removal in Facing (G72)65, 288
Offset Number	Stock Removal in Turning (G71)52, 273
Offset Number and Offset Value151	Straight cutting cycle31, 251
Operation to be performed if an interference is judged	Straight threading cycle
to occur217	SUBPROGRAM CALLING249
Outer Diameter / Internal Diameter Drilling Cycle	<t></t>
(G75)79, 303	T Code for Tool Offset137
Outer Diameter/Internal Diameter Cutting Cycle	Taper cutting cycle
(G90)31, 251	Taper threading cycle
Outputting tool offset / 2nd geometry data352	Tapping Cycle (G84)
Outputting Y-axis Offset Data	Threading Cycle (G92)
Override during Rigid Tapping115	Tool Geometry Offset and Tool Wear Offset136
Override signal	Tool Movement in Offset Mode
OVERVIEW OF CUTTER COMPENSATION	Tool Movement in Offset Mode Cancel
(G40-G42)164	Tool Movement in Start-up
OVERVIEW OF TOOL NOSE RADIUS	TOOL OFFSET
COMPENSATION (G40-G42)146	Tool Selection
<p></p>	TORQUE LIMIT SKIP (G31 P99)27
PARAMETERS387	
Pattern Repeating (G73)	< <i>V&gt;</i>
Peck Drilling Cycle (G83)	VECTOR RETENTION (G38)227
Peck Rigid Tapping Cycle (G84 or G88)109	<w></w>
Precautions to Be Taken by Operator	Workpiece Position and Move Command154
PREPARATORY FUNCTION (G FUNCTION)	
Prevention of Overcutting Due to Cutter or Tool Nose	<y></y>
Radius Compensation	Y Axis Offset
Restrictions on Canned Cycles	
· <r></r>	
Restrictions on Canned Cycles	
Restrictions on Multiple Repetitive Cycle313	
Restrictions on Multiple Repetitive Cycle (G70-G76)88	
RIGID TAPPING102	

Revision Record

FANUC Series 30i/300i/300is-MODEL A, Series 31i/310i/310is-MODEL A5, Series 31i/310i/310is-MODEL A. Series 32i/320i/320is-MODEL A. USER'S MANUAL (For Lathe System)(B-63944EN-1)

				Contents
				Date
				Edition
		Addition of functions Addition of following models - Series 31i /310i /310is-MODEL A5 - Series 32i /320i /320is-MODEL A		Contents
		Jun, 2004	Jul., 2003	Date
		02	01	Edition