

---

# **PROGRAMMING MANUAL (NC)**

Read this manual before performing work.

**brother**

---

## Introduction

**Thank you for purchasing this machine.**

**Always be sure to read this manual carefully first, in order to use the machine functions properly and safely.**

**Always be sure to carefully read this Programming Manual as well as the Operation Manual before use.**

**This manual gives a description of NC programming.**

**This machine can be used to carry out drilling, tapping and facing.**

**This machine manual is divided into the following sections.**

- **Operation Manual**  
This manual describes the operation procedure for the machine.
- **Installation Manual**  
This manual describes the machine's installation procedure and inspections.
- **Programming Manual**  
This manual provides a program description.

**Brother is not responsible or liable for accidents that occur during special machine use or handling that does not follow the general safety usage guidelines.**

**Keep this manual for future reference.**

**Attach this manual to the machine if it is resold.**

**Contact the nearest Brother sales office or Brother approved service dealer if this manual or the safety labels are damaged, lost or missing. (Charges apply)**

**The re-exporting and resale of this machine is regulated by Japan's export laws and regulations in accordance with international export management.**  
**When exporting, permission from the exporting country's government and/or from the Japanese government may be required.**  
**Contact a Brother Industries dealer in advance before re-transferring, reselling or re-exporting this machine.**

**Copying and reprinting all or part of the content in this manual without permission is illegal.**

**The content of this manual may be changed without prior notice.**

**Brother has taken steps to ensure this manual is accurate and complete. However, if you notice or suspect that there is an error, please contact the nearest Brother sales office or Brother approved service dealer.**

## How to Read This Manual

**This manual is divided into the following sections.**

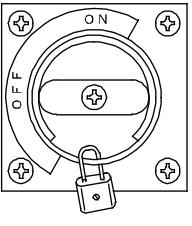
- (1) Overview**----- A summary of the content is provided for the corresponding section.
- (2) Warning**----- A warning is provided for any hazards that could potentially cause serious bodily injury, death or damage to the machine.  
The hazards are described in the following order.
  - (2-1) Hazard level
  - (2-2) Type of hazard
  - (2-3) Potential damage
  - (2-4) Safety directions to avoid danger
- (3) Operation procedure**--- The procedure describes how to operate each function.
- (4) Screenshot**---- A screenshot is inserted into places to highlight certain points in the operation procedure. The screenshot messages are shown at an approximate position and may differ slightly from the actual position of the line or column. The same applies to the font.
- (5) Illustration**----- Illustrations, such as explanatory drawings, diagrams that show the dimensions, positioning, ranges, figures or configurations, are used in certain places where a written explanation alone may be hard to understand.

(2) Warning
(3) Operation procedure
(1) Overview

1.3 Take the necessary steps to ensure safe operation

**▲ WARNING**


You may break or fracture your foot if you drop a heavy object on it.  
Therefore, use safety footwear when lifting or carrying heavy objects.



1 - 2

**1.3.1 Before starting operation**  
Review the following items before starting operation.

- (1) Turn the main power breaker handle on the control box door to the OFF position.  
Do not touch the primary power and terminal for the main power breaker because it has a high voltage.
- (2) A sign must be posted to warn others that work is in progress.
- (3) Do not let others approach the machine or the moving parts on the machine.
- (4) Make sure that there are no unnecessary objects around the machine.



1 - 3

(5) Illustration
(4) Screenshot

---

(This page was intentionally left blank.)

<b>CHAPTER 1</b>	<b>PROGRAM STRUCTURE</b>	<b>1-1</b>
1.1	Program Types and Structure	1-2
1.2	Block Structure	1-2
1.3	Word Structure	1-3
1.4	Number Commands	1-3
1.5	Sequence Numbers	1-5
1.6	Optional Block Skip	1-5
1.7	Control Out-In Function	1-5
<b>CHAPTER 2</b>	<b>COORDINATE COMMANDS</b>	<b>2-1</b>
2.1	Coordinate System and Coordinate Commands	2-2
2.2	Machine Zero Point and Machine Coordinates	2-2
2.3	Workpiece Coordinates	2-3
<b>CHAPTER 3</b>	<b>PREPARATION FUNCTION</b>	<b>3-1</b>
3.1	Outline of G Code	3-2
3.2	Positioning (G00, G60)	3-5
3.2.1	Positioning (G00)	3-5
3.2.2	Single Direction Positioning Function (G60)	3-6
3.2.3	Precautions for Programming Involving Use of Rotation Axis (Index Table)	3-7
3.3	Cutting (G01 to 03, G12/13, G102/103, G202/203)	3-8
3.3.1	Linear Interpolation (G01)	3-8
3.3.1.1	Chamfering to Desired Angle and Cornering R	3-9
3.3.2	Circular Interpolation/Helical Thread Cutting Interpolation	3-12
3.3.2.1	Circular Interpolation (G02, G03)	3-12
3.3.2.2	XZ Circular Interpolation (G102, G103)	3-13
3.3.2.3	YZ Circular Interpolation (G202, G203)	3-14
3.3.2.4	Precautions for Circular Interpolation	3-15
3.3.2.5	Helical Thread Cutting Interpolation	3-18
3.3.2.6	Spiral Interpolation	3-19
3.3.2.7	Conical Interpolation	3-21
3.3.2.8	Cutter Compensation for Spiral and Conical Interpolation	3-24
3.3.2.9	Involute interpolation	3-25
3.3.3	Circle Cutting (G12, G13)	3-26
3.4	Dwell (G04)	3-28
3.5	Exact Stop Check (G09, G61, G64)	3-29
3.6	Programmable Data Input (G10)	3-30
3.6.1	Entering Workpiece Coordinate Zero	3-30
3.6.2	Entering Tool Length/Cutter Compensation Data	3-30
3.6.3	Tool Wear Compensation	3-31
3.6.4	Tool Offset Data Input	3-31
3.6.5	Tool Offset wear Data input	3-32
3.6.6	Data Input of Extended Workpiece Coordinate	3-32
3.6.7	Workpiece Coordinate System Using the Results of Measurements	3-32
3.6.8	Data Input of Extended Workpiece Coordinate Zero Based on Measurement Results	3-33
3.6.9	Data Input for Reference Rotary Fixture Offset	3-33
3.6.10	Tool Information Input	3-34
3.7	Coordinate System (G17 to 19, G52 to 59, G54.1, G92)	3-35
3.7.1	Plane Selection (G17, G18, G19)	3-35
3.7.2	Machine Coordinate System Selection (G53)	3-35
3.7.3	Workpiece Coordinate System Selection (G54 to G59)	3-35
3.7.4	Extended Workpiece Coordinate System Selection (G54.1)	3-36
3.7.5	Workpiece Coordinate System Setting (G92)	3-36
3.7.6	Local Coordinate System Function (G52)	3-39
3.7.7	Rotary Fixture Offset Function (G54.2)	3-39
3.7.7.1	Overview	3-39
3.7.7.2	Command Format	3-40
3.7.8	Tilt Plane Coordinate Setting Function	3-41

3.7.8.1	Overview	3-41
3.7.8.2	Command Format	3-42
3.7.8.3	Restrictions	3-47
<b>3.8</b>	<b>Soft Limit</b>	<b>3-48</b>
3.8.1	Stroke	3-48
3.8.2	Stroke Limit	3-48
3.8.3	Programmable Stroke Limit (G22)	3-49
<b>3.9</b>	<b>Reference Position (G28 to G30)</b>	<b>3-50</b>
3.9.1	Return to the Reference Point (G28)	3-50
3.9.2	Return from the Reference Point (G29)	3-50
3.9.3	Return to the 2nd to 6th Reference Point (G30)	3-51
<b>3.10</b>	<b>Skip Function (G31, G131/G132)</b>	<b>3-52</b>
3.10.1	Before using the skip function	3-52
3.10.2	Skip Function (G31, G131/G132)	3-52
3.10.3	Continuous Skip Function (G31)	3-53
<b>3.11</b>	<b>Scaling (G50/G51)</b>	<b>3-54</b>
<b>3.12</b>	<b>Programmable Mirror Image (G50.1/51.1)</b>	<b>3-57</b>
<b>3.13</b>	<b>Rotational Transformation Function (G68/69, 168)</b>	<b>3-60</b>
3.13.1	Coordinate Rotation (G68/69)	3-60
3.13.2	Coordinate Rotation Using Measured Results (G168)	3-61
<b>3.14</b>	<b>Absolute Command and Incremental Command (G90/91)</b>	<b>3-62</b>
<b>3.15</b>	<b>Change of Tap Twisting Direction (G133/134)</b>	<b>3-64</b>
<b>3.16</b>	<b>G Code Priority</b>	<b>3-64</b>
<b>3.17</b>	<b>Programmable Data Input (High Accuracy) (G210)</b>	<b>3-71</b>
3.17.1	High Accuracy Mode A Parameter Changes	3-71
3.17.2	High Accuracy Mode B Parameter Changes	3-72
<b>3.18</b>	<b>Thread Cutting</b>	<b>3-73</b>
3.18.1	Single Start Lead Thread Cutting (G33)	3-73
3.18.2	Thread Cutting Cycle (G392)	3-75
3.18.2.1	Thread Cutting Cycle for Straight Screws	3-75
3.18.2.2	Thread Cutting Cycle for Taper Screws	3-76
3.18.3	Thread Cutting in Complex Thread Cutting Cycle (G376)	3-78
3.18.3.1	Infeed	3-79
3.18.3.2	Detailed Description of Operation	3-80
3.18.4	Thread Runout	3-83
3.18.5	Program Example	3-84
3.18.5.1	Machining and Thread Cutting Straight Screws	3-84
3.18.5.2	Machining and Thread Cutting Taper Screws	3-85
<b>3.19</b>	<b>Lathe Machining Infeed Direction</b>	<b>3-86</b>

## CHAPTER 4 PREPARATION FUNCTIONS (COMPENSATION FUNCTIONS) ----- 4-1

<b>4.1</b>	<b>Cutter Compensation (G40, G41 and G42)</b>	<b>4-2</b>
4.1.1	Cutter Compensation Function	4-2
4.1.1.1	Cutter Wear Offset	4-2
4.1.2	Cancel Mode	4-3
4.1.3	Startup	4-4
4.1.3.1	Inner Side Cutting ( $180^\circ \leq \theta$ )	4-4
4.1.3.2	Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )	4-5
4.1.3.3	Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )	4-6
4.1.4	Offset Mode	4-7
4.1.4.1	Inner Side Cutting ( $180^\circ \leq \theta$ )	4-7
4.1.4.2	Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )	4-9
4.1.4.3	Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )	4-10
4.1.4.4	Exceptional Cases	4-11
4.1.5	Offset Cancel	4-12
4.1.5.1	Inner Side Cutting ( $180^\circ \leq \theta$ )	4-12
4.1.5.2	Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )	4-13
4.1.5.3	Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )	4-14

4.1.6	G40 Individual Command	4-15
4.1.7	Compensation Direction Change in Offset Mode	4-16
4.1.8	Offset Direction Change in Offset Mode	4-17
4.1.8.1	When There is an Intersection	4-17
4.1.8.2	When There is No Intersection	4-18
4.1.8.3	When an Arc Laps Around in a Circle	4-19
4.1.8.4	Arc angle check when offset direction changes	4-20
4.1.9	G Code Command for Cutter Compensation in Offset Mode	4-21
4.1.10	Special Notes for Cutter Compensation	4-22
4.1.11	Override Function Related to Cutter Compensation	4-28
4.1.11.1	Automatic Corner Override	4-28
4.1.11.2	Inner Arc Override	4-29
4.1.12	Arc Angle Check During Inner Side Cutting	4-30
4.1.13	Interference Check for Cutter Compensation / Nose R Compensation	4-31
4.1.13.1	Interference detection method	4-31
4.1.13.2	Operation when interference is detected	4-33
4.1.13.3	Restrictions and special notes	4-33
<b>4.2</b>	<b>Tool Length Offset (G43, G44 and G49)</b>	<b>4-34</b>
4.2.1	Tool Length Offset Function	4-34
4.2.2	Tool Length Wear Offset	4-35
4.2.3	Z-axis Travel with Tool Length Offset Command	4-35
<b>4.3</b>	<b>Nose R Compensation (G141 and G142 - Option)</b>	<b>4-37</b>
4.3.1	Command Format	4-38
4.3.1.1	Nose R Wear Offset	4-39
4.3.2	Virtual Teeth	4-40
4.3.3	Cancel Mode	4-40
4.3.4	Startup	4-41
4.3.4.1	Inner Side Cutting ( $180^\circ \leq \theta$ )	4-43
4.3.4.2	Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )	4-44
4.3.4.3	Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )	4-45
4.3.5	Offset Mode	4-46
4.3.5.1	Inner Side Cutting ( $180^\circ \leq \theta$ )	4-46
4.3.5.2	Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )	4-48
4.3.5.3	Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )	4-49
4.3.5.4	Exceptional Cases	4-50
4.3.6	Offset Cancel	4-51
4.3.6.1	Inner Side Cutting ( $180^\circ \leq \theta$ )	4-51
4.3.6.2	Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )	4-52
4.3.6.3	Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )	4-53
4.3.7	G40 Individual Command	4-54
4.3.8	Compensation Direction Change in Offset Mode	4-55
4.3.9	Offset Direction Change in Offset Mode	4-55
4.3.9.1	When There is an Intersection	4-55
4.3.9.2	When There is No Intersection	4-56
4.3.10	G Code Command for Nose R Compensation in Offset Mode	4-58
4.3.11	Special Notes for Nose R Compensation	4-59
4.3.12	Override Function Related to Nose R Compensation	4-64
4.3.12.1	Automatic Corner Override	4-64
4.3.12.2	Inner Arc Override	4-65
4.3.13	Arc Angle Check During Inner Side Cutting	4-66
<b>4.4</b>	<b>Tool Position Compensation (G143, G144 and G49 - Option)</b>	<b>4-67</b>
4.4.1	Tool Position Compensation Function	4-67
4.4.2	Axis Travel with Tool Position Compensation	4-68
<b>CHAPTER 5</b>	<b>PREPARATION FUNCTION (CANNED CYCLE)</b>	<b>5-1</b>
<b>5.1</b>	<b>Outline</b>	<b>5-2</b>
<b>5.2</b>	<b>List of Canned Cycle Function</b>	<b>5-2</b>
<b>5.3</b>	<b>Basic Operation of Canned Cycle</b>	<b>5-3</b>
<b>5.4</b>	<b>General Rules of Canned Cycle</b>	<b>5-4</b>
5.4.1	Canned Cycle Operation Commands	5-4
5.4.2	Data in Absolute and Incremental Mode	5-4

5.4.3	Types of Return Points (G98, G99)-----	5-5
5.4.4	Canned Cycle Operating Conditions -----	5-5
5.4.5	Canned Cycle Machining Data -----	5-6
5.4.6	Canned Cycle Repetition Count -----	5-7
<b>5.5</b>	<b>Details of Canned Cycle -----</b>	<b>5-8</b>
5.5.1	High Speed Peck Drilling Cycle (G73) -----	5-8
5.5.2	Reverse Tapping Cycle (G74) -----	5-9
5.5.3	Fine Boring Cycle (G76) -----	5-10
5.5.4	Tapping Cycle (Synchro Mode) (G77)-----	5-11
5.5.5	Reverse Tapping Cycle (Synchro Mode) (G78) -----	5-12
5.5.6	Drilling Cycle (G81, G82)-----	5-14
5.5.7	Peck Drilling Cycle (G83)-----	5-15
5.5.8	Tapping Cycle (G84) -----	5-16
5.5.9	Boring Cycle (G85, G89)-----	5-17
5.5.10	Boring Cycle (G86)-----	5-18
5.5.11	Back Boring Cycle (G87)-----	5-19
5.5.12	End Milling/Tapping Cycle (G177)-----	5-20
5.5.13	End Milling/Tapping Cycle (G178)-----	5-21
5.5.14	Double Drilling Cycle (G181, G182)-----	5-22
5.5.15	Double Boring Cycle (G185, G189)-----	5-23
5.5.16	Double Boring Cycle (G186) -----	5-24
5.5.17	Deep Hole Tapping Cycle (Synchro Mode) (G277) -----	5-25
5.5.18	Reverse Deep Hole Tapping Cycle (Synchro Mode) (G278)-----	5-26
5.5.19	Reducing Step of Canned Cycle-----	5-27
5.5.20	Canned Cycle Cancel (G80) -----	5-34
5.5.21	General Precautions for Canned Cycle-----	5-34
<b>5.6</b>	<b>One-shot Canned Cycle-----</b>	<b>5-35</b>
5.6.1	High Speed Peck Drilling Cycle (G173)-----	5-35
5.6.1.1	High Speed Peck Drilling Cycle (G173) (Reducing Step)-----	5-36
5.6.2	Peck Drilling Cycle (G183)-----	5-37
5.6.2.1	Peck Drilling Cycle (G183) (Reducing Step)-----	5-38
<b>5.7</b>	<b>Canned Cycle for Tool Change (Non-stop ATC) (G100) -----</b>	<b>5-39</b>
5.7.1	S300X1/S500X1/S700X1/S1000X1/R450X1/R650X1/ S300X2/S500X2/S700X2/F600X1/R450X2/R650X2 -----	5-39
5.7.2	M140X1/M140X2/M200X3/M300X3 -----	5-41
5.7.2.1	Operation When Spindle is Selected (M141 Modal) -----	5-41
5.7.2.2	Operation When Turning Spindle is Selected (M142 Modal)-----	5-41
5.7.2.3	Tool change operation when lathe spindle is selected -----	5-42
5.7.2.4	Notes -----	5-42
5.7.3	R650X2 40MG-----	5-43
5.7.3.1	Notes -----	5-44
5.7.3.2	When changing from standard tool to standard tool or from large tool to large tool-----	5-45
5.7.3.3	When changing from standard tool to large tool or from large tool to standard tool-----	5-45
5.7.3.4	Next tool preparation operation -----	5-46
5.7.4	Simultaneously Commandable M Codes -----	5-46
5.7.5	Automatic Command of Tool Data in Tool Change -----	5-49
<b>5.8</b>	<b>Coordinate Calculation Function -----</b>	<b>5-50</b>
5.8.1	Outline -----	5-50
5.8.2	Coordinate Calculation -----	5-50
5.8.3	C Coordinates Calculation Parameters -----	5-50
5.8.4	Description of Coordinate Calculation Function -----	5-51
5.8.4.1	Bolt Hole Circle (G36)-----	5-51
5.8.4.2	Line (Angle) (G37)-----	5-51
5.8.4.3	Line (X, Y) (G38) -----	5-52
5.8.4.4	Grid (G39) -----	5-53
5.8.5	Examples of Application -----	5-54
<b>CHAPTER 6</b>	<b>MACRO -----</b>	<b>6-1</b>
<b>6.1</b>	<b>What is Macro? -----</b>	<b>6-2</b>
<b>6.2</b>	<b>Variables Function-----</b>	<b>6-4</b>
6.2.1	Outline-----	6-4



6.2.2	Expression of Variables	6-4
6.2.3	Undefined Variables	6-4
6.2.4	Types of Variables	6-5
6.2.5	Display and Setting of Variables	6-6
6.2.6	System Variables	6-6
6.2.6.1	Interface I/O Signals	6-6
6.2.6.2	Workpiece Coordinate Zero Point	6-7
6.2.6.3	Tool Data	6-7
6.2.6.4	Alarm Display	6-8
6.2.6.5	Message Display and Stop	6-8
6.2.6.6	Time	6-8
6.2.6.7	Operation Control	6-8
6.2.6.8	Mirror Image	6-9
6.2.6.9	Modal Info	6-9
6.2.6.10	Current Position	6-11
6.2.6.11	ATC Tool	6-11
6.2.6.12	Workpiece Counter	6-12
6.2.6.13	Result of Auto Workpiece Measurement	6-12
6.2.6.14	Rotary Fixture Offset	6-12
6.2.6.15	Machining load monitor	6-13
<b>6.3</b>	<b>Calculation Function</b>	<b>6-14</b>
6.3.1	Types of Calculation	6-14
6.3.2	Precedence of Calculation	6-14
6.3.3	Precautions for Calculation	6-15
<b>6.4</b>	<b>Control Function</b>	<b>6-16</b>
6.4.1	GOTO Instruction (Unconditional Branching)	6-16
6.4.2	IF Instruction (Conditional Branching)	6-16
6.4.3	WHILE Instruction (Repetition)	6-17
6.4.4	Precautions for Control Function	6-18
<b>6.5</b>	<b>Call Function</b>	<b>6-21</b>
6.5.1	Simple Call Function	6-21
6.5.2	Modal Call Function	6-22
6.5.3	G Code Macro Call	6-23
6.5.4	M Code Macro Call	6-24
6.5.5	Macro Call Arguments	6-25
6.5.6	Difference between G65 and M98	6-27
6.5.7	Multiple Call	6-27
<b>6.6</b>	<b>External Output Function</b>	<b>6-28</b>
6.6.1	POPEN	6-28
6.6.2	BPRNT	6-29
6.6.3	DPRNT	6-30
6.6.4	PCLOS	6-31
6.6.5	External Output to Memory Card	6-31
6.6.6	External Output to FTP Server	6-32
6.6.7	Precautions for External Output Command	6-34
<b>6.7</b>	<b>Interrupt Macro (Option)</b>	<b>6-35</b>
6.7.1	Interrupt Type	6-37
6.7.2	Call Type	6-39
6.7.3	Acceptance Type	6-39
6.7.4	Interrupt Macro and Modal Information	6-40
6.7.5	Interrupt Macro and Current Position	6-41
6.7.6	Macro Statement and NC Statement	6-41
6.7.7	Restrictions	6-42
6.7.8	Reference Folder When Interrupt Type Macro is Executed	6-43
<b>CHAPTER 7</b>	<b>AUTOMATIC WORKPIECE MEASUREMENT</b>	<b>7-1</b>
7.1	List of Automatic Workpiece Measurement Functions	7-2
7.2	Before Automatic Workpiece Measurement	7-4
7.3	Setting Data for Automatic Workpiece Measurement	7-5
7.4	Command Procedure for Automatic Workpiece Measurement	7-10

7.4.1	Corner-----	7-10
7.4.2	Parallel-----	7-14
7.4.3	Circle Center-----	7-16
7.4.4	Z axis Height -----	7-20
7.4.5	Positioning to Measurement Position -----	7-20
<b>7.5</b>	<b>Measurement Results Processing -----</b>	<b>7-21</b>
7.5.1	Display Screen for Measurement Results-----	7-21
7.5.2	Apply Measurement Results to Workpiece Coordinates -----	7-22
<b>7.6</b>	<b>Lock Key Operations-----</b>	<b>7-23</b>
<b>7.7</b>	<b>Program Restart Operation -----</b>	<b>7-23</b>
<b>CHAPTER 8</b>	<b>SUB PROGRAM FUNCTION -----</b>	<b>8-1</b>
<b>8.1</b>	<b>Overview -----</b>	<b>8-2</b>
<b>8.2</b>	<b>Create Sub Program-----</b>	<b>8-3</b>
<b>8.3</b>	<b>Simple Call Function -----</b>	<b>8-4</b>
<b>8.4</b>	<b>Specify Return Number from Sub Program Function -----</b>	<b>8-6</b>
<b>8.5</b>	<b>Call Specifying Sequence No. -----</b>	<b>8-8</b>
<b>CHAPTER 9</b>	<b>FEED FUNCTION -----</b>	<b>9-1</b>
<b>9.1</b>	<b>Feed Function -----</b>	<b>9-2</b>
9.1.1	Feedrate per Minute (G94) -----	9-2
9.1.2	Feedrate per Rotation (G95) -----	9-2
9.1.3	Inverse Time Feed (G93)-----	9-2
9.1.4	Command Range-----	9-3
9.1.5	Switching Between Feedrate per Minute / Feedrate per Rotation / Inverse Time Feed-----	9-4
<b>9.2</b>	<b>Automatic F Command at Tool Change -----</b>	<b>9-4</b>
<b>CHAPTER 10</b>	<b>SPINDLE RELATED FUNCTIONS</b>	
	<b>(S FUNCTION)-----</b>	<b>10-1</b>
<b>10.1</b>	<b>S Function-----</b>	<b>10-2</b>
10.1.1	Spindle Speed Command -----	10-2
10.1.2	Constant Peripheral Speed Control (G96, G97) (Option) -----	10-2
10.1.3	Spindle Speed Clamp (G92) (Option) -----	10-4
10.1.4	Automatic S Command at Tool Change -----	10-4
10.1.5	Register Maximum Speed -----	10-4
<b>10.2</b>	<b>M Function (Spindle Control)-----</b>	<b>10-5</b>
10.2.1	Spindle Normal Rotation (M03)-----	10-5
10.2.2	Spindle Reverse Rotation (M04) -----	10-5
10.2.3	Spindle Stop (M05)-----	10-5
10.2.4	Spindle Orientation (M19) -----	10-5
10.2.4.1	Spindle Orientation to a Given Angle -----	10-5
10.2.5	Spindle Orientation (M111) -----	10-5
<b>10.3</b>	<b>M Function (Lathe Spindle Control)-----</b>	<b>10-6</b>
10.3.1	Lathe Spindle Normal Rotation (M303) -----	10-6
10.3.2	Lathe Spindle Reverse Rotation (M304)-----	10-6
10.3.3	Lathe Spindle Stop (M305)-----	10-6
<b>10.4</b>	<b>M Function (Spindle Selection) -----</b>	<b>10-6</b>
10.4.1	Spindle Selection (M141) -----	10-6
10.4.2	Lathe Spindle Selection (M142)-----	10-6
<b>CHAPTER 11</b>	<b>TOOL RELATED FUNCTIONS (T FUNCTION)----</b>	<b>11-1</b>
<b>11.1</b>	<b>T Function -----</b>	<b>11-2</b>
11.1.1	When Issuing a Command Using the Tool Number -----	11-2
11.1.2	When Issuing a Command Using the Pot Number (Magazine Number) -----	11-2
11.1.3	When Issuing a Command Using the Group Number -----	11-2
<b>11.2</b>	<b>M Function (Tool Control) -----</b>	<b>11-3</b>
11.2.1	Tool Change (M06)-----	11-3

11.2.2	Tool Life Counters (M230 to M231)	11-3
11.2.3	Changing ATC Arm Turn Speed (M420 to M423, M432)	11-3
11.2.4	Shutter and Cover Related M Codes (M434, M438, M439)	11-4
11.2.5	Specify Magazine Turn Speed (M435, M436, M437)	11-4
11.2.6	Magazine Turns to the Tool Installation Position (M501 to M599)	11-4
11.2.7	Machining Load Monitor Function (M340 to M343)	11-4
<b>CHAPTER 12</b>	<b>M FUNCTION</b>	<b>12-1</b>
<b>12.1</b>	<b>Outline of M Function</b>	<b>12-2</b>
<b>12.2</b>	<b>M Code List</b>	<b>12-3</b>
12.2.1	Multiple M Code Commands in One Block	12-6
<b>12.3</b>	<b>M Function (Program control)</b>	<b>12-9</b>
12.3.1	Program Stop (M00)	12-9
12.3.2	Optional Stop (M01)	12-9
12.3.3	End of Program (M02, M30)	12-9
12.3.4	Workpiece Counter Specification (M211 to M214)	12-9
12.3.5	Workpiece Counter Cancel (M221 to M224)	12-9
12.3.6	To Prohibit Reading Ahead (M159)	12-9
<b>12.4</b>	<b>M Function (Signal Control)</b>	<b>12-10</b>
12.4.1	2-digit BCD Signal Output	12-10
12.4.2	To Check TOUCH Signal (M120, M121)	12-10
12.4.3	M Signal Level Outputs (M400 to M409, M480 to M487)	12-10
12.4.4	One-shot Output (M450, M451, M455, M456)	12-10
12.4.5	Waiting until Response is Given (M460 to M469)	12-10
12.4.6	Signal Output to PLC (M801 to M899)	12-11
12.4.7	Expansion Signal Output (M900 to M999)	12-11
12.4.8	Checking Measurement Instrument Detection Signal (M320, M321)	12-11
<b>12.5</b>	<b>M Function (Additional Axis Control)</b>	<b>12-12</b>
12.5.1	Pallet-related M Codes (M410, M411, M430, M431)	12-12
12.5.2	C-axis Unclamp/Clamp (M444/M430 and M445/M431)	12-12
12.5.3	B-axis Unclamp/Clamp (M440 and M441)	12-13
12.5.4	A-axis Unclamp/Clamp (M442 and M443)	12-13
<b>12.6</b>	<b>M Function (Positioning Check Distance)</b>	<b>12-14</b>
12.6.1	Positioning Finished Check Distance (M270 to M279)	12-14
<b>12.7</b>	<b>M Function (Time Constant Switch)</b>	<b>12-14</b>
12.7.1	Tap Time Constant Selection (M241 to M249, M250, M251, M252 to M254)	12-14
<b>CHAPTER 13</b>	<b>HIGH-ACCURACY MODE</b>	<b>13-1-1</b>
<b>13.1</b>	<b>High-accuracy Mode AIII</b>	<b>13-1-1</b>
1	Outline	13-1-2
1.1	Outline of High-accuracy Mode AIII	13-1-2
1.2	Functions of High-accuracy Mode AIII	13-1-2
2	How to Use	13-1-4
2.1	To Select a Machining Level	13-1-4
2.2	How to Use the Program	13-1-7
2.3	Usable Conditions	13-1-7
2.4	Conditions to be Cancelled	13-1-7
3	Restrictions	13-1-8
3.1	Commandable Functions	13-1-8
3.2	Additional Axis Movement Commands	13-1-9
3.3	Imposition Checked by High-accuracy Mode A Command	13-1-9
3.4	Feedrate when Commanding of High-accuracy Mode A	13-1-9
3.5	Notes in Motions by Smooth Path Offset Function	13-1-9
4	Detailed Explanations and Adjustments of Parameters	13-1-10
4.1	Detailed Explanations	13-1-10
4.2	Explanation About Parameters	13-1-13
4.3	Relationship Between Parameters and Shape Errors	13-1-16
4.4	Adjusting Parameters	13-1-18
<b>13.2</b>	<b>High-accuracy Mode B</b>	<b>13-2-1</b>

1	Outline-----	13-2-2
2	How to Use -----	13-2-3
2.1	To Select a Machining Level-----	13-2-3
2.2	How to Use Programs -----	13-2-3
2.3	Usable Conditions-----	13-2-3
2.4	Conditions to be Cancelled -----	13-2-3
3	Restrictions -----	13-2-4
3.1	Commandable Functions-----	13-2-4
3.2	Temporary Stop of Operation -----	13-2-5
3.3	Single Operation -----	13-2-5
3.4	Cutting Override -----	13-2-5
3.5	Dry Run-----	13-2-5
3.6	Feedrates when Commanding the High-accuracy Mode B -----	13-2-5
3.7	Notes in Smooth Ppath Offset Function-----	13-2-5
3.8	Additional Axis Travel Command -----	13-2-5
4	Explanations on Parameters -----	13-2-6
<b>13.3</b>	<b>Quick Setting for High Accuracy Mode -----</b>	<b>13-3-1</b>
1	Overview -----	13-3-2
2	High Accuracy (Adjustment) Function -----	13-3-2
2.1	Usage -----	13-3-3
2.1.1	Machining mode -----	13-3-3
2.1.2	Machining mode adjustment-----	13-3-4
2.2	Restrictions -----	13-3-7
3	High Accuracy (Accuracy Specification) Function -----	13-3-8
3.1	Usage -----	13-3-8
3.2	Restrictions -----	13-3-9
4	Mode Selection Using Program Command -----	13-3-10
5	Check Current Mode-----	13-3-12

<b>CHAPTER 1</b>	<b>PROGRAM STRUCTURE</b>	<b>1</b>
<b>CHAPTER 2</b>	<b>COORDINATE COMMANDS</b>	<b>2</b>
<b>CHAPTER 3</b>	<b>PREPARATION FUNCTION</b>	<b>3</b>
<b>CHAPTER 4</b>	<b>PREPARATION FUNCTIONS (COMPENSATION FUNCTIONS)</b>	<b>4</b>
<b>CHAPTER 5</b>	<b>PREPARATION FUNCTION (CANNED CYCLE)</b>	<b>5</b>
<b>CHAPTER 6</b>	<b>MACRO</b>	<b>6</b>
<b>CHAPTER 7</b>	<b>AUTOMATIC WORKPIECE MEASUREMENT</b>	<b>7</b>
<b>CHAPTER 8</b>	<b>SUB PROGRAM FUNCTION</b>	<b>8</b>
<b>CHAPTER 9</b>	<b>FEED FUNCTION</b>	<b>9</b>
<b>CHAPTER 10</b>	<b>SPINDLE RELATED FUNCTION (S FUNCTION)</b>	<b>10</b>
<b>CHAPTER 11</b>	<b>TOOL RELATED FUNCTION (T FUNCTION)</b>	<b>11</b>
<b>CHAPTER 12</b>	<b>M FUNCTION</b>	<b>12</b>
<b>CHAPTER 13</b>	<b>HIGH-ACCURACY MODE</b>	<b>13</b>

(This page was intentionally left blank.)

# CHAPTER 1

## PROGRAM STRUCTURE

- 1.1 Program Types and Structure**
- 1.2 Block Structure**
- 1.3 Word Structure**
- 1.4 Number Commands**
- 1.5 Sequence Numbers**
- 1.6 Optional Block Skip**
- 1.7 Control Out-In Function**

# 1.1 Program Types and Structure

The programs are divided into two types: main programs and sub programs.

## 1. Main programs

This program is used for machining one workpiece. Sub programs are called in the middle of the main program in order to create a program more efficiently.

Lastly, M02 (or M30) is used to end the program.

Main program

N0001 G92X100;
N0002 G00Z30;
:
:
:
:
M02;

## 2. Sub programs

Sub programs are used and called from the main program or other sub programs.

Lastly, M99 is used to end the program.

Sub program

N0010 G91X10
:
:
:
:
M99;

# 1.2 Block Structure

The programs are made up of a number of commands. One command unit is referred to as a block.

A block is made up of one or multiple words.

One block is delimited by an end of block (EOB) code.

(The explanation in this manual uses a semicolon “;” for the end of block code.)

...	;	N0001G92X100	;	...	;	M02	;
	←	Block	→			←Block→	

**(Note 1)** The end of block code in ISO code is “LF” 0A (hexadecimal).

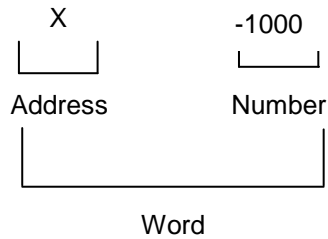
In EIA code, it is “CR” 80 (hexadecimal).

**(Note 2)** The maximum number of characters in 1 block is 128.



## 1.3 Word Structure

A word is made up of an address followed by a number of multiple digits.  
(There may be a positive (+) or negative (-) sign that is used before the number.)



**(Note 1)** The address is one character that is a letter between A and Z.

**(Note 2)** The address "0" is only used for comments.

## 1.4 Number Commands

### 1. Decimal point command

There are two systems (Base/Least) when issuing a number command.

- Coordinate value and travel amount commands

Select the <Program unit> in User parameter (Switch 1) for the coordinate values and travel amount.

In addition, the minimum unit setting (Type 1 (micron) and Type 2 (submicron)) can be selected using an optional function.

<Program unit>: 0 (Base)

Specifying program	Command axis	Actual amount (mm)	Actual amount (inches)
1	Feed axis	1 mm	1 inch
	Rotation axis	1 deg	1 deg
1.	Feed axis	1 mm	1 inch
	Rotation axis	1 deg	1 deg

<Program unit>: 1 (Least)

Specifying program	Command axis	Actual amount (mm)	Actual amount (inches)
1	Feed axis	0.001 mm	0.0001 inch
	Rotation axis	0.001 deg	0.001 deg
1.	Feed axis	1 mm	1 inch
	Rotation axis	1 deg	1 deg

However, if Type 2 (submicron) and <Program unit>: 1 (Least) are used for the minimum unit settings, the following applies.

<Program unit>: 1 (Least)

Specifying program	Command axis	Actual amount (mm)	Actual amount (inches)
1	Feed axis	0.0001 mm	0.00001 inch
	Rotation axis	0.0001 deg	0.0001 deg
1.	Feed axis	1 mm	1 inch
	Rotation axis	1 deg	1 deg

- Feedrate command  
Use <Program feed rate unit> in User parameter (Switch 1) to select the feedrate.

<Program feed rate unit>: 0 (Basic)

Specifying program	Command axis	Actual amount (mm)	Actual amount (inches)
1	Feed axis	1 mm/min 1 mm/rev 1(1/min)	1 inch/min 1 inch/rev 1(1/min)
	Rotation axis	1 deg/min 1 deg/rev 1(1/min)	1 deg/min 1 deg/rev 1(1/min)
1.	Feed axis	1 mm/min 1 mm/rev 1(1/min)	1 inch/min 1 inch/rev 1(1/min)
	Rotation axis	1 deg/min 1 deg/rev 1(1/min)	1 deg/min 1 deg/rev 1(1/min)

<Program feed rate unit>: 1 (Minimum)

Specifying program	Command axis	Actual amount (mm)	Actual amount (inches)
1	Feed axis	0.01 mm/min 0.0001 mm/rev 1(1/min)	0.001 inch/min 0.00001 inch/rev 1(1/min)
	Rotation axis	0.01 deg/min 0.0001 deg/rev 1(1/min)	0.001 deg/min 0.00001 deg/rev 1(1/min)
1.	Feed axis	1 mm/min 1 mm/rev 1(1/min)	1 inch/min 1 inch/rev 1(1/min)
	Rotation axis	1 deg/min 1 deg/rev 1(1/min)	1 deg/min 1 deg/rev 1(1/min)

However, if Type 2 (submicron) and <Program feed rate unit>: 1 (Minimum) are used for the minimum unit settings, the following applies.

<Program feed rate unit>: 1 (Minimum)

Specifying program	Command axis	Actual amount (mm)	Actual amount (inches)
1	Feed axis	0.001 mm/min 0.00001 mm/rev 1(1/min)	0.0001 inch/min 0.000001 inch/rev 1(1/min)
	Rotation axis	0.001 deg/min 0.00001 deg/rev 1(1/min)	0.0001 deg/min 0.000001 deg/rev 1(1/min)
1.	Feed axis	1 mm/min 1 mm/rev 1(1/min)	1 inch/min 1 inch/rev 1(1/min)
	Rotation axis	1 deg/min 1 deg/rev 1(1/min)	1 deg/min 1 deg/rev 1(1/min)

2. Range for address number commands  
The range for commands varies depending on the significance or use of the address.  
Anything below the minimum unit setting in a command is rounded off.

## 1.5 Sequence Numbers

A sequence number (1 to 999999) can be attached to address N for each block unit.

Command format N\*\*\*\*\*;

1. Specify a number between 0 and 9 after N.
2. A sequence number can be specified using a maximum of 6 digits.

**(Note 1)** Do not use “N0”.

**(Note 2)** Specify the header for the block.  
N0100 G90X100;

If an optional block skip (“/”) is used at the header for the block, a command can be issued before or after that.

N0100/ G90X100;

or

/N0100 G90X100;

**(Note 3)** The sequence numbers can be in any given order and do not necessarily have to be sequential.

**(Note 4)** The sequence number is recognized as a numerical value.  
Therefore, 0001, 001, 01, and 1 are treated as the same number.

## 1.6 Optional Block Skip

This method ignores the block information specified during automatic operation. A slash (“/”) is used at the header of the block.

Turn ON the [B.SKIP] key on the operation panel to ignore the block information during automatic operation.

Turn OFF the [B.SKIP] key to enable the block information.

The block skip ignores the entire block.

.....;                      /N0100G00X100.....;                      N0101.....;  
    | ←This range is ignored→                      |

**(Note 1)** If a slash “/” is not inserted at the header of the block, an alarm is triggered.

Note that it can be inserted immediately after the sequence number.

**(Note 2)** When the [B.SKIP] key is turned ON in single block mode during automatic operation, it does not stop at the block with a slash “/” but the next block thereafter.

## 1.7 Control Out-In Function

A comment can be inserted to make the program easier to read.

A comment must be enclosed with parentheses “( )” to distinguish it from the operation information.

Control out                      (                      )                      Control in  
 code                                      Comment                                      code

Ex: N1000 G00X200 (PRO-1);

**(Note)** Make sure that the comment including the control out and control in codes fit inside 1 block.

(This page was intentionally left blank.)

## CHAPTER 2

# COORDINATE COMMANDS

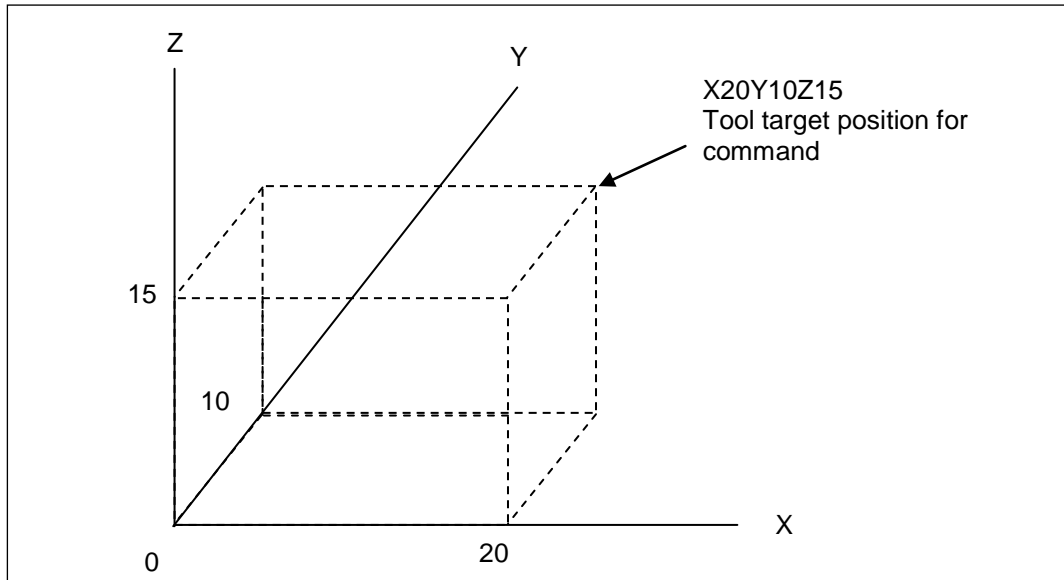
- 2.1     Coordinate System and Coordinate Commands**
- 2.2     Machine Zero Point and Machine Coordinates**
- 2.3     Workpiece Coordinates**

## 2.1 Coordinate System and Coordinate Commands

Coordinate commands in a coordinate system are required to move the tool to the target position. There are two types of coordinate systems shown below.

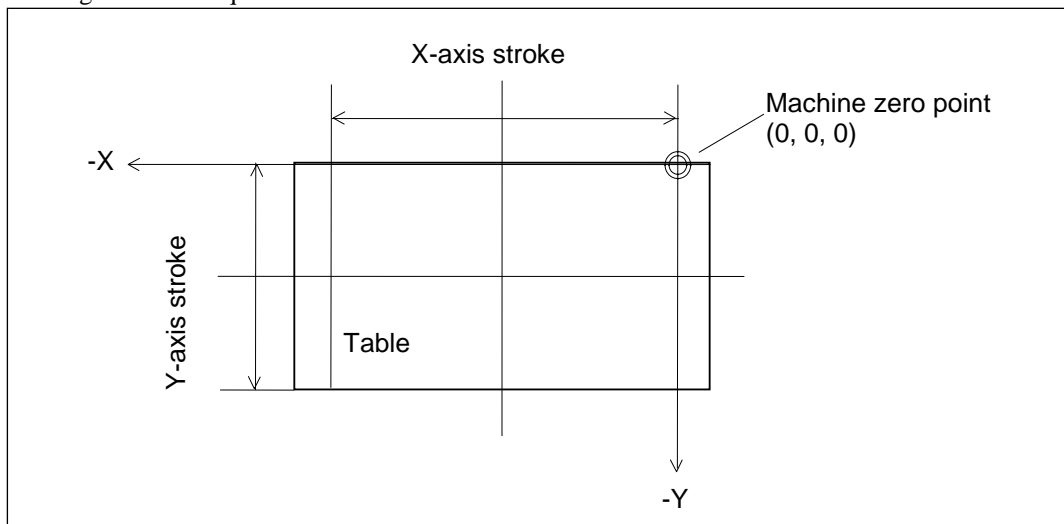
1. Machine coordinates
2. Workpiece coordinates

The coordinates are expressed in components using the program axes (X-, Y- and Z-axes on this machine).



## 2.2 Machine Zero Point and Machine Coordinates

1. Machine zero point  
The machine zero point refers to the origin that acts as a reference point on the machine.
2. Machine coordinates  
The machine coordinates refer to the coordinate system that uses the machine zero point as the origin and is unique to each machine.



## 2.3 Workpiece Coordinates

The workpiece coordinates refer to the coordinate system that is used to specify machining on each workpiece.

Select 1 set of coordinates that is preset in the data bank for machining.

Those coordinates are used to issue commands in the program thereafter.

However, each set of coordinates is configured following the amount that is offset between the machine zero point and the workpiece zero point.

(This page was intentionally left blank.)



## CHAPTER 3

## 3

# PREPARATION FUNCTION

- 3.1 Outline of G Code
- 3.2 Positioning (G00, G60)
- 3.3 Cutting (G01 to 03, G12/13, G102/103, G202/203)
- 3.4 Dwell (G04)
- 3.5 Exact Stop Check (G09, G61, G64)
- 3.6 Programmable Data Input (G10)
- 3.7 Coordinate System (G17 to 19, G52 to 59, G54.1, G92)
- 3.8 Soft Limit
- 3.9 Reference Position (G28 to 30)
- 3.10 Skip Function (G31, G131/G132)
- 3.11 Scaling (G50/G51)
- 3.12 Programmable Mirror Image (G50.1/51.1)
- 3.13 Rotational Transformation Function (G68/69, G168)
- 3.14 Absolute Command and Incremental Command (G90/91)
- 3.15 Change of Tap Twisting Direction (G133/134)
- 3.16 G code Priority
- 3.17 Programmable Data Input (High Accuracy) (G210)
- 3.18 Thread Cutting
- 3.19 Lathe Machining Infeed Direction

## 3.1 Outline of G Code

Within 3-digit number following the address G determines the meaning of the command of the block concerned.

The G codes are divided into the following two types.

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

List of G code

The G codes with \* mark indicates the modal status when the power is turned ON.

Group	G code	Description	Modal/One-shot
	G00*	positioning	Modal
	G01	Linear interpolation	
	G02	Circular/ helical interpolation (CW)	
	G03	Circular/ helical interpolation (CCW)	
	G02.2	Involute interpolation (CW)	
	G03.2	Involute interpolation (CCW)	
	G102	XZ Circular interpolation (CW)	
	G103	XZ Circular interpolation (CCW)	
	G202	YZ Circular interpolation (CW)	
	G203	YZ Circular interpolation (CCW)	
	G33	Thread cutting	
	G392	Thread cutting cycle	
	G04	Dwell	One-shot
	G09	Exact stop check	One-shot
	G10	Programmable data input	One-shot
	G12	Circular cutting CCW	One-shot
	G13	Circular cutting CCW	One-shot
	G17*	XY plane selection	Modal
	G18	ZX plane selection	
	G19	YZ plane selection	
	G22*	Programmable stroke limit ON	Modal
	G23	Programmable stroke limit cancel	
	G28	Return to the reference point	One-shot
	G29	Return from the reference point	
	G30	Return to the 2 <sup>nd</sup> to 6 <sup>th</sup> reference point	
	G31	Skip function	One-shot
	G36	Coordinate calculation function	One-shot
	G37	Coordinate calculation function (Line-angle)	
	G38	Coordinate calculation function (Line-X, Y)	
	G39	Coordinate calculation function (grid)	
	G40*	Tool diameter / nose R compensation cancel	Modal
	G41	Tool dia. offset left	
	G42	Tool dia. offset right	
	G141	Nose R left compensation	
	G142	Nose R right compensation	
	G43	Tool length offset +	Modal
	G44	Tool length offset -	
	G143	Tool position offset +	
	G144	Tool position offset -	
	G49*	Tool length/position offset cancel	
	G50*	Scaling cancel	Modal
	G51	Scaling	
	G50.1*	Mirror image cancel	Modal
	G51.1	Mirror image	
	G52	Local coordinate system	One-shot
	G53	Machine coordinate system selection	
	G53.1	Feature coordinate index	
	G54*	Working coordinate system selection 1	Modal
	G55	Working coordinate system selection 2	

Group	G code	Description	Modal/One-shot
	G56	Working coordinate system selection 3	
	G57	Working coordinate system selection 4	
	G58	Working coordinate system selection 5	
	G59	Working coordinate system selection 6	
	G54.1	Extended working coordinate system selection	
	G54.2P0*	Rotary fixture offset cancel	Modal
	G54.2Pn	Rotary fixture offset (n: 1 to 8)	
	G60	Single direction positioning	One-shot
	G61	Exact stop mode	Modal
	G64*	Cutting mode	
	G65	Macro call	One-shot
	G66	Macro modal call	Modal
	G67*	Cancel macro modal call	
	G68	Coordinate rotation function	Modal
	G69*	Coordinate rotation function cancel Feature coordinate manufacturing mode cancel	
	G168	Coordinate rotation using measured results	
	G68.2	Feature coordinate setting	
	G73	Canned cycle (High-speed peck drilling cycle)	Modal
	G74	Canned cycle (Reverse tapping cycle)	
	G76	Canned cycle fine boring cycle	
	G77	Canned cycle tapping cycle (synchro mode)	
	G78	Canned cycle (Reverse tapping cycle) (synchro mode)	
	G80*	Canned cycle (Cancel mode)	
	G81	Canned cycle (Drill, spot drilling cycle)	
	G82	Canned cycle (Drill, spot drilling cycle)	
	G83	Canned cycle (Peck drilling cycle)	
	G84	Canned cycle (Tapping cycle)	
	G85	Canned cycle (Boring cycle)	
	G86	Canned cycle (Boring cycle)	
	G87	Canned cycle (Back balling cycle)	
	G89	Canned cycle (Boring cycle)	
	G177	Canned cycle (End mill tapping cycle)	
	G178	Canned cycle (End mill tapping cycle)	
	G181	Canned cycle (Double drilling cycle)	
	G182	Canned cycle (Double drilling cycle)	
	G185	Canned cycle (Double boring cycle)	
	G186	Canned cycle (Double boring cycle)	
	G189	Canned cycle (Double boring cycle)	
	G277	Canned cycle (Deep hole tapping cycle) (synchro mode)	
	G278	Canned cycle (Counter deep hole tapping cycle) (synchro mode)	
	G173	Canned cycle (High-speed peck drilling cycle)	One-shot
	G183	Canned cycle (Peck drilling cycle)	One-shot
	G100	Non-stop automatic tool change	One-shot
	G90*	Absolute command	Modal
	G91	Incremental command	
	G92	Workpiece coordinate system setting Spindle speed clamp	One-shot
	G93	Inverse time feed	Modal
	G94*	Feed rate per minute	
	G95	Feed rate per revolution	
	G96	Constant peripheral speed control	Modal
	G97	Constant peripheral speed control cancel	
	G98*	Return to the initial point level	Modal
	G99	Return to the R point level	
	G120	Positioning to the measuring point	One-shot

Group	G code	Description	Modal/One-shot
	G121	Automatic measurement Corner (Boss)	One-shot
	G122	Automatic measurement Parallel (Groove)	
	G123	Automatic measurement Parallel (Boss)	
	G124	Automatic measurement Circle center (Hole, 3 points)	
	G125	Automatic measurement circle center (Boss, 3 points)	
	G126	Automatic measurement Circle center (Hole, 4 points)	
	G127	Automatic measurement Circle center (Boss, 4 points)	
	G128	Automatic measurement Z axis height	
	G129	Automatic measurement Corner (Groove)	
	G131	Measurement feed	One-shot
	G132	Measurement feed	
	G133	Changeover of tap twisting direction (CW)	One-shot
	G134	Changeover of tap twisting direction (CW)	
	G210	Programmable data input (high accuracy)	One-shot
	G321	Lathe machining infeed direction on X-axis	Modal (Note)
	G322	Lathe machining infeed direction on Y-axis	
	G323	Lathe machining infeed direction on Z-axis	
	G376	Thread angle in complex thread cutting cycle	One-shot

**(Note)** Specify the modal during power startup in the user parameter (switch 1) <Lathe machining infeed direction when power is turned ON>.

## 3.2 Positioning (G00, G60)

### 3.2.1 Positioning (G00)

Command format **G00 X\_ Y\_ Z\_ A\_ B\_ C\_;**

If an additional axis command is issued when there is no additional axis option, an alarm is triggered.

When carrying out G00 based positioning, it first performs an in-position check (Note 1) and then proceeds to the next block.

One of the following options can be selected for the tool path in the user parameter <Positioning method>.

<0: Non-linear interpolation type positioning>

Positioning is carried out on each axis independently using rapid feedrate on each axis. The tool path is not a straight line.

<1: Linear interpolation type - Positioning method 1>

Positioning is carried out so the tool path is a straight line and is completed in the shortest time period at a speed that does not exceed the rapid feedrate on each axis. However, the alarm <<Positioning command error for linear interpolation type>> is triggered when there are simultaneous commands issued for 2 or more additional axes.

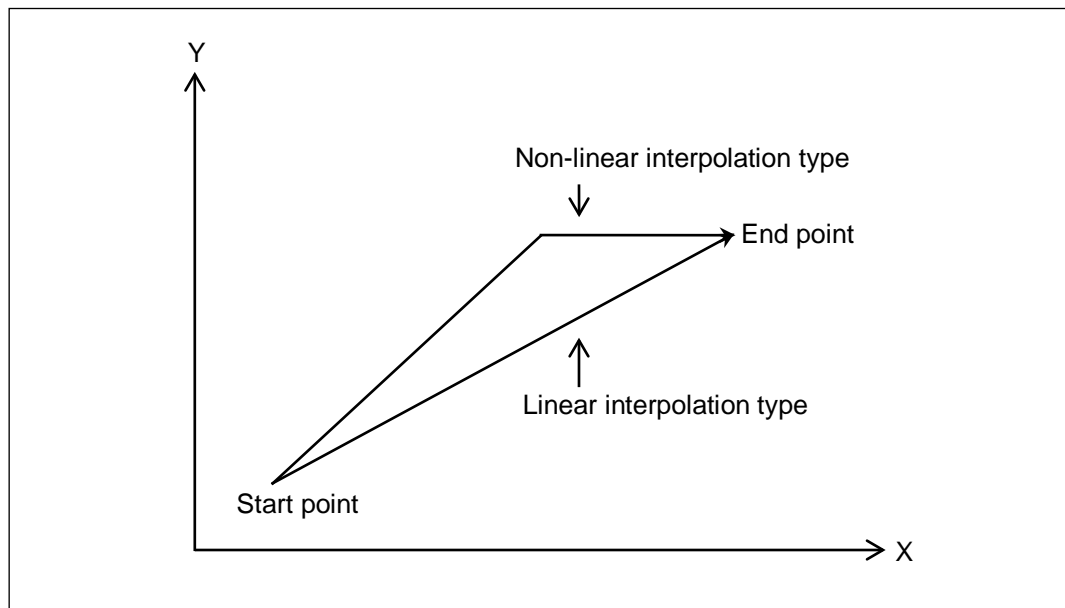
Be careful of collisions because if the margin of error on the path for positioning (including the additional axis) is greater than the positioning only on the linear axis, then the positioning changes depending on the feedrate and the load.

<2: Linear interpolation type - Positioning method 2>

Positioning is carried out so the tool path is a straight line and is completed in the shortest time period at a speed that does not exceed the rapid feedrate on each axis. However, the positioning operation on the additional axis is a non-linear interpolation type.

If the user parameter <Positioning method> is changed, use idling, for example, to check the tool path and cycle time before operating, because the tool path during rapid feed or the positioning time changes.

In addition, be careful because the margin of error due to thermal distortion also changes, adversely affecting the machining accuracy.

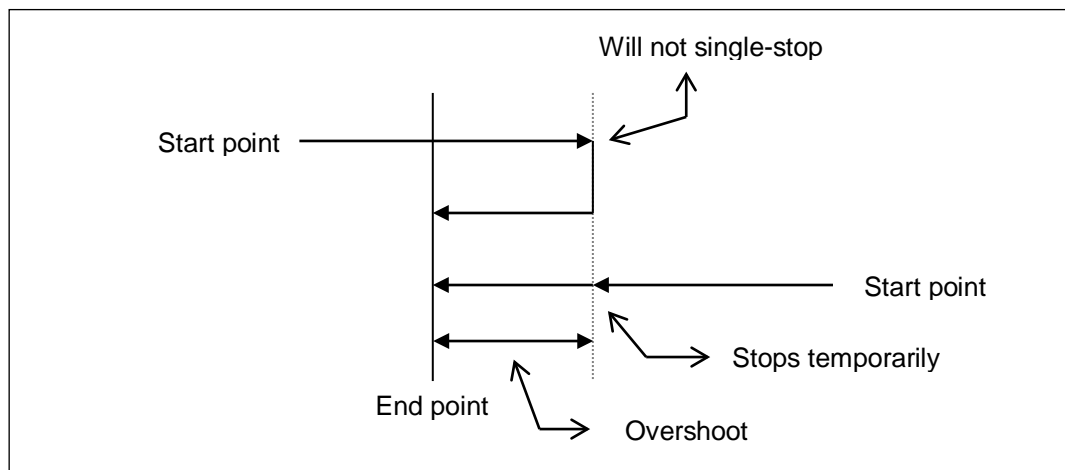


- (Note 1) The in-position check refers to checking whether or not the position detected on the machine has reached a range within the target position (end point).  
However, when there are successive Z-axis only operations, such as G0Z → G0Z, the in-position check may not be carried out.
- (Note 2) The in-position range used for the in-position check varies according to the <Machine parameter> that is used for the command that follows.  
When the command sequence is G0→G0, the <Positioning end check distance> applies, and when the command sequence is a cutting command such as G0→G1/G2, the <In-position width> applies. However, when the command sequence is an operation in the same direction such as G0Z\_ → G1Z\_, the <Positioning end check distance> applies.
- (Note 3) The rapid feedrate is set for each axis in the machine parameter. As a result, an F command for the feedrate cannot be carried out.
- (Note 4) The positioning operation during a tool change (G100 and M06) is a non-linear interpolation type regardless of the type that is selected in the user parameter <Positioning method>.

### 3.2.2 Single Direction Positioning Function (G60)

Command format **G60 X\_ Y\_ Z\_ A\_ B\_ C\_;**

X,Y,Z,A,B,C: Command value of the axis for which single direction positioning is performed.  
Coordinate of end point for G90 and travel amount for G91



When the above command is executed, the axis moves from the end point for the preset travel amount, and then moves to the end point.

G60 is a one shot command and the axis travel path is the same as that for G00.

Overshoot is set by the <single direction positioning excess travel amount> of <user parameter>.

- (Note 1) Z axis will not perform single direction positioning in canned cycle operation. This also applies to X and Y axes traveling a shift amount in G76 and G87 cycles.
- (Note 2) Single direction positioning is not performed for any axis that does not have the travel amount set for the parameter.
- (Note 3) Single direction positioning is performed even when 0 is specified for the travel amount.
- (Note 4) The alarm <<Compensating diameter>> is triggered when the G60 command is issued during cutter compensation and nose R compensation operations.
- (Note 5) Travel to the end point from a position that is passed the end point and travel from a temporary stop position to the end point are a non-linear interpolation type regardless of the type that is selected in the user parameter <Positioning method>.

### 3.2.3 Precautions for Programming Involving Use of Rotation Axis (Index Table)

Be sure to insert indexing commands for rotation axes A and B before cutting commands in the program when using an index table.

When opening a door with the door interlock function enabled, the axis may become misaligned due to the load of the workpiece, etc., because the servo for the rotation axis is turned OFF. Thereafter, if the shift amount when closing the door is smaller than the machine parameter <Return angle with servo controller ON>, then it returns to the position prior to turning OFF the servo. However, the alarm <<Return distance too long>> is triggered when the shift amount is larger than the <Return angle with servo controller ON>. If you continue in this state, the workpiece will not be machined in correct position.

### 3.3 Cutting (G01 to 03, G12/13, G102/103, G202/203)

#### 3.3.1 Linear Interpolation (G01)

Linear interpolation moves a tool linearly from the current position to the target position at the specified feed rate.

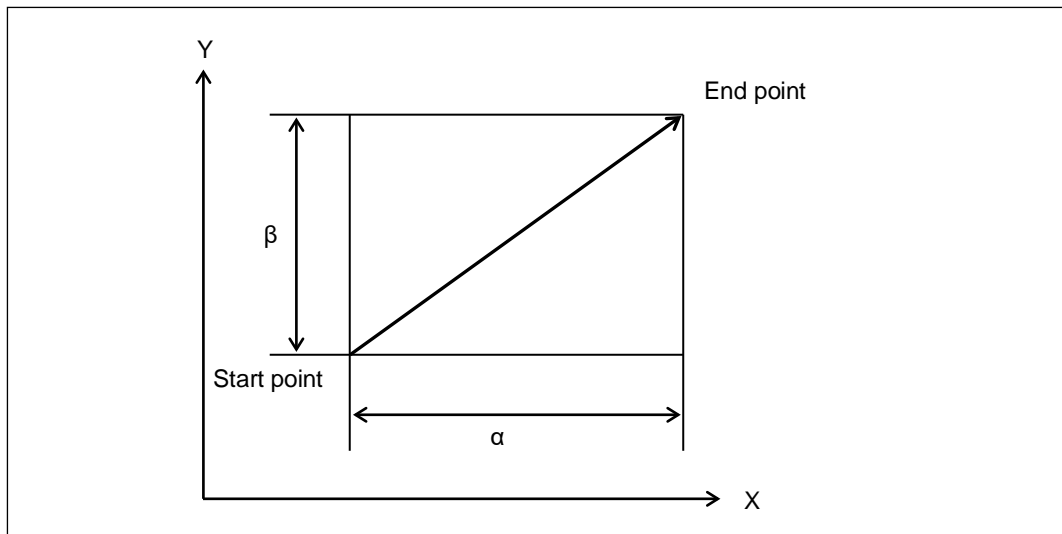
Command format

```
G01 X_ Y_ Z_ A_ F_;
G01 X_ Y_ Z_ B_ F_;
G01 X_ Y_ Z_ C_ F_;
```

Axis command is effective for up to 3 linear axes plus one additional axis. The alarm <<Invalid Command>> occurs when you command 2 additional axes simultaneously.

The alarm <<No \*axis Option>> occurs when you command additional axes in their absence. The feedrate is set by address F. Once the feed rate is commanded, it is effective until another value is specified.

X, Y, and Z axes, when commanded, are recognized with the feed rate of mm/min (mm/rev) and additional axis °/min (°/rev).



Feed rate of each axis is calculated in the equation below.

When "G01 G91 Xα Yβ Zγ Ff;" is programmed:

Feed rate along X axis: 
$$F_x = \frac{\alpha}{L} \cdot f$$

Feed rate along Y axis: 
$$F_y = \frac{\beta}{L} \cdot f$$

Feed rate along Z axis: 
$$F_z = \frac{\gamma}{L} \cdot f$$

$$(L = \sqrt{\alpha^2 + \beta^2 + \gamma^2})$$



The example below shows linear interpolation of linear axis and rotation axis.

When "G01 G91 X $\alpha$  Y $\beta$  Z $\gamma$  B $\delta$  Ff" is programmed

$$\text{Time for B axis distribution} \quad T_b = \frac{L}{f}$$

$$\text{Feed rate along B axis} \quad F_b = \frac{\delta}{T_b}$$

$$\text{Feed rate along X axis} \quad F_x = \frac{\alpha}{L} \cdot f$$

$$\text{Feed rate along Y axis} \quad F_y = \frac{\beta}{L} \cdot f$$

$$\text{Feed rate along Z axis} \quad F_z = \frac{\gamma}{L} \cdot f$$

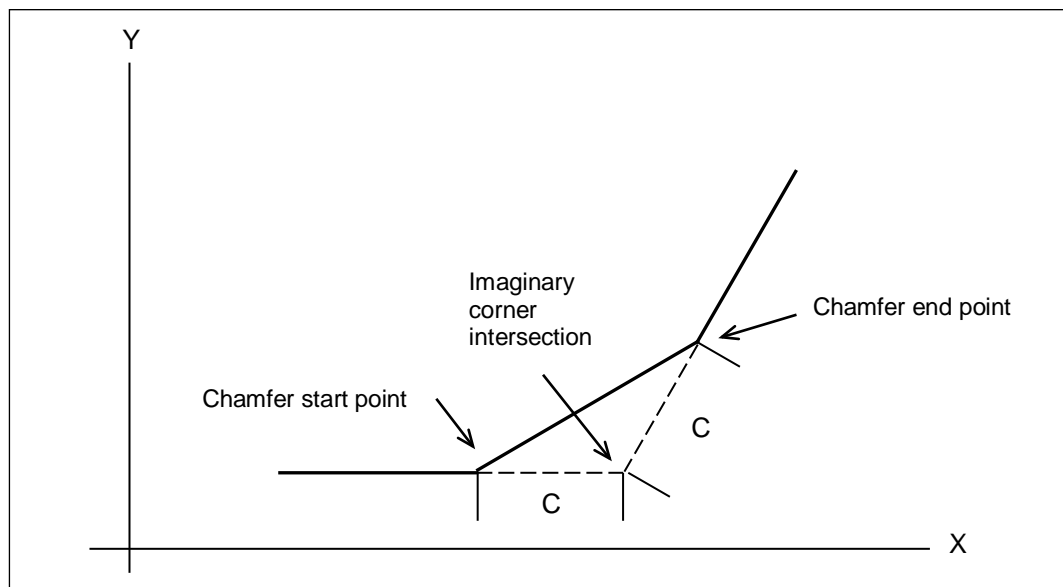
$$(L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \delta^2})$$

### 3.3.1.1 Chamfering to Desired Angle and Cornering R

Chamfering to the desired angle or rounding can be performed between interpolation commands.

Chamfering Command format **G01 X\_ Y\_, C\_;**

C : Distance from virtual corner to the chamfer start point and end point.  
This can be commanded only for the selected plane surface.



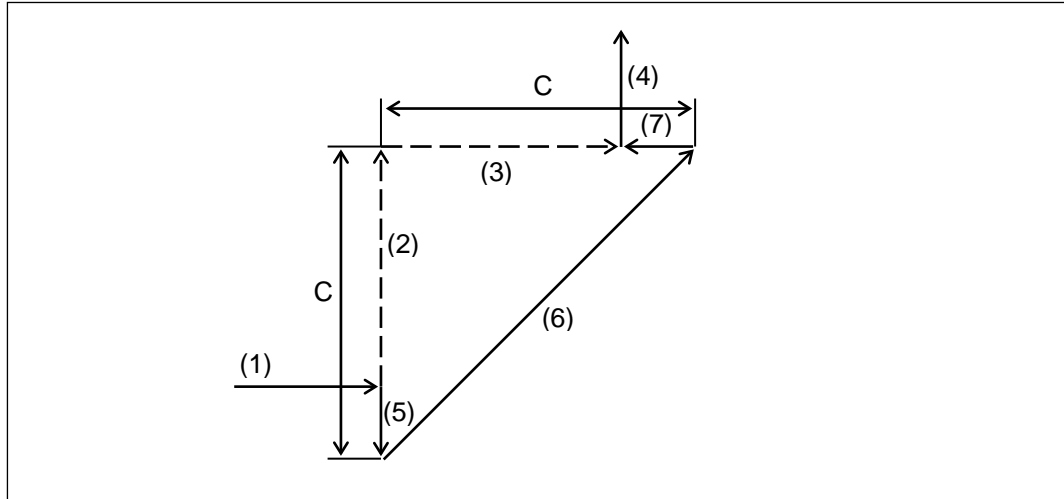
(Note 1) The corner chamfering command block and subsequent block must contain the interpolation command (G01-G03). The alarm <<Specified G Code Cannot Be Used>> occurs when interpolation command is missing or the next block is not a travel command.

(Note 2) The inserted block belongs to the corner chamfer command block. Even if the feed rate of the next block is different than that of the corner chamfer command block, the inserted block moves at the feed rate of the corner chamfer command block. In single block operation, the system does not stop before the inserted block but stops after it.

**(Note 3)** The cutter compensation / nose R compensation is applied to the shape after the corner chamfering is performed.

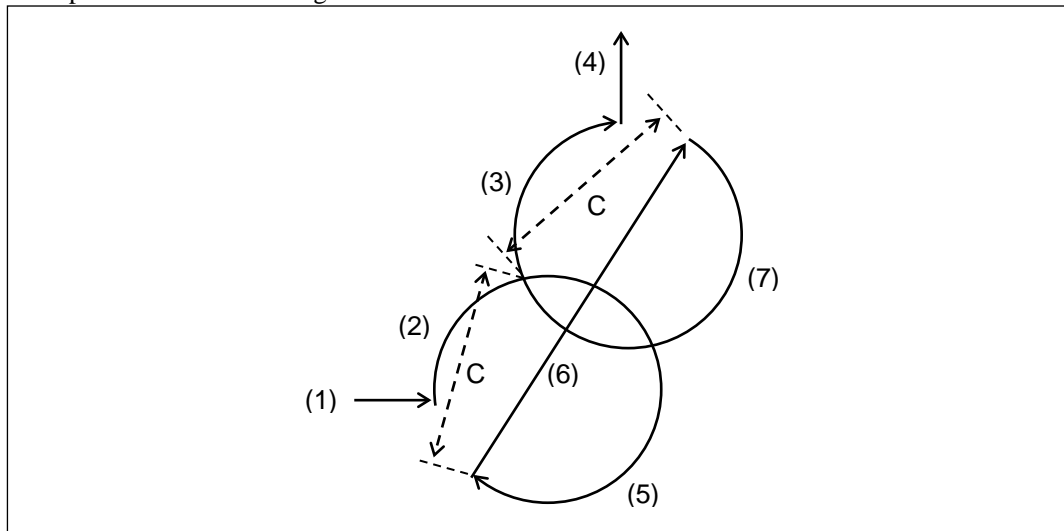
**(Note 4)** When the commanded chamfering size is greater than travel distance of the chamfer command block or the next block, the block is extended as required to define the chamfer start or end point.

Example.1: Linear cutting



When set the programmed path to (1)→(2)→(3)→(4) and the block C as (2), operate to (1)→(5)→(6)→(7)→(4).

Example.2: Circular cutting

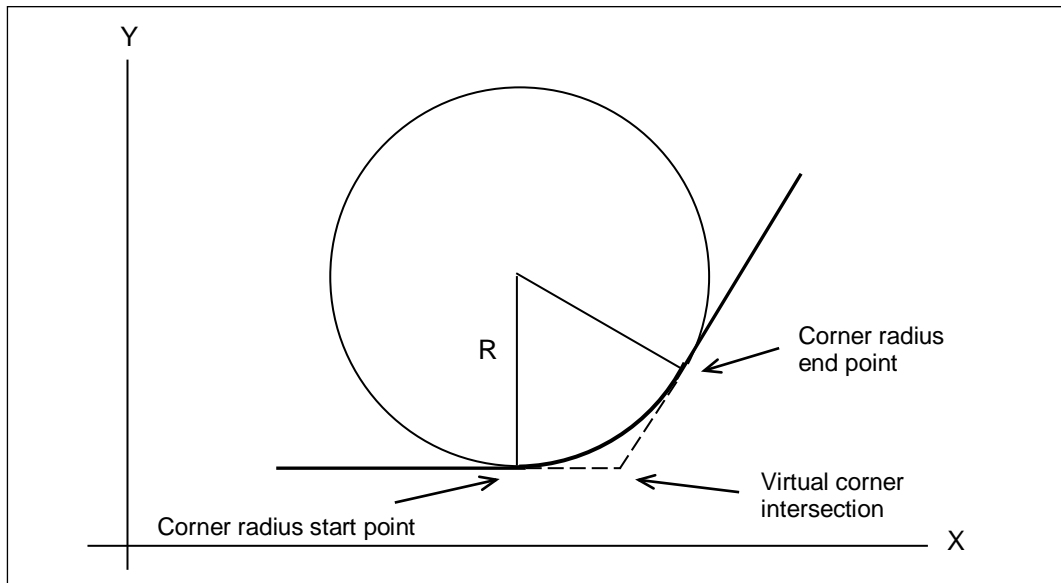


When the programmed path is (1)→(2)→(3)→(4) and the block (2) is C-specified, the machine operates through (1)→(5)→(6)→(7)→(4).

Cornering R Command format

**G01 X\_ Y\_, R\_;**

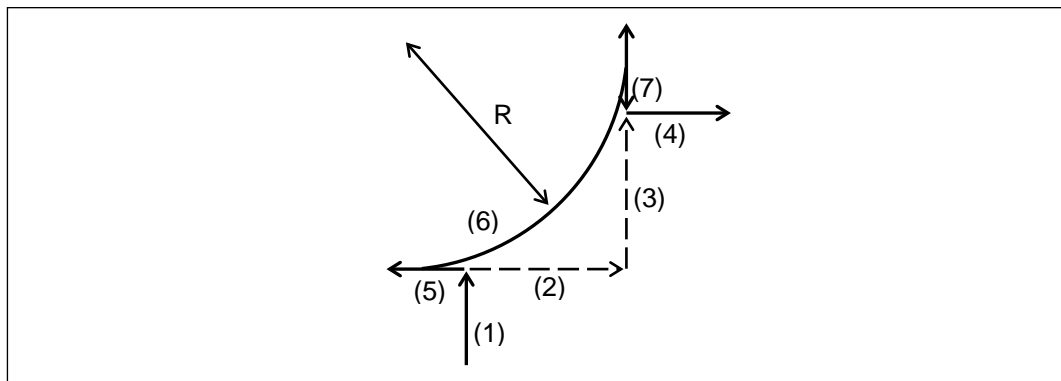
R : Radius of cornering  
This can be commanded only for the selected plane surface.



3

- (Note 1) Corner rounding command block and the next block must be a corner interpolation command (G01 to G03). The alarm <<Specified G Code Cannot Be Used>> occurs if they are not an interpolation command or the next corner is unable to execute travel command.
- (Note 2) The inserted block belongs to the corner rounding command block. Even if the feed rate of the next block is different than that of the corner rounding command block, the inserted block moves at the feed rate of the corner rounding command block. In single block operation, the system does not stop before the inserted block but stops after it.
- (Note 3) The cutter compensation / nose R compensation is applied to the shape after the corner R operation is performed.
- (Note 4) When the commanded radius is greater than corner rounding command block or the next command block, the block is extended as required to define the corner rounding start or end point
- (Note 5) After the feature coordinate setting, the corner R command cannot be issued before the feature coordinate index.

Example.1: Linear cutting



When the programmed path is (1)→(2)→(3)→(4) and the block (2) is R-specified, the machine operates through (1)→(5)→(6)→(7)→(4).

### 3.3.2 Circular Interpolation / Helical Thread Cutting Interpolation

#### 3.3.2.1 Circular Interpolation (G02, G03)

Circular interpolation moves a tool along a circular arc from the current position to the end point at the specified feed rate.

Command  
format

**XY plane:**  
**G17 G02 X\_ Y\_  $\left( \begin{smallmatrix} I_ & J_ \\ & R_ \end{smallmatrix} \right) F_;$**

**G17 G03 X\_ Y\_  $\left( \begin{smallmatrix} I_ & J_ \\ & R_ \end{smallmatrix} \right) F_;$**

**ZX plane:**  
**G18 G02 Z\_ X\_  $\left( \begin{smallmatrix} K_ & I_ \\ & R_ \end{smallmatrix} \right) F_;$**

**G18 G03 Z\_ X\_  $\left( \begin{smallmatrix} K_ & I_ \\ & R_ \end{smallmatrix} \right) F_;$**

**YZ plane:**  
**G19 G02 Y\_ Z\_  $\left( \begin{smallmatrix} J_ & K_ \\ & R_ \end{smallmatrix} \right) F_;$**

**G19 G03 Y\_ Z\_  $\left( \begin{smallmatrix} J_ & K_ \\ & R_ \end{smallmatrix} \right) F_;$**

The commands are given in the following format:

Rotation direction		G02	Clockwise (CW).
		G03	Counterclockwise (CCW).
End point	G90 mode	X,Y,Z	End point in the working coordinate system.
	G91 mode	X	Distance from the start point to the end point in the X direction.
		Y	Distance from the start point to the end point in the Y direction.
		Z	Distance from the start point to the end point in the Z direction.
Distance between start point and arc center		I	Distance from the start point to the center of arc in the X direction.
		J	Distance from the start point to the center of arc in the Y direction.
		K	Distance from the start point to the center of arc in the Z direction.
Arc radiusArc radius		R	Arc radiusArc radius
Feedrate		F	Feedrate in the tangential direction of circular arc.

Clockwise and counterclockwise are the rotation direction viewed from the positive direction to the negative direction on the Z axis of the X-Y plane.

### 3.3.2.2 XZ Circular Interpolation (G102, G103)

Circular interpolation moves a tool along a circular arc from the current position to the end point at the specified feed rate.

Command  
format

$\begin{pmatrix} \text{G102} \\ \text{G103} \end{pmatrix}$	$\text{X\_Z\_}$	$\begin{pmatrix} \text{I\_K\_} \\ \text{R\_} \end{pmatrix}$	$\text{F\_};$
--	-----------------	---	---------------

The commands are given in the following format:

Rotation direction		G102	Clockwise (CW).
		G103	Counterclockwise (CCW).
End point	G90 mode	X,Z	End point in the working coordinate system.
	G91 mode	X	Distance from the start point to the end point in the X direction.
		Z	Distance from the start point to the end point in the Z direction.
Distance between start point and arc center		I	Distance from the start point to the center of arc in the X direction.
		K	Distance from the start point to the center of arc in the Z direction.
Arc radiusArc radius		R	Arc radiusArc radius
Feedrate		F	Feedrate in the tangential direction of circular arc.

Clockwise and counterclockwise are the rotation direction viewed from the positive direction to the negative direction on the Y axis of the X-Z plane.

**(Note)** When X- and Y- arcs differ, and when any of the following commands are issued: diameter compensation command (G41 and G42), nose R compensation command (G141 and G142) and rotational transformation command (G68 and G168), the respective alarms <<Compensating diameter>> and <<During rotational transformation>> are triggered, and operation stops.  
In addition, the alarm <<Feature coordinate manufacturing mode engaged>> is triggered while in feature coordinate manufacturing mode (G68.2 modal in progress).

### 3.3.2.3 YZ Circular Interpolation (G202, G203)

Circular interpolation moves a tool along a circular arc from the current position to the end point at the specified feed rate.

Command Format	$\begin{pmatrix} \text{G202} \\ \text{G203} \end{pmatrix} \quad Y\_Z\_ \quad \begin{pmatrix} J\_K\_ \\ R\_ \end{pmatrix} \quad F\_;$
-------------------	--

The commands are given in the following format:

Rotation direction		G202	Clockwise (CW).
		G203	Counterclockwise (CCW).
End point	G90 mode	Y,Z	End point in the working coordinate system.
	G91 mode	Y	Distance from the start point to the end point in the Y direction.
		Z	Distance from the start point to the end point in the Z direction.
Distance between start point and arc center		J	Distance from the start point to the center of arc in the Y direction.
		K	Distance from the start point to the center of arc in the Z direction.
Arc radiusArc radius		R	Arc radiusArc radius
Feedrate		F	Feedrate in the tangential direction of circular arc.

Clockwise and counterclockwise are the rotation direction viewed from the positive direction to the negative direction on the X axis of the Y-Z plane.

**(Note)** When X- and Y- arcs differ, and when any of the following commands are issued: diameter compensation command (G41 and G42), nose R compensation command (G141 and G142) and rotational transformation command (G68 and G168), the respective alarms <<Compensating diameter>> and <<During rotational transformation>> are triggered, and operation stops.  
In addition, the alarm <<Feature coordinate manufacturing mode engaged>> is triggered while in feature coordinate manufacturing mode (G68.2 modal in progress).

### 3.3.2.4 Precautions for Circular Interpolation

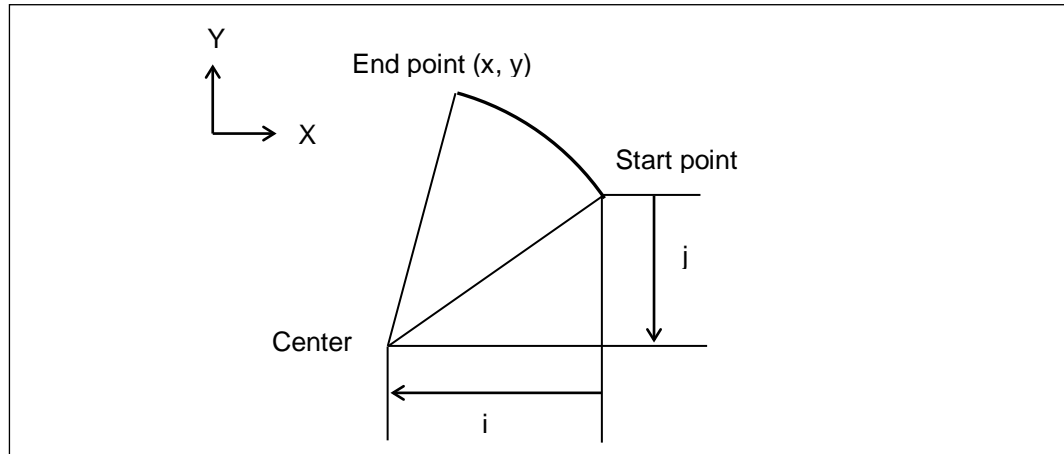
End point of an arc is set by an absolute or incremental value by G90 and G91, respectively.

Incremental value is the distance from start to end point of the arc.

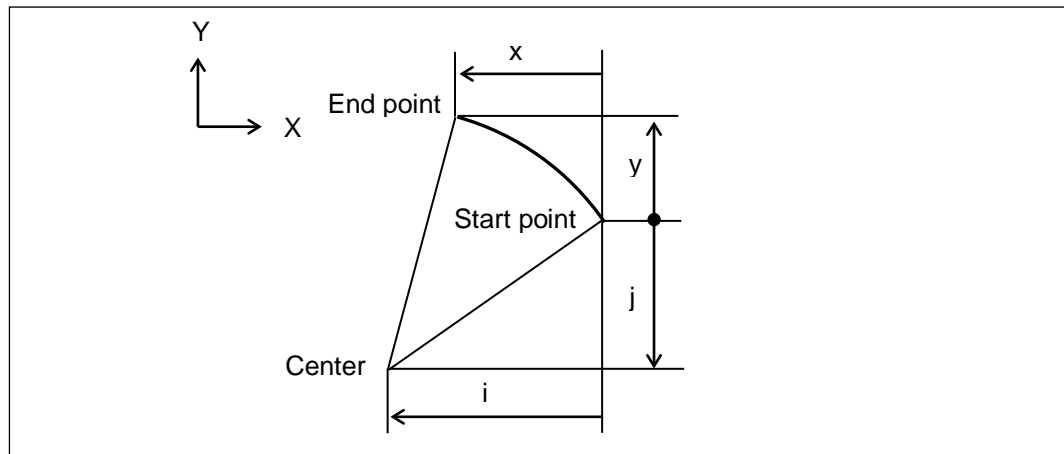
Center of the arc is specified by I, J, and K with respect to X, Y, and Z axis.

Vectors I, J, and K view the center of the arc from the start point. They are always specified by incremental values irrespective of G90 and G91.

- Absolute positioning  
G90 G03 Xx Yy Ii Jj Ff;



- Incremental positioning  
G91 G03 Xx Yy Ii Jj Ff;

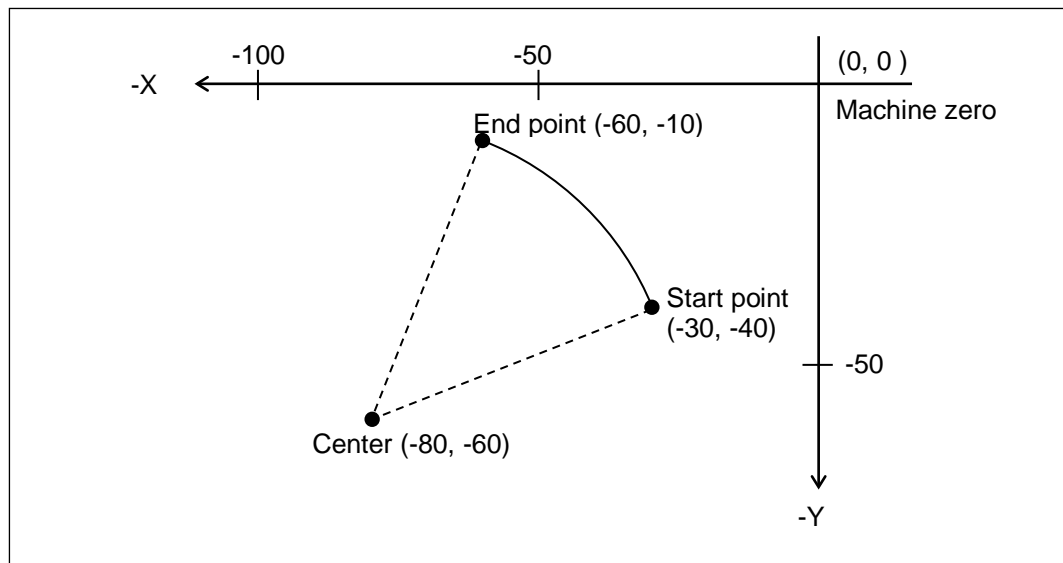


Absolute command

G03 X-60. Y-10. I-50. J-20. F1000;

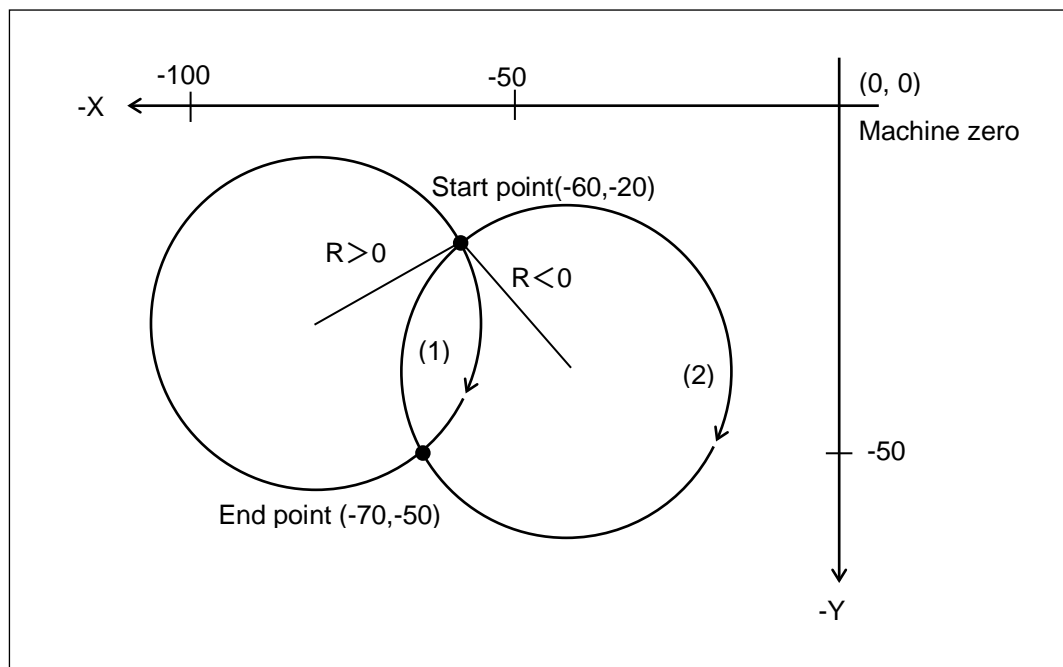
Incremental command

G03 X-30. Y30. I-50. J-20. F1000;



You may define the center with radius R instead of using vectors I, J, and K. Two different arcs are possible: an arc less than and the other greater than a half circle. Use a negative value for radius R when commanding an arc greater than a half circle.

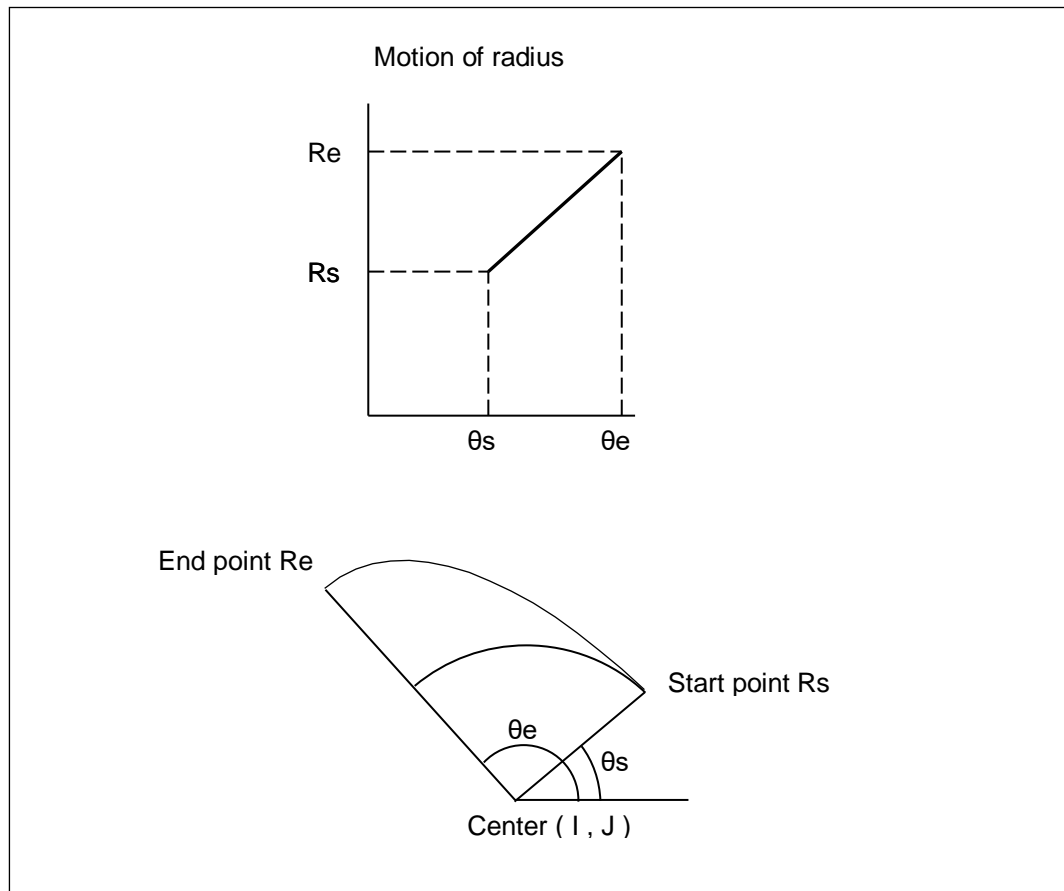
1. G02 X-70. Y-50. R25. F1000;
2. G02 X-70. Y-50. R-25. F1000;



- I, J, and K are considered zero if not specified.
- You cannot command an arc of R=0.
- When you omit X, Y, and Z, the end point is considered the same as start point and:  
 (1) an arc of 360° (a circle) is commanded when the center is defined by I, J, and K; or  
 (2) the alarm <<Arc Command Error>> occurs when the center is defined by R.
- You cannot define R and the set of I, J, and K simultaneously. The alarm <<Arc Command Error>> occurs.



- The tool moves as shown below when an end point is missing on the arc defined by a start point and the center of the arc.



- The alarm <<Curve Speed Error>> may occur when commanding an end point radius that is much greater than the start point radius.
- You may not command G36 to G39 in the circular mode. The alarm During <<Arc Mode>> occurs.
- After the feature coordinate setting, an arc command cannot be issued before the feature coordinate index.

### 3.3.2.5 Helical Thread Cutting Interpolation

Putting the other than selected plane axis command in the circular arc block permits a helical thread cutting.

Command format

<b>XY plane:</b>	<b>G17</b>	$\begin{pmatrix} \text{G02} \\ \text{G03} \end{pmatrix}$	X_ Y_ Z_	$\begin{pmatrix} \text{I_} \text{J_} \\ \text{R_} \end{pmatrix}$	$\begin{pmatrix} \text{A_} \\ \text{B_} \\ \text{C_} \end{pmatrix}$	F_;
<b>ZX plane:</b>	<b>G18</b>	$\begin{pmatrix} \text{G02} \\ \text{G03} \end{pmatrix}$	Z_ X_ Y_	$\begin{pmatrix} \text{K_} \text{I_} \\ \text{R_} \end{pmatrix}$	$\begin{pmatrix} \text{A_} \\ \text{B_} \\ \text{C_} \end{pmatrix}$	F_;
<b>YZ plane:</b>	<b>G19</b>	$\begin{pmatrix} \text{G02} \\ \text{G03} \end{pmatrix}$	Y_ Z_ X_	$\begin{pmatrix} \text{J_} \text{K_} \\ \text{R_} \end{pmatrix}$	$\begin{pmatrix} \text{A_} \\ \text{B_} \\ \text{C_} \end{pmatrix}$	F_;

Up to one linear axis and one additional axis can be controlled simultaneously when commanded for the surface other than selected plane.

The F code commands the feedrate in the circular interpolation axis..

The alarm <<Feed Rate Error>> occurs when F is greater than the maximum cutting travel speed or rapid feed.

The feedrate in the other than selected plane axis is determined by the values of "feedrate" in the circular interpolation axis, "end point X", "end point Y" and "end point Z". It can be calculated as follows:

$$F_z = \frac{180 \times L}{\pi \times R \times \theta} \times F$$

F	: Command speed (Selected plane axis)
R	: Radius
$\theta$	: Angle
Fz	: Other than selected plane of feedrate speed.
L	: Other than selected plane of feed distance.

Ex)

Setting following values: F=500 (mm/min), R=10 (mm),  
 $\theta=360$  (°), L=2 (mm)

$$F_z = (180 \times 2 \times 500) / (\pi \times 10 \times 360)$$

$$\approx 15.9 \text{ (mm/min)}$$

The alarm <<Feed Rate Error>> occurs when the feed rate calculated for the axis of other than the selected plane is greater than the maximum cutting travel speed of Machine parameter (System 1) or rapid feed.

When tool dia offset command is given, an offset is applied to the selected plane.

Helical thread cutting interpolation command is not possible while in the inverse time feed (G93) modal.

If a command is issued, the alarm <<Command not possible during inverse time feed>> is triggered.

### 3.3.2.6 Spiral Interpolation

An increment or decrement per rotation is specified for the circular interpolation command to perform spiral interpolation.

Command format

**XY plane:**

```
{ G17 } G02 X_ Y_ I_ J_ Q_ L_ F_ ;
{ G17 } G03 X_ Y_ I_ J_ Q_ L_ F_ ;
```

**ZX plane:**

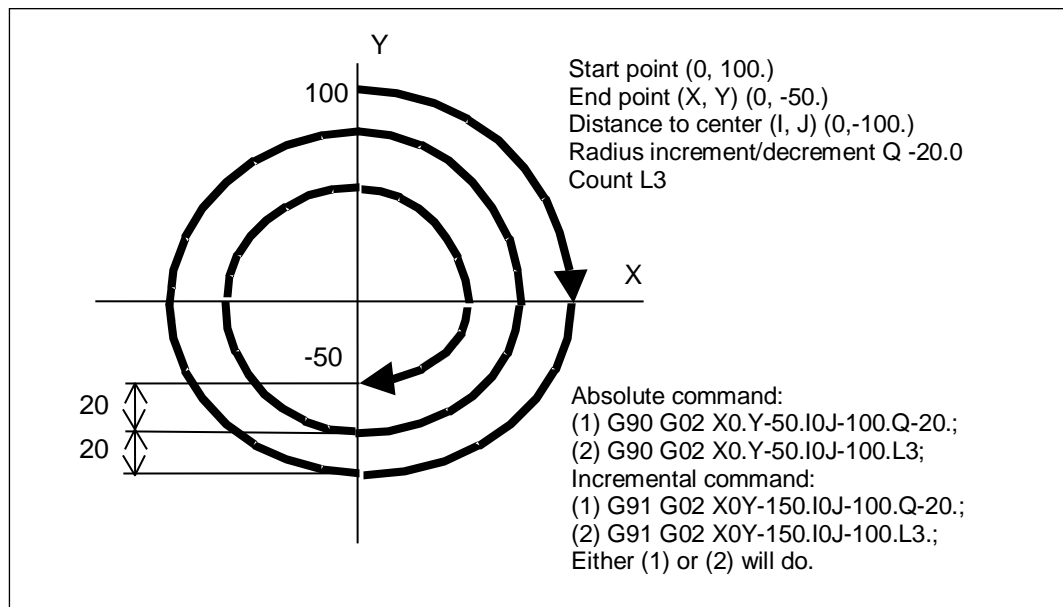
```
{ G18 } G02 Z_ X_ K_ I_ Q_ L_ F_ ;
{ G18 } G03 Z_ X_ K_ I_ Q_ L_ F_ ;
```

**YZ plane:**

```
{ G19 } G02 Y_ Z_ J_ K_ Q_ L_ F_ ;
{ G19 } G03 Y_ Z_ J_ K_ Q_ L_ F_ ;
```

- G02 : Clockwise cutting direction  
 G03 : Counterclockwise cutting direction  
 XYZ : End point coordinates  
 L : Number of rotations (An integer number is used to command. When the number is with decimal point, the number is rounded off.)  
 Example: Set "L6" for five and 1/4 rotations (5.25 rotations).  
 Q : Increment or decrement in radius per rotation  
 Setting a positive value increases the radius for each rotation. Setting a negative value decreases the radius for each rotation.  
 IJK : Vector (distance and direction) from the start point to the center (the same as circular interpolation)  
 F : Cutting speed

\* Either L (number of rotations) or Q (increment/decrement in radius) can be omitted. If "L" and "Q" are used together, "Q" is used.

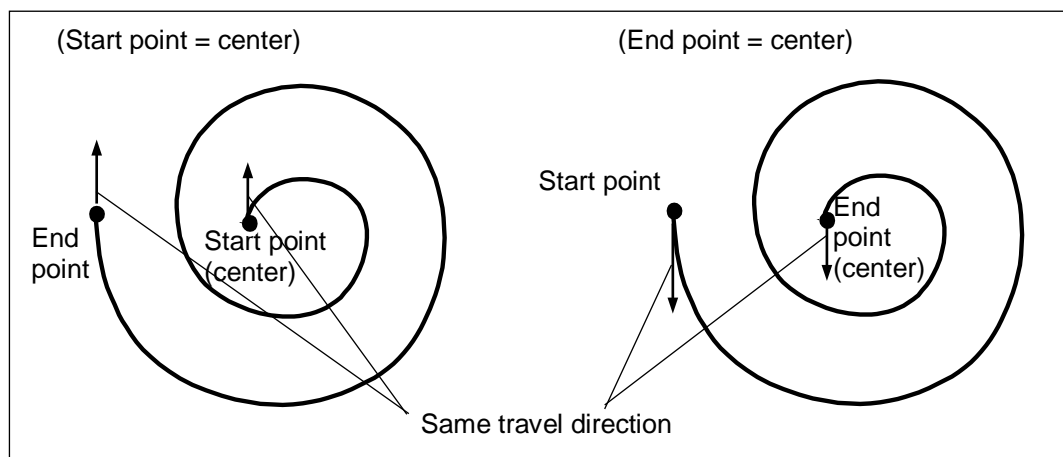


(Note 1) Tool dia offset can be performed only in the offset mode. The alarm

<<Cutter Compensation Error>> occurs in the startup and cancel mode.

(Note 2) The setting for [Tool dia offset] is applied relative to the start point and end point specified in the program during tool dia offset. The alarm <<Cutter Compensation Error>> occurs if the tool path after compensation intersects or touches the center of the spiral.

- (Note 3) The alarm <<Cutter compensation error>> is triggered when the compensated tool path crosses or touches the center of the spiral.
- (Note 4) The alarm Arc Radius Error Limit Exceeded occurs when the spiral end point calculated from per-rotation radius increment/decrement does not agree with the programed spiral end point and the difference is greater than the specified <arc radius error limit> of User parameter (Switch 1).
- (Note 5) The alarm <<Specified G Code Cannot Be Used>> occurs when corner CR is specified in the immediately preceding block.
- (Note 6) Corner CR is not used for spiral interpolation. The alarm Specified Numerical Data Cannot Be Used occurs.
- (Note 7) The alarm <<Arc Command Error>> occurs when the radius becomes 0 or less (including negative values) as a result of designation of per-rotation radius increment/decrement and rotation counts.
- (Note 8) If the rotation count exceeds 9999 when the amount of increase or decrease in the radius per rotation and the number of rotations are specified, then the alarm <<Arc command error>> is triggered.
- (Note 9) The alarm <<Arc Command Error>> occurs when a radius is specified by R parameters.
- (Note 10) The alarm <<Arc Command Error>> occurs when radius increment/decrement is zero.
- (Note 11) Do not use a Q0 command when start point radius is the same as end point radius (use an L command). The alarm <<Arc Command Error>> occurs.
- (Note 12) Even cutter compensation for the outside of a circle cannot be performed if start and end points are set on the center of a circle. The alarm <<Cutter Compensation Error>> occurs.
- (Note 13) Direction of travel on the start point when this is at the center (on the end point when this is at the center) is the same as the direction of travel on the end point (on the start point).



- (Note 14) Not commanded when mirror image is effective. The alarm <<Mirror Image Mode>> Is ON occurs.
- (Note 15) Spiral interpolation cannot be commanded when scaling is effective. The alarm <<Scaling>> occurs.
- (Note 16) When spiral interpolation and cutter compensation are followed by a cutter compensation release command in the succeeding block, the end point will be where a vertical vector is set up at the end point of the spiral interpolation.
- (Note 17) In-position check is performed in the blocks before and after spiral interpolation.
- (Note 18) Spiral interpolation command is not possible while in the inverse time feed (G93) modal. If a command is issued, the alarm <<Command not possible during inverse time feed>> is triggered.

### 3.3.2.7 Conical interpolation

The travel command of another axis in addition to the spiral interpolation command is added and an increment and decrement is specified for that axis per spiral rotation to perform conical interpolation.

Command format

**XY plane:**

{G17} G02 X\_ Y\_ Z\_ I\_ J\_ K\_ Q\_ L\_ (A\_ B\_ C\_) F\_;

{G17} G03 X\_ Y\_ Z\_ I\_ J\_ K\_ Q\_ L\_ (A\_ B\_ C\_) F\_;

**ZX plane:**

{G18} G02 Z\_ X\_ Y\_ K\_ I\_ J\_ Q\_ L\_ (A\_ B\_ C\_) F\_;

{G18} G03 Z\_ X\_ Y\_ K\_ I\_ J\_ Q\_ L\_ (A\_ B\_ C\_) F\_;

**YZ plane:**

{G19} G02 Y\_ Z\_ X\_ J\_ K\_ I\_ Q\_ L\_ (A\_ B\_ C\_) F\_;

{G19} G03 Y\_ Z\_ X\_ J\_ K\_ I\_ Q\_ L\_ (A\_ B\_ C\_) F\_;

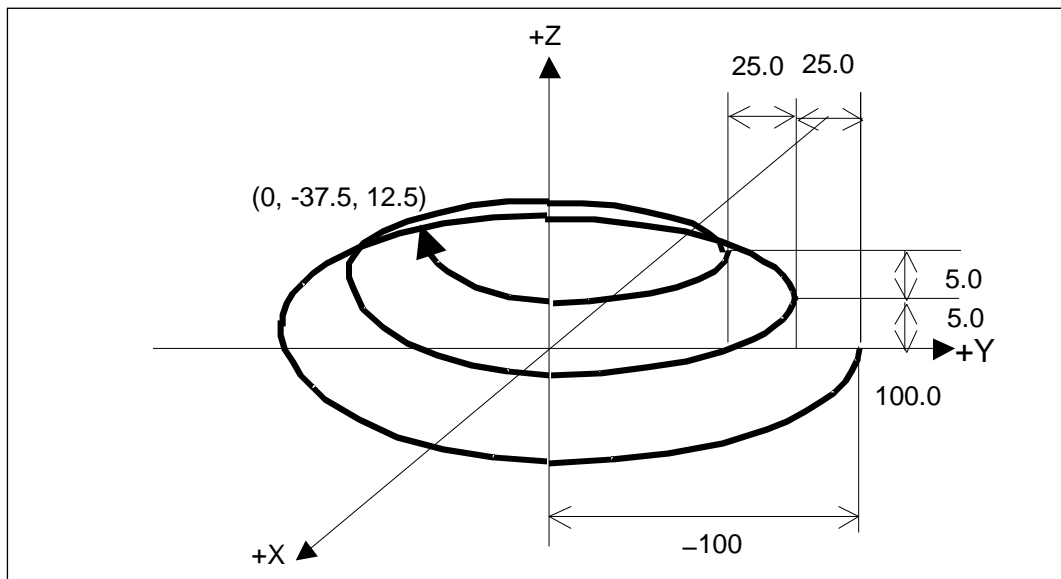
Up to one axis (linear axis or additional axis) can be controlled when commanded for the surface other than selected plane.

- G02 : Clockwise cutting direction
- G03 : Counterclockwise cutting direction
- XYZ : Coordinates of end point
- L : Number of rotations (An integer number is used to command. When the number is with decimal point, the number is rounded off.)  
Example: Set "L6" for five and 1/4 rotations (5.25 rotations).
- Q : Increment or decrement in radius per rotation.  
Setting a positive value increases the radius for each rotation.  
Setting a negative value decreases the radius for each rotation.
- IJK : Two axes are vectors from start point to center. The remaining axis specifies per-rotation spiral increment/decrement for conical interpolation. \*

Plane to be set	Vector from start point to center	Increment and decrement in height per spiral rotation
G17 XY	I, J	K
G18 ZX	K, I	J
G19 YZ	J, K	I

F : Cutting speed (Selected plane axis)

- \* I, J, K, L, and Q (Incremental height, rotation count, and incremental radius):  
Specify one, and you may omit the other two.  
If L and Q are contradictory to each other, Q has priority.  
If there is a discrepancy between "L" and the increment/decrement in height, the latter is used.  
If there is a discrepancy between "Q" and the increment/decrement in height, the former is used.  
Priority Higher ← "Q" > Increment/decrement in height > "L" → Lower



Example of program: The orders of the numerical values in the brackets ( ) are X, Y, and Z.

Start point	(0., 100., 0.)
End point	(0., -37.5, 12.5)
Distance to the center	(0., -100.)
Increment/decrement in radius	-25.
Increment/decrement in height	5.
No. of rotations	3

Absolute command

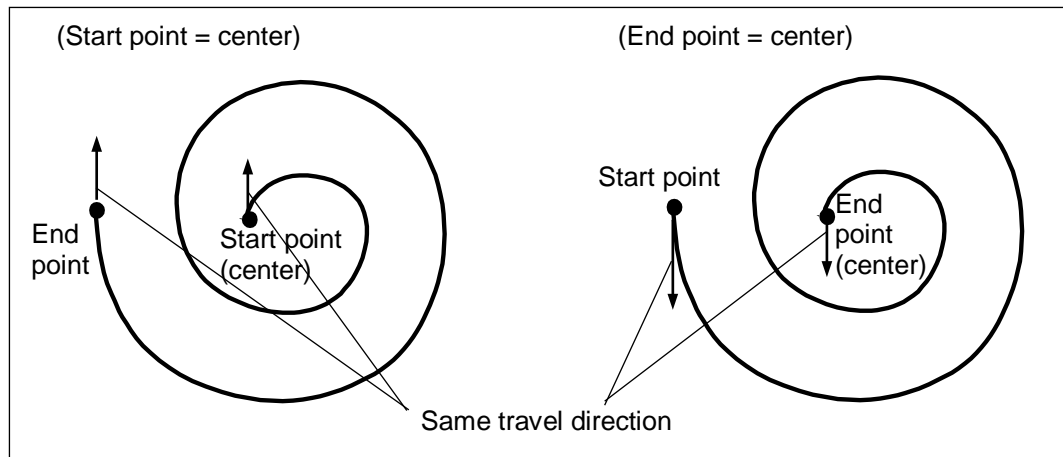
```
G90 G02 X0. Y-37.5 Z12.5 I0. J-100.  $\begin{bmatrix} K5 \\ Q-25 \\ L3 \end{bmatrix}$  F300.;
```

Incremental command

```
G91 G02 X0. Y-137.5 Z12.5 I0. J-100.  $\begin{bmatrix} K5 \\ Q-25 \\ L3 \end{bmatrix}$  F300.;
```

- (Note 1) Tool dia offset can be performed only in the offset mode. The alarm <<Cutter Compensation Error>> occurs in the startup and cancel mode.
- (Note 2) During execution of the cutter compensation, tool diameter offset compensation is performed, on the selected plane, for the start and end points specified by the program. The alarm <<Cutter Compensation Error>> occurs when the tool path after compensation intersects or touches the center of the cone.
- (Note 3) The alarm <<Cutter compensation error>> is triggered when the compensated tool path crosses or touches the center of the cone.
- (Note 4) The alarm Arc Radius Error Limit Exceeded occurs when the end point of the cone calculated from per-rotation radius increment/decrement does not agree with the programmed end point of the cone and the difference is greater than the specified <arc radius error limit> of User parameter (Switch 1).
- (Note 5) The alarm <<Specified G Code Cannot Be Used>> occurs when corner CR is specified in the immediately preceding block.
- (Note 6) Corner CR is not used for conical interpolation. The alarm Specified Numerical Data Cannot Be Used occurs.

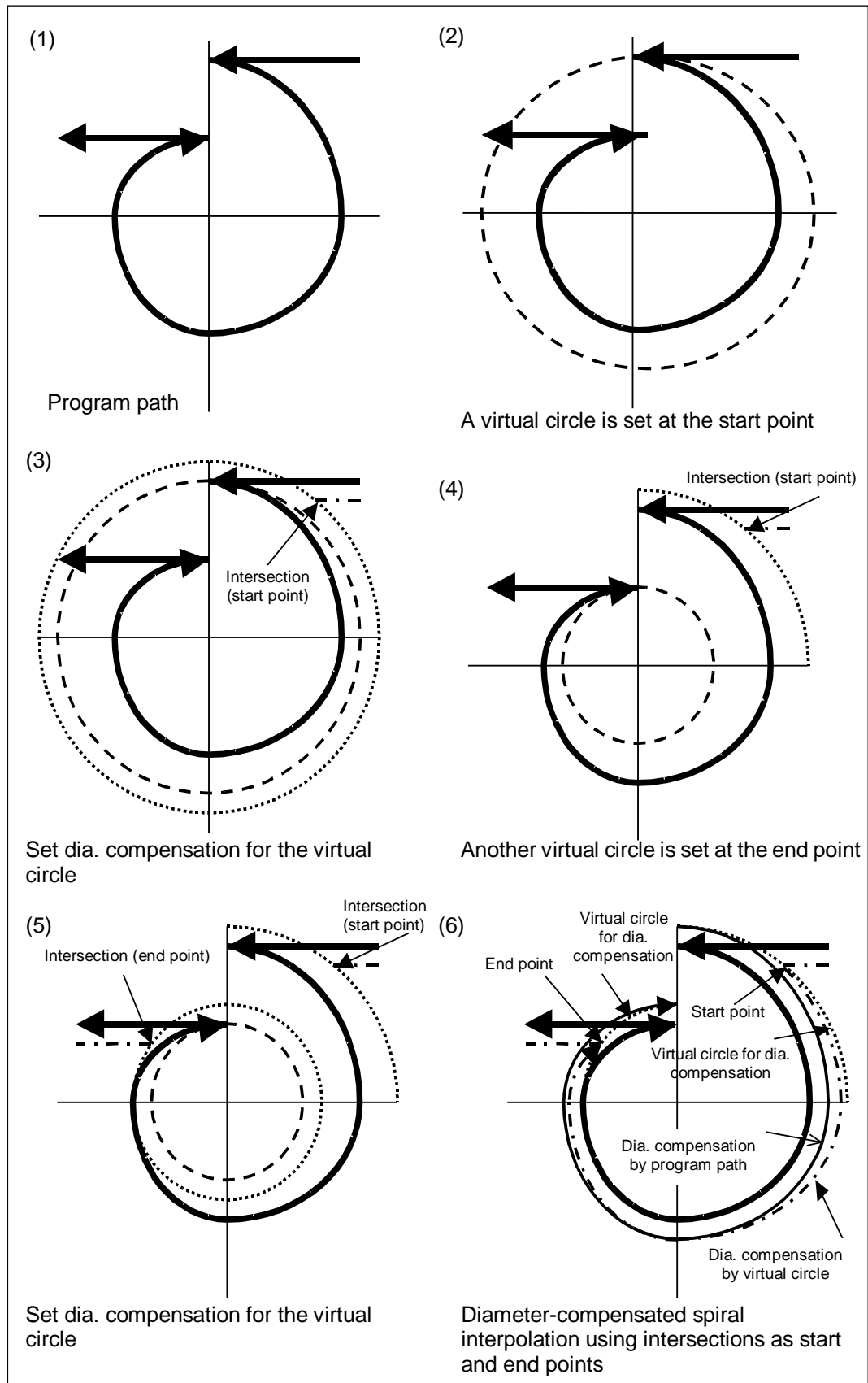
- (Note 7) The alarm <<Cutter Compensation Error>> occurs when you change the direction of cutter compensation (G41, G42) in the immediately preceding and succeeding blocks of conical interpolation.
- (Note 8) If the rotation count exceeds 9999 when the amount of increase or decrease in the radius per rotation and the number of rotations are specified, then the alarm <<Arc command error>> is triggered.
- (Note 9) The alarm <<Arc Command Error>> occurs when a radius is specified by R parameters.
- (Note 10) The alarm <<Arc Command Error>> occurs when radius increment/decrement is zero.
- (Note 11) Do not use a Q0 command when start point radius is the same as end point radius (use an L command). The alarm <<Arc Command Error>> occurs.
- (Note 12) Cutter compensation for the outside of a circle cannot be performed if start and end points are set on the center of a circle. The alarm <<Cutter Compensation Error>> occurs.
- (Note 13) Direction of travel on the start point when this is at the center (on the end point when this is at the center) is the same as the direction of travel on the end point (on the start point).



- (Note 14) Conical interpolation cannot be commanded when mirror image is effective. The alarm <<Mirror Image Mode>> Is ON occurs.
- (Note 15) Conical interpolation cannot be commanded when scaling is effective. The alarm <<Scaling>> occurs.
- (Note 16) When conical interpolation and cutter compensation are followed by a cutter compensation release command in the succeeding block, the end point will be where a vertical vector is set up at the end point of the conical interpolation.
- (Note 17) In-position check is performed in the blocks before and after conical interpolation.
- (Note 18) Conical interpolation command is not possible while in the inverse time feed (G93) modal. If a command is issued, the alarm <<Command not possible during inverse time feed>> is triggered.

### 3.3.2.8 Cutter Compensation for Spiral and Conical Interpolation

Virtual circles, with their centers set on the center of spiral interpolation, are created at the start and end points of the program. Cutter compensation is performed relative to the virtual circles and spiral interpolation is executed for the result of the compensation.





### 3.3.2.9 Involute interpolation

The involute interpolation is carried out at the command feedrate and travels along an involute curve from the current position to the end point.

Command format

**For X-Y plane:**  
 {G17} G02.2 X\_ Y\_ Z\_ I\_ J\_ R\_ F\_ ;  
 {G17} G03.2 X\_ Y\_ Z\_ I\_ J\_ R\_ F\_ ;

**For Z-X plane:**  
 {G18} G02.2 Z\_ X\_ Y\_ K\_ I\_ R\_ F\_ ;  
 {G18} G03.2 Z\_ X\_ Y\_ K\_ I\_ R\_ F\_ ;

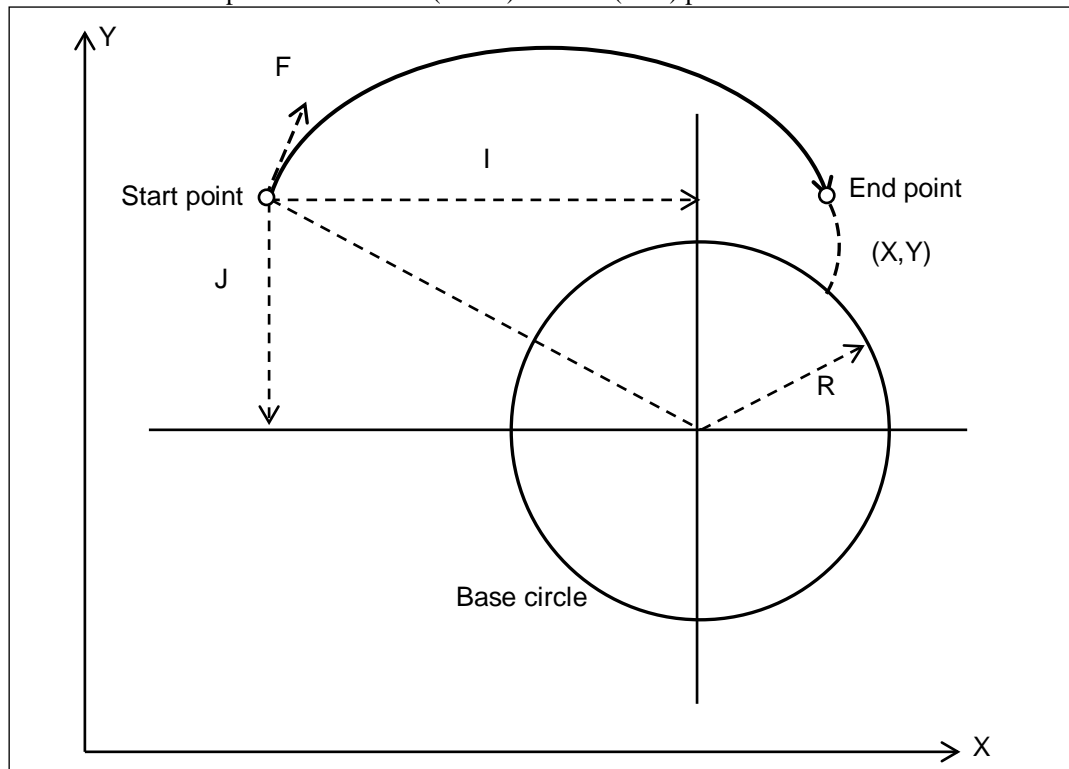
**For Y-Z plane:**  
 {G19} G02.2 Y\_ Z\_ X\_ J\_ K\_ R\_ F\_ ;  
 {G19} G03.2 Y\_ Z\_ X\_ J\_ K\_ R\_ F\_ ;

3

A description of the commands is shown below.

Rotation direction		G02.2	Clockwise (CW)
		G03.2	Counterclockwise (CCW)
End point	G90 mode	X, Y, Z	End position in workpiece coordinate system
	G91 mode	X	Distance in X-axis direction between the start and end points
		Y	Distance in Y-axis direction between the start and end points
		Z	Distance in Z-axis direction between the start and end points
Distance from start point to center point of base circle		I	Distance in X-axis direction from the start point to the center point of the base circle
		J	Distance in Y-axis direction from the start point to the center point of the base circle
		K	Distance in Z-axis direction from the start point to the center point of the base circle
Base circle radius		R	Radius of base circle
Feedrate		F	Speed along tangent line of involute curve

Ex: Involute interpolation command (G02.2) for G17 (X-Y) plane



(Note 1) The involute interpolation option is required to enable and use this function.

(Note 2) This function can only be used in NC language mode.

Refer to “Chapter 12 Options (23) Involute interpolation function” in the Operation Manual for further details.

### 3.3.3 Circle Cutting (G12, G13)

The tool starts at the center of a circle, cuts the inside, and returns to the center.

Command format

```
G12 I_D_F_;
G13 I_D_F_;
```

G12 : Clockwise cutting direction

G13 : Counterclockwise cutting direction

I : Radius of a circle. Signs + and - are ignored; always considered +.

D : Specify compensation amount.

Compensation amount is commanded by tool number.

For a positive value, the inside of the radius specified by I parameter is cut along the circle. For a negative one, the outside of the radius specified by I parameter is cut.

F : Cutting speed

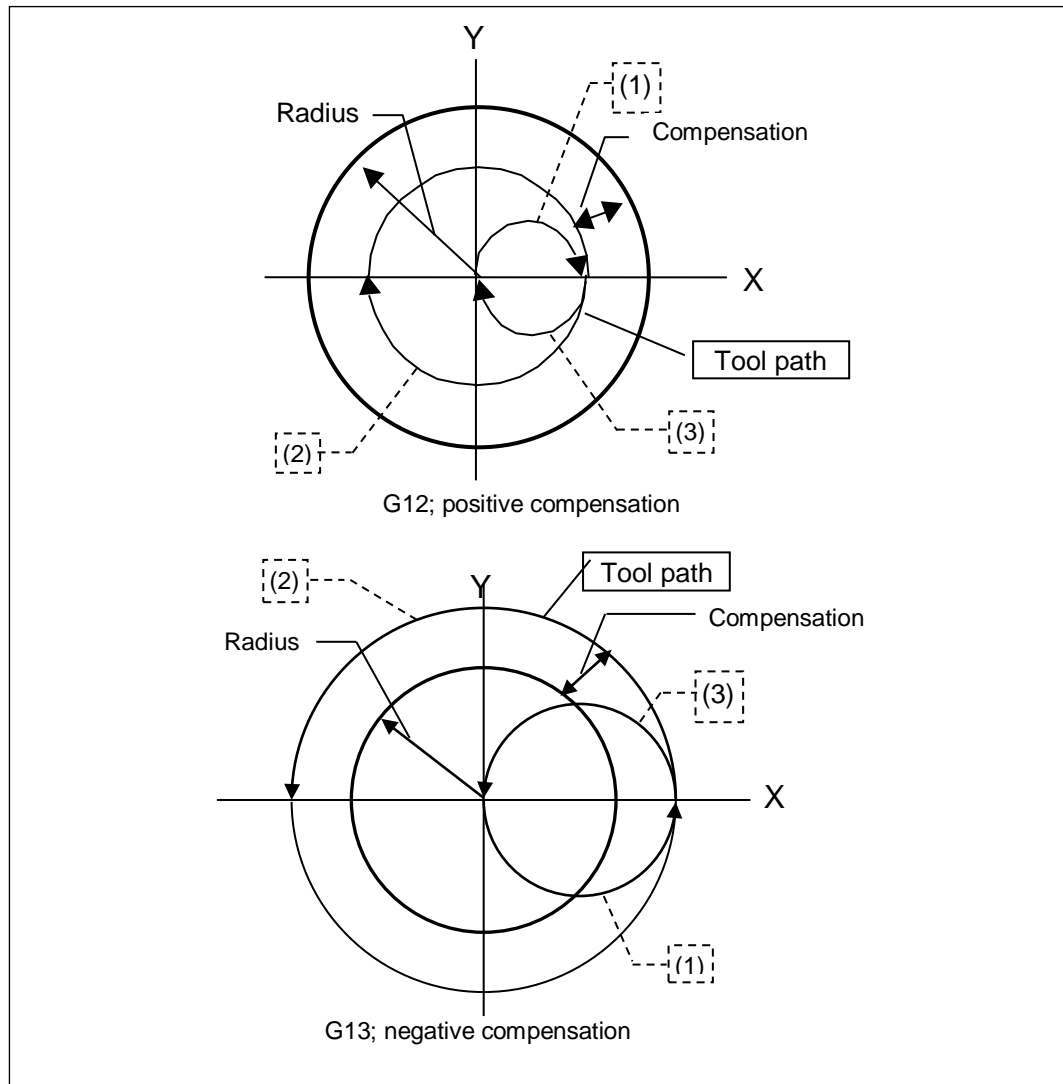
Operation (assuming an XY plane has been selected)

The tool starts from the center of the circle in the X-axis direction along a half-circle arc. Rotating direction is as determined by G12 or G13.

From (1), the tool moves along the entire circle in the direction specified by G12 or G13.

From the end point of (2), the tool moves to the center of the circle following the half-circle arc.

Rotating direction is as determined by G12 or G13.



- (Note 1) The alarm <<Arc Command Error>> occurs when the D command is omitted.
- (Note 2) The alarm <<Arc Command Error>> occurs if the radius (I command) is zero or a negative value after subtracting compensation amount.
- (Note 3) The alarm <<Cutter Compensation Error>> occurs when you specify cutter compensation control command (G40, G41, G42) (startup and cancel modes) and circle cutting command (G12, G13) simultaneously.
- (Note 4) Corner CR may not be set in the circle cutting command and immediately preceding block command. The alarms <<Specified G Code Cannot Be Used>> or Specified Numerical Data Cannot Be Used occur, respectively.
- (Note 5) The alarm <<Cutter Compensation Too Large>> occurs when the radius after compensation is smaller than tool diameter.
- (Note 6) Circle cutting is performed on the currently selected plane (G17, G18, G19).
- (Note 7) Start and end points are identical in circle cutting.
- (Note 8) When circle cutting (G12, G13) is executed during cutter compensation (G41, G42), the latter is effective for the path compensated by D command.
- (Note 9) After the feature coordinate setting, a circular cutting command cannot be issued before the feature coordinate index.
- (Note 10) Circular cutting command is not possible while in the inverse time feed (G93) modal. If a command is issued, the alarm <<Command not possible during inverse time feed>> is triggered.

## 3.4 Dwell (G04)

When a block is finished, the next block starts after a specified time period on completing the in-position check.

Command format

<b>G04 P_;</b>
----------------

or

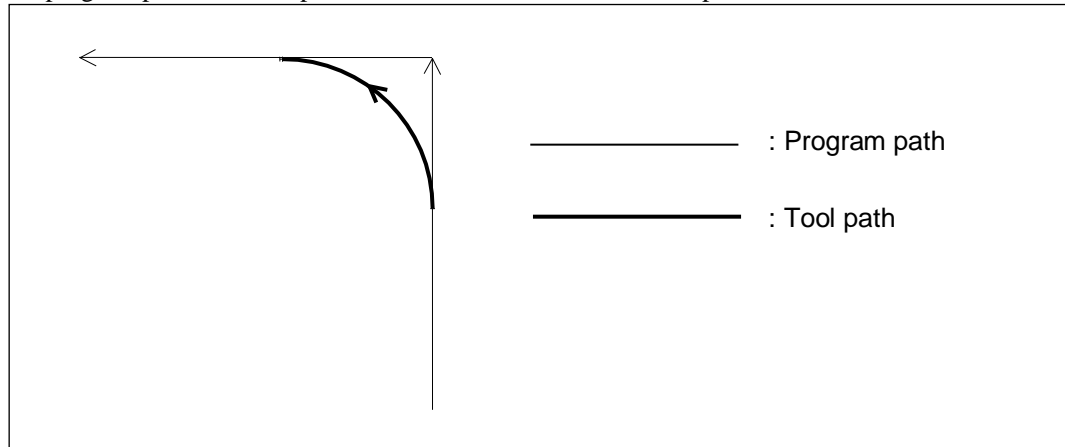
<b>G04 X_;</b>
----------------

P, X : Dwell time (sec.)

**(Note)** When a decimal point command is absent in dwell time designation, seconds or milliseconds apply if <program unit> of <user parameters> is <Base> or <Minimum> respectively.

### 3.5 Exact Stop Check (G09, G61, G64)

Each axis is accelerated / decelerated respectively. If, therefore, there is a large change in axis speed (cutting feed rate) between two neighboring blocks, the tool will run along an inside path relative to the program path. Exact stop check function is used to avoid this problem.



#### 1. Exact stop check (G09)

Command format

**G09;**

In-position check is performed at the end of each block before the tool proceeds to the next block.

**(Note 1) G09 is effective only in the commanded block.**

**(Note 2) Exact stop check is always performed in the positioning mode (G00) irrespective of presence or absence of the G09 command.**

#### 2. Exact stop check (G61)

Command format

**G61;**

The exact stop check is carried out at the end of each block after this command is issued and until the cutting mode (G64) command is issued.

#### 3. Cutting mode (G64)

Command format

**G64;**

From this command until G61 is specified, the tool starts, at each joint of blocks, traveling according to the instructions in the succeeding block as soon as possible, to avoid deceleration to the extent possible.

**(Note 1) Exact stop check is performed also in the cutting mode if a block includes a positioning mode (G00) or exact stop check (G09), or in the cutting feed rate blocks that are not continuous.**

**(Note 2) The finishing operation in the thread cutting cycle is always performed in the cutting mode regardless of this command.**

**(Note 3)**

Current block \ Succeeding block	Positioning	Cutting feed	No traveling
Positioning	×	×	×
Cutting feed	×	○	×
No traveling	×	×	×

○ Cutting mode

× Exact stop check mode

Exact stop check is always performed when the current block clamps at additional axis travel and when the succeeding block unclamps at additional axis travel.

## 3.6 Programmable Data Input (G10)

### 3.6.1 Entering Workpiece Coordinate Zero

Command format

G10 L2 Pn X_ Y_ Z_ A_ B_ C_;
------------------------------

n=0	:	[External]
n=1	:	G54
n=2	:	G55
n=3	:	G56
n=4	:	G57
n=5	:	G58
n=6	:	G59

The entered value becomes the new compensation amount in the G90 (absolute command) mode while it is added to the currently set compensation amount to specify the new one in the G91 (incremental command) mode.

**(Note 1)** When the additional axis is commanded and the optional additional axis is not installed, an alarm will occur.

**(Note 2)** The workpiece coordinate zero cannot be changed while in feature coordinate manufacturing mode (G68.2 modal in progress). The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.

### 3.6.2 Entering Tool Length / Cutter Compensation Data

Command format

Tool length compensation data

G10 L10 P_ R_;
----------------

Cutter compensation data

G10 L12 P_ R_;
----------------

P	:	Compensation number (1 to 99)
R	:	Compensation amount

The entered value becomes the new compensation amount in the G90 (absolute command) mode while it is added to the currently set compensation amount to specify the new one in the G91 (incremental command) mode.

**(Note)** The alarm <<Comm. issued to area other than (tool) data area.>> is triggered when a value is input that is outside of the range set in the <Tool data range> under the <Tool data> parameter.

### 3.6.3 Tool Wear Compensation

When a tool length/cutter compensation is commanded by the program, the wear compensation data relevant to the specified tool compensation command is automatically retrieved and considered in the cutting operation.

Command format

Tool length wear compensation **G10 L11 P\_ R\_;**

Cutter wear offset **G10 L13 P\_ R\_;**

P : Compensation number (1 to 99)

R : Compensation amount

The commanded value is used as the new compensation amount in the absolute (G90) mode while it is added to the current set compensation amount in the incremental (G91) mode.

Set range:  $\pm 99.999\text{mm}$   $\pm 9.9999\text{inch}$  (for Type 1 least input increment)

**(Note)** The alarm <<Comm. issued to area other than (tool) data area.>> is triggered when a value is input that is outside of the range set in the <Tool data range> under the <Tool data> parameter.

### 3.6.4 Tool Offset Data Input

\* Available when equipped with a lathe function

Tool length compensation data **G10 L90 P\_X\_Y\_Z\_R\_T\_;**

P : Tool number (1 – 99)

X : Tool position offset (X)

Y : Tool position offset (Y)

Z : Tool length offset (Z)

R : Tool diameter / nose R compensation

T : Virtual teeth number (0 – 9)

The compensation specified on X / Y / Z / R becomes the new compensation for the tool number specified by P when G90 (absolute command) mode is enabled. The new compensation is the sum of compensation specified on X / Y / Z / R plus the current compensation setting for the tool number specified by P when G91 (incremental command) mode is enabled. T (Virtual teeth number) is not based on G90 / G91.

**(Note)** The alarm <<Comm. issued to area other than (tool) data area.>> is triggered when a value is input that is outside of the range set in the <Tool data range> under the <Tool data> parameter.

### 3.6.5 Tool Offset Wear Data Input

\* Available when equipped with a lathe function

Command format **G10 L91 P\_ X\_ Y\_ Z\_ R\_;**

P : Tool number (1 – 99)  
 X : Tool position wear offset data (X)  
 Y : Tool position wear offset data (Y)  
 Z : T length wear offset (Z)  
 R : Tool diameter / nose R wear offset

The compensation specified on X / Y / Z / R becomes the new compensation for the tool number specified by P when G90 (absolute command) mode is enabled. The new compensation is the sum of compensation specified on X / Y / Z / R plus the current compensation setting for the tool number specified by P when G91 (incremental command) mode is enabled.

**(Note)** The alarm <<Comm. issued to area other than (tool) data area. >> is triggered when a value is input that is outside of the range set in the <Tool data range> under the <Tool data> parameter.

### 3.6.6 Data Input of Extended Workpiece Coordinate

Command format **G10 L20 Pn X\_ Y\_ Z\_ A\_ B\_ C\_;**

Pn : Code specified in extended workpiece coordinate system  
 n : 1 through 48  
 X, Y, Z : Workpiece origin offset

**(Note)** The workpiece coordinate zero cannot be changed while in feature coordinate manufacturing mode (G68.2 modal in progress). The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.

The entered values become the new offset values in the absolute mode (G90) while they are added to the current offset value to make new ones in the incremental mode (G91).

### 3.6.7 Workpiece Coordinate System Using the Results of Measurements

Command format **G10 L99 Pn X\_ Y\_ Z\_ Q\_;**

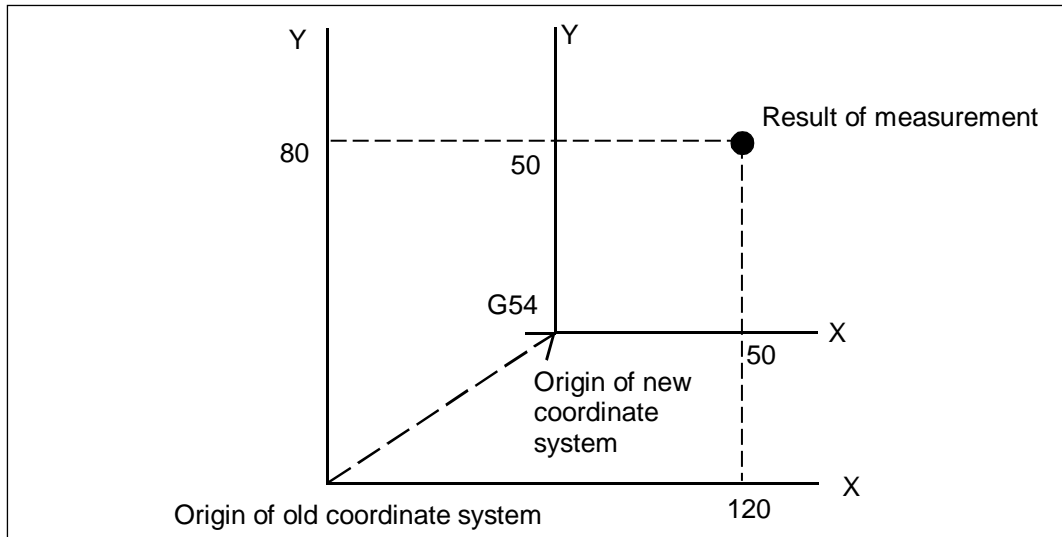
n = 1 : G54  
 n = 2 : G55  
 n = 3 : G56  
 n = 4 : G57  
 n = 5 : G58  
 n = 6 : G59  
 Q : Storage number (1 to 4) for results of measurement

A new coordinate system is set using the result of automatic measurement (G121 to 129).

Ex) Automatic measurement is performed on G54 coordinate system, yielding the data (120, 80).  
 We now set a new coordinate system moving this position to (50, 50).

G54 G121 X100. Y100. I20. J20. Z-10. R10. ; (angle measurement)  
 G10 L99 X50. Y50.;





**(Note)** The workpiece coordinate zero cannot be changed while in feature coordinate manufacturing mode (G68.2 modal in progress). The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.

### 3.6.8 Data Input of Extended Workpiece Coordinate Zero Based on Measurement Results

Command format

```
G10 L98 Pn X_ Y_ Z_ Q_;
```

- n : Extended workpiece coordinate system (1 to 48)  
 Q : Storage number (1 to 4) for results of measurement

**(Note)** The workpiece coordinate zero cannot be changed while in feature coordinate manufacturing mode (G68.2 modal in progress). The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.

### 3.6.9 Data Input for Reference Rotary Fixture Offset

Command format

```
G10 L21 Pn X_ Y_ Z_ A_ B_ C_ Qm;
```

- n : Reference rotary fixture offset number (1 to 8)  
 X, Y and Z : Reference offset  
 A, B and C : Reference angle  
 Q : Axis for calculation  
   m = 0 : A-axis  
   m = 1 : B-axis  
   m = 2 : C-axis  
   m = 3 : AB-axis  
   m = 4 : AC-axis  
   m = 5 : BC-axis

When G90 (absolute command) mode is enabled, the command values for X, Y, Z, A, B and C are set by the reference fixture offset that is specified in P. When G91 (incremental command) mode is enabled, the sum of the values specified for X, Y, Z, A, B and C is set by the current values set in the reference fixture offset specified in P. Q (Axis for calculation) is not based on G90 / G91.

### 3.6.10 Tool Information Input

Command format

G10 L97 P\_ Q\_ R\_ W\_ V\_ F\_ S\_ U\_ J\_;

- P : Tool number (1 to 99)  
 Q : Life type  
     1. Not counted  
     2 Time (minutes)  
     3 Drilling (holes)  
     4 Program (cycles)  
 R : Tool life  
 W : Life warning  
 V : Initial life / end life (switched by the setting of <Tool Life Counting Method> of <User Parameters>)  
 F : F command value (metric: 0.01 to 999999.99; inch: 0.001 to 99999.999 \*)  
 S : S command value (1 to 99999)  
 U : Maximum speed (-1, 0, 1 to 99999)  
 J : Extended information  
     0 Tool cleaning enabled / CTS enabled  
     1 Tool cleaning disabled / CTS enabled  
     2 Tool cleaning enabled / CTS disabled  
     3 Tool cleaning disabled / CTS disabled

Not rotatable is determined if you set the maximum speed to zero. Unset is indicated if you select -1.  
 Feed rate commanded by F is set as feed per minute even in the G95 (feed per revolution) mode.

\* For Type 1 least input increment.

## 3.7 Coordinate System (G17 to 19, G52 to 59, G54.1, G92)

### 3.7.1 Plane Selection (G17, G18, G19)

Plane selection specifies the plane where the following operations are performed: circular interpolation, cutter compensation, nose R compensation, rotational transformation, corner CR, circular cutting, spiral interpolation, conical interpolation and involute interpolation.

XY plane selection

Command format

**G17**

ZX plane selection

Command format

**G18**

YZ plane selection

Command format

**G19**

Tool length compensation always applies to Z axis irrespective of plane selection.

G17 only is valid for canned cycle, automatic workpiece measurement, coordinate calculation and feature coordinate setting, and an alarm is triggered for G18 and G19.

Corner CR is only executed when the current and the succeeding block share the same plane. An alarm occurs if different planes are selected in the blocks.

An alarm is triggered when a plane is selected that is different from the current modal during diameter compensation and nose R compensation.

### 3.7.2 Machine Coordinate System Selection (G53)

The below-mentioned command is used to specify coordinate values in a machine coordinate system.

Command format

**G53;**

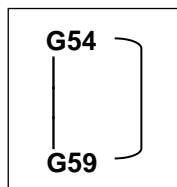
Coordinate values commanded in the same block as G53 are recognized in the machine coordinate system.

**(Note)** A G53 command is ignored when specified in the incremental mode (G91).

### 3.7.3 Workpiece Coordinate System Selection (G54 to G59)

Set 6 sets of workpiece-specific coordinate systems in the data, and you can call the required workpiece coordinate system using the G54 to G59 commands.

Command format



- G54 : Workpiece Coordinate System (\*1)
- G55 : Workpiece Coordinate System (\*2)
- G56 : Workpiece Coordinate System (\*3)
- G57 : Workpiece Coordinate System (\*4)
- G58 : Workpiece Coordinate System (\*5)
- G59 : Workpiece Coordinate System (\*6)

Data setting

1. Reference and setting can be performed on the <Workpiece coord zero> screen of the data bank.
2. Setting can be performed from the program using the G10 command.  
Refer to "3.6.1 Entering Workpiece coordinate zero" for the description of command format.

### 3.7.4 Extended Workpiece Coordinate System Selection (G54.1)

Command format

**G54.1 Pn;**

Pn : Code specified in extended workpiece coordinate system  
n : 1 to 48

The above command allows you to select 48 sets of workpiece coordinate systems.

The same function is available using G54 in place of G54.1.

When there is no P code on the same block after G54.1, the extended workpiece coordinate system 1 is selected.

Data setting

1. Reference and setting can be performed on the <Workpiece coord zero> screen of the data bank.
2. Setting can be performed from the program using the G10 command.  
Refer to "3.6.6 Data Input of Extended Workpiece Coordinate Zero" for further details.

### 3.7.5 Workpiece Coordinate System Setting (G92)

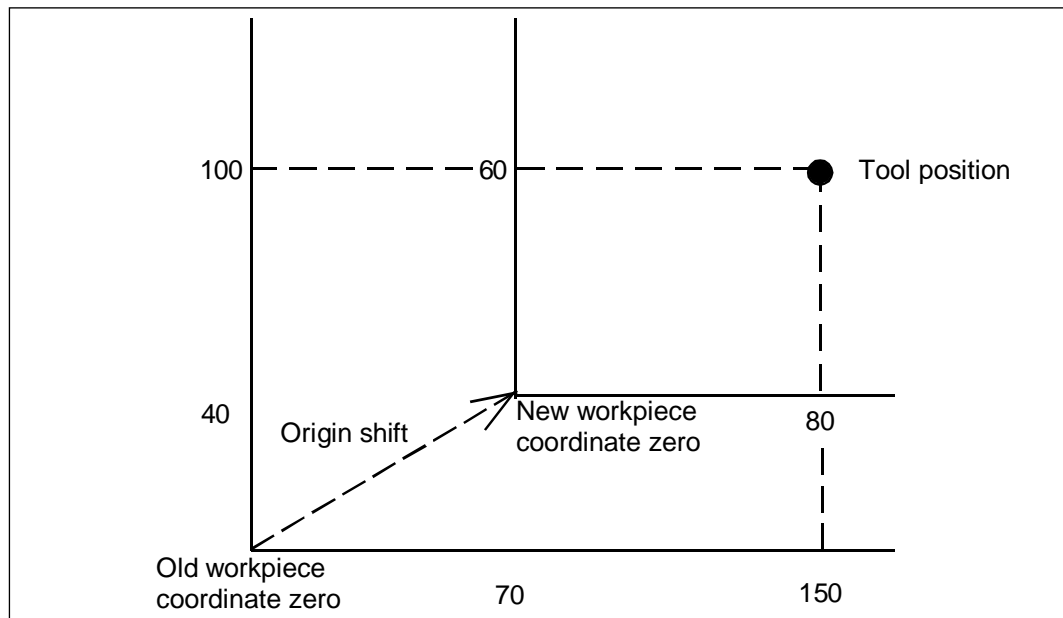
Change of workpiece zero position can be commanded as follows:

Command format

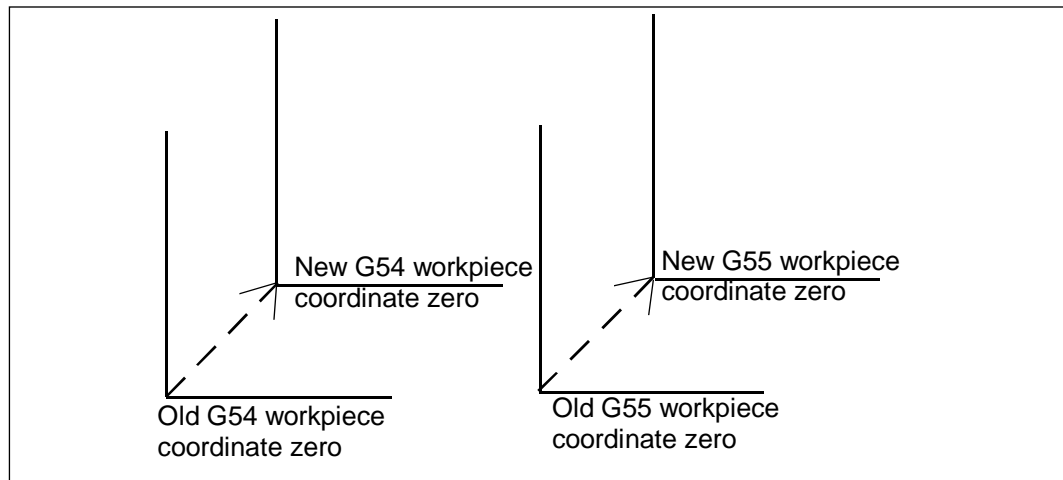
**G92 X\_ Y\_ Z\_ A\_ B\_ C\_;**

This command shifts the zero position in the working coordinate system so that the current tool position becomes to the commanded coordinate values.

G92 X80 Y60: Current tool position (150, 100) is changed to absolute coordinate position (80, 60) by this command.

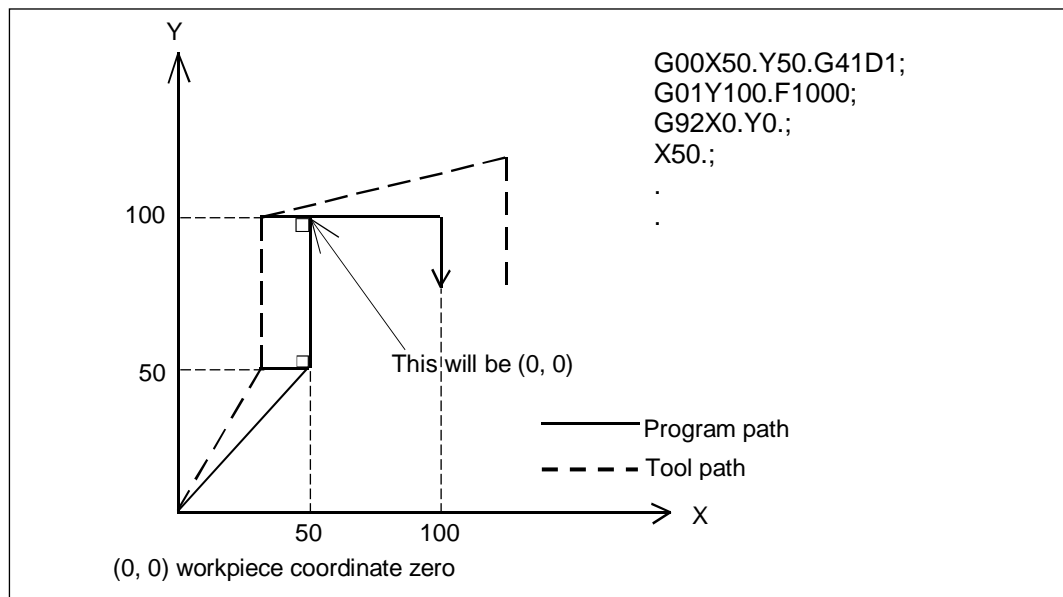


- (Note 1) The commanded coordinate values are always absolute regardless of G90 and G91.
- (Note 2) The working coordinate values of the not commanded axes do not change.
- (Note 3) The current working zero position shifts when G92 is executed, and other working zero positions also shift the same amount accordingly.

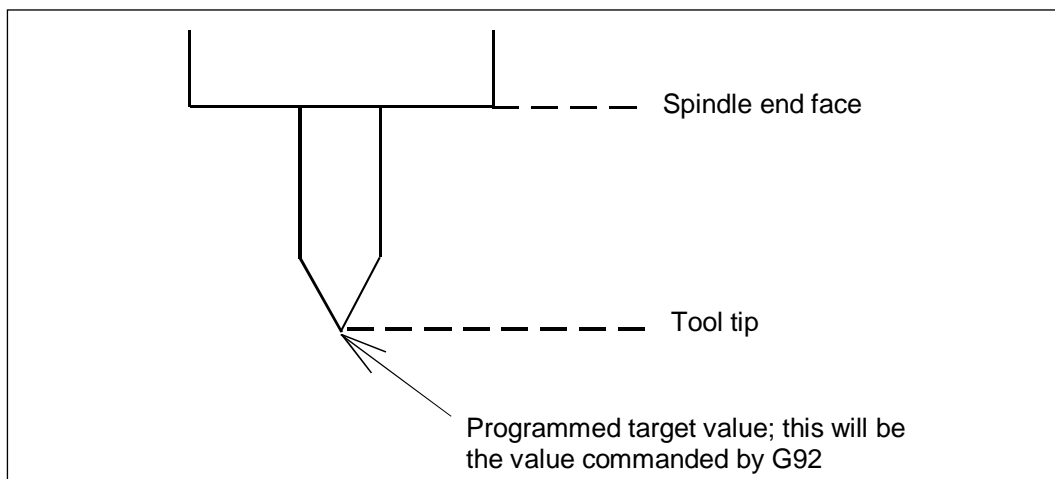


In the above figures, G92 is commanded in the coordinate system of G54. When the working zero position of G54 shifts, the other working zero positions of G55 through G59 also shift the same amount as G54.

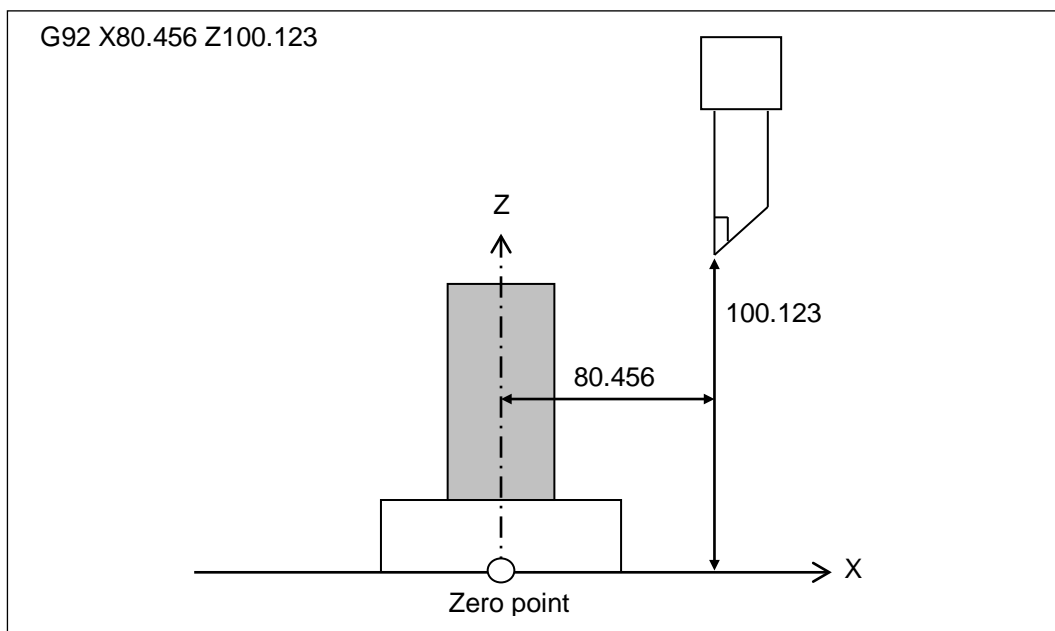
- (Note 4) When a command is issued during the cutter compensation and nose R compensation operations, it travels in a travel direction to a location where it is able to stand vertically for the offset vector of the plane axis that was previously selected (for X- and Y-axes when using the G17 modal). The workpiece coordinates, which are used in the G92 command that is issued, are created for the current position in the program path.



(Note 5) When G92 is commanded during tool length compensation, a new workpiece coordinate system will be created where the programmed target value of Z axis coincides with the coordinates commanded by G92.



(Note 6) When the G92 command is issued during the tool position compensation operation, the workpiece coordinates are created and used for the tool teeth in the G92 command that is issued.



(Note 7) When the additional axis is commanded and the optional additional axis is not installed, an alarm will occur.

### 3.7.6 Local Coordinate System Function (G52)

Command format

**G52 X\_ Y\_ Z\_ A\_ B\_ C\_;**

X, Y, Z, A, B, C: Amount of shift from workpiece coordinate zero point

Operation will be the same regardless of G90 or G91.

Amount of shift is applied only to the specified axis.

1. Executing this command creates a local coordinate system in all coordinate systems from G54 to G59.
2. The workpiece coordinate system does not vary even when this command is executed.
3. The local coordinate system of the specified axis is canceled when G92 command is executed.
4. An error will occur when this command is executed during coordinate rotation, scaling or miller imaging
5. When a command is issued during the cutter compensation and nose R compensation operations, it travels to a position where it is able to stand for the vertical vector at the end point for the plane axis that was last selected (for X- and Y-axes when using the G17 modal).
6. The local coordinate system is canceled when any of the following operations are performed:
  - When "0" is specified for the command value on the axis in the G52 command
  - G92 command
  - M02 (M30) command
  - Operation resetting

### 3.7.7 Rotary Fixture Offset Function (G54.2)

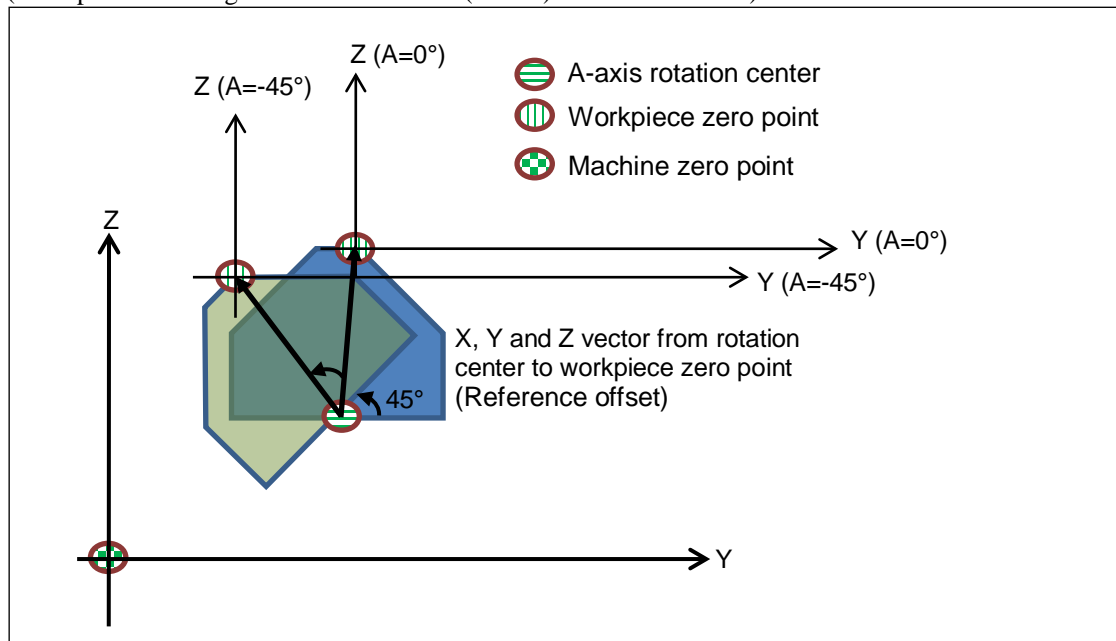
#### 3.7.7.1 Overview

This function is available on machines with an index table.

\* The option for rotary fixture offset function is required.

By setting the rotation center for the index table in the workpiece coordinate system (G54 to G59, G54.1, G92, G52 and external workpiece coordinate system shift) as well as setting the X, Y and Z vector from the rotation center to the workpiece zero point, when the table rotates, this function automatically calculates if the workpiece zero point has moved to another point in the machine coordinates. And, it sets a new workpiece coordinate system based on that position.

(Example when using it on the index table (A-axis) for an X-axis turn)



### 3.7.7.2 Command Format

Rotary fixture offset command

Command format

**G54.2 Pn**

n: Reference rotary fixture offset No. (1 to 8)

Cancel command for rotary fixture offset

Command format

**G54.2 P0**

G54.2 is a modal G code. Specify from 1 to 8 for the G54.2 P address in order to enable the rotary fixture offset function. Specify 0 for the P address to cancel or disable the function.

- (Note 1)** When there is no option for the rotary fixture offset function and the G54.2 command is issued, an alarm is triggered. Also, if the P address is not specified for the G54.2 command, then an alarm is triggered.
- (Note 2)** An alarm is triggered when the G54.2 command is issued in the following situations.
- When the G68/G168 modal is operating (alarm: <<During rotational transformation>>)
  - When the G51 modal is operating (alarm: <<Scaling>>)
  - When the G51.1 modal is operating (alarm: <<Miror image mode>>)
- (Note 3)** The alarm <<Rotary fixture offset in progress>> is triggered when the following commands are issued while the rotary fixture offset function is enabled.
- G68/G168
  - Additional axis command during thread cutting interpolation for helical screw
- (Note 4)** When the rotary fixture offset function is enabled, the coordinate system is not reset by the rotary fixture offset, even if the rotation axis coordinates change due to a command or edit operation. When a G54.2 command or a rotation command is issued from the next block, the coordinate system is reset by the rotary fixture offset.
- Workpiece coordinate system setting (G54 to G59, G54.1, G92, G52 and external workpiece coordiante system shift)
  - G10
  - G53
  - G28/G30
  - Motion from the G29 reference position to the middle position
  - Reference angle for reference rotary fixture offset
- (Note 5)** When simultaneous travel commands are issued for the X-, Y- and Z-axes and for the additional axis while the rotary fixture offset function is enabled, the X-, Y- and Z-axes are offset based on the angle that is specified for the additional axis.
- (Note 6)** The additional axis angle, which corresponds to the turning spindle when the turning spindle is selected, does not take into account the rotary fixture offset calculation. (When the G54.2 command or another travel command on the additional axis is issued, the current rotary fixture offset is the same as when positioning the additional axis, which corresponds to the turning spindle, to the reference angle.)

Program example (NC language)

```
N1 G90 G54 G00 X0 Y0 Z200. A0. C0.
N2 G54.2 P1
N3 G00 A-45.
N4 G01 X2. F1000
N5 G00 A-60. C270.
N6 G54.2 P0
```



### 3.7.8 Feature Coordinate Setting Function

#### 3.7.8.1 Overview

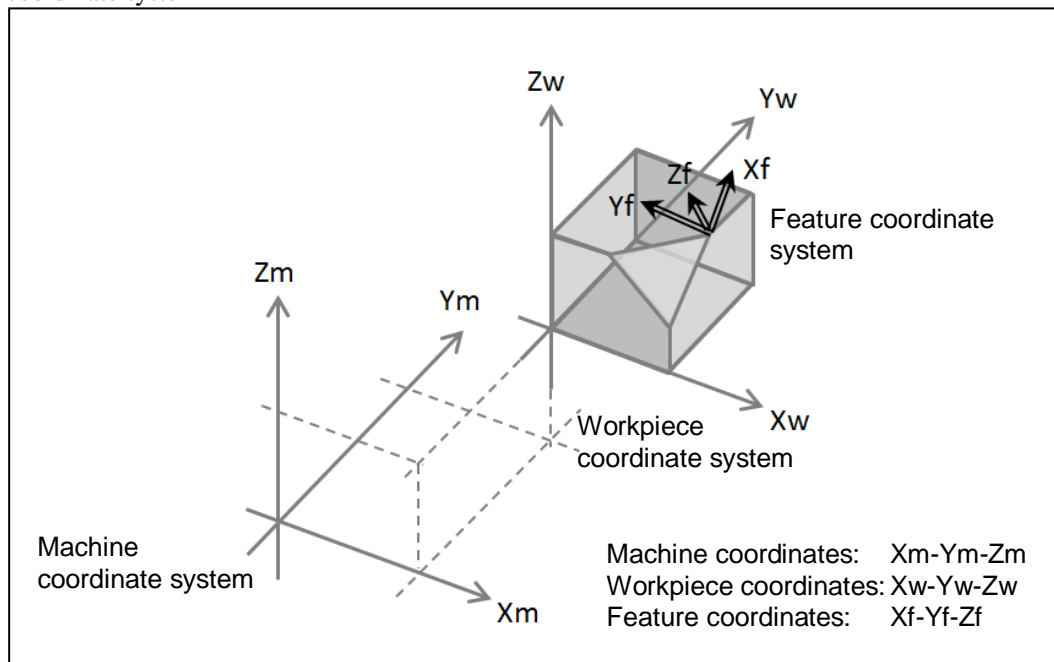
When machining on a tilt plane, set a new coordinate system on this plane to issue position commands in order to make the program simpler.

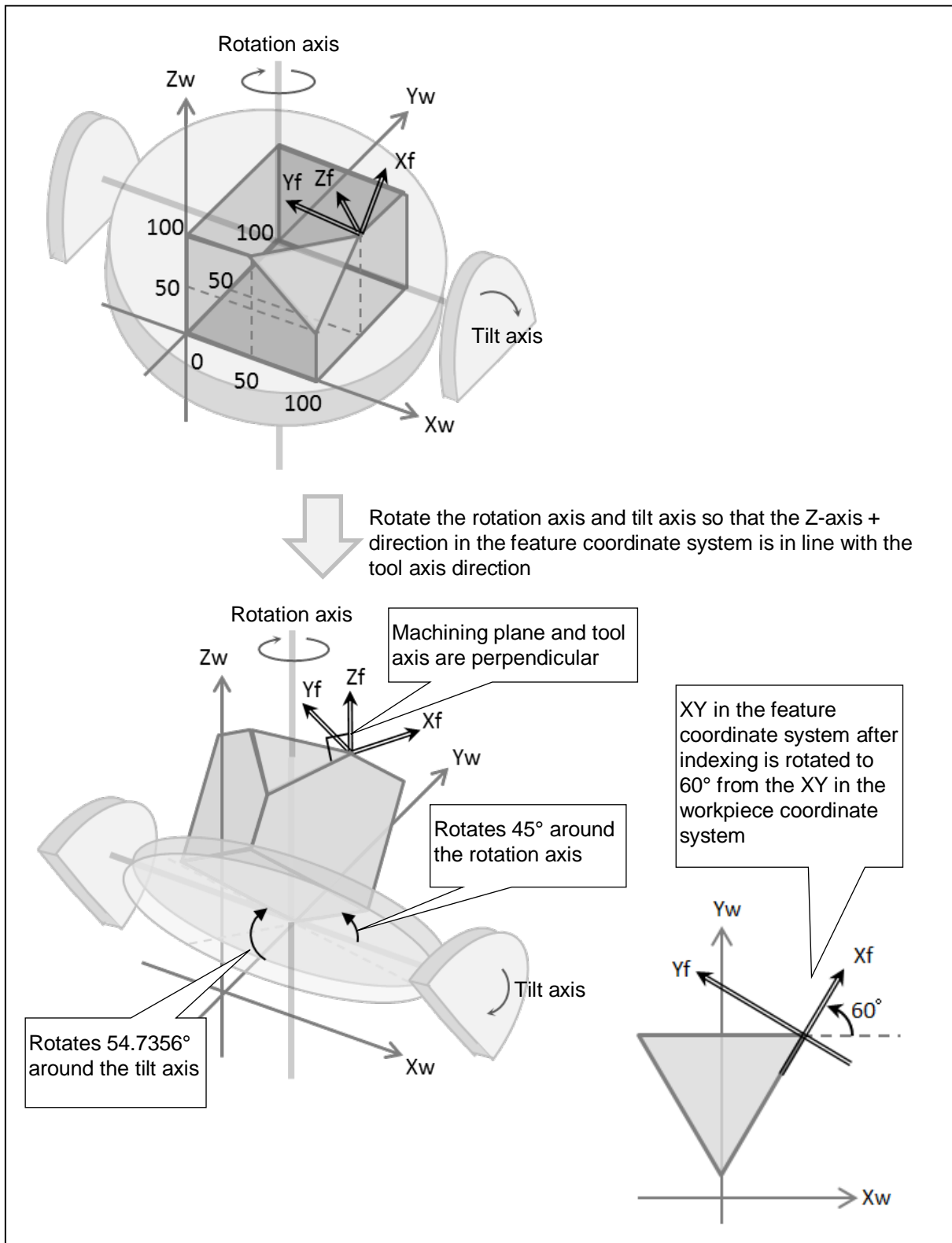
This new coordinate system is called the feature coordinate system and is set in the workpiece coordinate system.

When the feature coordinate system is set, this is called feature coordinate manufacturing mode. The commands are controlled as commands in the feature coordinate system until the feature coordinate manufacturing mode is cancelled.

Feature coordinate index refers to when the additional axis is moved so that the Z-axis + direction in the feature coordinate system is in line with the tool axis direction

Relationship between machine coordinate system, workpiece coordinate system and feature coordinate system





#### Usage conditions

The feature coordinate setting function is an option that is required to enable and use this function. This function can only be used in NC language mode.

### 3.7.8.2 Command format

When setting the feature coordinate system, the feature coordinate manufacturing mode is enabled. The command coordinates are processed as coordinates in the feature coordinate system on the block until the feature coordinate manufacturing mode is cancelled.

1. Feature coordinate setting command (G68.n)  
The feature coordinate system is set.

Do not rotate the additional axis after setting the feature coordinate system. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.

- (1) Feature coordinate setting using Euler angles (G68.2)

Command format

**G68.2 XxYyZzIαJβKγ;**

x, y, z: Feature coordinate zero point

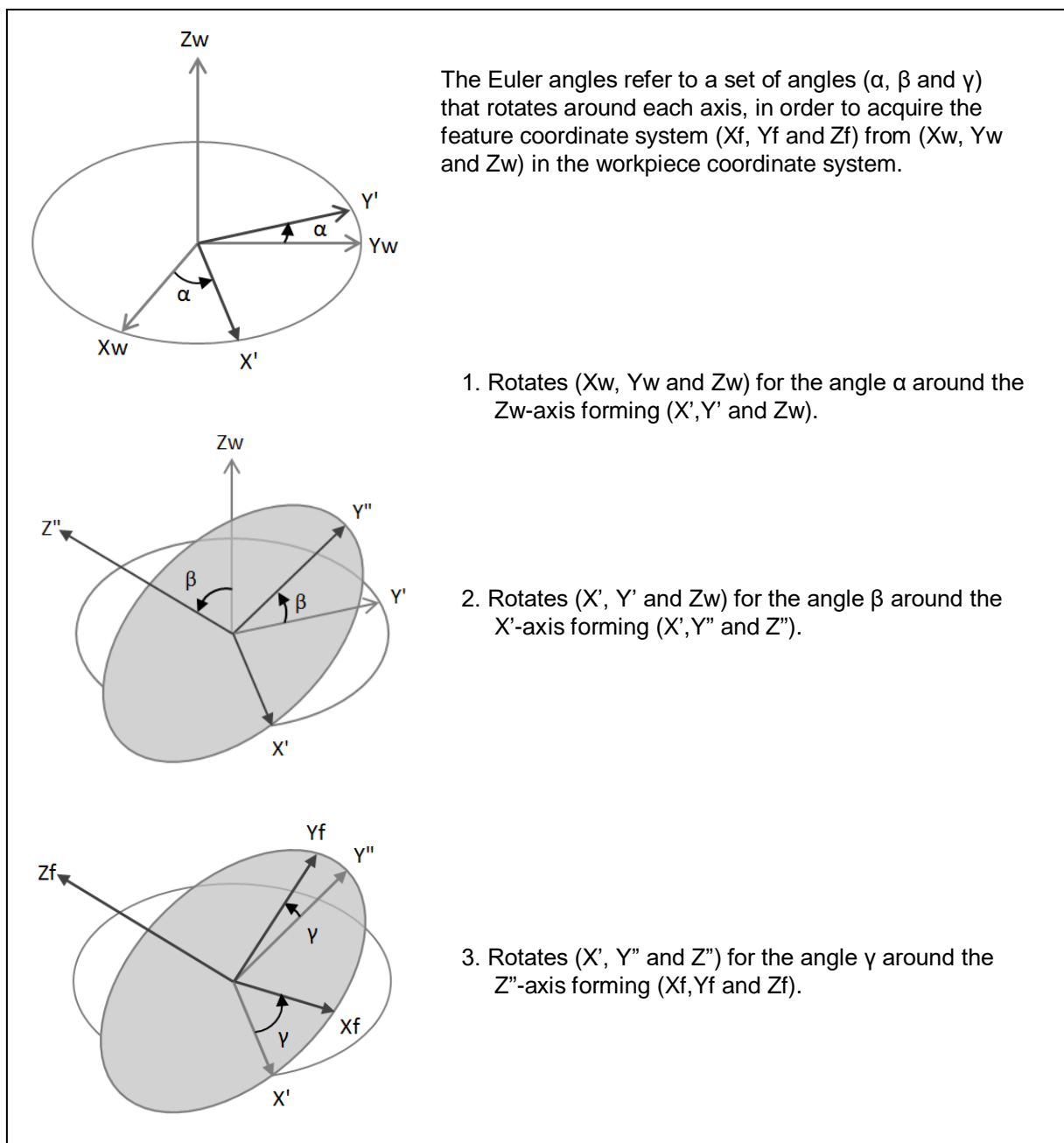
α, β, γ: Euler angles

Specify within the range: -360.000 to 360.000.

When one of the following is omitted: X, Y, Z, I, J or K, the command is processed as a 0 value.

The alarm <<Feature coordinate command error>> is triggered when I, J and K are all omitted.

3



- (2) Multiple absolute commands (G68.2)  
By executing another feature coordinate setting command (G68.2) while in feature coordinate manufacturing mode (and not cancelling feature coordinate manufacturing mode), a new feature coordinate system can be set for the workpiece coordinate system.
- (3) Special notes
  - ( i ) The original workpiece coordinate system cannot be changed while in feature coordinate manufacturing mode. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.
  - ( ii ) When <Tilt axis\*> or <Rotation axis\*> is set in the user parameter (feature coordinate setting) for an axis where the machine parameter (system 2) <Optional \*-axis> is set to <Not equipped>, if a feature coordinate setting command is issued, the alarm <<Parameter set. error in feature coordinate set. function>> is triggered.
  - ( iii ) If a feature coordinate setting command is issued when there is a shift in the local coordinate system (G52), the alarm <<Feature coordinate command error>> is triggered.
  - ( iv ) The feature coordinate setting command can only be executed in memory operation mode. <<Specified G code cannot be used>> is triggered when an attempt to issue a command is made in MDI mode.  
MDI intervention is not possible while in feature coordinate manufacturing mode. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when an attempt is made to change the mode.
  - ( v ) A feature coordinate setting command cannot be issued in an interrupt program. The alarm <<Feature coordinate command error>> is triggered.  
Interrupt type macro cannot be executed while in feature coordinate manufacturing mode. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.
  - ( vi ) A command or operation that requires additional axis travel cannot be executed while in feature coordinate manufacturing mode. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.
  - ( vii ) A feature coordinate setting command can only be executed when XY plane is selected (G17 modal). When a command is issued in another plane besides the XY plane (G18 and G19), the alarm <<Selected plane error>> is triggered.  
While in feature coordinate manufacturing mode, the alarm <<Feature coordinate manufacturing mode engaged>> is triggered when an attempt is made to issue a plane selection command to change to another plane besides the XY plane (G18 and G19 modal).

2. Feature coordinate manufacturing mode cancel command (G69)  
Feature coordinate manufacturing mode is cancelled.

Command format

**G69;**

The axis does not travel with this command, it only cancels the feature coordinate manufacturing mode.

When a modal is enabled for tool length offset, tool position compensation, cutter compensation and canned cycle after the feature coordinate setting, cancel it before this command. The alarm <<Feature coordinate command error>> is triggered if the feature coordinate manufacturing mode is cancelled before cancelling the above modals.

The feature coordinate manufacturing mode is cancelled by resetting operation with M02/M30.

3. Feature coordinate index command (G53.1)  
The additional axis rotates to index the tilt plane so that the Z-axis + direction in the feature coordinate system, set by the feature coordinate setting command, is in line with the tool axis direction.

Command format

**G53.1;**

Use this command individually.

When another GM code command is issued on the same block, the alarm <<Invalid command>> is triggered.

However, this excludes when issuing a simultaneous command with G100/M6, which will be described later.

The additional axis may rotate too much.

Issue a command beforehand to ensure the tool is in a safe position so that the tool and the workpiece do not collide.

Issue a feature coordinate index command while in feature coordinate manufacturing mode. The alarm <<Feature coordinate command error>> is triggered when an attempt is made to issue a feature coordinate index command in a mode other than feature coordinate manufacturing mode.

The additional axis travel is for a positioning operation that does not rely on the modal.

- (1) Simultaneous command with the canned cycle (G100/M6) for tool change  
The feature coordinate index command (G53.1) can be issued only on the same block as the canned cycle (G100/M6) for tool change.

Issue a command with the coordinate values in the feature coordinate system for the X, Y and Z coordinates in G100/M6.

Do not issue a travel command on the additional axis. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.

- (2) How the travel angle on the additional axis is established  
There are normally 2 sets of angles within the 0° to 360° range for the combination of the tilt axis and rotation axis angles when moving the additional axis during the feature coordinate index. When there are two or more sets of angle combinations, which also include angles of integral multiples of 360°, the angle to move the axis is determined by the following methods.

- ( i ) Travel amount of tilt axis is a small angle.
- ( ii ) Travel amount of rotation axis is a small angle when the travel amount in the positive and negative directions is the same for (i).
- ( iii ) Angle of the tilt axis' travel destination is an angle close to a multiple of 360° when the set of angles is not decided by (i) or (ii).
- ( iv ) Angle of the rotation axis' travel destination is an angle close to a multiple of 360° when the set of angles is not decided by (i), (ii) or (iii).
- ( v ) Angle where the tilt axis travels in a positive direction when the set of angles is not decided by (i), (ii), (iii) or (iv).

A short cut is taken when traveling to the established angle. A positive direction is taken when traveling if the travel amount in the positive and negative directions is the same.

When there is an axis that is set as a tilt axis or rotation axis and the user parameter (switch 2) <\*-axis stroke control> is set to <1: Yes>, select an angle within the stroke range. In addition, select an angle where the axis can travel within the stroke range. The alarm <<Feature coordinate command error>> is triggered when an angle cannot be found for the axis to travel. The angle is established using the method above when there are still two or more sets of angles.

Ex: Angle calculated from command is (tilt axis, rotation axis) = (54.7356, 45)(-54.7356, 135)

Case 1

Tilt axis with stroke between -95 and +5, rotation axis without stroke  
Current position (tilt axis, rotation axis) = (30, 0)

Select “-54.7356” where the tilt axis is within the stroke range.

Set of angles (-54.7356, 135) is used because there is no stroke for the rotation axis.

### Case 2

Tilt axis with stroke between -60 and +60, rotation axis with stroke between 0 and 180

Current position (tilt axis, rotation axis) = (30, 0)

Both tilt axis and rotation axis are within the stroke range.

The set of angles (54.7356, 45) is used because there is little travel from the current position of the tilt axis.

### Case 3

Tilt axis with stroke between -800 and +800, rotation axis without stroke

Current position (tilt axis, rotation axis) = (350, 0)

Both tilt axis and rotation axis are within the stroke range.

The travel amount of the tilt axis is  $350 - 54.7356 = 295.2644$  and  $350 - (-54.7356) =$

$404.7356$ , and “54.7356” has less travel. However, “350” is the same position as “-10.”

The actual travel amount is  $(-10) - 54.7356 = -64.7256$  and  $(-10) - (-54.7356) = 44.7356$ .

“-54.7356” has less travel so the set of angles (-54.7356, 135) is used.

### Case 4

Tilt axis with stroke between -60 and +210, rotation axis without stroke

Current position (tilt axis, rotation axis) = (205, 0)

Both tilt axis and rotation axis are within the stroke range.

The shorter travel from the current position on the tilt axis is “305.2644”, the same

position as “-54.7356”. However, the travel goes outside of the stroke range, and

therefore, the set of angles (54.7356, 45) is used.

### Case 5

Tilt axis with stroke between -30 and +120, rotation axis with stroke between -30 and +120

Current position (tilt axis, rotation axis) = (30, 0)

Select “-54.7356” where the tilt axis is within the stroke range.

The alarm <<Feature coordinate command error>> is triggered because the “135” that makes the pair is outside of the rotation axis stroke range and there is no other set of angles that can be used.

### 3.7.8.3 Restrictions

1. Simultaneous use with other functions
  - A setting command for the feature coordinate system is not possible when the cutter compensation is ON. The alarm <<Feature coordinate command error>> is triggered.
  - Rotational transformation, rotary fixture offset, scaling and mirror image commands are not possible while in feature coordinate manufacturing mode. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.
  - After the feature coordinate setting, the following commands cannot be used before the feature coordinate index. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered.
    - Cutter compensation
    - Canned cycle
    - Corner R
    - Circular interpolation, helical thread cutting interpolation
    - Circular cutting
    - Involute interpolation

Refer to “3.16 G Code Priority” for further details.

2. When lathe spindle is selected
 

The feature coordinate setting command cannot be issued when the lathe spindle is (M142 modal in progress) selected.

The alarm <<A command unable to select the lathe spindle>> is triggered.

In addition, if the detection signal is ON when travel starts, it does not operate.

After changing from the lathe spindle (M142) to the spindle (M141), the feature coordinate setting command cannot be issued even when the following commands are issued to the additional axis that is assigned as the lathe spindle. The alarm << Feature coordinate command error >> is triggered.

  - Axis travel command for G90 modal
  - Reference position return command (G28, G30) when there is no travel to the middle position

The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when the lathe spindle (M142) is selected after the feature coordinate setting.
3. Special notes
 

This function is a coordinate conversion function and does not offer compensation for indexing accuracy that is specific to each machine.

## 3.8 Soft Limit

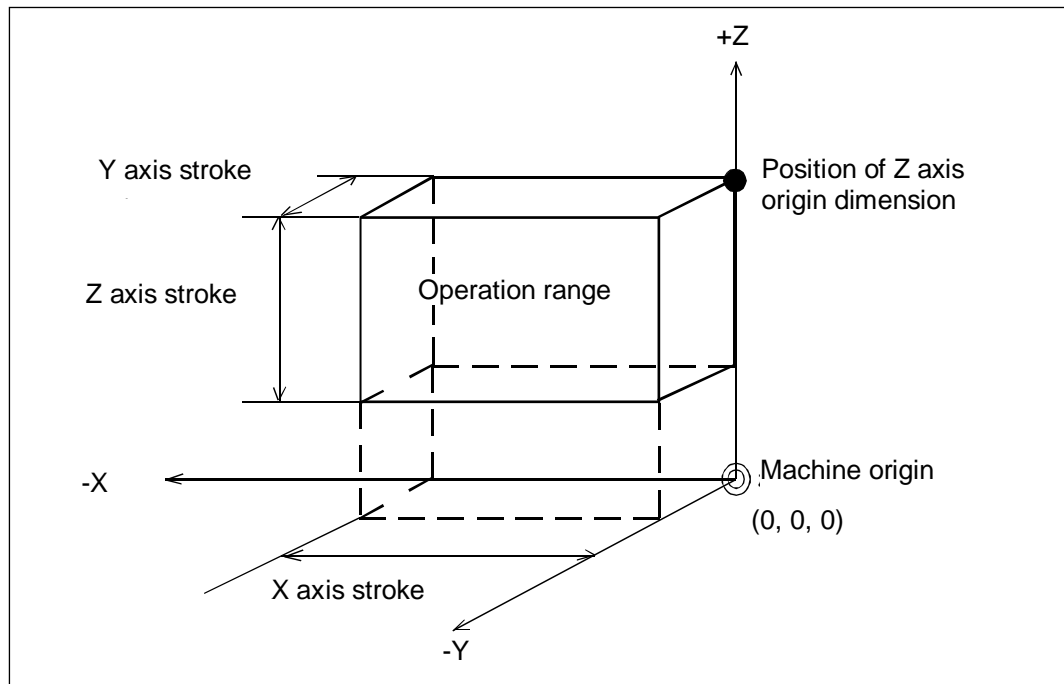
Tool operation range is set by software.

Tool operation range is specified by one of the three methods below.

1. Set strokes by machine parameters.
2. Set stroke limit by user parameters.
3. Set programmable stroke limit by G22 code

### 3.8.1 Stroke

The maximum stroke of the machine has been set at the factory using <Stroke 1X to Z axis (+/-)> of <Machine Parameters>. Do not change it.



**(Note)** Z axis origin dimension is set by <Distance to Z-axis zero> of <Machine Parameters>.

### 3.8.2 Stroke Limit

<Stroke limit 1X to Z axis (+/-)> of <User Parameters> is used to set operation range for X, Y, and Z axis, respectively.



### 3.8.3 Programmable Stroke Limit (G22)

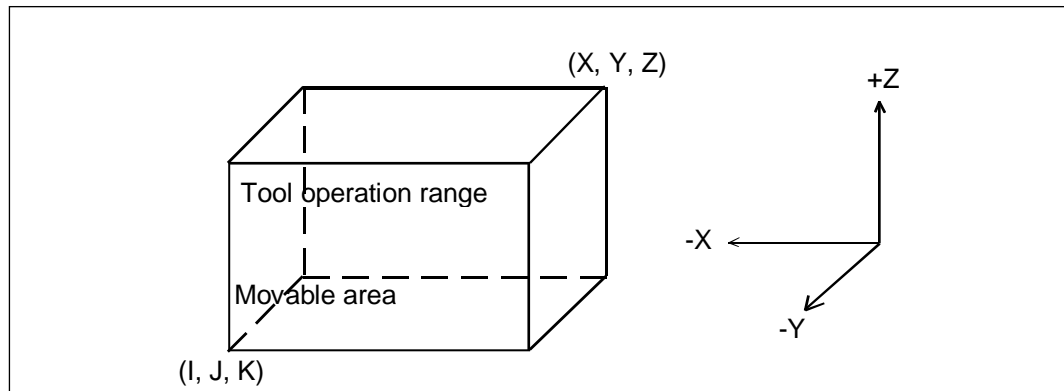
Tool operation range is commanded by program.

Command format

**G22 X\_ Y\_ Z\_ I\_ J\_ K\_;**

- X : Programmable stroke limit for X axis in + direction
- Y : Programmable stroke limit for Y axis in + direction
- Z : Programmable stroke limit for Z axis in + direction
- I : Programmable stroke limit for X axis in - direction
- J : Programmable stroke limit for Y axis in - direction
- K : Programmable stroke limit for Z axis in - direction

These are commanded by coordinate values of the machine coordinate system.  
Absolute values are used irrespective of G90 and G91.



**(Note 1) Selection criterion for using stroke or programmable stroke as soft limit on the program:**

- G22 ---- Check using programmable stroke limit as soft limit
- G23 ---- Check using stroke as soft limit

**(Note 2) Stroke limit of User Parameters is effective immediately after starting up the machine. Then, change of User Parameters and commanding by G22, whichever takes place later, is effective.**  
Stroke limit of User Parameters is recognized as command value for those axes that are not specified by G22. When you change the stroke limit of User Parameters, however, the User Parameters' value applies to all axes including those that are not changed.

**(Note 3) Stroke by machine parameters is always effective.**

## 3.9 Reference Position (G28 to G30)

### 3.9.1 Return to the Reference Point (G28)

Command format

G28 X_ Y_ Z_ A_ B_ C_;
------------------------

The axis returns to the reference point via an intermediate point.

X\_ Y\_ Z\_ A\_ B\_ C is a travel command to intermediate points, given in absolute (G90) or incremental (G91) values.

Coordinates of the intermediate point for the axis commanded in this block are memorized.

The axis goes to the intermediate point by rapid feed, and then moves to the reference point also by rapid feed.

- (Note 1) Intermediate point coordinates are memorized for only the axis for which travel is specified in the G28 block.  
For the axes not specified in the G28 block, G28 intermediate point coordinates defined previously will be used as they are.
- (Note 2) Reference points are set by <Reference Points X to Z / Axes 4 to 8> of <User Parameters>.
- (Note 3) Travel to intermediate and reference point is a positioning operation. No interpolation is performed.
- (Note 4) The tool stops at the intermediate point in a single block operation.
- (Note 5) Coordinate values of intermediate points are memorized in absolute values of workpiece coordinate system. When workpiece coordinate system is changed after commanding a G28, the intermediate point also moves to the new workpiece coordinate system.
- (Note 6) An alarm occurs when an additional axis is commanded in the absence of the optional additional axis.
- (Note 7) When a command is issued during the tool length offset, the tool length offset stays enabled while traveling to the middle point. When travelling to the reference position, the tool length offset is cancelled temporarily. The same also applies during tool position compensation.

### 3.9.2 Return from the Reference Point (G29)

Command format

G29 X_ Y_ Z_ A_ B_ C_;
------------------------

Axes move to the specified position via intermediate point. In the case of incremental commands, motion from the intermediate point is given in an incremental value. The specified axis goes to the intermediate point by rapid feed and then to the final position.

- (Note 1) Travel to intermediate and reference point is a positioning operation. No interpolation is performed.
- (Note 2) Axes pass the intermediate point commanded by G28 or G30 whichever is the later.
- (Note 3) The tool stops at the intermediate point in a single block operation.
- (Note 4) For the axes which intermediate point is not memorized by G28 or G30, the current position is used as the intermediate point.
- (Note 5) An alarm occurs when an additional axis is commanded in the absence of the optional additional axis.

### 3.9.3 Return to the 2<sup>nd</sup> to 6<sup>th</sup> Reference Point (G30)

Command format

**G30 P\_ X\_ Y\_ Z\_ A\_ B\_ C\_;**

- P2 : Return to the 2<sup>nd</sup> reference point
- P3 : Return to the 3<sup>rd</sup> reference point
- P4 : Return to the 4<sup>th</sup> reference point
- P5 : Return to the 5<sup>th</sup> reference point
- P6 : Return to the 6<sup>th</sup> reference point

G30 is the same as G28 except that the axes return to the 2<sup>nd</sup> through 6<sup>th</sup> reference points.  
You can use G29 for G30 in the same way as you use it for G28.

- (Note 1)** The 2nd through 6th reference points are set by <2nd through 6th Reference Points X through Z / Axes 4 through 8> of <User Parameters>.
- (Note 2)** Omit P\_, and return to the 2nd reference point is selected.
- (Note 3)** An alarm occurs when an additional axis is commanded in the absence of the optional additional axis.

## 3.10 Skip Function (G31, G131/G132)

### 3.10.1 Before using the skip function

Set the following item in the “User parameter (switch 1)”.

Item name	Setting range	Description
Probe detection signal method	0: Method 1 1: Method 2	Set the accuracy of the detection signal that is obtained from the touch probe. Even if the <1: Method 2> increases the travel speed of the probe more than <0: Method 1>, there is less variation in the measurement values. As a result, the speed can be set higher than <Automatic centering speed 2> and <Measurement speed 2>. However, <1: Method 2> cannot be set due to the machine structure, and an alarm is triggered.

### 3.10.2 Skip Function (G31, G131/G132)

The tool moves linearly (linear interpolation) at the specified feed rate from the current position to the target position or until the detection signal turns ON.

Command format

```
G31   X_ Y_ Z_ F_;
G131  X_ Y_ Z_ F_;
G132  X_ Y_ Z_ F_;
```

Up to three linear axes (X,Y,Z) can be controlled simultaneously.

The feedrate is set by address F. Once the feed rate is commanded, it is effective until another value is specified.

For G131, the <<SENSOR SIGNAL OFF>> alarm occurs when the tool has moved to the target position without the detection signal turning ON. For G31, G132, an alarm does not occur.

When using G31, after the detection signal turns ON, it decelerates and then stops. When using G131/G132, after the detection signal turns ON, it decelerates and stops. Then, it returns back to the same coordinate where the detection signal turned ON.

As the coordinate value when detection signal turns ON is stored in system variables (#5061 ~ #5068, #5071 ~ #5074) of the macro, it can be used in the macro.

In feature coordinate manufacturing mode, the coordinates in the feature coordinate system are stored in the system variables: #5161 to #5168 and #5171 to #5174.

- (Note 1) The alarm <<Compensating diameter>> is triggered when cutter compensation mode is enabled.
- (Note 2) The tool does not move during a dry run state.
- (Note 3) The tool moves to the target position during a machine lock state.
- (Note 4) When the detection signal is already ON, the operation is not performed.
- (Note 5) The appropriate feedrate varies depending on the probe being used. Therefore, contact the probe manufacturer when deciding on the rate.
- (Note 6) The “Coordinate when the detection signal turns ON” is acquired by this function, and the true value varies depending on factors such as the delay that is unique to the probe. Therefore, contact the probe manufacturer and adjust it accordingly.
- (Note 7) Make sure that no chips or shavings are stuck to the end of the measurement probe or on the measurement surface. In addition, make sure that there is no disturbance (caused by vibrations from outside of the machine) that adversely affects the machine. We cannot guarantee the measurement accuracy when there are chips or other factors that can cause inaccuracy.
- (Note 8) While in the inverse time feed (G93) modal, the alarm <<Command not possible during inverse time feed>> is triggered.

### 3.10.3 Continuous Skip Function (G31)

The tool moves linearly (linear interpolation) at the specified feedrate from the current position to the target position. If the detection signal turns ON in the meantime, the coordinate value when the detection signal turns ON is stored in the system variables (#5061~#5068, #5071~#5074) of macro. In feature coordinate manufacturing mode, the coordinates in the feature coordinate system are stored in the system variables: #5161 to #5168 and #5171 to #5174.

Command format

<b>G31 P90</b>	<b>X_F_;</b>
<b>G31 P90</b>	<b>Y_F_;</b>
<b>G31 P90</b>	<b>Z_F_;</b>

- (Note 1) The alarm <<Compensating diameter>> is triggered when cutter compensation mode is enabled.
- (Note 2) The tool does not move during a dry run state.
- (Note 3) The tool moves to the target position during a machine lock state.
- (Note 4) The appropriate feedrate varies depending on the probe being used. Therefore, contact the probe manufacturer when deciding on the rate.
- (Note 5) The “Coordinate when the detection signal turns ON” is acquired by this function, and the true value varies depending on factors such as the delay that is unique to the probe. Therefore, contact the probe manufacturer and adjust it accordingly.
- (Note 6) Make sure that no chips or shavings are stuck to the end of the measurement probe or on the measurement surface. In addition, make sure that there is no disturbance (caused by vibrations from outside of the machine) that adversely affects the machine. We cannot guarantee the measurement accuracy when there are chips or other factors that can cause inaccuracy.
- (Note 7) While in the inverse time feed (G93) modal, the alarm <<Command not possible during inverse time feed>> is triggered.

## 3.11 Scaling (G50/G51)

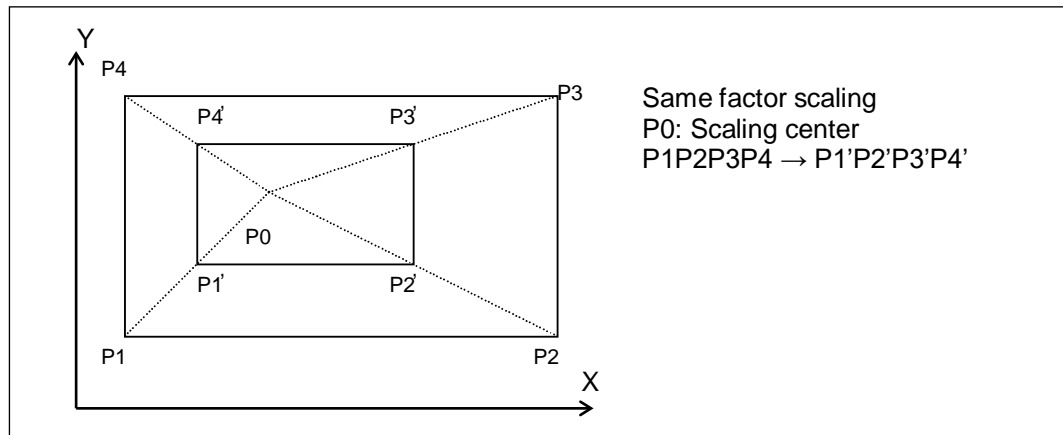
The programmed shape can be enlarged or reduced by the desired scaling factor. Scaling is possible using the same ratio for all axes or a different ratio for each axis.

Scaling using the same ratio for all axes:

Command format

**G51 X\_ Y\_ Z\_ P\_;**

X, Y, Z : Scaling center coordinate axes (workpiece coordinates)  
P : Scaling factor

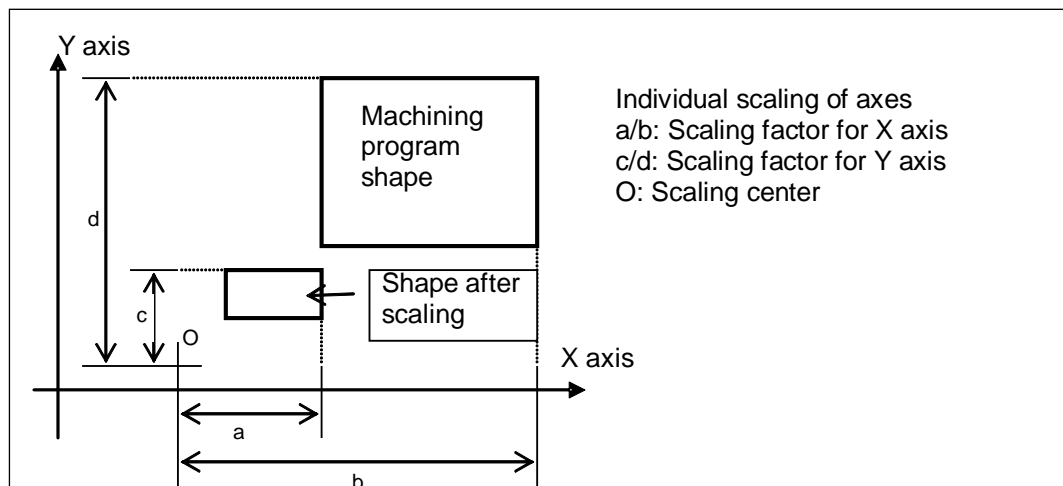


Scaling using a different ratio for each axis

Command format

**G51 X\_ Y\_ Z\_ I\_ J\_ K\_;**

X, Y, Z : Scaling center coordinate axes (workpiece coordinates)  
I, J, K : Scaling factors for X, Y, Z axes



Scaling cancel:

Command format

**G50;**

The following user parameters are used in scaling:

1. Scaling factor (same or individual scaling of axes) is specified by <Scaling>.
2. Set value of <Scaling Factor> is used when the scaling factor command (P or IJK) is omitted.
3. Scaling factor is set in the unit of 0.001 or 0.00001 according to the Unit of Scaling Factor. The range of scaling factor command (P or IJK) or scaling factor parameters is  $\pm 1$  to  $\pm 999999$ . Accordingly, the valid scaling range is  $\pm 0.001$  to  $\pm 999.999$  or  $\pm 0.00001$  to  $\pm 9.99999$ .

**Precautions for use of scaling function:**

(Note 1) Do not use other GM codes in a bloc. The alarm <<Invalid Command>> will occur.

(Note 2) When scaling center coordinates (XYZ) is omitted, tool position at G51 command is used as the center coordinates.

(Note 3) Scaling ON (G51) and Scaling OFF (G50) do not entail axis motion.

(Note 4) Setting a different scaling ratio for each axis in circular interpolation mode does not result in elliptical interpolation.

(Note 5) When a different scaling ratio is set for each axis and radius R of the arc is specified in circular interpolation, the diameter will be related to the axis of the greater scaling factor on the plane where the arc lies.

Ex) Arc using command "R": The left and right command formats are equivalent.

G90 G00 X0.Y100.; G51 X0.Y0.Z0.I2000J1000; G02 X100.Y0.R100.F500;	=	G90 G00 X0.Y100.; G02 X200.Y0.R200.F500;
---	---	---

(Note 6) When a different scaling ratio is set for each axis and the center (I, J) of the arc is specified in circular interpolation mode, the distance from the start point to the center (I, J) is not subject to scaling.

Ex) Arc using commands "I" "J": The left and right command formats are equivalent.

G90 G00 X0.Y100.; G51 X0.Y0.I2000J1000; G02 100.Y0.I0.J-100.F500;	=	G90 G00 X0.Y100.; G02 X200.Y0.I0.J-100.F500;
---	---	---

(Note 7) When scaling is invalid

The scaling command does not affect the following items:

1. Cutter compensation during the scaling setting operation, tool length offset, nose R compensation and tool offset for the tool position compensation
2. Additional axis
3. Travel amounts in manual intervention
4. The following motion in canned cycle:
  - infeed amount "Q" and relief amount "d" of deep hole cycle (G83, G73, G173, G183)
  - XY-axes shift "Q" of fine balling (G76) and back balling (G87).

The alarm <<Scaling>> occurs when a canned cycle is executed with Z axis specified for scaling.

(Note 8) Traveling axes when performing scaling or programmable mirror image: Irrelevant axes may move during scaling and programmable mirror imaging depending on which axes and/or coordinates are commanded. If this occurs, operation may stop due to lock signal check of non-specified axes or Z axis may move due to automatic triggering of dry run offset. An alarm can also occur due to restriction of commandable axis.

(Note 9) Cases when an alarm will occur:

1. The alarm <<Scaling>> occurs when the following instructions are given during scaling setting:
  - Reference-related commands (G28 to G30) are specified
  - When a coordinate change command (G10L2/20/98/99, G22 to G23, G52 to G59, G92 and external workpiece coordinate system shift) is specified
  - Single direction positioning (G60) is specified
  - Auto workpiece measurement commands (G120 to G129) are specified
  - When a tool change, XZ or YZ arc (G102/103, 202/203), circular cutting spiral interpolation, conical interpolation or involute interpolation (G02.2/G03.2) command is carried out
  - Corner C, R is specified
  - Circle cutting (G12, 13) is specified
  - A canned cycle is specified during setting of Z axis scaling
  - When a thread cutting (G33, G392 and G376) command is issued
  - When a feature coordinate setting command (G68.2) is issued

2. The alarm <<Scaling Address Error>> occurs when scaling is commanded to the axis which has been set to <0: Scaling Invalid> in <X/Y/Z axes Scaling> of User parameter (Switch 1).
3. The alarm <<Feed Rate Error>> occurs when a dry run is specified immediately before activation of circular interpolation command where X-Y axes travel amount becomes zero in the G17 modal mode due to scaling.
4. The alarm <<Specified G Code Cannot Be Used>> occurs when scaling is specified in MDI operation.
5. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when a scaling command is issued while in feature coordinate manufacturing mode (G68.2 modal in progress).

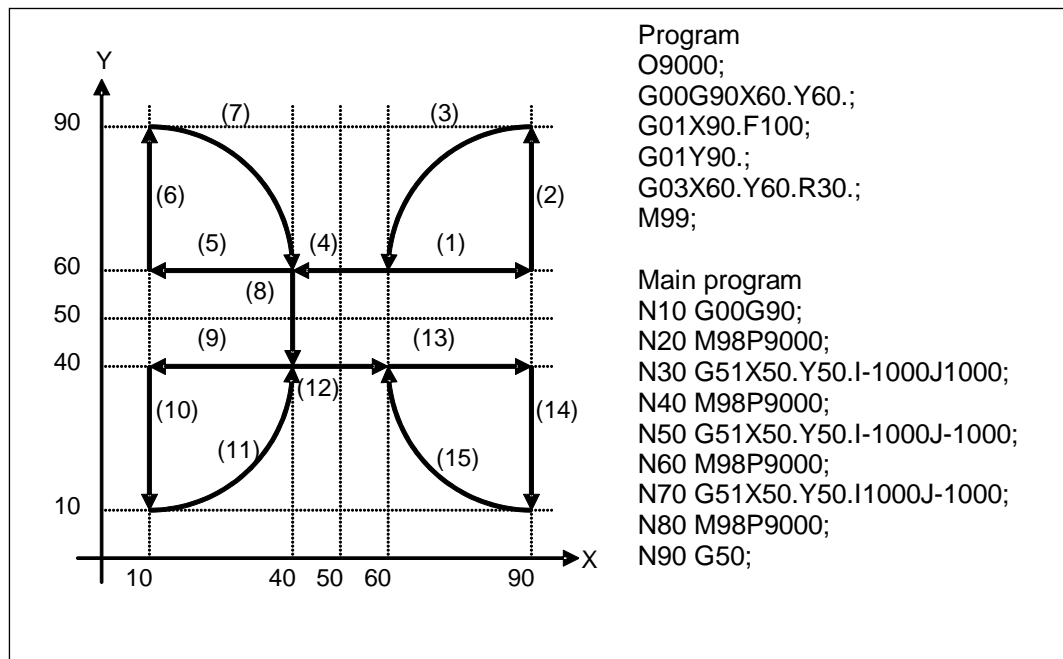
(Note 10) Scaling is cancelled when M02 or M30 is used or operation is reset.

(Note 11) Scaling is executed in the order of mirror, scaling, and coordinate rotation. Therefore, specify them in the order to designate them in a program. To cancel scaling, specify them in the reverse order. If the order is incorrect, the alarm <<scaling>> or <<During coordinate rotation>> occurs.

(Note 12) When scaling while in the inverse time feed (G93) modal, the feedrate is calculated based on the travel distance before scaling.

Program example of mirror image using scaling function:

When a negative number is specified for the scaling factor, programmable mirror image is applied. When a negative value is specified for the scaling factor and there is only one scaling axis, CW and CCW of circular travel will be reversed.



Do not use the first feed rate command for circular interpolation or helical screw cut interpolation (G02, G03), after commanded by mirror image of scaling. When use it, positioning error occurred between start point, end point and center point that cause of distortion in the circular arc.

Mirror image is applied to scaling center coordinates and programmed path while the mirror image (G51.1) is valid.



## 3.12 Programmable Mirror Image (G50.1/51.1)

Mirror image is applied to the program commands for the axes specified in the program.

Mirror image

Command format **G51.1 X\_ Y\_ Z\_;**

Mirror image cancel

Command format **G50.1 X\_ Y\_ Z\_;**

Mirror image setting can be applied simultaneously for the 1st to 3rd axes.

Set the mirror image axis in workpiece coordinates.

Set the mirror image axis. Omit this for axes about which a mirror image is not created.

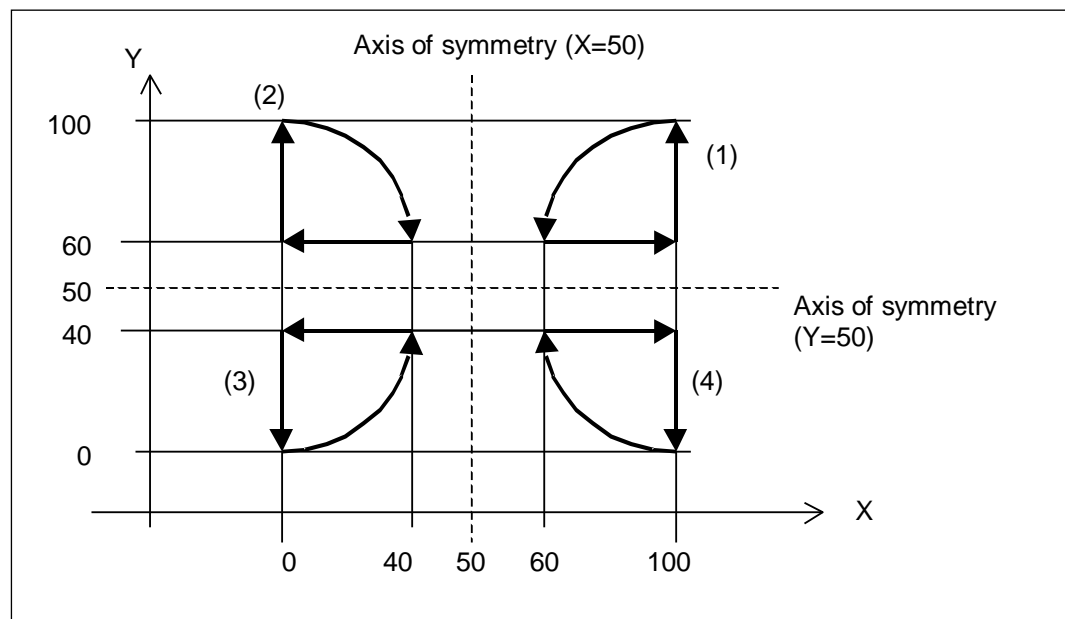
Using G50.1 command is valid while setting a mirror image.

Coordinate values are arbitrarily set.

**(Note 1) Using G51.1 command is valid while setting a mirror image. It is regarded as an addition of mirror axes or a change of the mirror axis coordinates.**

**(Note 2) Mirror cancel command for the axes that are not mirror-designated will not invoke an alarm.**

An example of programmable mirror imaging



- (1) Original program command
- (2) Mirror axis is set for position X50.
- (3) Mirror axis is set for position X50, Y50.
- (4) Mirror axis is set for position Y50.

**Precautions for use of programmable mirror image****(Note 1)** When programmable mirror image is invalid:

1. The tool length offset and the tool position compensation do not apply to the mirror setting compensation.
2. The spindle rotation direction does not change during mirror image setting.
3. The thread cutting direction does not change during mirror image setting.
4. Manual intervention allows the axis travel while ignoring the mirror image setting. Note that, however, axis travel by manual intervention during mirror imaging is applicable to mirror image-processed tool path.

**(Note 2)** Traveling axes when performing scaling or programmable mirror image: Irrelevant axes may move during scaling and programmable mirror imaging depending on which axes and/or coordinates are commanded. If this occurs, operation may stop due to lock signal check of non-specified axes or Z axis may move due to automatic triggering of dry run offset. An alarm can also occur due to restriction of commandable axes depending on specifications.

**(Note 3)** Cases when an alarm will occur:

1. The alarm <<Mirror image mode>> occurs when the following instructions are given during mirror setting:
  - Reference-related commands (G28 to G30) are specified
  - When a coordinate change command (G10L2/20/98/99, G22 to G23, G52 to G59, G92 and external workpiece coordinate system shift) is specified
  - Single direction positioning (G60) is specified
  - Auto workpiece measurement (G120 to G129, etc.) is specified
  - Skip function (G31, G131, G132) is specified.
  - When a tool change, XY or YZ arc (G102/103, G202/203), circular cutting spiral interpolation, conical interpolation, involute interpolation (G02.2/G03.2), circular cutting or coordinate calculation command (G36 to G39) is carried out
  - A canned cycle is specified during setting of Z axis mirror imaging
  - When a thread cutting (G33, G392 and G376) command is issued
  - When a feature coordinate setting command (G68.2) is issued
2. The alarm <<Scaling>> or Rotational Transformation Going On occurs when mirror image commands (G50.1, G51.1) are specified during scaling or coordinate rotation.
3. The alarm <<Specified G Code Cannot Be Used>> occurs when mirror imaging is specified in MDI operation.
4. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when a mirror command (G50.1, G51.1) is issued while in feature coordinate manufacturing mode (G68.2 modal in progress).

**(Note 4)** Mirror image is cancelled when M02 or M30 is used or operation is reset.

**(Note 5)** Do not use a circular interpolation, an involute interpolation (G02.2/G03.2), or a helical thread cutting interpolation (G02, G03) command for the first travel command after a mirror image command. Positioning error will occur among start point, end point, and center, resulting in distortion of the arc.

Transformation of Programmable Mirror Image:

Coordinates are calculated according to the following sequence: mirror, scaling, and then rotational transformation. Accordingly, set these in this order in a program. Set these in the reverse order to cancel previous settings. The alarm <<Scaling>> or Rotational Transformation Going On occurs if this order is not observed.

When mirror image is set for only one axis on the selected plane, change the following commands:

Circular interpolation	:	Rotation direction
Tool dia. Offset	:	Compensation direction
Nose R compensation	:	Compensation direction
Coordinate rotation function	:	Rotation direction
Circle cutting	:	Rotation direction

While the mirror image function is enabled, the stroke limit is checked using the coordinates after the mirror image is created.

The axis does not travel while setting or canceling a mirror image.

## 3.13 Rotational Transformation Function (G68/69, 168)

### 3.13.1 Coordinate Rotation (G68/69)

The shape specified in the program is rotated.

Coordinate rotation function

Command format

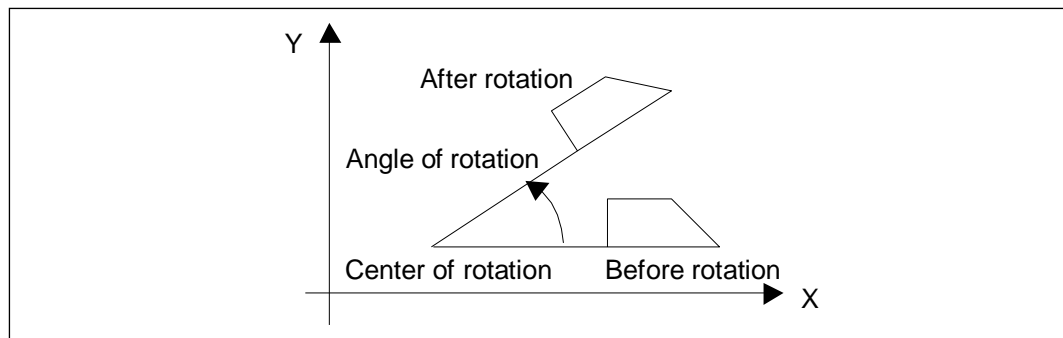
G17  
G18  
G19  
G68  $\alpha$ \_ $\beta$ \_R\_;

Coordinate rotation function cancel

Command format

G69;

- $\alpha$ ,  $\beta$  : Rotation center coordinates  
Recognize coordinates consistently that commanded absolute value.  
When omit it, position G69 to G68 is a center.
- R : Rotation angle (based on CCW))  
Selectable between -360.000 and 360.0000.  
Command "R" cannot be omitted. The alarm <<Invalid data>> occurs.



Plane section command can be omitted. The plane currently selected is valid when it is omitted.  
Relationship between selected plane and  $\alpha\beta$ .

Selected plane	$\alpha$	$\beta$
G17	X	Y
G18	Z	X
G19	Y	Z

The rotation angle in incremental programming mode is determined in reference to the angle after the previous rotational transformation, and in reference to the  $\alpha$  axis when it is the first rotational transformation.

#### Precautions for using coordinate rotation:

- (Note 1) When the center coordinates are omitted for rotational transformation, the coordinates of the spindle's current position are regarded as the rotation center coordinates.
- (Note 2) The rotation angle of the rotational transformation is not subject to scaling.
- (Note 3) When the rotational transformation command is used while the mirror image and scaling functions are valid, calculation is performed according to the following sequence:
1. Change of rotational transformation center coordinates due to mirror image function
  2. Change of rotation angle direction for rotational transformation when there is only one mirror axis
  3. Rewrite of center coordinates of coordinate rotation by scaling

**(Note 4) Cases when an alarm will occur**

1. The alarm <<During Rotational Transformation>> occurs when the following is performed during coordinate rotation:
  - Reference point return-related commands (G28 to G30) are specified
  - Local coordinate setting (G52) or workpiece coordinate system setting (G92) is specified
  - Auto workpiece measurement commands (G131, G132, G120 to G129) are specified
  - Plane selection commands (G17, G18, G19) are specified
  - Compensation commands for linear axes (X, Y, Z axes) and rotation axes (A, B, C axes) are specified simultaneously
  - When a thread cutting (G33, G392 and G376) command is issued
  - When a feature coordinate setting command (G68.2) is issued
2. The alarm <<Selected Plane Error>> occurs when the selected plane and the center axis of coordinate rotation are contradictory to each other
3. The alarm <<Specified G Code Cannot Be Used>> occurs when coordinate rotation is commanded in MDI operation
4. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when a rotational transformation command is issued while in feature coordinate manufacturing mode (G68.2 modal in progress).

**(Note 5)** Rotational transformation is cancelled when M02/M03 is used or operation is reset.

**(Note 6)** Coordinates are calculated according to the following sequence: mirror, scaling, and then rotational transformation. Set these in this order to write a program. To cancel the previous setting, set these in reverse order. The alarm <<Scaling>> or Rotational Transformation Going On occurs if this order is not observed.

### 3.13.2 Coordinate Rotation Using Measured Results (G168)

Command format

**G168 X\_ Y\_ Q\_;**

- X, Y : Rotation center coordinate value  
 Q : Selects the desired measured result by setting "1" to "4".  
 When the selection is omitted, the setting is considered to be "1".

Recognize coordinates consistently that commanded absolute value.

When this setting is omitted, the position in which the block has shifted from G69 to G168 (or G68) is considered the center.

The coordinate is rotated using the angle obtained from the measurement.  
 Other features are the same as those for the coordinate rotation function.  
 The shape specified in the program is rotated.

The rotation <angle> of the rotational transformation is not subject to scaling.

Rotational transformation is cancelled when M02 or M30 is used or operation is reset.

When the center coordinates are omitted for rotational transformation, the coordinates of the spindle's current position are regarded as the rotation center coordinates.

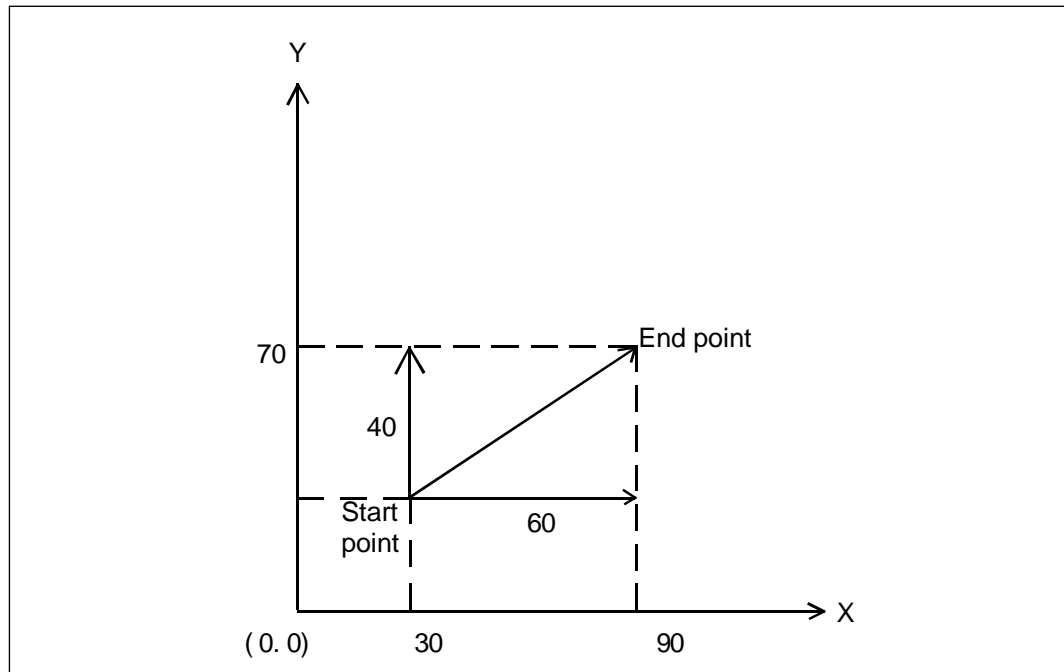
Rotational transformation may be commanded during a rotational transformation.

Coordinates are calculated according to the following sequence: mirror, scaling, and then rotational transformation. Accordingly, set these in this order in a program. Set these in the reverse order to cancel previous settings. An alarm will occur when the specified sequence is not followed.

## 3.14 Absolute Command and Incremental Command (G90/G91)

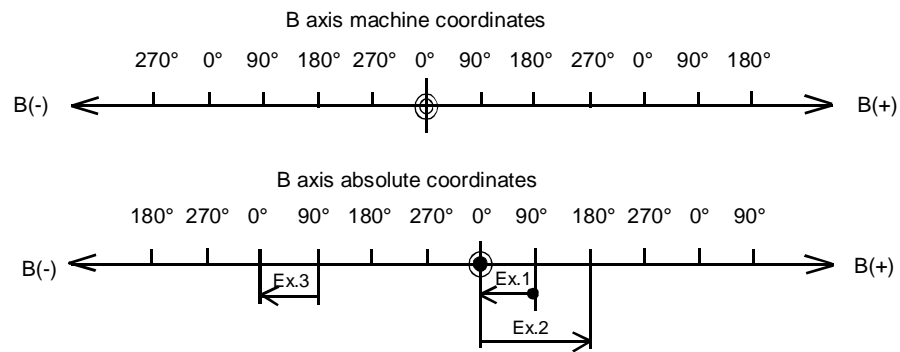
The axis movement amount can be specified by either the absolute command or the incremental command.

1. Absolute command (G90)  
G90 code specifies the end point of a block using coordinate values of the workpiece coordinate system.
2. Incremental command (G91)  
The axis movement amount can be specified by either the absolute command or the incremental command.
  - Absolute command  
G90 X90 Y70;
  - Incremental command  
G91 X60 Y40;



3. Additional axis command
  - (1) Absolute command (Example: B axis)
    - <Bth-axis stroke control> of <user parameter> is set to <1: Yes>, the B axis rotates to the commanded angle.
    - <Bth-axis stroke control> of <user parameter> is set to <0: No>, the B axis rotates in the direction closer to the commanded angle.  
When the turning angle is the same in either positive or negative direction (e.g., 180°), the axis rotates toward positive.
    - <Bth-axis stroke control> of <user parameter> is set to <0: No>, even a larger angle than 360 degrees is commanded, this is handled within 360 degrees.

When B STROKE is set to 0: NO



Ex. 1: Enter B 0.000 to rotate B axis 90 deg in negative direction  
 Ex. 2: Enter B 180.000 to rotate B axis 180 deg in positive direction.  
 Ex. 3: Enter B 0.000 to rotate B axis 90 deg in negative direction.

- B axis machine origin
- B axis machining origin  
(set to 90 deg in this example)
- B axis current position (angle) before travel

(2) Incremental command (Ex. B axis)

Regardless of the setting of <Bth-axis stroke control (1: Yes or 0: No)> of user parameter, the axis rotates for the commanded angle.

However, <Bth-axis stroke control> of user parameter is set to <1: Yes>, STROKE OVER or LIMIT OVER alarm may occur due to stroke and stroke limit control.

## 3.15 Change of Tap Twisting Direction (G133/134)

Command format

<div style="display: inline-block; vertical-align: middle;"> <div style="border: 1px solid black; padding: 5px; margin-bottom: 5px;">G133</div> <div style="border: 1px solid black; padding: 5px;">G134</div> </div> <div style="display: inline-block; vertical-align: middle; margin: 0 10px;">Z_</div> <div style="display: inline-block; vertical-align: middle;"> <div style="border: 1px solid black; padding: 5px; margin-bottom: 5px;">I_</div> <div style="border: 1px solid black; padding: 5px;">J_</div> </div> <div style="display: inline-block; vertical-align: middle;">S_;</div>
--

Commanding G133 and G134 rotates the spindle clockwise and counterclockwise, respectively.

Z : Z axis target position  
As specified by G90/G91.  
I : Thread pitch  
J : Number of threads  
S : Spindle speed

The Z axis is moved synchronously with the spindle.

These are one shot G codes. Use G133/G134 codes each time even for continuous operation.

**(Note 1)** When the screw pitch is less than the <Minimum tapping pitch> in the <Machine parameter>, the alarm <<Pitch data error>> is triggered.

**(Note 2)** A command to change the tap twist direction is not possible while in the inverse time feed (G93) modal.  
If a command is issued, the alarm <<Command not possible during inverse time feed>> is triggered.

## 3.16 G code Priority

- 1 Executed correctly
- 2 Error
- 3 The last G command is effective
- 4 One-shot is executed and the modal is updated
- 5 G00 group is only modal updated
- 6 G00 group is executed and canned cycle is cancelled
- 7 G100 is executed and canned cycle is cancelled
- 8 An error occurs when the XY plane (G17) is not selected
- 9 An error occurs during coordinate rotation (G68, G168 modal)
- 10 An error occurs when Z-axis is mirror mod
- 11 An error occurs when the coordinate system used is changed
- 12 An error occurs during tool diameter offset (G41, G42 modal)
- 13 An error occurs during tool diameter offset (G41, G42 modal) and the selected plane is different
- 14 An error occurs other than during tool diameter offset (G41, G42 modal)
- 15 An error is triggered when switching between cutter compensation and nose R compensation, except when the tool change command is issued on the same block.
- 16 An error is triggered when switching between cutter compensation and tool position compensation, except when the tool change command is issued on the same block.
- 17 An error is triggered if the travel of X, Y or Z axis when tool length/tool position offset is changed is set to "1: Type2".
- 18 An error is triggered during tool position compensation (G143 and G144 modal).
- 19 Error when not using feed rate per rotation (G95)
- 20 Error when a G100 simultaneous command is not possible (due to priority position of G100)



## ■ When a command is given to the same block (modal - modal)

Front	Back	Back																																					
		G0 G1	G2 G3	G2 .2 G3.2	G102 G103 G202 G203	G33 G392	G17	G18 G19	G22	G23	G40	G41 G42 G141 G142	G43 G44	G143 G144	G49	G50	G51	G50.1	G51.1	G54	G54.1P	G54.2	G61	G66	G67	G68	G168	G69	G68.2	G73 G83	G80	G90	G93	G94 G95	G96	G97	G98 G99	G321 G322 G323	
	G0,G1	3	3	3	3	3	1	1	5	1	1	1	1	1	2	2	5	5	1	1	1	1	2	2	2	5	5	1	5	5	1	1	1	1	1	1	1	1	1
	G2,G3	3	3	3	3	3	1	1	5	1	12	14	2	2	2	2	5	5	1	1	1	1	2	2	5	5	1	5	5	1	1	1	1	1	1	1	1	1	1
	G2.2,G3.2	3	3	3	3	3	1	1	5	1	12	14	2	2	2	2	5	2	1	1	1	1	2	2	2	2	1	2	5	1	1	2	1	1	1	1	1	1	
	G102,G103 G202,G203	3	3	3	3	3	1	2	5	1	12	2	2	2	2	2	5	2	1	1	1	1	2	2	2	2	1	2	5	1	1	1	1	1	1	1	1	1	
	G33,G392	3	3	3	3	3	1	1	2	1	2	2	2	2	2	2	2	2	1	1	2	1	2	2	2	2	1	2	2	1	1	2	19	1	1	1	1	1	
	G17	1	1	1	1	1	3	3	2	1	13	2	1	1	1	2	2	2	2	1	1	1	1	2	2	9	9	1	1	1	1	1	1	1	1	1	1	1	
	G18,G19	1	1	1	2	1	3	3	2	1	13	2	1	1	1	2	2	2	2	1	1	1	1	2	2	9	2	1	1	2	1	1	1	1	1	1	1	1	1
	G22	5	5	5	5	2	2	2	3	3	2	2	2	2	2	2	2	2	1	1	1	1	2	2	2	2	1	2	2	1	1	1	1	1	1	1	1	1	
	G23	1	1	1	1	1	1	1	3	3	2	2	2	2	2	2	2	2	1	1	1	1	2	2	2	2	1	2	2	1	1	1	1	1	1	1	1	1	
	G40	1	12	12	12	2	13	13	2	2	3	3	1	1	1	2	2	2	2	1	1	1	1	2	2	2	2	1	2	2	1	1	1	1	1	1	1	1	1
	G41,G42 G141,G142	1	14	14	2	2	2	2	2	3	3	1	1	1	2	2	2	2	1	1	1	1	2	2	2	2	2	2	2	1	1	1	1	1	1	1	1	1	1
	G43,G44	1	20	20	2	2	1	1	2	2	1	1	3	3	3	2	2	2	2	1	1	1	1	2	2	2	2	2	2	1	1	1	1	1	1	1	1	1	1
	G143,G144	1	20	20	2	2	1	1	2	2	1	1	3	3	3	2	2	2	2	1	1	1	1	2	2	2	2	2	2	1	1	1	1	1	1	1	1	1	1
	G49	1	2	2	2	2	1	1	2	2	1	1	3	3	3	2	2	2	2	1	1	1	1	2	2	2	2	1	2	18	1	1	1	1	1	1	1	1	
	G50	2	2	2	2	2	2	2	2	2	2	2	2	2	3	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	
	G51	2	2	2	2	2	2	2	2	2	2	2	2	2	2	3	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	
	G50.1	5	5	5	5	2	2	2	2	2	2	2	2	2	2	2	3	3	2	2	2	1	2	2	2	2	2	2	2	2	1	1	1	1	1	1	1	1	
	G51.1	5	5	2	2	2	2	2	2	2	2	2	2	2	2	3	3	2	2	2	2	1	2	2	2	2	2	2	2	2	1	1	1	1	1	1	1	1	
	G54	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	3	3	2	1	2	2	1	1	1	1	1	1	1	1	1	1	2	1	1	1	
	G54.1P	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	3	3	2	1	2	2	2	1	1	2	2	1	1	1	1	1	2	1	1	1	
	G54.2P	1	2	2	2	2	1	1	1	1	1	1	1	1	1	2	2	2	2	2	2	3	1	2	2	2	2	1	2	2	1	1	1	1	1	2	1	1	
	G61	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	3	2	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
	G66	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	
	G67	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	
	G68	5	5	2	2	2	9	9	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	3	3	3	2	2	1	1	1	1	1	1	1	1	1	
	G168	5	5	2	2	2	9	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	3	3	3	2	2	1	1	1	1	1	1	1	1	1	
	G69	1	1	1	1	1	1	1	1	1	1	2	2	2	1	2	2	2	2	1	1	1	1	2	2	3	3	3	3	1	1	1	1	1	1	1	1	1	
	G68.2	5	5	2	2	2	1	1	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	3	3	2	1	1	1	1	1	2	1	1	2	
	G73,G83	6	6	6	6	2	1	2	2	2	2	1	2	18	2	2	2	2	1	2	2	1	2	2	2	2	1	2	3	3	1	2	1	2	2	1	1		
	G80	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	1	1	1	1	2	2	1	1	1	1	3	3	1	1	1	1	1	1	1	1	
	G90	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	2	2	1	1	1	1	1	1	3	1	1	1	1	1	1	1	
	G94,G95	1	1	2	1	19	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	3	3	1	1	1	1	
	G93	1	1	1	1	2	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	2	2	1	1	1	2	1	1	3	3	2	1	1	1	1	1	
	G96	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	2	2	2	1	2	2	1	1	2	2	1	1	2	1	3	3	1	1	1	
	G97	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	2	2	1	1	1	1	2	1	1	1	1	3	3	1	1	1	
	G98,G99	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	3	1	1
	G321,G322, G323	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	2	2	1	1	1	2	1	1	1	1	1	1	1	1	1	3	

## ■ When a command is given to the same block (modal - one-shot)

Front \ Back	G4	G9	G10	G12	G28	G30	G31	G36	G52	G53	G53.1	G60	G65	G92	G100	G120	G121	G131	G133	G173	G210	G376
G0,G1	4	1	4	4	4	4	4	4	4	1	20	4	2	4	4	4	4	4	4	5	2	5
G2,G3	4	1	4	4	4	4	4	2	4	1	2	4	2	4	4	4	2	4	4	5	2	5
G2.2,G3.2	4	1	4	2	4	4	4	2	4	1	2	4	2	4	4	4	2	4	4	5	2	5
G102,G103, G202,G203	4	1	4	4	4	4	4	2	4	1	2	4	2	4	4	4	2	4	4	5	2	5
G33,G392	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G17	2	1	2	2	2	2	2	1	2	1	2	1	2	2	2	2	1	2	1	1	2	1
G18,G19	2	1	2	2	2	2	2	2	2	1	2	1	2	2	2	2	2	2	2	2	2	1
G22	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G23	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G40	2	1	2	2	2	2	2	2	2	1	20	2	2	2	1	2	2	2	2	2	2	2
G41,G42 G141,G142	2	1	2	2	2	2	2	2	2	1	2	2	2	2	8	2	2	2	2	2	2	2
G43,G44	2	1	2	2	1	1	1	1	2	17	20	1	2	2	1	1	1	1	2	1	2	2
G143,G144	2	1	2	2	1	1	1	2	2	17	20	1	2	2	1	1	2	1	2	2	2	2
G49	2	1	2	2	1	1	1	1	2	1	20	1	2	2	1	1	18	1	2	18	2	2
G50	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G51	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G50.1	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G51.1	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G54	1	1	1	1	1	1	1	2	1	4	2	1	2	1	1	1	1	1	1	1	2	1
G54.1P	2	1	2	1	2	2	2	2	1	4	2	1	2	1	1	1	1	1	1	2	2	2
G54.2P	2	1	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	2	2	2
G61	1	1	1	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	2	1
G66	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G67	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G68	2	1	2	1	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G168	2	1	2	1	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G69	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1
G68.2	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G73	2	1	2	2	2	2	2	2	2	1	2	1	2	2	2	2	2	2	2	3	2	2
G80	1	1	1	1	4	4	4	4	4	1	20	4	2	4	4	4	4	4	4	3	2	1
G90	1	1	1	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	2	1
G94,G95	1	1	1	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	2	19
G93	1	1	1	2	1	1	2	2	1	1	20	1	1	2	2	1	1	1	1	1	1	2
G96	2	1	2	2	1	2	2	2	1	1	20	1	2	1	1	1	2	2	2	2	2	2
G97	1	1	2	2	1	1	1	2	1	1	20	1	2	2	1	1	2	1	2	2	2	2
G98,G99	1	1	1	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	2	1
G321,G322,G323	1	1	1	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	2	1

## ■ When a command is given to the same block (one-shot - modal)

Back Front	Which a Command is given to the same block (one-shot - modular)																																G321 G322 G323				
	G0 G1	G2 G3	G2. 2 G3.2	G102 G103 G202 G203	G33 G392	G17	G18 G19	G22	G23	G40	G41 G42 G141 G142	G43 G44	G143 G144	G49	G50	G51	G50.1	G51.1	G54	G54.1	G54.2	G61	G66	G67	G68	G168	G69	G68.2	G73	G80	G90	G93		G94 G95	G96	G97	G98 G99
G4	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	2	1	2	2	1	1	1	1	2	1	1	1
G9	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	
G10	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	2	1	2	2	1	1	1	1	2	2	1	1
G12	4	4	2	4	2	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	1	1	1	2	2	2	1	1	2	1	2	2	1	1
G28	4	4	4	4	2	2	2	2	2	2	2	1	1	1	2	2	2	2	1	2	2	1	2	2	2	2	1	2	2	4	1	1	1	1	1	1	1
G30	4	4	4	4	2	2	2	2	2	2	2	1	1	1	2	2	2	2	1	2	2	1	2	2	2	2	1	2	2	4	1	1	1	2	1	1	1
G31	4	4	4	4	2	2	2	2	2	2	2	1	1	1	2	2	2	2	1	2	2	1	2	2	2	2	1	2	2	4	1	2	1	2	1	1	1
G36	4	2	2	2	2	1	2	2	2	2	2	1	2	1	2	2	2	1	2	2	1	2	2	2	2	1	2	2	4	1	2	1	2	2	1	1	
G52	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	2	2	2	1	2	2	4	1	1	1	1	1	1	1
G53	1	1	1	1	1	1	1	1	1	1	1	17	17	1	2	2	2	2	4	4	2	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	
G53.1	20	2	2	2	2	2	2	2	2	20	2	20	20	20	2	2	2	2	2	2	20	2	2	2	2	2	2	2	2	20	20	20	20	2	20	20	20
G60	4	4	4	4	2	1	1	2	2	2	2	1	1	1	2	2	2	2	1	1	1	1	2	2	2	2	1	2	1	4	1	1	1	1	1	1	1
G65	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	
G92	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	2	2	1	2	2	4	1	1	1	1	2	2	1	1
G100	4	4	4	4	2	2	2	2	2	1	8	1	1	1	2	2	2	2	1	1	1	1	2	2	2	2	1	2	2	4	1	1	1	1	1	1	1
G120	4	4	4	4	2	2	2	2	2	2	2	1	1	1	2	2	2	2	1	1	2	1	2	2	2	2	1	2	2	4	1	1	1	1	1	1	1
G121	4	2	2	2	2	1	2	2	2	2	2	1	2	18	2	2	2	2	1	1	2	1	2	2	2	2	1	2	2	4	1	2	1	2	2	1	1
G131	4	4	4	4	2	2	2	2	2	2	2	1	1	1	2	2	2	2	1	1	2	1	2	2	2	2	1	2	2	4	1	2	1	2	1	1	1
G133	4	4	4	4	2	1	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	2	2	1	2	2	4	1	2	1	2	2	1	1	
G173	6	6	6	6	2	1	2	2	2	2	2	1	2	18	2	2	2	2	1	2	2	1	2	2	2	2	1	2	3	3	1	2	1	2	2	1	1
G210	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G376	6	6	6	6	2	1	1	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	1	2	2	1	1	2	19	2	2	1	1	

## ■ When a command is given to the same block (one-shot - one-shot)

	G4	G9	G10	G12	G28	G30	G31	G36	G52	G53	G53.1	G60	G65	G92	G100	G120	G121	G131	G133	G173	G210	G376
G4	3	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G9	1	3	1	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	2	1
G10	2	1	3	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G12	2	1	2	3	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G28	2	1	2	2	3	3	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G30	2	1	2	2	3	3	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G31	2	1	2	2	2	2	3	2	2	1	2	2	2	2	2	2	2	3	2	2	2	2
G36	2	1	2	2	2	2	2	3	2	1	2	1	2	2	2	2	2	2	2	2	2	2
G52	2	1	2	2	2	2	2	2	3	1	2	2	2	2	2	2	2	2	2	2	2	2
G53	1	1	1	1	1	1	1	1	1	3	2	1	2	1	1	1	1	1	1	1	2	1
G53.1	2	20	2	2	2	2	2	2	2	2	3	2	2	2	1	2	2	2	2	2	2	2
G60	2	1	2	2	2	2	2	1	2	1	2	3	2	2	2	2	2	2	2	1	2	2
G65	2	2	2	2	2	2	2	2	2	2	2	2	3	2	2	2	2	2	2	2	2	2
G92	2	1	2	2	2	2	2	2	2	1	2	2	3	2	2	2	2	2	2	2	2	2
G100	2	1	2	2	2	2	2	2	2	1	1	2	2	2	3	2	2	2	2	2	2	2
G120	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	3	2	2	2	2	2	2
G121	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	3	2	2	2	2	2
G131	2	1	2	2	2	2	3	2	2	1	2	2	2	2	2	2	2	3	2	2	2	2
G133	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	3	2	2	2
G173	2	1	2	2	2	2	2	2	2	1	2	1	2	2	2	2	2	2	2	3	2	2
G210	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	3	2
G376	2	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	3

## ■ When a command is given during modal call

Modal Comman d	G0 G1	G2 G3	G2.2 G3.2	G33 G392	G17	G18 G19	G22	G23	G40	G41 G42 G141 G142	G43 G44	G143 G144	G49	G50	G51	G50.1	G51.1	G54	G54.1	G54.2P 0	G54.2P 1	G61	G66	G67	G68 G168	G69	G68.2 Before G53	G68.2 After G53.1	G73 G83	G74 G84	G80	G90	G93	G94 G95	G96	G97	G98 G99	G321 G322 G323
G0 G1	3	3	3	3	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	6	6	1	1	1	1	1	1	1	1
G2 G3	3	3	3	3	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	6	6	1	1	1	1	1	1	1	1
G2.2 G3.2	3	3	3	3	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	2	1	6	6	1	1	2	1	1	1	1	1
G102 G103 G202 G203	3	3	3	3	1	2	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	2	2	6	6	1	1	1	1	1	1	1	1
G33 G392	3	3	3	3	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	2	2	6	6	1	1	2	19	1	1	1	1
G17	1	1	1	1	1	3	3	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	
G18 G19	1	1	1	1	1	3	3	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	2	2	2	1	1	1	1	1	1	1	1
G22	1	1	1	1	1	1	3	3	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G23	1	1	1	1	1	1	3	3	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G40	1	1	1	2	1	1	1	1	3	3	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G41 G42 G141 G142	1	1	1	2	1	1	1	1	3	15	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	2	1	2	2	1	1	1	1	1	1	1	
G43 G44	1	1	2	2	1	1	1	1	1	1	16	16	3	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G143 G144	1	1	2	2	1	1	1	1	1	1	16	16	3	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	
G49	1	1	1	2	1	1	1	1	1	1	3	3	3	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G50	1	1	1	1	1	1	1	1	1	1	1	1	1	3	3	1	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	
G51	1	1	2	2	1	1	1	1	1	1	1	1	1	3	3	1	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	
G50.1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	3	3	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	
G51.1	1	1	2	2	1	1	1	1	1	1	1	1	1	2	3	3	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	
G54	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	3	3	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	
G54.1P	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	3	3	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	
G54.2P0	1	1	1	2	1	1	1	1	1	1	1	1	1	2	1	2	1	1	3	3	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	
G54.2P1	1	1	1	2	1	1	1	1	1	1	1	1	1	2	1	2	1	1	3	3	1	1	1	2	1	2	2	2	1	1	1	1	1	1	1	1	1	
G61	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G66	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	2	3	1	1	1	1	1	1	1	1	1	1	1	1	1	
G67	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	3	1	1	1	1	1	1	1	1	1	1	1	1	1	
G68	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	3	3	2	2	1	1	1	1	1	1	1	1	1	1	
G168	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	3	3	2	2	1	1	1	1	1	1	1	1	1	1	
G69	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	3	1	1	1	1	1	1	1	1	1	1	1	
G68.2	1	1	1	2	1	2	1	1	1	2	1	2	1	2	1	2	1	1	1	2	1	1	1	2	1	1	1	1	1	1	1	1	1	2	1	1	1	
G73 G83	1	1	1	2	1	2	1	1	1	2	1	2	1	1	1	1	10	1	1	1	1	1	1	1	1	1	2	1	3	3	3	1	2	1	1	1	1	
G74 G84	1	1	1	2	1	2	1	1	1	2	1	2	1	1	1	1	10	1	1	1	1	1	1	1	1	2	1	3	3	3	1	2	1	2	1	1	1	
G80	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	3	3	1	1	1	1	1	1	1	
G90	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	1	1	1	1	1	1	
G93	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	3	3	2	1	1	1	
G94 G95	1	1	1	19	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	3	1	1	1	1	
G96	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	2	1	3	3	1	1	1	
G97	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	3	1	1	1	

Modal Comman d	G0 G1	G2 G3	G2.2 G3.2	G33 G392	G17	G18 G19	G22	G23	G40	G41 G42 G141 G142	G43 G44	G143 G144	G49	G50	G51	G50.1	G51.1	G54	G54.1	G54.2P 0	G54.2P 1	G61	G66	G67	G68 G168	G69	G68.2 Before G53	G68.2 After G53.1	G73 G83	G74 G84	G80	G90	G93	G94 G95	G96	G97	G98 G99	G321 G322 G323
G98	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	1	
G321, G322, G323	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	
G4	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G9	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G10	1	1	1	1	1	1	1	1	1	1	1	1	1	1	11	1	11	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G12	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	2	1	1	1	1	
G28	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	
G30	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	
G31	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	1	1	2	2	1	1	2	1	1	1	1	
G36	1	2	2	2	1	2	1	1	1	2	1	2	1	1	2	1	2	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	2	1	1	1	1	
G52	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	1	
G53	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G53.1	1	1	1	2	1	2	1	1	1	2	1	2	1	1	2	1	2	1	1	1	2	1	1	1	2	1	1	1	1	1	1	1	1	2	1	1	1	
G60	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G65	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G92	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	
G100	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	7	7	1	1	1	1	1	1	1	
G120	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	
G121	1	2	1	2	1	2	1	1	1	2	1	2	1	1	2	1	2	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	2	1	1	1	1	
G131	1	1	2	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	1	2	2	1	1	2	1	1	1	1	1	
G133	1	1	1	1	1	2	1	1	1	2	1	2	1	1	1	1	10	1	1	1	1	1	1	1	1	2	2	1	1	1	1	2	1	2	1	1	1	
G173	1	1	1	1	1	2	1	1	1	2	1	2	1	1	1	1	10	1	1	1	1	1	1	1	1	2	2	1	1	1	1	2	1	1	1	1	1	
G210	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
G376	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	2	19	1	1	1	

## 3.17 Programmable Data Input (High Accuracy) (G210)

### 3.17.1 High Accuracy Mode A Parameter Changes

Command format

G210 L31 Pn I\_ J\_ K\_ W\_ Q\_ X\_ R\_ B\_;

n = 1	:	High accuracy mode A (Level 1) (M260)
n = 2	:	High accuracy mode A (Level 2) (M261)
n = 3	:	High accuracy mode A (Level 3) (M262)
n = 4	:	High accuracy mode A (Level 4) (M263)
n = 5	:	High accuracy mode A (Level 5) (M264)
n = 6	:	High accuracy mode A (Level 6) (M265)
n = 7	:	High accuracy mode A (Level 7) (M266)
n = 8	:	High accuracy mode A (Level 8) (M267)
I	:	Corner deceleration override (%)
J	:	Arc deceleration override (%)
K	:	Curve approximation deceleration override (%)
W	:	Smooth path offset level
Q	:	Smooth override (%)
X	:	Cutting feed time constant selection
R	:	Minute block deletion distance (mm) (inch)
B	:	Accuracy level (mm) (inch)

- (Note 1) Refer to “Chapter 13 (1) High accuracy mode A III” for further details on high accuracy mode A.
- (Note 2) The set parameters can be cancelled using M30, reset or turning OFF the power.
- (Note 3) An alarm is triggered when L or P is omitted, or when only the L and P commands are issued.
- (Note 4) When another address is omitted, the current value for the omitted parameter does not change.
- (Note 5) The range that can be set for the numerical values of IJKWQXRB is the same as the corresponding user parameter. Refer to the “Chapter 2.5 User Parameter” in the Operation Manual (Data) for further details.

### 3.17.2 High Accuracy Mode B Parameter Changes

Command format

**G210 L32 Pn I\_ J\_ K\_ W\_ Q\_ R\_ Y\_;**

- n = 1 : High accuracy mode B (Level 1) (M280)
- n = 2 : High accuracy mode B (Level 2) (M281)
- n = 3 : High accuracy mode B (Level 3) (M282)
- n = 4 : High accuracy mode B (Level 4) (M283)
- n = 5 : High accuracy mode B (Level 5) (M284)
- n = 6 : High accuracy mode B (Level 6) (M285)
- n = 7 : High accuracy mode B (Level 7) (M286)
- n = 8 : High accuracy mode B (Level 8) (M287)

- I : Corner deceleration override (%)
- J : Arc deceleration override (%)
- K : Curve approximation deceleration override (%)
- W : Smooth path offset level
- Q : Smooth override (%)
- R : Minute block deletion distance (mm) (inch)
- Y : Smooth override type

- (Note 1)** The set parameters can be cancelled using M30, reset or turning OFF the power.
- (Note 2)** An alarm is triggered when L or P is omitted, or when only the L and P commands are issued.
- (Note 3)** When another address is omitted, the current value for the omitted parameter does not change.
- (Note 4)** The range that can be set for the numerical values of JKWQRY is the same as the corresponding user parameter.  
Refer to the “Chapter 2.5 User Parameter” in the “Operation Manual (Data)” for further details.



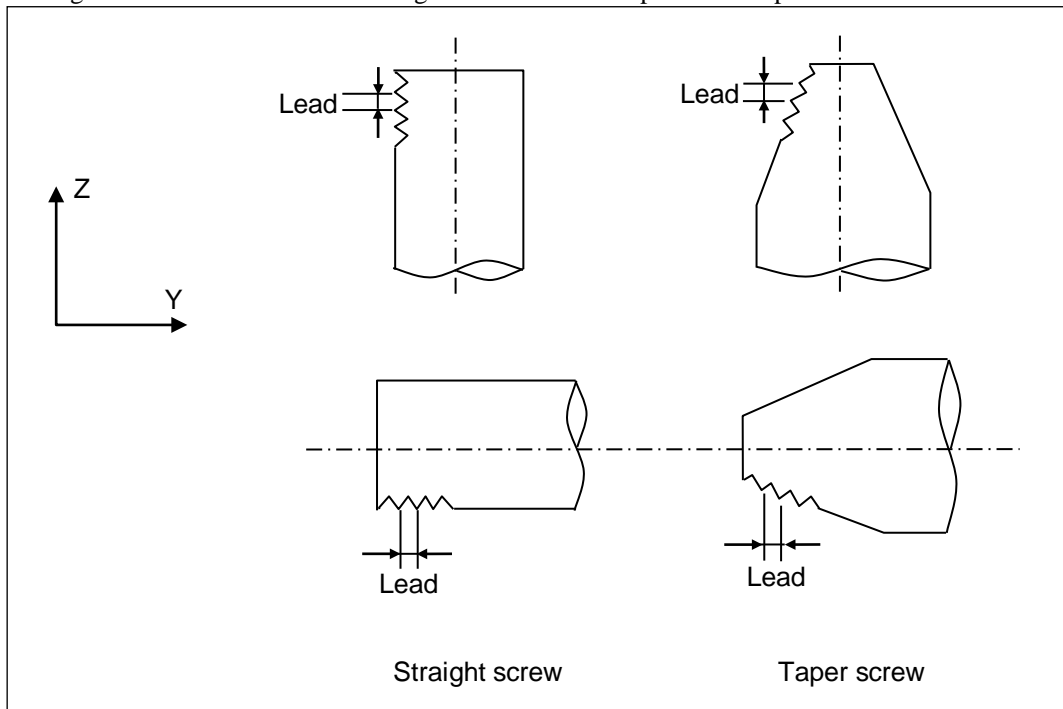
## 3.18 Thread Cutting

The thread cutting function is available when equipped with the lathe function.

A thread cutting command can be issued when using the feed rate per rotation (G95) modal. If issuing a command using the feed rate per minute (G94) modal, the alarm <<Thread cutting command not possible in feed rate per minute>> is triggered. In addition, if a command is issued while in the inverse time feed (G93) modal, the alarm <<Command not possible during inverse time feed>> is triggered.

### 3.18.1 Single Start Lead Thread Cutting (G33)

Cutting the lead for the thread on a single start screw and taper screw is possible.



Command format

**When cutting a straight screw**

**G33**  $\begin{bmatrix} X\_ \\ Y\_ \\ Z\_ \end{bmatrix}$  **F\_ Q\_**

**When cutting a taper screw**

**G33**  $\begin{bmatrix} X\_Y\_ \\ X\_Z\_ \\ Y\_Z\_ \end{bmatrix}$  **F\_ Q\_**

X, Y, Z: Screw end point (mm) (inch)

F : Screw lead (mm) (inch)

The command range depends on the user parameter (switch 1) <Machine unit system>.

Metric: 0.0001 to 999.9999 (mm)

Inch: 0.00001 to 99.99999 (inch)

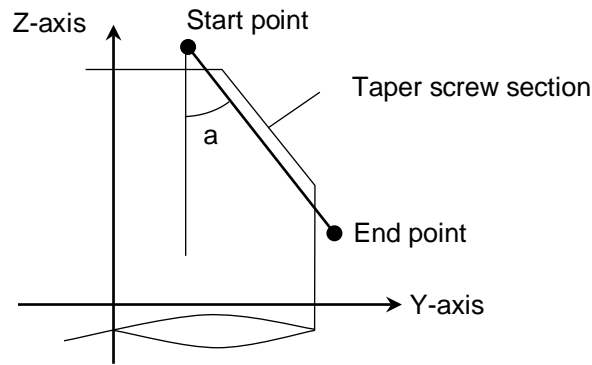
Q : Start angle for thread cutting (0.000° to 360.000°)

If this is omitted, the default is 0 degrees.

The value for the command at address F is also reflected in the F code modal.

When cutting a taper screw, address F issues a command for the lead on the long axis direction.

When specifying the start angle on the spindle for thread cutting or when machining multiple start screws, address Q issues a command per block.



When  $a < 45^\circ$  → The lead is direction of the Z-axis.  
 When  $a > 45^\circ$  → The lead is direction of the Y-axis.  
 When  $a = 45^\circ$  → The lead can either be the direction of the Y- or Z-axis.

- (Note 1) During a single block operation, it stops after the first block is executed that is not a thread cutting block (after thread cutting).
- (Note 2) When a command is issued while the lathe spindle is not rotating, a <<Feedrate error>> is triggered.
- (Note 3) During thread cutting, the SPINDLE OVERRIDE and FEEDRATE OVERRIDE are disabled, and the setting is fixed at 100%.
- (Note 4) During thread cutting as well, the constant peripheral speed control is enabled. However, the correct thread lead may not be maintained if the spindle speed is changed. When cutting a taper screw, do not use the constant peripheral speed control.
- (Note 5) When one of the following operations is performed or when a stop level 3 or 4 alarm is triggered during thread cutting, it stops after the first block is executed that is not a thread cutting block (after thread cutting).
- [FEED HOLD] switch is pressed.
  - [RST] key is pressed.
  - [M.LCK] key is pressed.
  - [SINGL] key is pressed.
  - Mode is changed.
  - [Mode of Safe Operation] switch setting is changed.
  - [Service Mode] switch setting is changed.

### 3.18.2 Thread Cutting Cycle (G392)

The thread cutting cycle is a canned cycle that performs thread cutting for straight and taper screws.

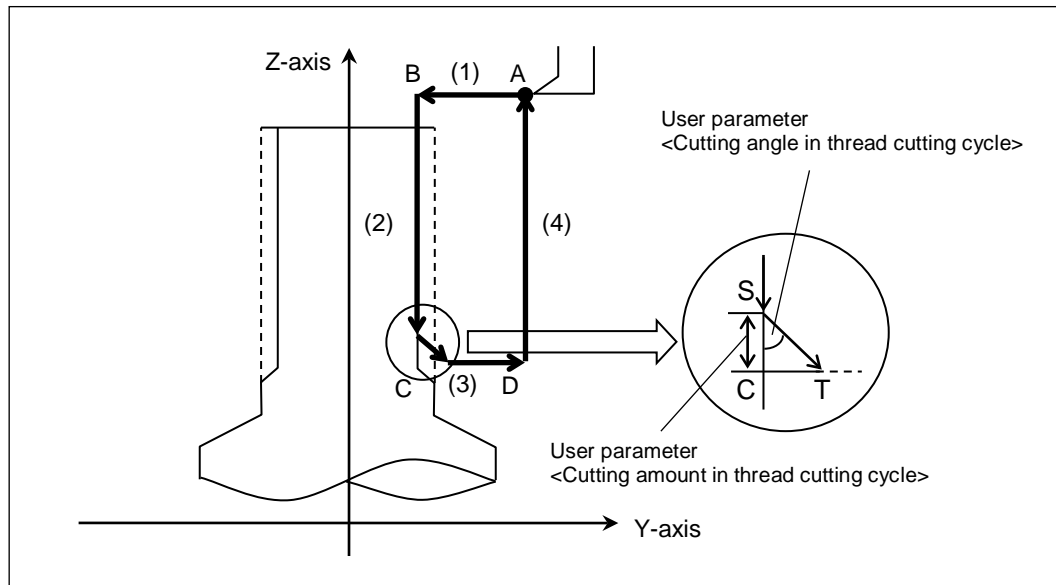
#### 3.18.2.1 Thread Cutting Cycle for Straight Screws

Command format

G392	$\begin{bmatrix} X\_Y\_ \\ X\_Z\_ \\ Y\_Z\_ \end{bmatrix}$	F\_ Q\_

X, Y, Z: Screw end point (mm) (inch)  
 F : Screw lead (mm) (inch)  
 Q : Start angle for thread cutting (0.000° to 360.000°)

3



The thread cutting cycle for straight screws performs 4 operations.

- (1) It travels using rapid feed from start point A to start point B for thread cutting.  
 The travel direction is set by the command axis address and lathe machining infeed direction. Refer to “3.19 Lathe machining infeed direction” for further details.
- (2) It travels using the cutting feed from start point B for thread cutting until the end point for thread cutting. While the M323 modal is in progress, it starts the finishing operation on the screw according to the value set in the user parameter (switch 1) <Finishing amount in thread cutting cycle>, the amount from point S to point C. The finishing angle is the value that is set in the user parameter (switch 1) <Finishing angle in thread cutting cycle>.  
 The speed for the infeed direction during finishing is calculated as shown below. When the speed of the infeed direction exceeds the machine parameter (system 1) <Max. cutting feedrate>, the speed of the infeed is controlled or limited by the maximum cutting feedrate. However, the finishing angle is the same angle as specified in the parameter.  

$$V = F \times S \times \tan(a)$$

V: Infeed direction speed (mm/min)  
 F: Screw lead (mm)  
 S: Spindle speed (min<sup>-1</sup>)  
 a: User parameter (switch 1) <Cutting angle in thread cutting cycle>
- (3) It travels using rapid feed from point C (M323 modal in progress at point T) until point D.
- (4) It travels using rapid feed from point D until start point A.

**(Note 1)** During a single block operation, operations (1), (2), (3) and (4) are performed all at once.

**(Note 2)** When the travel distance between C and D is 0, the alarm <<No relief amount>> is triggered before starting operation (1).

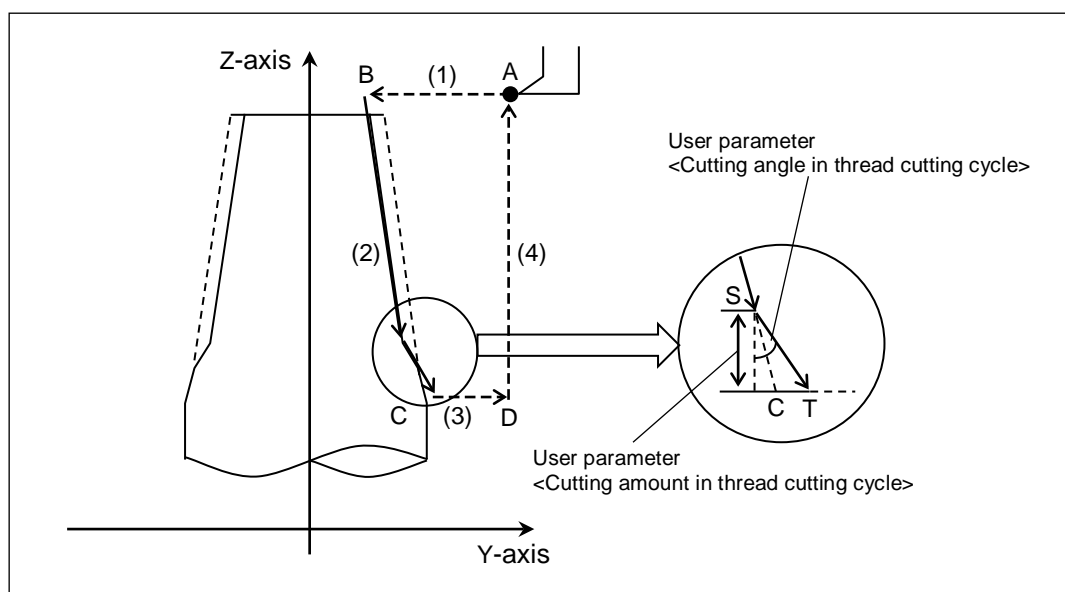
- (Note 3) When one of the conditions below applies, distance between C and D is 0, the alarm <<Cutting amount error in thread cutting cycle>> is triggered before starting operation (1).
- When the travel distance between S and C is greater than the travel distance between B and C.
  - When the travel distance between C and T is greater than the travel distance between C and D.
- (Note 4) When a thread cutting cycle command on a straight screw is issued while the lathe spindle is not rotating, a <<Feedrate error>> is triggered before starting operation (2).
- (Note 5) During operation (2), the SPINDLE OVERRIDE and FEEDRATE OVERRIDE are disabled, and the setting is fixed at 100%.
- (Note 6) When one of the following operations is performed or when a stop level 3 or 4 alarm is triggered during operation (2), it stops after operation (3).
- [FEED HOLD] switch is pressed.
  - [RST] key is pressed.
  - [M.LCK] key is pressed.
  - Mode is changed.
- (Note 7) When a stop level 3 alarm is triggered during any of the following operations: (1), (2), (3) or (4), the machine stops after operation (4).
- (Note 8) RAPID TRAVERSE OVERRIDE is always valid. When a thread cutting cycle command is issued for a straight screw in M322 modal (finishing OFF) and operation has been stopped using a RAPID TRAVERSE OVERRIDE after operation (2), the shape of the thread runout changes. In addition, be careful because the tool may become damaged.

### 3.18.2.2 Thread Cutting Cycle for Taper Screws

Command format

G392	$\begin{bmatrix} X\_Y\_ \\ X\_Z\_ \\ Y\_Z\_ \end{bmatrix}$	R_ F_ Q_

X, Y, Z: Screw end point (mm) (inch)  
 R : Taper amount (mm) (inch)  
 F : Screw lead (mm) (inch)  
 Q : Start angle for thread cutting (0.000° to 360.000°)



The thread cutting cycle for taper screws performs 4 operations.

- (1) It travels using rapid feed from start point A to start point B for thread cutting.  
The travel direction is set by the command axis address and lathe machining infeed direction. Refer to “3.19 Lathe machining infeed direction” for further details.
- (2) It travels using the cutting feed from start point B for thread cutting until the end point of the thread. At this time, thread cutting is performed where the lead for the cutting direction (Z-axis in the figure above) is the command value issued at F. The finishing is the same as the thread cutting cycle for straight screws. However, the finishing angle is the angle for the cutting direction (See figure above).

When the travel distance for the cutting direction is 0, the alarm <<Thread cutting command error>> is triggered before starting operation (1).

When the distance between C and D is greater than the distance between T and D, the alarm <<Cutting angle error in thread cutting cycle>> is triggered, and it stops before starting operation (1).

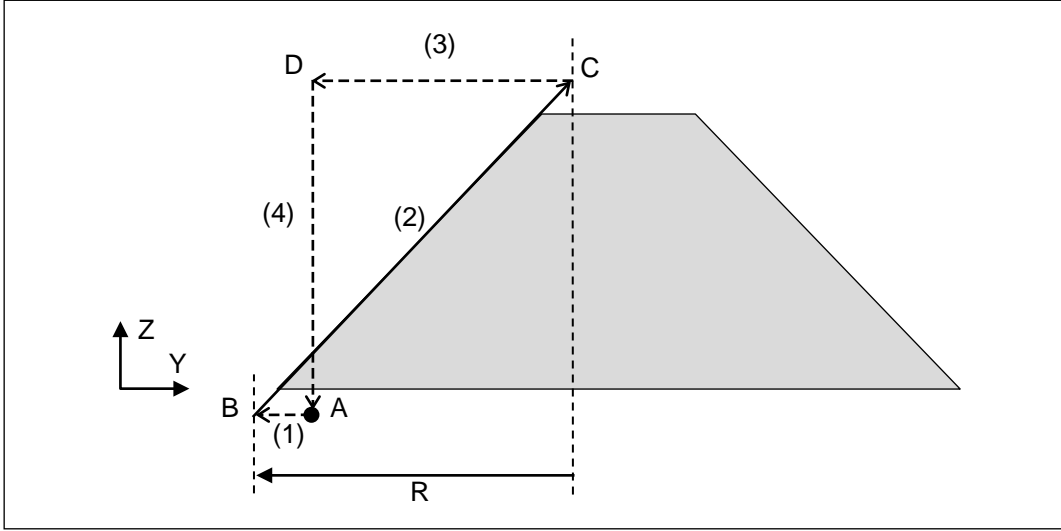
Operations (3) to (4) are the same as the thread cutting cycle for straight screws.

The diagram below shows an example of machining the outer and inner diameters when thread cutting on a taper screw with an infeed on the Y-axis.

The taper amount command is issued in increments in the infeed direction starting from the screw end point, regardless of the absolute mode (G90) and incremental mode (G91).

Taper amount (R) sign	Outer diameter machining	Inner diameter machining
+	1	2
-	3	4

When a taper command is issued in which the taper amount exceeds start point A from end point C on the screw, as shown in the diagram, the alarm <<Taper amount too large>> is triggered and it stops before starting operation (1).



The special notes above for the thread cutting cycle of straight screws is the same for the thread cutting cycle of taper screws.

3.18.3 Thread Cutting in Complex Thread Cutting Cycle (G376)

The complex thread cutting cycle is a cycle that performs thread cutting slowly in steps, breaking up the threads into multiple cuts.

Command format	<div><div>G376</div><div><div>X_Y_ X_Z_ Y_Z_</div></div><div>R_P_Q_F_D_E_I_J_K_L_</div></div>
----------------	---

- X, Y, Z:
- Screw end point (mm) (inch)
- R
- :
- Taper amount (mm) (inch)
- P
- :
- Height of screw thread (mm) (inch)
- Specifies the distance from the end point of the infeed direction for the screw, regardless of the absolute mode (G90) and incremental mode (G91).
- When there is an omission, the alarm <<Thread cutting cycle address error>> is triggered.
- Q
- :
- Initial infeed amount (mm) (inch)
- Specifies the distance from the height of the screw thread, regardless of the absolute mode (G90) and incremental mode (G91).
- When there is an omission, the alarm <<Thread cutting cycle address error>> is triggered.
- F
- :
- Screw lead (mm) (inch)
- D
- :
- Angle of screw thread (0.000° to 120.000°)
- E
- :
- Finishing amount (0.0 to 99.9)
- The finishing amount is the product when the thread lead F (mm) (inch) is multiplied by the command value.
- I
- :
- Finishing count (1 to 99.9)
- J
- :
- Minimum infeed amount (mm) (inch)
- The sign is ignored even when a negative value command is issued.
- K
- :
- Finishing allowance (mm) (inch)
- Specifies the distance from the end point of the infeed direction for the screw, regardless of the absolute mode (G90) and incremental mode (G91). In addition, the sign is ignored even when a negative value command is issued.
- L
- :
- Start angle for thread cutting (0.000° to 360.000°)

A user parameter with the same definition exists for D, E, I, J and K.

When a command is omitted, the operation is carried out according to the value set in the user parameter.

D: User parameter (switch 1) <Thread angle in complex thread cutting cycle>

E: User parameter (switch 1) <Cutting amount in thread cutting cycle>

I: User parameter (switch 1) <Cutting count in complex thread cutting cycle>

J: User parameter (switch 1) <Minimum infeed amount in complex thread cutting cycle>

K: User parameter (switch 1) <Finishing allowance in complex thread cutting cycle>

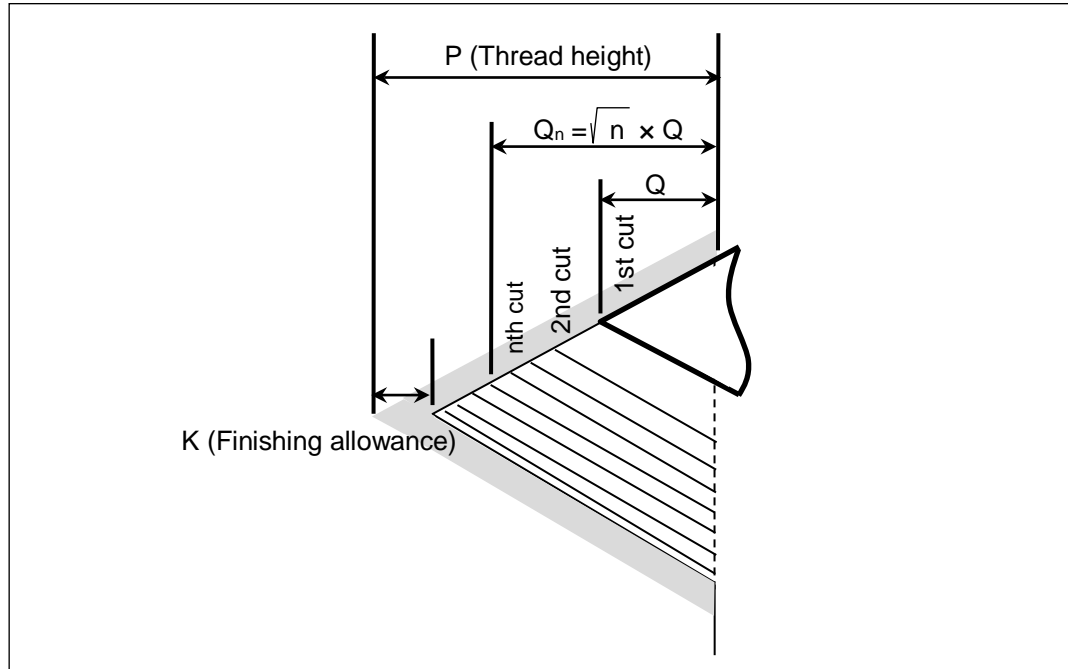
### 3.18.3.1 Infeed

When a command for the complex thread cutting cycle is issued, the infeed is carried out for a “fixed cutting amount and single edge cutting” until the finishing operation. The infeed amount  $Q_n$  for nth cut is as follows:

$$Q_n = \sqrt{n} \times Q \quad Q: \text{Initial infeed amount (mm)(inch)}$$

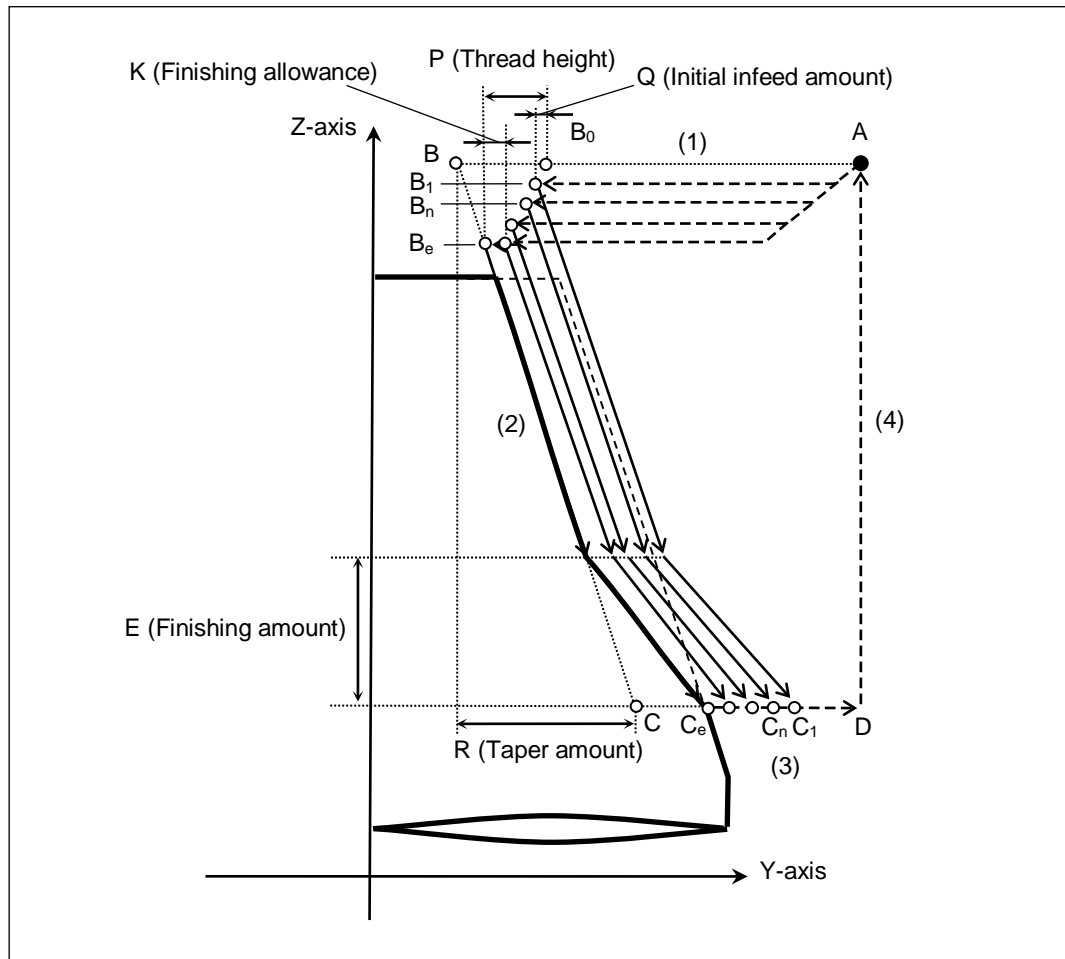
If the infeed amount after the previous cut is less than the minimum infeed amount, the infeed amount becomes the sum of the previous infeed amount plus the minimum infeed amount.

In addition, if the infeed overlaps the finishing section (gray area below), then the infeed amount becomes the difference when finishing allowance is subtracted from the thread height.



### 3.18.3.2 Detailed Description of Operation

In a complex thread cutting cycle, the rough cutting operation all the way to the finishing operation are carried out all at once as follows.



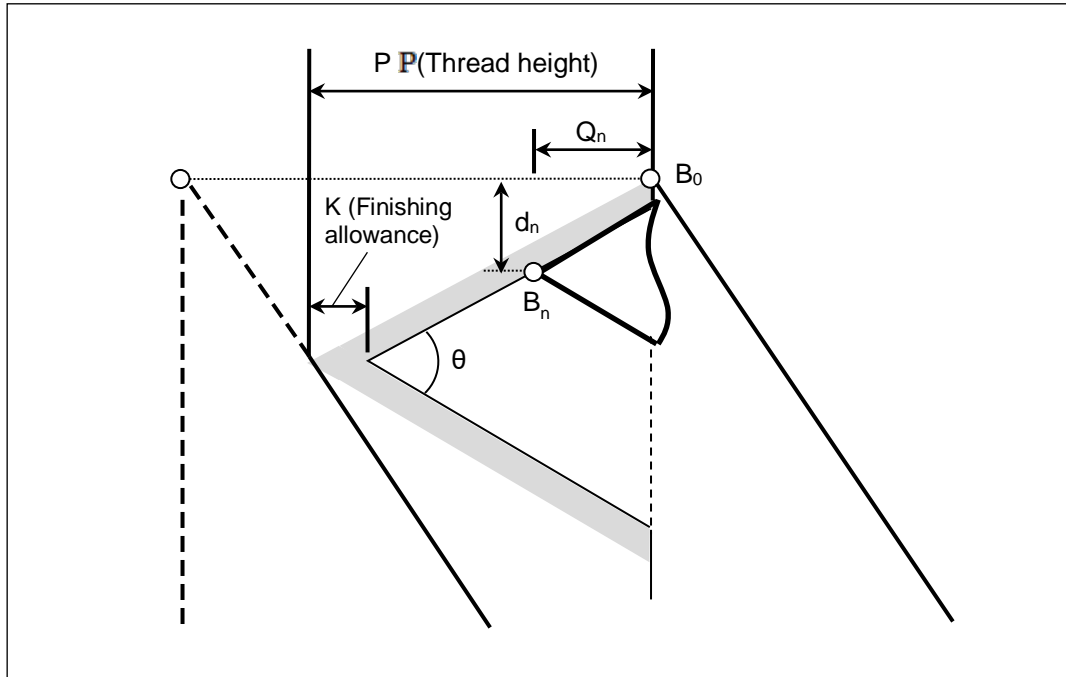


Rough cutting

- (1) It travels using rapid feed from start point A to start point B<sub>n</sub> for thread cutting on the n<sup>th</sup> cut. The infeed amount is calculated based on the formula noted in “3.18.3.1 Infeed” and it travels in the cutting direction only for the distance that is calculated from the following formula.

$$d_n = (Q_n + K) \times \tan(\theta/2)$$

- $d_n$  : Distance that is traveled in the cutting direction from the start point for thread cutting on the n<sup>th</sup> cut (mm)(inch)  
 $Q_n$  : Infeed amount for nth cut (mm) (inch)  
 $K$  : Finishing allowance (mm) (inch)  
 $\theta$  : Thread angle (°)



- (2) It travels using the cutting feed from start point B<sub>n</sub> for thread cutting on the n<sup>th</sup> cut until the end point C<sub>n</sub> for thread cutting on the nth cut. The thread finishing is the same as the thread cutting cycle (G392).  
 (3) It travels using the rapid feed from end point C<sub>n</sub> for thread cutting on the nth cut until point D.  
 (4) It travels using rapid feed from point D until start point A.

## Finishing

- (1) It travels from start point A to start point B<sub>e</sub> for finishing. It travels in the cutting direction only for the distance d<sub>e</sub> that is calculated from the following formula.

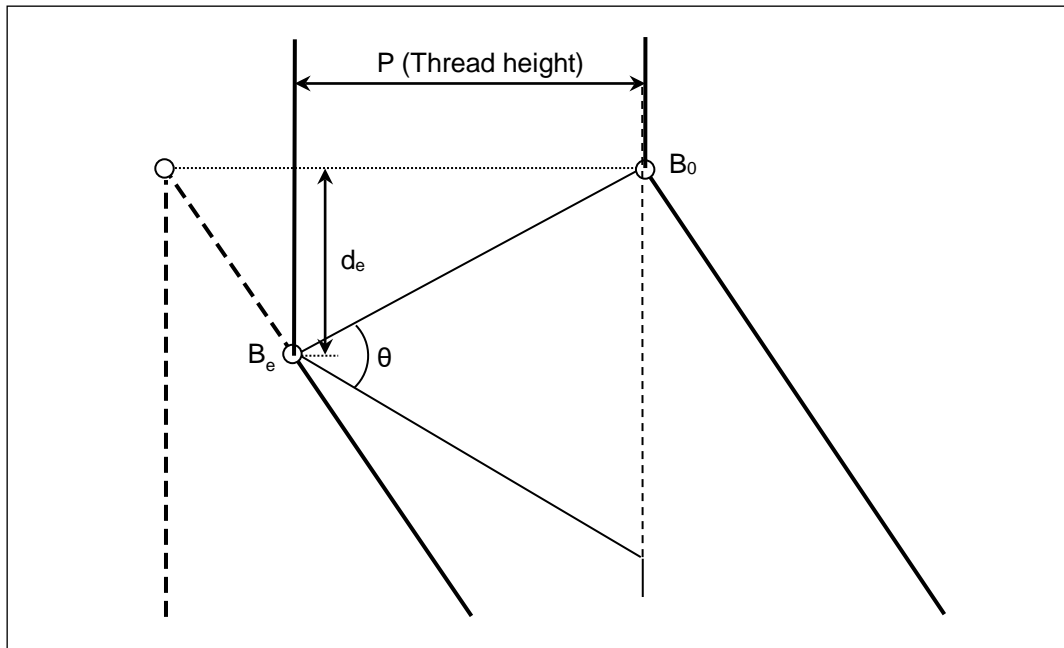
$$d_e = P \times \tan(\theta/2)$$

d<sub>e</sub> : Distance that is traveled in the cutting direction from the start point for finishing (mm)(inch)

P : Height of screw thread (mm)(inch)

θ : Thread angle (°)

When d<sub>e</sub> is greater than the travel distance in the cutting direction between B and C, the alarm <<Cutting amount error in thread cutting cycle>> is triggered, and it stops before starting the complex thread cutting cycle operation.



- (2) It travels using the cutting feed from start point B<sub>e</sub> for finishing until the end point C<sub>e</sub> of the thread. The thread finishing is the same as the thread cutting cycle (G392).
- (3) It travels using the rapid feed from end point C<sub>e</sub> of the thread until point D.
- (4) It travels using rapid feed from point D until start point A.

(1) through (4) is repeated according to the number in the finishing count.

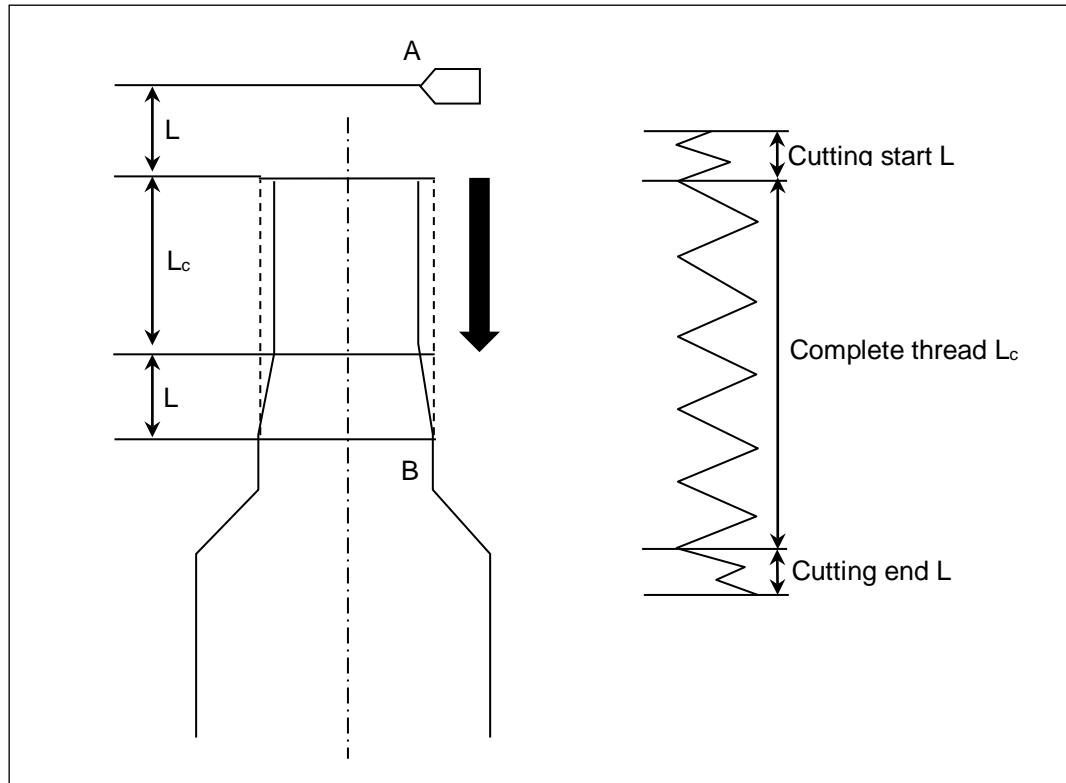
The special notes for the operations are the same as the thread cutting cycle (G392).

### 3.18.4 Thread Runout

A thread runout for the lead can be made on the cutting start and cutting end sections because the motor accelerates and decelerates when the thread cutting starts and ends. This is called a thread runout.

For example, when thread cutting from point A to point B as shown in the diagram below, it accelerates at point A (cutting start) and decelerates at point B (cutting end) in order to make a thread runout for length L.

To make a thread runout  $L_c$ , the thread cutting length for  $(2L + L_c)$  is required. Be aware of this when making a program.



#### Calculating the thread runout

The thread runout is calculated using the following formula.

$$L = K \times (N / 60) \times P$$

L : Thread runout (mm)

K : Constant (Refer to table below)

N : Spindle speed ( $\text{min}^{-1}$ )

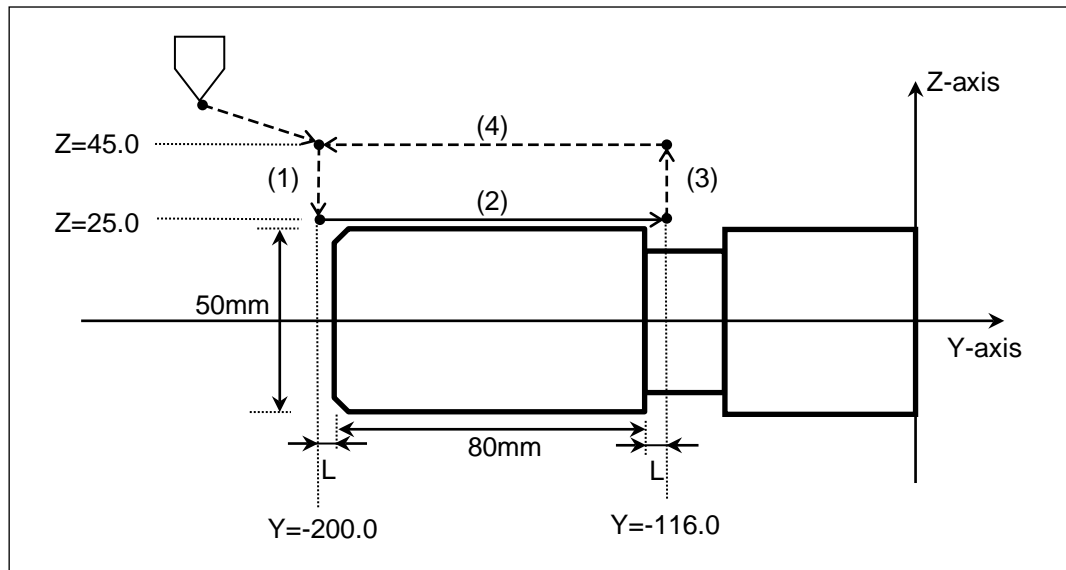
P : Lead (mm)

Model	K
M140X1	0.047
M140X2	0.047
M200X3	0.047
M300X3	0.047

This is an example of a program when thread cutting with a SPEEDIO M140X1.

Screw type	Right screw
Screw diameter	50 mm
Lead	2 mm
Spindle speed	1000 min <sup>-1</sup>

Thread runout L is set to a size that is greater than the calculation above. The diagram below shows an example when the thread runout is 2 mm.



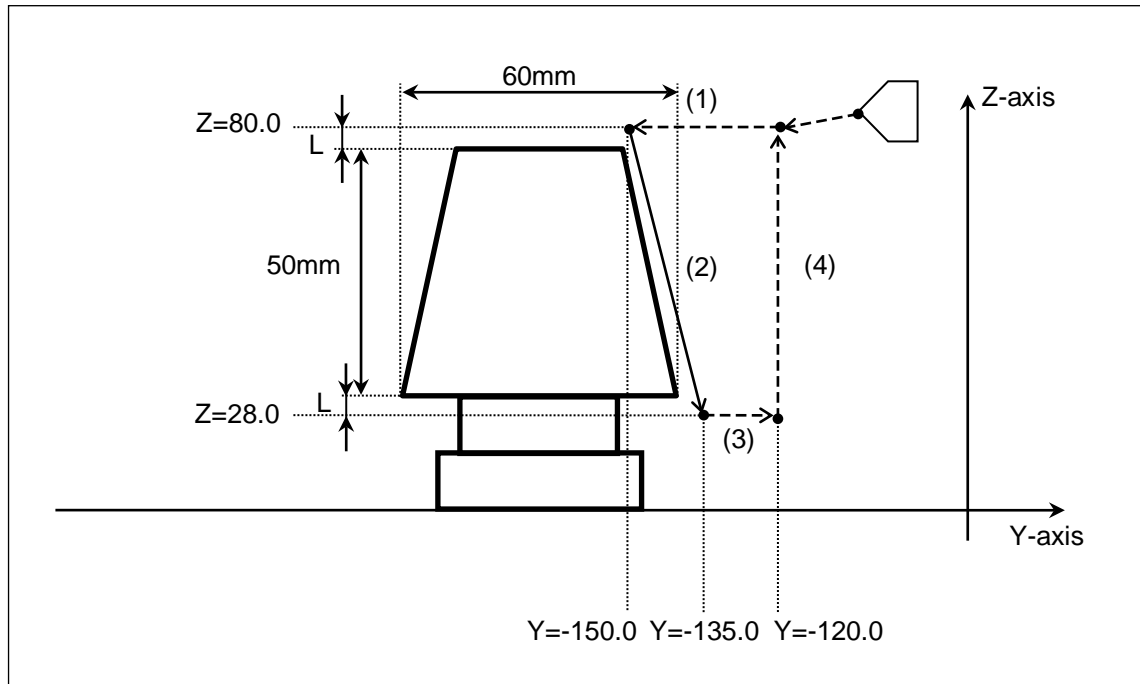
When made using G33	When made using G392	When made using G376
M303 S1000	M303 S1000	M303 S1000
G00 Y-200. Z45.	G00 Y-200. Z45.	G00 Y-200. Z45.
Z25. (1)	G392 G322 Y-116. Z25. F2. M323	G376 G322 Y-116. Z23.5 P2. Q0.5 F2. M323
G33 Y-116. F2. (2)		
G00 Z45. (3)		
Y-200. (4)		
Z24.5	Z24.5	
G33 Y-116.		
G00 Z45.		
Y-200.		
Z24.	Z24.	
G33 Y-116.		
G00 Z45.		
Y-200.		
Z23.5	Z23.5	
G33 Y-116.		
G00 Z45.		
Y-200.		

### 3.18.5.2 Machining and Thread Cutting Taper Screws

Screw type	Right screw
Screw diameter	60 mm
Lead	2 mm
Spindle speed	500 min <sup>-1</sup>

Thread runout  $0.047 \times (500 / 60) \times 2 = 0.783(\text{mm})$

Thread runout L is set to a size that is greater than the calculation above. The diagram below shows an example when the thread runout is 1 mm.



When made using G33	When made using G392	When made using G376
M303 S500 G00 Y-120. Z80. Y-150. (1) G33 Y-135. Z28. F2. (2) G00 Y-120. (3) Z80. (4)	M303 S500 G00 Y-120. Z80. G392 G322 Y-135. Z28. R-15. F2. M323	M303 S500 G00 Y-120. Z80. G376 G322 Y-136.5 Z28. R-15. P2. Q0.5 F2. M323
Y-150.5 G33 Y-135.5 Z28. F2. G00 Y-120. Z80. Y-151. G33 Y-136. Z28. F2. G00 Y-120. Z80. Y-151.5 G33 Y-136.5 Z28. F2. G00 Y-120. Z80.	Y-135.5    Y-136.   Y-136.5	

## 3.19 Lathe Machining Infeed Direction

It specifies the infeed direction during lathe machining.

Infeed direction on X-axis

Command format

**G321**

Infeed direction on Y-axis

Command format

**G322**

Infeed direction on Z-axis

Command format

**G323**

In the canned cycle for lathe machining, first, it travels using rapid feed from the start point to the start point for cutting.

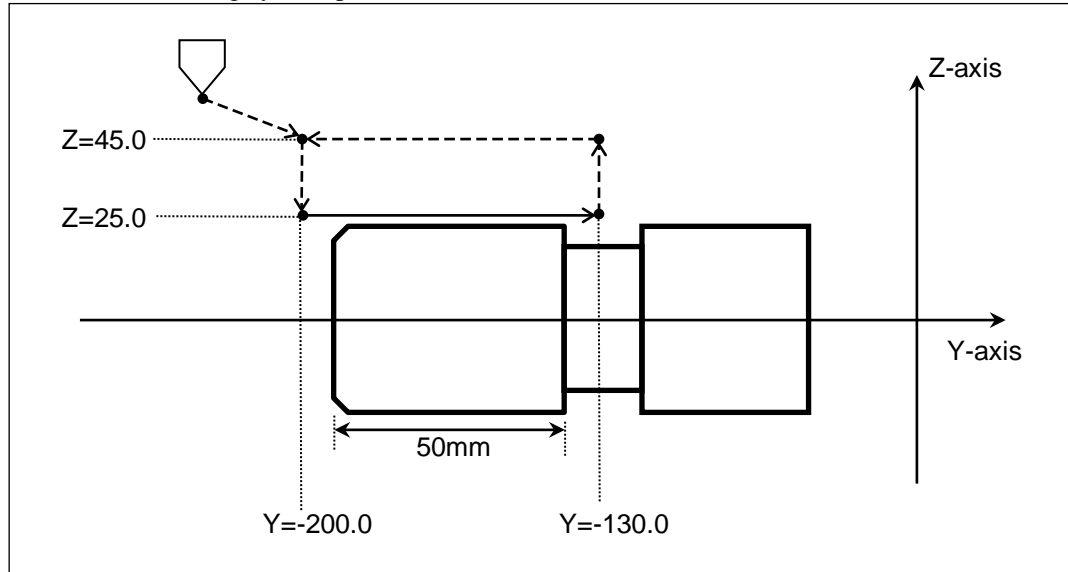
At this time, the travel direction is set by the command axis address and the G code modal (Refer to table below). If that setting is invalid, the alarm <<Invalid command>> is triggered at the time of the command.

Command axis address	G code modal for infeed direction	Travel direction
X_ Y_	G321	X-axis
	G322	Y-axis
	G323	<<Invalid command>>
X_ Z_	G321	X-axis
	G322	<<Invalid command>>
	G323	Z-axis
Y_ Z_	G321	<<Invalid command>>
	G322	Y-axis
	G323	Z-axis

The modal can be specified during power startup in the user parameter (switch 1) <Lathe machining infeed direction when power is turned ON>.

Program example

When a thread cutting cycle is performed with the Z-axis as the infeed direction



Program	Description
M303 S1000	Lathe spindle rotates clockwise at 1000 min <sup>-1</sup>
G00 Y-200. Z45.	Positioning to start point for thread cutting cycle
G323	Specifies Z-axis as infeed direction
G392 Y-130. Z25. F2. M323	Infeeds on Z-axis and cuts thread on Y-axis

(This page was intentionally left blank.)



## CHAPTER 4

# PREPARATION FUNCTIONS (COMPENSATION FUNCTIONS)

4

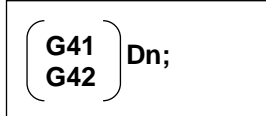
- 4.1 Cutter Compensation (G40, G41 and G42)
- 4.2 Tool Length Offset (G43, G44 and G49)
- 4.3 Nose R Compensation (G141 and G142 - Option)
- 4.4 Tool Position Compensation (G143, G144 and G49 - Option)

## 4.1 Cutter Compensation (G40, G41 and G42)

### 4.1.1 Cutter Compensation Function

The shape of the workpiece is programmed ahead of time and then the cutter compensation function is used when actually machining the workpiece. This function takes into account the radius of the tool used and creates a tool center path that is offset according to the shape of the workpiece.

Command format



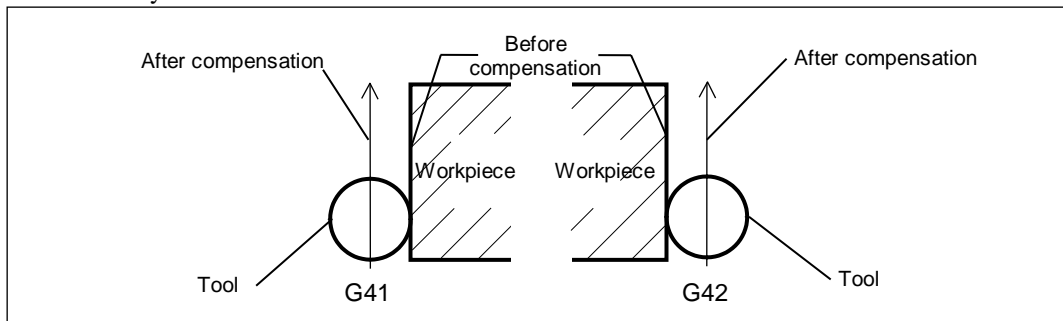
G code and D code used for cutter compensation

G40: Cutter compensation cancels (This mode is used when the power is turned ON.)

G41: Left side compensation (Offsets to left side for direction of tool travel)

G42: Right side compensation (Offsets to right side for direction of tool travel)

If either the command G41 or G42 is issued, the cutter compensation mode is enabled. This mode is cancelled by G40.



Dn: Tool No. (n = 0 to 99), or Group No. (n = 901 to 930)

The amount that is offset for D0 is always 0.

The offset can be set on the tool list screen, or by inputting (G10) the tool data.

- (Note 1) When a command is issued with zero travel, or when there are no travel commands on the X- and Y-axes for more than 3 blocks, the infeed will be too much or too little.
- (Note 2) When the cutter compensation range is set for the tool specified in D code, the range is checked. The alarm <<Comm. issued to area other than (tool) data area>> is triggered when the command area is outside of the range.
- (Note 3) If a cutter compensation command (G41 and G42) is issued during G141 and G142 modals, an alarm is triggered. Note, if a tool change command (G100 and M06) is issued on the same block, no alarm is triggered.
- (Note 4) When a cutter compensation command (G141 and G142) is issued while the feature coordinate is being set (after G68.2 command and before G53.1 command), the alarm <<Feature coordinate manufacturing mode engaged>> is triggered. A command is possible while the feature coordinate is being indexed (after G53.1 command).

#### 4.1.1.1 Cutter Wear Offset

When G41 or G42 command is issued in the program, the cutter wear offset is added to the cutter compensation on the tool number for the command. The cutter wear offset can be set on the tool list screen.

Cutter offset = Cutter compensation + Cutter wear offset

- (Note) When the cutter wear offset range is set for the tool specified in D code, the range is checked. The alarm <<Comm. issued to area other than (tool) data area>> is triggered when the command area is outside of the range.

### 4.1.2 Cancel Mode

This mode refers to when the cutter compensation is disabled such as when the power is turned ON or when the **[RST]** key is pressed.

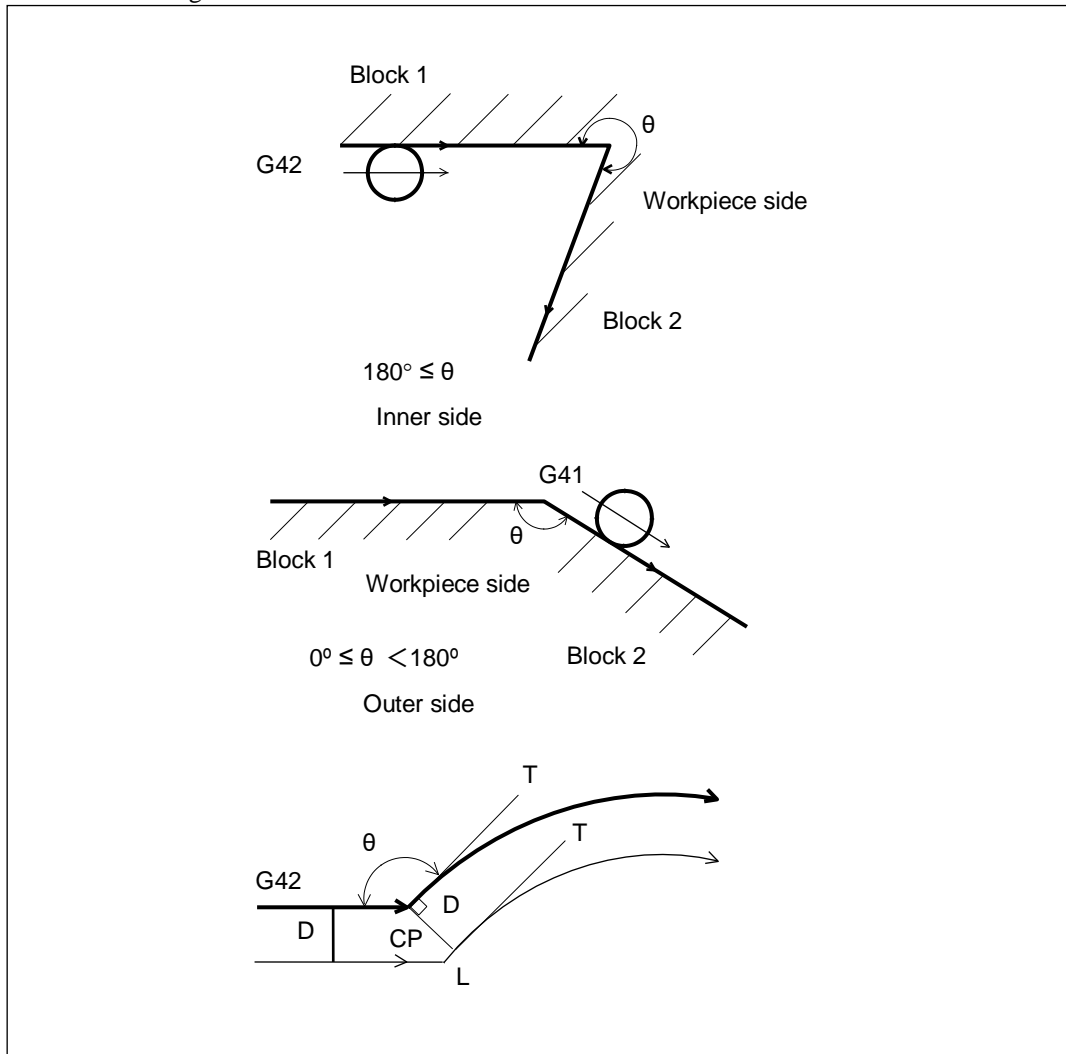
Normally, the tool path that is programmed matches the center path of the actual tool.

<Term / symbol explanation>: The terms and symbols that will appear hereafter in the program explanation are described below.

1. Description of inner side and outer side

The terms outer side and inner side refer to the intersecting angle for the travel command.

Inner side means that the angle measures more than  $180^\circ$  on the workpiece side. Outer side means the angle measures between  $0$  and  $180^\circ$ .



2. Explanation of symbols in diagram

	: Program path
	: Tool center path
	: Auxiliary line
L	: Straight line
C	: Arc
D	: Cutter compensation
$\theta$	: Angle on workpiece side
T	: Arc tangent line
CP	: Intersection
S	: Single block stop point

### 4.1.3 Startup

Offset mode is enabled for the control when a command that meets all the conditions below is executed for cancel mode. Startup refers to the travel operation in this situation.

1. G41 or G42 command is issued.
2. G0 or G1 travel command is issued, and the travel amount  $\neq 0$ .

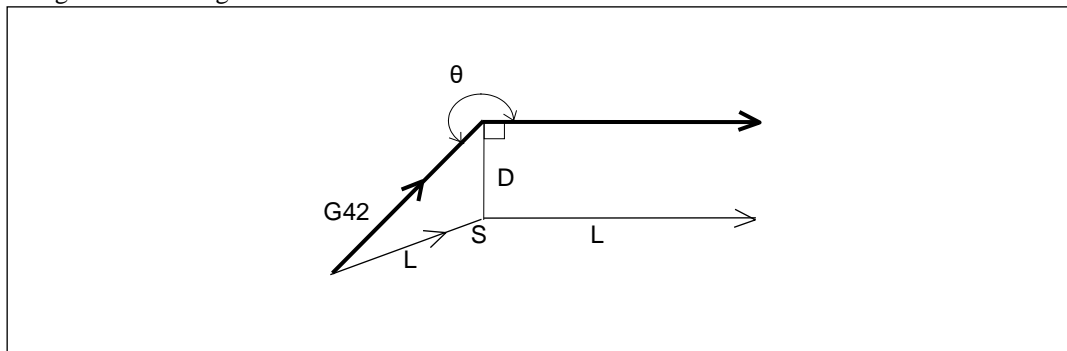
(Note 1) The alarm <<Cutter compensation error>> is triggered when an arc command or an involute interpolation is issued.

(Note 2) Execute one of the following travel commands (G0, G1, G2, G3, G02.2 or G03.2) first before issuing a G41 or G42 command.

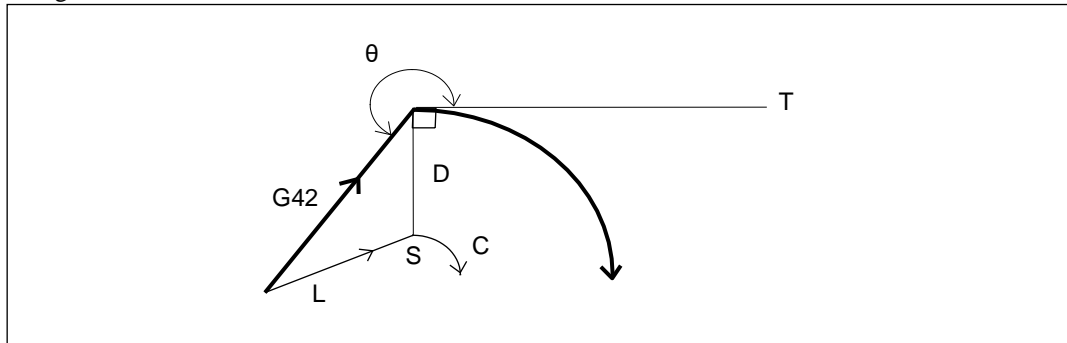
(Note 3) There are two setting types for the startup and cancel operations: <Type 1 (shortcut)> and <Type 2 (detour)>. Use the user parameter <Startup/cancel> to set one of the types.

#### 4.1.3.1 Inner Side Cutting ( $180^\circ \leq \theta$ )

Straight line to straight line

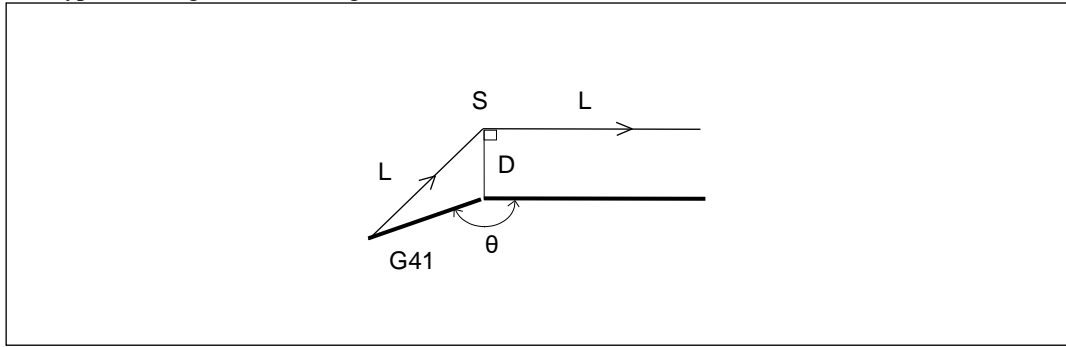


Straight line to arc

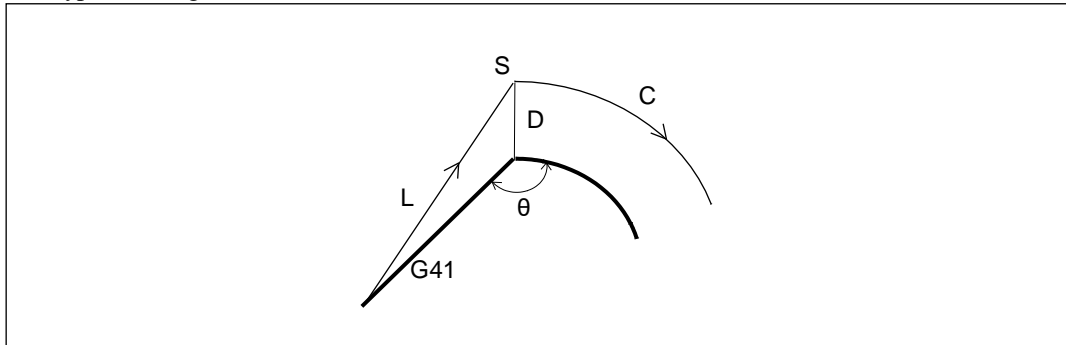


### 4.1.3.2 Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )

1. Type 1: Straight line to straight line

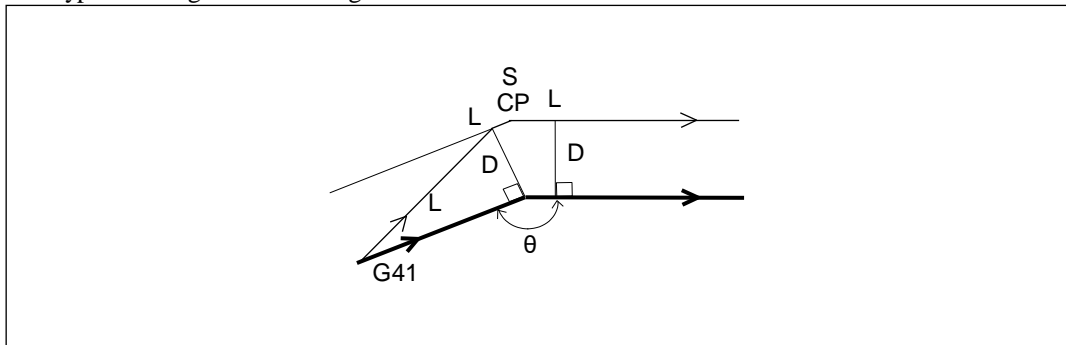


Type 1: Straight line to arc

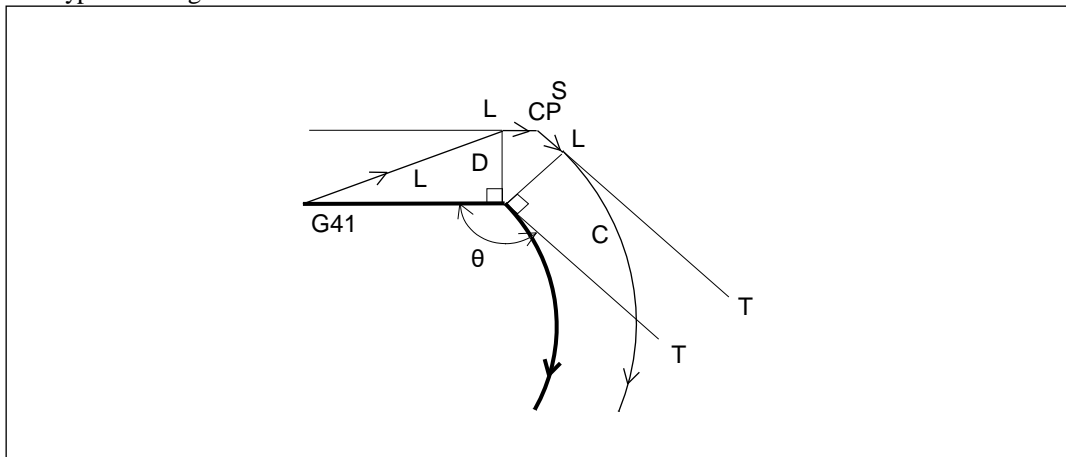


4

2. Type 2: Straight line to straight line

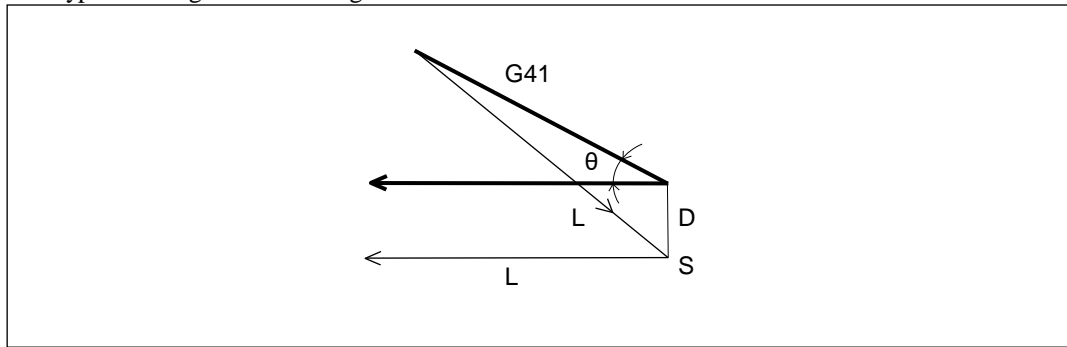


Type 2: Straight line to arc

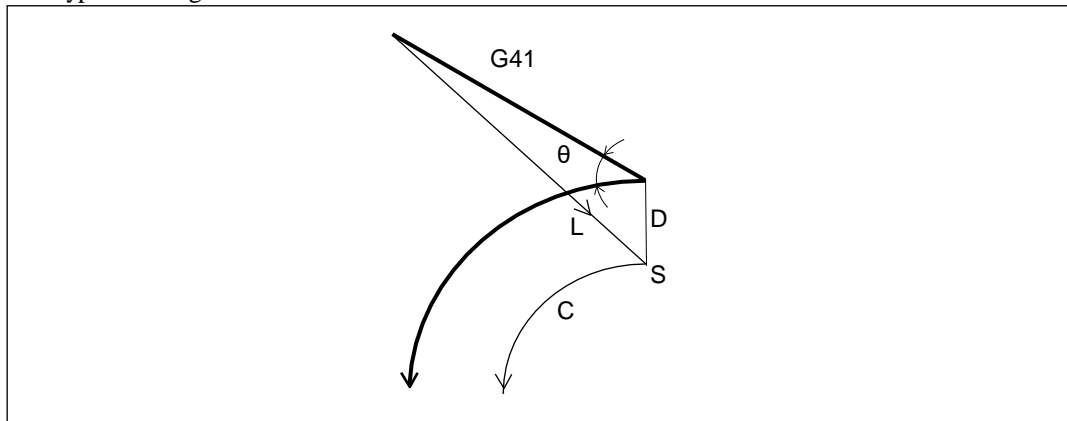


### 4.1.3.3 Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )

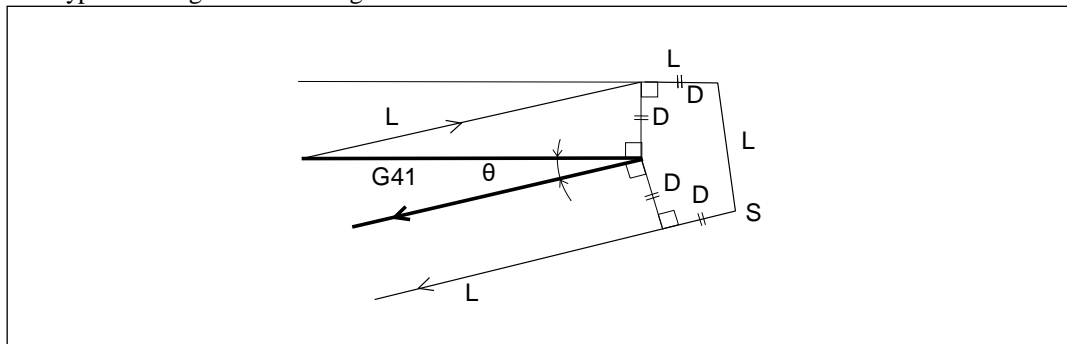
1. Type 1: Straight line to straight line



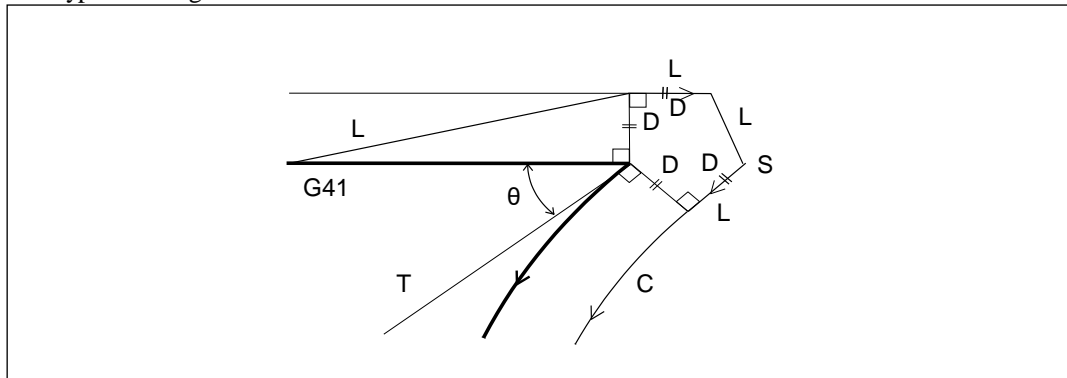
Type 1: Straight line to arc



2. Type 2: Straight line to straight line



Type 2: Straight line to arc



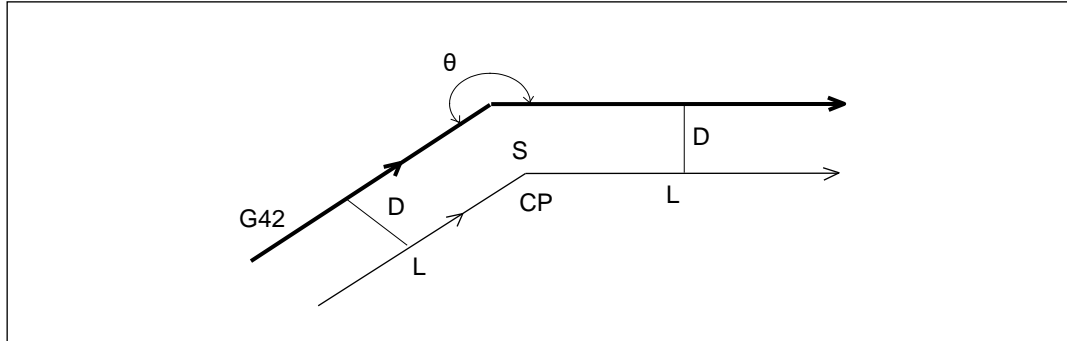
**(Note)** When  $\theta \leq 1^\circ$ , the setting <0: Type 1 (shortcut)> is used or enabled, even if <1: Type 2 (detour)> is specified for the user parameter <Start up/cancel>.

### 4.1.4 Offset Mode

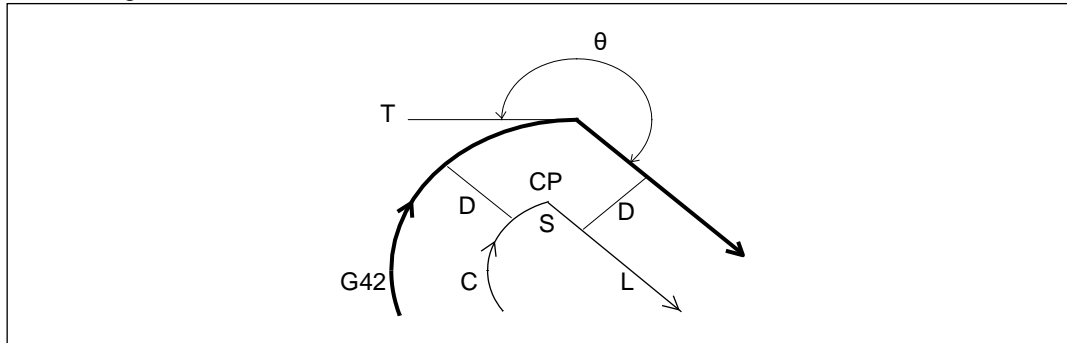
The travel commands in offset mode include: positioning, linear interpolation, circular interpolation, helical interpolation and involute interpolation.

#### 4.1.4.1 Inner Side Cutting ( $180^\circ \leq \theta$ )

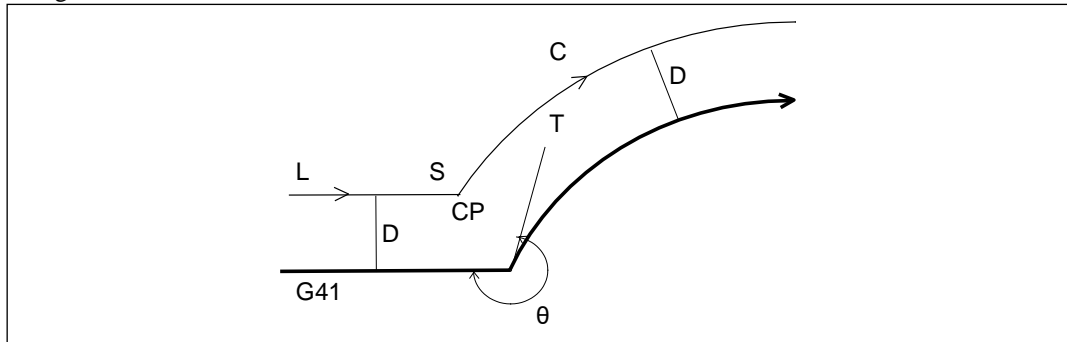
Straight line to straight line



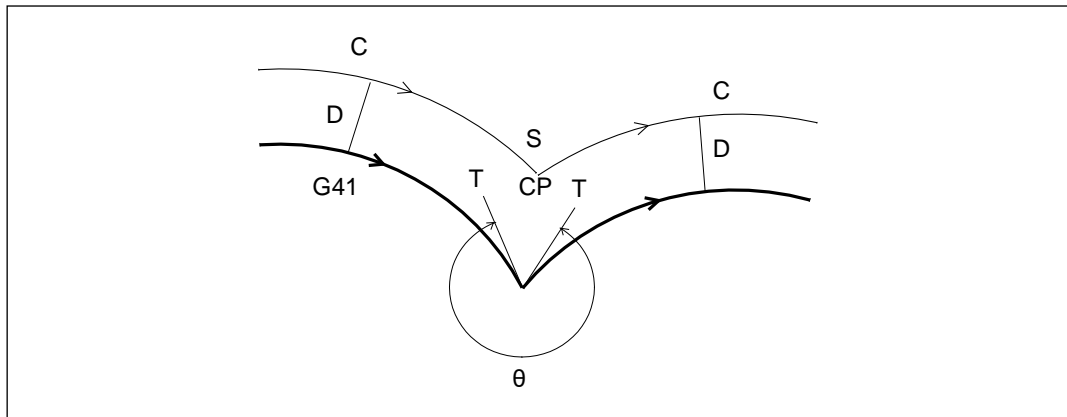
Arc to straight line



Straight line to arc

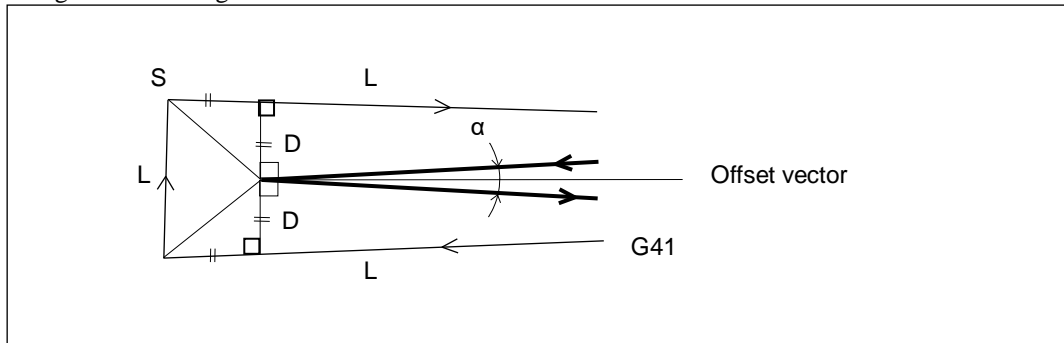


Arc to arc

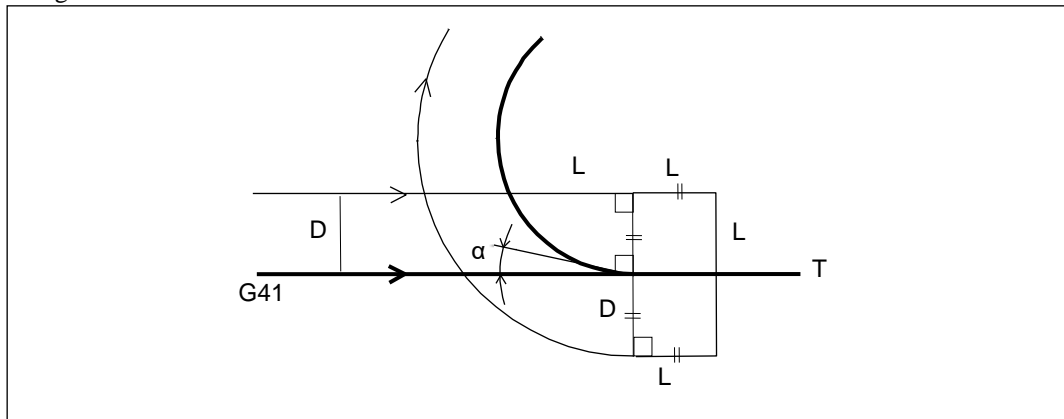


(Note 1) When turning on the inner side of a narrow angle ( $\alpha < 1^\circ$ ), and the offset vector is abnormally large.

Straight line to straight line

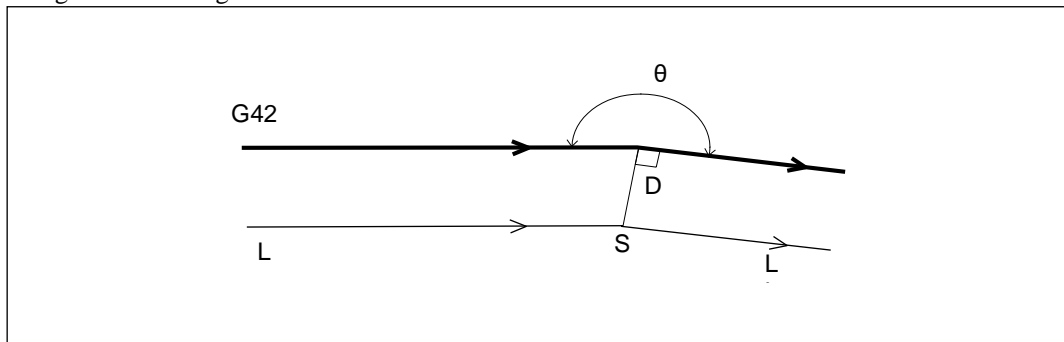


Straight line to arc



(Note 2) When turning on the inner side of an angle that is almost parallel ( $180^\circ \leq \theta < 181^\circ$ ).

Straight line to straight line

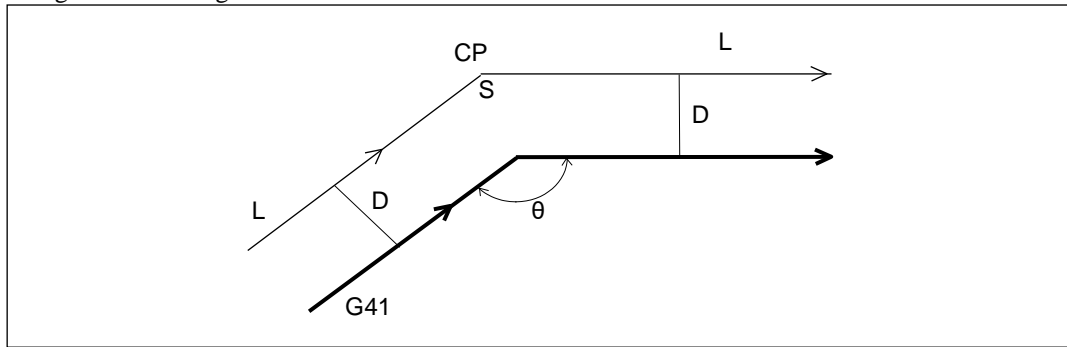


The processing is the same for: arc  $\rightarrow$  straight line, straight line  $\rightarrow$  arc and arc  $\rightarrow$  arc.

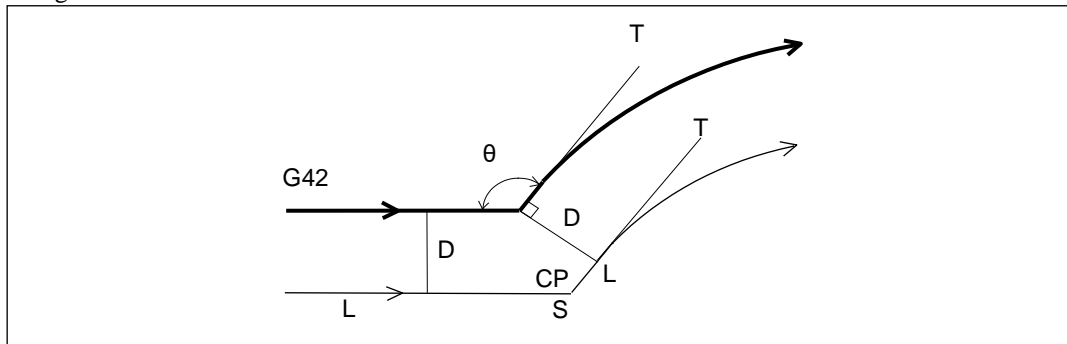


### 4.1.4.2 Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )

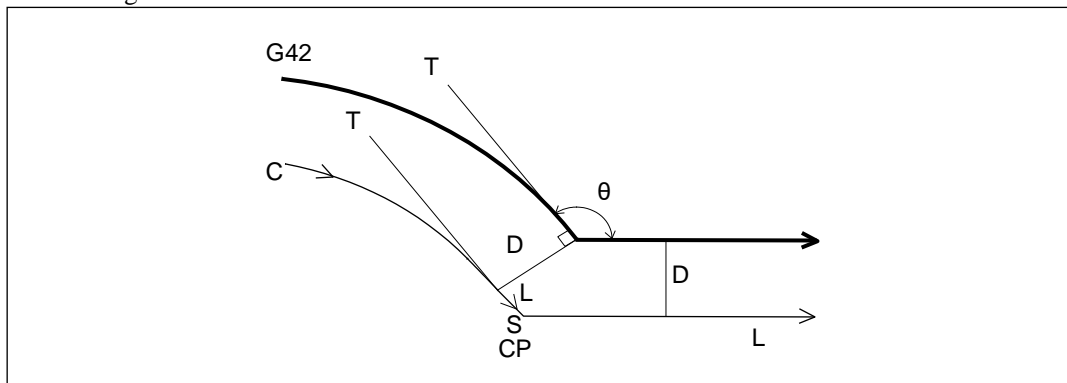
Straight line to straight line



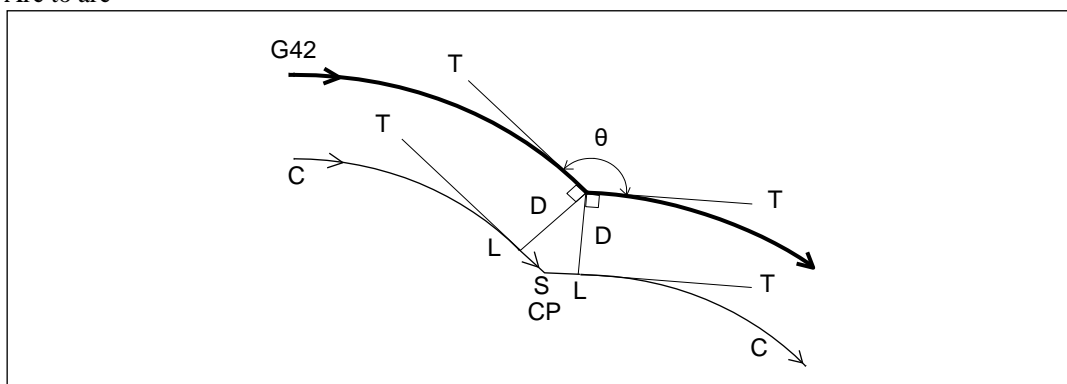
Straight line to arc



Arc to straight line

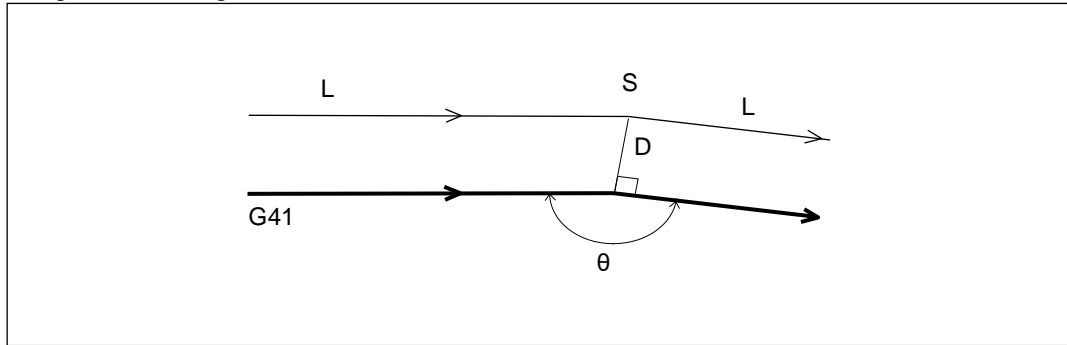


Arc to arc



(Note) When turning on the outer side of an angle that is almost parallel ( $179^\circ \leq \theta < 180^\circ$ ).

Straight line to straight line

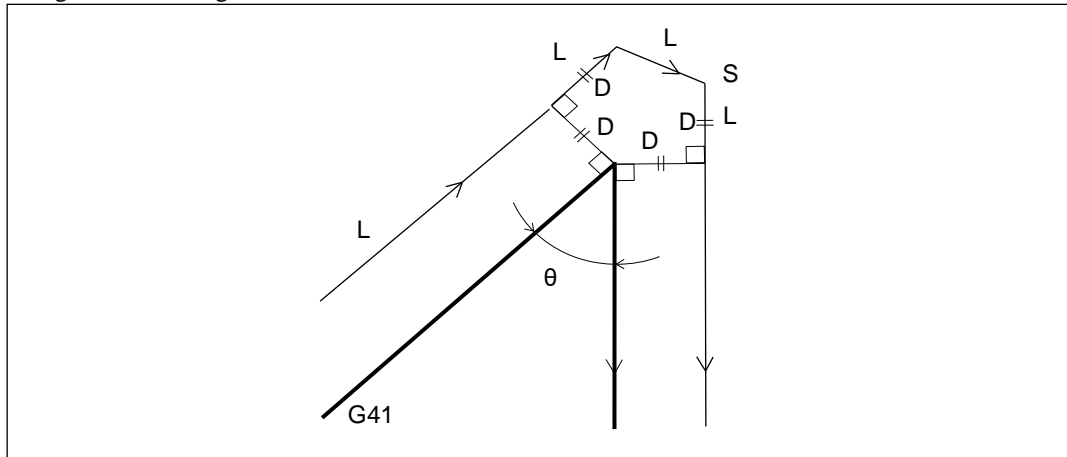


The processing is the same for: arc  $\rightarrow$  straight line, straight line  $\rightarrow$  arc and arc  $\rightarrow$  arc.

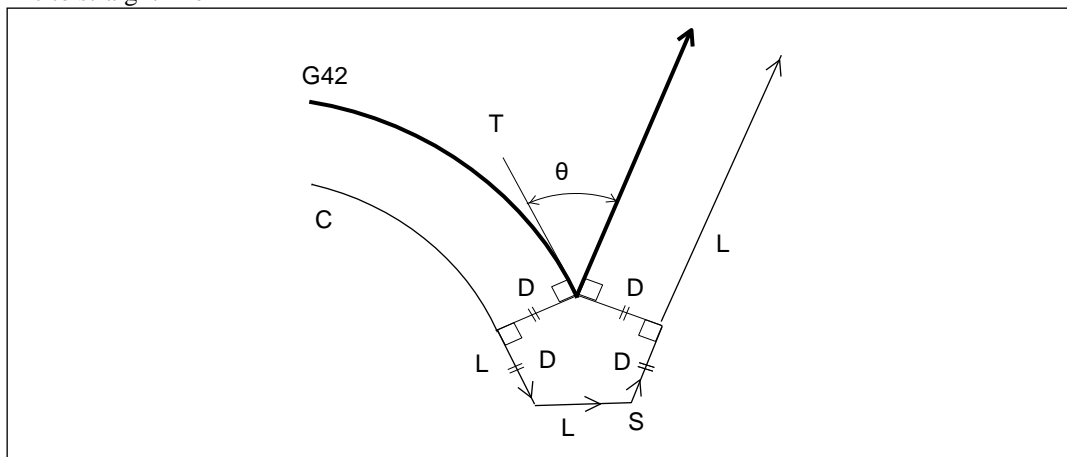
4

### 4.1.4.3 Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )

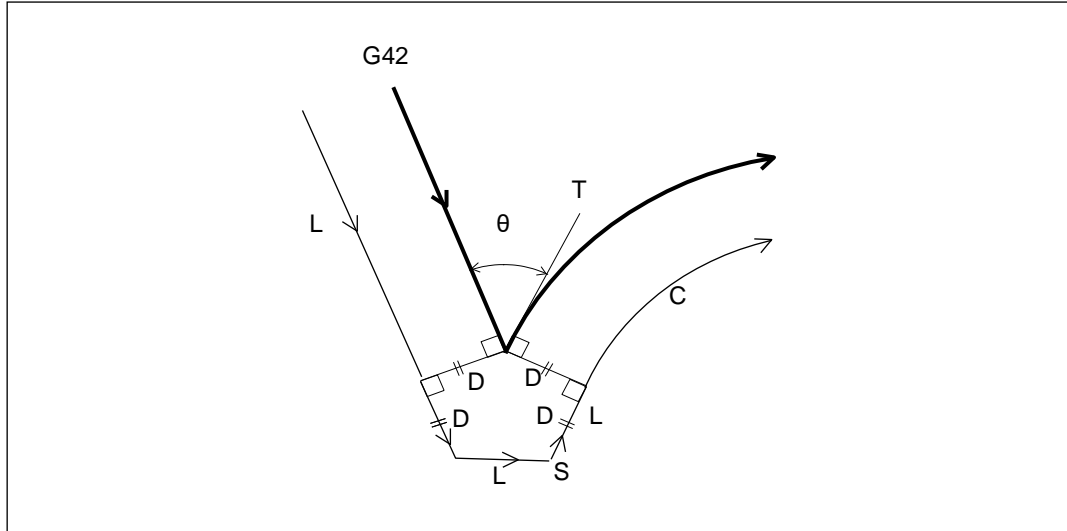
Straight line to straight line



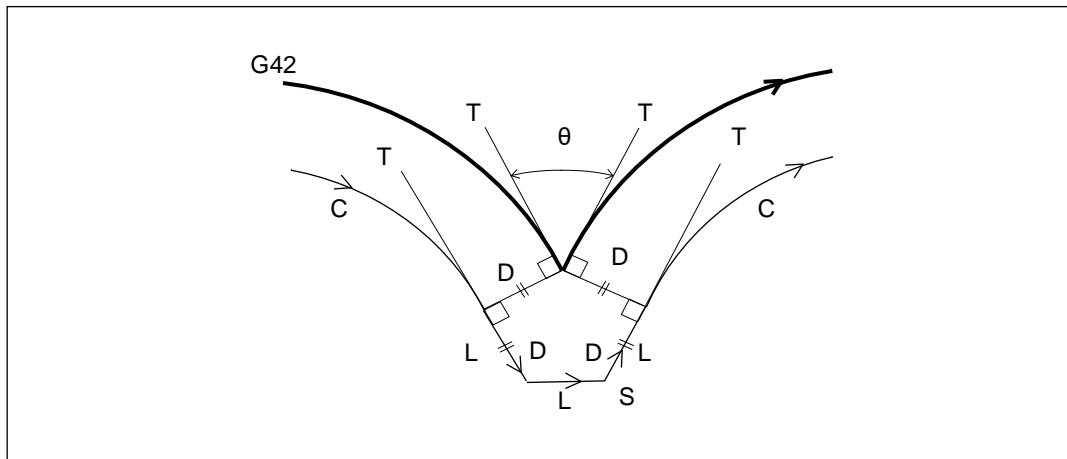
Arc to straight line



Straight line to arc

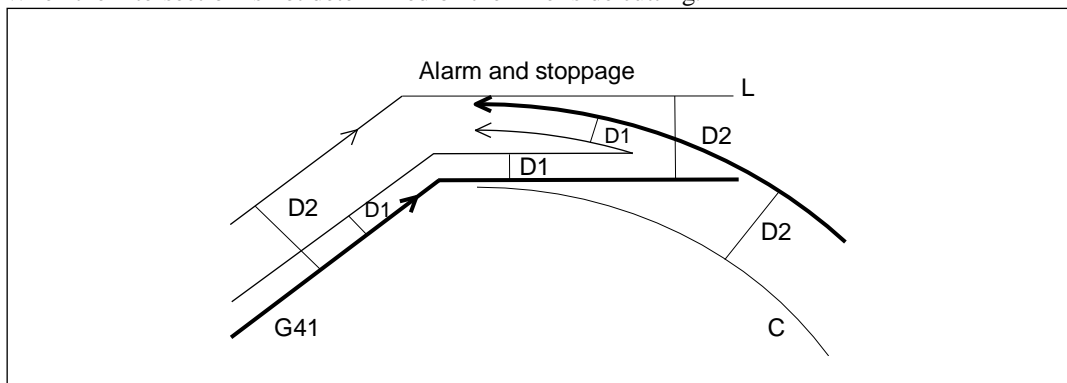


Arc to arc



#### 4.1.4.4 Exceptional Cases

When the intersection is not determined on the inner side cutting.



When the compensation is a small amount as shown in the diagram above, even if an intersection exists, increasing the compensation may cause the intersection to disappear. In this situation, the alarm <<Cutter compensation too large>> is triggered at the previous block and operation stops.

### 4.1.5 Offset Cancel

Cancel mode is enabled for the control when a command that meets all the conditions below is executed for offset mode. Offset cancel refers to the travel operation in this situation.

1. G40 command is issued.

Command format

**G40;**

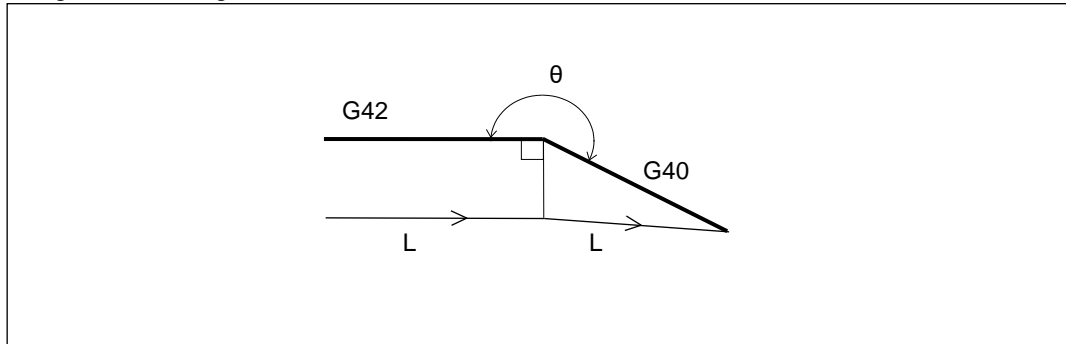
2. A travel command is issued excluding an arc, involute interpolation and thread cutting command.

**(Note 1)** When an arc, involute interpolation or thread cutting command is issued, an alarm is triggered.

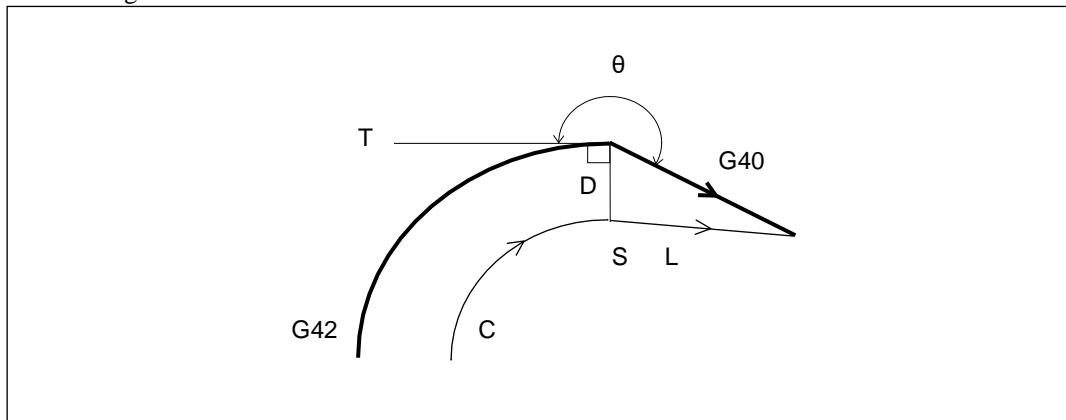
**(Note 2)** There are two setting types for the startup and cancel operations: <Type 1 (shortcut)> and <Type 2 (detour)>. Use the user parameter <Start up/cancel> to set one of the types.

#### 4.1.5.1 Inner Side Cutting ( $180^\circ \leq \theta$ )

Straight line to straight line

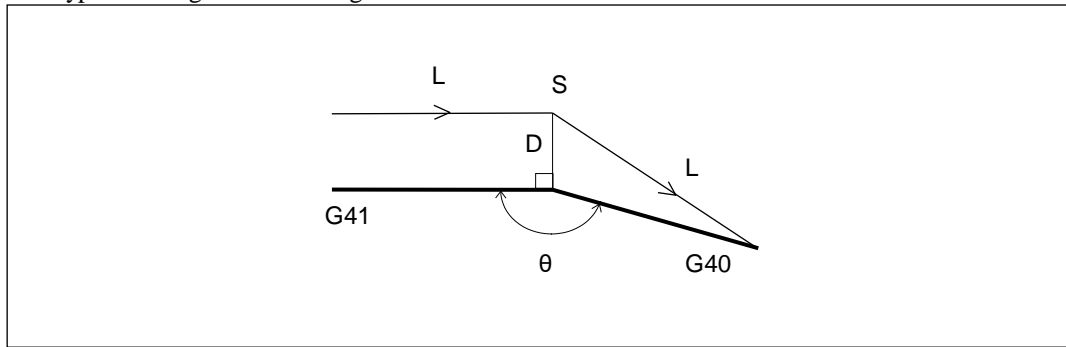


Arc to straight line

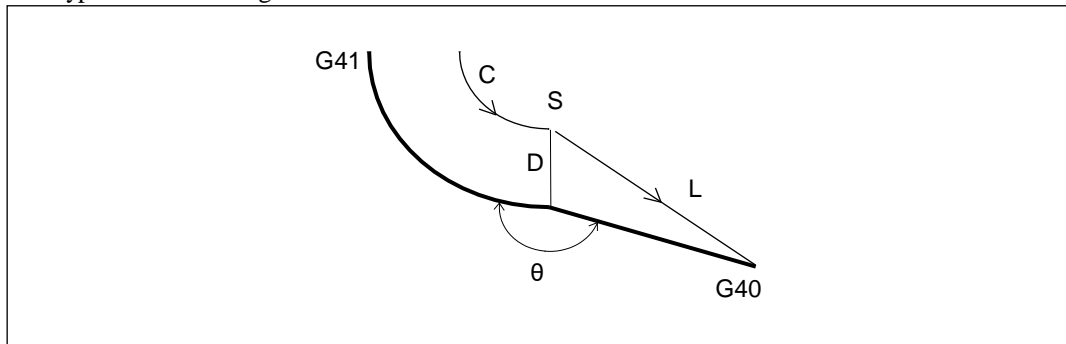


### 4.1.5.2 Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )

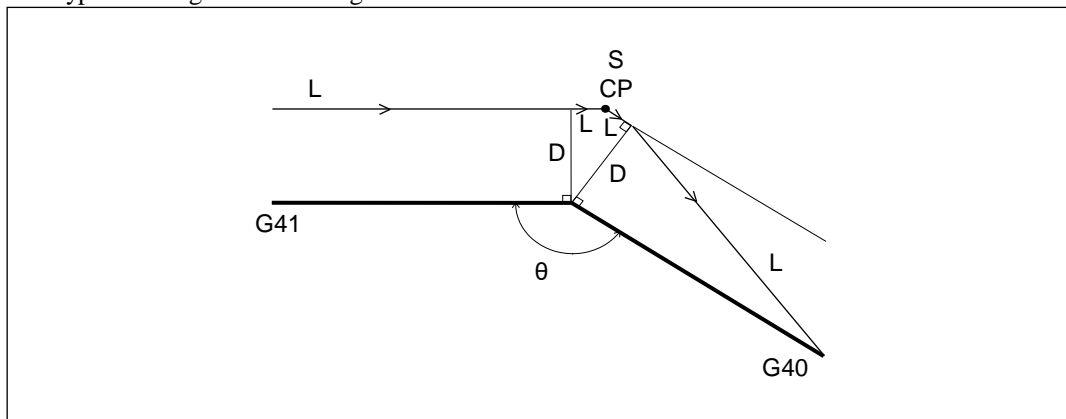
1. Type 1: Straight line to straight line



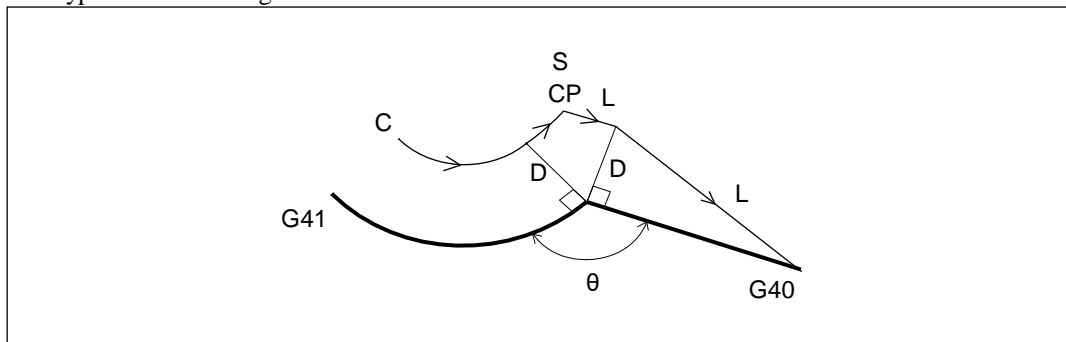
Type 1: Arc to straight line



2. Type 2: Straight line to straight line



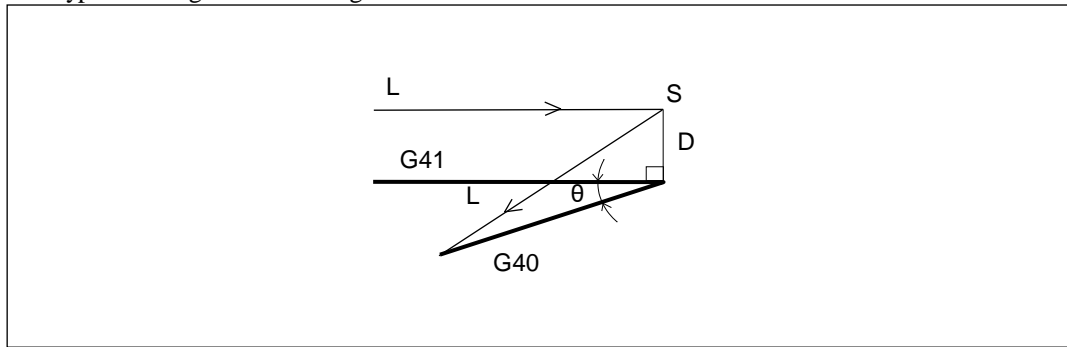
Type 2: Arc to straight line



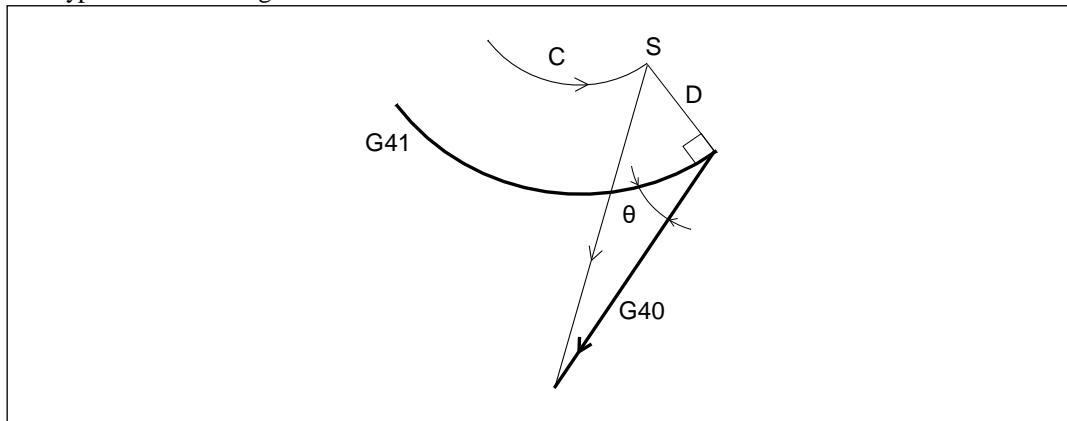
**(Note)** When  $179^\circ \leq \theta \leq 180^\circ$ , the setting <0: Type 1 (shortcut)> is used or enabled, even if <1: Type 2 (detour)> is specified for the user parameter <Start up/cancel>.

### 4.1.5.3 Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )

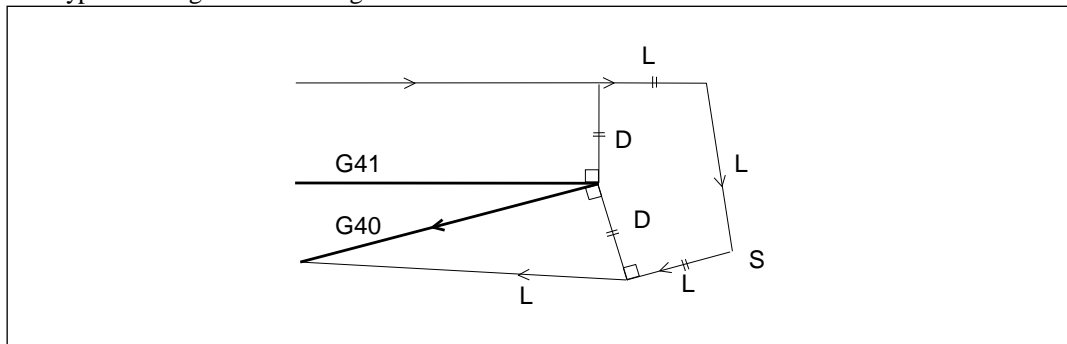
1. Type 1: Straight line to straight line



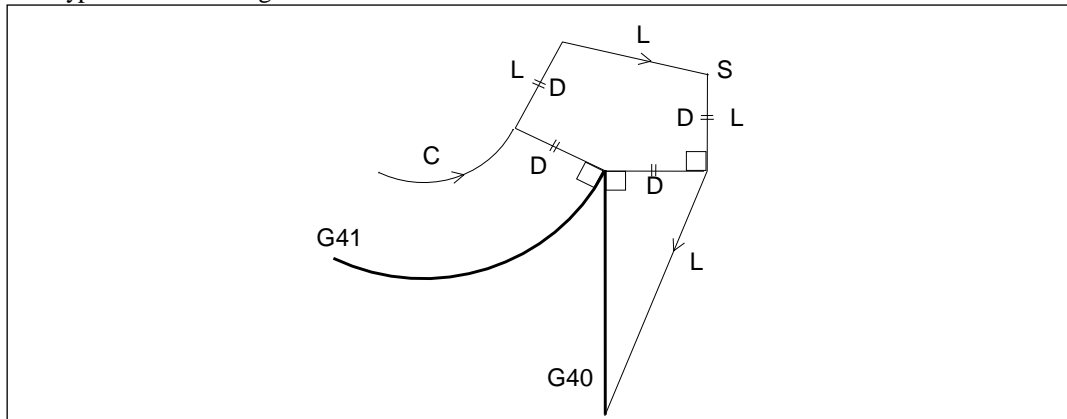
- Type 1: Arc to straight line



2. Type 2: Straight line to straight line



- Type 2: Arc to straight line

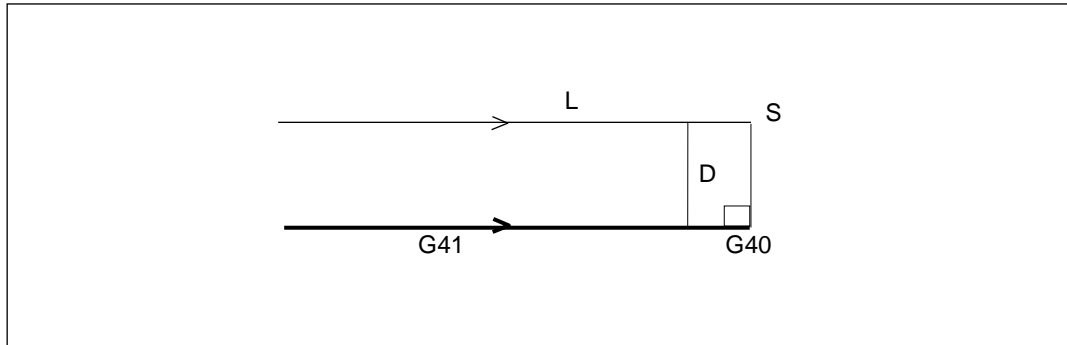


### 4.1.6 G40 Individual Command

When G40 is issued as an individual command, the machine travels to a position for the cutter compensation that is offset perpendicularly from the command value for the previous block.

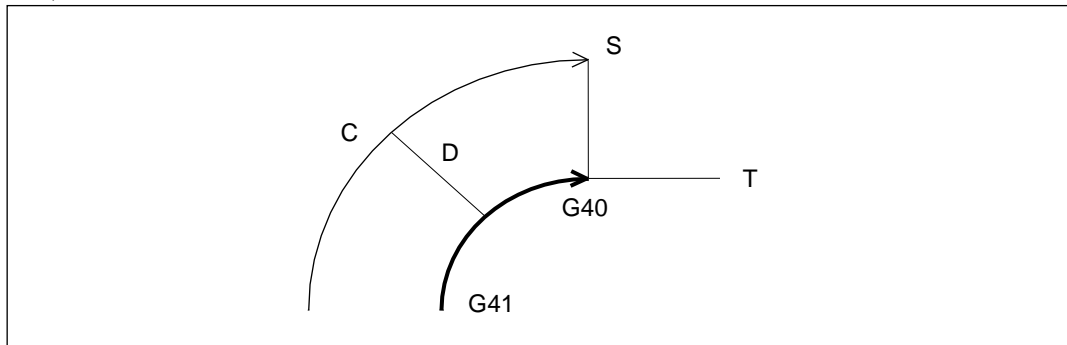
Straight line to straight line

G41 X\_Y\_D\_;  
G40;



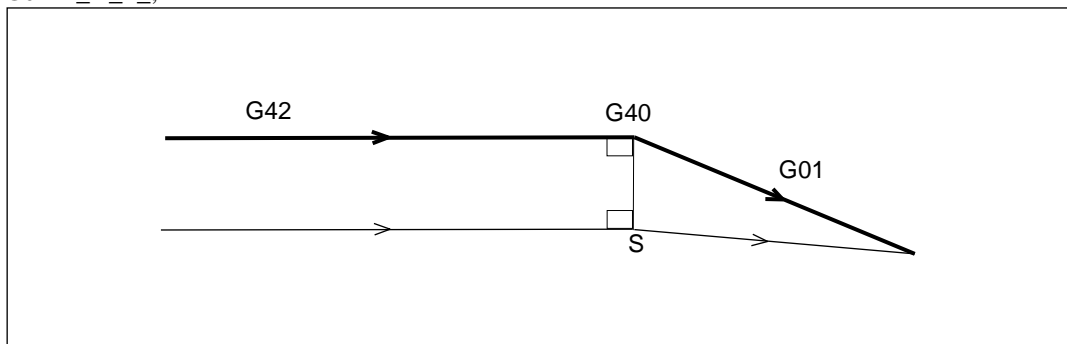
Arc to straight line

G41 X\_Y\_D\_;  
G40;



**(Note)** The remaining offset is cancelled together with the next travel command.

G42 X\_Y\_D\_;  
G40 ;  
G01 X\_Y\_F\_;



### 4.1.7 Compensation Direction Change in Offset Mode

The compensation direction can be changed even while offset mode is enabled, by issuing a G41 or G42 command, or reversing the positive/negative sign for the compensation.

However, when compensation is set to 0, the compensation is processed as a positive amount.

Note, the block following the startup block cannot be changed.

In addition, the compensation direction also cannot be changed even when changed using the mirror (single axis specification) or D address value, etc.

G code \ Offset sign	+	-
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

Execution conditions

Offset mode	Command	Straight line to straight line	Straight line to arc	Arc to straight line	Arc to arc
G41	G41	Executes (The stop point is offset by the cutter compensation perpendicular to the end point of the previous block.)			
G42	G42				
G41	G42	Executes		Executes	
G42	G41				

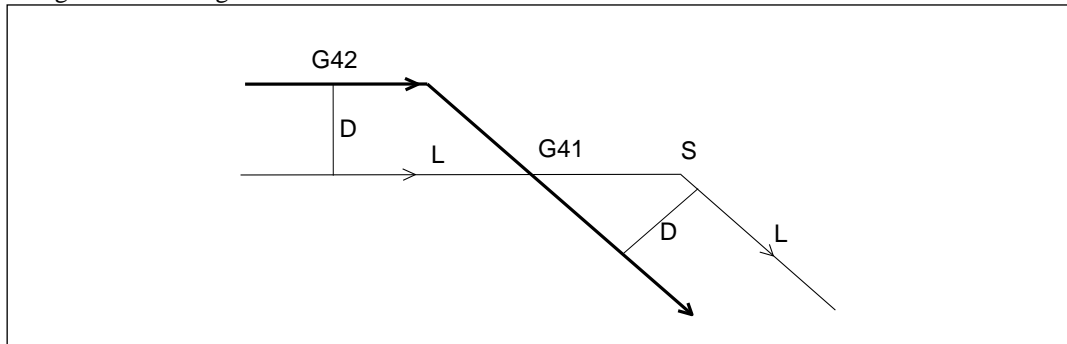
There is no distinction between the inner side and outer side cutting when changing the compensation direction, but it varies depending on whether the intersection exists or not. In the following explanation, the compensation is positive.



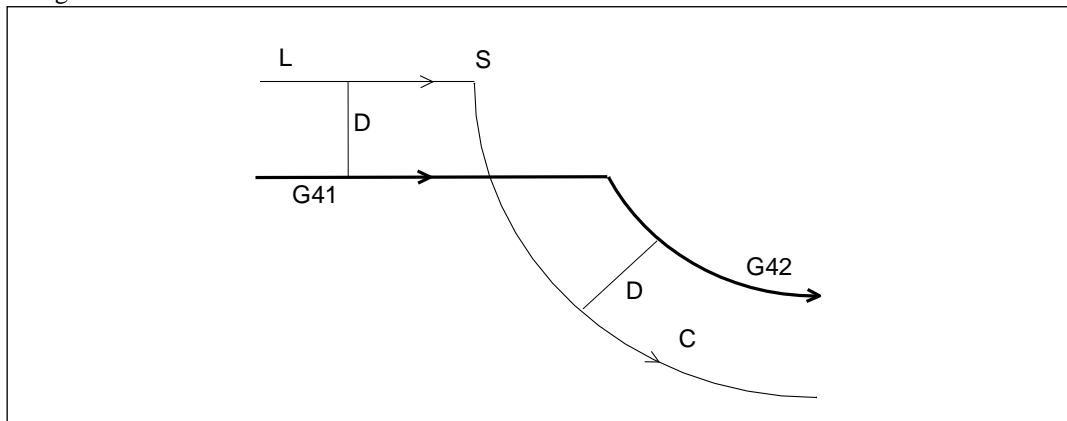
## 4.1.8 Offset Direction Change in Offset Mode

### 4.1.8.1 When There is an Intersection

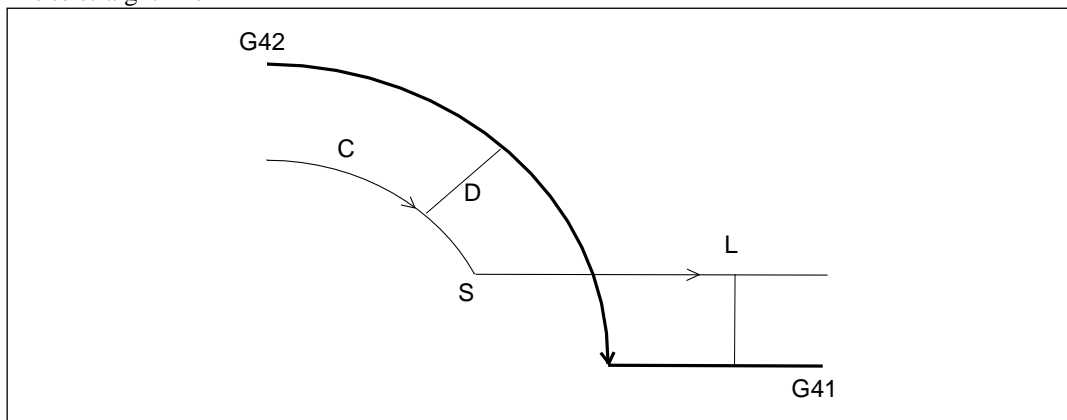
Straight line to straight line



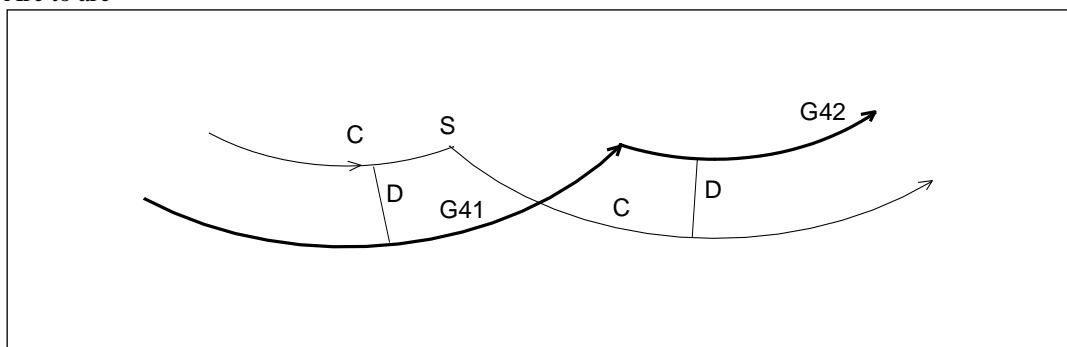
Straight line to arc



Arc to straight line

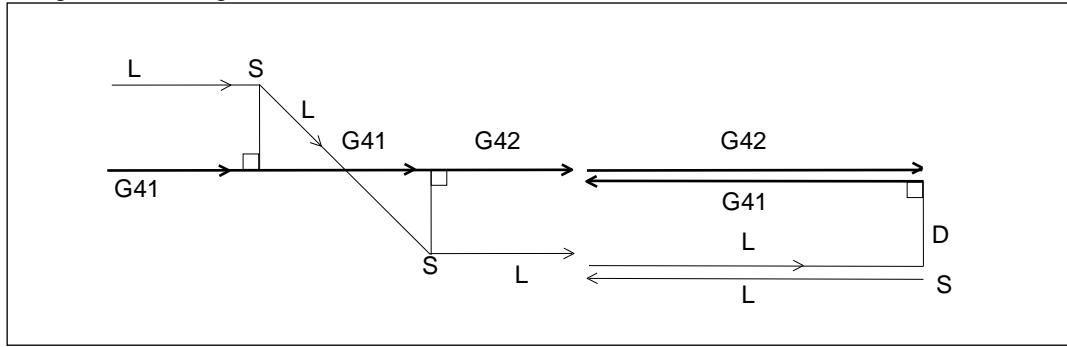


Arc to arc

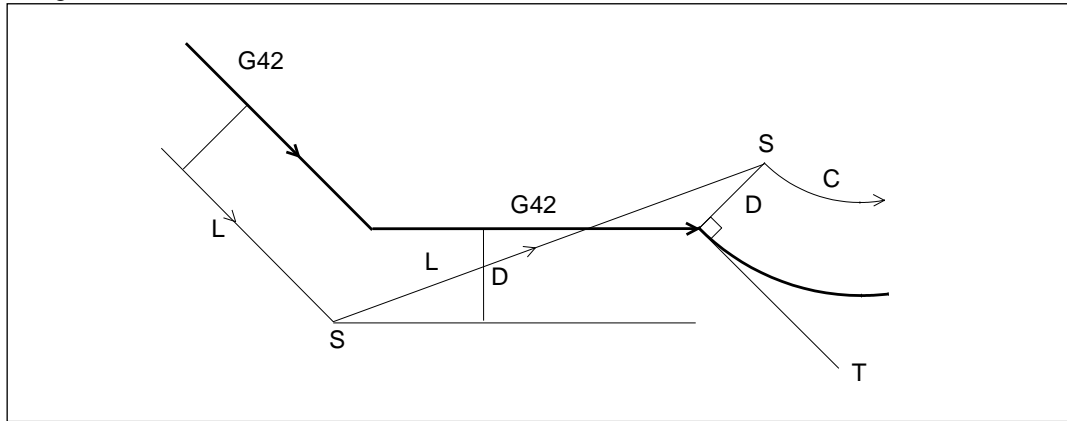


### 4.1.8.2 When There is No Intersection

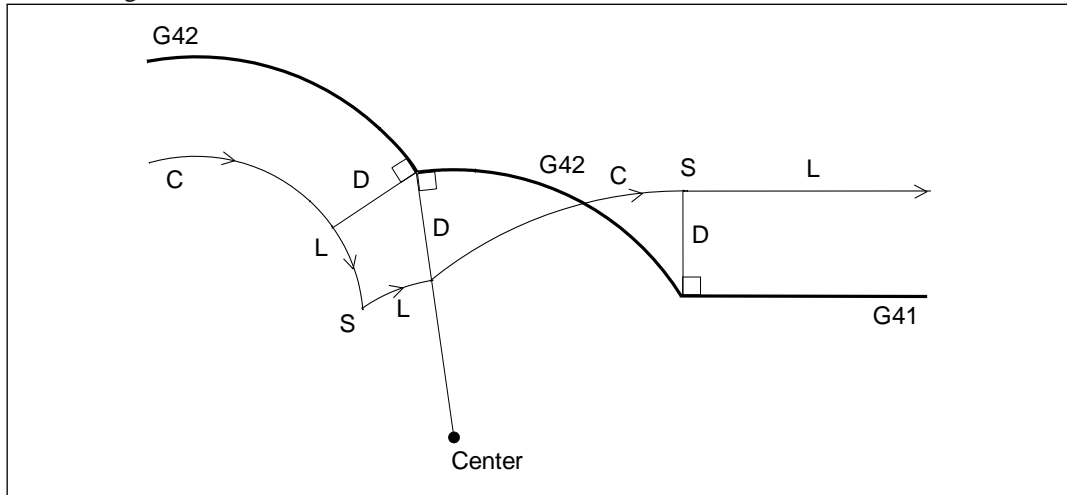
Straight line to straight line



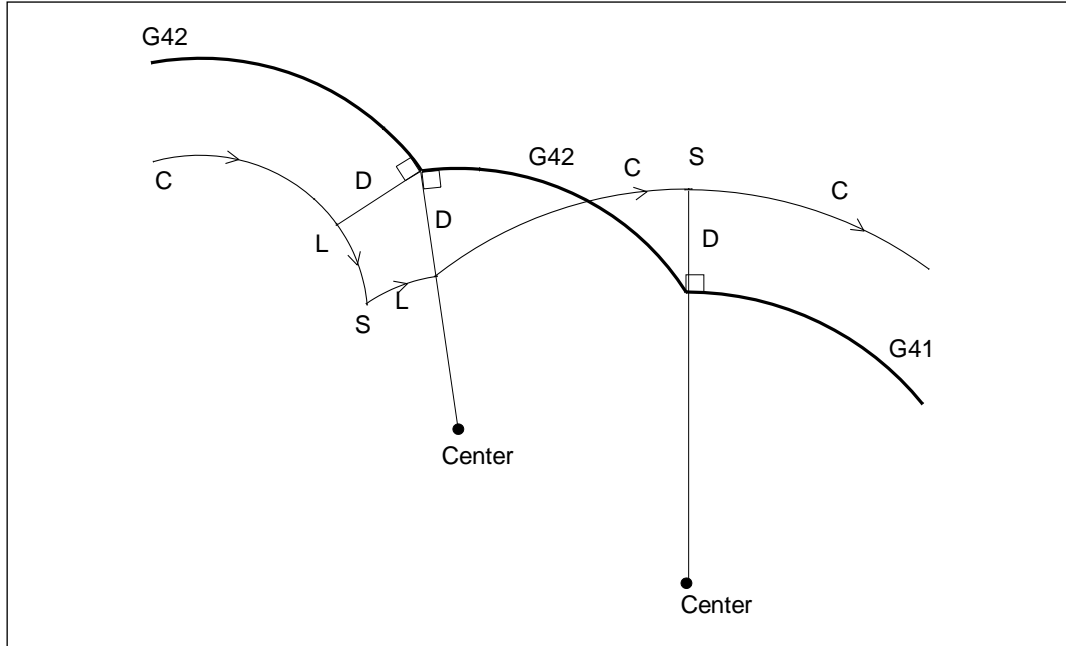
Straight line to arc



Arc to straight line



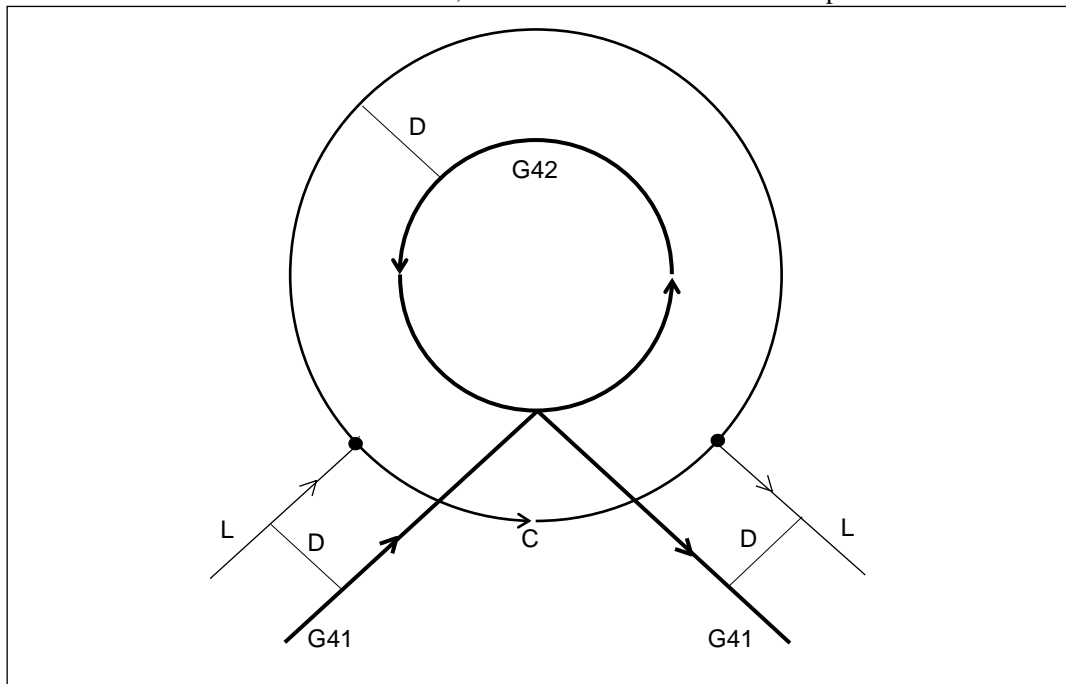
Arc to arc



4

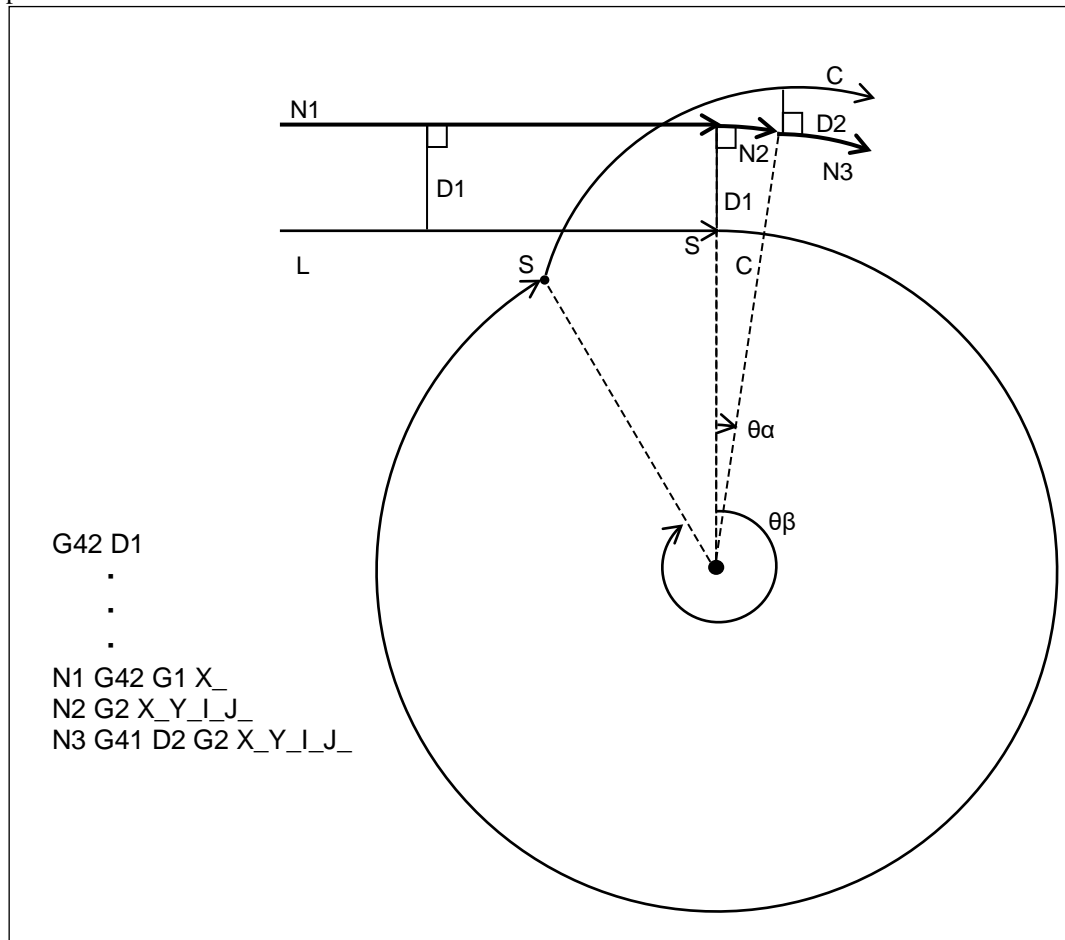
#### 4.1.8.3 When an Arc Laps Around in a Circle

If the direction of the compensation is changed and the arc laps around in a circle, a short arc is executed as shown below. In this situation, use commands to divide the arc up.



#### 4.1.8.4 Arc angle check when offset direction changes

When a G42 command is issued for a G41 modal (same for G41 command for a G42 modal) and the offset direction changes on the next block for an arc command (due to a change in cutter compensation during operation), this function stops operation before the arc operation if the arc angle for the program path is significantly different from the offset arc angle for the tool center path.



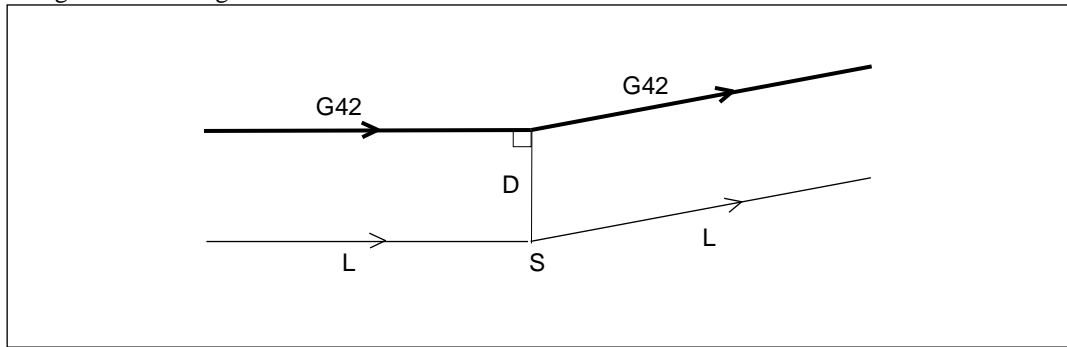
This function checks the difference between the arc angle ( $\theta\alpha$ ) at the start and end points in the N2 program path and the arc angle ( $\theta\beta$ ) at the start and end points in tool center path after being offset. If  $\theta\beta$  is greater than  $\theta\alpha$  by  $180^\circ$  or more, then an alarm is triggered and operation stops before executing N2.

**(Note)** This function carries out the check at the end point after 3 blocks of travel. If one of the situations below applies while the offset mode is enabled, this check function may not work properly because the tool center path start and end points change after being offset.

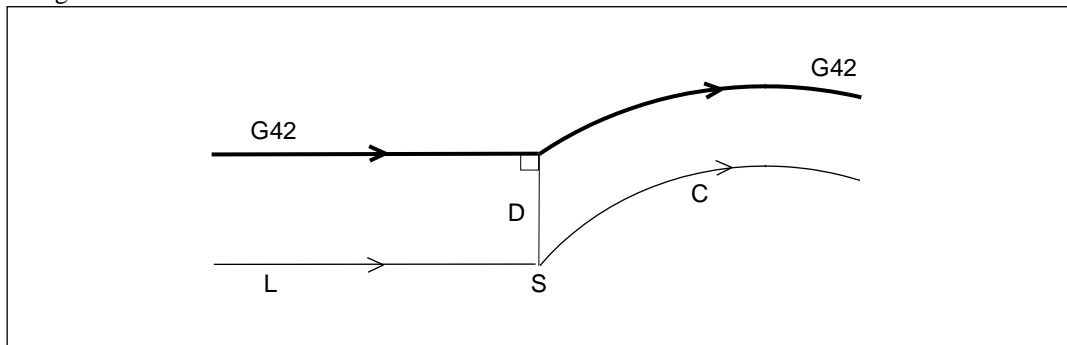
- When there is a command that sets a perpendicular vector
- When there are zero travel commands for more than 3 blocks

### 4.1.9 G Code Command for Cutter Compensation in Offset Mode

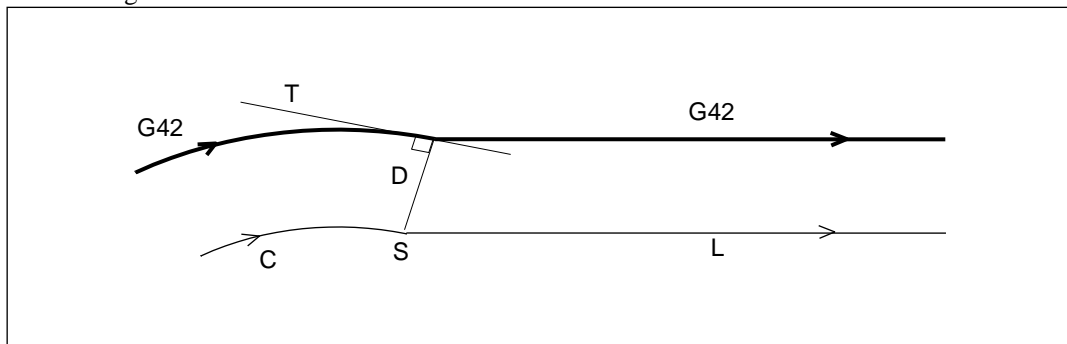
Straight line to straight line



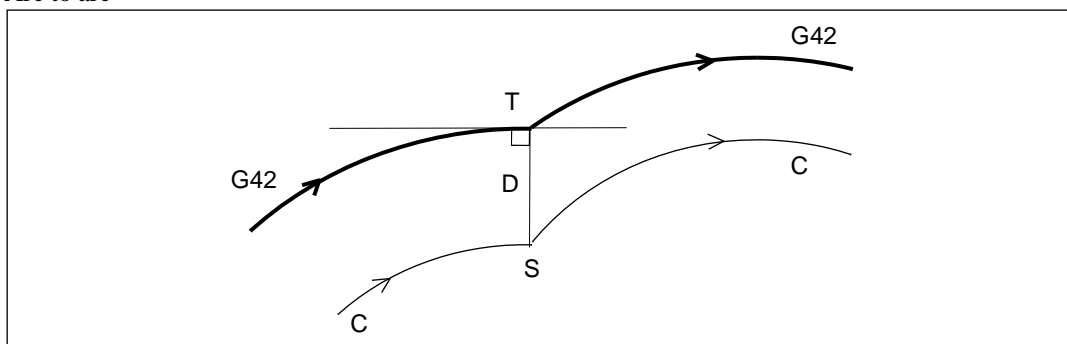
Straight line to arc



Arc to straight line



Arc to arc



### 4.1.10 Special Notes for Cutter Compensation

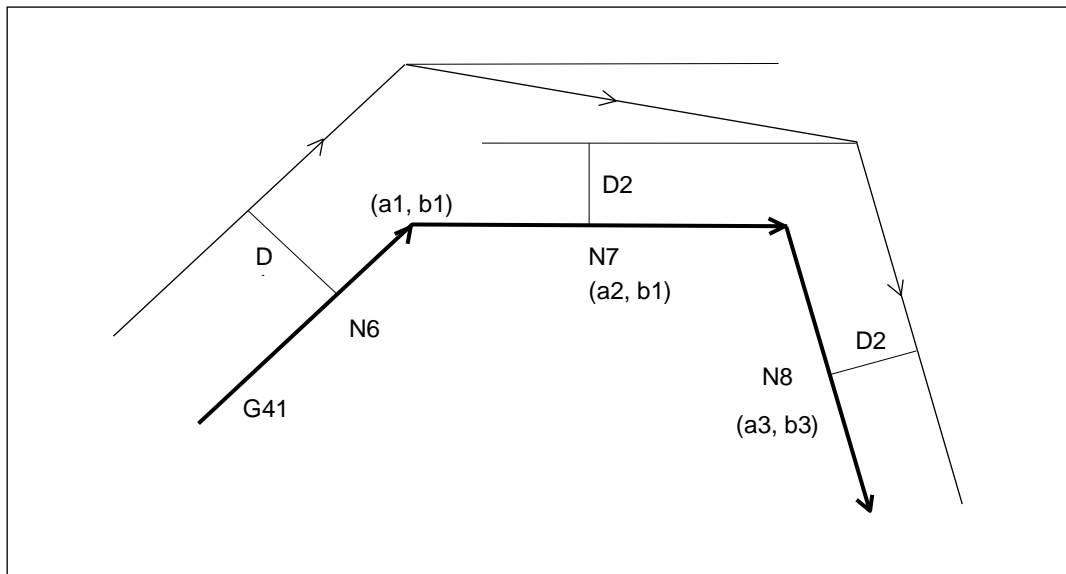
1. Tool diameter offset commands  
The offset amount is specified in the number for the D command. The command is issued for the same block when issuing a G41 or G42 command, but if that command is omitted, then the number is used for the D command that is issued previously.
2. Tool diameter offset change  
When the offset is changed while in offset mode, the new offset applies after the end point of that block.

N1 G41 X\_Y\_D1;

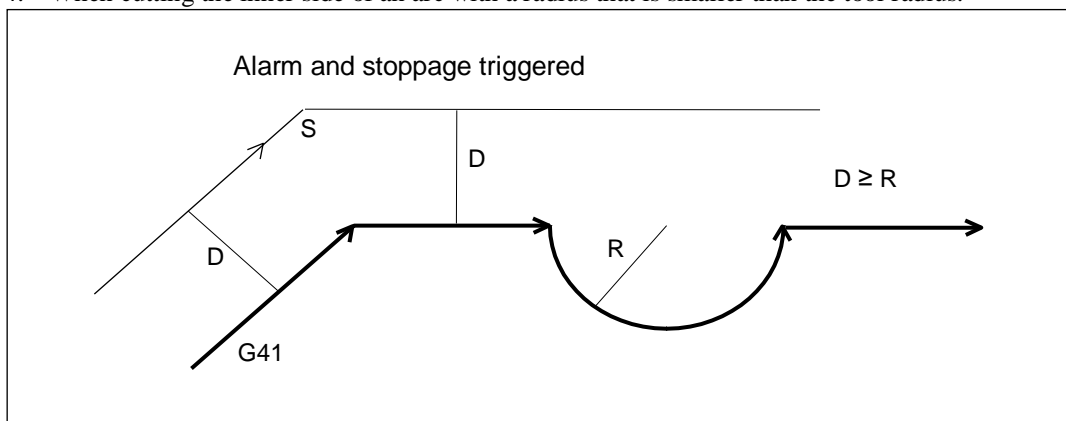
N6 Xa1 Yb1;

N7 Xa2 D2; ..... Offset changed

N8 Xa3 Yb3;



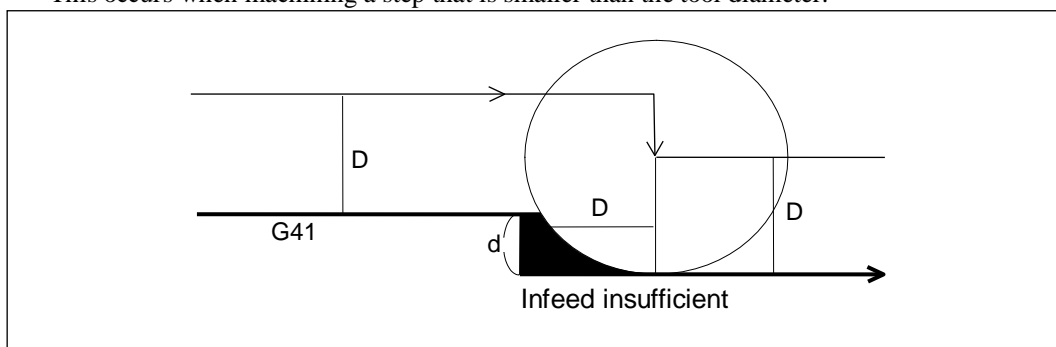
3. Current position display  
The current position display shows the tool center position.
4. When cutting the inner side of an arc with a radius that is smaller than the tool radius.



In this situation, the alarm <<Cutter compensation too large>> and stoppage are triggered because the infeed is not possible. It stops at the end point of the previous block.

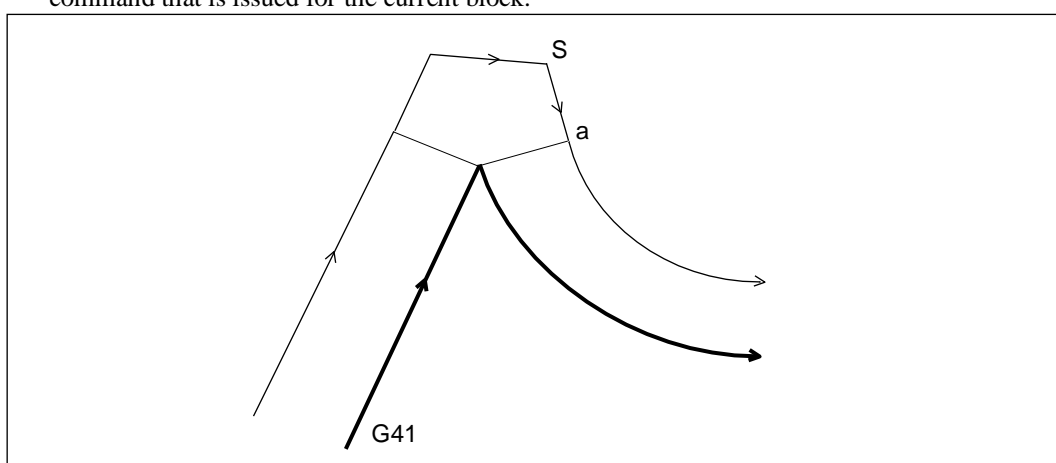
### 5. Infeed insufficient

This occurs when machining a step that is smaller than the tool diameter.

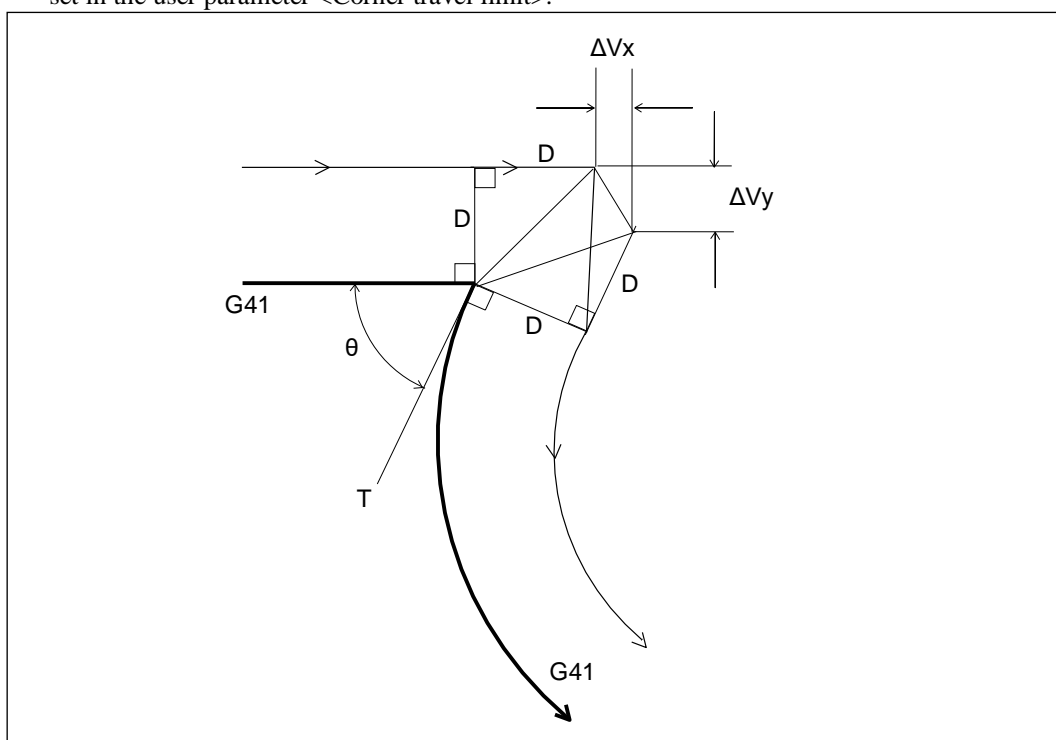


## 6. Corner travel

When cutting the outer side, it can turn on a corner with many angles. The travel mode and feedrate when turning the corner up to point *a* in the diagram below are based on the command that is issued for the current block.

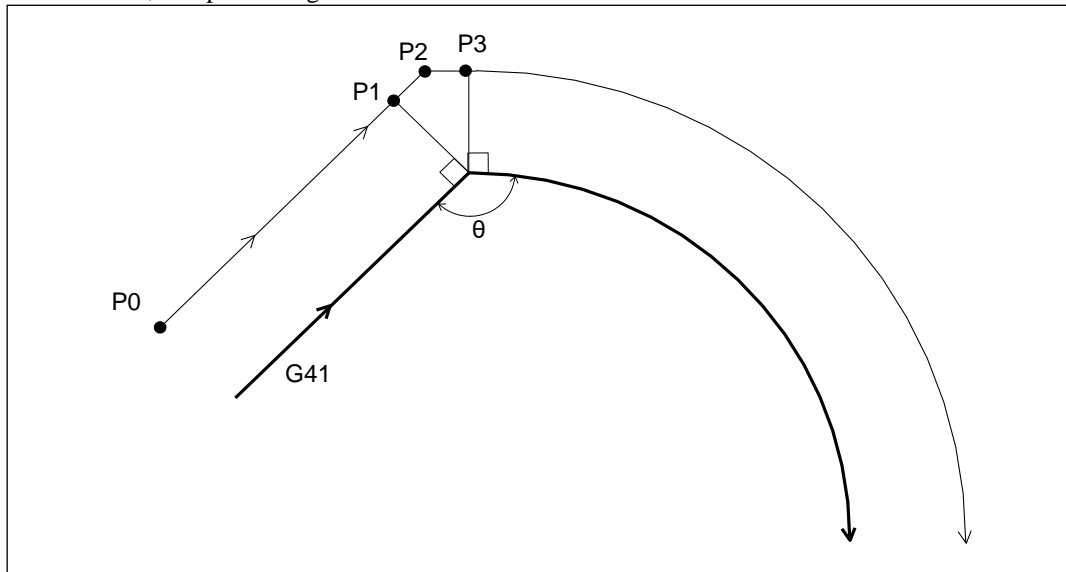


In addition, as shown in the diagram below, the travel operation is ignored when the corner travel distance is extremely small, and when  $\Delta VX \leq \Delta V$  and  $\Delta VY \leq \Delta V$ . The value for  $\Delta V$  is set in the user parameter <Corner travel limit>.



As a result, extremely small travel operations for a corner can be kept to a minimum.

However, this processing is not carried out when the next block is a circle.



Breakdown of travel in the above diagram:

P0-P1-P2      Travel in a straight line

P2-P3      Travel in a straight line

Thereafter, it travels in a circular arc with P3 as the target position.

In this situation, if extremely small travel operations are processed, the travel from P2 to P3 is ignored.

P0-P1-P2      Travel in a straight line

P2-P3      Travel in an arc (extremely small)

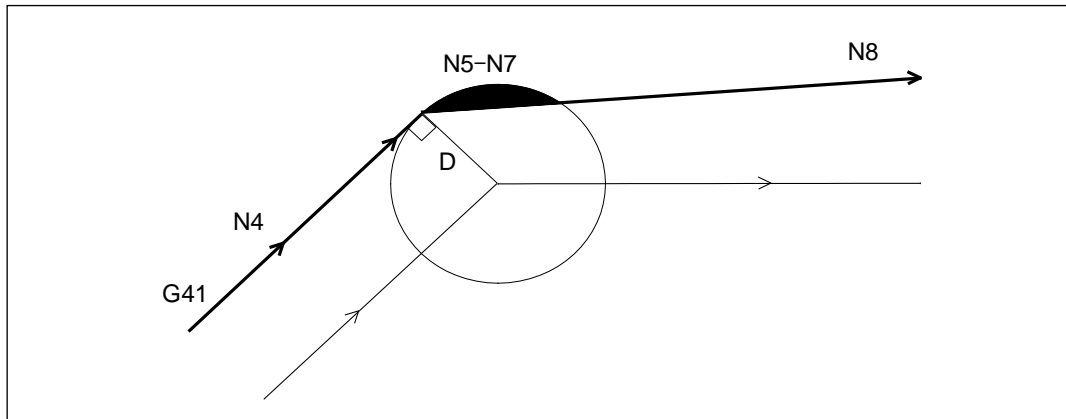
Traveling in a circular arc is ignored and travel from P2 to P3 becomes a small arc pattern. Therefore, this processing is not carried out.



7. Blocks without travel operations

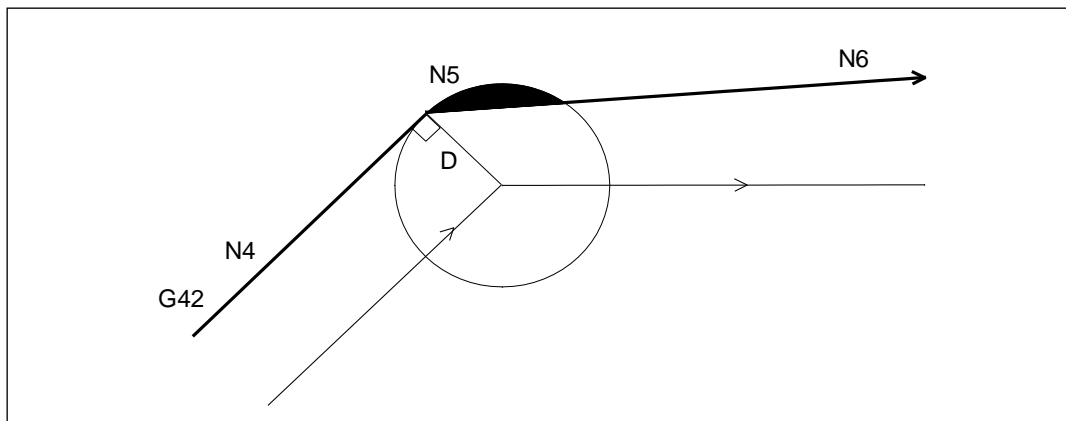
While in cutter compensation mode, if a command is issued for which the 2 axes on the selected planes do not travel for more than 3 blocks, the infeed will be too much or too little, as shown in the diagram below. Therefore, please avoid issuing those types of commands.

```
N4 X_Y_;
N5 Z_;
N6 F_;
N7 Z_;
N8 X_;
```



**(Note 1)** The same infeed problem arises as noted above for a block with zero travel.

```
N4 G91 X_Y_;
N5 X0;
N6 X_;
```



**(Note 2)** If there is no travel command for 2 axes on the selected planes during startup, the startup operation is performed when a travel command is executed thereafter even on a single axis for either the X- or Y-axis (when travel amount  $\neq 0$ ).

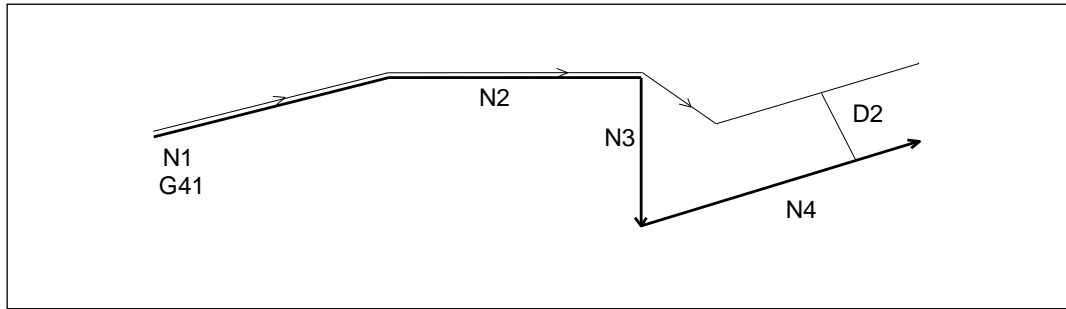
### 8. Tool movement when the offset for cutter compensation is 0

#### (1) Startup

The offset mode is enabled when the G41 and G42 commands are issued while in cancel mode, but the startup operation is not performed because offset = 0.

The operation thereafter is the same as described in the section 2. “Tool diameter offset change” when changed to an offset number where offset  $\neq 0$ .

```
N1 G41 X_Y_D1; (D1=0)
N2 X_;
N3 Y_D2; (D2≠0)
N4 X_Y_;
```



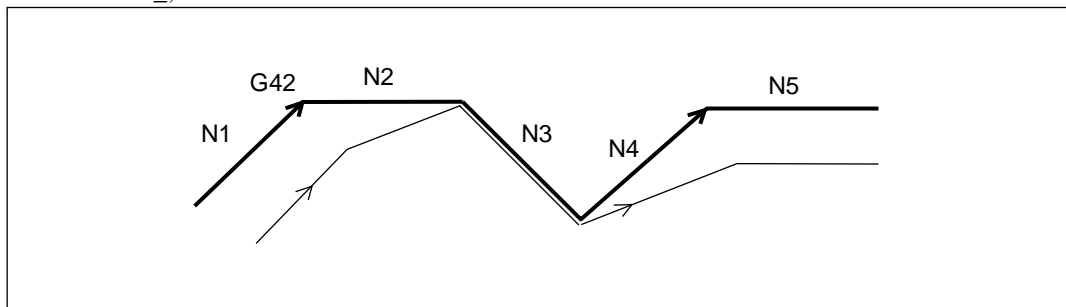
#### (2) Offset mode enabled

Cancel mode is not enabled even if changed to an offset number where the offset amount = 0 while in offset mode.

The operation is the same as described in section 2. “Tool diameter offset change”.

The operation thereafter is the same as described in the section “2. Tool diameter offset change” when changed to an offset number where offset  $\neq 0$ .

```
N1 X_Y_;
N2 X_D1; (D1=0)
N3 X_Y_;
N4 X_Y_D2; (D2≠0)
N5 X_;
```



### 9. Commands issued during cutter compensation that cause exception processing or that trigger alarms

#### (1) Command that sets a perpendicular vector

G10	:	Programmable data input
G52	:	Local coordinate system setting
G92	:	Coordinate system setting
G210	:	Programmable data input (high accuracy)
#3000	:	Alarm display
#3006	:	Message display & stoppage

If the command noted above is issued, the machine travels to a position that is offset by the cutter compensation using the value from the last X- and Y-axes travel command.

- (2) Command that forces the cutter compensation to cancel  
M06 : Tool change  
G100 : Nonstop ATC

If the command noted above is issued, G40 (cutter compensation cancel) is automatically triggered. Therefore, the machine travels to a position that is offset by the cutter compensation using value from the last X- and Y- axes travel command.

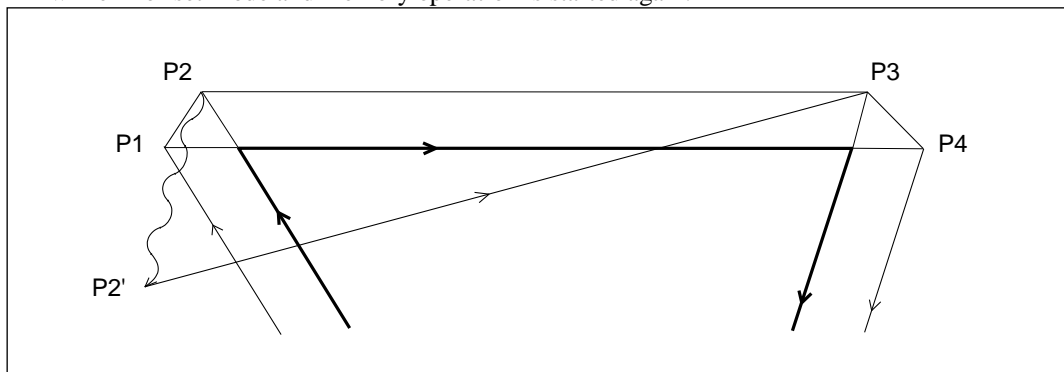
- (3) Command that triggers the alarm <<Compensating diameter>>
- |                    |  |
|--------------------|--|
| G17~G19            | : Plane selection                              |
| G28                | : Reference position return                    |
| G29                | : Return from reference position               |
| G30                | : No. 2 to 6 reference position return         |
| G33, G376, G392    | : Thread cutting                               |
| G36~G39            | : Coordinate calculation                       |
| G60                | : Single direction positioning                 |
| G66                | : Macro program modal call                     |
| G68.2              | : Feature coordinate setting                   |
| G73~G89, G173~G189 | : Canned cycle                                 |
| G120               | : Positioning to measurement position          |
| G121~G129          | : Automatic measurement                        |
| G31, G131, G132    | : Skip feed                                    |
| G133, G134         | : Change tap twist direction                   |
| M410, M411         | : Pallet index                                 |
| G2, G3             | : Arc with 0 start point or 0 end point radius |

10. Input command from MDI operation

The alarm <<Specified G code cannot be used>> is triggered when there is an input related to cutter compensation (G40, G41 or G42) in MDI operation mode.

11. Manual operation intervention

The correct offset path is enabled on block 2 when the tool is moved using manual operation while in offset mode and memory operation is started again.



- \* When operation stops at the end point (P2) of a block and the tool is then moved manually, the tool travels from P2' to P3, and the correct path is enabled from P3.

12. Command after cancelling cutter compensation

If a G17 to G19 (plane selection) command is issued when the G40 command is issued individually and there is a remaining offset amount, the alarm <<Cutter compensation error>> is triggered. When a travel command is issued for the same block as G40 or after the G40 command, issue a command after cancelling the offset amount.

## 4.1.11 Override Function Related to Cutter Compensation

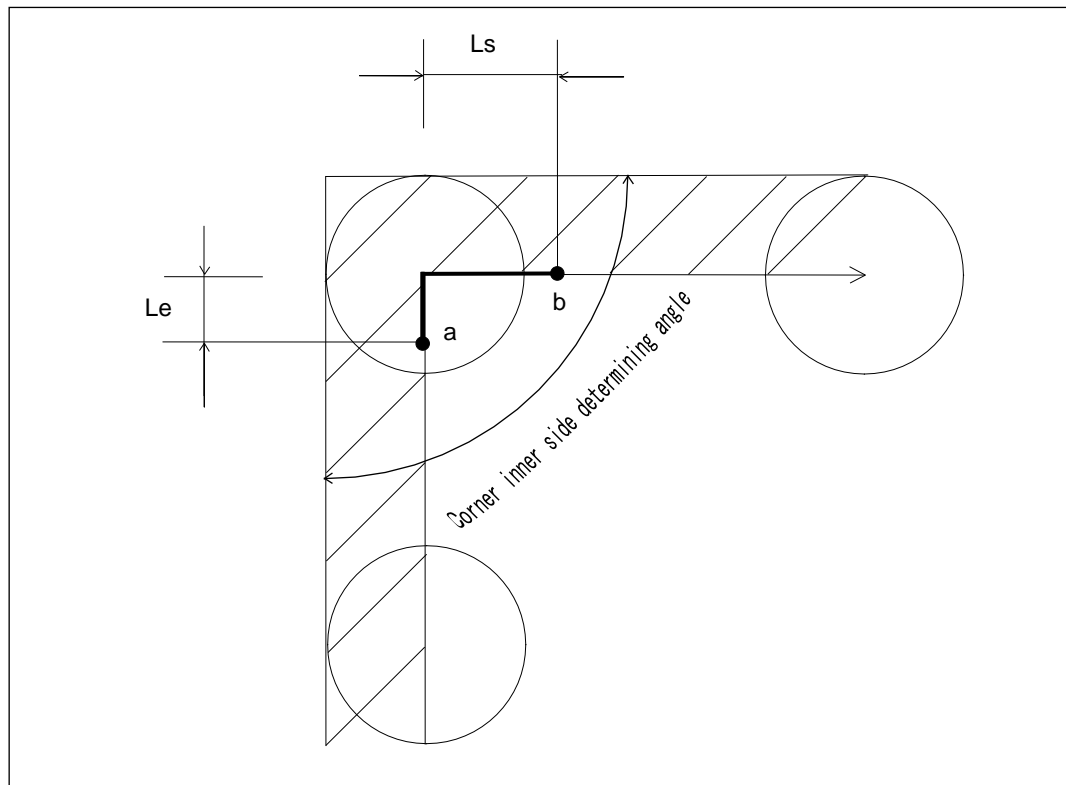
### 4.1.11.1 Automatic Corner Override

When both the block before and after the corner of the inner side meet the following conditions while in offset mode, the override function is automatically enabled in order to reduce the load on the tool.

1. G01, G02 or G03 travel operation. (Excluding spiral/conical interpolation)
2. Offset  $\neq 0$  when offset mode is enabled.
3. The corner's inner side angle is less than the user parameter <Automatic corner override (angle)>.
4. The block does not include the following commands: G41, G42 and G40.
5. The compensation direction does not change.

The following items are configured in the user parameter settings.

- (1) Automatic corner override (length 1) : Corner end point deceleration distance  $L_e$
- (2) Automatic corner override (length 2) : Corner start point deceleration distance  $L_s$
- (3) Automatic corner override (ratio) : Deceleration ratio (%)  $Y$
- (4) Automatic corner override (angle) : Corner inner side determining angle  $\theta$



The override applies to the section  from point a to point b.

$$\text{Actual feedrate} = \text{Command speed} \times (\text{Deceleration ratio}/100)$$

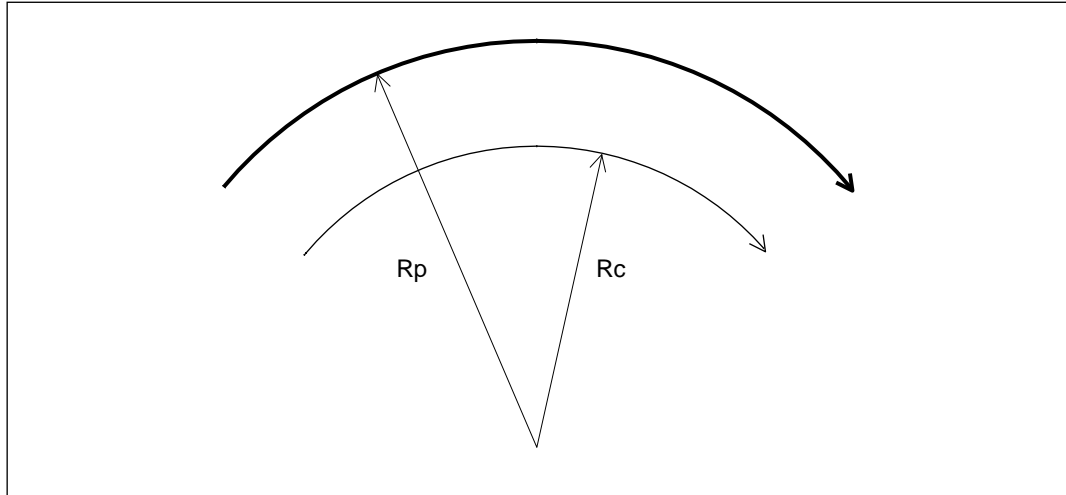
### 4.1.11.2 Inner Arc Override

When performing arc cutting that is offset on the inner side during offset mode, the actual feedrate is the product of  $R_c/R_p$  for the feedrate command that is issued.

Actual feedrate = Command speed  $\times R_c/R_p$

$R_p$ : Program radius

$R_c$ : Tool center path radius



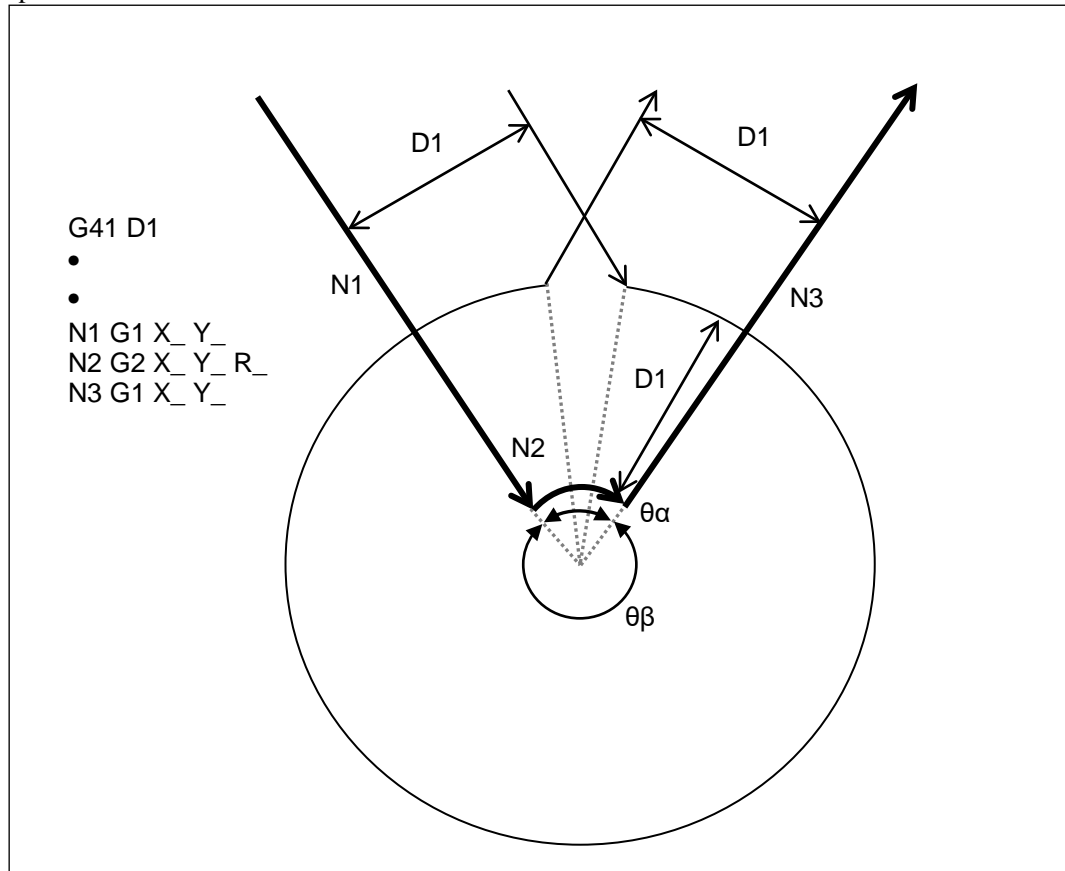
(Note 1) When  $R_c/R_p$  is less than the <Inner arc override limit> that is set in the user parameter, that parameter value is multiplied as an alternative to  $R_c/R_p$ .

Actual feedrate = Command speed  $\times$  (Inner arc override limit / 100)

(Note 2) Refer to “Chapter 12 (23) Involute interpolation function” in the Operation Manual for details about override for an involute interpolation command.

### 4.1.12 Arc Angle Check During Inner Side Cutting

During inner side cutting, if the arc angle in the program path for the arc command and the arc angle in tool center path after being offset are significantly different, this function stops the operation before the arc motion.



This function checks the arc angle ( $\theta\alpha$ ) at the start and end points in the N2 program path and the arc angle ( $\theta\beta$ ) at the start and end points in tool center path after being offset. If the angle is greater than  $180^\circ$ , then an alarm is triggered and operation stops before executing N2.

- (Note 1)** When an alarm is triggered, the infeed may already be too great (The infeed is too great for the workpiece on the N3 side after N1 is executed in the above example).
- (Note 2)** This function carries out the check at the end point after 3 blocks of travel. If one of the situations below applies while the offset mode is enabled, this check function may not work properly because the tool center path start and end points change after being offset.
- When there is a cutter compensation G code command or a command that sets a perpendicular vector
  - Zero travel commands for more than 3 blocks

### 4.1.13 Interference Check for Cutter Compensation / Nose R Compensation

When a tool cuts into the workpiece too much, that is called interference. The function that detects that interference beforehand and prevents it from cutting into too much is called the interference check.

This function cannot always detect the interference, and it also may detect interference when the tool has not actually cut in too much.

#### 4.1.13.1 Interference detection method

This detection method uses two types of check: a direction check and an arc angle check.

The user parameter (switch 1) <Cutter compensation interference check> is used to set which method is used for the detection.

##### 1. Direction check

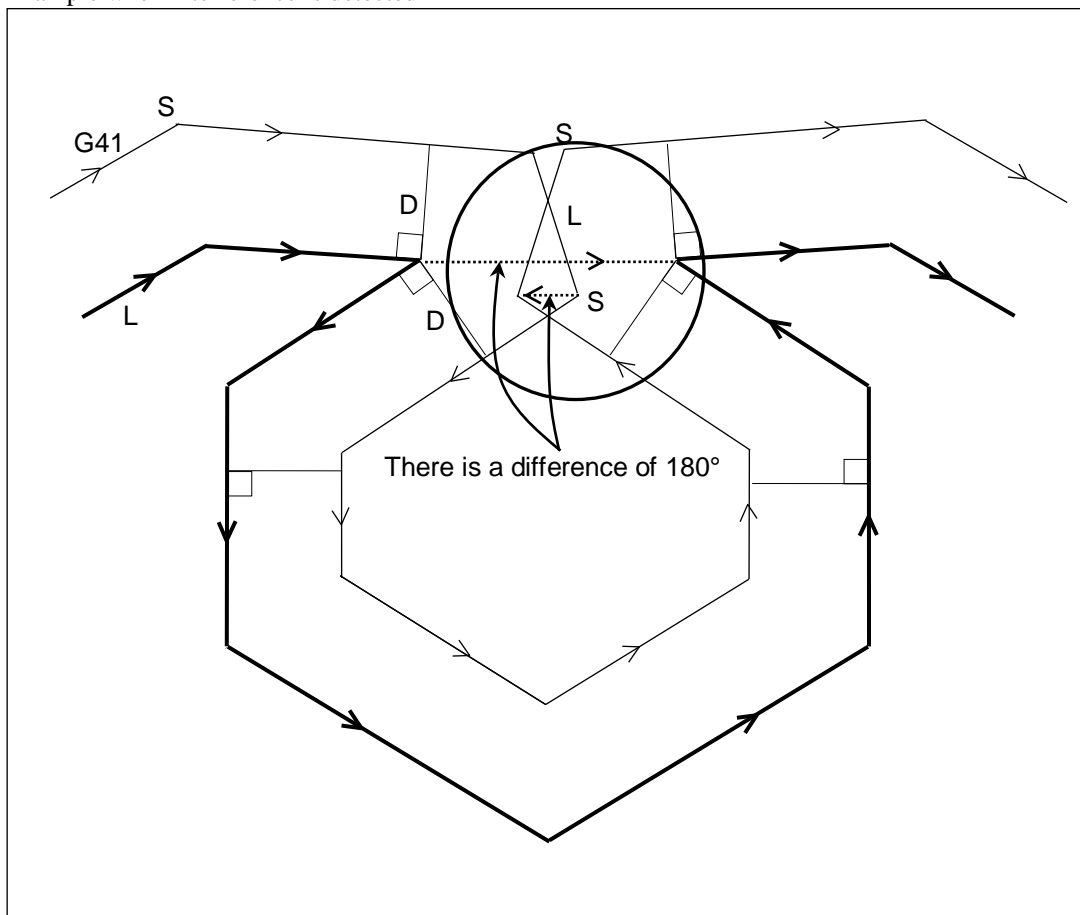
The cutter compensation function checks between the offset vectors that occur at the end point of the block for the program path.

The check range is set in the user parameter (switch 1) <Number of blocks to check cutter compensation interference>.

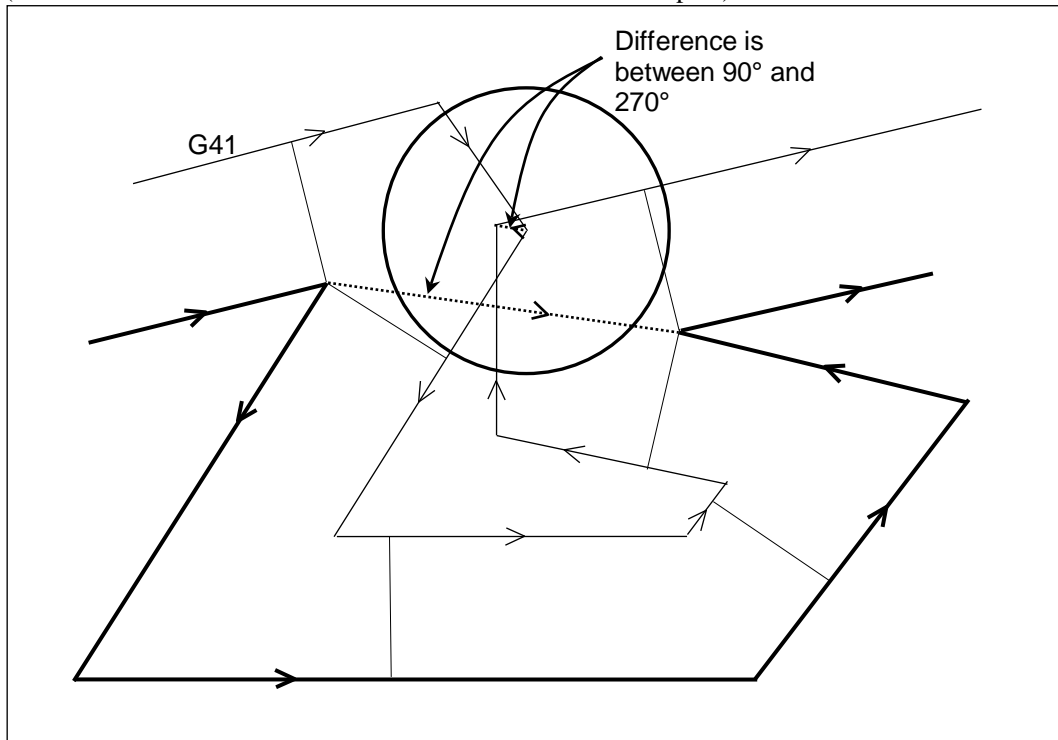
It assumes there is interference when the difference between the angle for the end points on the program path block and the angle for the offset vectors is greater than  $90^\circ$  and less than  $270^\circ$ .

When there is a block without travelling included, the interference check range becomes smaller.

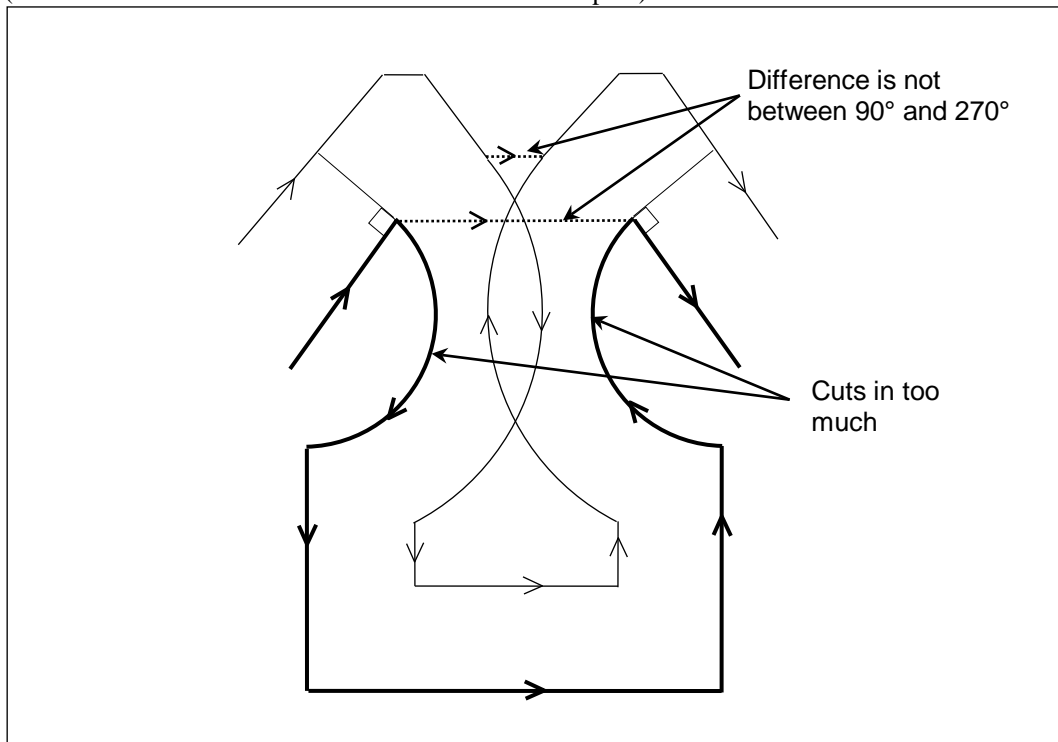
Example when interference is detected



Example when interference is detected but there is actually no interference  
(Tool does not cut in too much even when it follows the offset path)



Example when interference is not detected but there is actually interference  
(Interference is not detected in the middle of the offset path)

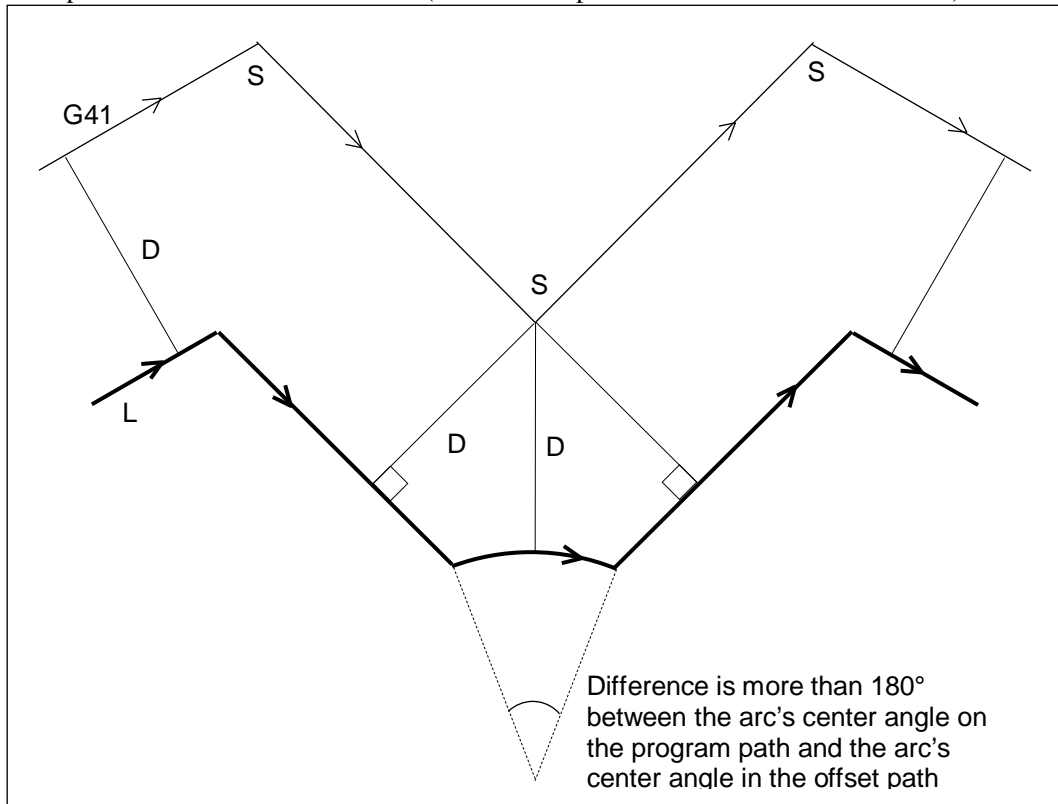




## 2. Arc angle check

When there is an arc, this check detects interference if the difference is more than  $180^\circ$  between the arc's center angle on the program path and the arc's center angle in the offset path.

Example when interference is detected (start and end points are the same in the arc block)



### 4.1.13.2 Operation when interference is detected

When interference is detected, the alarm <<Cutter compensation interference check error>> is triggered and machine operation is stopped.

### 4.1.13.3 Restrictions and special notes

- (1) When the user parameter (switch 1) <Number of blocks to check cutter compensation interference> is set to a value that is larger than the machine parameter (system 1) <Maximum blocks to check cutter compensation interference>, the alarm <<User param. setting error (switch 1)>> is triggered.
- (2) Interference is not detected when it falls outside of the range set in the user parameter (switch 1) <Number of blocks to check cutter compensation interference>.
- (3) The interference check range covers from where the cutter compensation offset mode is enabled until where that mode is cancelled. When the cutter compensation is temporarily cancelled, the interference check range ends. The interference check range starts again when the check is restored.
- (4) Detection is not possible when the tool cuts in too much in the middle of the travel path, because the interference check uses the offset vectors that form at the end point of the block.
- (5) When there are multiple travel commands that form from 1 block in the NC program, each command is counted as 1 block in the interference check range.
- (6) Cutting in too much can be prevented for all the commands that fall in the range set by the number of blocks in the user parameter (switch 1) <Number of blocks to check cutter compensation interference>.

## 4.2 Tool Length Offset (G43, G44 and G49)

### 4.2.1 Tool Length Offset Function

This function offsets the tool position so that the end of the tool moves into the position that is programmed. Even in an absolute command or an incremental command, the coordinates that are offset only for the offset amount specified in H code become the actual end point for the coordinates of the Z-axis travel command end point that is programmed.

1. Tool length offset (+)

Command format **G43 Hn;**

Hn : Tool number (n = 0 to 99), or group number (n = 901 to 930)

**(Note)** The amount that is offset for H0 is always 0.  
The offset can be set on the tool list setting screen.  
The tool length offset is performed for Z-axis.

2. Tool length offset (-)

Command format **G44 Hn;**

Hn : Tool number (n = 0 to 99), or group number (n = 901 to 930)

3. Tool length offset cancel

Command format **G49;**

**(Note 1)** When the tool length offset is cancelled, it is cancelled by the G49 command or by issuing 0 for the tool number.

**(Note 2)** The tool length offset is cancelled by the M06 (tool change) or by the G100 (nonstop ATC) command.

**(Note 3)** Refer to “4.2.3 Z-axis travel with tool length offset command” for travel when there is no Z-axis command for G43H\_, G44H\_, or tool length offset G49 and H0 command blocks.

**(Note 4)** When a Z-axis command is issued during the tool length offset for reference position return (G28) or No. 2 to 6 reference position return (G30), the tool length offset stays enabled while traveling to the middle point. And, the tool length offset is cancelled temporarily while travelling to the reference position.

Refer to “4.2.3 Z-axis travel with tool length offset command” for travel when the tool length offset operation that was cancelled resumes. When the tool length offset resumes, if the incremental mode is enabled, it is the equivalent of traveling from the absolute coordinates right before.

**(Note 5)** If the G53Z \_; or the G120Z\_: command is issued with the tool length offset enabled, the tool length offset is temporarily cancelled and the travel operation is carried out accordingly.

**(Note 6)** When the tool length offset range is set for the tool specified in H code, the range is checked. The alarm <<Comm. issued to area other than (tool) data area>> is triggered when the command area is outside of the range.

**(Note 7)** If a tool length offset command (G43 and G44) is issued during G143 and G144 modals, an alarm is triggered. Note, if a tool change command (G100 and M06) is issued on the same block, no alarm is triggered.

## 4.2.2 Tool Length Wear Offset

When a G43 or G44 command is issued in the program, the tool length wear offset is added to the tool length on the tool number for the command.

Tool length offset = Tool length offset + Tool length compensation

The tool length wear offset can be set on the tool list screen.

**(Note)** When the tool length wear offset range is set for the tool specified in H code, the range is checked. The alarm <<Comm. issued to area other than (tool) data area>> is triggered when the command area is outside of the range.

## 4.2.3 Z-axis Travel with Tool Length Offset Command

1. Z-axis travel with tool length offset command

When there is a Z-axis travel command for the command block G43 / G44 / G49, such as G43 H\_Z\_, G44 H\_Z\_ and G49Z\_, the Z-axis in that block travels to the canceled (G49) position that takes into account the compensation (G43/G44) specified in H code.

When there is no Z-axis travel command for G43 / G44 / G49 / H0 command block, such as G43H\_, G44H\_ or tool length offset G49 and H0, the operation follows the user parameter noted below.

User parameter <Travel of X, Y or Z axis when tool length/tool position offset is changed>	Type 1		Type 2
User parameter <Error check when traveling during tool length/tool position offset cancel>	Check	No check	-
G43/G44 command block without Z-axis command Ex.1) G43H1; Ex.2) G43; Ex.3) H1; (Tool length offset enabled)	Compensation specified in H code and Z-axis travel		Axis does not travel (Travels to offset position specified in H code for the next Z-axis travel command)
G49/H0 command block without Z-axis command (Tool length offset enabled) Ex.1) G49; Ex.2) H0;	When the current compensation is another value other than 0, the alarm <<Tool length offset cancel error>> is triggered (Note 2, 3 and 4).	Current tool length offset and Z-axis travel (Note 2)	Axis does not travel (Travels to position where the current tool length offset is cancelled for the next Z-axis travel command)

Z-axis motion example

(Workpiece coordinate zero → X: -200.000, Y: -200.000, Z: 50.000, H1 offset: 120.000)

User parameter <Travel of X, Y or Z axis when tool length/tool position offset is changed>	Type 1		Type 2
User parameter <Error check when traveling during tool length/tool position offset cancel>	Check	No check	-
	Machine coordinate	Machine coordinate	Machine coordinate
G90G0Z100.;	150.000	150.000	150.000
G43H1;	270.000	270.000	150.000
G0Z80.;	250.000	250.000	250.000
G49;	The alarm <<Tool length offset cancel error>> is triggered.	130.000	250.000

- (Note 1) When a circular interpolation command, an involute interpolation command or a thread cutting command is issued during travel, an alarm is triggered.
- (Note 2) When the tool length offset is temporarily canceled as per G28 or G53, cancel travel for the compensation does not occur even for a G49/H0 command block without a Z-axis command. As a result, the alarm <<Tool length offset cancel error>> is not triggered even when the parameter <Travel of X, Y or Z axis when tool length/tool position offset is changed> is set to <0: Type 1> and the parameter <<Error check when traveling during tool length/tool position offset cancel>> is set to <0: No check>.
- G90  
G43 H1 Z100.  
G91 G28 Z0  
G49 ← No Z-axis travel
- (Note 3) An alarm is not triggered during scaling, mirror imaging and rotational transformation because the axis travel command is on the same block as G49 and H0 and an axis other than the axis that was specified may operate.
- (Note 4) When the parameter <X-, Y- and Z-axes travel during current rotary fixture offset change> is set to <Type 2>, the alarm is not triggered if the Z-axis operates following a change in the workpiece coordinates.
- (Note 5) When the parameter <Travel of X, Y or Z axis when tool length/tool position offset is changed> is set to <1: Type 2> and the G43 or G44 command is issued on the same block as G53, an alarm is triggered.
- (Note 6) When <Travel of X, Y or Z axis when tool length/tool position offset is changed> is set to <0: Type 1> and the X-, Y- and Z-axes travel in another modal besides G0, the alarm <<Feedrate not specified>> is triggered if there is no F command during G93 (inverse time feed).

## 2. Tool length offset resumed

The tool length offset can be temporarily canceled due to one of the following Z-axis commands: reference position return (G28/G30), machine coordinate selection (G53), positioning to the measurement position (G120) or external indexing on the pallet (M410/M411). The compensation or offset is resumed during the next Z-axis travel command, H command or G43/G44 command.

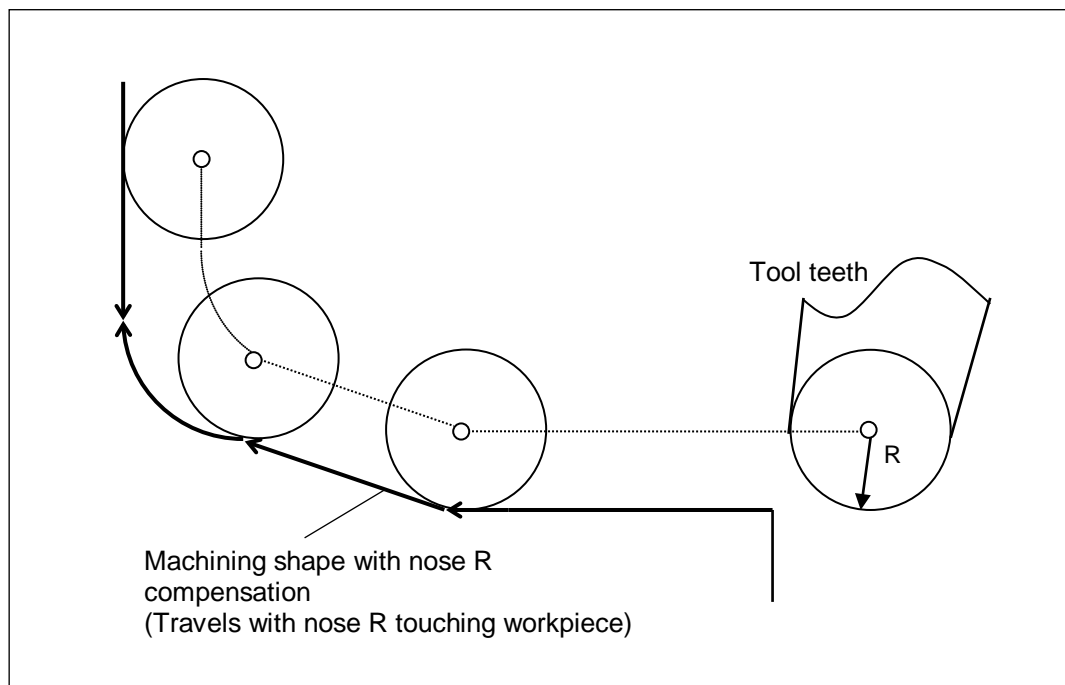
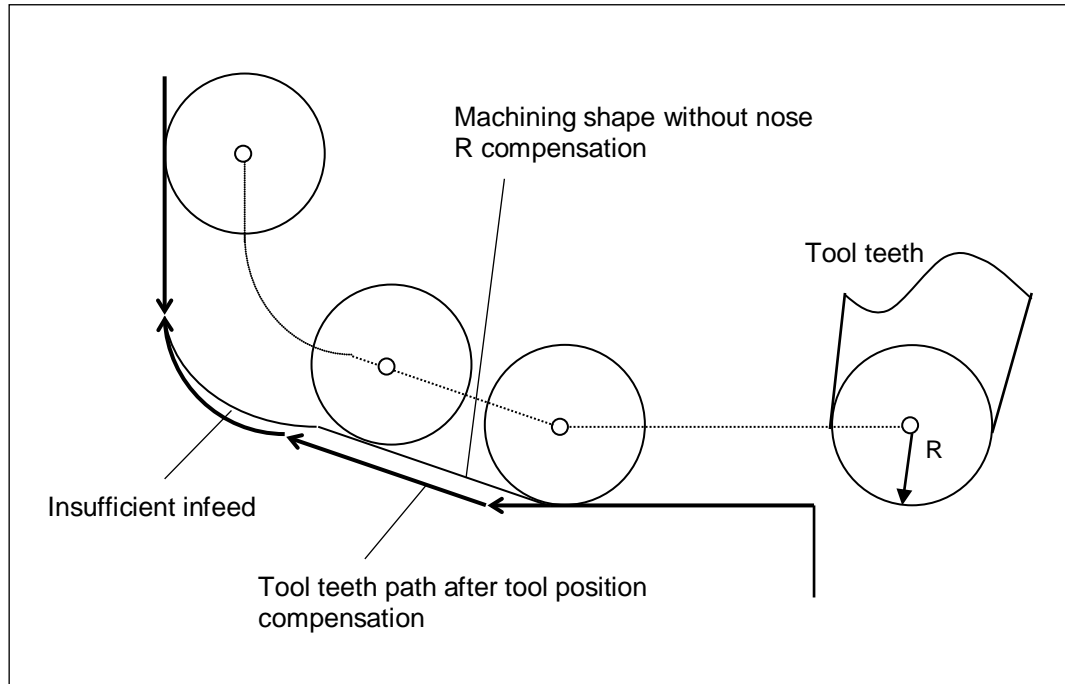
Ex:

G28 X-50. Y-50. Z400.;	←	Tool length offset temporarily canceled
...;	}	← During this time, operation is performed while tool length offset stays cancelled
G0 X-100. Y-100.;		
...;		
G0 Z300.;	←	Tool length offset resumed

### 4.3 Nose R Compensation (G141 and G142 - Option)

\* Available when equipped with a lathe function

The nose R compensation function is used on a lathe tool that has rounded teeth (nose R). It automatically compensates for the difference between the tool teeth that is offset by the tool position compensation operation and the actual nose R (cutting point).



### 4.3.1 Command Format

Command format

<b>G141</b> <b>G142</b>	<b>Dn;</b>
----------------------------	------------

G code and D code used for nose R compensation

- G40: Nose R compensation cancels (This mode is used when the power is turned ON.)
- G141: Left side compensation (Offsets to left side for direction of tool travel)
- G142: Right side compensation (Offsets to right side for direction of tool travel)

Dn : Tool number (n = 0 to 99), or group number (n = 901 to 930)

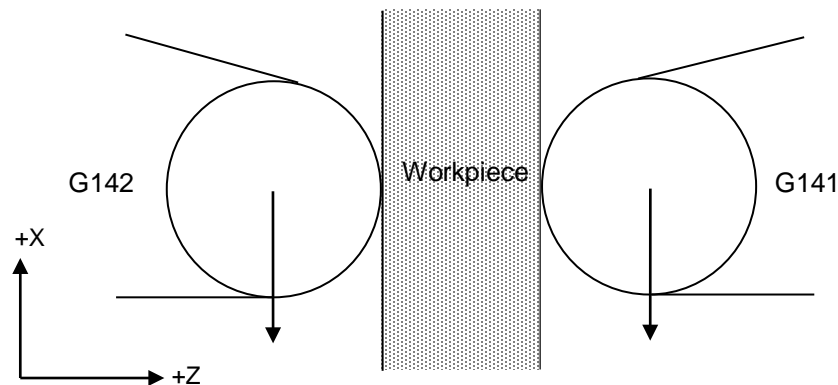
The nose R compensation for D0 is always 0.

The nose R compensation can be set on the tool list screen, or by inputting (G10) the tool data.

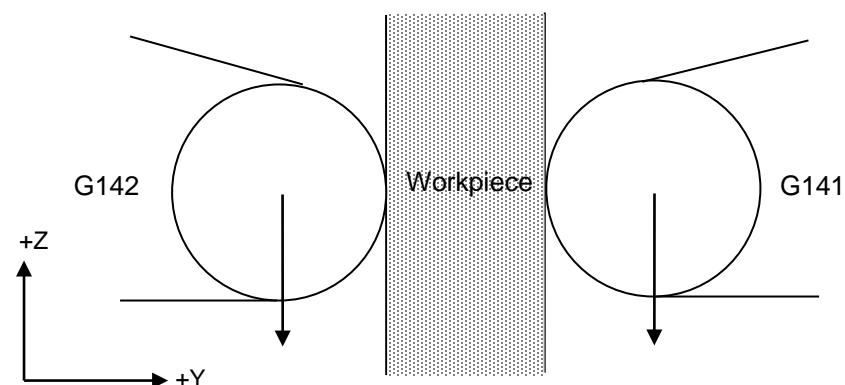
If either the command G141 or G142 is issued, the nose R compensation mode is enabled. This mode is cancelled by G40.

The nose R compensation target axis depends on the plane selection command (G17 to G19).

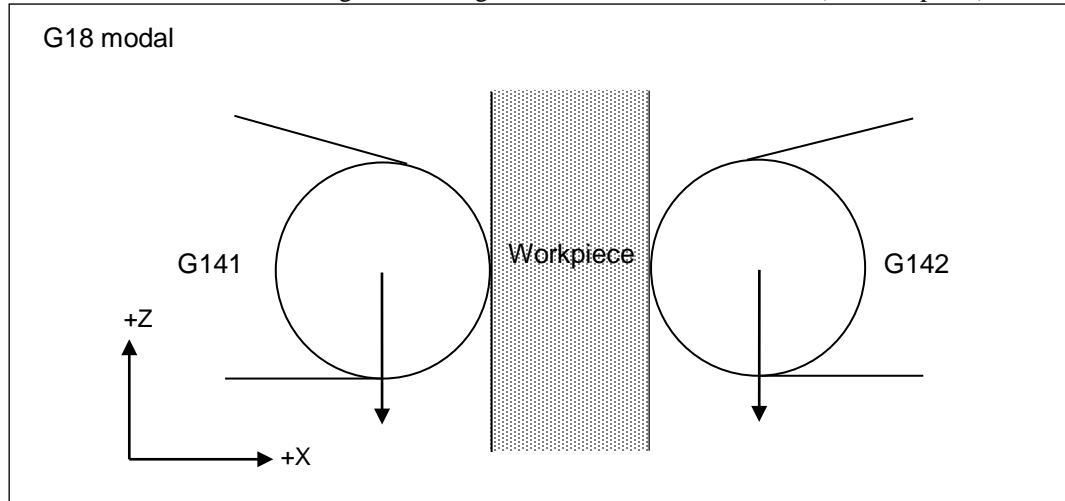
G18 modal



G19 modal



When the coordinates are configured looking from the front of the machine (X and Z plane)



- (Note 1)** During nose R compensation, when a command is issued with zero travel, or when there are no travel commands for more than 3 blocks, the infeed will be too much or too little.
- (Note 2)** When the cutter/nose R compensation range is set for the tool specified in D code, the range is checked. The alarm <<Comm. issued to area other than (tool) data area>> is triggered when the command area is outside of the range.
- (Note 3)** If a nose R compensation command (G141 and G142) is issued during G41 and G42 modals, an alarm is triggered. Note, if a tool change command (G100 and M06) is issued on the same block, no alarm is triggered.
- (Note 4)** When a nose R compensation command (G141 and G142) is issued while the feature coordinate is being set (after G68.2 command and before G53.1 command), the alarm <<Feature coordinate manufacturing mode engaged>> is triggered.  
A command is possible while the feature coordinate is being indexed (after G53.1 command).

#### 4.3.1.1 Nose R Wear Offset

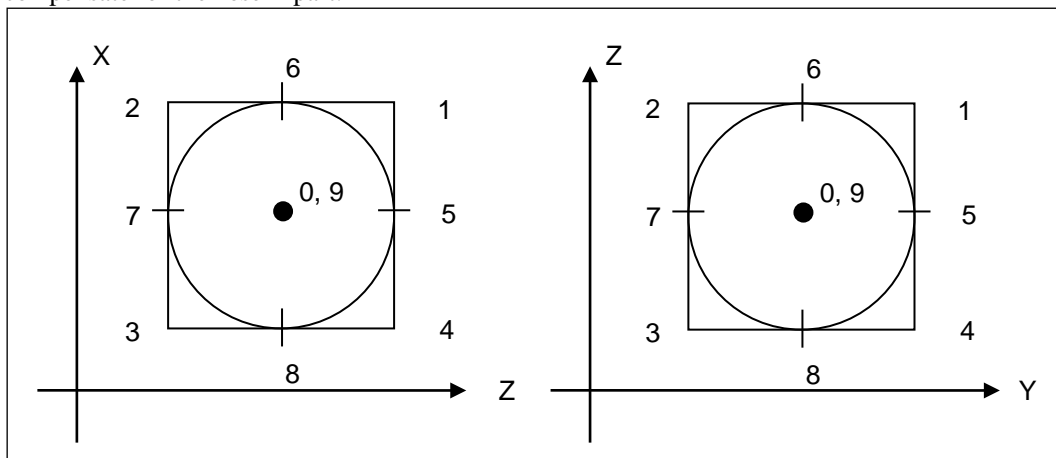
When a G141 or G142 command is issued in the program, the nose R wear offset is added to the nose R compensation on the tool number for the command. The nose R wear offset can be set on the tool list screen.

Nose R offset = Nose R compensation + Nose R wear offset

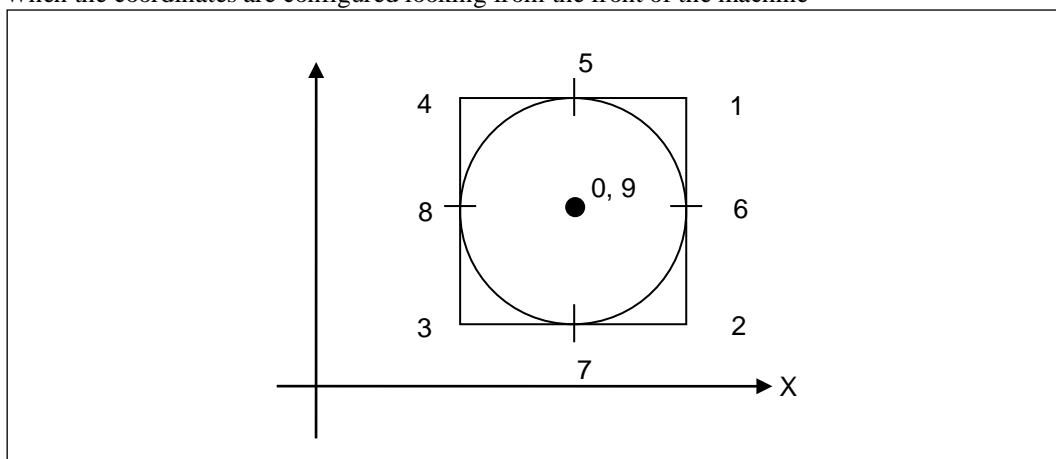
- (Note)** When the cutter/nose R wear range is set for the tool specified in D code, the range is checked. The alarm <<Comm. issued to area other than (tool) data area>> is triggered when the command area is outside of the range.

### 4.3.2 Virtual Teeth

The virtual teeth are the tool teeth after the tool position compensation operation. Set the virtual teeth direction (0 to 9) on the tool list screen when looking at it from the nose R center, in order to compensate for the nose R part.



When the coordinates are configured looking from the front of the machine



### 4.3.3 Cancel Mode

This mode refers to when the nose R compensation is disabled such as when the power is turned ON or when the [RST] key is pressed.

Normally, the tool path that is programmed matches the path of the virtual teeth.

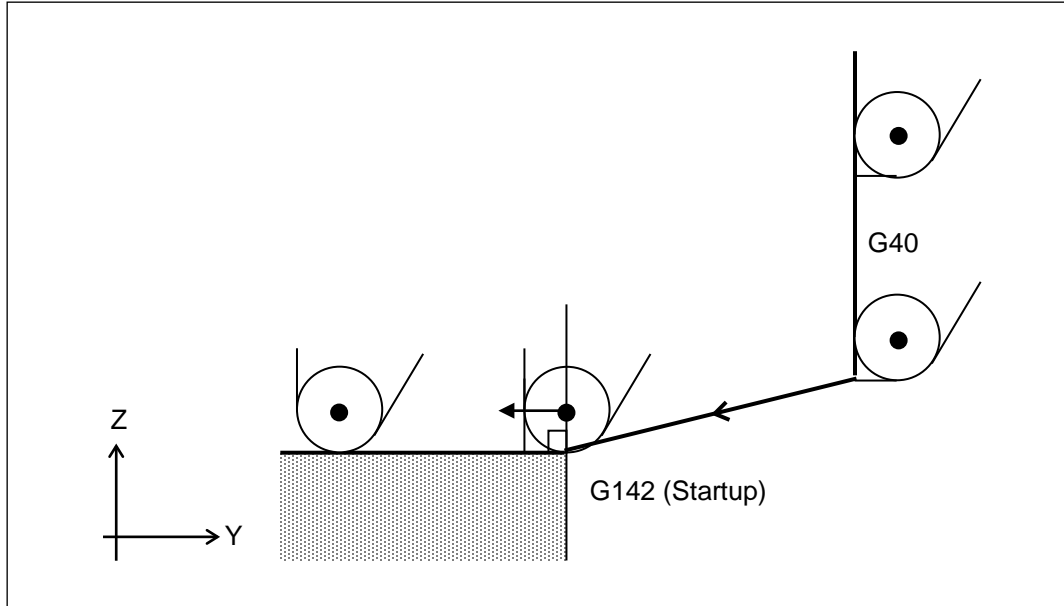


### 4.3.4 Startup

Offset mode is enabled for the control when a command that meets all the conditions below is executed for cancel mode. Startup refers to the travel operation in this situation.

- a) G141 or G142 command is issued.
- b) G0 or G1 travel command is issued, and the travel amount  $\neq 0$ .

In the startup operation, the travel operation occurs so that the nose R center comes to a position that is perpendicular to the start point of the next block.



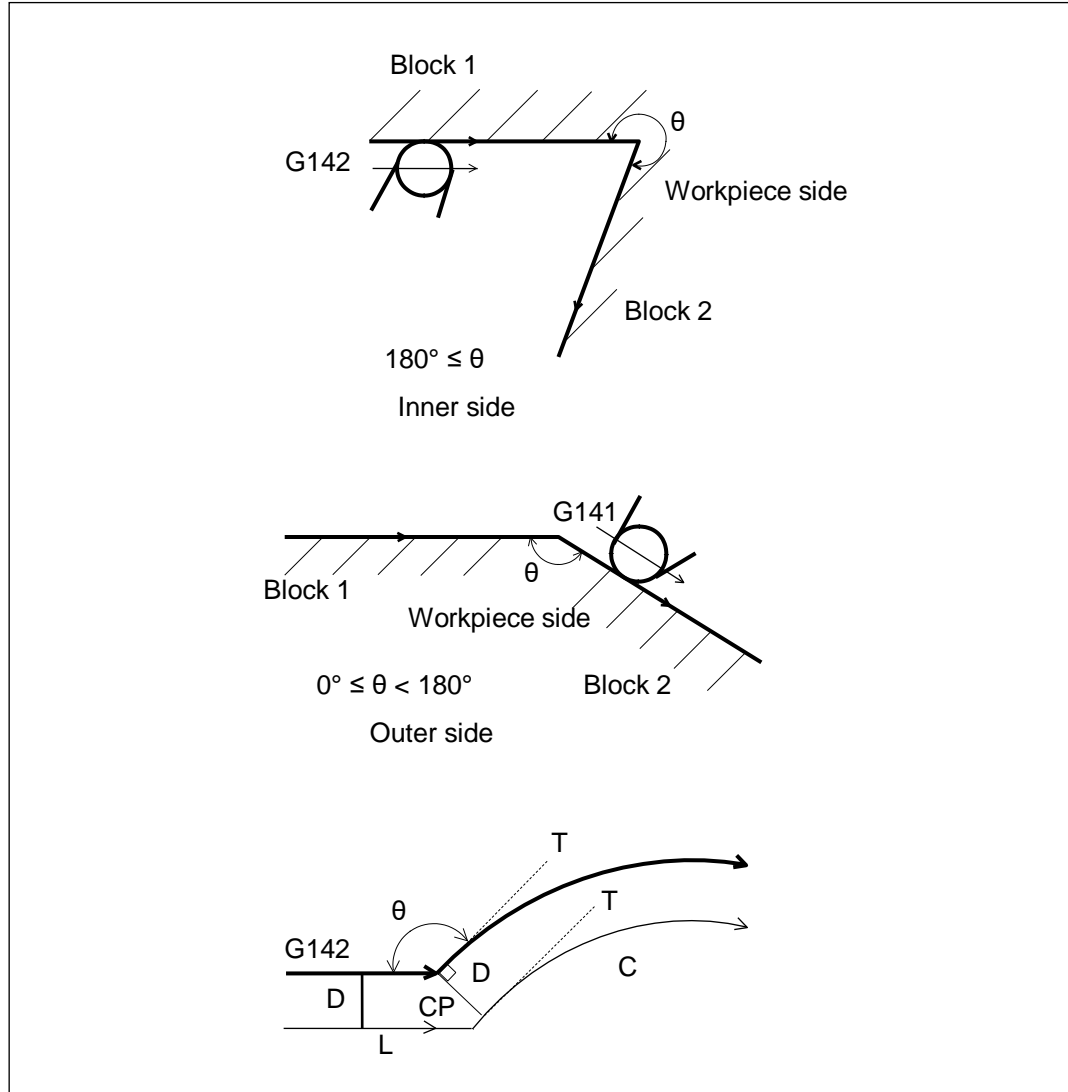
- (Note 1) The alarm <<Cutter compensation error>> is triggered when an arc command is issued.
- (Note 2) There are two setting types for the startup and cancel operations: <Type 1 (shortcut)> and <Type 2 (detour)>. Use the user parameter <Startup/cancel> to set one of the types.

The following explanation applies to the Y- and Z-planes. When considering the X- and Z-planes if looking from the front of the machine, the compensation direction is reversed (G141 → G142, G142 → G141).

<Term / symbol explanation>: The terms and symbols that will appear hereafter in the explanations are described below.

### 1. Description of inner side and outer side

The terms outer side and inner side refer to the intersecting angle for the travel command. Inner side means that the angle measures more than  $180^\circ$  on the workpiece side. Outer side means the angle measures between  $0$  and  $180^\circ$ .

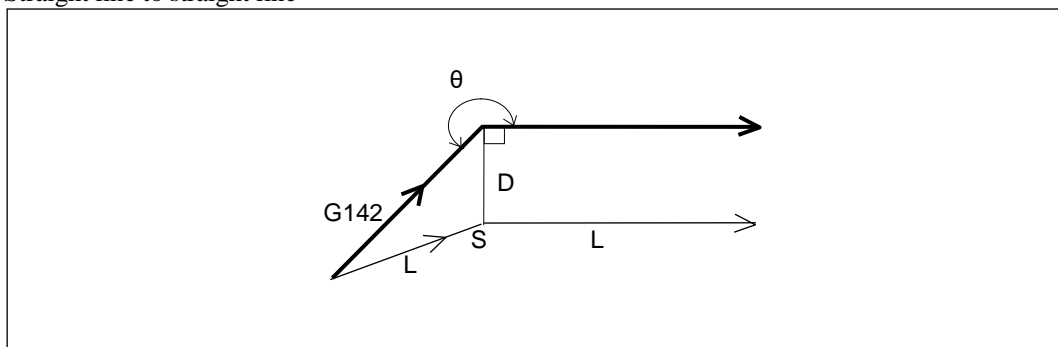


### 2. Explanation of symbols in diagram

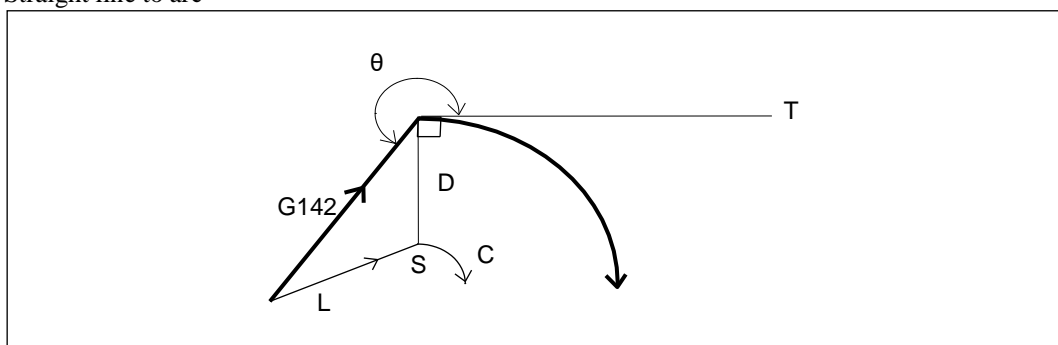
<b>————</b>	:	Program path
<b>————</b>	:	Nose R center path
<b>- - - - -</b>	:	Auxiliary line
<b>L</b>	:	Straight line
<b>C</b>	:	Arc
<b>D</b>	:	Nose R radius
<b><math>\theta</math></b>	:	Angle on workpiece side
<b>T</b>	:	Arc tangent line
<b>CP</b>	:	Intersection
<b>S</b>	:	Single block stop point

### 4.3.4.1 Inner Side Cutting ( $180^\circ \leq \theta$ )

Straight line to straight line

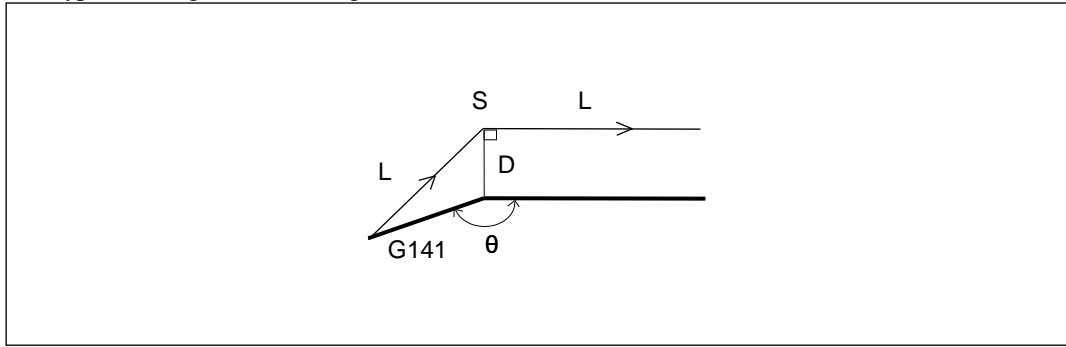


Straight line to arc

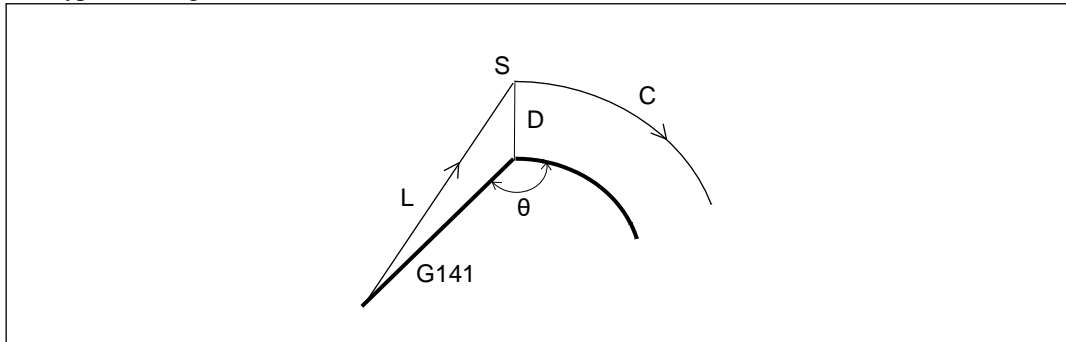


### 4.3.4.2 Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )

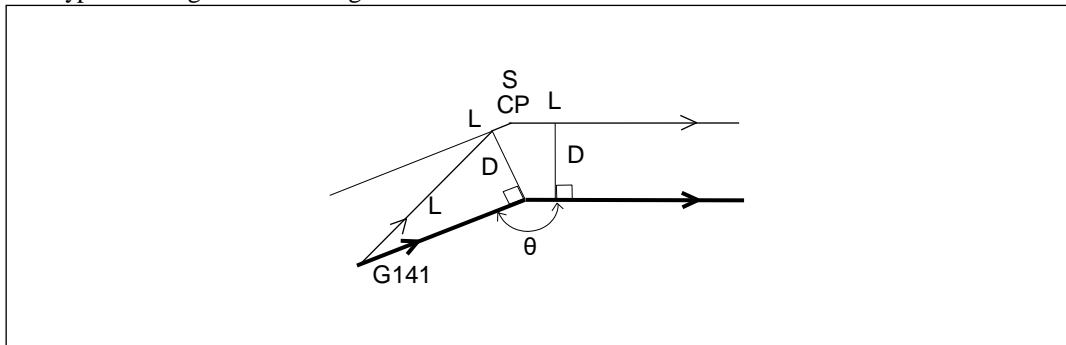
1. Type 1: Straight line to straight line



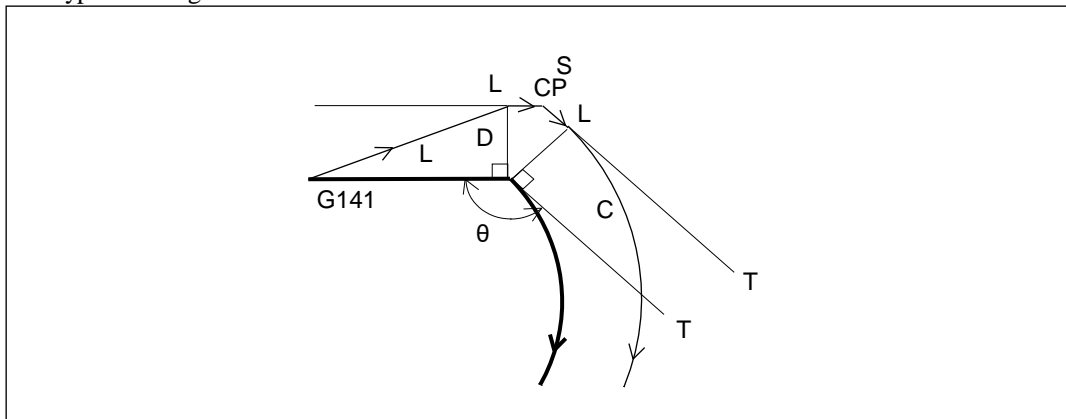
Type 1: Straight line to arc



2. Type 2: Straight line to straight line

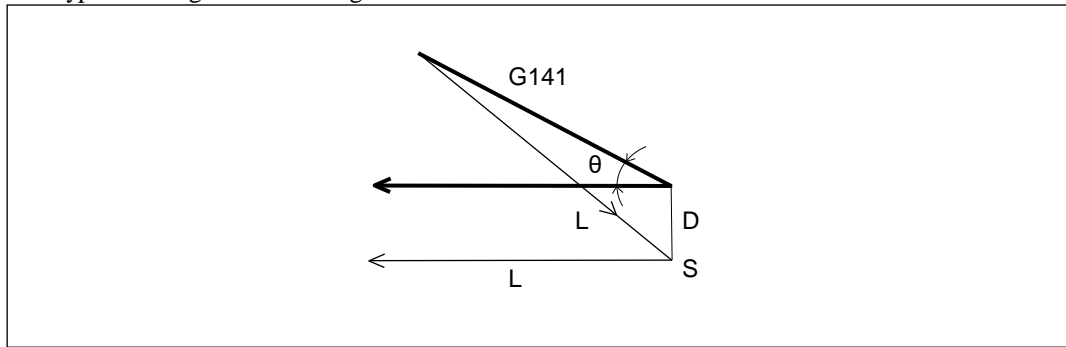


Type 2: Straight line to arc

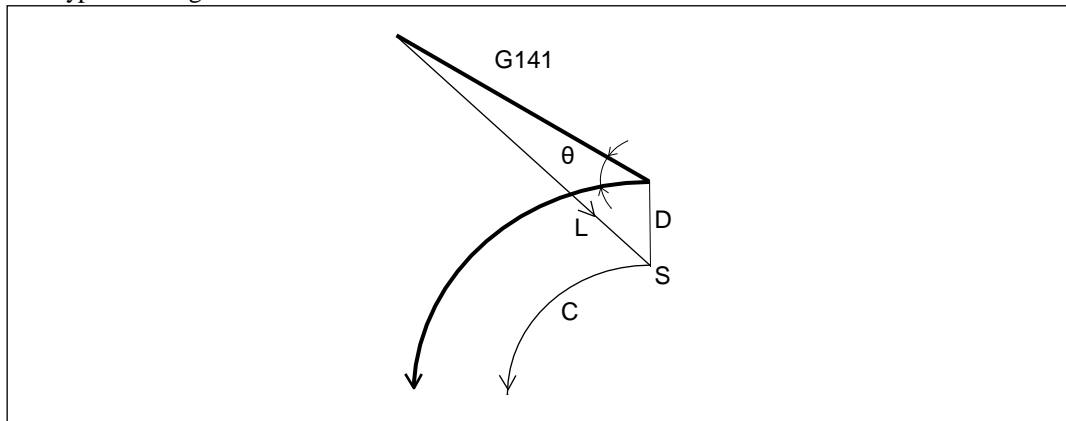


### 4.3.4.3 Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )

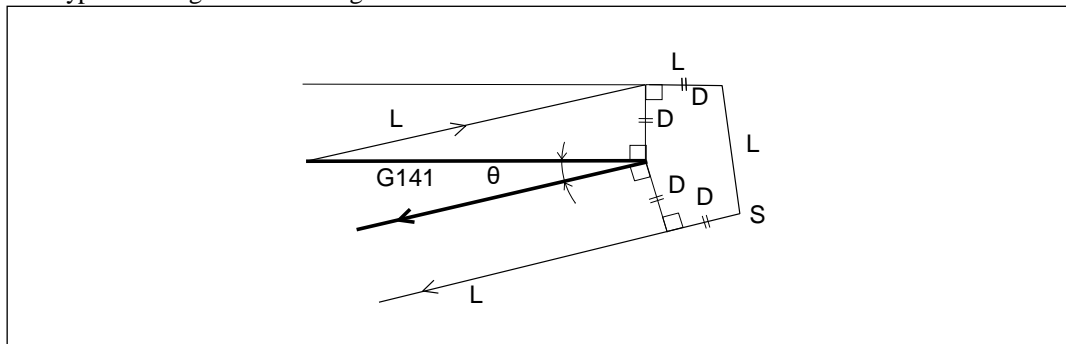
1. Type 1: Straight line to straight line



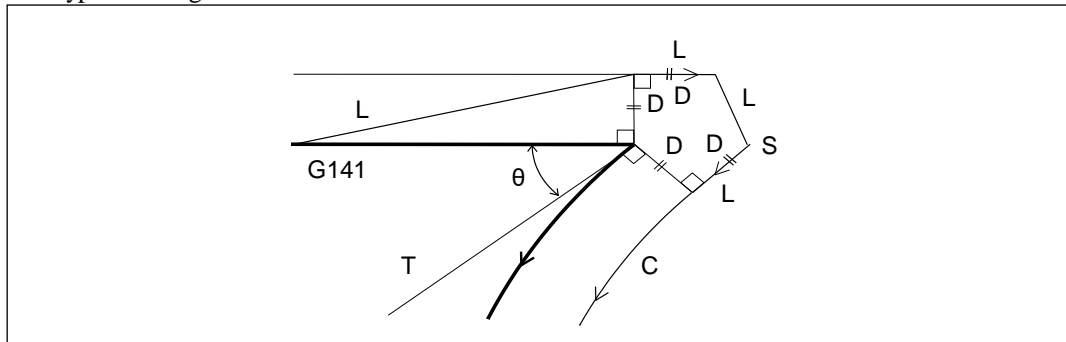
Type 1: Straight line to arc



2. Type 2: Straight line to straight line



Type 2: Straight line to arc



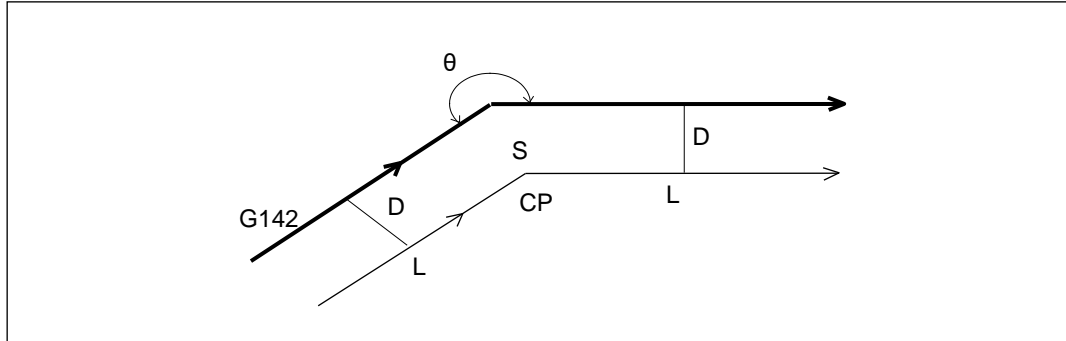
(Note) When  $\theta \leq 1^\circ$ , the setting <0: Type 1 (shortcut)> is used or enabled, even if <1: Type 2 (detour)> is specified for the user parameter <Start up/cancel>.

### 4.3.5 Offset Mode

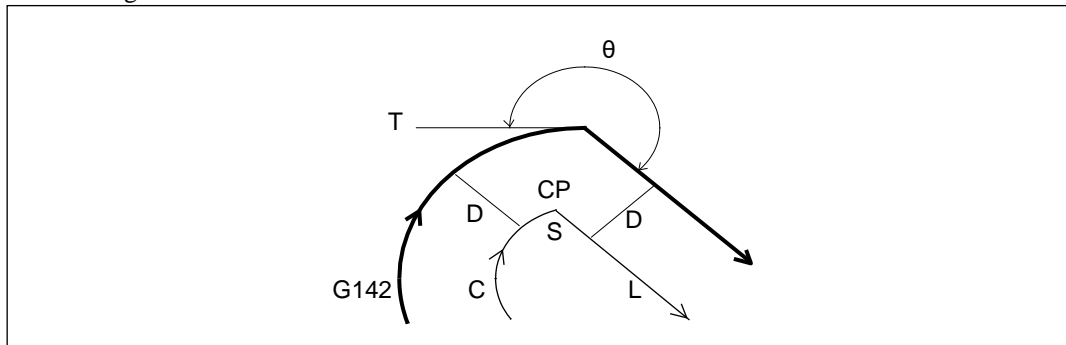
The travel commands in offset mode include: positioning, linear interpolation and circular interpolation.

#### 4.3.5.1 Inner Side Cutting ( $180^\circ \leq \theta$ )

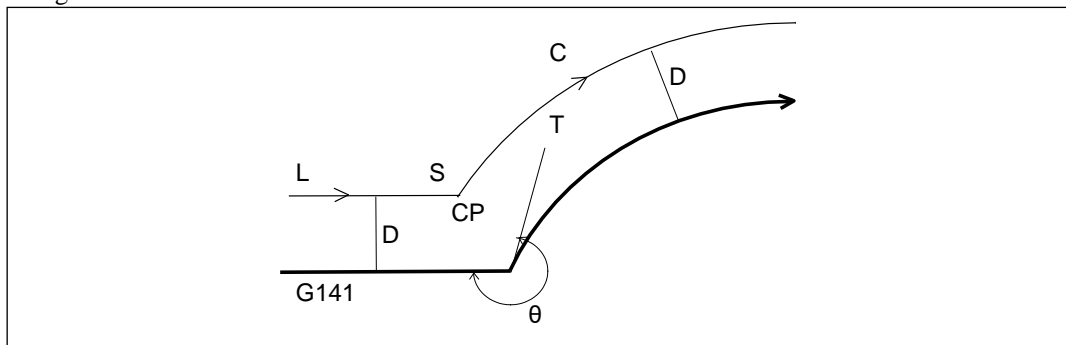
Straight line to straight line



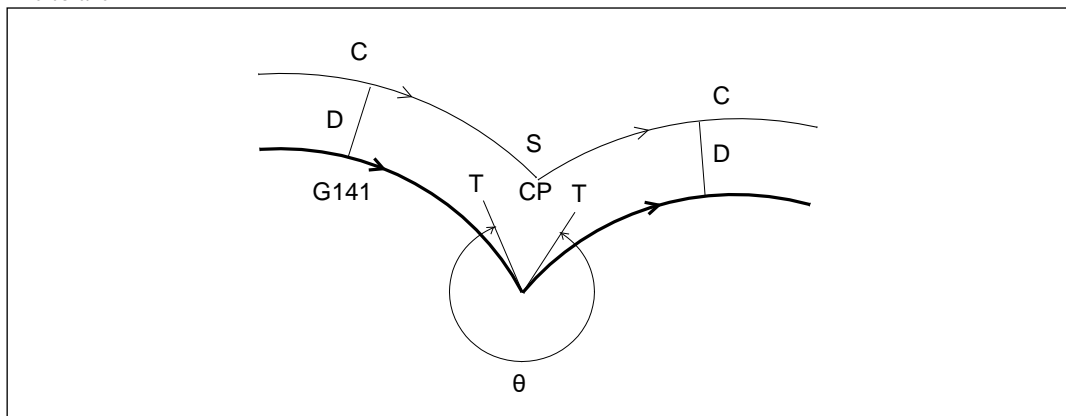
Arc to straight line



Straight line to arc

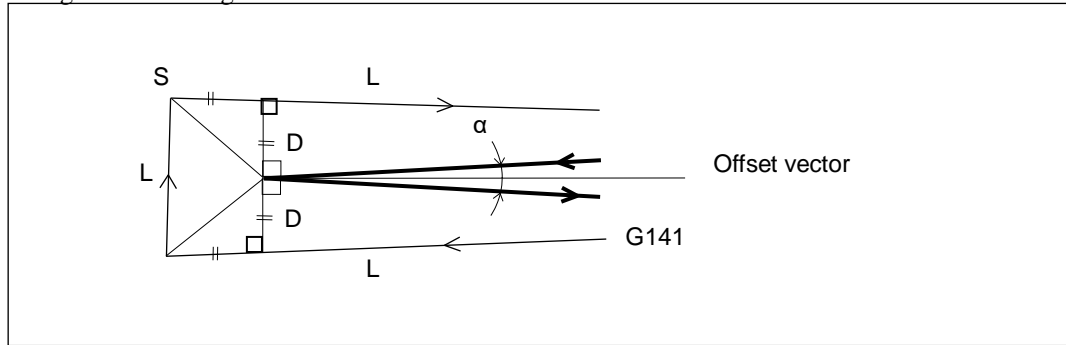


Arc to arc

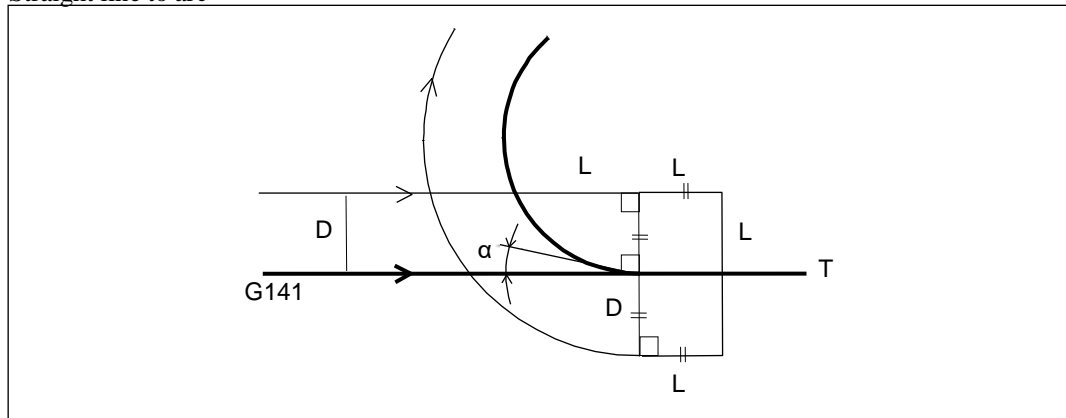


**(Note 1) When turning on the inner side of a narrow angle ( $\alpha < 1^\circ$ ), and the offset vector is abnormally large.**

Straight line to straight line

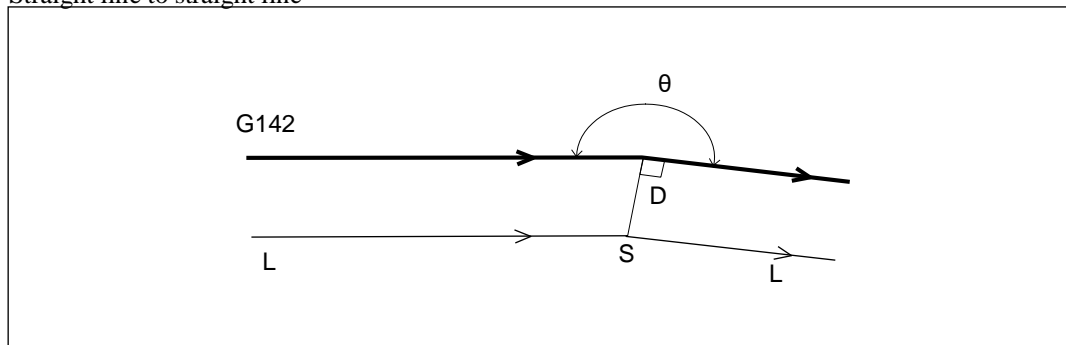


Straight line to arc



**(Note 2) When turning on the inner side of an angle that is almost parallel ( $180^\circ \leq \theta < 181^\circ$ ).**

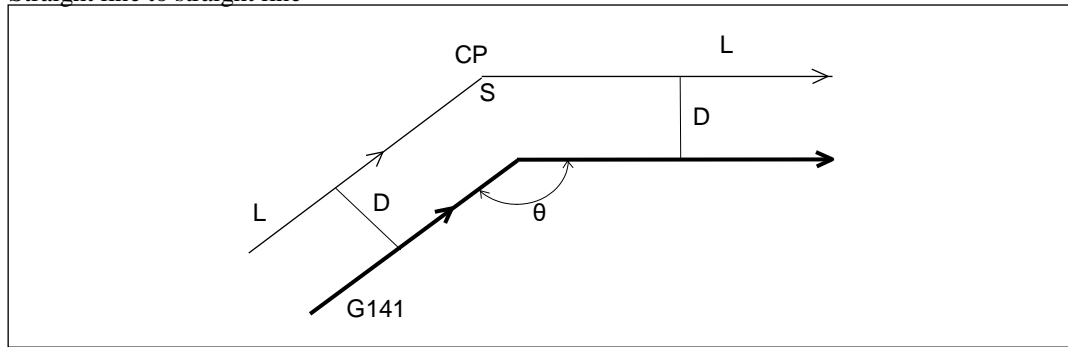
Straight line to straight line



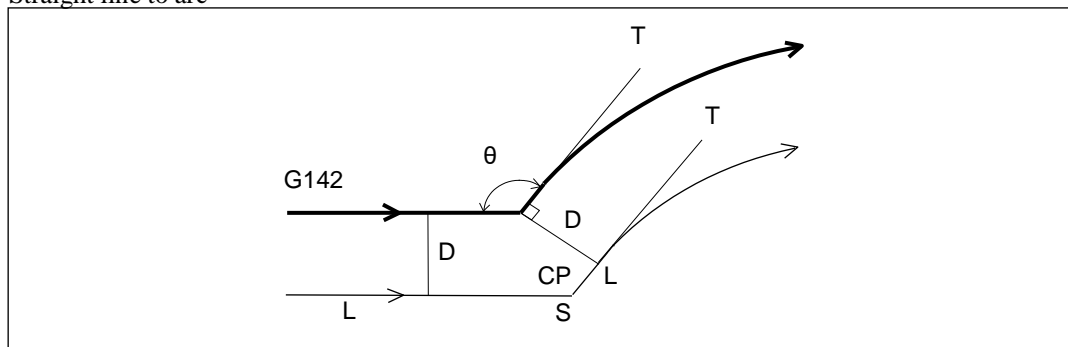
The processing is the same for: arc → straight line, straight line → arc and arc → arc.

### 4.3.5.2 Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )

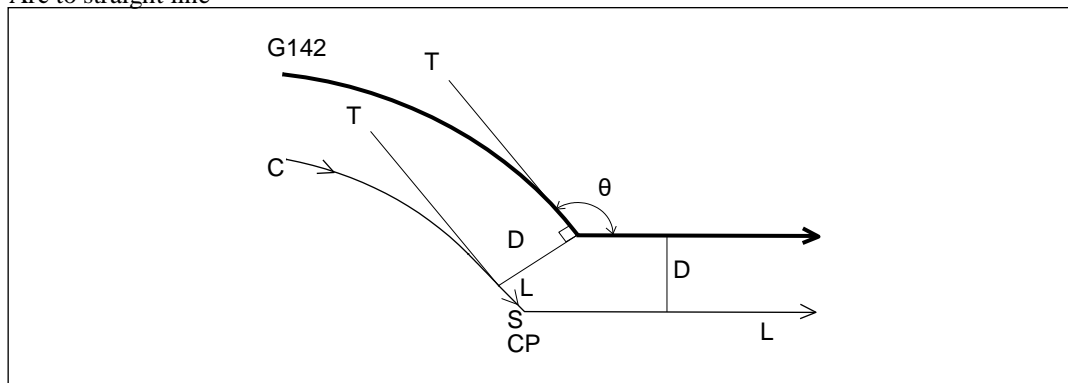
Straight line to straight line



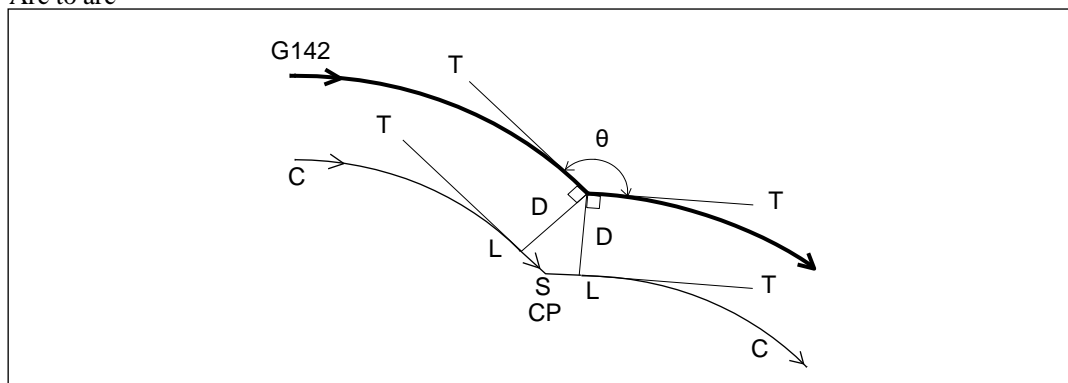
Straight line to arc



Arc to straight line



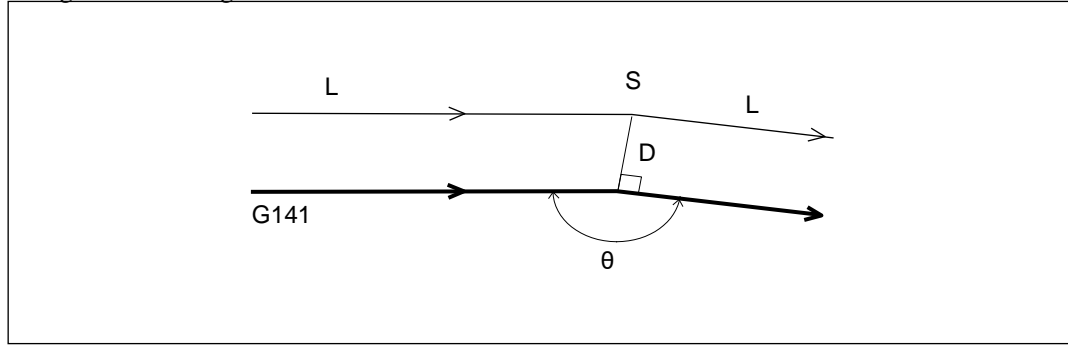
Arc to arc





(Note) When turning on the outer side of an angle that is almost parallel ( $179^\circ \leq \theta < 180^\circ$ ).

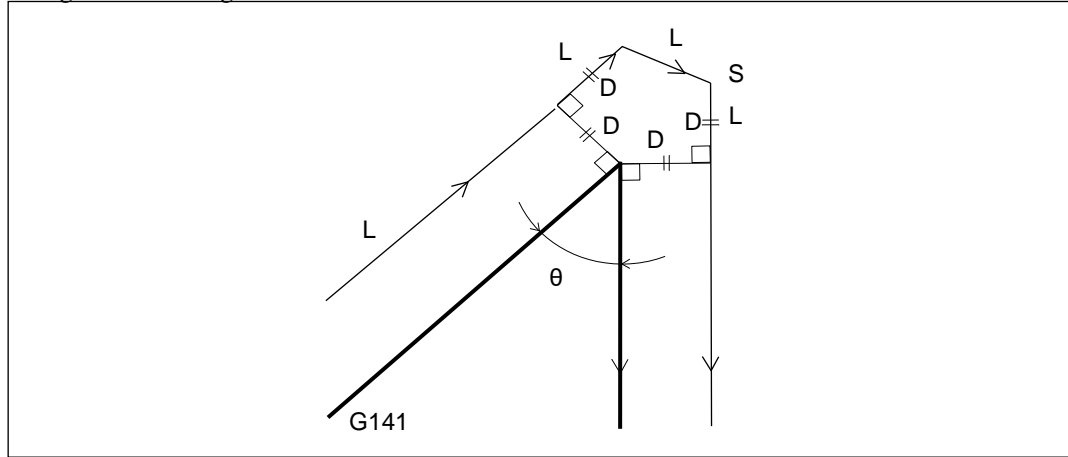
Straight line to straight line



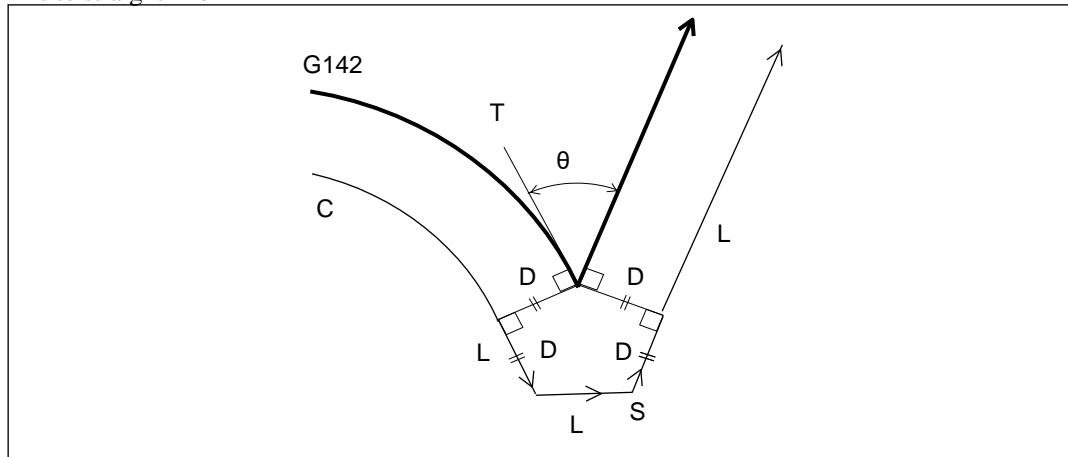
The processing is the same for: arc  $\rightarrow$  straight line, straight line  $\rightarrow$  arc and arc  $\rightarrow$  arc.

#### 4.3.5.3 Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )

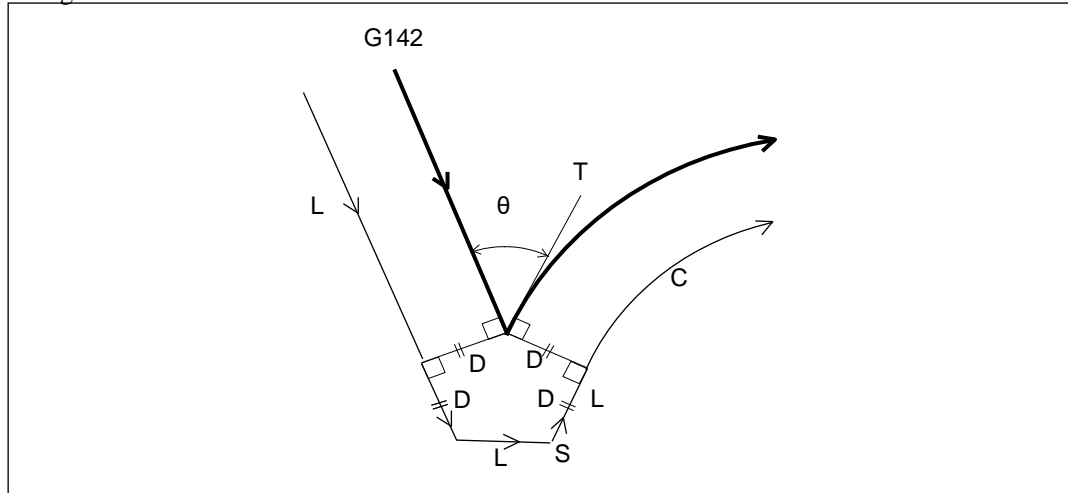
Straight line to straight line



Arc to straight line

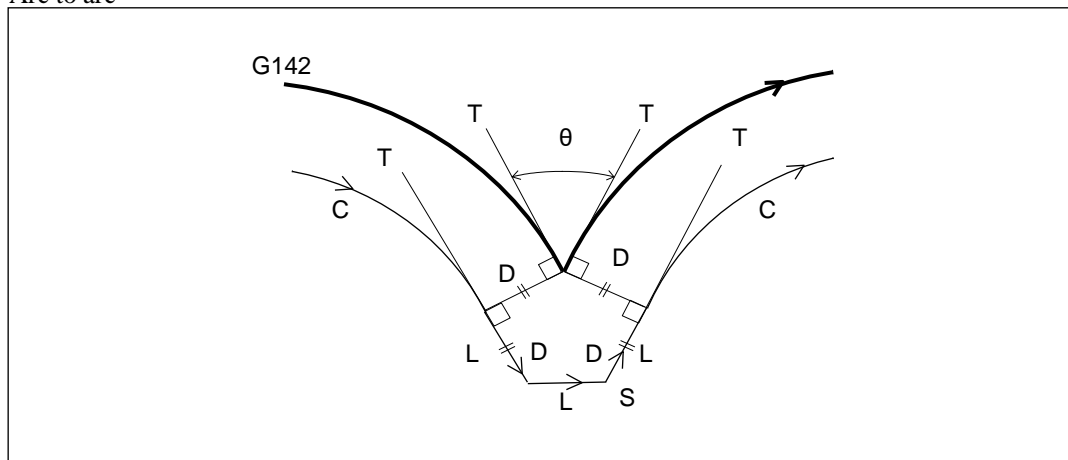


Straight line to arc



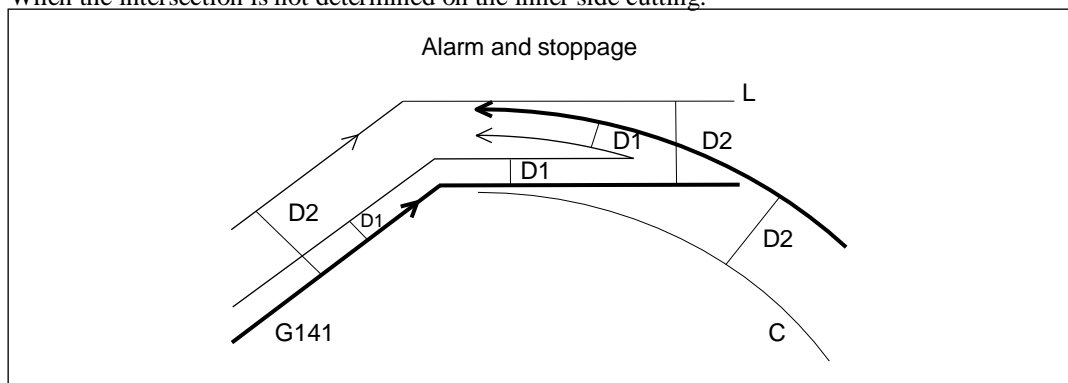
4

Arc to arc



### 4.3.5.4 Exceptional Cases

When the intersection is not determined on the inner side cutting.



When the compensation is a small amount as shown in the diagram above, even if an intersection exists, increasing the compensation may cause the intersection to disappear. In this situation, the alarm <<Cutter compensation too large>> is triggered at the previous block and operation stops.

### 4.3.6 Offset Cancel

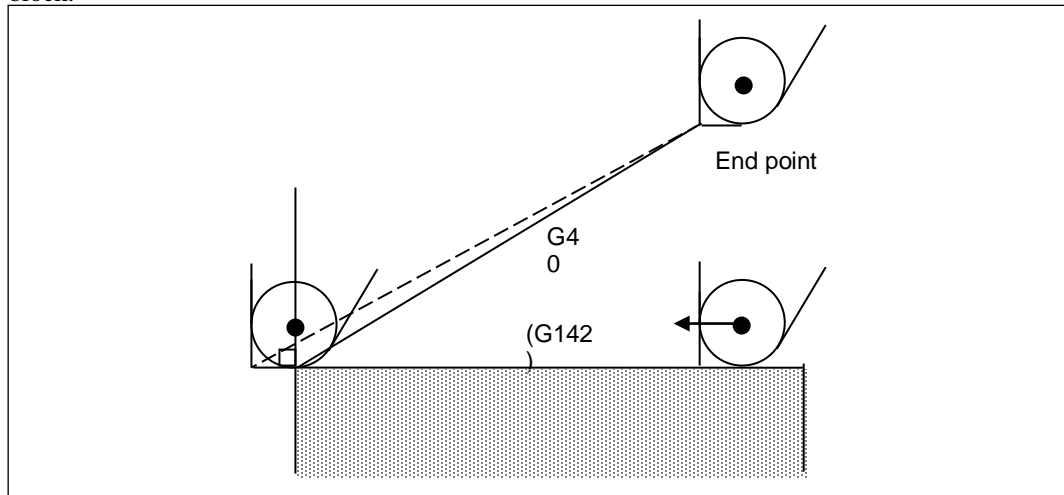
Cancel mode is enabled for the control when a command that meets all the conditions below is executed for offset mode. Offset cancel refers to the travel operation in this situation.

- a) G40 command is issued.

Command format **G40;**

- b) Travel command, excluding arcs and thread cutting, is issued.

In the offset cancel operation, the travel operation occurs so that the nose R center comes to a position that is perpendicular to the end point of the block that comes before the offset cancel block.

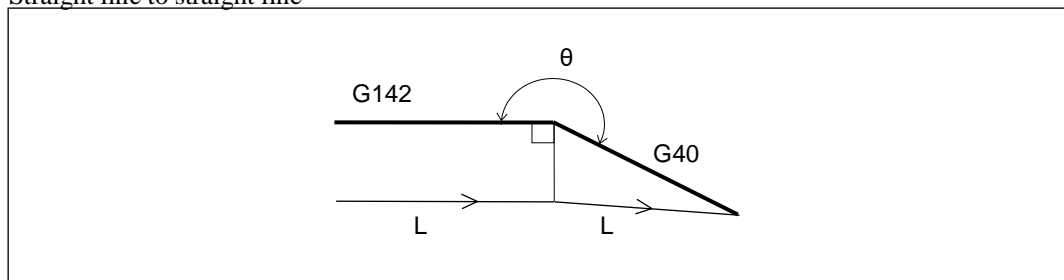


(Note 1) When an arc and thread cutting command are issued, an error is triggered.

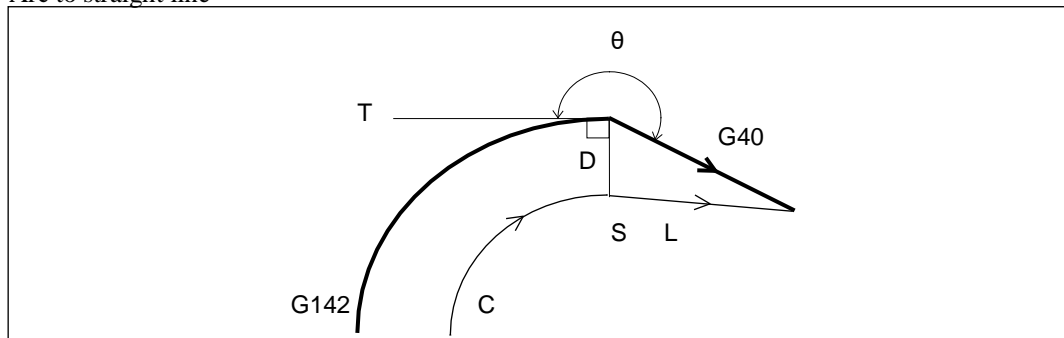
(Note 2) There are two setting types for the startup and cancel operations: <Type 1 (shortcut)> and <Type 2 (detour)>. Use the user parameter <Start up/cancel> to set one of the types.

#### 4.3.6.1 Inner Side Cutting ( $180^\circ \leq \theta$ )

Straight line to straight line

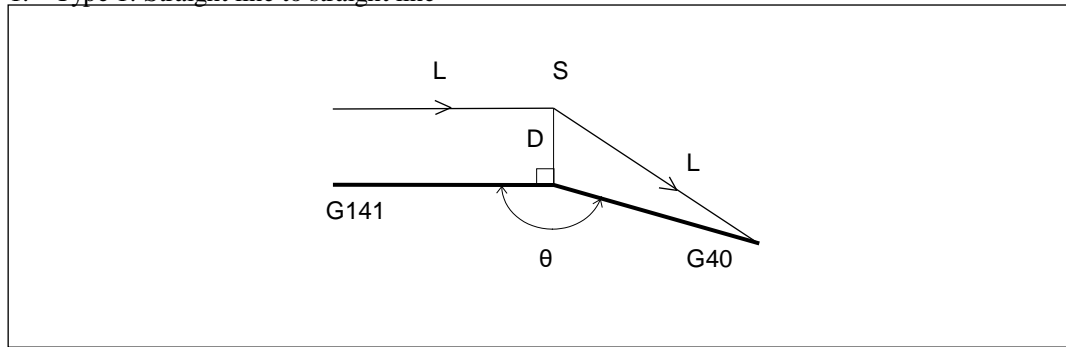


Arc to straight line

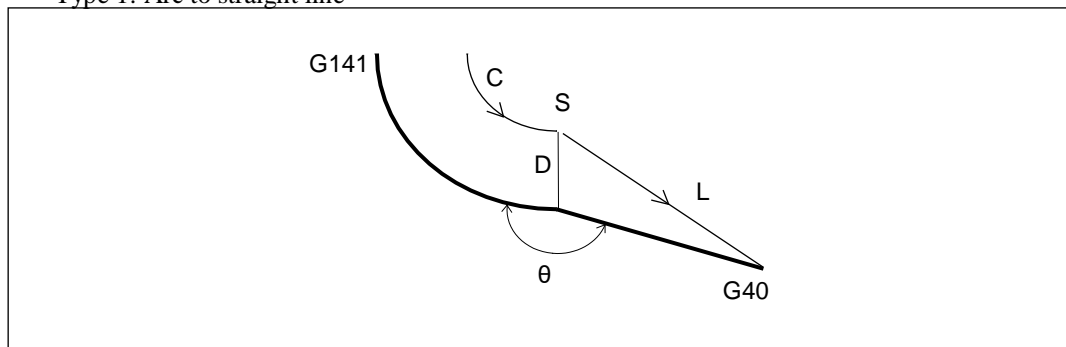


### 4.3.6.2 Outer Side (Obtuse Angle Cutting) ( $90^\circ \leq \theta < 180^\circ$ )

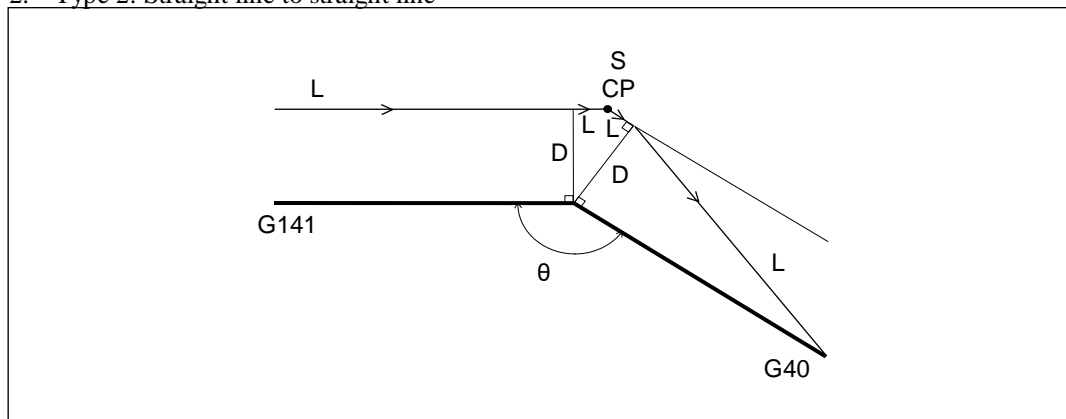
#### 1. Type 1: Straight line to straight line



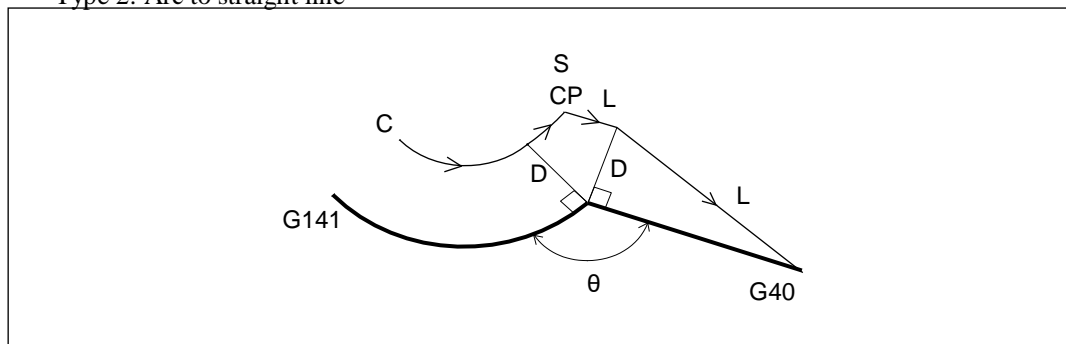
#### Type 1: Arc to straight line



#### 2. Type 2: Straight line to straight line



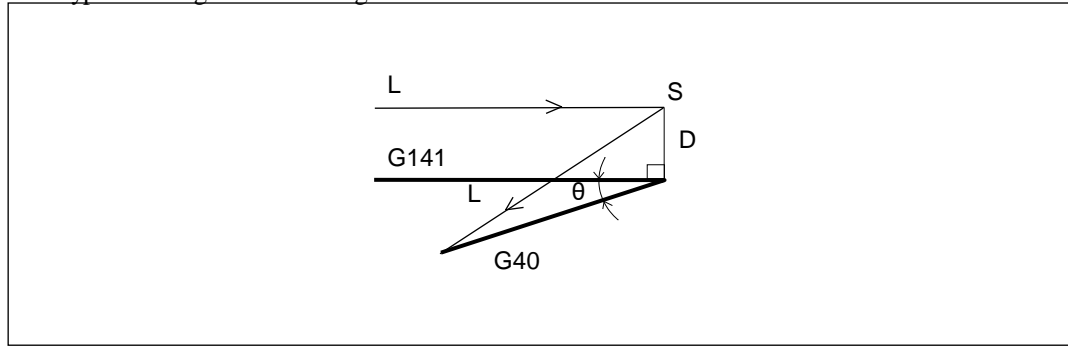
#### Type 2: Arc to straight line



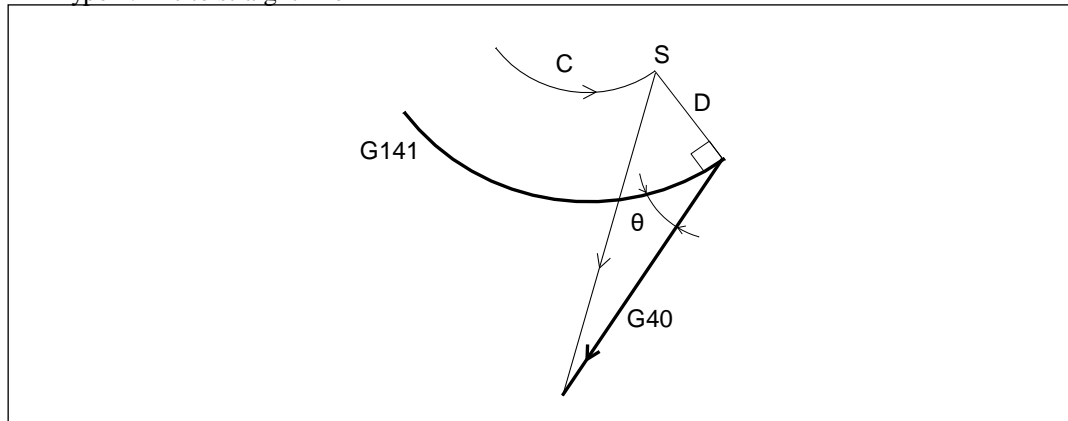
**(Note)** When  $179^\circ \leq \theta \leq 180^\circ$ , the setting <0: Type 1 (shortcut)> is used or enabled, even if <1: Type 2 (detour)> is specified for the user parameter <Start up/cancel>.

### 4.3.6.3 Outer Side (Acute Angle Cutting) ( $\theta < 90^\circ$ )

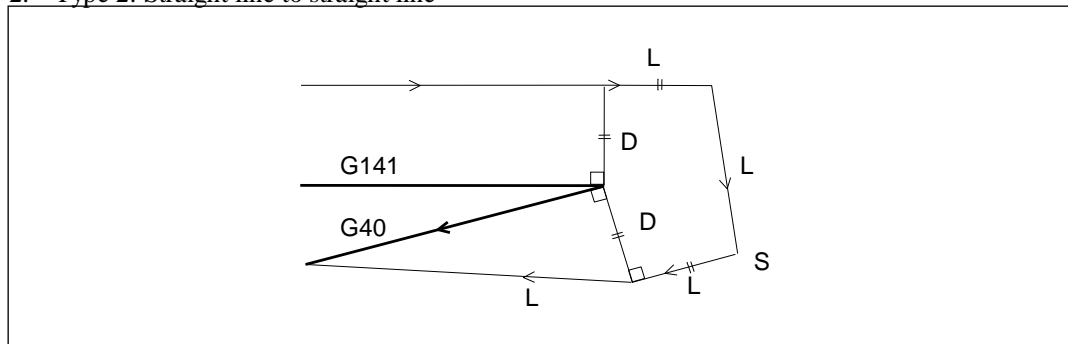
1. Type 1: Straight line to straight line



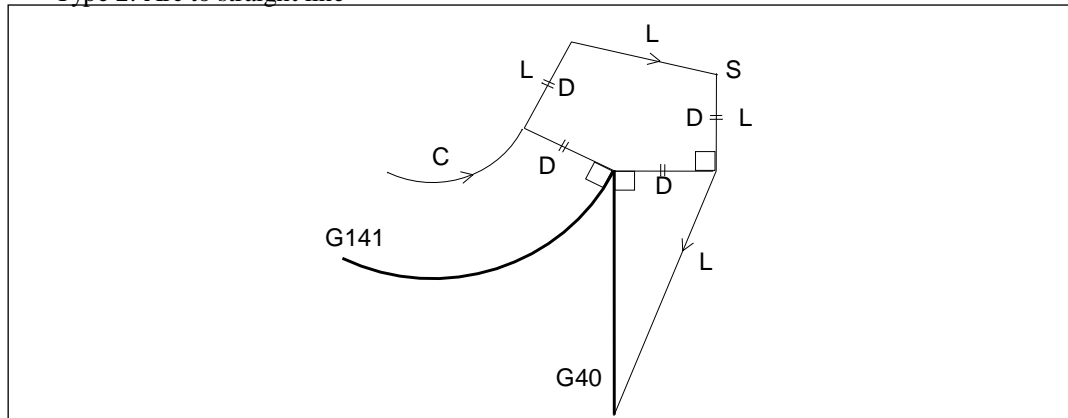
- Type 1: Arc to straight line



2. Type 2: Straight line to straight line



- Type 2: Arc to straight line

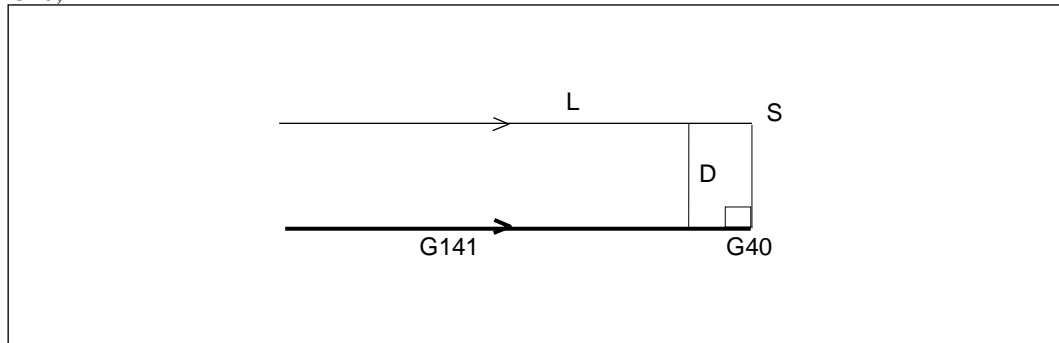


### 4.3.7 G40 Individual Command

When G40 is issued as an individual command, the nose R center comes to a position that is perpendicular to the command value for the previous block.

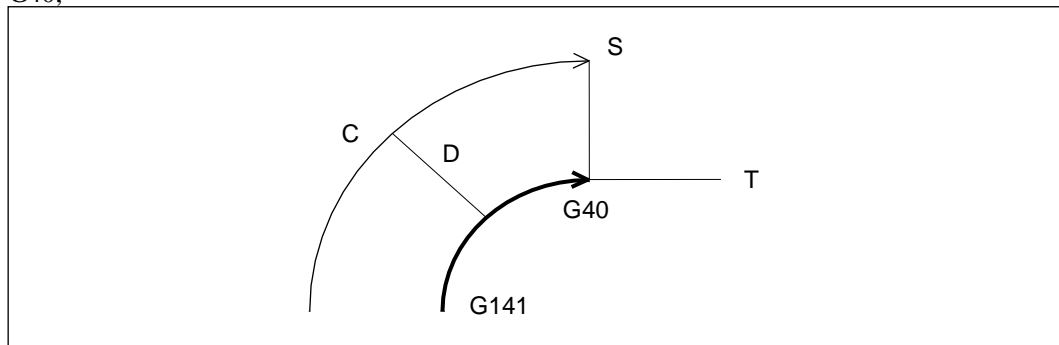
Straight line

G141 Y\_Z\_D\_;  
G40;



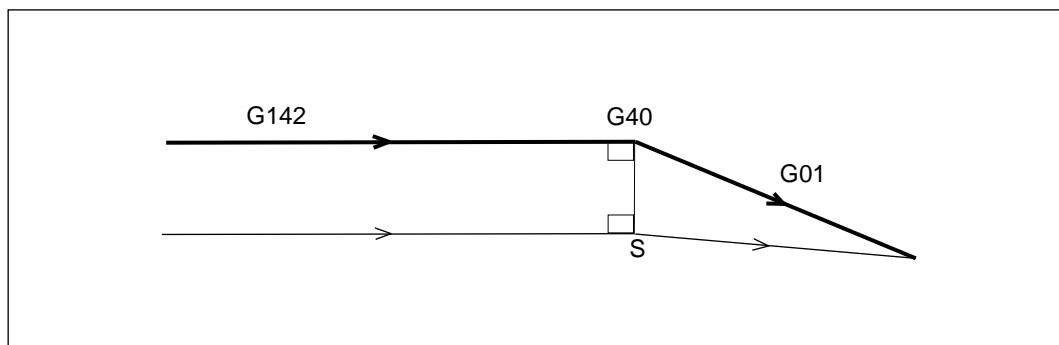
Arc

G141 Y\_Z\_D\_;  
G40;



**(Note)** The remaining offset is cancelled together with the next travel command.

G142 Y\_Z\_D\_;  
G40;  
G01 Y\_Z\_F\_;



### 4.3.8 Compensation Direction Change in Offset Mode

The compensation direction can be changed even while offset mode is enabled, by issuing a G141 or G142 command, or reversing the positive/negative sign for the compensation.

Note, the block following the startup block cannot be changed.

In addition, the compensation direction also cannot be changed even when changed using the mirror (single axis specification) or D address value, etc.

Compensation sign G code	+	-
G141	Left side offset	Right side offset
G142	Right side offset	Left side offset

#### Execution conditions

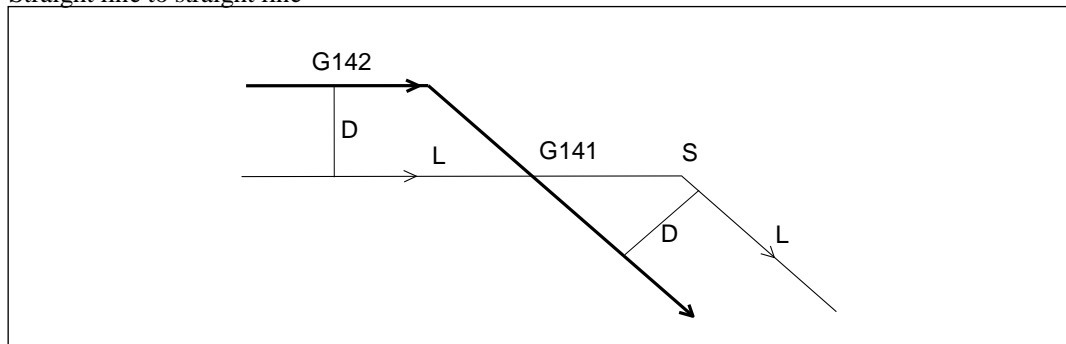
Offset mode	Command	Straight line to straight line	Straight line to arc	Arc to straight line	Arc to arc
G141	G141	Executes (The location that is offset by the nose R radius at a position perpendicular to the end point of the previous block becomes the stop point.)			
G142	G142				
G141	G142	Executes		Executes	
G142	G141				

There is no distinction between the inner side and outer side cutting when changing the compensation direction, but it varies depending on whether the intersection exists or not. In the following explanation, the compensation is positive.

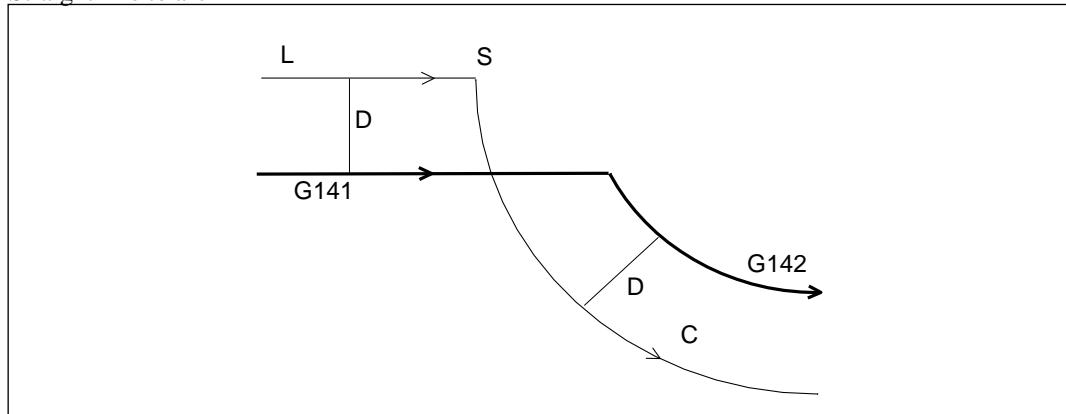
### 4.3.9 Offset Direction Change in Offset Mode

#### 4.3.9.1 When There is an Intersection

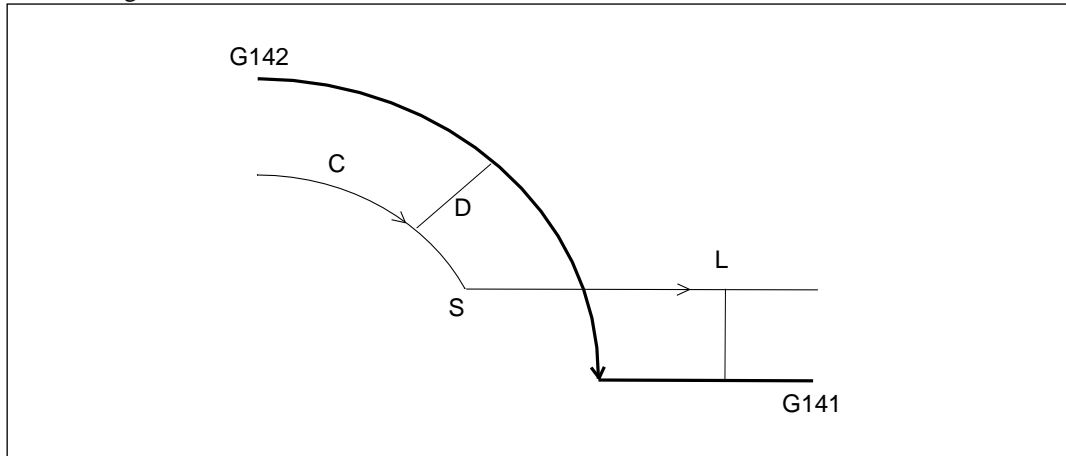
Straight line to straight line



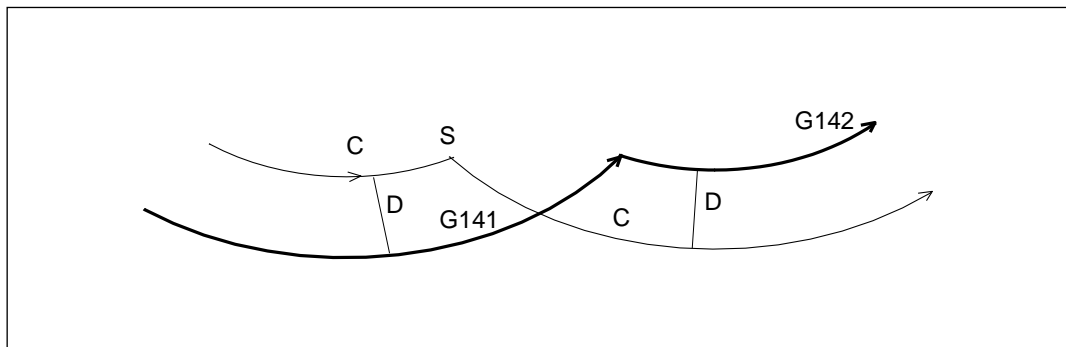
Straight line to arc



Arc to straight line

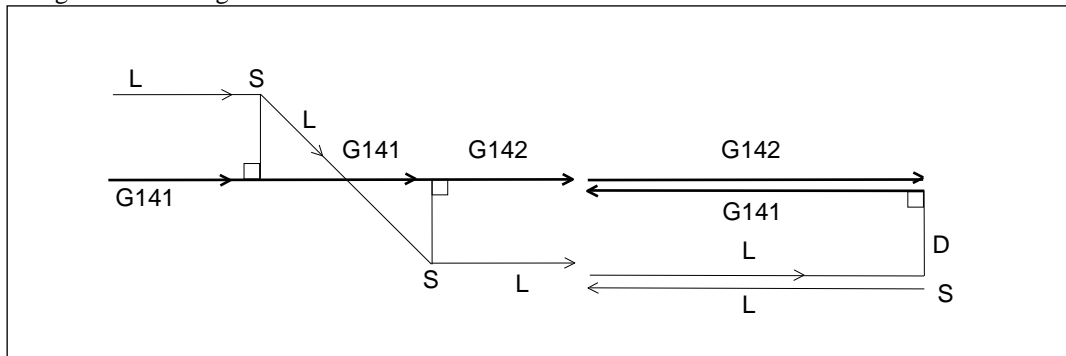


Arc to arc

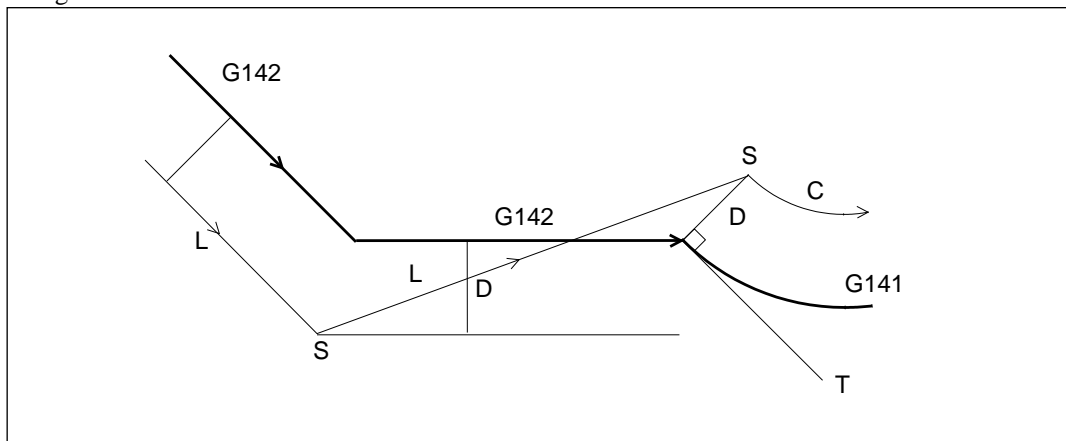


### 4.3.9.2 When There is No Intersection

Straight line to straight line

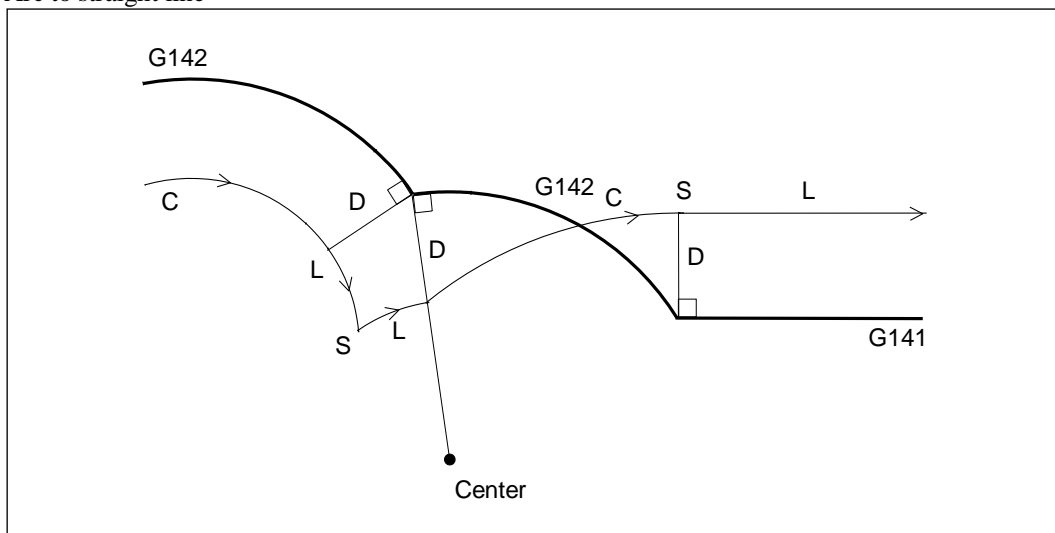


Straight line to arc

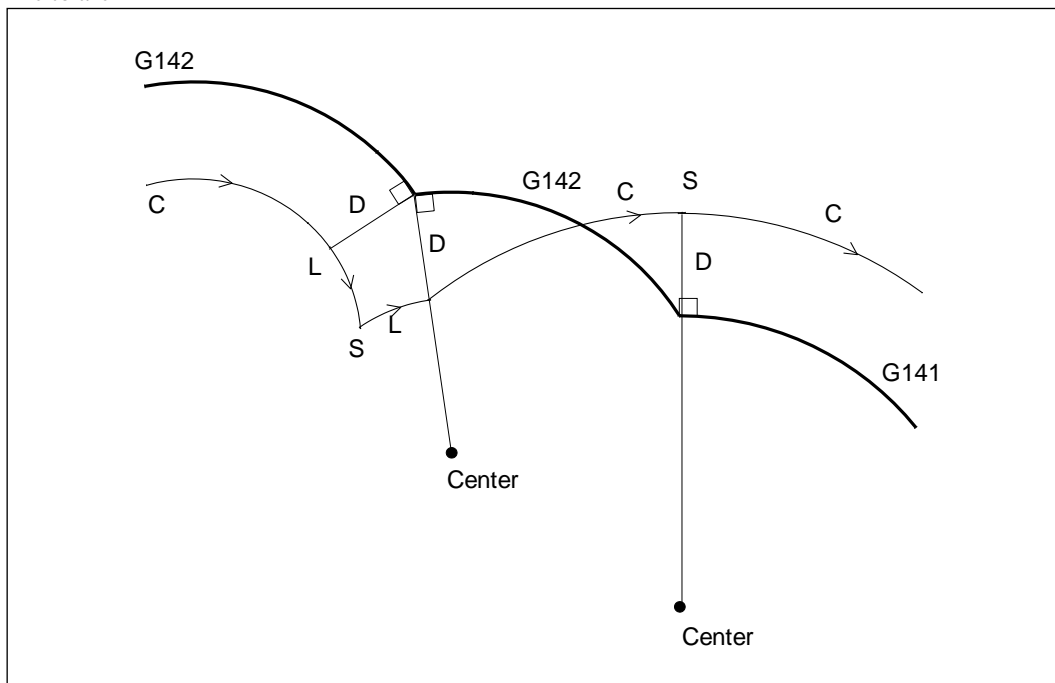




Arc to straight line

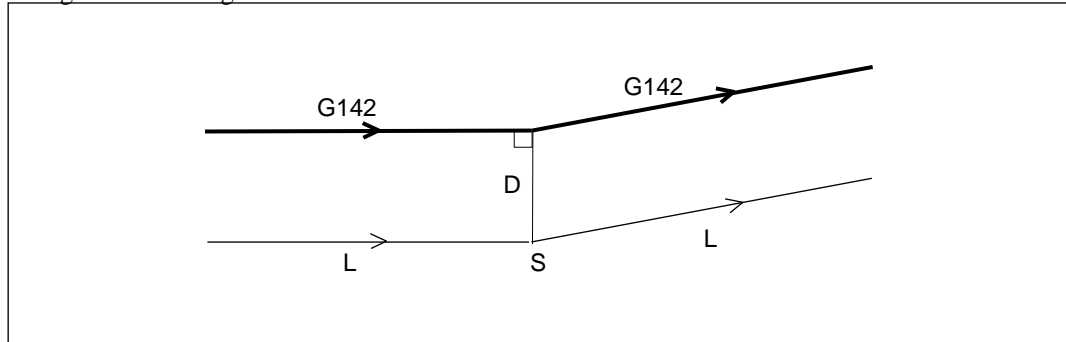


Arc to arc

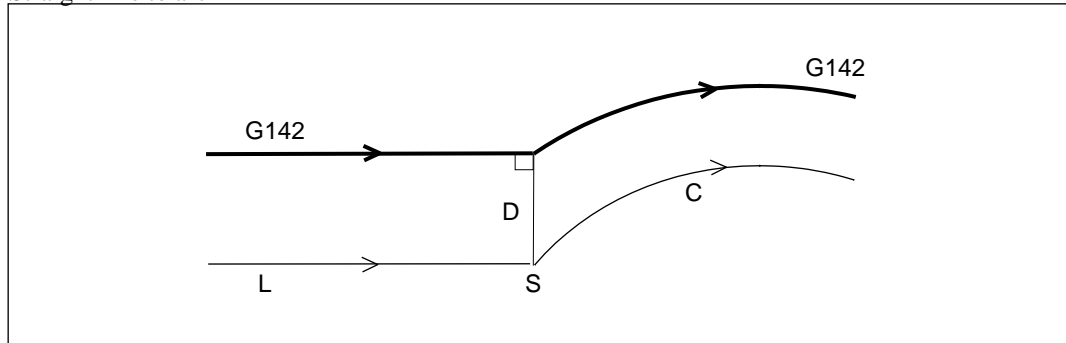


### 4.3.10 G Code Command for Nose R Compensation in Offset Mode

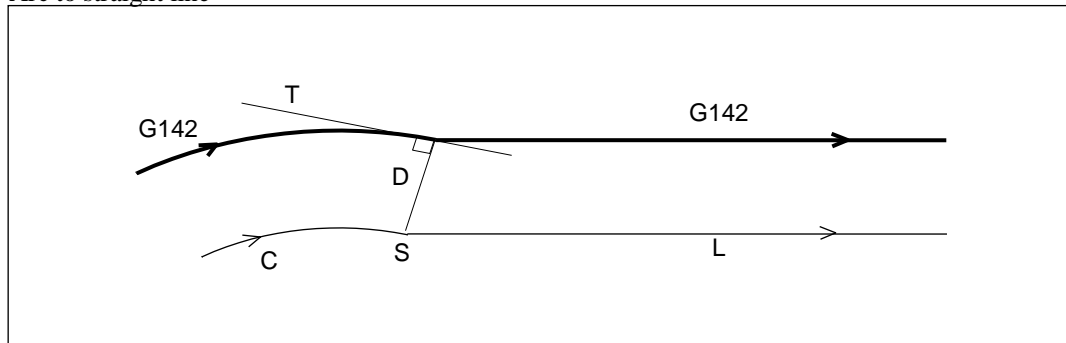
Straight line to straight line



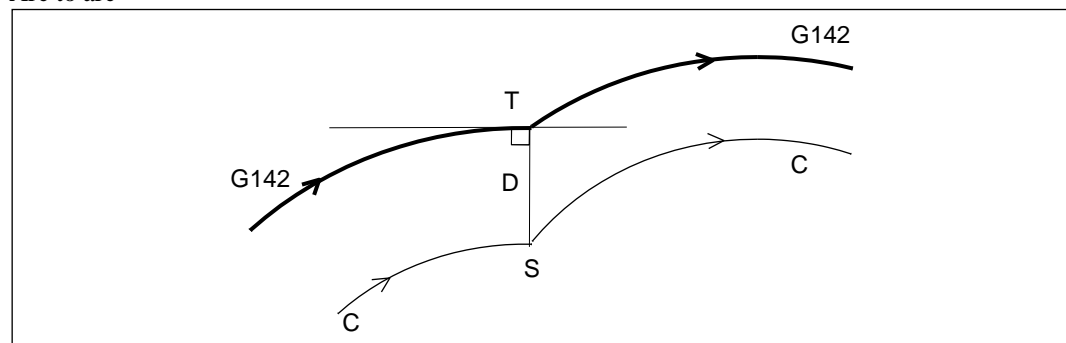
Straight line to arc



Arc to straight line



Arc to arc



### 4.3.11 Special Notes for Nose R Compensation

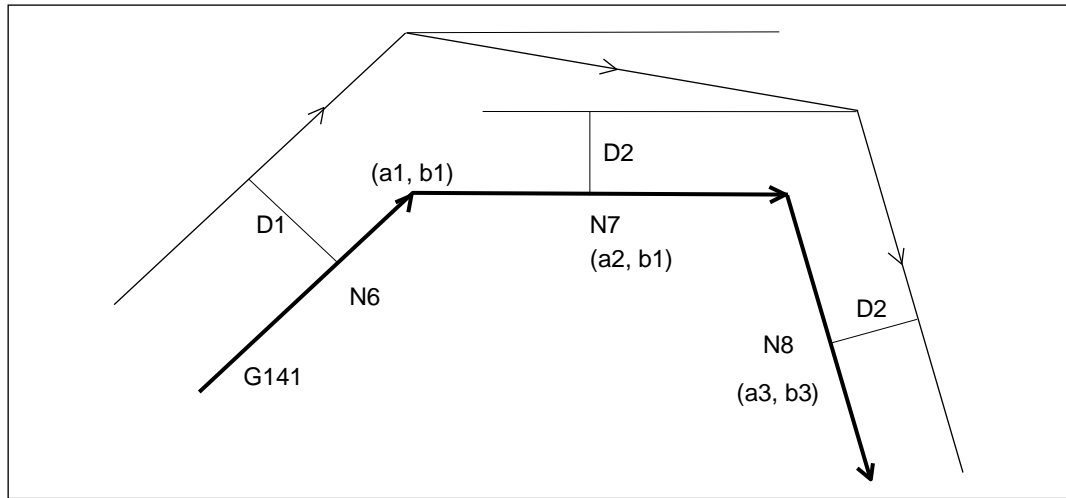
1. Nose R compensation command  
The nose R compensation command is specified by the number in the D command. The command is issued for the same block as the G141 or G142 command, but if that command is omitted, then the number for the D command that is issued previously is used.
2. Nose R compensation change  
When the nose R compensation is changed while in offset mode, that compensation change applies from the compensation for the end point of that block.

N1 G141 Y\_Z\_D1;

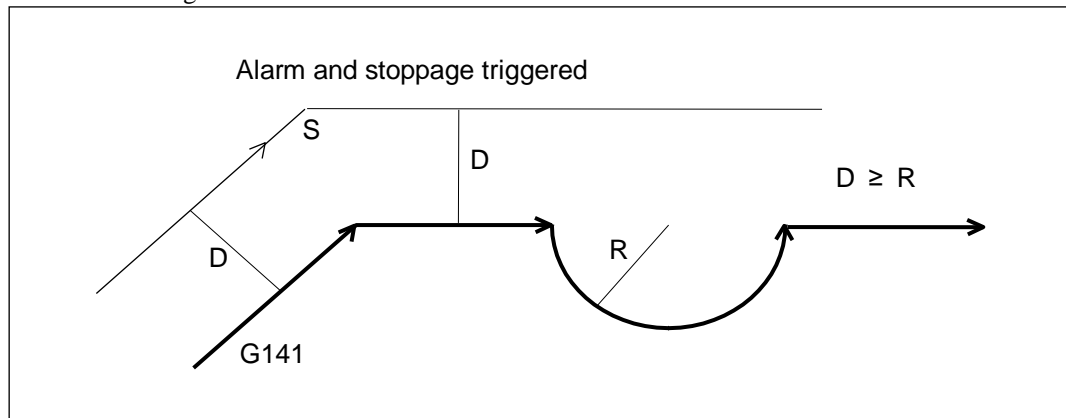
N6 Ya1 Zb1;

N7 Ya2 D2; ..... Compensation changed

N8 Ya3 Zb3;



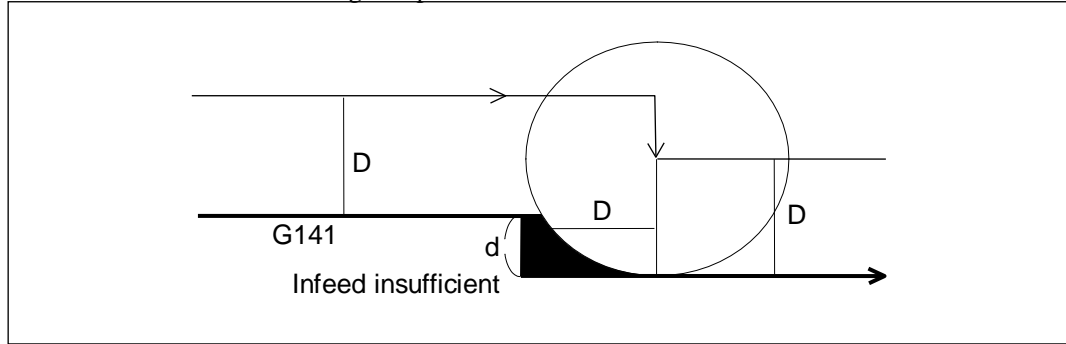
3. Current position display  
The current position display shows the virtual teeth position.
4. When cutting the inner side of an arc with a radius that is smaller than the nose R radius.



In this situation, the alarm <<Cutter compensation too large>> and stoppage are triggered because the infeed is not possible. It stops at the end point of the previous block.

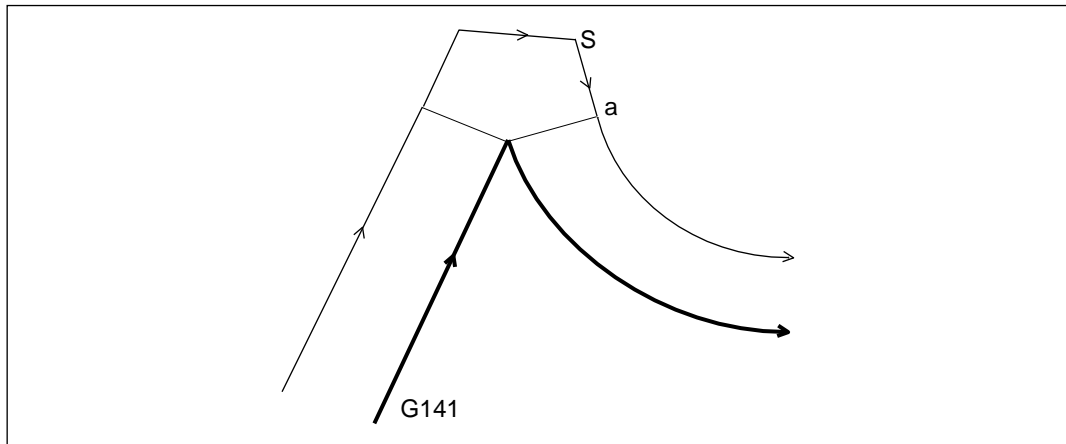
### 5. Infeed insufficient

This occurs when machining a step that is smaller than nose R.



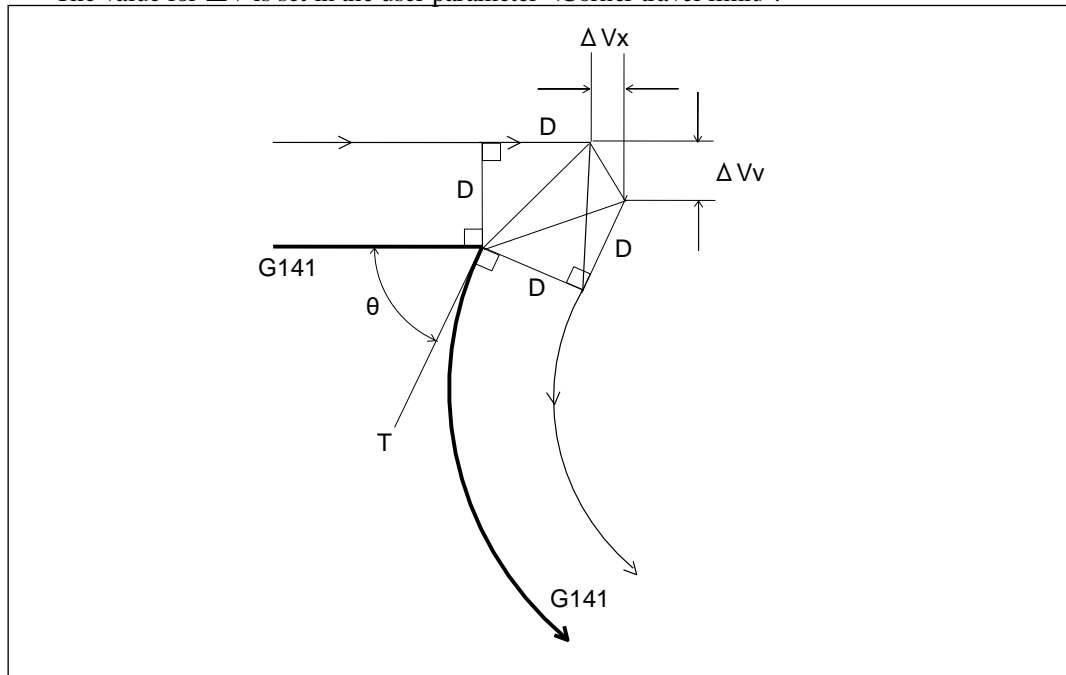
### 6. Corner travel

When cutting the outer side, it can turn on a corner with many angles. The travel mode and feedrate when turning the corner up to point *a* in the diagram below are based on the command that is issued for the current block.



In addition, as shown in the diagram below, the travel operation is ignored when the corner travel distance is extremely small, and when  $\Delta V_x \leq \Delta V$  and  $\Delta V_y \leq \Delta V$ .

The value for  $\Delta V$  is set in the user parameter <Corner travel limit>.

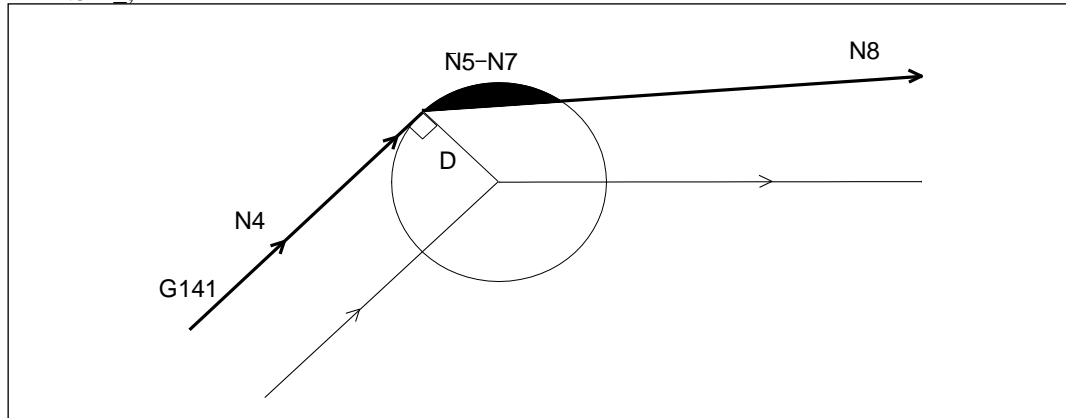


As a result, extremely small travel operations for a corner can be kept to a minimum.

7. Blocks without travel operations

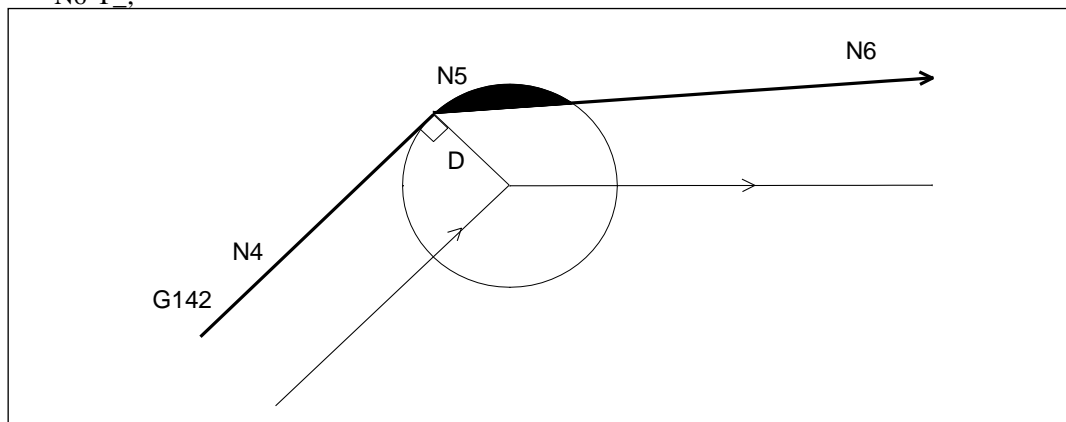
While in offset mode, if a command is issued for which the 2 axes on the selected planes do not travel for more than 3 blocks, the infeed will be too much or too little, as shown in the diagram below. Therefore, please avoid issuing those types of commands.

```
N4 Y_Z_;
N5 X_;
N6 F_;
N7 X_;
N8 Y_;
```



**(Note 1)** The same infeed problem arises as noted above for a block with zero travel.

```
N4 G91 Y_Z_;
N5 Y0;
N6 Y_;
```



**(Note 2)** If there is no travel command for 2 axes on the selected planes during startup, the startup operation is performed when a travel command is executed thereafter even on a single axis for either the Y- or Z-axis (when travel amount  $\neq 0$ ).

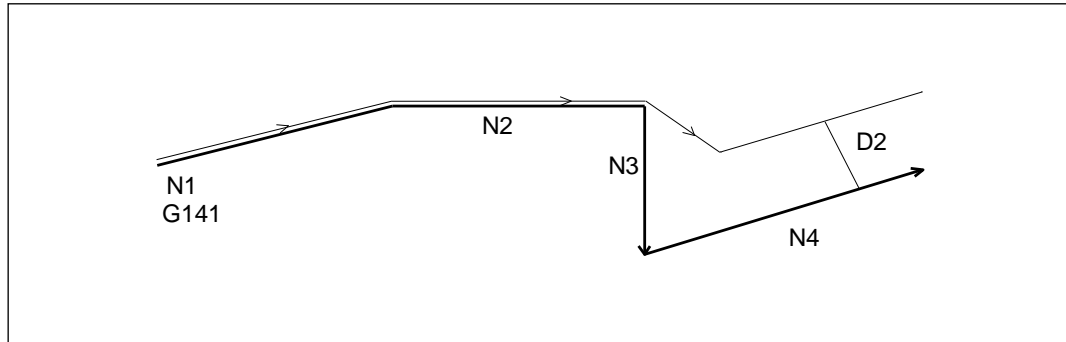
### 8. Tool movement when the nose R compensation is 0

#### (1) Startup

The offset mode is enabled when the G141 and G142 commands are issued while in cancel mode, but the startup operation is not performed because offset = 0.

The operation thereafter is the same as described in the section “2. Nose R compensation change” when changed to an offset number where offset  $\neq 0$ .

```
N1 G141 Y_Z_D1; (D1=0)
N2 Y_ ;
N3 Z_D2; (D2≠0)
N4 Y_Z_ ;
```



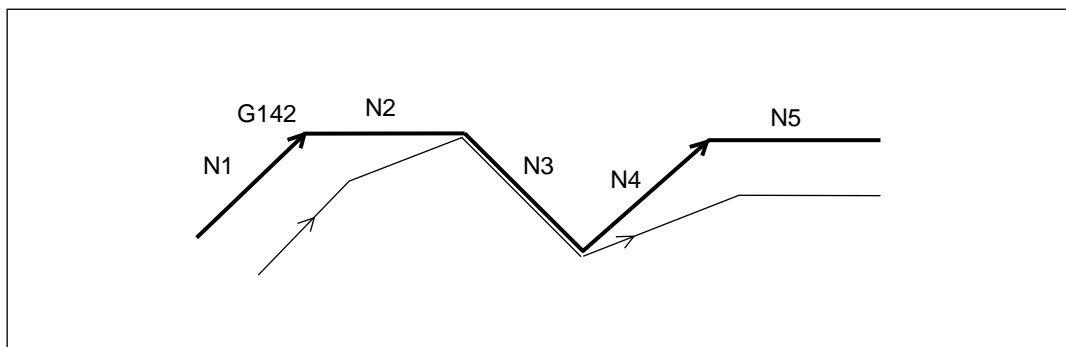
#### (2) Offset mode enabled

Cancel mode is not enabled even if changed to a D number where the nose R compensation = 0 while in offset mode.

The operation is the same as described in section “2. Nose R compensation change”.

The operation thereafter is the same as described in the section “2. Nose R compensation change” when changed to a D number where nose R compensation  $\neq 0$ .

```
N1 Y_Z_ ;
N2 Y_D1; (D1=0)
N3 Y_Z_ ;
N4 Y_Z_D2; (D2≠0)
N5 Y_ ;
```



9. Commands issued during nose R compensation that cause exception processing or that trigger alarms

- (1) Command that sets a perpendicular vector
  - G10 : Programmable data input
  - G52 : Local coordinate system setting
  - G92 : Coordinate system setting
  - G210 : Programmable data input (high accuracy)
  - #3000 : Alarm display
  - #3006 : Message display & stoppage

If the command noted above is issued, the machine travels to a position that is offset by the nose R compensation using the value from the last Y- and Z-axes travel command.

- (2) Command that forces the nose R compensation to cancel
  - M06 : Tool change
  - G100 : Nonstop ATC

If the command noted above is issued, G40 (nose R compensation cancel) is automatically triggered. Therefore, the machine travels to a position that is offset by the nose R compensation using the value from the last Y- and Z-axes travel command.

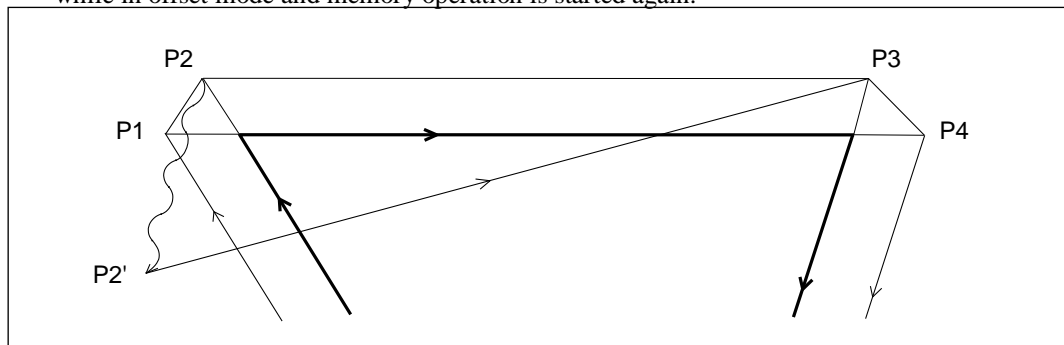
- (3) Command that triggers the alarm <<Compensating diameter>>
  - G17~G19 : Plane selection
  - G28 : Reference position return
  - G29 : Return from reference position
  - G30 : No. 2 to 6 reference position return
  - G33,G376,G392 : Thread cutting
  - G36~G39 : Coordinate calculation
  - G60 : Single direction positioning
  - G66 : Macro program modal call
  - G68.2 : Feature coordinate setting
  - G73~G89,G173~G189 : Canned cycle
  - G120 : Positioning to measurement position
  - G121~G129 : Automatic measurement
  - G31,G131,G132 : Skip feed
  - G133,G134 : Change tap twist direction
  - M410,M411 : Pallet index
  - G2,G3 : Arc with 0 start point or 0 end point radius

10. Input command from MDI operation

The alarm <<Specified G code cannot be used>> is triggered when there is an input related to nose R compensation (G40, G141 or G142) in MDI operation mode.

11. Manual operation intervention

The correct offset path is enabled on block 2 when the tool is moved using manual operation while in offset mode and memory operation is started again.



- \* When operation stops at the end point (P2) of a block and the tool is then moved manually, the tool travels from P2' to P3, and the correct path is enabled from P3.

## 12. Command after cancelling nose R compensation

If a G17 to G19 (plane selection) command is issued when the G40 command is issued individually and there is a remaining offset amount, the alarm <<Cutter compensation error>> is triggered. When a travel command is issued for the same block as G40 or after the G40 command, issue a command after cancelling the offset amount.

### 4.3.12 Override Function Related to Nose R Compensation

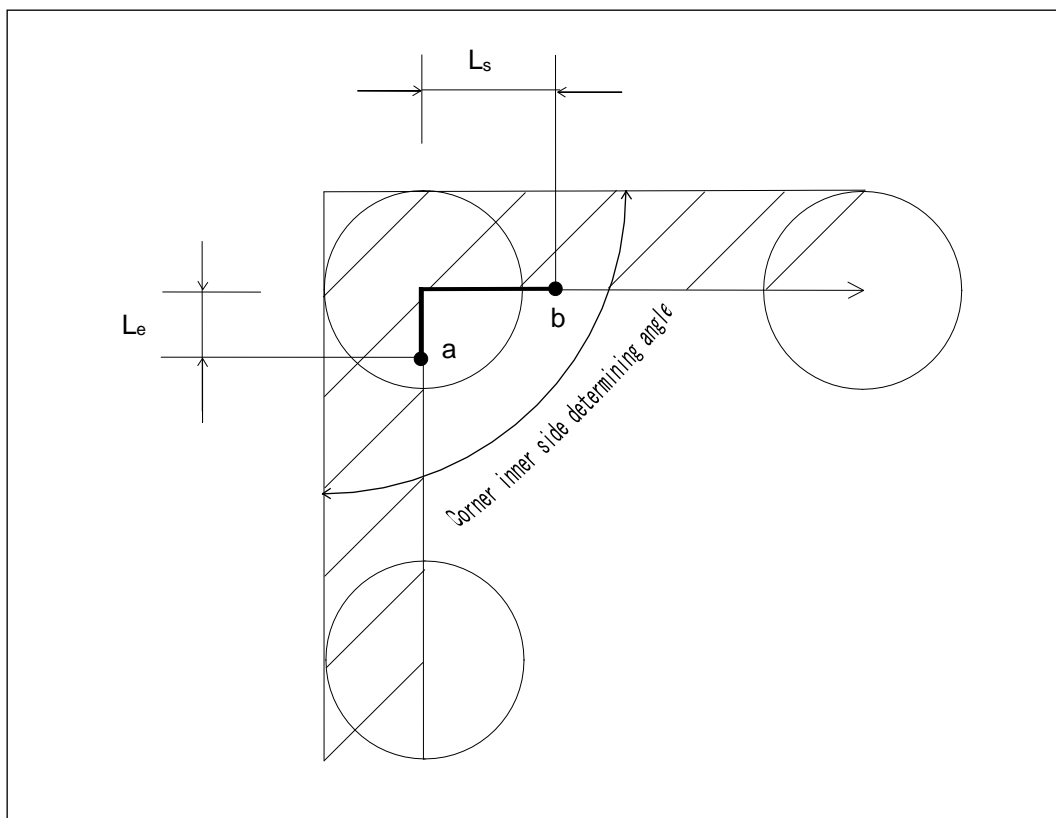
#### 4.3.12.1 Automatic Corner Override

When both the block before and after the corner of the inner side meet the following conditions while in offset mode, the override function is automatically enabled in order to reduce the load on the tool.

1. G01, G02 or G03 travel operation. (Excluding spiral/conical interpolation)
2. Offset  $\neq 0$  when offset mode is enabled.
3. The corner's inner side angle is less than the user parameter <Automatic corner override (angle)>.
4. The block does not include the following commands: G141, G142 and G40.
5. The compensation direction does not change.

The following items are configured in the user parameter settings.

- (1) Automatic corner override (length 1) : Corner end point deceleration distance  $L_e$
- (2) Automatic corner override (length 2) : Corner start point deceleration distance  $L_s$
- (3) Automatic corner override (ratio) : Deceleration ratio (%)  $Y$
- (4) Automatic corner override (angle) : Corner inner side determining angle  $\theta$



The override applies to the section — from point  $a$  to point  $b$

$$\text{Actual feedrate} = \text{Command speed} \times (\text{Deceleration ratio}/100)$$



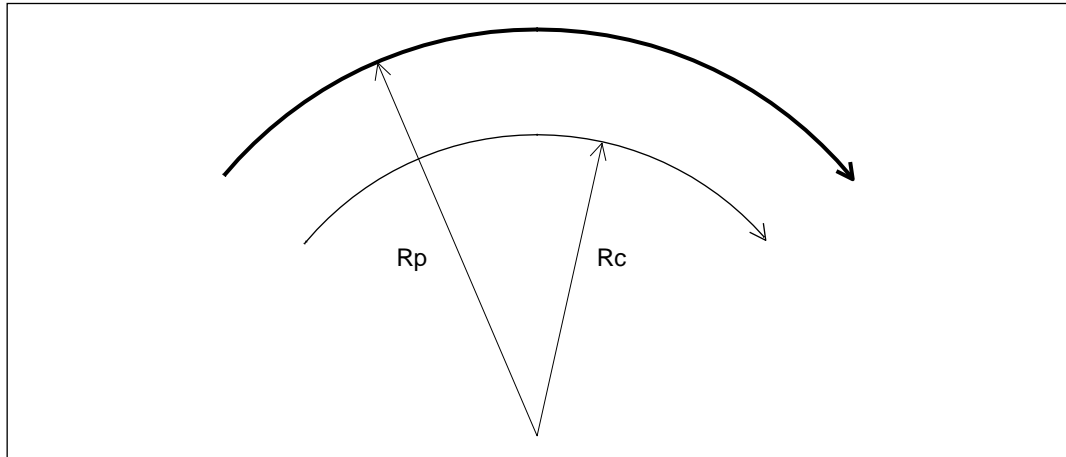
### 4.3.12.2 Inner Arc Override

When performing arc cutting that is offset on the inner side during offset mode, the actual feedrate is the product of  $R_c/R_p$  for the feedrate command that is issued.

$$\text{Actual feedrate} = \text{Command speed} \times R_c/R_p$$

$R_p$ : Program radius

$R_c$ : Tool center path radius

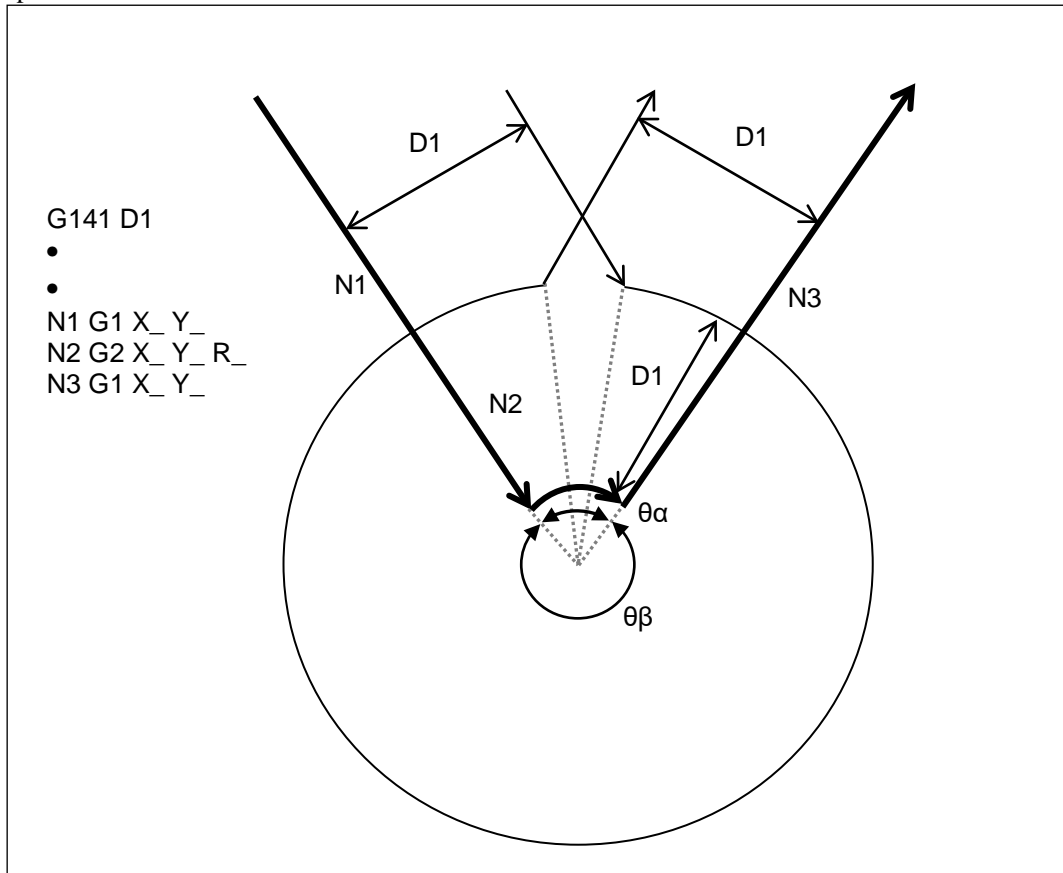


**(Note)** When  $R_c/R_p$  is less than the <Inner arc override limit> that is set in the user parameter, that parameter value is multiplied as an alternative to  $R_c/R_p$ .

$$\text{Actual feedrate} = \text{Command speed} \times (\text{Inner arc override limit} / 100)$$

### 4.3.13 Arc Angle Check During Inner Side Cutting

During inner side cutting, if the arc angle in the program path for the arc command and the arc angle in tool center path after being offset are significantly different, this function stops the operation before the arc motion.



This function checks the arc angle ( $\theta\alpha$ ) at the start and end points in the N2 program path and the arc angle ( $\theta\beta$ ) at the start and end points in tool center path after being offset. If the angle is greater than  $180^\circ$ , then an alarm is triggered and operation stops before executing N2.

**(Note 1)** When an alarm is triggered, the infeed may already be too great (The infeed is too great for the workpiece on the N3 side after N1 is executed in the above example).

**(Note 2)** This function carries out the check at the end point after 3 blocks of travel. If one of the situations below applies while the offset mode is enabled, this check function may not work properly because the tool center path start and end points change after being offset.

- When there is a nose R compensation G code command or a command that sets a perpendicular vector
- Zero travel commands for more than 3 blocks

## 4.4 Tool Position Compensation (G143, G144 and G49 - Option)

### 4.4.1 Tool Position Compensation Function

\* Available when equipped with a lathe function

This function offsets the tool position so that the teeth move to the position that is programmed. Even in an absolute command or an incremental command, the coordinates that are offset only for the tool number compensation specified in H code become the actual end point for the coordinates of the travel command end point that is programmed.

The compensation can be set on the tool list setting screen.

X-axis compensation: Tool position offset (X) + Tool position wear offset (X)

Y-axis compensation: Tool position offset (Y) + Tool position wear offset (Y)

Z-axis compensation: Tool length offset (Z) + Tool length wear offset (Z)

Offset is performed for X-, Y- and Z-axes. Set 0 for the axis where the compensation does not apply.

1. Tool position compensation (+)

Command format

**G143 Hn;**

Hn : Tool number (n = 0 to 99), or group number (n = 901 to 930)

2. Tool position compensation (-)

Command format

**G144 Hn;**

Hn : Tool number (n = 0 to 99), or group number (n = 901 to 930)

3. Tool position compensation cancel

Command format

**G49;**

**(Note 1)** When the tool position compensation is cancelled, it is cancelled by the G49 command or by issuing 0 for the tool number.

**(Note 2)** The tool position compensation is cancelled by the M06 (tool change) or by the G100 (nonstop ATC) command.

**(Note 3)** Refer to the next section “Axis travel with tool position compensation command” for axis operation when there are no travel commands for G143H\_, G144H\_, or for G49 during tool position compensation and H0 command blocks.

**(Note 4)** When an X-, Y and Z-axes command is issued during tool position compensation for reference position return (G28) or No. 2 to 6 reference position return (G30), the tool position compensation stays enabled while traveling to the middle point. And, the tool position compensation is cancelled temporarily while travelling to the reference position. Refer to the next section “Resume tool position compensation” for travel when the tool position compensation that was cancelled resumes. When the tool position compensation resumes, if the incremental mode is enabled, it is the equivalent of traveling from the absolute coordinates right before.

**(Note 5)** If the G53X\_Y\_Z\_ command is issued during the tool position compensation, the tool position compensation is temporarily cancelled during travel.

**(Note 6)** A range check is performed when the range for the tool that is specified in H code is set for the following items: the tool length offset, the tool length wear offset, the tool position offset and the tool position wear offset. The alarm <<Comm. issued to area other than (tool) data area>> is triggered when the command area is outside of the range.

(Note 7) If a tool length offset command (G143 and G144) is issued during G43 and G44 modals, an alarm is triggered. Note, if a tool change command (G100 and M06) is issued on the same block, no alarm is triggered.

## 4.4.2 Axis Travel with Tool Position Compensation

1. X-, Y- and Z-axis travel for tool position compensation command

When there are X-, Y- and Z-axes travel commands for the command block G143 / G144 / G49, such as G143 H\_X\_Y\_Z\_, G144 H\_X\_Y\_Z\_ and G49 X\_Y\_Z\_, the X-, Y-, and Z- axes in that block travel to the position that takes into account the compensation specified in H code.

When there are no X-, Y- and Z-axis travel commands for G143 / G144 / G49 / H0 command blocks, such as G143H\_, G144H\_ or tool position compensation G49 and H0, the operation follows the user parameter noted below.

User parameter <Travel of X, Y or Z axis when tool length/tool position offset is changed>	Type 1		Type 2
User parameter <Error check when traveling during tool length/tool position offset cancel>	Check	No check	-
G143/G144 command block without X-, Y- and Z-axes command Ex.1) G143H1; Ex.2) G143; Ex.3) H1; (Tool position compensation enabled)	Compensation specified in H code and X-, Y- and Z-axes travel		Axis does not travel (Travels to compensated position specified in H code for the next X-, Y- and Z-axes travel command)
G49/H0 command block without X-, Y- and Z-axes command (Tool position compensation enabled) Ex.1) G49; Ex.2) H0;	When the current compensation for X-, Y- and Z-axes is another value other than 0, the alarm <<Tool position offset cancel error>> is triggered. (Note 2, 3 and 4)	Current tool position compensation and X-, Y- and Z-axes travel (Note 2)	Axis does not travel (Travels to position where the current tool position compensation is cancelled for the next X-, Y- and Z-axis travel command)

X-, Y- and Z-axes motion example

(Workpiece coordinate zero → X: -100.000, Y: -200.000, Z: 50.000, H1 offset → X: 5.000, Y: 0.000 and Z: 120.000)

	<Travel of X, Y or Z axis when tool length/tool position offset is changed> : <Type 1> <Error check when traveling during tool length/tool position offset cancel> : <Yes>		
	X	Y	Z
	Machine coordinate	Machine coordinate	Machine coordinate
G90G0X0.Y0.Z100.;	-100.000	-200.000	150.000
G143H1;	-95.000	-200.000	270.000
G0X10.Y20.Z80.;	-85.000	-180.000	250.000
G49;	The alarm <<Tool position offset cancel error>> is triggered.		

	<Travel of X, Y or Z axis when tool length/tool position offset is changed> : <Type 1> <Error check when traveling during tool length/tool position offset cancel> : <No>		
	X	Y	Z
	Machine coordinate	Machine coordinate	Machine coordinate
G90G0X0.Y0.Z100.;	-100.000	-200.000	150.000
G143H1;	-95.000	-200.000	270.000
G0X10.Y20.Z80.;	-85.000	-180.000	250.000
G49;	-90.000	-180.000	130.000

	<Travel of X, Y or Z axis when tool length/tool position offset is changed> : <Type 2>*		
	X	Y	Z
	Machine coordinate	Machine coordinate	Machine coordinate
G90G0X0.Y0.Z100.;	-100.000	-200.000	150.000
G143H1;	-100.000	-200.000	150.000
G0X10.Y20.Z80.;	-85.000	-180.000	250.000
G49;	-85.000	-180.000	250.000

- \* The operation is the same regardless if the parameter <Error check when traveling during tool length/tool position offset cancel> is set to <Yes> or <No>.

- (Note 1) When a circular interpolation command or an involute interpolation command is issued during travel, an alarm is triggered.
- (Note 2) When the tool position compensation is temporarily canceled as per G28 or G53, cancel travel for the compensation does not occur even for a G49/H0 command block without a X-, Y- or Z-axis command. As a result, the alarm <<Tool position offset cancel error>> is not triggered even when the parameter <Travel of X, Y or Z axis when tool length/tool position offset is changed> is set to <0: Type 1> and the parameter <<Error check when traveling during tool length/tool position offset cancel>> is set to <0: No>.
- ```

G90
G143 H1 X0. Y0. Z100.
G91 G28 X0 Y0 Z0
G49    ←No X-, Y- and Z-axes travel

```
- (Note 3) An alarm is not triggered during scaling, mirror imaging and rotational transformation because the axis travel command is on the same block as G49 and H0 and an axis other than the axis that was specified may operate.
- (Note 4) When the parameter <X-, Y- and Z-axes travel during current rotary fixture offset change> is set to <Type 2>, the alarm is not triggered if the X-, Y- and Z-axes operate following a change in the workpiece coordinates.
- (Note 5) When the parameter <Travel of X, Y or Z axis when tool length/tool position offset is changed> is set to <1: Type 2> and the G143 or G144 command is issued on the same block as G53, an alarm is triggered.

## 2. Tool position compensation resumed

The tool position compensation can be temporarily canceled due to one of the following X-, Y- and Z-axes commands: reference position return (G28/G30), machine coordinate selection (G53), positioning to the measurement position (G120) or external indexing on the pallet (M410/M411). The compensation or offset is resumed during the next X-, Y- or Z-axis travel command, H command or G143/G144 command.

Ex:

|                        |   |                                                    |
|------------------------|---|----------------------------------------------------|
| G28 X-50. Y-50. Z400.; | ← | Temporarily cancels the tool position compensation |
| ...;                   |   |                                                    |
| G0 X-100. Y-100.;      | } | ← X- and Y-axes tool position compensation resumes |
| ...;                   |   |                                                    |
| G0 Z300.;              | ← | Z-axis tool position compensation resumes          |

(This page was intentionally left blank.)

## CHAPTER 5

# PREPARATION FUNCTION (CANNED CYCLE)

5

- 5.1 Outline
- 5.2 List of Canned Cycle Function
- 5.3 Basic Operation of Canned Cycle
- 5.4 General Rules of Canned Cycle
- 5.5 Details of Canned Cycle
- 5.6 One-shot Canned Cycle
- 5.7 Canned Cycle for Tool Change (Non-stop ATC)  
(G100)
- 5.8 Coordinate Calculation Function

## 5.1 Outline

A sequence of machine operations mostly in the tool axis direction is included in a single block and initiated by a single G code.

## 5.2 List of Canned Cycle function

Functions of canned cycle

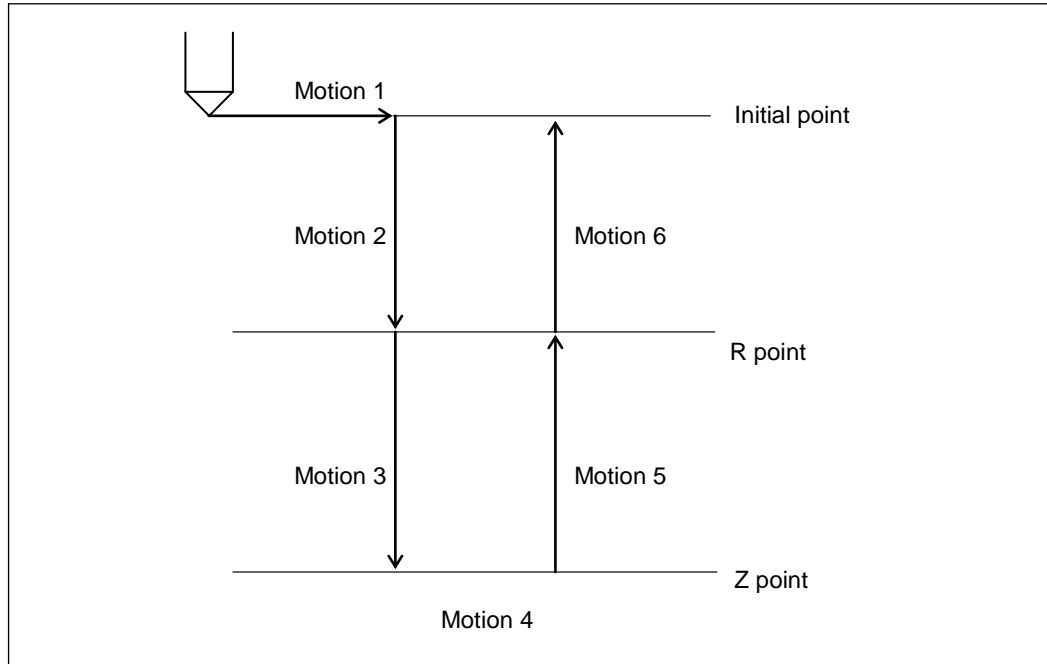
| G code | Application                                    | Boring            | Operations at hole position | Retract                        | Spindle rotation at return point |
|--------|------------------------------------------------|-------------------|-----------------------------|--------------------------------|----------------------------------|
| G73    | High speed peck drilling                       | Intermittent feed | Dwell                       | Rapid traverse                 |                                  |
| G74    | Reverse tapping                                | Cutting feed      | Dwell→Spindle CW            | Cutting feed                   | Stop                             |
| G76    | Fine balling                                   | Cutting feed      | Dwell→Orientation           | Rapid traverse                 | normal rotation                  |
| G77    | Tapping (Synchro mode)                         | Intermittent feed | Spindle CCW                 | Cutting feed                   | Stop                             |
| G78    | Reverse tapping (Synchro mode)                 | Intermittent feed | Spindle CW                  | Cutting feed                   | Stop                             |
| G80    | Cancel                                         | -                 | -                           | -                              | -                                |
| G81    | Drilling                                       | Cutting feed      | Dwell                       | Rapid traverse                 |                                  |
| G82    | Drilling                                       | Cutting feed      | Dwell                       | Rapid traverse                 |                                  |
| G83    | Peck drilling                                  | Intermittent feed | Dwell                       | Rapid traverse                 |                                  |
| G84    | Tapping                                        | Cutting feed      | Dwell→Spindle CCW           | Cutting feed                   | Stop                             |
| G85    | Boring                                         | Cutting feed      | Dwell                       | Cutting feed                   |                                  |
| G86    | Boring                                         | Cutting feed      | Dwell→Spindle stop          | Rapid traverse                 | normal rotation                  |
| G87    | Back balling                                   | Cutting feed      | Dwell→Orientation           | Rapid traverse                 | normal rotation                  |
| G89    | Boring                                         | Cutting feed      | Dwell                       | Cutting feed                   |                                  |
| G177   | End milling/tapping                            | Cutting feed      | Spindle CCW                 | Cutting feed                   | Stop                             |
| G178   | End milling/tapping                            | Cutting feed      | Spindle CW                  | Cutting feed                   | Stop                             |
| G181   | Double drilling                                | Cutting feed      | Dwell                       | Rapid traverse                 |                                  |
| G182   | Double drilling                                | Cutting feed      | Dwell                       | Rapid traverse                 |                                  |
| G185   | Double boring                                  | Cutting feed      | Dwell                       | Cutting feed<br>Rapid traverse |                                  |
| G186   | Double boring                                  | Cutting feed      | Dwell→Spindle stop          | Rapid traverse                 | normal rotation                  |
| G189   | Double boring                                  | Cutting feed      | Dwell                       | Cutting feed<br>Rapid traverse |                                  |
| G277   | Deep hole tapping cycle (synchro mode)         | Intermittent feed | Spindle CCW                 | Cutting feed                   | Stop                             |
| G278   | Deep hole reverse tapping cycle (synchro mode) | Intermittent feed | Spindle CW                  | Cutting feed                   | Stop                             |



## 5.3 Basic Operation of Canned Cycle

A canned cycle generally consists of six motions:

- Motion 1 ..... Move the tool to hole machining position (X-Y axes) by rapid feed
- Motion 2 ..... Move the tool to R point by rapid feed
- Motion 3 ..... Hole machining (cutting feed)
- Motion 4 ..... Operation at the bottom of a hole
- Motion 5 ..... Retreat the tool to R point by rapid feed / cutting feed
- Motion 6 ..... Return the tool to initial point by rapid feed



In a single block operation, the control stops at the end of motions 1, 2, and 6, respectively.

**(Note) Temporary stop availability in tapping cycles (G74, G77, G78, G84, G177, G178, G277, and G278)**

1. Temporary stop is available for motions 1, 2, and 6.
2. Temporary stop is not available for motions 3 to 5. When temporary stop-involving operations are performed (pressing the stop switch, selecting the manual mode, etc.), the control stops at the end of motion 5. This also applies when you press the reset key in motion 3, 4, or 5.

### 5.4.1 Canned Cycle Operation Commands

### (I) Data format

(II)Return level

Please refer to “5.2 List of canned cycle function”

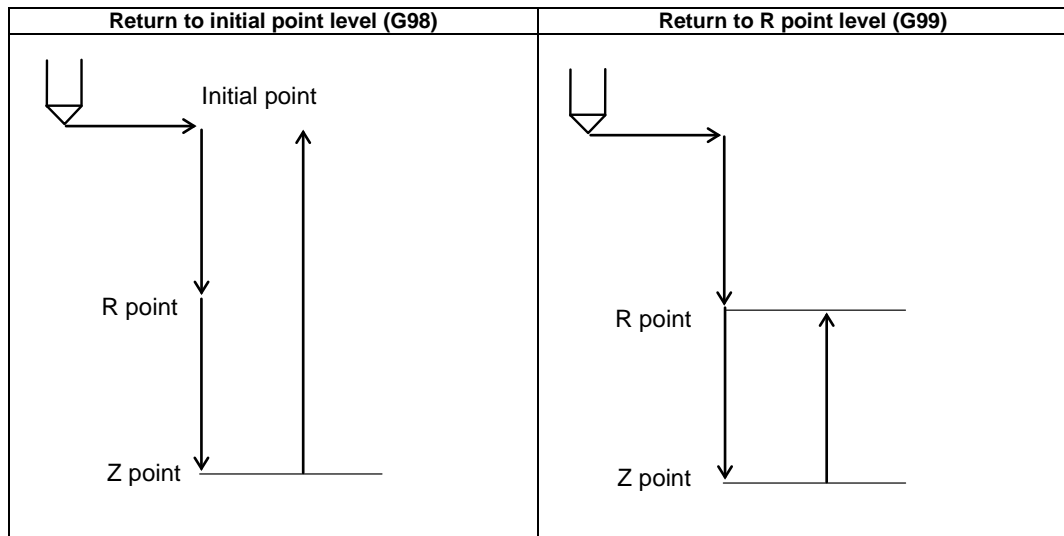
The diagram illustrates the difference between Absolute and Incremental coordinate systems for a vertical axis.

**Absolute:** The vertical axis has a fixed reference point labeled  $Z=0$ . A point is labeled "R point" and another point further down is labeled "Z point". The distance from the  $Z=0$  line to the "R point" is labeled  $R$ . The distance from the  $Z=0$  line to the "Z point" is labeled  $Z$ .

**Incremental:** The vertical axis has a fixed reference point labeled "Initial point". A point is labeled "R point" and another point further down is labeled "Z point". The distance from the "Initial point" to the "R point" is labeled  $R$ . The distance from the "R point" to the "Z point" is labeled  $Z$ .

### 5.4.3 Types of Return Points (G98, G99)

Two available tool return levels on completing a canned cycle operation are Initial Point (G98) and R Point (G99) level.



(Note 1) G98 and G99 are modal commands. G98 is always effective when the machine is powered on.

(Note 2) When a canned cycle command and a tool length offset command exist in the same block, the tool length offset command is executed after the tool reaches the R point. Accordingly, the initial point is saved without tool length offset.

(Note 3) Initial point is the Z axis position in the machine coordinates when the control is switched from the canned cycle cancel mode to the canned cycle mode.

### 5.4.4 Canned Cycle Operating Conditions

Canned cycle operations are performed under the following conditions:

1. The control is in one of the hole machining blocks (G73, G74, G76 to G78, G81 to G87, G89, G177, G178, G181 to G182, G185, G186, G189, G277, and G278) and at least one of X, Y, Z, R, A, B, and C is included in the block.
2. Any block, including at least one of X, Y, Z, R, A, B, and C, that occurs following a hole machining block, up until the canned cycle command is canceled in a later block.

(Note 1) In a canned cycle operation, if any of X, Y, Z, R, A, B, or C is non-existent and other hole machining data is commanded, nothing is performed except that the relevant hole machining data is saved.

(Note 2) The canned cycle command cannot be issued when using the turning spindle in the following situations. After switching modals, issue the command.

- M142 modal in progress
- G143/G144 modal in progress

(Note 3) A canned cycle command cannot be issued while the feature coordinate is being set (after G68.2 command and before G53.1 command). A command is possible while the feature coordinate is being indexed (after G53.1 command).

## 5.4.5 Canned Cycle Machining Data

Command  
format

|                     |                              |                                 |                   |
|---------------------|------------------------------|---------------------------------|-------------------|
| <b>Gxx</b><br>[ ]   | <b>X_ Y_ A_ B_ C_</b><br>[ ] | <b>Z_ R_ Q_ P_ F_ S_</b><br>[ ] | <b>K_;</b><br>[ ] |
| Hole machining mode | Hole position data           | Hole machining data             | Repetition count  |

G codes : G73, G74, G76 to G78, G81 to G87, G89, G177, G178, G181 to G182, G185, G186, G189, G277, and G278

All canned cycle G codes are modal.

- X,Y,A,B,C** : Drilling position  
Rapid feed is used to go to the drilling position.  
An alarm occurs when an additional axis is commanded when it is not installed.
- Z** : Hole bottom position  
If you are in the incremental mode, Z refers to the distance from R point to the bottom of a hole.
- R** : R point position  
If you are in the incremental mode, R refers to the distance from the point before the control enters the canned cycle mode to R point.
- Q** : Cutting depth, shift value, and distance to feed rate switching point.  
(I) G73, G83 – Cutting depth for each stroke  
(II) G77, G78, G277, G278 – Cutting depth for each stroke  
(III) G76, G87 – Shift value  
(IV) G177, G178 – Distance to feed rate switching point  
(V) G86, G186 – Orientation angle
- P** : Dwell time (unit of time is the same as G04 designation)  
**F** : Cutting feed rate  
**S** : Spindle speed  
**K** : Canned cycle repetition count

**(Note)** Canned cycle is canceled (G80) if you command the following codes during a canned cycle operation:

- Tool change canned cycle (G100/M06)
- Changing from spindle to turning spindle (M141 to M142)
- G00 group  
(G00/G01/G02/G03/G02.2/G03.2/G102/G103/G202/G203/G33/G392)  
commands

### 5.4.6 Canned Cycle Repetition Count

You can specify the repetition count for canned cycle operation using address K when the same cycle is to be repeated such as drilling multiple holes at regular intervals.

The range of K is 0 to 9999.

K is effective only for the block in which it is specified.

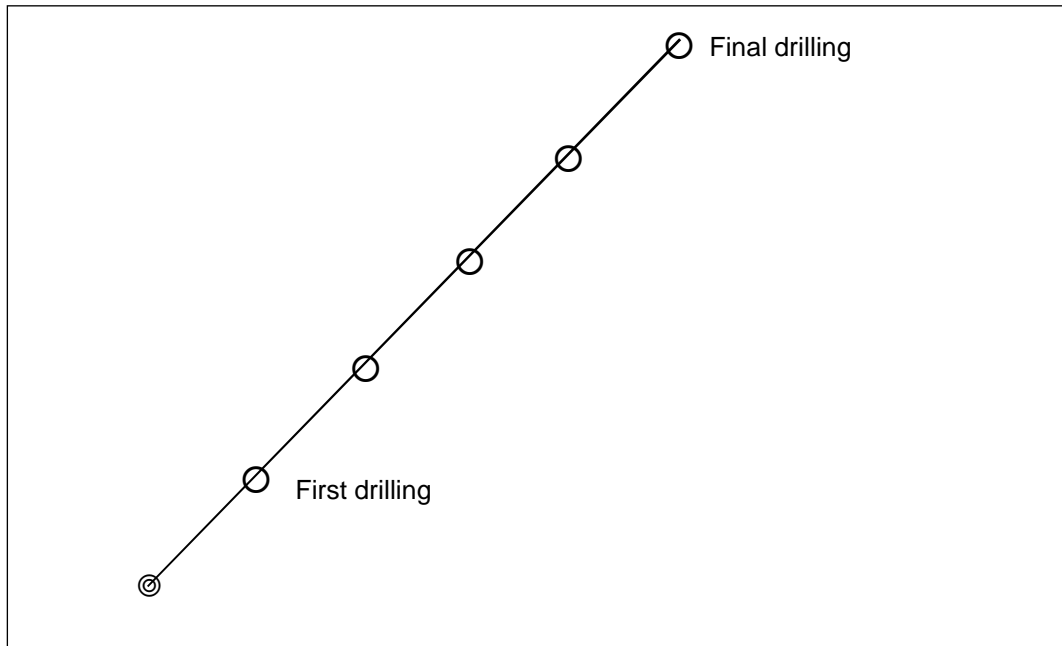
K is assumed to be 1 if a value is not specified.

When K0 is commanded, drilling will not occur; commanded hole machining data is saved; and X / Y axes will travel if the relevant commands exist.

X\_Y\_ designates the first drilling position in incremental values (G91).

If the position is given in absolute values (G90), drilling is repeated in the same position.

Ex)



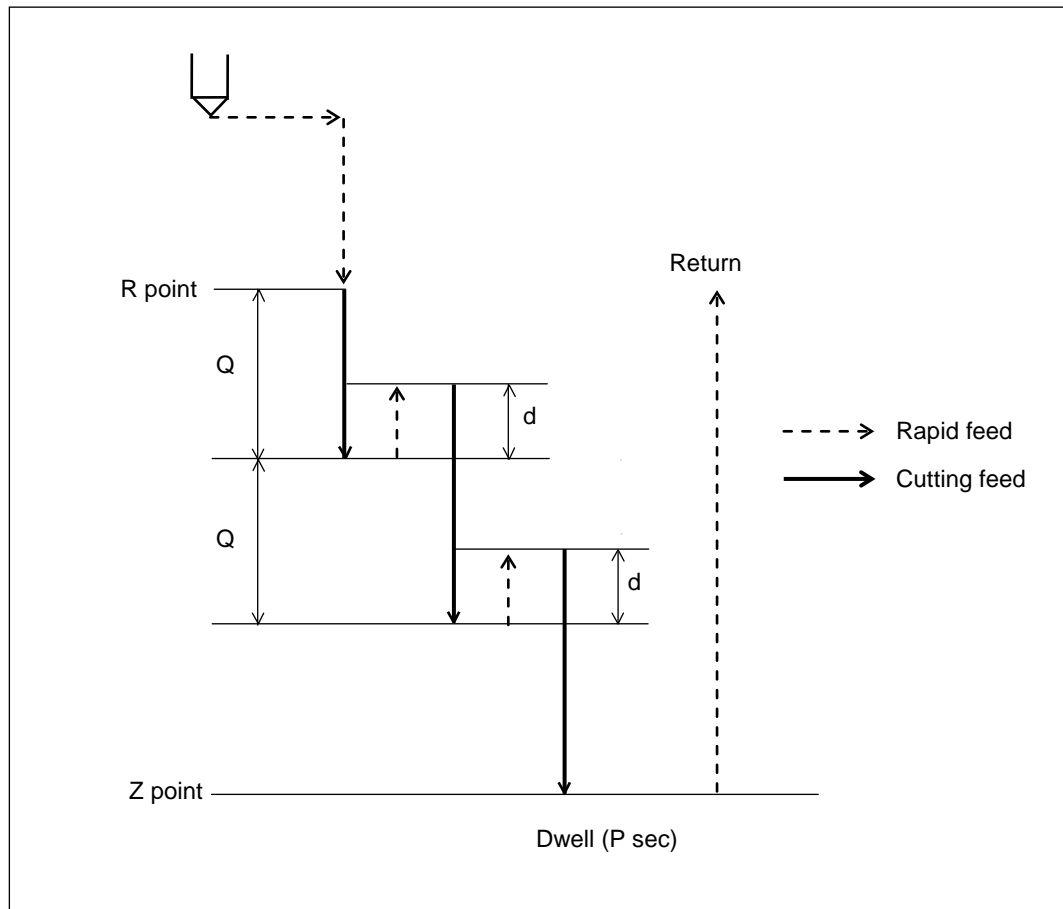
G81X\_Y\_Z\_R\_K5F\_; (G91 mode)

## 5.5 Details of Canned Cycle

### 5.5.1 High Speed Peck Drilling Cycle (G73)

Command format

G73 X\_Y\_Z\_R\_P\_Q\_F\_;

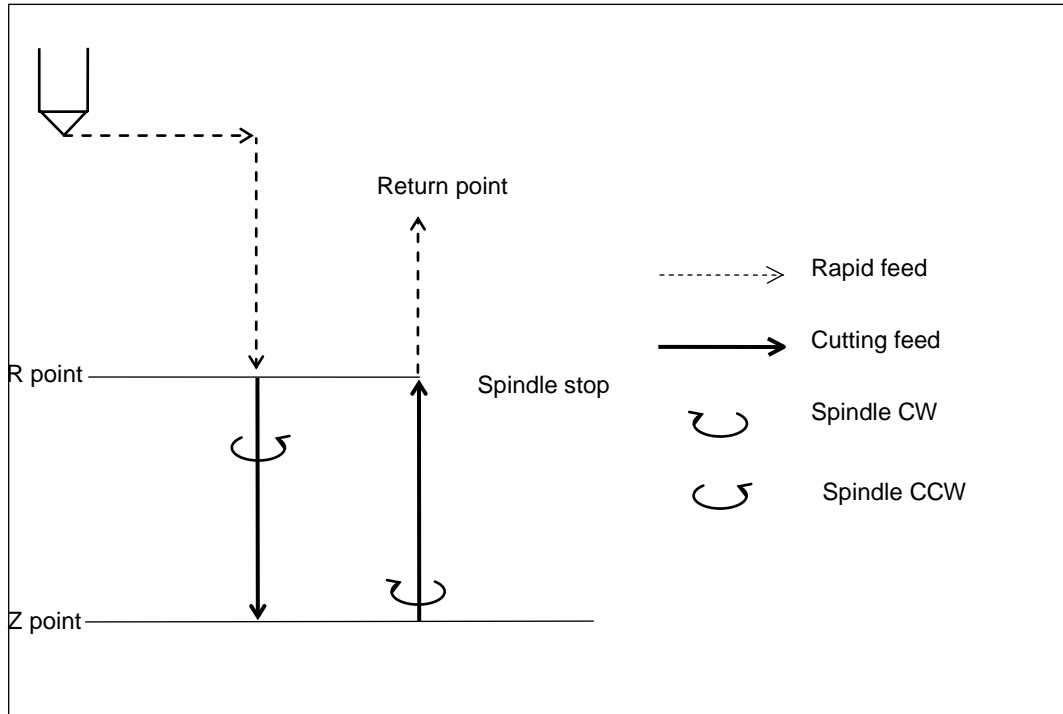


- Retract value  $d$  is set on the <G73 relief amount> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount  $Q$ , the algebraic symbol (-) is ignored.

## 5.5.2 Reverse Tapping Cycle (G74)

Command format

**G74 X\_ Y\_ Z\_ R\_ P\_ F\_ S\_;**

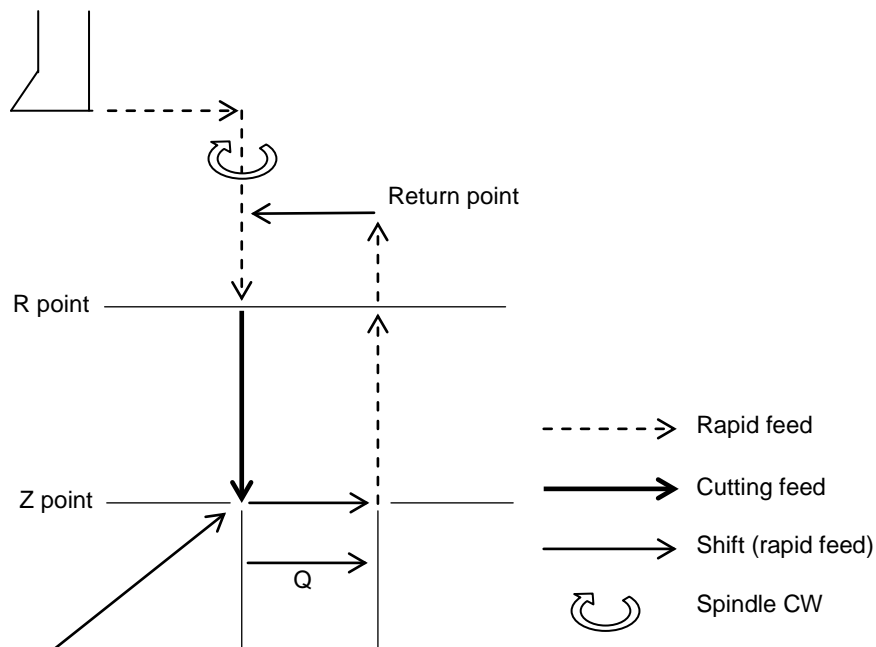


Spindle stops at Z point; dwells for P sec; and rotates clockwise.

- The tool stops on finally reaching the R point when a temporary stop is commanded at any moment en route from R to Z point and further back to the R point.
- For feed per minute (G94), thread pitch = cutting feed rate ÷ spindle speed.
- For feed per revolution (G95), thread pitch = cutting feed rate.
- The alarm <<Spindle Speed Error>> occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1)
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.
- During the tapping operation, the operation is carried out with the FEEDRATE OVERRIDE and the SPINDLE OVERRIDE at 100%.

### 5.5.3 Fine Boring Cycle (G76)

Command format

**G76 X\_ Y\_ Z\_ R\_ Q\_ V\_ P\_ F\_ S\_;**


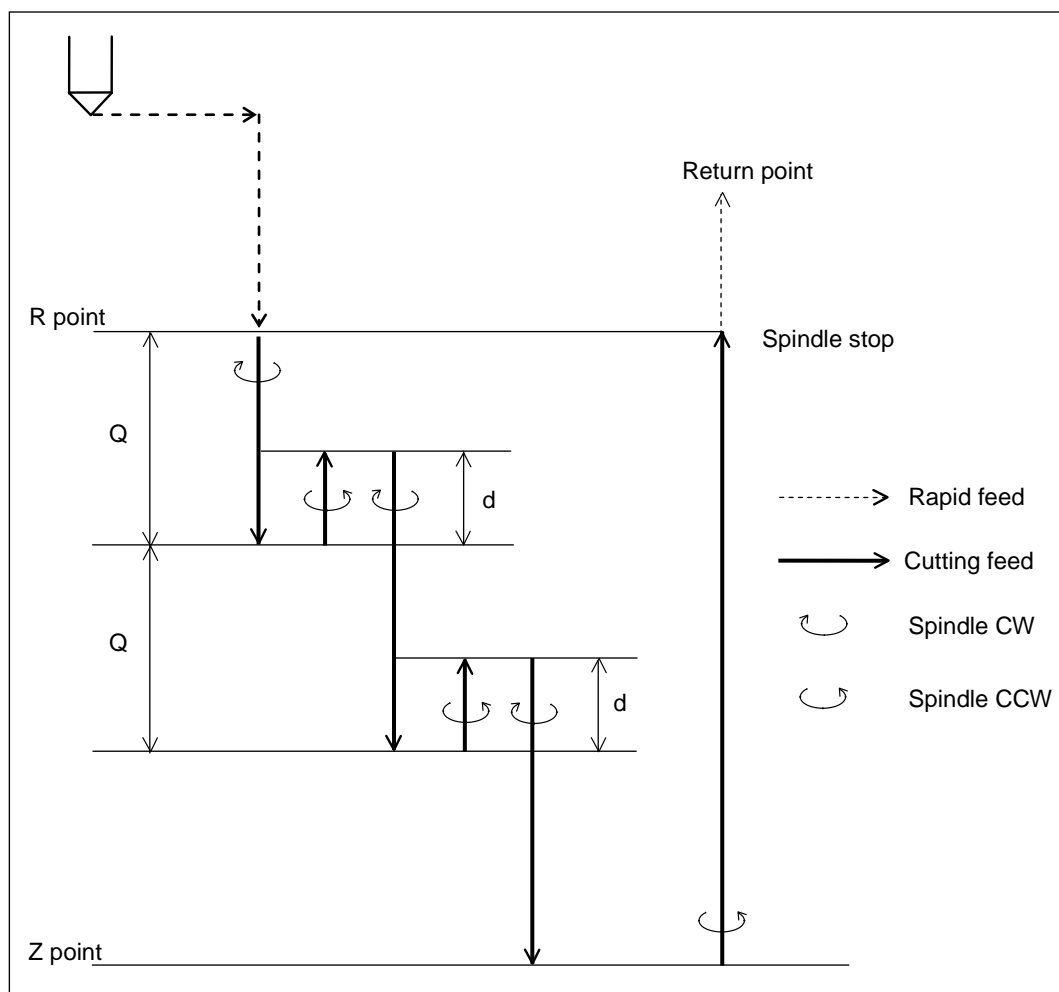
- Dwell (P sec)
- Spindle stops in V-specified rotational position

- The sign is ignored even when a negative value command is issued for the amount of shift for Q.
- Shift direction, one from among +X, -X, +Y, and -Y, is selected and set beforehand using <G76,G87 shift direction> of User parameter (Switch 1).
- Shift direction is selectable from only 4 directions of +X, -X, +Y, and -Y. Install the tool such that the bit faces one of these directions when the spindle stops at the specified rotational position.
- 0° is considered commanded when you omit V.



### 5.5.4 Tapping Cycle (Synchro Mode) (G77)

Command format

$$\text{G77 X\_Y\_Z\_R\_} \begin{pmatrix} \text{I\_} \\ \text{J\_} \end{pmatrix} \text{Q\_S\_};$$


- Retract value d is set on the <G77,G78 relief amount> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount Q, the algebraic symbol (-) is ignored.
- The tool stops on finally reaching the R point when a temporary stop is commanded at any moment en route from R to Z point and further back to the R point.
- Screw pitch or number of threads must be designated.  
Enter the value after I and J, respectively.
- When I and J exist in the same block, the former is used.
- The alarm <<Spindle Speed Error>> occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- The tool returns with the <Max. tapping speed> of Machine parameter (System 1) when <Tapping cycle return rotational frequency> of User parameter (Switch 1) is selected to <1:Max. speed.>
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.

- During the tapping operation, the operation is carried out with the FEEDRATE OVERRIDE and the SPINDLE OVERRIDE at 100%.

### High speed tap return

Spindle speed at return of tapping cycle (synchro mode) (G77) is varied.

Command format

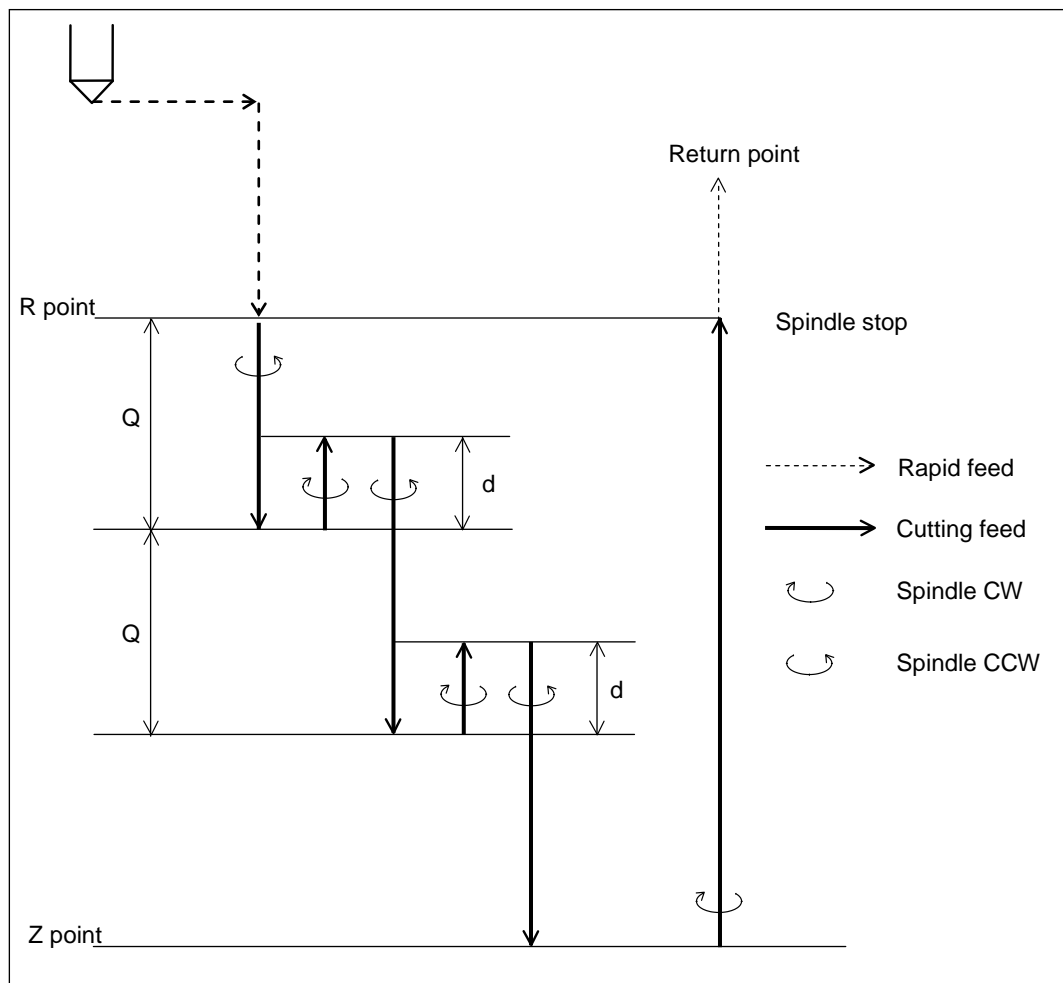
$$G77 \ X\_Y\_Z\_R\_Q\_ \begin{bmatrix} I\_ \\ J\_ \end{bmatrix} S\_L\_;$$

- Address L commands the spindle speed at return.
- Spindle speed at infeed and return is identical when address L is omitted.
- Once commanded, address L behaves in a modal manner throughout the canned cycle mode.
- The alarm <<Spindle Speed Error>> occurs if address L command value exceeds the <Max. tapping speed> of Machine parameter (System 1).
- The tool moves according to the address S value when address L value is smaller than it.
- Address L is prioritized even if <Tapping cycle return rotational frequency> of User parameter (Switch 1) is set to <1: Max. speed>.

## 5

### 5.5.5 Reverse Tapping Cycle (Synchro Mode) (G78)

Command format

$$G78 \ X\_Y\_Z\_R\_ \begin{bmatrix} I\_ \\ J\_ \end{bmatrix} Q\_S\_;$$


- Retract value  $d$  is set on the <G77,G78 relief amount> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount  $Q$ , the algebraic symbol (-) is ignored.
- The tool stops on finally reaching the R point when a temporary stop is commanded at any moment en route from R to Z point and further back to the R point.
- Screw pitch or number of threads must be designated.  
Enter the value after I and J, respectively.
- When I and J exist in the same block, the former is used.
- The alarm <<Spindle Speed Error>> occurs if  $S$  exceeds the <Max. tapping speed> of Machine parameter (System 1).
- The tool returns with the <Max. tapping speed> of Machine parameter (System 1) when <Tapping cycle return rotational frequency> of User parameter (Switch 1) is selected to <1:Max. speed.>
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.
- During the tapping operation, the operation is carried out with the FEEDRATE OVERRIDE and the SPINDLE OVERRIDE at 100%.

#### High speed tap return

Spindle speed at return of reverse tapping cycle (synchro mode) (G78) is varied.

Command format

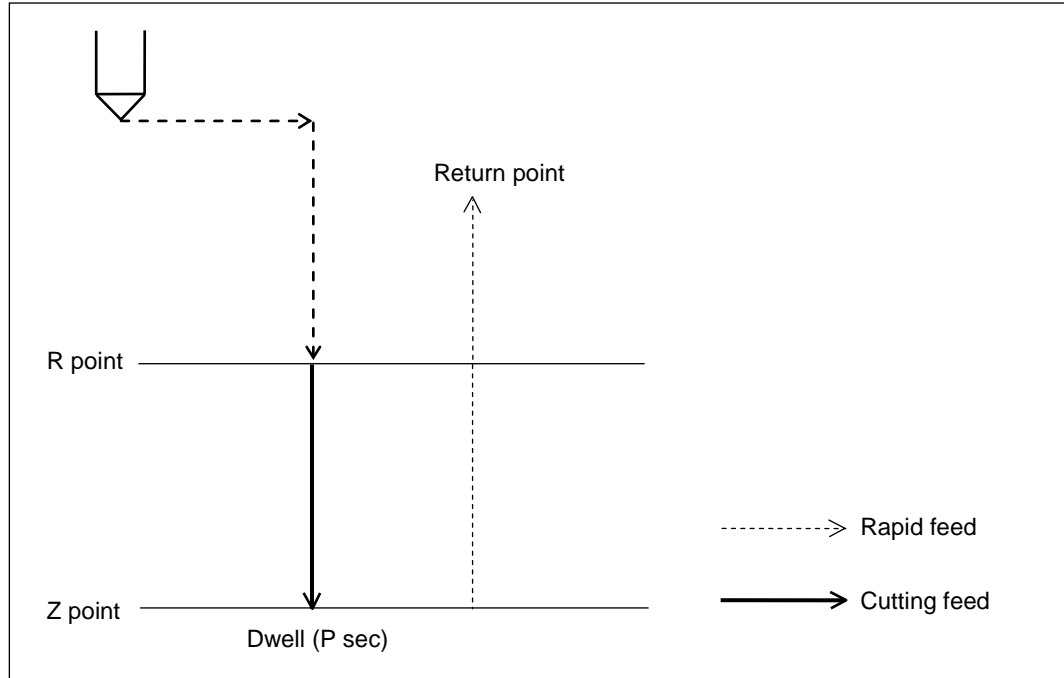
|                                                                                                                                                                                                   |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>G78 X_ Y_ Z_ R_ Q_</b> <div style="display: inline-block; vertical-align: middle; text-align: center;"> <math>\left[ \begin{array}{c} I_ \\ J_ \end{array} \right]</math> </div> <b>S_ L_;</b> |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|

- Address L commands the spindle speed at return.
- Spindle speed is the same at infeed and return if you omit address L.
- Once commanded, address L behaves in a modal manner throughout the canned cycle mode.
- The alarm <<Spindle Speed Error>> occurs if address L command value exceeds the <Max. tapping speed> of Machine parameter (System 1).
- The tool moves according to the address S value when address L value is smaller than it.
- Address L is prioritized even if <Tapping cycle return rotational frequency> of User parameter (Switch 1) is set to <1: Max. Speed>.

## 5.5.6 Drilling Cycle (G81, G82)

Command format

|                                                                                                                        |                      |
|------------------------------------------------------------------------------------------------------------------------|----------------------|
| <div style="display: inline-block; vertical-align: middle; text-align: center;"> <b>G81</b><br/><br/><b>G82</b> </div> | <b>X_Y_Z_R_P_F_;</b> |
|------------------------------------------------------------------------------------------------------------------------|----------------------|



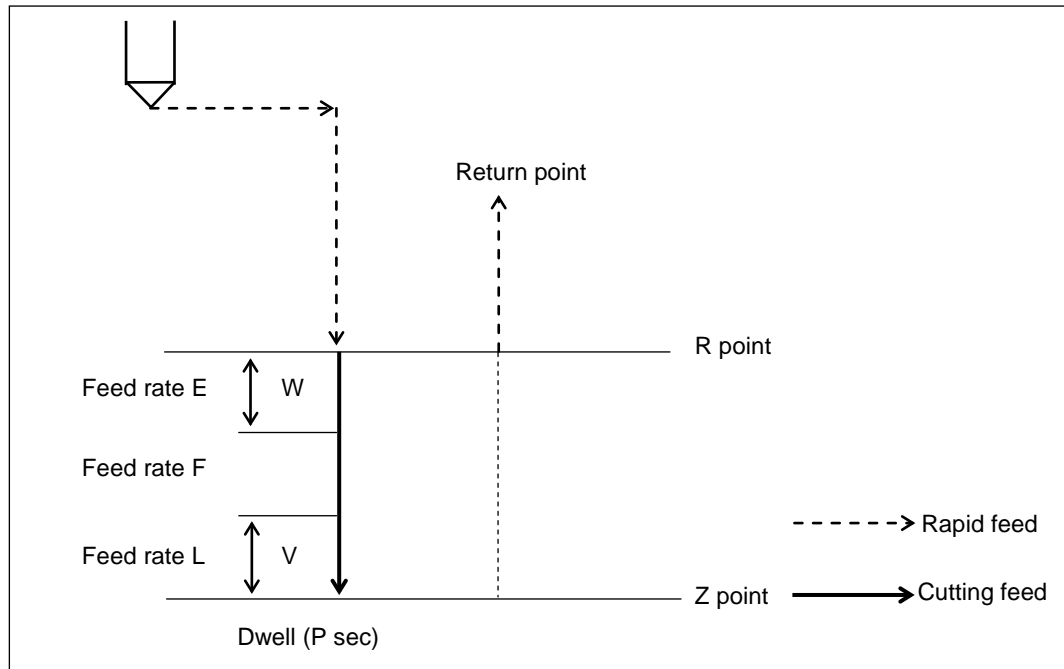
### High speed cycle

Feed rate at start and end of cutting in drilling cycle (G81 or G82) is varied.

Command format

|                                                                                                                        |                              |
|------------------------------------------------------------------------------------------------------------------------|------------------------------|
| <div style="display: inline-block; vertical-align: middle; text-align: center;"> <b>G81</b><br/><br/><b>G82</b> </div> | <b>X_Y_Z_R_P_W_V_F_E_L_;</b> |
|------------------------------------------------------------------------------------------------------------------------|------------------------------|

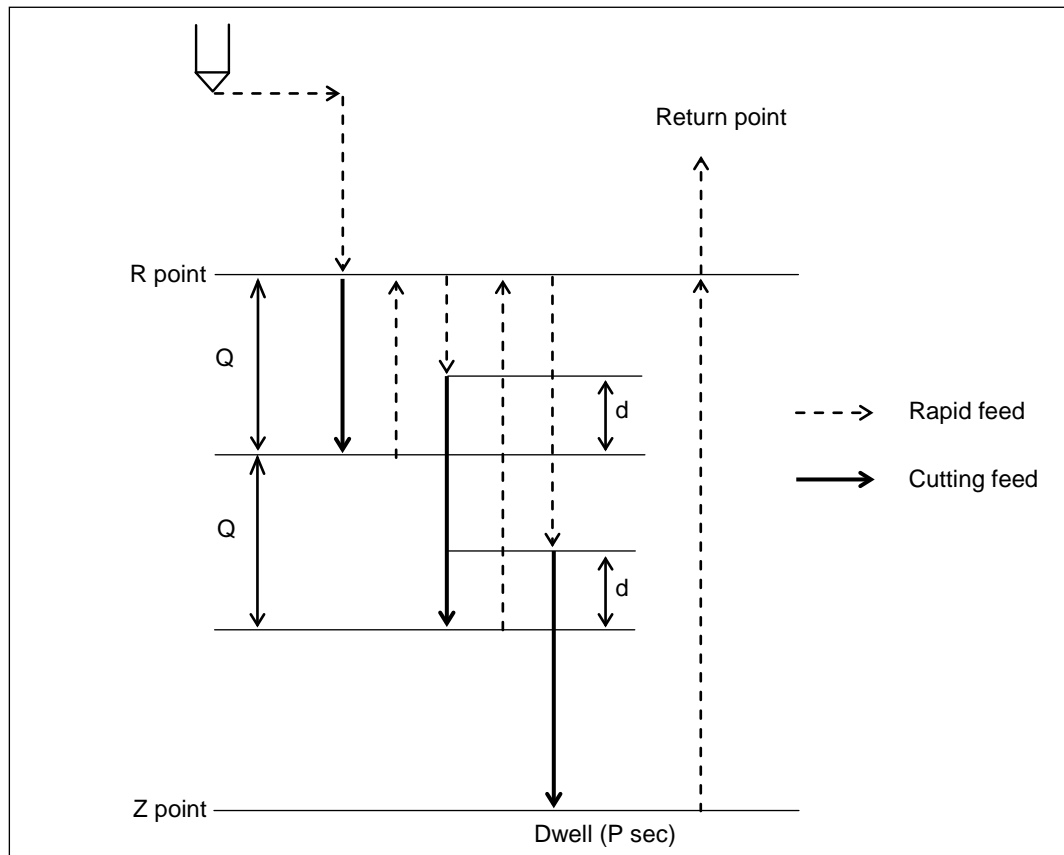
- W : Feed rate switching point  
Distance from R point irrespective of absolute (G90) or incremental (G91) mode.
- V : Feed rate switching point  
Distance from Z point irrespective of absolute (G90) or incremental (G91) mode.
- E : Feed rate for range W from R point
- L : Feed rate for range V from Z point



### 5.5.7 Peck Drilling Cycle (G83)

Command format

**G83 X\_ Y\_ Z\_ R\_ P\_ Q\_ F\_;**

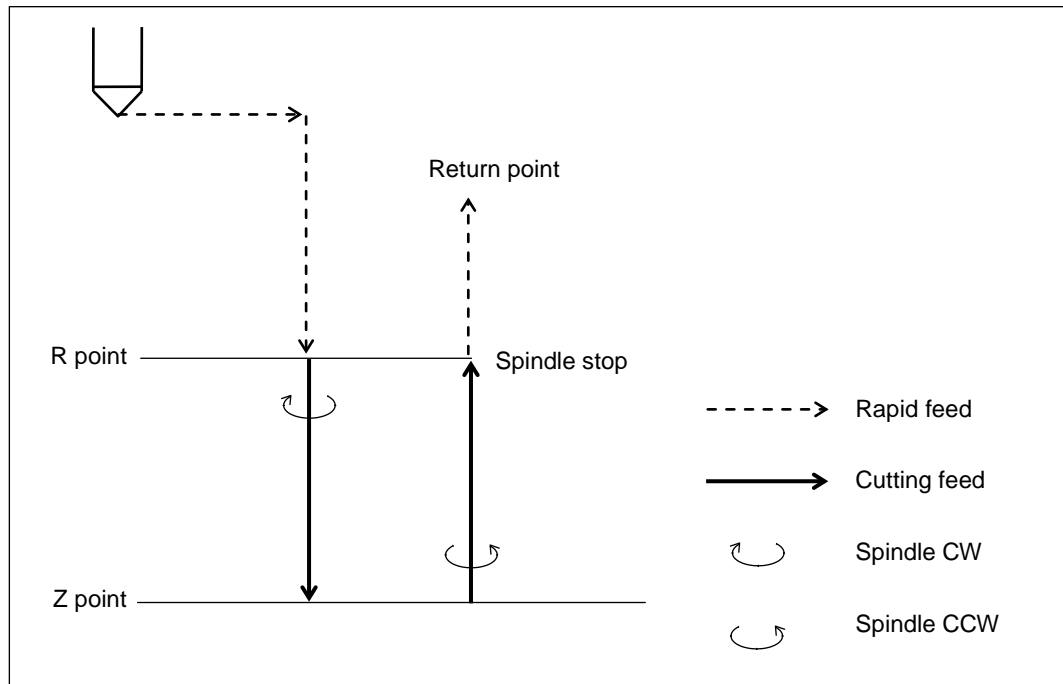


- Cutting start point d is set on the <G83 cutting start position> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount Q, the algebraic symbol (-) is ignored.

## 5.5.8 Tapping Cycle (G84)

Command format

**G84 X\_ Y\_ Z\_ R\_ P\_ F\_ S\_;**



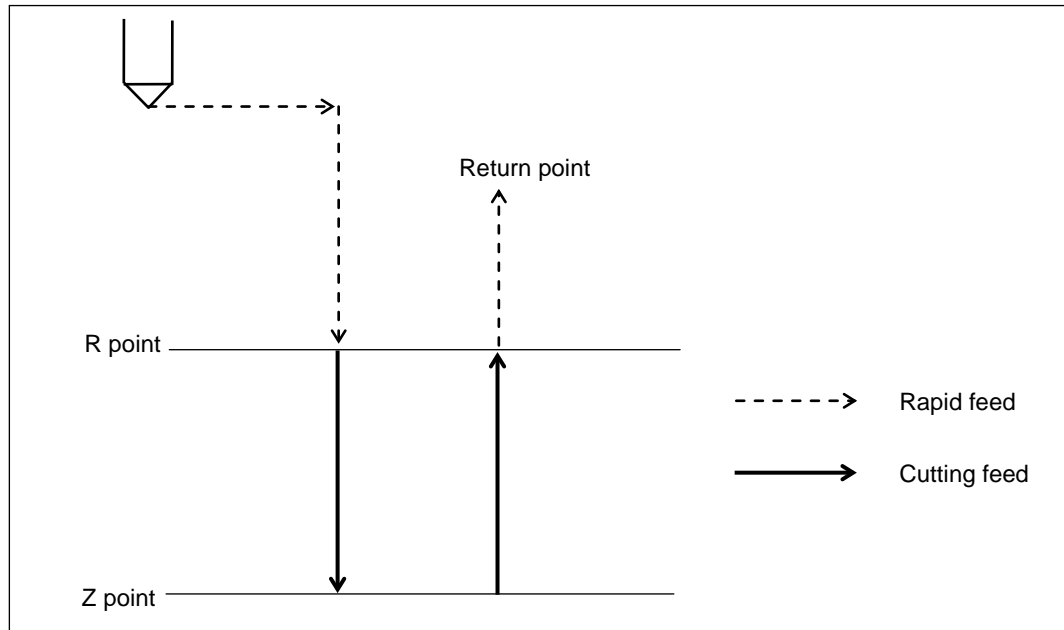
The spindle stops at Z point, dwells for P sec, and then rotates reversely.

- The tool stops on finally reaching the R point when a temporary stop is commanded at any moment en route from R to Z point and further back to the R point.
- For feed per minute (G94), thread pitch = cutting feed rate ÷ spindle speed.
- For feed per revolution (G95), thread pitch = cutting feed rate.
- The alarm <<Spindle Speed Error>> occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.
- During the tapping operation, the operation is carried out with the FEEDRATE OVERRIDE and the SPINDLE OVERRIDE at 100%.

### 5.5.9 Boring Cycle (G85, G89)

Command format

|            |                      |
|------------|----------------------|
| <b>G85</b> | <b>X_Y_Z_R_P_F_;</b> |
| <b>G89</b> |                      |



#### High speed cycle

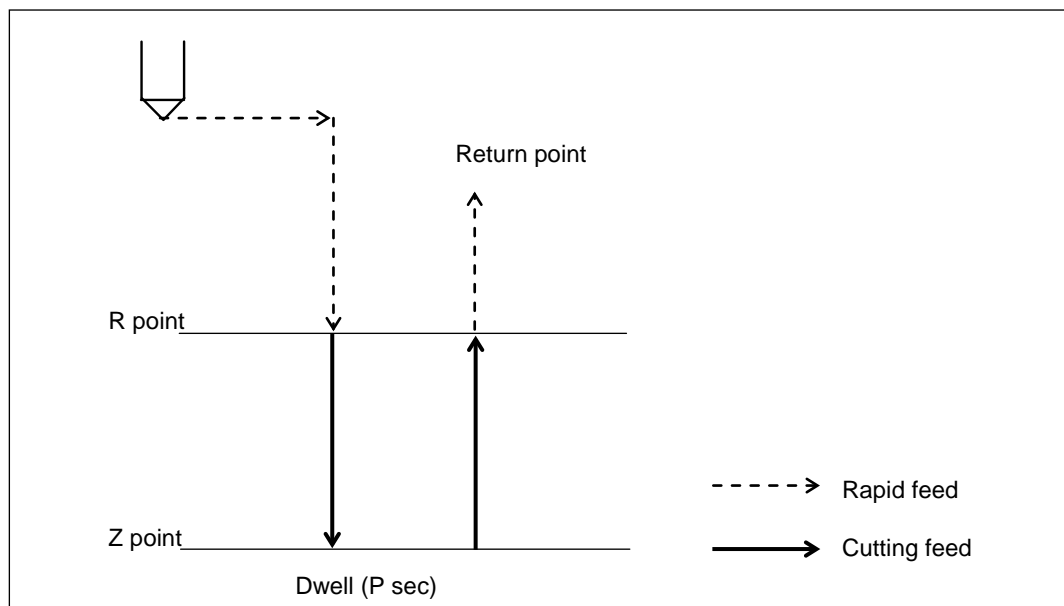
Feed rate at return of boring cycle (G85 or G89) is varied.

Command format

|            |                        |
|------------|------------------------|
| <b>G85</b> | <b>X_Y_Z_R_P_F_E_;</b> |
| <b>G89</b> |                        |

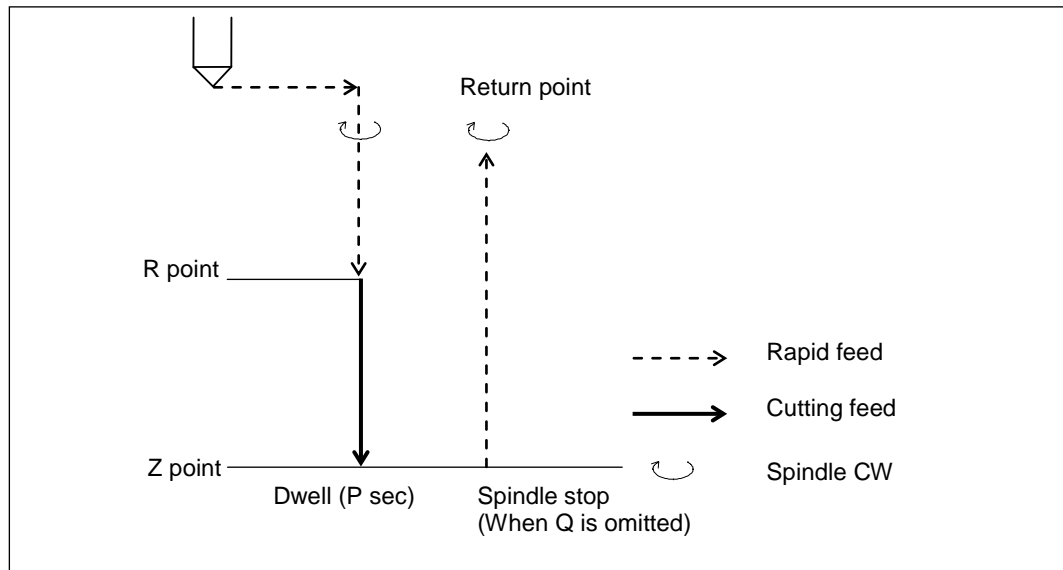
F : Feed rate from R to Z point

E : Feed rate from Z to R point



### 5.5.10 Boring Cycle (G86)

Command format

**G86 X\_ Y\_ Z\_ R\_ P\_ Q\_ F\_ S\_;**


#### High speed cycle

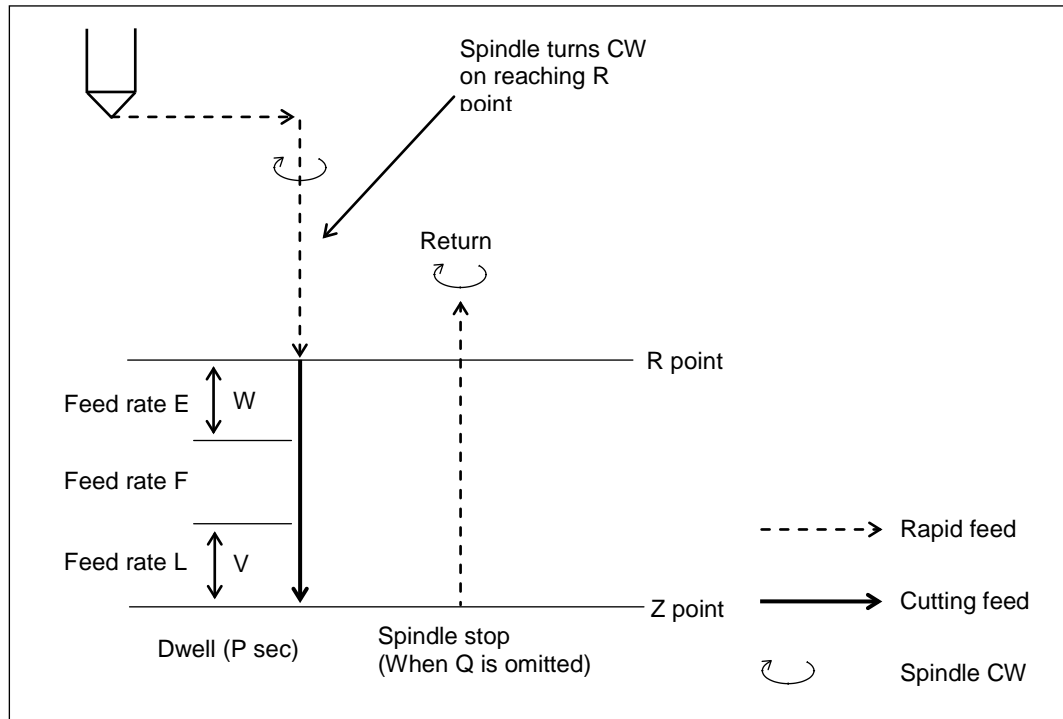
Feed rate at start and end of cutting in boring cycle (G86) is varied.

Command format

**G86 X\_ Y\_ Z\_ R\_ P\_ W\_ V\_ Q\_ F\_ E\_ L\_ S\_;**

- W : Feed rate switching point  
Distance from R point irrespective of absolute (G90) or incremental (G91) mode.
- E : Feed rate for range W from R point
- V : Feed rate switching point  
Distance from Z point irrespective of absolute (G90) or incremental (G91) mode.
- L : Feed rate for range V from Z point
- Q : To stop the spindle at a specified rotational position at Z point, specify the desired angle. If the entry is omitted, the spindle just stops.

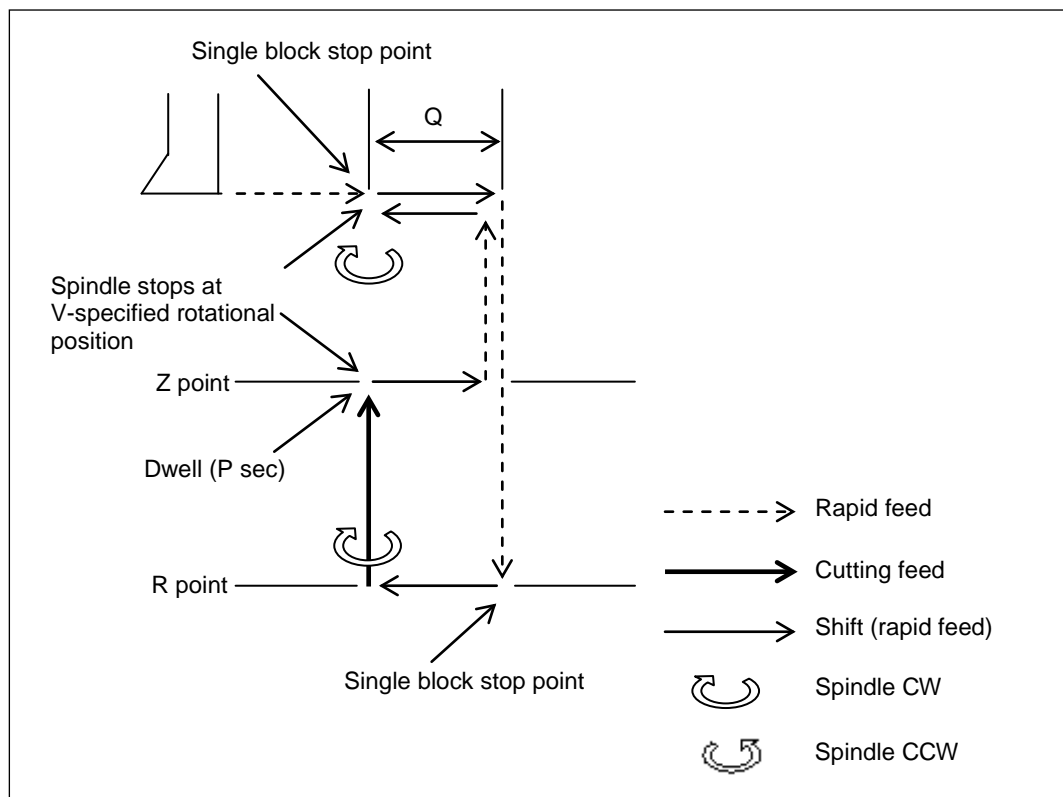




### 5.5.11 Back Boring Cycle (G87)

Command format

**G87 X\_Y\_Z\_R\_Q\_P\_V\_F\_S\_;**



- The sign is ignored even when a negative value command is issued for the amount of shift for Q.
- Shift direction, one from among +X, -X, +Y, and -Y, is selected and set beforehand using <G76,G87 shift direction> of User parameter (Switch 1).

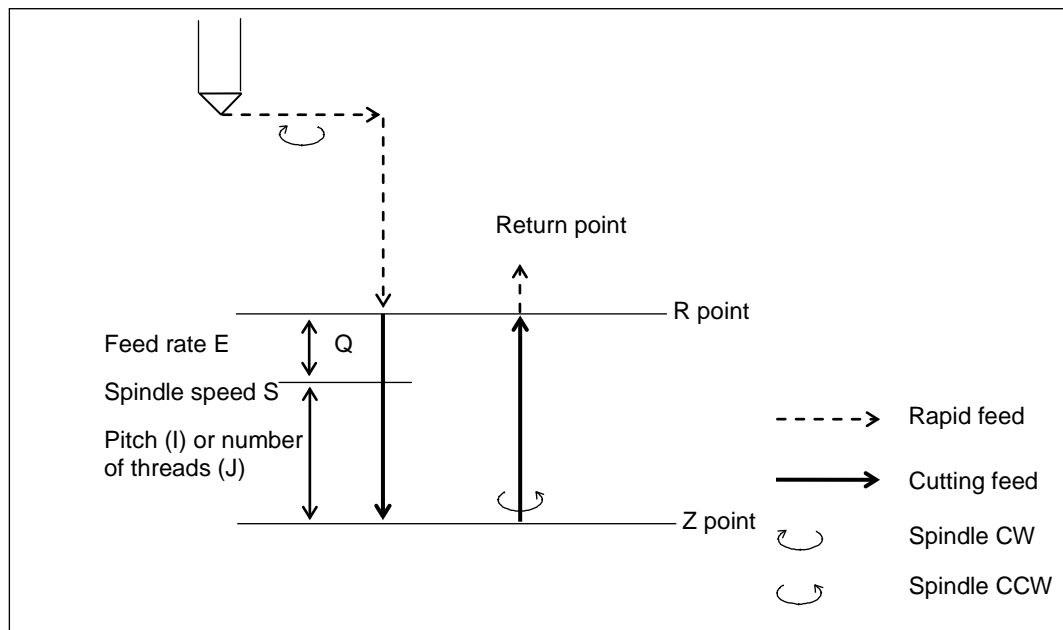
- Shift direction is selectable from only 4 directions of +X, -X, +Y, and -Y. Install the tool such that the bit faces one of these directions when the spindle stops at the specified rotational position.
- G99 (Return to R point level) does not exist.
- 0° is considered commanded when you omit V.

### 5.5.12 End Milling/Tapping Cycle (G177)

Command format

**G177 X\_ Y\_ Z\_ R\_  $\begin{bmatrix} I_ \\ J_ \end{bmatrix}$  S\_ L\_ Q\_ E\_;**

- I : Screw pitch in the tapping section
- J : Number of threads in the tapping section
- S : Spindle speed  
The spindle starts rotation simultaneously with the X/Y axes traveling
- L : Spindle return speed from Z to R point  
S-specified speed applies in the absence of the command
- Q : Feed rate switching point  
Distance from R point irrespective of absolute (G90) or incremental (G91) mode.  
Tapping starts from this position
- E : Feed rate in section Q



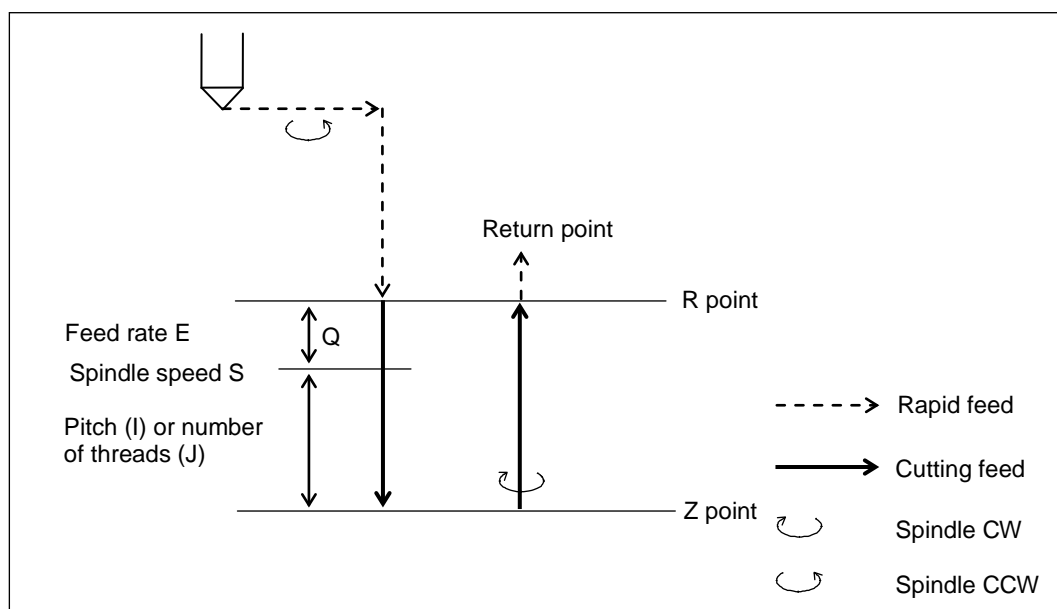
- The alarm <<Spindle Speed Error>> occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.

### 5.5.13 End Milling/Tapping Cycle (G178)

Command format

|                                                                                                      |
|------------------------------------------------------------------------------------------------------|
| <b>G178 X_ Y_ Z_ R_</b> $\left[ \begin{array}{c} I\_ \\ J\_ \end{array} \right]$ <b>S_ L_ Q_ E_;</b> |
|------------------------------------------------------------------------------------------------------|

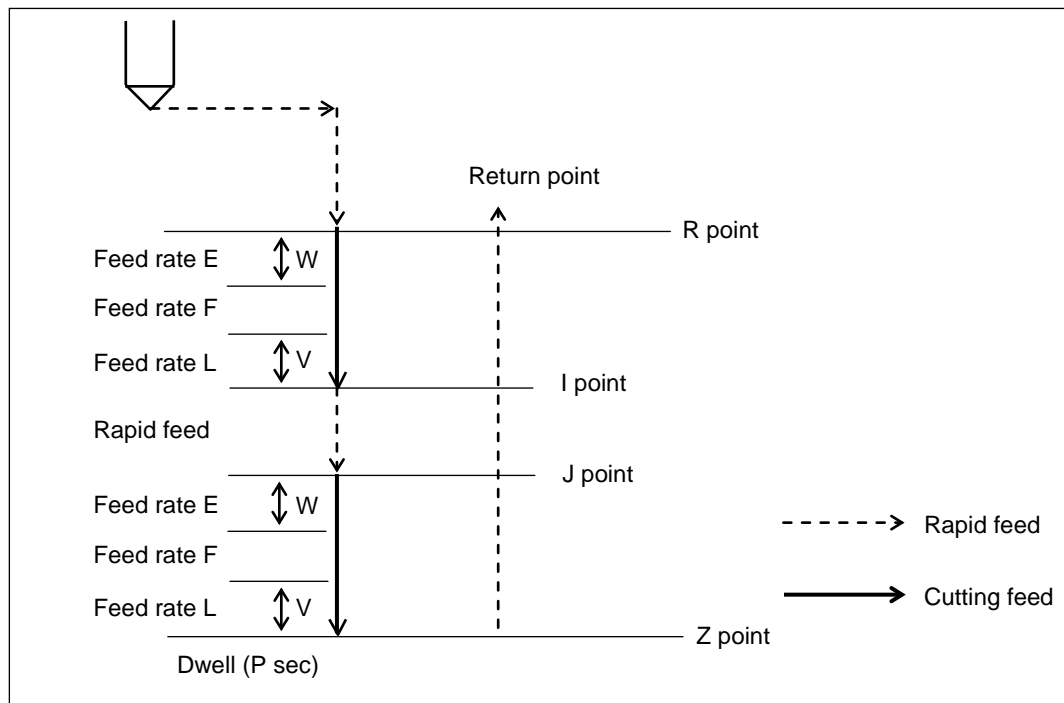
- I** : Screw pitch in the tapping section  
**J** : Number of threads in the tapping section  
**S** : Spindle speed  
       The spindle starts rotation simultaneously with the X/Y axes traveling  
**L** : Spindle return speed from Z to R point  
       S-specified speed applies in the absence of the command  
**Q** : Feed rate switching point  
       Distance from R point irrespective of absolute (G90) or incremental (G91) mode  
       Tapping starts from this position  
**E** : Feed rate in section Q



- The alarm <<Spindle Speed Error>> occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.

## Command format

|   |   |                                                                                                |
|---|---|------------------------------------------------------------------------------------------------|
| I | : | Rapid feed start point (depends on G90/G91)<br>Distance from R point in the incremental mode   |
| J | : | Cutting feed start point (depends on G90/G91)<br>Distance from I point in the incremental mode |
| W | : | Feed rate switching point<br>Incremental mode irrespective of G90 or G91                       |
| V | : | Feed rate switching point<br>Incremental mode irrespective of G90 or G91                       |
| E | : | Feed rate for range W                                                                          |
| L | : | Feed rate for range V                                                                          |

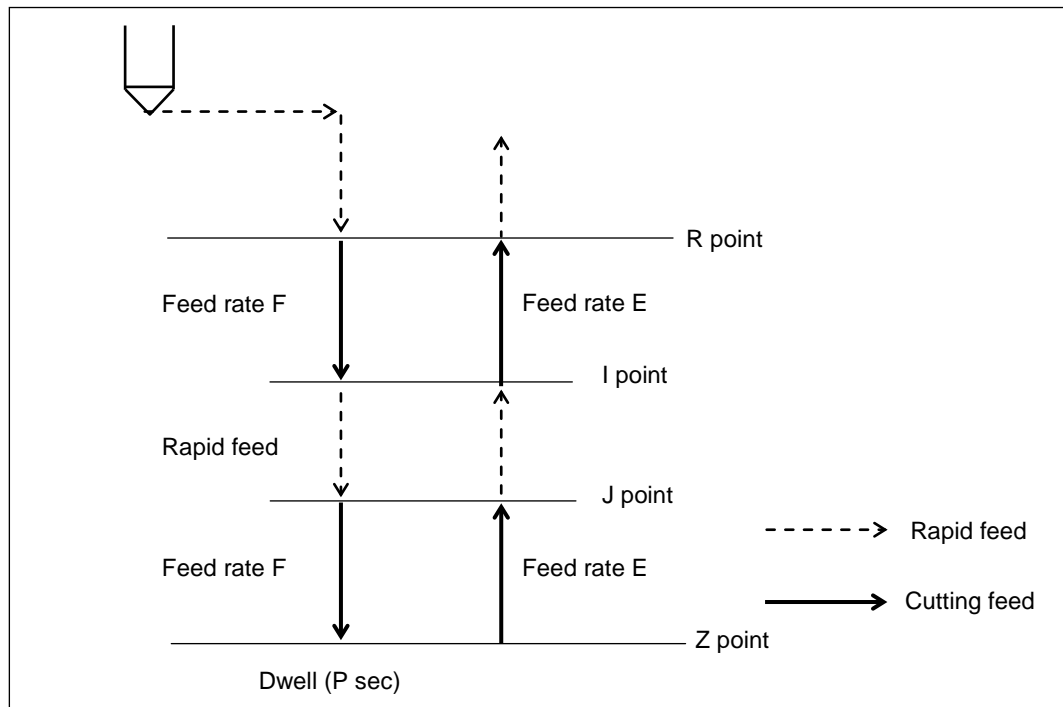


### 5.5.15 Double Boring Cycle (G185, G189)

Command format

|      |                     |
|------|---------------------|
| G185 | X_Y_Z_R_I_J_P_F_E_; |
| G189 |                     |

- I : Rapid feed start point (depends on G90/G91)  
Distance from R point in the incremental mode
- J : Cutting feed start point (depends on G90/G91)  
Distance from I point in the incremental mode
- F : Cutting feed rate from R to Z point
- E : Cutting feed rate from Z to R point

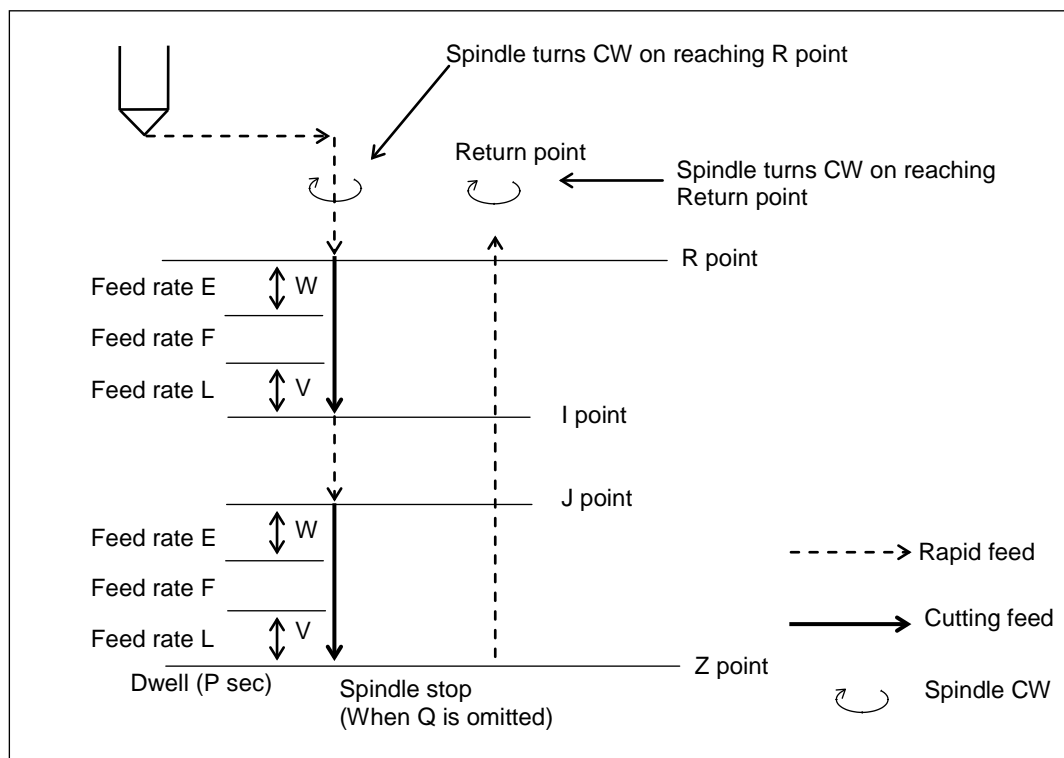


### 5.5.16 Double Boring Cycle (G186)

Command format

**G186 X\_ Y\_ Z\_ R\_ I\_ J\_ P\_ W\_ V\_ Q\_ F\_ E\_ L\_ S\_;**

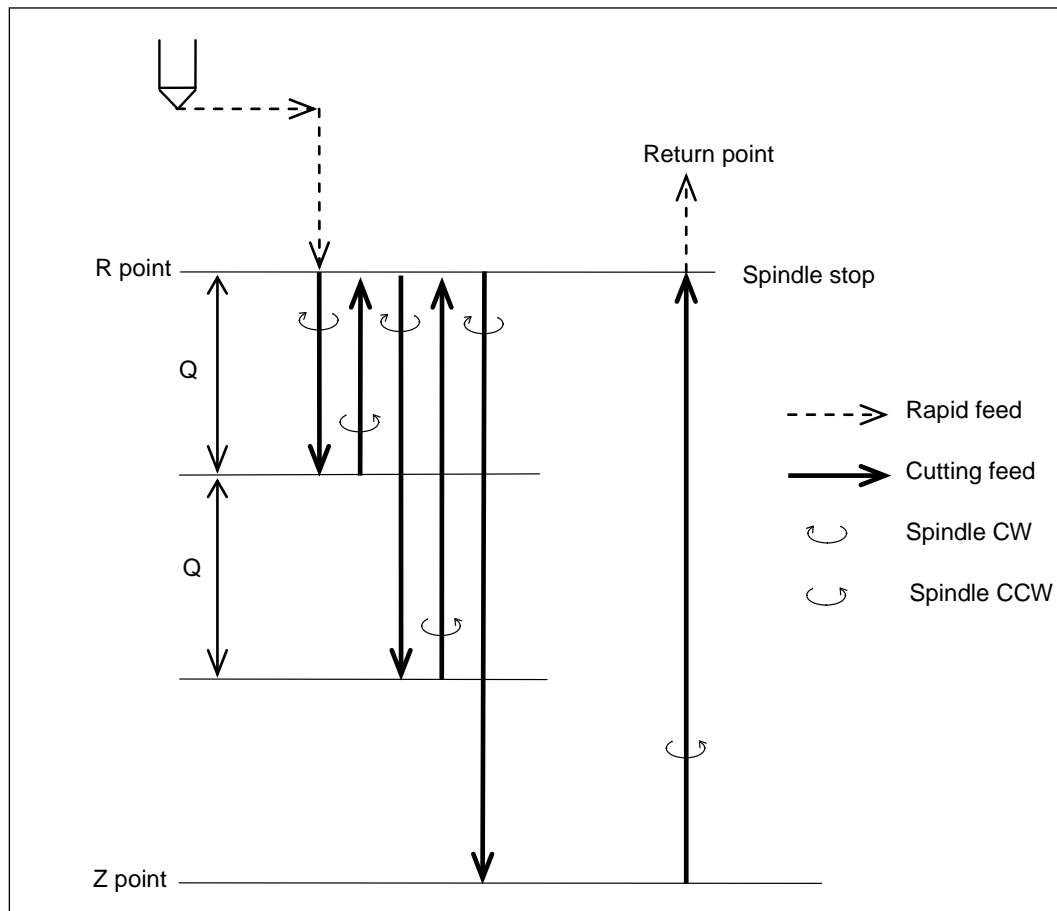
- I : Rapid feed start point (depends on G90/G91)  
Distance from R point in the incremental mode
- J : Cutting feed start point (depends on G90/G91)  
Distance from I point in the incremental mode
- W : Feed rate switching point  
Incremental mode irrespective of G90 or G91
- V : Feed rate switching point  
Incremental mode irrespective of G90 or G91
- Q : To stop the spindle at a specified rotational position at Z point, specify the desired angle. If the entry is omitted, the spindle just stops.
- E : Feed rate for range W
- L : Feed rate for range V



### 5.5.17 Deep Hole Tapping Cycle (Synchro Mode) (G277)

Command format

|                                                                                                |
|------------------------------------------------------------------------------------------------|
| <b>G277 X_ Y_ Z_ R_</b> $\left[ \begin{array}{c} I\_ \\ J\_ \end{array} \right]$ <b>Q_ S_;</b> |
|------------------------------------------------------------------------------------------------|

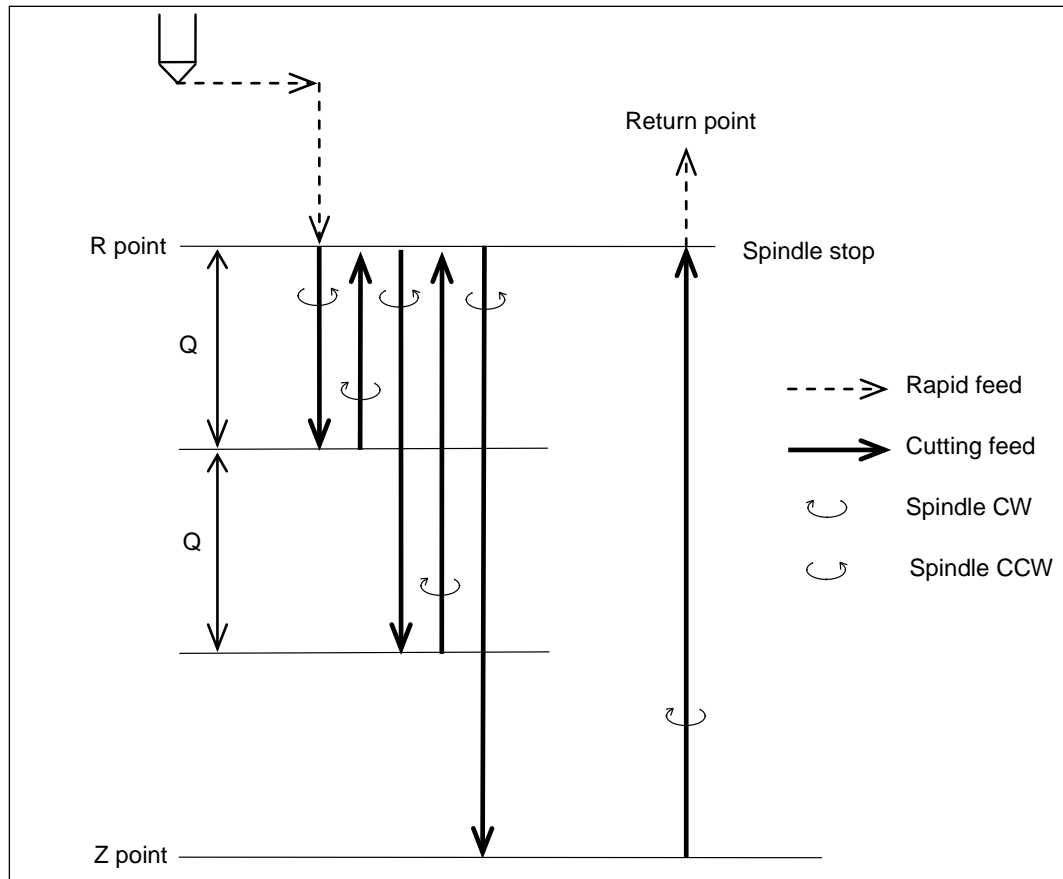


- If a negative value is entered for the cutting amount  $Q$ , the algebraic symbol (-) is ignored.
- When a temporary stop is commanded during cutting, the control stops on returning from the hole bottom position  $Z$  to the  $R$  point.
- Screw pitch or number of threads must be designated.  
Enter the value after  $I$  and  $J$ , respectively.
- When  $I$  and  $J$  exist in the same block, the former is used.
- An alarm occurs if  $S$  exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When <Tapping cycle return rotational frequency> of User parameter (Switch 1) is set to <1: Max. speed>, the tool returns with <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the  $Z$ -axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.
- During the tapping operation, the operation is carried out with the FEEDRATE OVERRIDE and the SPINDLE OVERRIDE at 100%.

### 5.5.18 Reverse Deep Hole Tapping Cycle (Synchro Mode) (G278)

Command format

|                  |                                          |        |
|------------------|------------------------------------------|--------|
| G278 X_ Y_ Z_ R_ | $\begin{bmatrix} I_ \\ J_ \end{bmatrix}$ | Q_ S_; |
|------------------|------------------------------------------|--------|



- If a negative value is entered for the cutting amount Q, the algebraic symbol (-) is ignored.
- When a temporary stop is commanded during cutting, the control stops on returning from the hole bottom position Z to the R point.
- Screw pitch or number of threads must be designated.  
Enter the value after I and J, respectively.
- When I and J exist in the same block, the former is used.
- An alarm occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1)
- When <Tapping cycle return rotational frequency> of User parameter (Switch 1) is set to <1: Max. speed>, the tool returns with <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the screw pitch is less than the <Minimum tapping pitch> in the Machine parameter (System 1), the alarm <<Pitch data error>> is triggered.
- During the tapping operation, the operation is carried out with the FEEDRATE OVERRIDE and the SPINDLE OVERRIDE at 100%.



High speed tap return

Spindle speed at return of synchro tap (G277, G278) is varied.

Command format

$$\begin{bmatrix} \text{G277} \\ \text{G278} \end{bmatrix} X\_Y\_Z\_R\_ \begin{bmatrix} I\_ \\ J\_ \end{bmatrix} Q\_S\_L\_;$$

- Address L commands spindle speed at return.
- Spindle speed at infeed and return is identical when address L is omitted.
- Once commanded, address L behaves in a modal manner throughout the canned cycle mode.
- An alarm occurs if address L command value exceeds the maximum tap speed.
- The tool moves according to the address S value when address L value is smaller than it.
- Address L is prioritized even if <Tapping cycle return rotational frequency> of User parameter (Switch 1) is set to <1: Max. Speed>.

### 5.5.19 Reducing Step of Canned Cycle

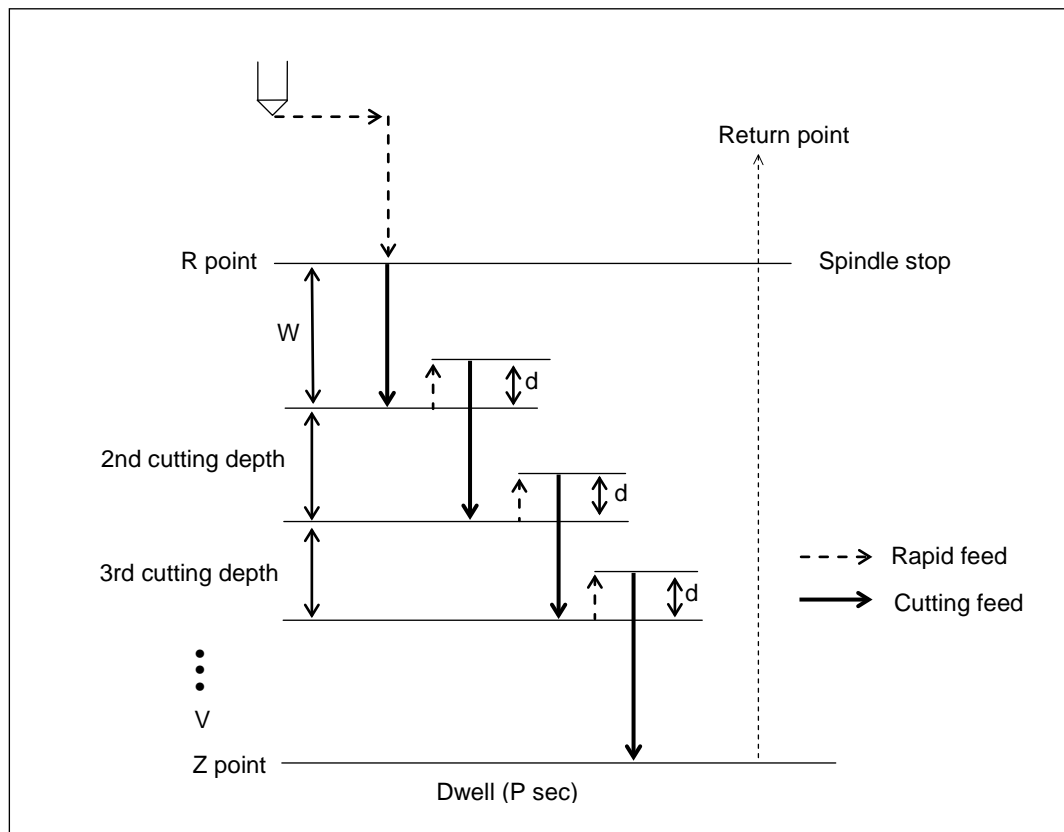
Cutting depth is gradually reduced in canned cycles of G73, G77, G78, G83, G173, and G183.

#### 1. High speed peck drilling cycle (G73) (reducing step)

Command format

**G73 X\_ Y\_ Z\_ R\_ P\_ W\_ V\_ F\_;**

W : 1st cutting feed  
V : Minimum cutting feed



- Retract value d is set on the <G73 relief amount> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

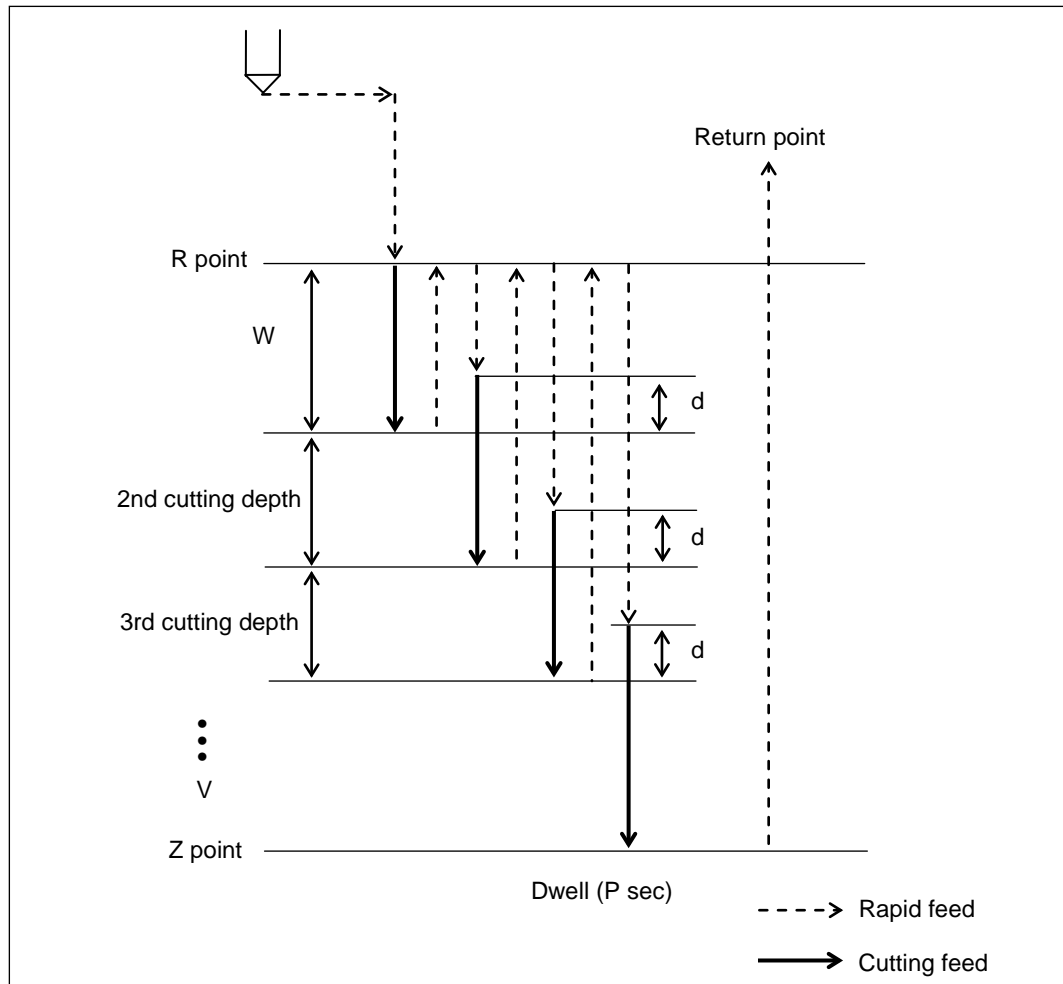
## 2. Peck drilling cycle (G83) (reducing step)

Command format

G83 X\_ Y\_ Z\_ R\_ P\_ W\_ V\_ F\_;

W : 1st cutting feed

V : Minimum cutting feed



- Cutting start point d is set on the <G83 cutting start position> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

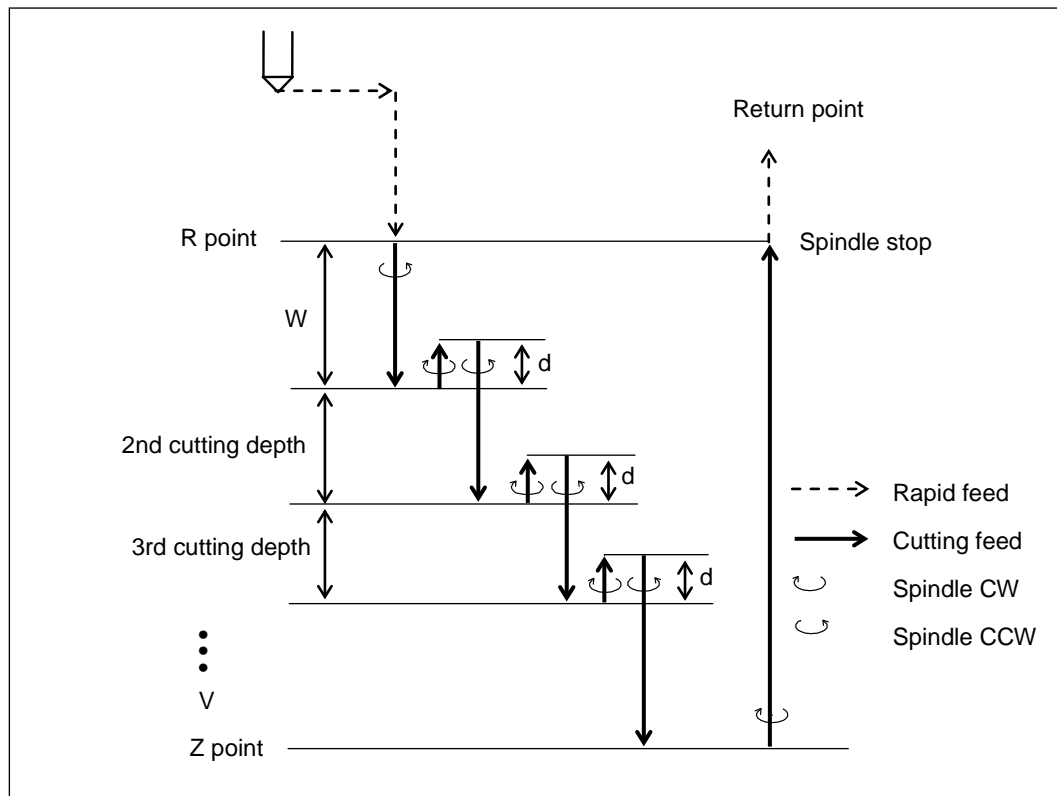
### 3. Tapping cycle (synchro mode) (G77) (reducing step)

Command format

|                                                                                    |
|------------------------------------------------------------------------------------|
| <b>G77 X_ Y_ Z_ R_</b> $\begin{bmatrix} I\_ \\ J\_ \end{bmatrix}$ <b>W_ V_ S_;</b> |
|------------------------------------------------------------------------------------|

W : 1st cutting feed

V : Minimum cutting feed



- Retract value d is set on the <G77,G78 relief amount> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.
- The tool stops on finally reaching the R point when a temporary stop is commanded at any moment en route from R to Z point and further back to the return point.
- Screw pitch or number of threads must be designated.  
Enter the value after I and J, respectively.
- When I and J exist in the same block, the former is used.
- The alarm <<Spindle Speed Error>> occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

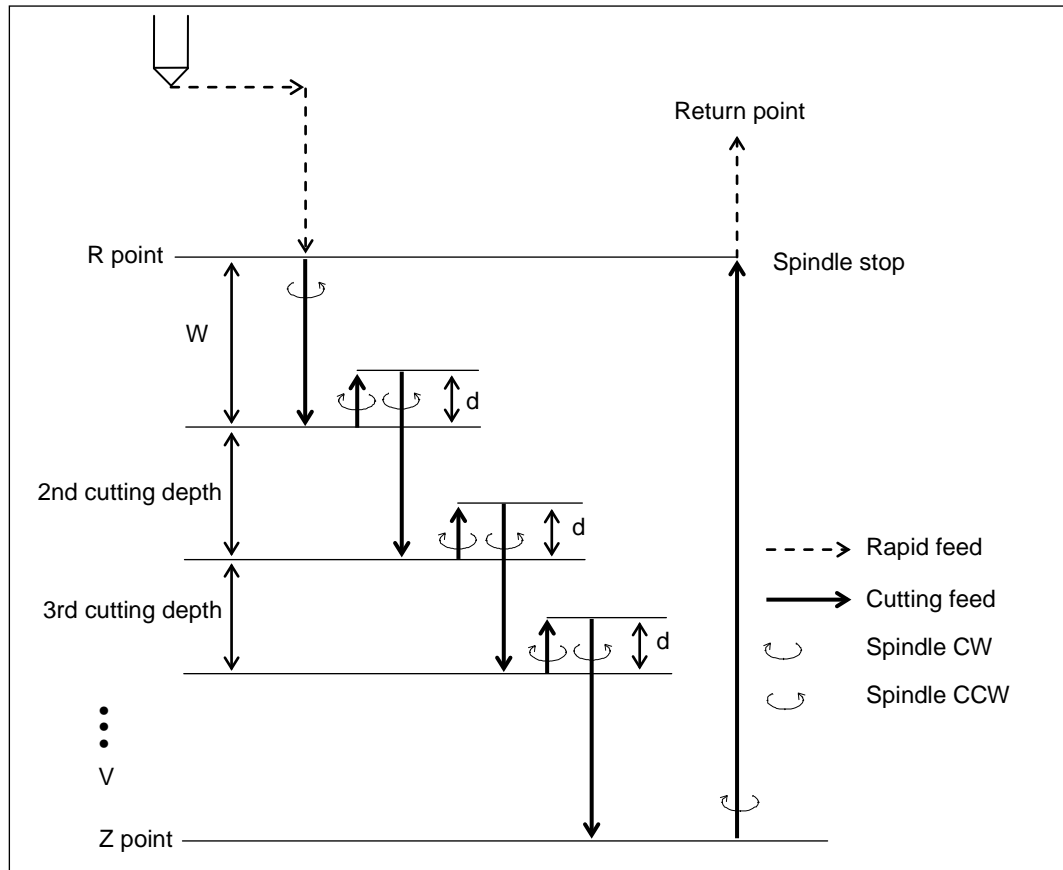
## 4. Reverse tapping cycle (synchro mode) (G78) (reducing step)

Command format

$$G78 \ X\_Y\_Z\_R\_ \left[ \begin{array}{c} I\_ \\ J\_ \end{array} \right] W\_V\_S\_;$$

W : 1st cutting feed

V : Minimum cutting feed



- Retract value d is set on the <G77,G78 relief amount> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.
- The tool stops on finally reaching the R point when a temporary stop is commanded at any moment en route from R to Z point and further back to the R point.
- Screw pitch or number of threads must be designated.  
Enter the value after I and J, respectively.
- When I and J exist in the same block, the former is used.
- The alarm <<Spindle Speed Error>> occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

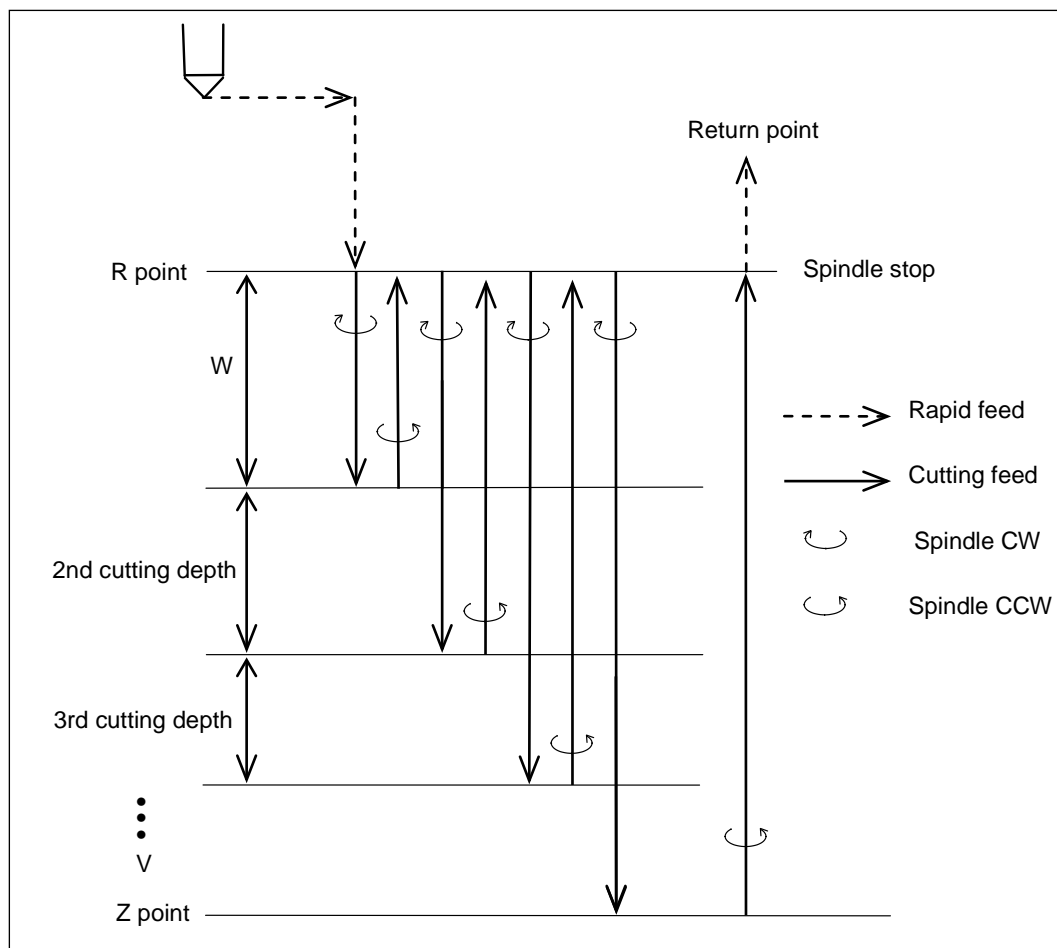
## 5. Deep hole tapping cycle (synchro mode) (G277) (reducing step)

Command format

|                                                                                     |
|-------------------------------------------------------------------------------------|
| <b>G277 X_ Y_ Z_ R_</b> $\begin{bmatrix} I\_ \\ J\_ \end{bmatrix}$ <b>W_ V_ S_;</b> |
|-------------------------------------------------------------------------------------|

W : 1st cutting feed

V : Minimum cutting depth



- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.
- When a temporary stop is commanded during cutting, the control stops on returning from the hole bottom position Z to the R point.
- Screw pitch or number of threads must be designated.  
Enter the value after I and J, respectively.
- When I and J exist in the same block, the former is used.
- An alarm occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

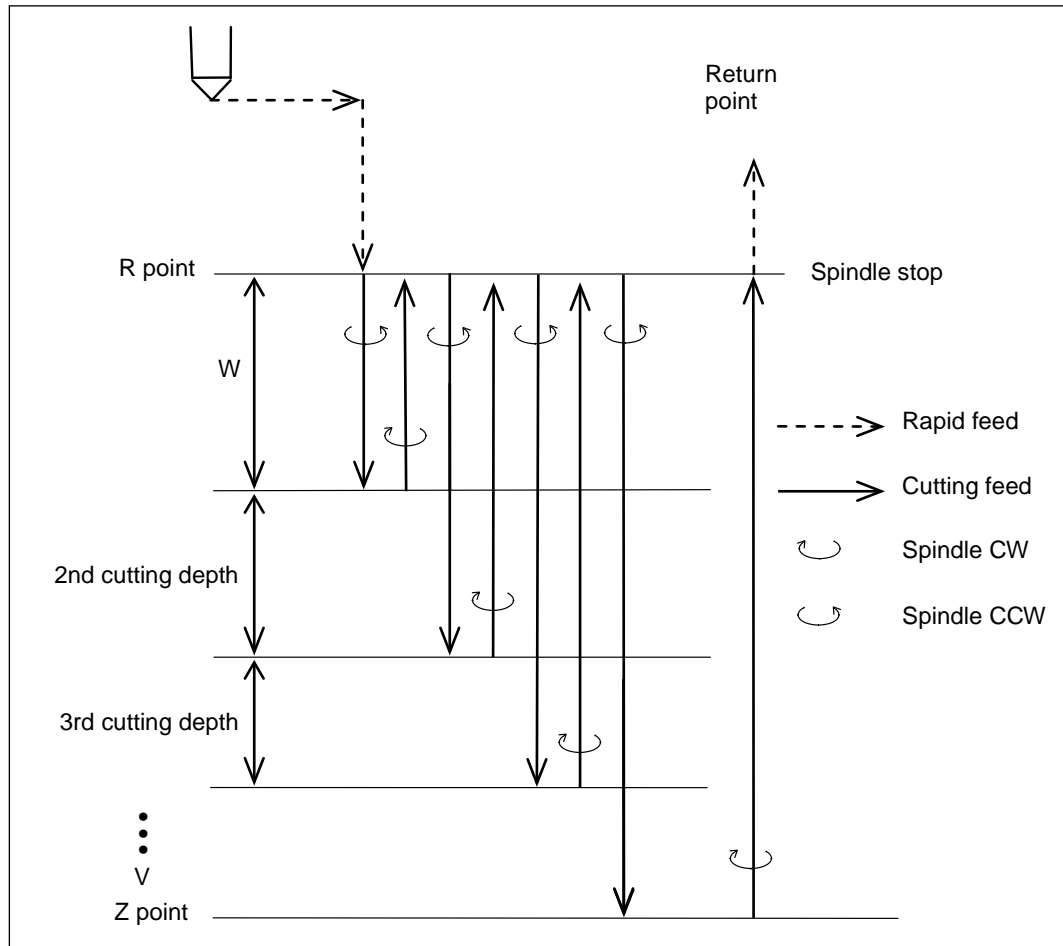
## 6. Reverse deep hole tapping cycle (synchro mode) (G278) (reducing step)

Command format

$$\text{G278 X\_Y\_Z\_R\_} \begin{bmatrix} \text{I\_} \\ \text{J\_} \end{bmatrix} \text{W\_V\_S\_};$$

W : 1st cutting feed

V : Minimum cutting feed



- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.
- When a temporary stop is commanded during cutting, the control stops on returning from the hole bottom position Z to the R point.
- Screw pitch or number of threads must be designated.  
Enter the value after I and J, respectively.
- When I and J exist in the same block, the former is used.
- An alarm occurs if S exceeds the <Max. tapping speed> of Machine parameter (System 1).
- When the synchronized error for the Z-axis and spindle exceeds the Machine parameter (System 1) <<Synchronized tapping error limit>> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <0: Level 4> and this alarm is triggered, the tapping operation in progress at this time continues until the block stop, and then it stops after the tapping operation is complete.  
When the User parameter (Switch 1) <The tapping synchronous error is too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

**7. 2<sup>nd</sup> and after cutting depth in G73, G83, G173, and G183**

2<sup>nd</sup> and after cutting depth in canned cycle G73, G83, G173, and G183 is shown below.

Cutting depth = Factor \* 1st cutting depth (W)

|        |       |       |       |       |      |     |       |       |
|--------|-------|-------|-------|-------|------|-----|-------|-------|
| Count  | 2     | 3     | 4     | 5     | 6    | 7   | 8     | 9     |
| Factor | 0.825 | 0.675 | 0.525 | 0.425 | 0.35 | 0.3 | 0.225 | 0.175 |

|      |     |     |       |       |
|------|-----|-----|-------|-------|
| 10   | 11  | 12  | 13    | 14    |
| 0.15 | 0.1 | 0.1 | 0.075 | 0.075 |

- Use the factor for the 14<sup>th</sup> count to obtain the 15<sup>th</sup> and after cutting depth.
- When the cutting depth gets smaller than V, the tool will cut by the V dimension.

**8. 2<sup>nd</sup> and after cutting depth in G77, G78, G273, and G283**

2<sup>nd</sup> and after cutting depth in canned cycle G77 and G78 is shown below.

Cutting depth = Factor \* 1st cutting depth (W)

|        |      |      |      |     |      |     |     |     |
|--------|------|------|------|-----|------|-----|-----|-----|
| Count  | 2    | 3    | 4    | 5   | 6    | 7   | 8   | 9   |
| Factor | 0.85 | 0.65 | 0.55 | 0.4 | 0.35 | 0.3 | 0.2 | 0.2 |

|      |     |     |      |      |
|------|-----|-----|------|------|
| 10   | 11  | 12  | 13   | 14   |
| 0.15 | 0.1 | 0.1 | 0.05 | 0.05 |

- Use the factor for the 14<sup>th</sup> count to obtain the 15<sup>th</sup> and after cutting depth.
- When the cutting depth gets smaller than V, the tool will cut by the V dimension.

**(Note) Reducing and constant steps are switched using W and Q commands.**

- When W and Q commands exist in the same block, the W command is used.
- When W and Q are not commanded, or zero is specified, the whole depth is cut by a single pass.
- When V is not commanded, or zero is specified, the following applies (for type 1 minimum setting unit):  
 V = 0.001 (metric system)  
 V = 0.0001 (inch system)

### 5.5.20 Canned Cycle Cancel (G80)

Canned cycles (G73, G74, G76 to G78, G81 to G87, G89, G177, G178, G181 to G182, G185, G186, G189, G277, and G278) are canceled and ordinary machining resumes. The hole machining data is canceled except for point R and point Z.

Command format

|             |
|-------------|
| <b>G80;</b> |
|-------------|

**(Note 1) Cancelling the canned cycle is possible by G80 or any of the following commands below.**

- Canned cycle (G100/M06) for tool change
- Change from spindle to lathe spindle (M141 → M142)
- G00 group  
(G00/G01/G02/G03/G02.2/G03.2/G102/G103/G202/G203/G33/G392)  
commands

**(Note 2) Axis travel is performed after canceling canned cycles if axis travel is commanded in the same G80 block.**

### 5.5.21 General Precautions for Canned Cycle

1. For a canned cycle without spindle rotation control (G73, G81 to G83, G85, G89, G181 to G182, G185, and G189), the spindle must be rotated beforehand by M code.
2. When M code is commanded in the same block together with canned cycle command, the M code is executed simultaneously with or after the first X/Y axis positioning. When the count (K) is specified, the M code is executed in the initial instance, and will not be executed thereafter.
3. When M00 or M01 is commanded in the same block as canned cycle command, spindle and coolant will stop after X/Y axes positioning, and no automatic recover is available. Command, when necessary, in manual or MDI operation.
4. The following occurs when G00 to G03, G02.2, and G03.2 are commanded in the same block together with canned cycle:
  - G00 G81 X\_ Y\_ Z\_ R\_ P\_ F\_;  
G00 turns modal and G81 canned cycle is executed.
  - G81 G00 X\_ Y\_ Z\_ R\_ P\_ F\_;  
X Y, and Z axes travel according to G00 and the canned cycle is not executed.
5. You may not command M200, M201, or M120 in the same block together with canned cycle. The alarm <<Invalid Command>> occurs.



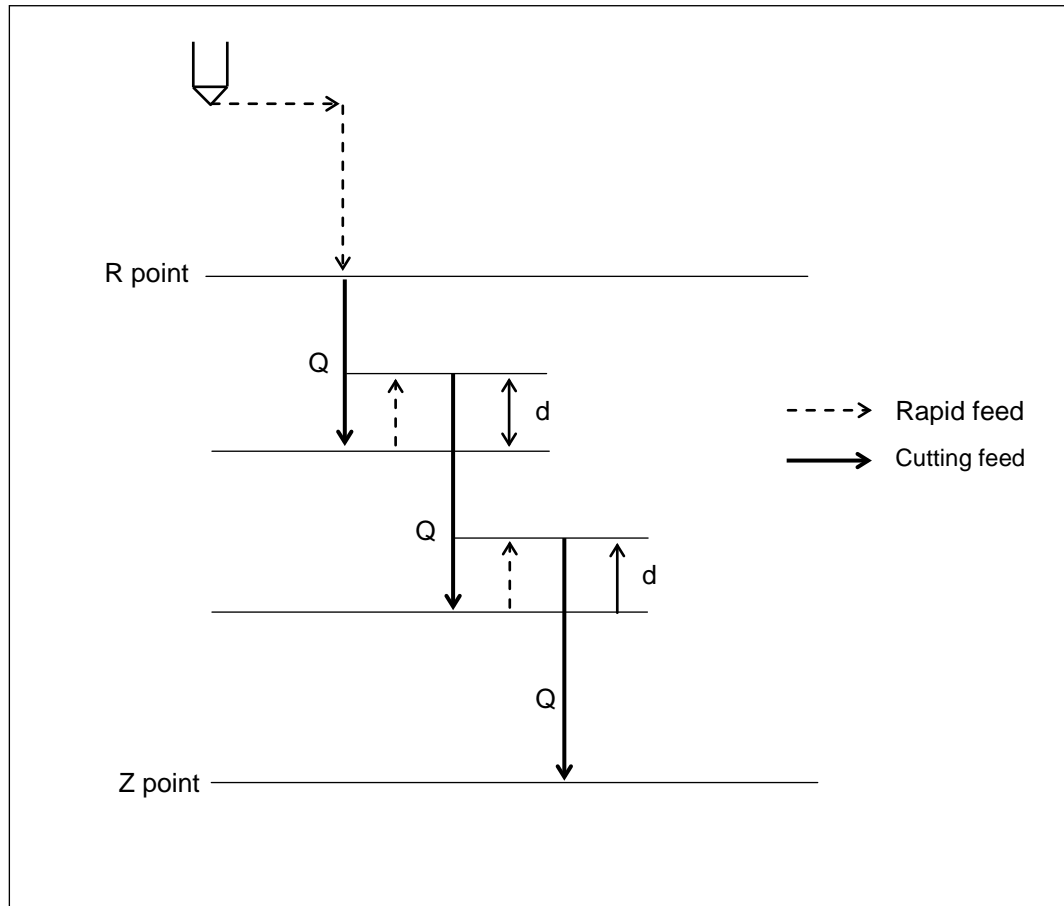
## 5.6 One-shot Canned Cycle

### 5.6.1 High Speed Peck Drilling Cycle (G173)

Command format

**G173 X\_ Y\_ Z\_ R\_ Q\_ F\_;**

This is the cycle where return operation is removed from G73.



- Address K is ignored.

### 5.6.1.1 High Speed Peck Drilling Cycle (G173) (Reducing Step)

Reducing step is available which reduces the cutting depth gradually.

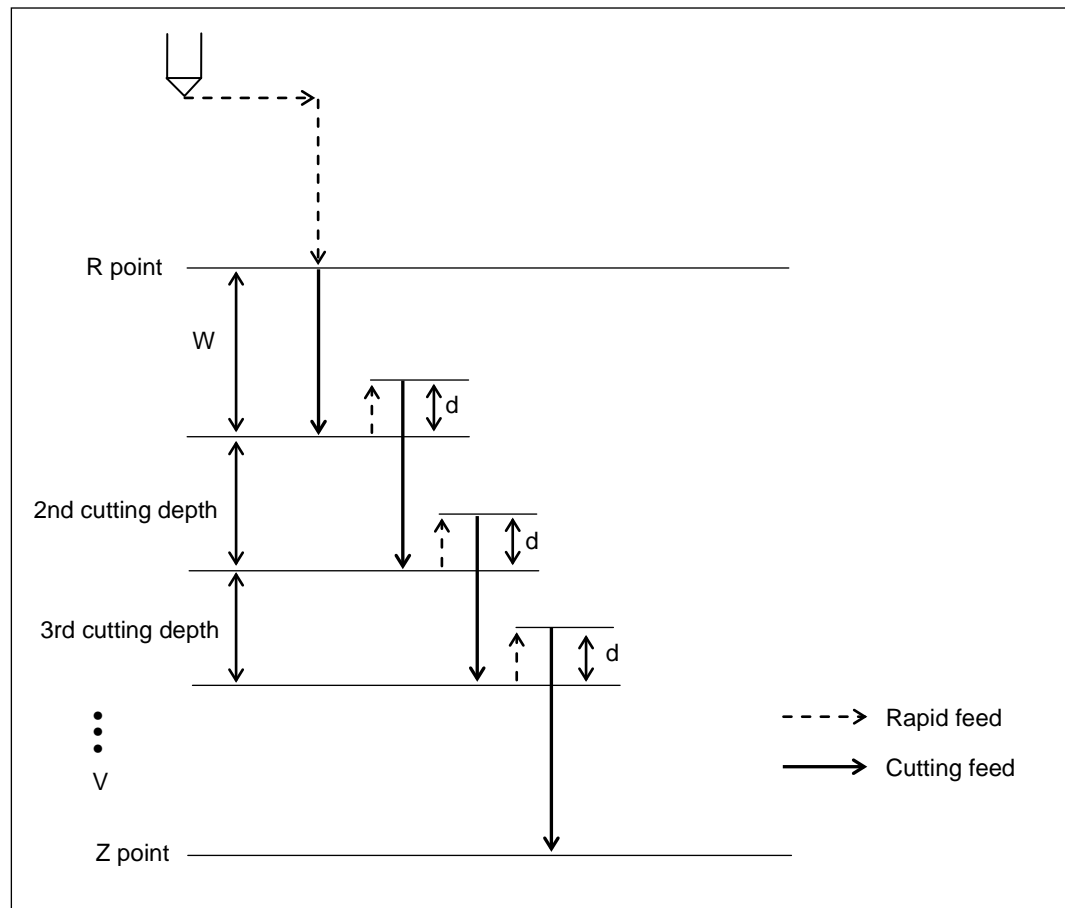
For the 2nd and after cutting depth, refer to 5.5.19 Reducing step of canned cycle.

Command format

**G173 X\_ Y\_ Z\_ R\_ W\_ V\_ F\_;**

W : 1st cutting feed

V : Minimum cutting feed



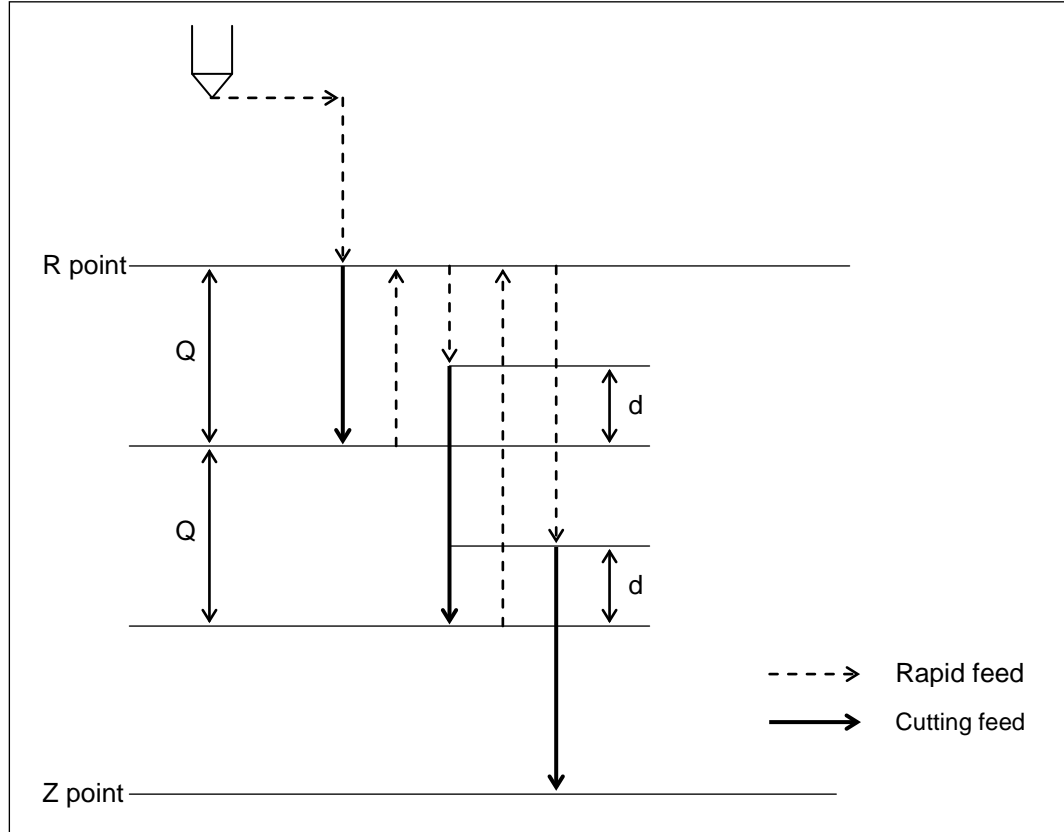
- Retract value d is set on the <G73 relief amount> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

## 5.6.2 Peck Drilling Cycle (G183)

Command format

**G183 X\_ Y\_ Z\_ R\_ Q\_ F\_;**

This is cycle where return operation is removed from G83.



- Address K is ignored.

### 5.6.2.1 Peck Drilling Cycle (G183) (Reducing Step)

Reducing step is available which reduces the cutting feed depth gradually.

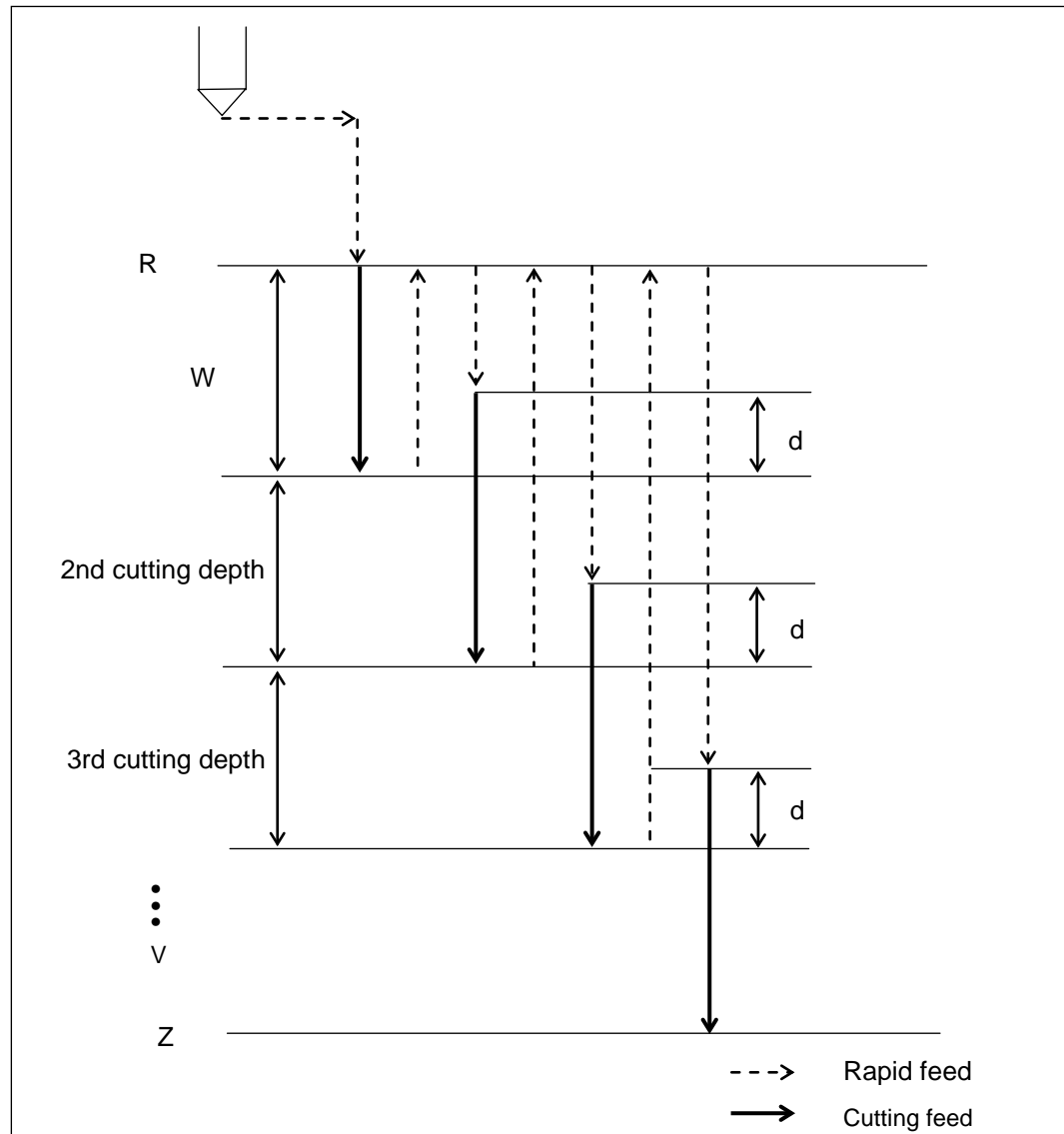
For the 2nd and after cutting depth, refer to 5.5.19 Reducing step of canned cycle.

Command format

**G183 X\_ Y\_ Z\_ R\_ W\_ V\_ F\_;**

W : 1st cutting feed

V : Minimum cutting feed



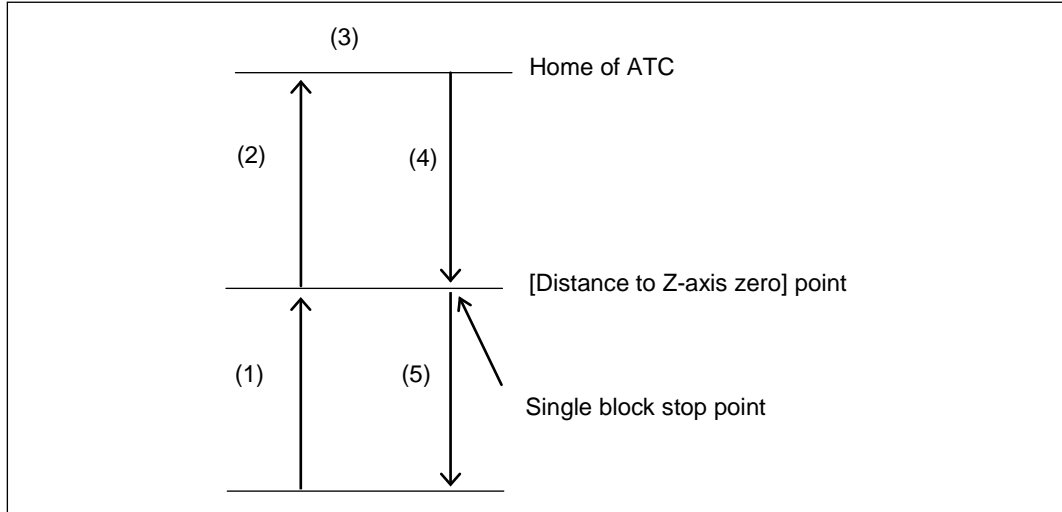
- Cutting start point is set on the <G83 cutting start position> of User parameter (Switch 1).
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

## 5.7 Canned Cycle for Tool Change (Non-stop ATC) (G100)

### 5.7.1 S300X1/S500X1/S700X1/S1000X1/R450X1/R650X1/ S300X2/S500X2/S700X2/F600X1/R450X2/R650X2

Command format

**G100 T\_X\_Y\_Z\_R\_A\_B\_C\_L\_;**



- T : Tool number (1-99), or pot number (101-149), or group number (901-930)
- X、Y、A、B、C : Target value when X, Y, A, B and C axes are moved at the same time as tool replacement operations.  
Movement is rapid feed.
- Z : For the target value for operation in (5), movement is rapid feed.
- R : R commands are ignored.
- L : Then L gives the command for tool number, magazine number and group number.  
The number specified by L is the T modal after G100.

#### Operation

- (1) When moving to the distance to Z-axis zero point, spindle orientation is carried out at the same time.  
(Unclamping of added axes is also carried out at the same time.)
- (2) Moves to ATC zero return.  
When X, Y, A, B and C axes are specified, they are moved to at the same time.
- (3) The magazine turns, and the tool specified by T is assigned.
- (4) Moves to Z-axis zero point position.
- (5) When a Z-axis command is made, the spindle moves to the instructed position.  
If a main spindle command (M03 group) is made, the main spindle moves at the same time.

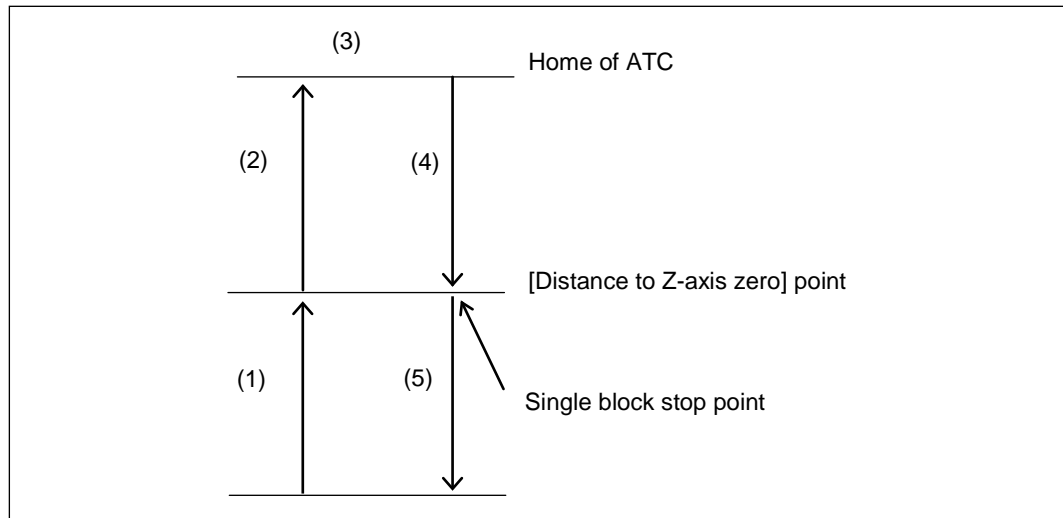
When <2: Equipped (lathe spindle)> is selected for the machine parameter <Optional 5th- to 8th-axis>, the lathe spindle follows the operation described later from section “5.7.2.1 Operation when spindle is selected (M141 modal)” to section “5.7.2.3 Tool change operation when lathe spindle is selected”.

- (Note 1) During cycle operation, (1) and (2), and also (4) and (5) are linked together in their movement during machining mode.
- (Note 2) The machine stops after operation (4) when the [RST] key or the [FEED HOLD] switch is pressed between (2) and (4). However, the X-, Y-, A-, B- and C-axis travel stops immediately at (2).
- (Note 3) Switching the mode is not possible between (1) and (4).
- (Note 4) It is possible to omit all addresses except the G100 address, but a T code command has to be issued once before the G100 command is issued.
- (Note 5) The cutter compensation and the nose R compensation is cancelled when the G100 command is issued. In addition, the tool length / tool position offset (Z-axis) is cancelled starting from (1).
- (Note 6) When cutter compensation / nose R compensation commands on the G100 block and travel commands on the selected plane axis are issued, the cutter compensation / nose R compensation startup operation is performed in the travel for the selected plane axis in (2). Regardless of the <Start up/cancel> setting in the User parameter (Switch 1) the operation is performed following the Type 1 setting.
- (Note 7) The X- and Y- axes compensation is enabled from operation (2) and the Z-axis compensation is enabled from (5) when the tool length / tool position offset command is issued on the G100 block.
- (Note 8) Only the M code commands listed in the “Simultaneous command M code” (described later) are valid for the G100 block. The alarm <<Specified M code cannot be used.>> is triggered when an unlisted or invalid M code command is issued.
- (Note 9) The alarm <<There is no \*-axis option>> is triggered if an A-axis, B-axis or C-axis command is issued when there is no A-axis, B-axis or C-axis option.
- (Note 10) The <ATC simultaneous operation start position> in the <User parameter> is not used.
- (Note 11) The <ATC reference tool length> in the <User parameter> is not used. Even if the G100 command is issued when the tool length / tool position offset is cancelled (G49), (1) travels the distance to Z-axis zero position. If there is no tool length / tool position offset command on the G100 block, (5) travels to the Z command position.
- (Note 12) When a feature coordinate index (G53.1) command is issued on the G100 block, travel (A-axis, B-axis or C-axis rotation) is performed for the feature coordinate index in (2).  
The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when an A-axis, B-axis or C-axis command is issued. Issue a command with the coordinate values in the feature coordinate system for the X-axis, Y-axis or Z-axis.  
When a command is issued on the same block as G100 and G53.1 in G01 modal, the A-axis, B-axis or C-axis travels at rapid feed.
- (Note 13) When a G100 command is issued, the machining load monitor (M340) is disabled in the machining load monitor function.

### 5.7.2 M140X1/M140X2/M200X3/M300X3

Command format

**G100 T\_ X\_ Y\_ Z\_ R\_ A\_ B\_ C\_ L\_;**



- T** : Tool number (1 – 99), or pot number (101 – 149), or group number (901 – 930)
- X, Y, A, B and C:** Target values for tool change operation and travel along X-, Y-, A-, B- and C-axes  
Travel is rapid feed
- Z** : Target values for operation (5) and travel is rapid feed.
- R** : R command is ignored.
- L** : Specifies the tool number, magazine number and group number after L.  
The number specified in L becomes the T modal after G100.

5

#### 5.7.2.1 Operation When Spindle is Selected (M141 Modal)

1. Travels the distance to Z-axis zero point and performs the spindle orientation at the same time. (Performs the C-axis unclamp operation at the same time as well.)
2. Travels to the ATC zero point.  
Travels along the corresponding axes at the same time when X-, Y-, A-, B- and C-axes are specified.
3. The magazine turns and indexes the tool specified in T.
4. Travels the distance to Z-axis zero position.
5. Travels to the position that is specified when the Z-axis command is issued.  
Performs the operation at the same time when the spindle command (M03 group) is issued.

#### 5.7.2.2 Operation When Turning Spindle is Selected (M142 Modal)

1. Travels the distance to Z-axis zero point and performs the spindle orientation at the same time. Performs the turning spindle operation at the same time when the turning spindle command (M303 – M305) is issued.
2. Travels to the ATC zero point.  
Travels along the corresponding axes at the same time when X-, Y-, A- and C-axes are specified.
3. The magazine turns and indexes the tool specified in T.
4. Travels the distance to Z-axis zero position.
5. Travels to the position that is specified when the Z-axis command is issued.

### 5.7.2.3 Tool change operation when lathe spindle is selected

When a tool change command is issued while the lathe spindle (M142 modal) is selected, the rotating lathe spindle continues to rotate without stopping. In addition, when a lathe spindle command (M303-M305) is issued on the same block, the lathe spindle rotation command is executed at the same time as the rise operation to the R-point on the Z-axis or "Distance to Z-axis zero". The axis operation apart from the lathe spindle follows the operation noted in the "5.7 Canned Cycle for Tool Change (Nonstop ATC) (G100)" in the NC Programming Manual.

- (Note 1) Pressing the [RST] key, after the tool change starts and before the X- and Y-axes start to travel, immediately stops the rotating lathe spindle. However, when the [RST] key is pressed after the X- and Y-axes start to travel, the rotating lathe spindle stops after the tool change is completed up until the single block stop.
- (Note 2) The lathe spindle rotation does not stop even when the [FEED HOLD] switch is pressed during a tool change.

### 5.7.2.4 Notes

- (Note 1) Travels in cutting mode when cycle operation is enabled and (1) and (2), and (4) and (5) are linked in the cutting mode.
- (Note 2) The machine stops after operation (4) when the [RST] key or the [FEED HOLD] switch is pressed between (2) and (4). However, the X-, Y-, A- and B-axis travel stops immediately at (2). When the C-axis is travelling to the command position, it also stops immediately in the same way as the X-, Y-, A- and B-axes.
- (Note 3) When the C-axis is operating as the lathe spindle, if the [RST] key is pressed between (2) and (4), it stops after operation (4) (Stops after the positioning operation is finished when transitioning from the lathe spindle rotation operation to the positioning operation for the command position).  
The C-axis does not stop even when the [FEED HOLD] switch is pressed.
- (Note 4) The mode cannot be changed between (1) and (4).
- (Note 5) It is possible to omit all addresses except the G100 address, but a T code command has to be issued once before the G100 command is issued.
- (Note 6) The tool diameter / nose R compensation is cancelled when the G100 command is issued. In addition, the tool length / tool position offset (Z-axis) is cancelled starting from (1).
- (Note 7) When cutter / nose R compensation travel commands are issued on the G100 block, the cutter / nose R compensation startup operation is performed using travel in (2). Regardless of the <Start up/cancel> setting in the User parameter (Switch 1) the operation is performed following the Type 1 setting.
- (Note 8) The X- and Y- axes compensation is enabled from operation (2) and the Z-axis compensation is enabled from (5) when the tool length / tool position offset command is issued on the G100 block.
- (Note 9) Only the M code commands listed in the "Simultaneous command M code" (described later) are valid for the G100 block. The alarm <<Specified M code cannot be used.>> is triggered when an unlisted or invalid M code command is issued.
- (Note 10) The alarm <<There is no \*-axis option>> is triggered if an A-axis, B-axis or C-axis command is issued when there is no A-axis, B-axis or C-axis option.
- (Note 11) The <ATC simultaneous operation start position> in the <User parameter> is not used.
- (Note 12) The <ATC reference tool length> in the <User parameter> is not used. Even if the G100 command is issued when the tool length / tool position offset is cancelled (G49), (1) travels the distance to Z-axis zero position.



(Note 13) When a feature coordinate index (G53.1) command is issued on the G100 block, travel (A-axis, B-axis or C-axis rotation) is performed for the feature coordinate index in (2).

The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when an A-axis, B-axis or C-axis command is issued. Issue a command with the coordinate values in the feature coordinate system for the X-axis, Y-axis or Z-axis.

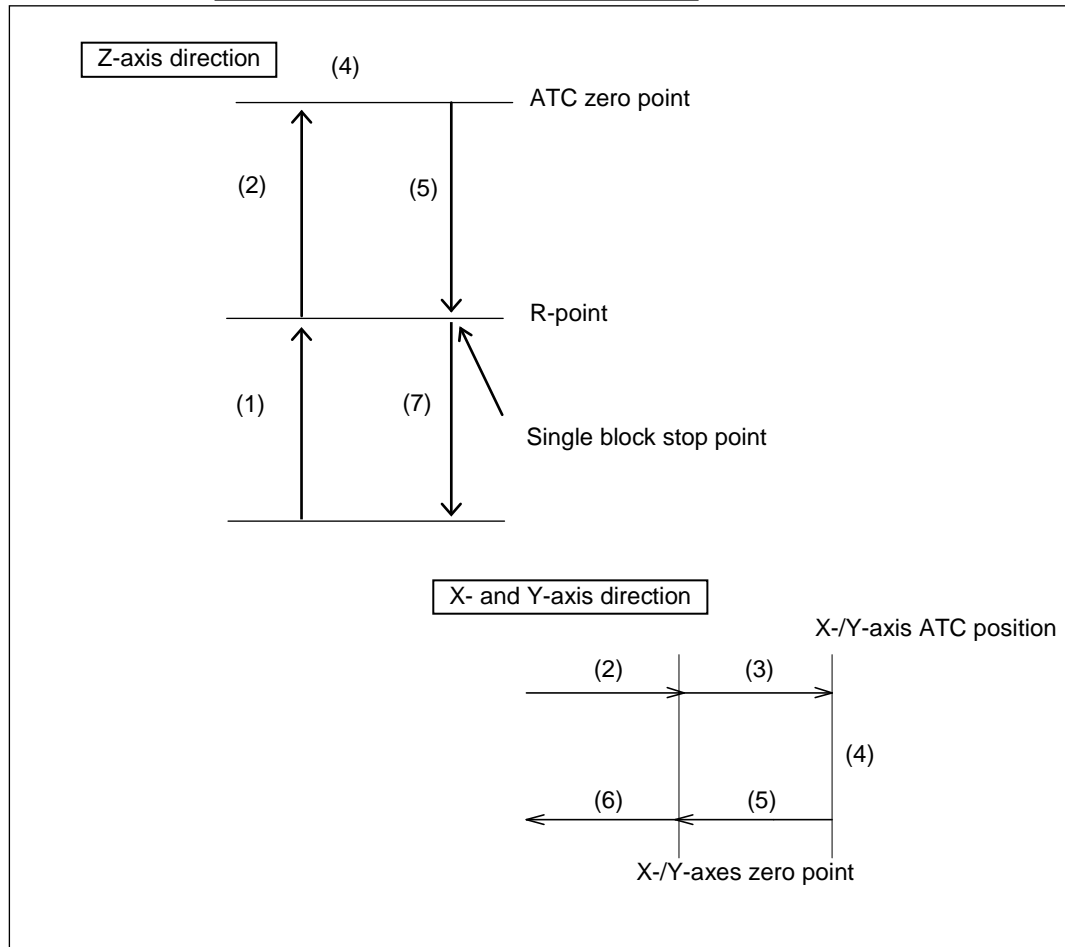
When a command is issued on the same block as G100 and G53.1 in G01 modal, the A-axis, B-axis or C-axis travels at rapid feed.

(Note 14) When a G100 command is issued, the machining load monitor (M340) is disabled in the machining load monitor function.

### 5.7.3 R650X2 40MG

Command format

**G100 T\_ X\_ Y\_ Z\_ R\_ A\_ B\_ C\_ L\_;**



- T : Tool number (1 – 99), or pot number (101 – 149), or group number (901 – 930).
- A, B and C : Target values when moving X-, Y-, A-, B- and C-axes at the same time as the tool change operation, and travel is rapid feed.
- X and Y : Target values for operation (6) and travel is rapid feed.
- Z : Target values for operation (7) and travel is rapid feed.
- R : Return height position before tool change (operates with tool length offset).  
When there is no R command, the user parameter (switch 1) <ATC simultaneous operation start position> is used as the R command value for operation.
- L : Specifies the tool number, pot number and group number after L.  
Pot with the corresponding tool is indexed after the tool change in operation (4).  
(Next tool preparation operation)

Operation

- (1) When traveling to the R-point on the Z-axis, the spindle orientation and the pot shutter open operations are carried out at the same time. When there is a T command, the magazine turns. Thereafter, the pot falls.
- (2) When the Z-axis travels to the ATC zero point and the X- and Y-axes travel to the zero point, travel to the command values on the A-, B- and C-axes also occurs at the same time.
- (3) X- and Y-axes travel to the ATC position.
- (4) The arm turns to carry out a tool change operation. When there is an L command, the magazine turns after the tool change. For the tool change, the operation varies depending on the settings in the ATC tool screen. For details about the tool change operation, refer to the tool change operation items.
- (5) The Z-axis travels to the R-point and the X- and Y-axes travel to the zero point. The operations are carried out at the same time when the spindle command (M03 group) is issued.
- (6) The X- and Y-axes travel to the command value.
- (7) The Z-axis travels to the command value.

**5.7.3.1 Notes**

- (Note 1) During cycle operation, travel connecting (1) and (2) as well as (5) and (7) on the Z-axis occurs in cutting mode.
- (Note 2) When the [RST] key or the [FEED HOLD] switch is pressed during operation (2), it stops after the Z-axis completes travel to the ATC zero point. However, the X-, Y-, A-, B- and C-axes travel stops immediately.
- (Note 3) When the [RST] key or the [FEED HOLD] switch is pressed between operations (3) and (5), it stops after operation (5). However, the A-, B- and C-axes travel stops immediately.
- (Note 4) Between operations (1) and (6), it does not stop at the single block.
- (Note 5) Operation (4) moves onto the next operation without checking the pot rising, the pot shutter closing or checking the preparations for the next tool.
- (Note 6) The mode cannot be changed between operations (1) and (6).
- (Note 7) It is possible to omit all addresses except the G100 address, but a T code command has to be issued once before the G100 command is issued.
- (Note 8) The cutter compensation is cancelled when the G100 command is issued. In addition, the tool length offset is canceled from (2).
- (Note 9) When a cutter compensation command is issued on the G100 block, the cutter compensation startup operation is performed in (6). Regardless of the user parameter (switch 1) <Start up/cancel> value, the startup operation follows the Type 1 (shortcut) setting.
- (Note 10) The X- and Y- axes compensation is enabled from operation (6) and the Z-axis compensation is enabled from operation (5) when the tool length offset command is issued on the G100 block.
- (Note 11) When the G100 command is issued while the tool length offset is cancelled (G49), the user parameter setting <ATC reference tool length> compensates for the tool length in operation (1).
- (Note 12) When there is no tool length/tool position offset command in the G100 block, the user parameter (switch 1) setting <ATC reference tool length> compensates for the tool length or position in operation (5).
- (Note 13) Only the M code commands listed in the “5.7.4 Simultaneous command M codes” (described later) are valid for the G100 block. The alarm <<Invalid command M code>> is triggered when an unlisted or invalid M code command is issued.
- (Note 14) The alarm <<There is no \*-axis option>> is triggered if an A-axis, B-axis or C-axis command is issued when there is no A-axis, B-axis or C-axis option.
- (Note 15) When the Z-axis travel falls for operation (1), the Z-axis does not travel in operation (1).
- (Note 16) When the Z-axis travel rises for operation (7), the Z-axis travels to the command position for operation (5), and does not travel for operation (7).
- (Note 17) It is only possible to intervene manually during a single stop operation at R-point after operation (5). Manual intervention is not possible during any other operation.

- (Note 18) When a feature coordinate index (G53.1) command is issued on the G100 block, travel (A-axis, B-axis or C-axis rotation) is performed for the feature coordinate index in (2). The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when an A-axis, B-axis or C-axis command is issued. Issue a command with the coordinate values in the feature coordinate system for the X-axis, Y-axis or Z-axis. When a command is issued on the same block as G100 and G53.1 in G01 modal, the A-axis, B-axis or C-axis travels at rapid feed.
- (Note 19) After the feature coordinate setting, an R command cannot be issued before the feature coordinate index. Otherwise, the alarm <<Feature coordinate command error>> is triggered.
- (Note 20) When the user parameter (quick table) <Pallet 1 loading> or <Pallet 2 loading> is set to <1:At 1st tool change>, if the R-point position for the first tool change command (G100/M06) is issued at a position lower than the machine parameter (system 1) <R-point lower limit for pallet loading during tool change>, the alarm <<Pallet turn restricted range error>> is triggered.
- (Note 21) When a G100 command is issued, the machining load monitor (M340) is disabled in the machining load monitor function.

### 5.7.3.2 When changing from standard tool to standard tool or from large tool to large tool

1. Magazine turn
2. Pot shutter open
3. Pot fall
4. Arm turn
5. Pot rise
6. Pot shutter close

When the operations above are carried out, the change is complete.

When the tool change is between standard tools, the arm turns at a speed that is set by the machine parameter (system 3) <Rapid feedrate AT-axis 1>.

When the tool change is between large tools, the arm turns at a speed that is set by the machine parameter (system 3) <Rapid feedrate AT-axis 2>.

### 5.7.3.3 When changing from standard tool to large tool or from large tool to standard tool

1. Magazine turn (empty pot is indexed)
2. Pot shutter open
3. Pot fall
4. Arm turn (spindle tool changes to an empty pot)
5. Pot rise
6. Magazine turn (specified pot is indexed)
7. Pot fall
8. Arm turn (specified pot tool changes to spindle)
9. Pot rise
10. Pot shutter close

When the operations above are carried out, the change is complete.

The procedure starts from step 2 when the empty pot is already indexed before the tool change. For the empty pot index in step 1, an empty pot for a large tool is indexed when the tool change is for a large tool, and an empty pot for a standard tool is indexed when the tool change is for a standard tool. In addition, the empty pot is indexed turning in the direction that is closest.

When the tool change is for a large tool, the arm turns for steps 4 and 8 at a speed that is set by the machine parameter (system 3) <Rapid feedrate AT-axis 2>. When the tool change is for a standard tool, the arm turns at a speed that is set by the machine parameter (system 3) <Rapid feedrate AT-axis 1>.

**(Note 1)** When changing from a large to standard tool and there is no empty pot for the large tool, the alarm <<No empty pot>> is triggered.

**(Note 2)** When changing from a standard to large tool and there is no empty pot for the standard tool, the alarm <<No empty pot>> is triggered.

### 5.7.3.4 Next tool preparation operation

The next tool preparation is carried out after the arm turns for the tool change and then after the pot rises. If there is no tool change, then only the next tool preparation operation is carried out. When the next tool is already indexed, the next tool preparation operation is not carried out.

The next tool preparation varies depending on the type of tool that is set on the ATC tool screen.

If the spindle tool type and the next tool type are the same, the specified tool is indexed. However, if the types are different, an empty pot that is the same type as the spindle tool type is indexed.

When the spindle tool is a standard tool and the next tool is a large tool, an empty pot for a standard tool is indexed.

When the spindle tool is a large tool and the next tool is a standard tool, an empty pot for a large tool is indexed.

**(Note 1)** When preparing for the next tool and changing from a large to standard tool, if there is no empty pot for the large tool, the alarm <<No empty pot>> is triggered.

**(Note 2)** When preparing for the next tool and changing from a standard to large tool, if there is no empty pot for the standard tool, the alarm <<No empty pot>> is triggered.

### 5.7.4 Simultaneously Commandable M Codes

The G100 (Canned cycle for tool change) can be commanded in the same block together with the M codes listed below.

If other M code is commanded, the <<Specified M code cannot be used>> alarm appears.

Simultaneously commandable M codes

| M code       | Contents                                    |
|--------------|---------------------------------------------|
| M03          | Spindle CW                                  |
| M04          | Spindle CCW                                 |
| M05          | Spindle stop                                |
| M06          | Tool change                                 |
| M19          | Spindle Orientation                         |
| M111         | Spindle orientation (180°)                  |
| M08          | Coolant pump ON                             |
| M09          | Coolant pump OFF                            |
| M141         | To select a spindle                         |
| M142         | To select a turning spindle                 |
| M230         | Tool life counter set                       |
| M231         | Tool life counter cancel                    |
| M290         | Tool replacement Z axis lower speed 100%    |
| M291 to M293 | Tool replacement Z axis lower speeds 1 to 3 |
| M294         | Tool wash filter check                      |
| M303         | turning spindle forward rotation            |
| M304         | turning spindle backward rotation           |
| M305         | turning spindle stop                        |
| M400         | M400 signal ON (Chip shower ON)             |
| M401         | M400 signal OFF (Chip shower OFF)           |
| M402         | M402 signal ON                              |
| M403         | M402 signal OFF                             |
| M404         | M404 signal ON                              |
| M405         | M404 signal OFF                             |
| M406         | M406 signal ON                              |
| M407         | M406 signal OFF                             |

| M code           | Contents                                                              |
|------------------|-----------------------------------------------------------------------|
| M408             | M408 signal ON                                                        |
| M409             | M408 signal OFF                                                       |
| M420             | ATC arm turn speed (maximum speed)                                    |
| M421             | ATC arm turn speed (large tool speed)                                 |
| M432             |                                                                       |
| M422             | ATC arm turn speed 1                                                  |
| M430             | C-axis unclamp<br>(Simultaneous command is not possible with QT-axis) |
| M431             | C-axis clamp<br>(Simultaneous command is not possible with QT-axis)   |
| M435             | Magazine rotation top speed                                           |
| M436             | Magazine rotational speed 1                                           |
| M437             | Magazine rotational speed 2                                           |
| M440             | B-axis unclamp                                                        |
| M441             | B-axis clamp                                                          |
| M442             | A-axis unclamp                                                        |
| M443             | A-axis clamp                                                          |
| M444             | C-axis unclamp                                                        |
| M445             | C-axis clamp                                                          |
| M450             | One shot signal output                                                |
| M451             | (Proceeds to the next block after the signal has turned off)          |
| M455             | One shot signal output                                                |
| M456             | (Proceeds to the next block without waiting for the signal off.)      |
| M460             | Waiting for M460 signal ON                                            |
| M461             | Waiting for M460 signal OFF                                           |
| M462             | Waiting for M462 signal ON                                            |
| M463             | Waiting for M462 signal OFF                                           |
| M464             | Waiting for M464 signal ON                                            |
| M465             | Waiting for M464 signal OFF                                           |
| M466             | Waiting for M466 signal ON                                            |
| M467             | Waiting for M466 signal OFF                                           |
| M468             | Waiting for M468 signal ON                                            |
| M469             | Waiting for M468 signal OFF                                           |
| M474             | Coil conveyor automatic mode: Enable                                  |
| M475             | Coil conveyor automatic mode: Disable                                 |
| M480             | M480 signal ON                                                        |
| M481             | M480 signal OFF                                                       |
| M482             | M482 signal ON                                                        |
| M483             | M482 signal OFF                                                       |
| M484             | M484 signal ON                                                        |
| M485             | M484 signal OFF                                                       |
| M486             | M486 signal ON                                                        |
| M487             | M486 signal OFF                                                       |
| M490 to M494     | Coolant through center ON                                             |
| M495             | Coolant through center OFF                                            |
| M497             | Tool replacement tool washing ON                                      |
| M498             | Tool wash ON                                                          |
| M499             | Tool wash OFF                                                         |
| M801 to M899     | Signal output * for PLC                                               |
| M900 to M999     | Extend signal output                                                  |
| 2-bit BCD signal | BCD signal output                                                     |

The M codes are simultaneously carried out when going up to an R point except the following cases.

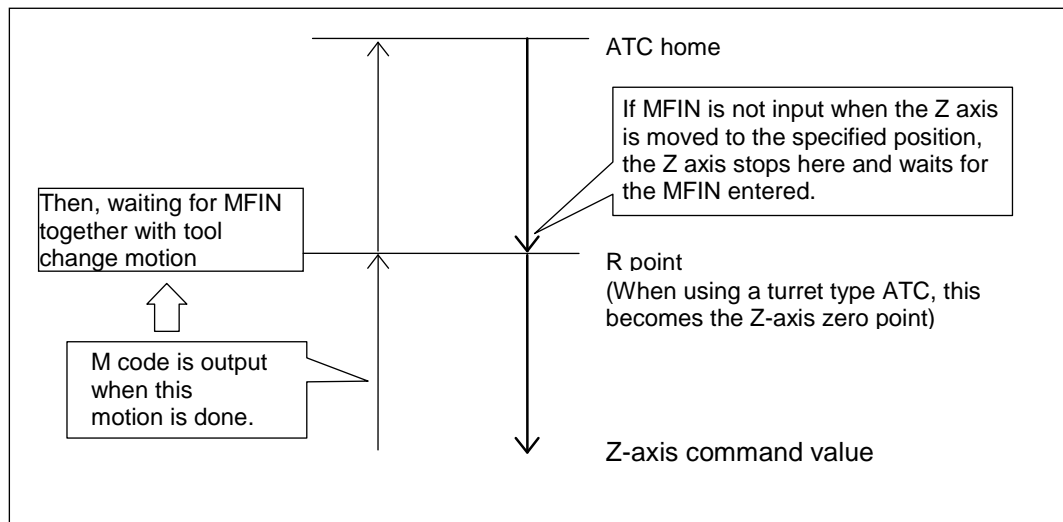
When using a turret type ATC mechanism, the following M code is output at the same time while traveling from the “Distance to Z-axis zero point” to the “Z-axis command point”. When using an arm type ATC mechanism, the code is output at the same time while traveling from the ATC zero point to the R-point, after the tool change.

- (1) M03 (Spindle forward rotation)
- (2) M04 (Spindle backward rotation)
- (3) M05 (Spindle stop)
- (4) M19 (Spindle orientation)
- (5) M111 (Spindle orientation to 180°)
- (6) M08 (Coolant pump ON)
- (7) M490 to M494 (Coolant through center ON)

When using an arm type ATC mechanism, the following M code is output at the same time while traveling from the R-point to the “Z-axis command point”, after the tool change. In addition, when a command is issued at the same time as a canned cycle (G100) for a tool change while in memory operation, it is valid after the magazine turn operation in preparation for the next tool. However, if a command is issued at the same time while in MDI operation, the alarm <<Specified M code cannot be used>> is triggered.

- M435 to M437 (specify maximum magazine turn)

Output of an M signal to wait for MFIN is always accepted during ATC. However, if no MFIN is input before the Z-axis finishes moving down to the R point (Z-axis zero point), the MFIN is waited. After the MFIN is input, the Z-axis moves down to the commanded position.



If the <Multiple M codes in one block> of the <user parameter> is set as <1: Yes>, up to 3 codes can be commanded in the same block.

However, only one of the 2-bit BCD signals, signal outputs (M801 to M899) to PLC, and expansion signal outputs (M900 to M999) can be commanded.

In addition, if M codes which work simultaneously are commanded at the same time, they are output simultaneously. If you want to know the order of output, command them separately in multiple blocks.

### 5.7.5 Automatic Command of Tool Data in Tool Change

You can set the following values <user parameters> registered in the tool data to automatically output for tool changes.

| User parameter                       | Tool data       |
|--------------------------------------|-----------------|
| Automatic command (S) in tool change | S command value |
| Automatic command (F) in tool change | F command value |

If the <Automatically use (S or F) command when changing tool> is set as <1:Yes> each value set in the tool data for the tool in the spindle is automatically updated and output in automatic tool change.

In addition, if a command is already in a same block of a program in NC, it is first commanded prior others.

- (Note 1) The spindle speed can only be automatically set when the constant peripheral speed control is cancelled (G97). The automatic setting does not perform when the constant peripheral speed is being controlled (G96).
- (Note 2) Feed rate can be automatically set only when the feed per minute is set (G94). It is not automatically set when the rotational speed is set (G95).
- (Note 3) If spindle rotational speeds and feed rates in the tool data are not set, the spindle moves in the current S/F mode.
- (Note 4) If dialogue tools are changed by a pot number command, the spindle rotational speed and feed rate are considered as unset and the spindle is moved in the current S/F mode.

## 5.8 Coordinate Calculation Function

### 5.8.1 Outline

Point cloud coordinates (linear, grid, and circular) are calculated in individual blocks.  
Point cloud drilling can be performed by a single command when this command is used in combination with the canned cycle function, etc.

### 5.8.2 Coordinate calculation

Coordinate calculation

| G code | Name             | Function                                                                   |
|--------|------------------|----------------------------------------------------------------------------|
| G36    | Bolt hole circle | Calculate coordinates of a point cloud on the circumference of a circle    |
| G37    | Line (angle)     | Calculate coordinates of a point cloud on a line by specifying angle       |
| G38    | Line (X,Y)       | Calculate coordinates of a point cloud on a line by specifying coordinates |
| G39    | Grid             | Calculate coordinates of a point cloud of a grid form                      |

### 5.8.3 Coordinates Calculation Parameters

Command format

|                               |                              |
|-------------------------------|------------------------------|
| <b>G36</b><br>:<br><b>G39</b> | <b>X_ Y_ I_ J_ K_ P_ Q_;</b> |
|-------------------------------|------------------------------|

X,Y : Datum point coordinates  
I, J, K, P, Q : Coordinates calculation parameters

- Datum point coordinates value (X, Y)
  - Workpiece coordinate system is used for specifying the datum point.
  - Current position is used as the datum point when X/Y values are omitted.
- Coordinate calculation parameters (I, J, K, P, Q)
  - Specify the parameters together with G36 to G39 codes in the same block.
  - The parameters are effective only in the current block. They are erased automatically on completing calculation.
  - Relationship between the functions and parameters are given in the list below.

|                  | G code | Parameter |   |   |   |   |
|------------------|--------|-----------|---|---|---|---|
|                  |        | I         | J | K | P | Q |
| Bolt Hole Circle | G36    | ●         | ● | ● | ● |   |
| Line (angle)     | G37    | ●         | ● | ○ |   |   |
| Line (X, Y)      | G38    |           |   |   |   |   |
| Grid             | G39    | ●         | ● | ● | ● | ● |

● : May not be omitted. An alarm occurs when they are missing.  
○ : May be omitted (assumed to be 1).  
Space : Data may be entered but is not used.

**(Note)** Coordinate calculation function command is not possible while in the inverse time feed (G93) modal. If a command is issued, the alarm <<Command not possible during inverse time feed>> is triggered.



## 5.8.4 Description of Coordinate Calculation Function

### 5.8.4.1 Bolt Hole Circle (G36)

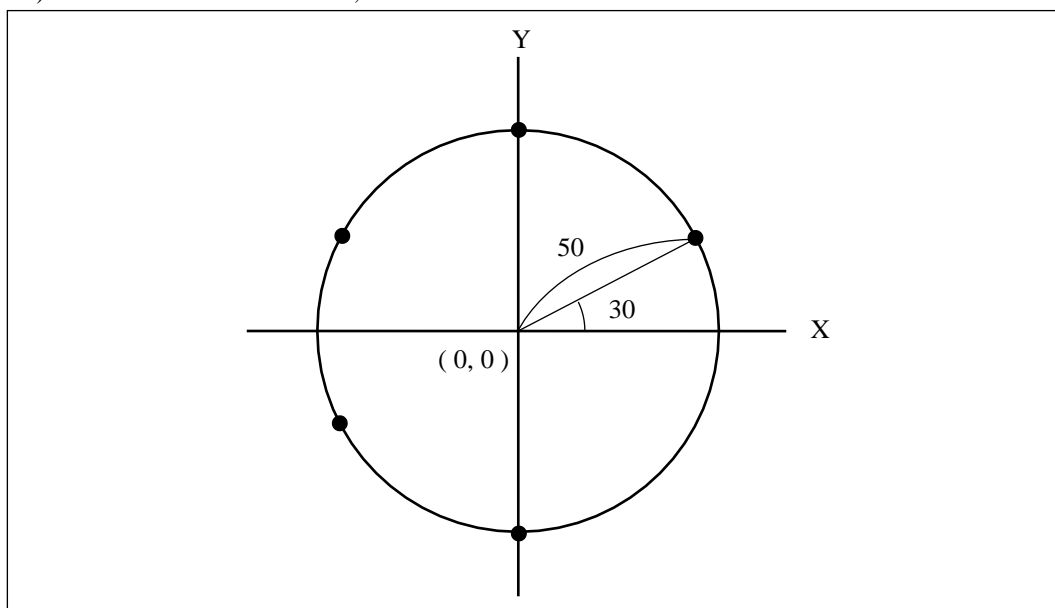
The commanded coordinate value is the center of the arc and the partitioned coordinate values are found for the discretionary points on the circumference as a starting point, etc.

Command format

**G36 X\_ Y\_ I\_ J\_ K\_ P\_;**

X, Y : Coordinate value at arc center  
 I : Arc radius  
 J : Angle with X-axis of the starting point  
 K : Number of drilling holes (999 holes or less)  
 P : Number of splits (max. 999.999)

Ex) G36 X0 Y0 I50 J30 K5 P6;



**(Note)** Coordinate values are measured counterclockwise from the start point.

### 5.8.4.2 Line (Angle) (G37)

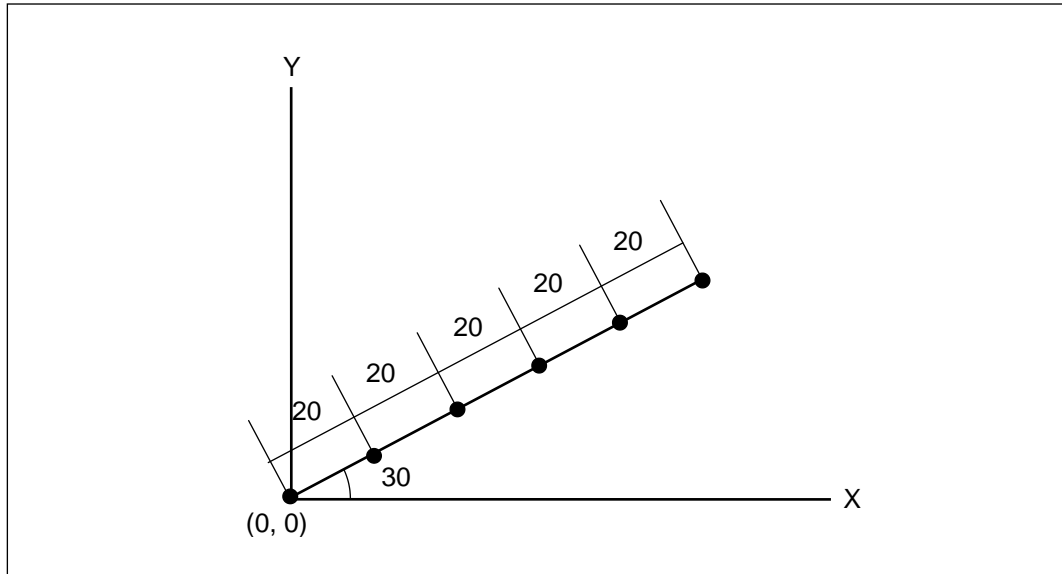
The commanded coordinate value is assumed as the datum point, and the coordinate value is found that lines up on the straight line in the angle ( $\theta^\circ$ ) direction for the X axis.

Command format

**G37 X\_ Y\_ I\_ J\_ K\_;**

X,Y : Datum point coordinates value  
 I : Interval with the point just prior  
 J : Angle with the X-axis.  
 K : Number of drilling holes (999 holes or less)

Ex) G37 X0 Y0 I20 J30 K6;



(Note 1) If omitted, K is considered 1.

(Note 2) The coordinates for the reference point are also output.

### 5.8.4.3 Line (X, Y) (G38)

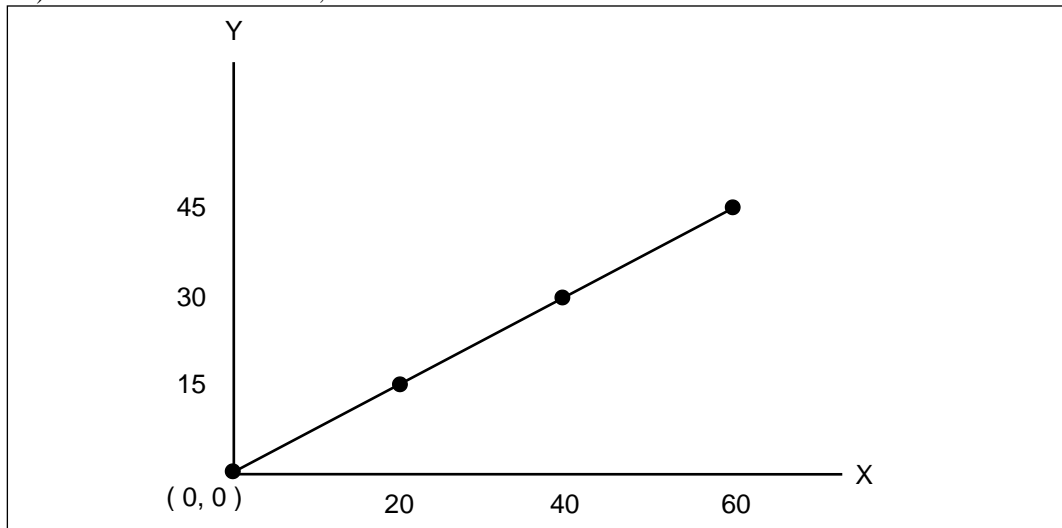
The commanded coordinate value is assumed to be a datum point, and the coordinate values are found that have a plus for both the X direction and the Y direction respectively.

Command format

**G38 X\_ Y\_ I\_ J\_ K\_;**

X, Y : Datum point coordinates value  
 I : Intervals in the direction of X  
 J : Intervals in the direction of Y  
 K : Number of drilling holes (999 holes or less)

Ex) G38 X0 Y0 I20 J15 K4;



(Note 1) If omitted, K is considered 1.

(Note 2) The coordinates for the reference point are also output.

### 5.8.4.4 Grid (G39)

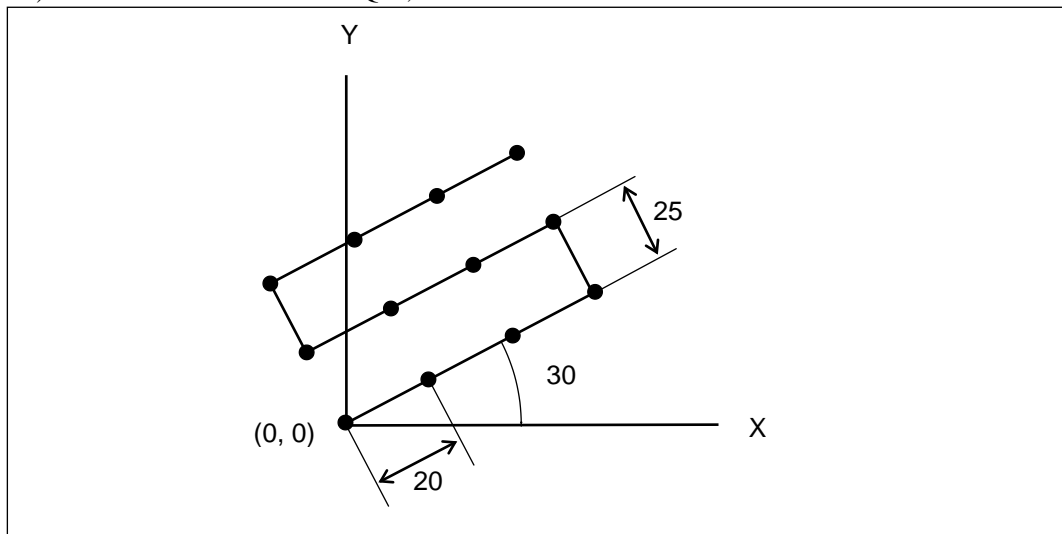
The commanded coordinate value is assumed to be a datum point, and the coordinate values are found for the grid that consists of the points that are lined up at equal intervals parallel, etc. to the X-axis direction as well as the points that are lined up at equal intervals parallel, etc. to the vertical axis. In addition, the whole can be inclined by specifying the angle to the X-axis.

Command format

**G39 X\_ Y\_ I\_ J\_ K\_ P\_ Q\_;**

X,Y : Datum point coordinates value  
 I : Interval in the direction of X axis  
 J : Interval in the direction of Y axis  
 K : Number in the X axis direction (max. 999)  
 P : Number in the Y axis direction (max. 999)  
 Q : Angle with the X axis.

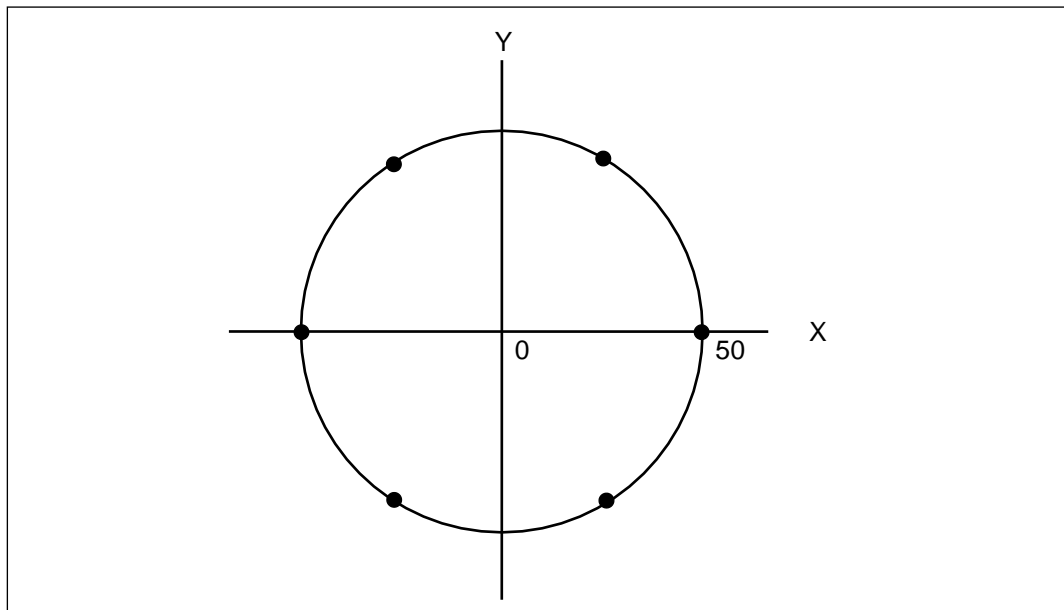
Ex) G39 X0 Y0 I20 J25 K4 P3 Q30;



(Note 1) The coordinates for the reference point are also output.

(Note 2) The direction of the X-axis is obtained from the reference point.

## 5.8.5 Examples of Application



Drilling 6 holes along the circumference of a circle of 50 radius.

```

:
N100 G81R2.Z-10.F1000K0;
N105 G36X0.Y0.I50.J0.K6P6.;
:

```

N100 memorizes canned cycle data, and N105 calculates coordinates to drill holes at the specified position.

# CHAPTER 6

## MACRO

- 6.1 What is Macro?
- 6.2 Variables Function
- 6.3 Calculation Function
- 6.4 Control Function
- 6.5 Call Function
- 6.6 External Output Function
- 6.7 Interrupt Macro (Option)

## 6.1 What is Macro?

Macro allows you to create unique canned cycles and highly versatile programs by incorporating identical motions repeatedly and using variables, calculations, and conditional branching.

Four major macro function groups are:

- Variables function
- Calculation function
- Control function
- Call function

Examples of combination of the functions are shown below (Examples 1 and 2).  
How to create a macro program is described on the following pages.

Ex. 1) Check for tool damage once every 10 machining rounds

```
N01 G90G0G54.....;
N02 ;

•
•Machining program
•
N50 #100=#100+1;          (count up)
N51 IF[#100LT10] GOTO 55; (go to N55 if the content of #100 is less than 10)
N53 M200;                 (detects tool break)
N54 #100=0;               (clear counter)
N55 M30;
```

Ex. 2) Arc machining by specifying the center, radius, and angle.

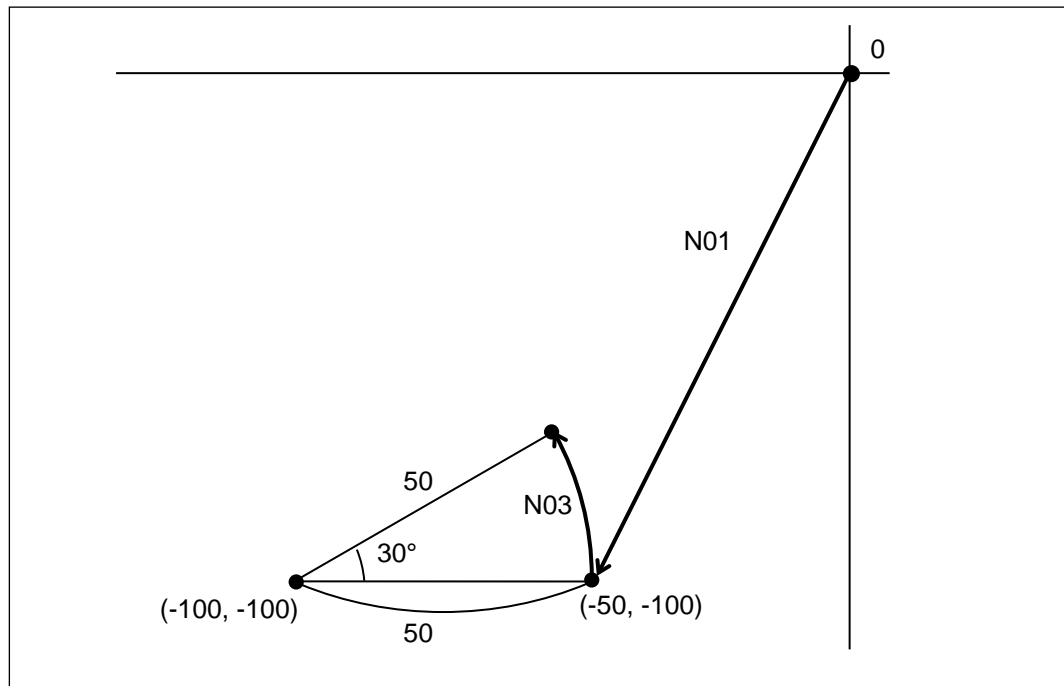
|                        |                        |                      |                       |
|------------------------|------------------------|----------------------|-----------------------|
| X: center X            | Y: center Y            | R: radius            | Z: Cutting position Z |
| W: Stop before workpc. | U: Cutting start angle | V: Cutting end angle | F: Feed rate          |

- Main program

```
N01 G90G54G0Z30.;
N02 G65P0042X-100.Y-100.R50.Z-3.W2.U0.V30.F1000;
```

- Macro program O0042

```
N01 G90G0X[#24+COS[#21]*#18]Y[#25+SIN[#21]*#18];
N02 Z#23;
N03 G1Z#26F#9;
N03 G3X[#24+COS[#22]*#18]Y[#25+SIN[#22]*#18] R#18;
N04 G0Z#23;
N05 M99;
```



## 6.2 Variables Function

### 6.2.1 Outline

Numbers are directly specified, such as G90 and X200, to command an operation in ordinary programs. Using macro variables, you can use the values stored in them as the command for G, X, etc.

The value of variables can be changed by program or MDI operation.

### 6.2.2 Expression of Variables

Each variable number is prefixed with the symbol "#."

Ex. 1) #100

Ex. 2) You can use values stored in a variable and also equations using brackets ([ ]).  
#100 = #[100+10]

The content of variable #110 is substituted in #100.

Ex. 3) For #1 = 9, #9 = 20, and #20 = 30,

assume #5 = #[#[#1]].

Then, this has the same meaning as #5 = 30.

Variables can be used in place of specifying values.

Ex. 4) #3 = #2 + 10;

G01X#3Y10;

The value of variable 2 plus 10 is specified as the X coordinate value.

(if #2 is 40, then X50 is meant.)

If a variable is used as data for an address such as in Example 4, the figure is rounded off to agreed with the number of significant digits of the data.

Ex.) Assume a command G00X#1; for equipment of significant digits 1/1000. When #1 is 12.345678, the command will be G00X12.346.

An alarm occurs when the maximum command value of the relevant address is exceeded.

Ex. 5) Address N cannot take a variable.

You cannot command N#20.

Ex. 6) G00X[#1+#2]; When an equation is used to specify the data for an address, the equation must be put in brackets.

### 6.2.3 Undefined Variables

<empty> is used to indicate the status of an undefined variable.

#0 is always an empty variable. You can read the value but cannot substitute it.

Ex. 1) When #1 is empty:

G01X#1Y100. → G01Y100.

G01X[#1+10.]Y100 → G01X10.Y100.

Ex. 2) Calculation

#0 + #0 → 0

#0 \* 5 → 0

Ex. 3) Conditional equation

| For #1=<empty> |                  | For #1=0 |                   |
|----------------|------------------|----------|-------------------|
| #1 EQ          | #0 → satisfied   | #1 EQ    | #0 → dissatisfied |
| #1 NE          | 0 → satisfied    | #1 NE    | 0 → dissatisfied  |
| #1 GE          | #0 → satisfied   | #1 GE    | #0 → satisfied    |
| #1 GT          | 0 → dissatisfied | #1 GT    | 0 → dissatisfied  |

<empty> is considered not equal to zero in EQ and NE.



## 6.2.4 Types of Variables

Two types of variables are:

1. Local variables (#1~#33), and
2. Common variables (#100 to #199 and #500 to #999)

Local variables are unique to respective macro program call level. When a macro program is called, the local variables of the calling program are saved, and the area for new local variables is prepared for the called macro program.

Local variables and levels are described in 6.5 Call Function.

Common variables can be called and written from any program and level.

Detailed specifications are given in the table below.

Types of variables

| Variable No. | Variable type    | Function                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                          |
|--------------|------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| #0           | Always empty     | Always empty; cannot enter a value.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                               |
| #1 to #33    | Local variables  | Used uniquely on respective levels of a macro program. Initialized to the empty status when power is turned off. The range of variables that can be input into local variables is:<br>-1.0×10 <sup>99</sup> to -1.0×10 <sup>-99</sup> , 0, 1.0×10 <sup>-99</sup> to 1.0×10 <sup>99</sup><br><b>(Note) All digits are not necessarily displayed on the screen but actual variables can be represented in the above range.</b>                                                                                                                                                                                                                                                                                                                                                                                                                      |
| #100 to #199 | Common variables | Used in all different macro programs commonly. Initialized to the empty status when power is turned off. The range of variables that can be input into these common variables is:<br>-1.0×10 <sup>99</sup> to -1.0×10 <sup>-99</sup> , 0, 1.0×10 <sup>-99</sup> to 1.0×10 <sup>99</sup><br><b>(Note) All digits are not necessarily displayed on the screen but actual variables can be represented in the above range. Initialization is also carried out if &lt;Conversation/NC language change&gt; is changed in the &lt;user parameter&gt;.</b>                                                                                                                                                                                                                                                                                               |
| #500 to #999 |                  | Used in all different macro programs commonly. The data is retained when power is turned off. The range and number of significant digits of variables that can be input into these common variables is:<br>(When the minimum set unit is type 1)<br>Metric: -999999.999 to 999999.999<br>(6 digits for integer and 3 for fractional portion)<br>Inch: -99999.9999 to 99999.9999<br>(5 digits for integer and 4 for fractional portion).<br>(When the minimum set unit is type 2)<br>Metric: -999999.9999 to 999999.9999<br>(6 digits for integer and 4 for fractional portion)<br>Inch: -99999.99999 to 99999.99999<br>(5 digits for integer and 5 for fractional portion).<br>When substituting a variable with a large number of significant digits at the fractional portion, the fractional portion is rounded to the above number of digits. |

**(Note)** When an attempt was made to assign a value that exceeds the variable range, the alarm <<Macro command error>> is triggered.

## 6.2.5 Display and Setting of Variables

Variables are displayed and manually set on the Data Bank screen.

To display macro variables, press **[3][ENT]** on the Data Bank Menu screen. Alternatively, select Menu No. 3 with the cursor and press the **[ENT]** key.

The values of common variables (#100 to #199 and #500 to #999) and local variables can be referenced and/or changed.

- Display of values  
The values of common variables from #100 to #199 and all local variables are displayed only when these variables are within the range given below.

(When the minimum set unit is type 1)

Metric: -999999.999 to 999999.999 (6 digits for integer and 3 for fractional portion)

Inch: -99999.9999 to 99999.9999 (5 digits for integer and 4 for fractional portion).

(When the minimum set unit is type 2)

Metric: -999999.9999 to 999999.9999 (6 digits for integer and 4 for fractional portion)

Inch: -99999.99999 to 99999.99999 (5 digits for integer and 5 for fractional portion).

The display turns < \*\*\*\*\* > if these ranges are exceeded.

All digits are not necessarily displayed even though the value is within the above range.

The rounded figures are shown on the display. The value appearing on the screen may therefore differ from the actual variable.

## 6.2.6 System Variables

### 6.2.6.1 Interface I/O Signals

|                                                 |                |     |
|-------------------------------------------------|----------------|-----|
| Signal input                                    | #1000 to #1031 | R   |
| Signal input                                    | #1200 to #1231 | R   |
| Signal input                                    | #1232 to #1263 | R   |
| Signal input                                    | #1264 to #1295 | R   |
| Signal output                                   | #1100 to #1131 | R/W |
| Signal output                                   | #1300 to #1331 | R/W |
| Signal output                                   | #1332 to #1363 | R/W |
| Signal output                                   | #1364 to #1395 | R/W |
| Signal batch read<br>(#1000 to #1031) (32-bit)  | #1032          | R   |
| Signal batch read<br>(#1200 to #1231) (32-bit)  | #1033          | R   |
| Signal batch read<br>(#1232 to #1263) (32-bit)  | #1034          | R   |
| Signal batch read<br>(#1264 to #1295) (32-bit)  | #1035          | R   |
| Signal batch write<br>(#1100 to #1131) (32-bit) | #1132          | R/W |
| Signal batch write<br>(#1300 to #1331) (32-bit) | #1133          | R/W |
| Signal batch write<br>(#1332 to #1363) (32-bit) | #1134          | R/W |
| Signal batch write<br>(#1364 to #1395) (32-bit) | #1135          | R/W |

[Typical application]

Signals are output from a program to port 10 of standard terminal block.

Assign #1100 to port 10 of the standard terminal block with <External Output Signal> of <External I/O Signal>.

- Write a command as follows on the program to output signals to port 10 of the standard terminal block.

```

•
•
#1100=1;
•
•

```

### 6.2.6.2 Workpiece Coordinate Zero Point

Workpiece coordinate zero point is read and written.

|                                  |                |     |
|----------------------------------|----------------|-----|
| Workpiece coordinates (external) | #5201 to #5206 | R/W |
| (G54)                            | #5221 to #5226 | R/W |
| (G55)                            | #5241 to #5246 | R/W |
| •                                | •              | •   |
| •                                | •              | •   |
| (G59)                            | #5321 to #5326 | R/W |
| (G54.1P1)                        | #7001 to #7006 | R/W |
| (G54.1P2)                        | #7021 to #7026 | R/W |
| •                                | •              | •   |
| •                                | •              | •   |
| (G54.1P48)                       | #7941 to #7946 | R/W |

### 6.2.6.3 Tool Data

Tool compensation/life data are read and written.

|                                      |               |                                                                                            |
|--------------------------------------|---------------|--------------------------------------------------------------------------------------------|
| Tool length offset                   | #11001~#11099 | R/W                                                                                        |
| T length wear offset                 | #10001~#10099 | R/W                                                                                        |
| Cutter compensation                  | #13001~#13099 | R/W                                                                                        |
| Tool diameter wear offset            | #12001~#12099 | R/W                                                                                        |
| Tool position offset (X)             | #25001~#25099 | R/W, Available when equipped with a lathe function                                         |
| Tool position wear offset data (X)   | #20001~#20099 | R/W, Available when equipped with a lathe function                                         |
| Tool length offset (Z)               | #26001~#26099 | R/W, Available when equipped with a lathe function                                         |
| T length wear offset (Z)             | #21001~#21099 | R/W, Available when equipped with a lathe function                                         |
| Tool diameter / nose R compensation  | #27001~#27099 | R/W, Available when equipped with a lathe function                                         |
| Tool diameter / nose R wear offset   | #22001~#22099 | R/W, Available when equipped with a lathe function                                         |
| Tool position offset (Y)             | #29001~#29099 | R/W, Available when equipped with a lathe function                                         |
| Tool position wear offset data (Y)   | #24001~#24099 | R/W, Available when equipped with a lathe function                                         |
| Virtual teeth direction              | #23001~#23099 | R/W, Available when equipped with a lathe function                                         |
| Tool life unit                       | #5501~#5599   | R/W<br>1: Is not counted<br>2: Time (minutes)<br>3: Drilling (holes)<br>4: Program (times) |
| Initial tool life / End of tool life | #5601~#5699   | R/W                                                                                        |
| Life warning                         | #5701~#5799   | R/W                                                                                        |
| Tool life                            | #5801~#5899   | R/W                                                                                        |

**(Note)** Some variables are not supported depending on models.

### 6.2.6.4 Alarm Display

#3000=n(ALARM MESSAGE)

Alarm number 9000+n(n:0~200) is generated and the alarm message in the parentheses is displayed (up to 20 characters from top; release level 2).

The values in brackets record the alarm history in single-byte alphanumeric characters only. (Typical application) When executing a block #3000=6(ABCD);, the alarm <<9006 \*ABCD>> occurs.

**(Note 1)** When commanded during cutter compensation, the tool moves to where an offset vector is set vertically perpendicular to the direction of travel of the previous axis travel.

**(Note 2)** If characters other than single-byte alphabet and numbers are described, the operation is not guaranteed.

### 6.2.6.5 Message Display and Stop

#3006=(MESSAGE)

A message of up to 20 characters in the parentheses is displayed after completing execution of the preceding block.

Twenty characters from top is displayed when the message contains 21 characters or more.

Alarm number is 9300 fixed.

(Stop level 1, Reset level 1)

**(Note)** When commanded during cutter compensation, the tool moves to where an offset vector is set vertically perpendicular to the direction of travel of the previous axis travel.

### 6.2.6.6 Time

|        |       |     |                                                                                                                                                                                  |
|--------|-------|-----|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Time 1 | #3001 | R/W | Timer in the unit of 10msec.<br>Clears after 42949672.96 sec (ca. 497 days). Cleared to zero on turning power on, and counts continuously.                                       |
| Time 2 | #3002 | R/W | Timer in the unit of 10msec.<br>Clears after 42949672.96 sec (ca. 497days). Counts the time when activation LED is lit (STL).<br>The value is retained when power is turned off. |
| Date   | #3011 | R   | Current date<br>Ex) January 20, 2007<br>#3011 = 20070120 (Note 1)                                                                                                                |
| Hour   | #3012 | R   | Current time (24-hour system)<br>Ex) 4H 17M 5S pm<br>#3012 = 161705 (Note 2)                                                                                                     |

**(Note 1)** The alarm <<Macro Command Error>> occurs when you attempt to substitute data (#3011) in common variables (#500 to #999).

**(Note 2)** The alarm <<Macro Command Error>> occurs when you attempt to substitute hour (#3012) in common variables (#500 to #999) when <Machine unit system> of <user parameter> is set to <1: Inch>.

### 6.2.6.7 Operation Control

|                   |       |     |                                                |
|-------------------|-------|-----|------------------------------------------------|
| Operation control | #3003 | R/W | MFIN<br>0 : Wait<br>2 : Does not wait          |
| Operation control | #3004 | R/W | Feed<br>[Override]<br>0 : Valid<br>2 : Invalid |

## #3003

- The default value 0 appears when turning power on.
- Turns 0 on resetting and with M30.
- The control goes to the next block without waiting for MFIN if this is set to <Does Not Wait>. MFIN OFF is not checked before outputting an M signal.  
When <Does Not Wait> is set, output time of M signals is set as < External signal output time when MFIN is invalid> of <user parameter>.
- When M signal blocks run successively with MFIN set to <Does Not Wait>, the next M signal is output after time elapse of the above parameter.

## #3004

- The default value 0 when turning power on.
- Turns 0 on resetting and with M30.
- When feed rate override is selected to Invalid, override is fixed at 100% irrespective of feed rate override on the operation panel.
- Spindle override and rapid feed override are also fixed at 100%.

### 6.2.6.8 Mirror Image

Status of mirror image of axes

Binary numbers are converted into decimal numbers for handling.

|              |                                                    |   |                          |
|--------------|----------------------------------------------------|---|--------------------------|
| Mirror image | #3007<br>bit0:X axis<br>bit1:Y axis<br>bit2:Z axis | R | 0 : Invalid<br>1 : Valid |
|--------------|----------------------------------------------------|---|--------------------------|

### 6.2.6.9 Modal Info

You can specify and read modal information.

Modal information (current block)

| Variable No. | Contents                                                                                                                                    |
|--------------|---------------------------------------------------------------------------------------------------------------------------------------------|
| #4001        | G00 to G03, G02.2, G03.2, G102, G103, G202, G203, G33, G392                                                                                 |
| #4002        | G17, G18, G19                                                                                                                               |
| #4003        | G90, G91                                                                                                                                    |
| #4004        | G22, G23                                                                                                                                    |
| #4005        | G93, G94, G95                                                                                                                               |
| #4006        | Inch→20, Meter→21                                                                                                                           |
| #4007        | G40, G41, G42, G141, G142                                                                                                                   |
| #4008        | G43, G44, G143, G144, G49                                                                                                                   |
| #4009        | G73, G74, G76 to G78, G80 to G87, G89, G173, G177, G178, G181 to 183, G185, G186, G189, G277, G278                                          |
| #4010        | G98, G99                                                                                                                                    |
| #4011        | G50, G51                                                                                                                                    |
| #4012        | G66, G67                                                                                                                                    |
| #4013        | G96, G97                                                                                                                                    |
| #4014        | G54 to G59, G54.1                                                                                                                           |
| #4015        | G61, G64                                                                                                                                    |
| #4016        | G68, G69, G168, G68.2                                                                                                                       |
| #4022        | G50.1, G51.1                                                                                                                                |
| #4107        | D code                                                                                                                                      |
| #4109        | F code                                                                                                                                      |
| #4111        | H code                                                                                                                                      |
| #4113        | M code                                                                                                                                      |
| #4114        | Sequence No.                                                                                                                                |
| #4115        | Program number                                                                                                                              |
| #4119        | S code                                                                                                                                      |
| #4120        | T code                                                                                                                                      |
| #4130        | P code (number in extended workpiece coordinate system currently selected)<br>(0 when extended workpiece coordinate system is not selected) |
| #34001       | G321 to G323                                                                                                                                |

## #4113

- The M code returns the M number commanded immediately before.  
The number of the last stated M code in the block is given when two or more M codes are commanded in a block, in the case when <Multiple M codes in one block > of <user parameter> is selected to <Yes>.

## #4114

- Sequence Number returns the N number commanded immediately before.  
(Not the block number currently being executed)  
N90 #100 = 0;  
N100 #100 = #4114;  
When commanded as above, 90 is substituted in #100.

## #4115

- Program number returns the subprogram number when it is being executed.

## #4120

- Pot number command value (101 to 1nn) is returned when T code is commanded by pot number. Group number command value (901 to 930) is returned when T code is commanded by group number.

Modal information (before commanding an interrupt type macro)

| Variable No. | Contents                                                                                                                                    |
|--------------|---------------------------------------------------------------------------------------------------------------------------------------------|
| #4401        | G00 to G03, G02.2, G03.2, G102, G103, G202, G203                                                                                            |
| #4402        | G17, G18, G19                                                                                                                               |
| #4403        | G90, G91                                                                                                                                    |
| #4404        | G22, G23                                                                                                                                    |
| #4405        | G93, G94, G95                                                                                                                               |
| #4406        | Inch→20, Meter→21                                                                                                                           |
| #4407        | G40, G41, G42, G141, G142                                                                                                                   |
| #4408        | G43, G44, G143, G144, G49                                                                                                                   |
| #4409        | G73, G74, G76 to G78, G80 to G87, G89, G173, G177, G178, G181 to 183, G185, G186, G189, G277, G278                                          |
| #4410        | G98, G99                                                                                                                                    |
| #4411        | G50, G51                                                                                                                                    |
| #4412        | G66, G67                                                                                                                                    |
| #4413        | G96, G97                                                                                                                                    |
| #4414        | G54 to G59, G54.1                                                                                                                           |
| #4415        | G61, G64                                                                                                                                    |
| #4416        | G68, G69, G168, G68.2                                                                                                                       |
| #4422        | G50.1, G51.1                                                                                                                                |
| #4507        | D code                                                                                                                                      |
| #4509        | F code                                                                                                                                      |
| #4511        | H code                                                                                                                                      |
| #4513        | M code                                                                                                                                      |
| #4514        | Sequence No.                                                                                                                                |
| #4515        | Program No.                                                                                                                                 |
| #4519        | S code                                                                                                                                      |
| #4520        | T code                                                                                                                                      |
| #4530        | P code (number in extended workpiece coordinate system currently selected)<br>(0 when extended workpiece coordinate system is not selected) |
| #34401       | G321 to G323                                                                                                                                |

### 6.2.6.10 Current Position

| Variable No.                   | Contents                           | Coordinate system                   | Tool offset  | Read while traveling |
|--------------------------------|------------------------------------|-------------------------------------|--------------|----------------------|
| #5001~#5008                    | End point coordinates              | Workpiece Coordinate System         | Not included | Yes                  |
| #5021~#5028<br>#5031~#5034     | Current position                   | Machine coordinate system selection | Included     | No                   |
| #5041~#5048<br>#5051~#5054     | Current position                   | Workpiece Coordinate System         | Included     | No                   |
| #5061~#5068<br>#5071~#5074     | Skip coordinates                   | Workpiece coordinate system         | Included     | Yes                  |
| #5081~#5088                    | Tool length / Tool position offset |                                     |              | No                   |
| #5101~#5108<br>#5111~#5114     | Servo deviation                    |                                     |              | No                   |
| #5161 ~ #5168<br>#5171 ~ #5174 | Skip coordinate                    | Feature coordinate system           | Included     | Possible             |

#5001 to #5008, #5021 to #5028, #5041 to #5048, #5061 to #5068, #5081 to #5088, #5101 to #5108, and #5161 to #5168 read the value of X, Y, Z, and additional axes, respectively.  
#5031 to #5034, #5051 to #5054, #5071 to #5074, #5111 to #5114, and #5171 to #5174 read the value of PLC1 to 4 axes, respectively.

Read while Traveling is specified No for current position, tool length offset, and servo deviation in the above table for reasons that the value cannot be warranted because it reflects the result of look-ahead.

To be specific:

X-10.;  
X-10.;  
X-10.;  
#100 = #5021;  
•  
•

In the above blocks, for example, the macro command looks ahead while the axes are traveling, and thus the position of axes in transit rather than the travel end position in the preceding block is read.

### 6.2.6.11 ATC Tool

Tool number set on the ATC Tool screen is read.

|       |         |   |
|-------|---------|---|
|       | Spindle | R |
| #3701 | Pot 1   | R |
| #3702 | Pot 2   | R |
| •     | •       | R |
| •     | •       | R |
| #3750 | Pot 50  | R |

0 : Cap designation  
1 to 99 : Tool number set in the NC language mode  
1001 to 1099 : Tool number set in the interactive mode  
Empty : Undefined

### 6.2.6.12 Workpiece Counter

The set values on the workpiece counter screen is read and written.

|       |                                |     |
|-------|--------------------------------|-----|
| #3801 | Workpiece counter 1 count      | R/W |
| #3802 | Workpiece counter 1 current    | R/W |
| #3803 | Workpiece counter 1 completion | R/W |
| #3804 | Workpiece counter 1 ending     | R/W |
| #3811 | Workpiece counter 2 count      | R/W |
| #3812 | Workpiece counter 2 current    | R/W |
| #3813 | Workpiece counter 2 completion | R/W |
| #3814 | Workpiece counter 2 ending     | R/W |
| #3821 | Workpiece counter 3 count      | R/W |
| #3822 | Workpiece counter 3 current    | R/W |
| #3823 | Workpiece counter 3 completion | R/W |
| #3824 | Workpiece counter 3 ending     | R/W |
| #3831 | Workpiece counter 4 count      | R/W |
| #3832 | Workpiece counter 4 current    | R/W |
| #3833 | Workpiece counter 4 completion | R/W |
| #3834 | Workpiece counter 4 ending     | R/W |

### 6.2.6.13 Result of Auto Workpiece Measurement

Results of auto workpiece measurements are read.

|                |                                                                                                                |   |
|----------------|----------------------------------------------------------------------------------------------------------------|---|
| #3601 to #3608 | Result of measurement 1 latest<br>X, Y, Z (Note 1), rotation, date, time, G code 1 (Note 2), G code 2 (Note 3) | R |
| #3611 to #3618 | Result of measurement 2 latest                                                                                 | R |
| #3821 to #3828 | Result of measurement 3 latest                                                                                 | R |
| #3631 to #3638 | Result of measurement 4 latest                                                                                 | R |

(Note 1) Results of measurement of X, Y, and Z axes use the machine coordinate system.

(Note 2) G code 1 takes the following values:

1 :G121; 2 :G122; 3 :G123; 4 :G124; 5 :G125; 6 :G126; 7 :G127; 9 :G129

Result of measurement in the case of G128 only is zero.

(Note 3) G code 2 takes the following value:

8 : G128

Result of measurement in the case of not including G128 is zero.

### 6.2.6.14 Rotary Fixture Offset

The reference rotary fixture offset and the current rotary fixture offset are read and written.

|                  |                                                            |     |
|------------------|------------------------------------------------------------|-----|
| #35501 to #35503 | Current rotary fixture offset                              | R   |
| #35520           | Axis for calculation for reference rotary fixture offset 1 | R/W |
| #35521 to #35523 | Reference offset for reference rotary fixture offset 1     | R/W |
| #35524 to #35526 | Reference angle for reference rotary fixture offset 1      | R/W |
| #35540           | Axis for calculation for reference rotary fixture offset 2 | R/W |
| #35541 to #35543 | Reference offset for reference rotary fixture offset 2     | R/W |
| #35544 to #35546 | Reference angle for reference rotary fixture offset 2      | R/W |
| •                | •                                                          | •   |
| •                | •                                                          | •   |
| #35660           | Axis for calculation for reference rotary fixture offset 8 | R/W |
| #35661 to #35663 | Reference offset for reference rotary fixture offset 8     | R/W |
| #35664 to #35666 | Reference angle for reference rotary fixture offset 8      | R/W |



### 6.2.6.15 Machining load monitor

The set value that is used in the machining load monitor function and the machining load value are read and monitored.

|               |                                                                    |   |
|---------------|--------------------------------------------------------------------|---|
| #36000        | Current machining load                                             | R |
| #36001        | Parameter number being used                                        | R |
| #36002        | Machining load monitor method for parameter number being used      | R |
| #36003        | Time constant for parameter number being used                      | R |
| #36004        | Stop level when maximum is reached for parameter number being used | R |
| #36005        | Stop level when minimum is reached for parameter number being used | R |
| #36006        | Maximum machining load for parameter number being used             | R |
| #36007        | Minimum machining load for parameter number being used             | R |
| #36008~#36009 | (Reserved)                                                         |   |
| #36010        | Most recent machining load for parameter No.1 (peak value)         | R |
| #36011        | Most recent machining load for parameter No.1 (average value)      | R |
| #36012~#36019 | (Reserved)                                                         |   |
| •             | •                                                                  | • |
| •             | •                                                                  | • |
| #36990        | Most recent machining load for parameter No.99 (peak value)        | R |
| #36991        | Most recent machining load for parameter No.99 (average value)     | R |
| #36992~#36999 | (Reserved)                                                         |   |

## 6.3 Calculation Function

### 6.3.1 Types of Calculation

The calculations mentioned below are implemented with variables and values.

| Definition of variable          | #i = #j                                                                                                                                                                                     | Definition and replacement                                                                                                                                                                             |
|---------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Addition-type calculation       | #i = #j + #k<br>#i = #j - #k<br>#i = #j OR #k<br>#i = #j XOR #k                                                                                                                             | Addition<br>Subtraction<br>Logical sum<br>Exclusive OR                                                                                                                                                 |
| Multiplication-type calculation | #i = #j * #k<br>#i = #j / #k<br>#i = #j AND #k                                                                                                                                              | Multiplication<br>Division<br>Logical multiplication                                                                                                                                                   |
| Function                        | #i = SIN [#k]<br>#i = COS [#k]<br>#i = TAN [#k]<br>#i = ATAN [#k]<br>#i = SQRT [#k]<br>#i = ABS [#k]<br>#i = BIN [#k]<br>#i = BCD [#k]<br>#i = ROUND [#k]<br>#i = FIX [#k]<br>#i = FUP [#k] | Sine<br>Cosine<br>Tangent<br>Arctangent<br>Square root<br>Absolute value<br>BCD to BIN conversion<br>BIN to BCD conversion<br>Rounding<br>Truncate decimal places<br>Round up to the next whole number |

**(Note)** i, j, and k in the #i, #j, and #k are numerical values.  
 They indicate macro variables such as #10.  
 #j and #k on the right side of an equation may be a constant.

### 6.3.2 Precedence of Calculation

Precedence of calculation is:

1. Function
2. Multiplication-type calculation
3. Addition-type calculation.

To specify precedence of calculation different from the above general rule, you may use square brackets ([ ]).

Up to five sets of square brackets, including those of the function, can be used.

### 6.3.3 Precautions for Calculation

**(Note 1) Equations**

Equations on the right side are constants, variables, or functions, or allow associativity by operators.

Constants without a decimal point are regarded as having a decimal point at the end of the figure.

Example: For #1 = 12;, #1 is 12.000.

**(Note 2) Angle calculation**

The unit of functions SIN, COS, TAN, and ATAN is degrees.

Example: 90 degrees and 30 minutes is commanded as 90.5 degrees.

**(Note 3) Logical operation**

Operations mentioned below are performed for the bits of the integer portion of a logical sum, logical multiplication, and exclusive OR. The fractional portion is 0.

| Target of operation | AND result | OR result | XOR result |
|---------------------|------------|-----------|------------|
| 0 and 0             | 0          | 0         | 0          |
| 0 and 1             | 0          | 1         | 1          |
| 1 and 0             | 0          | 1         | 1          |
| 1 and 1             | 1          | 1         | 0          |

**(Note 4) BCD-BIN conversion**

BIN means binary number. BCD means binary-coded decimal.

Each digit of a decimal number is expressed by a binary number of 4 bits.

Example: 12 = 0001(4 bits)0010(4 bits)

00010010 (a binary number) is equal to 18. When we

BIN-to-BCD convert 12, we obtain 18.

BCD-to-BIN conversion is the reversal of the above operation.

Fractional portion of the source figure is always treated as 0.

**(Note 5) Range of a constant**

The range of a constant used in equations is:

-9999999999 to -0.000000001,

0, and

+0.000000001 to +9999999999

Maximum number of digits available for a constant is 10 digits by decimal number.

**(Note 6) Accuracy of operation**

Certain numerical errors occur and increase with each operation of macro statements. However, the data is retained internally using the floating decimal point system to ensure about 15 significant digits (by decimal number) for a value to warrant accuracy.

## 6.4 Control Function

The flow of a program is changed under certain conditions using the control function. Three types of control functions are:

1. GOTO instruction (unconditional branching)
2. IF instruction (conditional branching)
3. WHILE instruction (repetition)

Use and control availability of these functions are described below.

### 6.4.1 GOTO Instruction (Unconditional Branching)

The program branches to sequence No. n (n: 1 to 99999) unconditionally.

Command format

**GOTO n;**

n : Sequence No.

The alarm <<No applicable sequence>> occurs when the specified sequence number n is not in the range between 1 and 99999, or there is no such sequence number.

Sequence number may be specified by an equation.

Ex) N1 GOTO 3;  
N2 GOTO #10;  
N3 ;

N2 (sequence No. 2) is skipped unconditionally.

If N2 is executed, the control skips to the sequence number of the value of #10.

GOTO instruction is effective within the program to which it belongs. It acts on the first sequence number that it encounters in the program in the direction toward the end of the program. On reaching the end of the program, the search starts from the beginning of the program.

**(Note)** If a command is issued in extended memory operation and tape operation, only searching within a range for approximately 60 KB is possible.

### 6.4.2 IF Instruction (Conditional Branching)

A conditional equation is specified after IF.

Command format

**IF [Conditional equation] GOTO n;**

n : 1~99999

The control branches to the sequence number n when the conditional equation is satisfied. If not, the succeeding block is executed.

Types of conditional equations are described below.

Types of conditional equations

|          |                                   |
|----------|-----------------------------------|
| #i EQ #j | #i is equal to #j                 |
| #i NE #j | #i is not equal to #j             |
| #i GT #j | #i is greater than #j             |
| #i LT #j | #i is less than #j                |
| #i GE #j | #i is equal to or greater than #j |
| #i LE #j | #i is less than #j                |

**(Note 1)** Conditional equation is placed within the brackets [ , ].

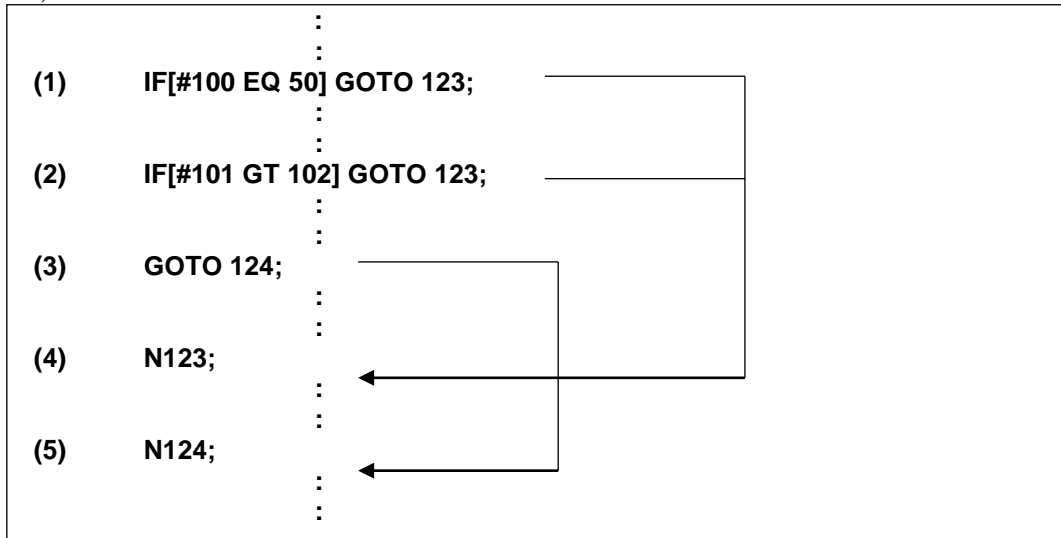
**(Note 2)** The range of values used in a conditional equation is given below.

**Metric:** -9223372036854775.808 to 9223372036854775.807

**Inch:** -922337203685477.5808 to 922337203685477.5807

The alarm <<Macro Command Error>> occurs when specifying a value exceeding the above range.

Ex)



In the above example,

At (1), if the variable of #100 is 50, the control skips to (4) or seq. No. 123.

If the variable of #100 is not 50, the control goes to the next block (2).

At (2), if #101 is greater than 102 ( $\#101 > 102$ ), the control goes to (4) or seq. No. 123.

If #101 is equal to or less than 102 ( $\#101 \leq 102$ ), the control goes to the next block (3).

At (3), the control skips to (5) unconditionally according to the GOTO instruction.

**(Note)** If a command is issued in extended memory operation and tape operation, only searching within a range for approximately 60 KB is possible.

### 6.4.3 WHILE Instruction (Repetition)

Command format

**WHILE [Conditional equation] DOm~ENDm;**

m = 1~4

A conditional equation is inserted after WHILE.

Programs between DO and END are executed as long as the conditional equation is met.

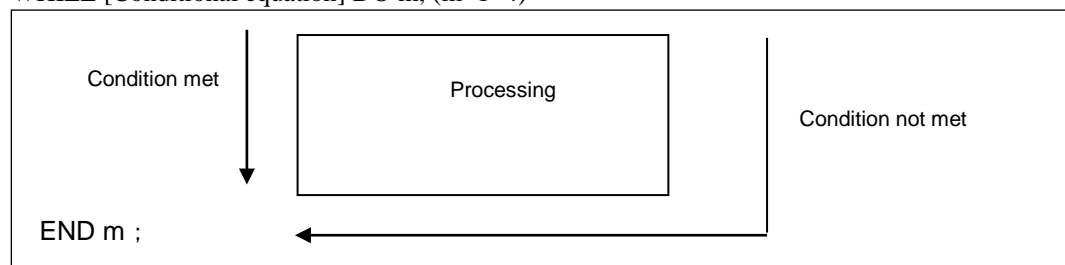
The control goes to the block next to END when the conditional equation is no longer satisfied.

When WHILE[Conditional equation] is omitted, programs between Dom and ENDm is repeated infinitely.

Conditional equation is included in the brackets [ , ].

Ex) WHILE instruction

WHILE [Conditional equation] DO m; (m=1~4)



**(Note 1)** The range of values used in a conditional equation is given below.

Metric: -9223372036854775.808 to 9223372036854775.807

Inch: -922337203685477.5808 to 922337203685477.5807

The alarm <<Macro Command Error>> occurs when specifying a value exceeding the above range.

**(Note 2)** If a command is issued in extended memory operation and tape operation, the <<Macro command error>> is triggered.

## 6.4.4 Precautions for control function

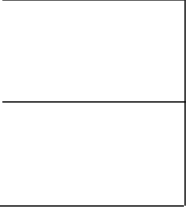
(Note 1) DOm and ENDm must have a one-to-one relationship in WHILE instructions.

The alarm <<Macro Command Error>> occurs.

```

:
WHILE [#100 LT 10] DO 1;
:
:
WHILE [#101 EQ 50] DO 1;
:
:
END 1;
:

```

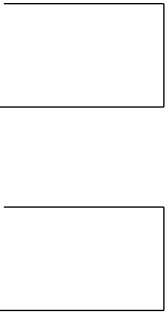


The identifier m(m=1 to 4) may be used as many times as you want provided that the one-to-one relationship is maintained.

```

:
WHILE [#100 LT 10] DO 1;
:
:
END 1;
:
:
WHILE [#101 EQ 50] DO 1;
:
:
END 1;
:

```



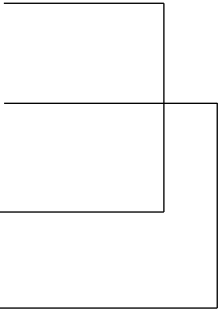
(Note 2) You may not cross DOm and ENDm in WHILE instructions.

The alarm <<Macro Command Error>> occurs.

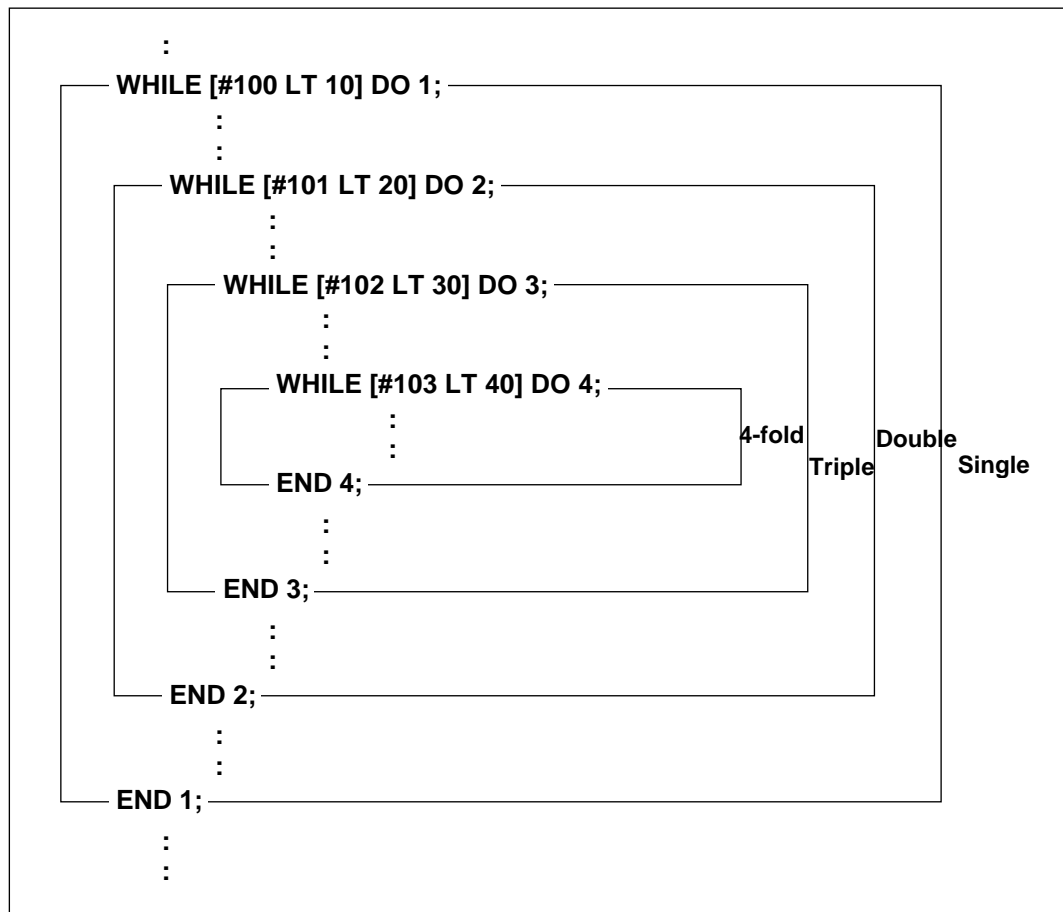
```

:
WHILE [#100 LT 10] DO 1;
:
:
WHILE [#101 EQ 50] DO 2;
:
:
END 1;
:
:
END 2;
:

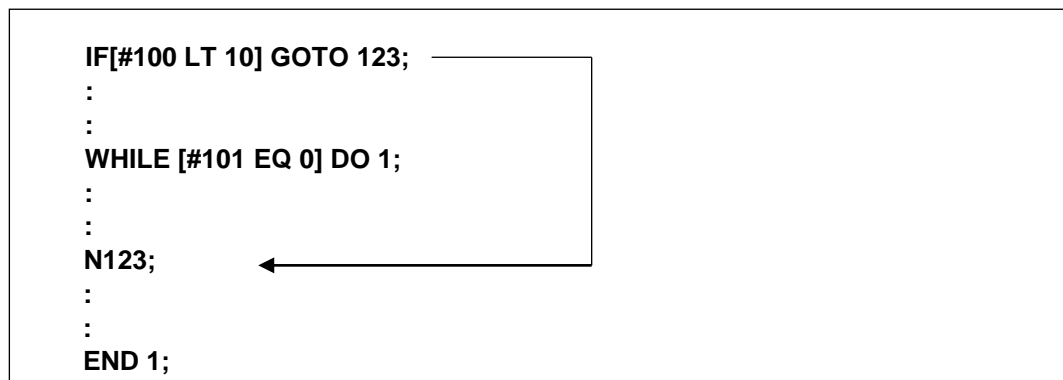
```



(Note 3) Multiplicity of DOs in WHILE instructions is up to 4-fold.  
If this is exceeded, the alarm <<Macro Command Error>> occurs.



(Note 4) IF and WHILE instructions  
An IF-GOTO instruction cannot branch into WHILE-END. The alarm  
<<Macro Command Error>> occurs.



**(Note 5) IF and WHILE instructions**

It is possible to branch off the WHILE-END to other part using IF-GOTO present in WHILE-END.

```
WHILE [#101 EQ 0] DO 1;
```

```
:  
:
```

```
IF[#101 LT 10] GOTO 123;
```

```
:  
:
```

```
END 1;
```

```
:  
:
```

```
N123;
```





## 6.5 Call Function

This function uses G65 or G66 and G code or M code that is registered in the data bank <G/M code macro> to call and execute a separate program. More specifically it could be called a macro calling function or a macro program, which is a program that is called.

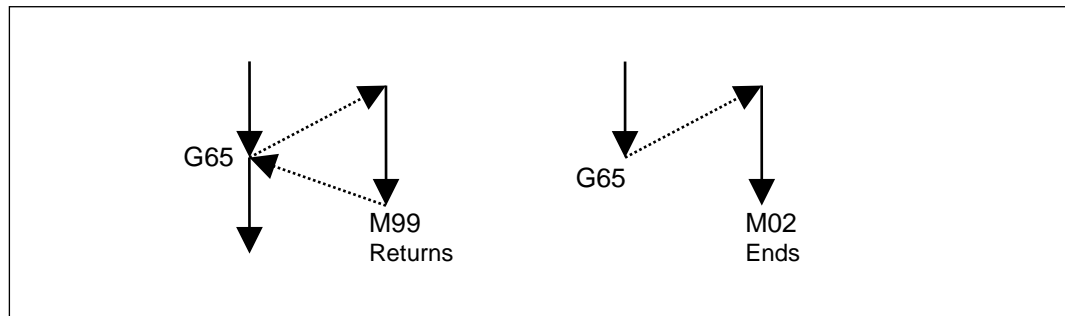
This function is used to help create a unique canned cycle for when the user wishes to repeat the same operation.

Macro program calling (G65, G66 and G code or M code that is registered in the data bank <G/M code macro>) can be executed in memory operation mode call, but it cannot be executed from MDI operation.

The macro program that is called returns to the calling source when M99 is executed.

When the macro program executes M02 or M30 (program end), the program ends without returning to the calling source. (Memory operation ends.)

Difference between M99 and M02



A macro program can call other macros with G65/G66. This parental relationship is available for up to 4 generations (4-fold multiplicity of macro program; this is referred to as multi-layered call).

When calling a macro program, the source program may deliver arbitrary numerical values to the called program as argument.

Macro program call commands (G65 and G66) are similar to call subprogram (M98). The difference is described in detail later.

These call functions are described below.

### 6.5.1 Simple Call Function

Command format

**G65 P\_ L\_ (argument);**

P : Macro program number to be called  
 L : Number of repeats of call (up to 9999)  
 A single call is determined if it is omitted  
 Commands cannot be given during extension memory operation/tape operation.  
 (Argument) : Data to be sent to the macro; may be omitted.

Ex. 1)

G65 P200;

This calls program No. 200 once.

Ex. 2)

G65 P200L2;

is the same as:

G65 P200;

G65 P200;

Program No. 200 is called twice in either case.

## 6.5.2 Modal Call Function

Once call is registered, the designated macro program is called automatically each time an axis travel command is executed.

Modal call is registered by G66 command. Registration is canceled by G67 command. Once modal call is registered, the designated macro program is executed after axes travel each time an axis travel is commanded.

Command format

**G66 P\_ L\_ (argument);**

P : Macro program number to be called  
 L : Number of repeats of call (up to 9999)  
 A single call is determined if it is omitted (see description of G65)  
 Commands cannot be given during extension memory operation/tape operation..  
 (Argument) : Data to be sent to the macro; may be omitted.

Cancel is commanded as follows:

Command format

**G67;**

Ex. 1)

|                   |                                    |
|-------------------|------------------------------------|
| G66 P10;          | (1) Register No. 10 ready for call |
| G01 X-10.0Y-10.0; | (2) Call No. 10 after execution    |
| G01 X-1.0Y-1.0;   | (3) Call No. 10 after execution    |
| G67;              | (4) Cancel registration            |
| G01 X-10.0Y-10.0; | (5) Do not call                    |
| G01 X-1.0Y-1.0;   | (6) Do not call                    |

Program No. 10 is called once after executing (2). It is also called once after executing (3). The program is not called after execution of (5) and (6).

**(Note 1) G67 must be commanded outside the called program.**

**G66 mode can also be canceled by M30.**

**(Note 2) G66 may not be commanded during the G66 mode.**

**(Note 3) G66 command registers a macro program, ready for being called. It does not actually call and execute a macro.**

**(Note 4) Macro variables may be used for specifying a macro program. It is necessary, in this case, to register all programs possibly called by macro variables by inserting M98P? (? = program number) after M30 (M02), such as M98P1 (calling program No. 1). This also applies when using G66 (G67) in place of M98.**

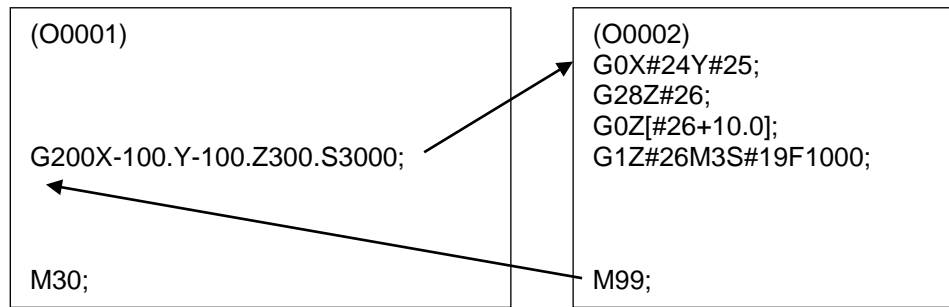
**Ex) Assume #100 can take values of 1, 5, and 100**

|                    |                                                                |
|--------------------|----------------------------------------------------------------|
| G66 P#100;         | ← a macro variable is used in macro program call               |
| G0 X10.Y10.;       |                                                                |
| G100 T1R150.Z100.; |                                                                |
| G67;               |                                                                |
| M30;               |                                                                |
| M98P1;             | } All possible M98P** commands are inserted after M30 command. |
| M98P5;             |                                                                |
| M98P100;           |                                                                |

**(Note 5) More than 2 G/M code macro commands cannot be used on the same block. Otherwise, the alarm <<Invalid command>> is triggered.**

### 6.5.3 G Code Macro Call

A macro program can be called using a registered G code simply by registering the G code number and the corresponding program number, used for macro calling, ahead of time in the parameters.



Command format

**GxxxL\_ (argument);**

- xxx : G code number registered in the data bank as a <G/M code macro>  
However, 0, 65, 66 and 67 cannot be used.
- L : Number of repeated calls (9999 times or less)  
If omitted, it is processed as 1 time (Refer to G65 description).  
Commands cannot be given during extended memory operation/tape operation.
- (Argument) : Data transferred to the macro. Omission is possible.

In order to use G code macro calling, preset the following parameters noted in <G/M code macro> in the data bank.

<G/M code macro (Common parameter)>

- <G code macro program folder>
- <Macro program call control type>

<G/M code macro (G code macro)>

- <G code number>
- <Program No.>
- <Call method>

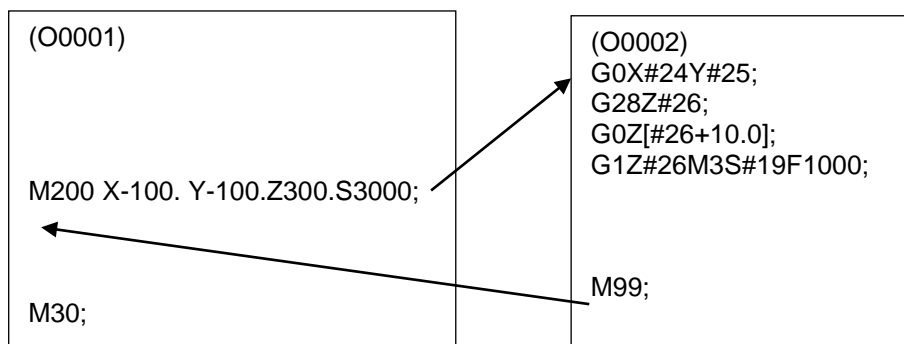
When an M98, G65 or G66 command is issued inside the macro program that is called using a G code, the program is called from a folder that is set in <G code macro program folder>. In addition, a macro call using G code cannot be used inside a program that is called. The system G code operation is carried out.

- (Note 1)** The program restrictions or the conditions that trigger an alarm, and the modal display or the conditions that cancel a modal are processed in the same way as a simple call (G65P\*\*\*\*) or a modal call (G66P\*\*\*\*) depending on the <Call method>.
- (Note 2)** If the G code number used in a macro call overlaps with a G code number that is used in the system, the macro call operation takes precedence.
- (Note 3)** A macro call using G code cannot be used inside a macro program that is called using G code. The system G code operation is carried out.
- (Note 4)** When a macro call using M code is carried out inside a macro program that is called using G code, the system M code operation is carried out when the <Macro program call control type> is set to type 1. The macro program is called when it is set to type 2.
- (Note 5)** More than 2 G/M code macro commands cannot be used on the same block. Otherwise, the alarm <<Invalid command>> is triggered.
- (Note 6)** After a G code based macro is called when the <Call type> is set to <1: Type 2>, the G code based macro call cannot be used until the modal call is canceled (G67 command is executed).

- (Note 7) When there is a G66 command inside the macro program that is called using a G code, if a travel axis command is issued after the macro program call is registered, the program is called from a folder that is set in <G code macro program folder>.
- (Note 8) In the following situations, operation is carried out according to the G code in the system even if executed in MDI operation mode.
- When <System G/M code calling during MDI operation> is set to <1: Call>
  - When executing after switching to MDI operation mode while a macro program called by G code is running
  - When executing after switching to MDI operation mode while a macro program called by M code is running (when <Macro program call control type> is set to <0: Type 1>)

### 6.5.4 M Code Macro Call

A macro program can be called using a registered M code simply by registering the M code number and the corresponding program number, used for macro calling, ahead of time in the parameters.



Command format

**Mxxx L\_ (argument);**

- xxx : M code (discussed later) registered in the parameter  
However, 0, 1, 2, 30, 96 to 99 and 203 cannot be used.
- L : Number of repeated calls (9999 times or less)  
If omitted, it is processed as 1 time (Refer to G65 description).  
Commands cannot be given during extended memory operation/tape operation.
- (Argument) : Data transferred to the macro. Omission is possible.

In order to use M code macro calling, preset the following parameters noted in <G/M code macro> in the data bank.

<G/M code macro (Common parameter)>

- <M code macro program folder>
- <Macro program call control type>

<G/M code macro (M code macro)>

- <M code number>
- <Program No.>

When an M98, G65 or G66 command is issued inside the macro program that is called using an M code, the program is called from a folder that is set in <M code macro program folder>. In addition, a macro call using M code cannot be used inside a program that is called. The system M code operation is carried out.

- (Note 1) The program restrictions or an alarm, and the modal display or the conditions that cancel a modal are processed the same way as a simple call (G65P\*\*\*\*).

- (Note 2) If the M code number used in a macro call overlaps with an M code number that is used in the system, the macro call operation takes precedence.
- (Note 3) A macro call using M code cannot be used inside a macro program that is called using M code. The system M code operation is carried out.
- (Note 4) When a macro call using G code is carried out inside a macro program that is called using M code, the system G code operation is carried out when the <Macro program call control type> is set to type 1. The macro program is called when it is set to type 2.
- (Note 5) More than 2 G/M code macro commands cannot be used on the same block. Otherwise, the alarm <<Invalid command>> is triggered.
- (Note 6) When there is a G66 command inside the macro program that is called using an M code, if a travel axis command is issued after the macro program call is registered, the program is called from a folder that is set in <M code macro program folder>.
- (Note 7) In the following situations, operation is carried out according to the M code in the system even if executed in MDI operation mode.
- When <System G/M code calling during MDI operation> is set to <1: Call>
  - When executing after switching to MDI operation mode while a macro program called by M code is running
  - When executing after switching to MDI operation mode while a macro program called by G code is running (when <Macro program call control type> is set to <0: Type 1>)

### 6.5.5 Macro Call Arguments

Arguments must be specified if it is necessary to send local variables to macro programs.

Method 1:

Arguments are specified with all addresses except G, L, N, O, and P.

| Address for specifying arguments | Variables in macro |
|----------------------------------|--------------------|
| A                                | #1                 |
| B                                | #2                 |
| C                                | #3                 |
| D                                | #7                 |
| E                                | #8                 |
| F                                | #9                 |
| G                                | -                  |
| H                                | #11                |
| I                                | #4                 |
| J                                | #5                 |
| K                                | #6                 |
| L                                | -                  |
| M                                | #13                |
| N                                | -                  |
| O                                | -                  |
| P                                | -                  |
| Q                                | #17                |
| R                                | #18                |
| S                                | #19                |
| T                                | #20                |
| U                                | #21                |
| V                                | #22                |
| W                                | #23                |
| X                                | #24                |
| Y                                | #25                |
| Z                                | #26                |

- (Note) When an M code based macro is called, the address M cannot be specified for an argument. Otherwise, the alarm <<Invalid data>> is triggered.

## Method 2

Number of repeats of I, J, and K and A, B, and C can be specified.

| Address for specifying arguments | Specified number of repeats | Variables in macro |
|----------------------------------|-----------------------------|--------------------|
| A                                | 1                           | #1                 |
| B                                | 1                           | #2                 |
| C                                | 1                           | #3                 |
| I                                | 1                           | #4                 |
| J                                | 1                           | #5                 |
| K                                | 1                           | #6                 |
| I                                | 2                           | #7                 |
| J                                | 2                           | #8                 |
| K                                | 2                           | #9                 |
| I                                | 3                           | #10                |
| J                                | 3                           | #11                |
| K                                | 3                           | #12                |
| I                                | 4                           | #13                |
| J                                | 4                           | #14                |
| K                                | 4                           | #15                |
| I                                | 5                           | #16                |
| J                                | 5                           | #17                |
| K                                | 5                           | #18                |
| I                                | 6                           | #19                |
| J                                | 6                           | #20                |
| K                                | 6                           | #21                |
| I                                | 7                           | #22                |
| J                                | 7                           | #23                |
| K                                | 7                           | #24                |
| I                                | 8                           | #25                |
| J                                | 8                           | #26                |
| K                                | 8                           | #27                |
| I                                | 9                           | #28                |
| J                                | 9                           | #29                |
| K                                | 9                           | #30                |
| I                                | 10                          | #31                |
| J                                | 10                          | #32                |
| K                                | 10                          | #33                |

(Note 1) Addresses that do not need designation need not be specified.

(Note 2) The value of local variables corresponding to non-specified address is empty.

(Note 3) Methods 1 and 2 are switched by the address in use. When both are used, the last specified value is effective.

(Note 4) Arguments must be commanded after G65 and G66.

Ex) G65 D1.0 E2.0 I3.0 J4.0 K5.0 I6.0 J7.0 K8.0 F9.0

(1) #7 ← 1.0

(2) #8 ← 2.0

(3) #4 ← 3.0

(4) #5 ← 4.0

(5) #6 ← 5.0

(6) #7 ← 6.0 (value specified by D is invalid)

(7) #8 ← 7.0 (value specified by E is invalid)

(8) #9 ← 8.0

(9) #9 ← 9.0 (value specified by K for the second time is invalid)

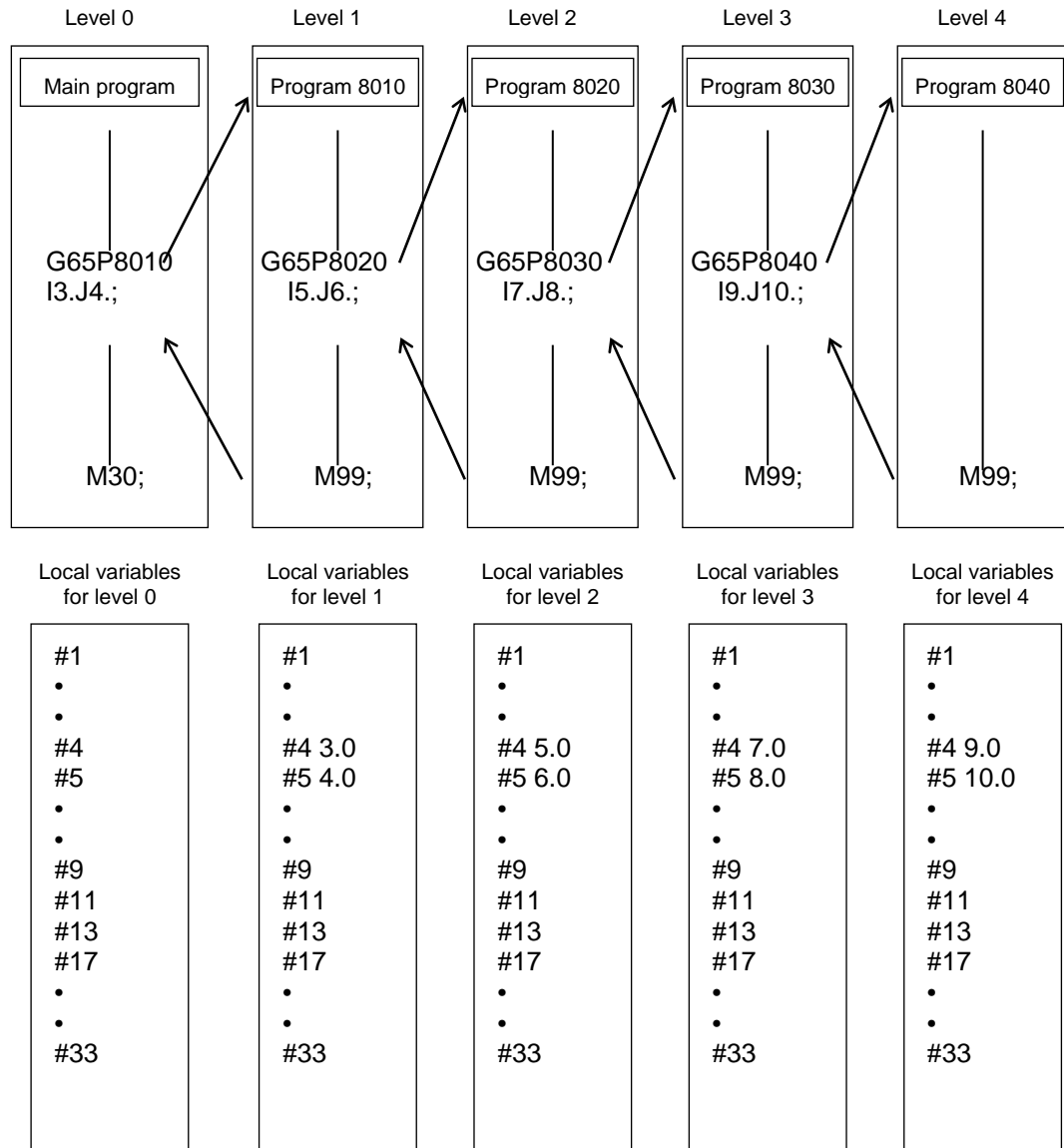
### 6.5.6 Difference Between G65 and M98

1. G65 can specify arguments but M98 cannot.
2. G65 can have local variables for respective levels of its multiplicity but M98 cannot.
3. Multiplicity of call of G65 is up to 8-fold including call by M98, and up to 4-fold by itself.

### 6.5.7 Multiple Call

Up to 4-level deep macro call is possible. Local variables (#1 to #33) are prepared for each level. When a macro is called by G65, etc., the current local variables are saved and those for the level of the called macro program are newly set for use. The saved local variables return when M99 is executed.

Common variables are read and written across all levels.



## 6.6 External Output Function

When executing an external output command shown below during memory operation, the macro variable values or characters can be output to external devices through RS-232C or can be output as a file to a memory card/FTP server.

1. POPEN ... Instruction that executes a preparatory processing of data output
2. BPRNT ... Instruction that executes an output of characters and a binary output of macrovariable values
3. DPRNT ... Instruction that executes an output of characters and a character string output of macro variable values
4. PCLOS ... Instruction that executes a terminating processing of data output

Figure 1 NC program that contains the external output command

```
G90;
:
POPEN;
BPRNT[#100[3]];
DPRNT[#2[63]];
PCLOS;
:
M30;
```

### 6

### 6.6.1 POPEN

This command links with an external connection. Specify this command prior to respective commands of BPRNT, DPRNT, and PCLOS.

Command format

**POPEN;**

When connecting to a <General COMM device>

The control code of “DC2” is output if the <Communication mode> of the <Communication parameter> is <1: Code 1> or <2: Code 2>. Nothing is output if <0: Line> is set.

When connecting to a <Memory card>

It opens the memory card file.

When connected to an <FTP server>

This connects to the FTP server.

**(Note 1)** If POPEN is specified when the POPEN state has already been established, it is ignored.

**(Note 2)** When connecting to a <Memory card>, be sure to keep the memory card inserted without removing it until the PCLOS command or the program is finished.



## 6.6.2 BPRNT

This command executes an output of characters and a binary output of macro variable values.

Command format

```
BPRNT[ xx #xx [x] ... ];
```

**(Note 1)** The alarm << POPEN is unable. >> occurs if BPRNT is commanded without first commanding POPEN.

**(Note 2)** Add the “end of block” code at the end of output data.

### 1. Output of characters

The following characters are output as they are.

Alphabets “A” to “Z”

Numbers “0” to “9”

Symbols “(” “)” “=” “/” “.” “+” “,” “-” “?”

A space is not output. Instead, “###” is output with a space code.

**(Note 1)** When connecting to a <General COMM device>, the output character code follows the <Communication parameter> <Send data code> setting. The output character code does not output “?” when using EIA format.

**(Note 2)** The alarm <<Macro Command Error>> occurs if “#”, “[” and “]” are output (used other than in the output format of macro variables).

**(Note 3)** Motion is not guaranteed when using characters that cannot be output.

### 2. Output of macro variables

Specify the number of significant digits after decimal point in square brackets following the variable command. Macro variable value is treated as 4-byte (32-bit) data, and it is output as binary data, starting from the high-order byte.

**(Note 1)** When a macro variable value is a negative value, it is output in the expression of two’s complement.

**(Note 2)** If the number of digits after decimal point of the data to be output is larger than the significant digits, the output data is rounded off.

**(Note 3)** The alarm << Too many external output command digit.>> occurs when the macro variable value exceeds the range of -2147483648 to 2147483647 as a result of forms processing.

BPRNT example:

```
BPRNT[DATA*X*#100[2] Y*#101[2] Z*#102[0]];
```

Variable values are

#100=123.456

#101=-123.456

#102=0.056

When output character code is set to ISO, and the end code of the block to CR, LF

|    |      |    |    |    |    |    |    |    |       |    |    |    |    |    |        |    |    |    |    |    |    |    |       |    |
|----|------|----|----|----|----|----|----|----|-------|----|----|----|----|----|--------|----|----|----|----|----|----|----|-------|----|
| 44 | 41   | D4 | 41 | A0 | D8 | A0 | 00 | 00 | 30    | 3A | 59 | A0 | FF | FF | CF     | C6 | 5A | A0 | 00 | 00 | 00 | 00 | 8D    | 0A |
|    | DATA |    | SP |    | X  |    | SP |    | 12346 |    | Y  |    | SP |    | -12346 |    | Z  |    | SP |    | 0  |    | CR,LF |    |

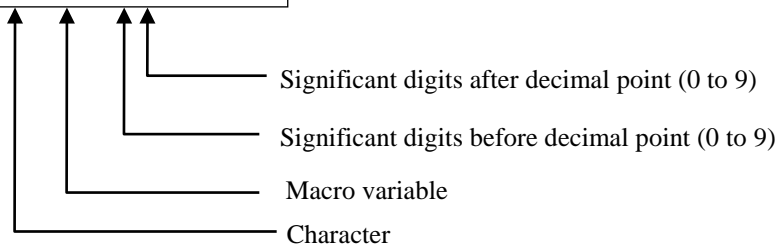
\* SP: space code

### 6.6.3 DPRNT

This command executes an output of characters and a character string output of macro variable values.

Command  
format

**DPRNT [ xx #xx [x x] ... ];**



**(Note 1) The alarm << POPEN is unable. >> occurs if DPRNT is commanded without first commanding POPEN.**

**(Note 2) Add the “end of block” code at the end of output data.**

1. Output of characters  
Same as the BPRNT command. Refer to 1. Output of Characters, 6.6.2 BPRNT.
2. Output of macro variables  
Of a macro variable value, specify necessary number of digits before and after decimal point respectively in square brackets. By this command, a macro variable value is output with the character codes including decimal point by the amount of number of digits specified, every digit starting from the high-order digit.

A space code is output for the high-order zero and a positive sign if the <leading zero suppression (DPRNT)> of the <Communication parameter> is set to <0: Type 1>. Nothing is output if <1: Type 2>.

When the number of digits after decimal point is other than 0, the decimal point and a value after decimal point are always output. If the number of digits after decimal point is 0, the decimal point is not output.

**(Note 1) If the number of significant digits specified is 1, the number of significant digits before decimal point is treated as 0.**

**(Note 2) A numerical value exceeding the number of significant digits is not output. If the number of digits after decimal point of the data to be output is larger than the significant digits, the output data is rounded off.**

**(Note 3) When the data is 0 as a result of rounding off, a sign depends on a numeric value before it was rounded off.**

DPRNT example:

**DPRNT[X\*#100[44] Y\*#101[22] Z\*#102[20] \*#100[2]];**

Variable values are

#100=-123.456

#101=123.456

#102=0.056

When output character code is set to ISO, and the end code of the block to CR, LF

(1) <Leading zero suppression (DPRNT)> of <Communication parameter> is set to <0: Type 1>

|                                                                                           |
|-------------------------------------------------------------------------------------------|
| D8 A0 2D A0 B1 B2 33 2E B4 35 36 59 A0 A0 B2 33 2E B4 36 5A A0 A0 A0 30 A0 2D 2E B4 8D 0A |
| 30 36                                                                                     |
| X  SP -SP123.4560  Y  SP SP23.46  Z  SP SPSP0  SP  -.46   CR,LF                           |

(2) <Leading zero suppression (DPRNT)> of <Communication parameter> is set to <1: Type 2>

|                                                                                     |
|-------------------------------------------------------------------------------------|
| D8 A0 2D B1 B2 33 2E B4 35 36 30 59 A0 B2 33 2E B4 36 5A A0 30 A0 2D 2E B4 36 8D 0A |
| X  SP -123.4560  Y  SP 23.46  Z  SP 0  SP  -.46   CR,LF                             |

\* SP: space code

## 6.6.4 PCLOS

This command cancels the link with the external connection. Specify this command after respective commands of POPEN, BPRNT, and DPRNT.

Command format

**PCLOS;**

When PCLOS is specified, if the data output by the BPRNT or DPRNT command is underway, the PCLOS processing is executed after the data output is completed.

When connecting to a <General COMM device>

The control code of "DC4" is output if the <Communication mode> of the <Communication parameter> is <1: Code 1> or <2: Code 2>. Nothing is output if <0: Line> is set.

When connecting to a <Memory card>

It closes the memory card file.

When connected to an <FTP server>

The connection to the FTP server is severed.

**(Note)** If PCLOS is specified when the PCLOS state has already been established (including the state in which POPEN is not executed), it is ignored.

## 6.6.5 External Output to Memory Card

The external output data is saved in the root folder under the file name: date + sequence number.

|                  |   |                              |
|------------------|---|------------------------------|
| Output folder    | : | Root folder                  |
| Output file name | : | yyyymmdd_*.log               |
|                  |   | yyyy: year mm: month dd: day |
|                  |   | **: sequence number (00~99)  |

The external output file saves the output data from POPEN to PCLOS inside the program, or until the program finishes.

A new output data file is created when an external output file with the same date does not exist in the memory card. When an external output file with the same date exists, it is added to the file with the largest sequence number.

When the file size exceeds 20 MB for POPEN, a file with the next available sequence number is created and saved.

A maximum of 100 external output files with the same date can be saved.

However, if a “yyyymmdd\_99.log” file already exists when creating a file, the alarm <<External output file cannot be created.>> is triggered and the file cannot be saved.

Even if there is an available number in the middle of the sequence, it is skipped over and the next biggest number is used. As a result, even when the maximum (100) is not exceeded, the same alarm is triggered and the file cannot be saved.

If there is not enough space available on the memory card and the file cannot be saved, the alarm <<Memory overflow (Memory card)>> is triggered.

The data that is output to the external output file can be opened using software that supports the following file formats.

Data output using BPRNT command: Binary format

Data output using DPRNT command: Text format

**(Note)** The output data is added even if the external output file attributes that exist in the memory card are changed to reading only.

## 6.6.6 External Output to FTP Server

The data that is output externally is saved under the file name (character string + date + sequence number) specified in the communication parameter <External output - FTP output name> and is saved to the folder specified in the communication parameter <External output - FTP output destination>.

|                     |                                                                                                      |
|---------------------|------------------------------------------------------------------------------------------------------|
| Output folder:      | Folder specified in the communication parameter <External output – FTP output destination>           |
| Output folder name: | SSSSSyyyymmdd_****.log                                                                               |
|                     | SSSSS: Character string specified in the communication parameter <External output – FTP output name> |
|                     | yyyy: Year                                                                                           |
|                     | mm: Month                                                                                            |
|                     | dd: Day                                                                                              |
|                     | ****: Sequence number (0000 to 9999)                                                                 |

The external output file uses the APPE command to save the output data from POPEN to PCLOS inside the program, or until the program finishes.

**(Note 1)** A server that supports the APPE command needs to be specified for the server used for saving. When the server does not support APPE commands, even if a new file is created successfully the first time, thereafter it cannot be created from the second time onward. When using an FTP server for the first time, execute and check the operation two times to make sure it is working properly.

**(Note 2)** This machine cannot be specified as a server for saving.

A new output data file is created when an external output file with the same date does not exist in the <Output folder>. When an external output file with the same date exists, it is added to the file with the largest sequence number.

When the file size exceeds 20 MB for POPEN, a file with the next available sequence number is created and saved.

A maximum of 10000 external output files with the same date can be saved.

However, if an “SSSSSyyyymmdd\_9999.log” file already exists when creating a file, the alarm <<External output file cannot be created.>> is triggered and the file cannot be saved.

Even if there is an available number in the middle of the sequence, it is skipped over and the next biggest number is used. As a result, even when the maximum (10000) is not exceeded, the same alarm is triggered and the file cannot be saved.

**(Note)** When there is a file on the server that has the same name as an external output file but the file names differ only in letter case, then the file may not output correctly due to the server configuration.

The data that is output to the external output file can be opened using software that supports the following file formats.

Data output using BPRNT command: Binary format

Data output using DPRNT command: Text format

When there are a lot of files inside the output folder, it may take some time to process them. In addition, it may take some time for a reply depending on the server and network configuration. Therefore, we recommend using a high speed server and network when possible. In particular, when the FTP server name is not specified in the IP address, the server name will take a few seconds to set during POPEN. If the name fails to set in the processing, it will automatically make another attempt, which may take more than 10 seconds to complete.

### 6.6.7 Precautions for External Output Command

- (Note 1) External output commands are used in memory operation (including extended memory operation) of NC language. The alarm <<Macro Command Error>> occurs when external output commands are issued in MDI and tape operation.
- (Note 2) Data is output also in dry run and machine lock.
- (Note 3) The mode cannot be changed when executing an external output command.
- (Note 4) At program restart, external output commands specified before the restart position are also executed.
- (Note 5) Variables that are empty are regarded as 0.
- (Note 6) Up to 10 macro variables may be specified in the output data of one block.
- (Note 7) It is not possible to specify macro variables and the number of significant digits using a macro variable.

```
BPRNT[#100][1];
```

```
DPRNT[#100][1];
```

- (Note 8) If the following operations are performed before the PCLOS command is executed, the communication is blocked without carrying out the PCLOS processing when connected to a <General COMM device>. When connected to an <FTP server> and a PCLOS command is processed, the connection to the FTP server is severed.

When connecting to a <General COMM device>

- [RST] key is pressed
- Operation is reset (except operation reset by M30)

When connecting to a <Memory card>

- [RST] key is pressed
- Operation reset
- Running program is stopped due to alarm (stop level 3 or higher)
- Running program is stopped when alarm (stop level 2 or higher) is triggered

When connected to an <FTP server>

- [RST] key is pressed
- Operation reset
- Running program is stopped due to alarm (stop level 3 or higher)
- Running program is stopped when alarm (stop level 2 or higher) is triggered

- (Note 9) When connected to a <General COMM device>, if the <Communication parameter> <Check DR signal> is set to <1: Yes>, the DR signal is checked between the POPEN command until the PCLOS command. The alarm <<DR signal off>> occurs when DR signals are turned off during this period.
- (Note 10) When connected to a <General COMM device>, a <Memory card> or an <FTP server> and another setting is configured, the alarm <<Connected to wrong device>> is triggered when the POPEN command is executed.

## 6.7 Interrupt Macro (Option)

Interrupt macro is a function to suspend the program currently being executed on detecting an interrupt signal (external input signal: UINT) and execute the commanded program. A program that is executed by interruption is called interrupt program.

Command format

**M96 P\_;**

P : Interrupt program No.

Use the command below to cancel an interruption.

Command format

**M97;**

The following conditions are required to use interrupt macros:

- The system is in the memory operation mode
- The system is in automatic operation (external output: STL signals ON)
- Not executing an interrupt macro

Any interrupt signal that is received under conditions other than the above is ignored.

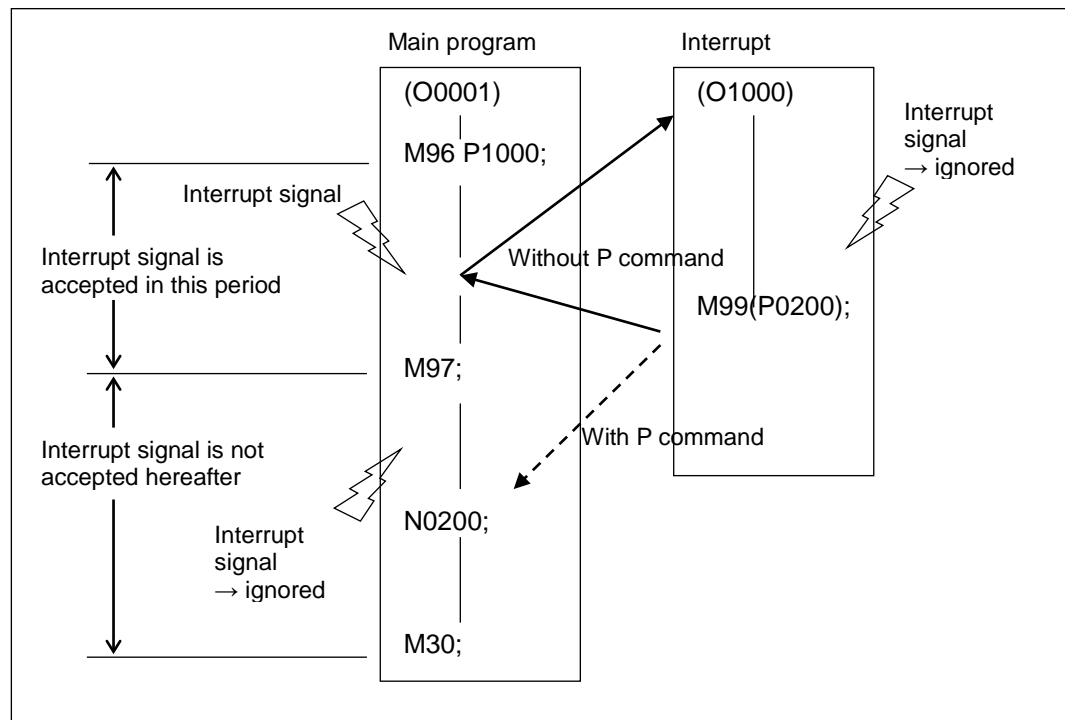
Subprograms and macros can be called from interrupt programs.

Insert M99 command to return from an interrupt macro to the main program. The sequence number in the main program to return to can be specified using P address.

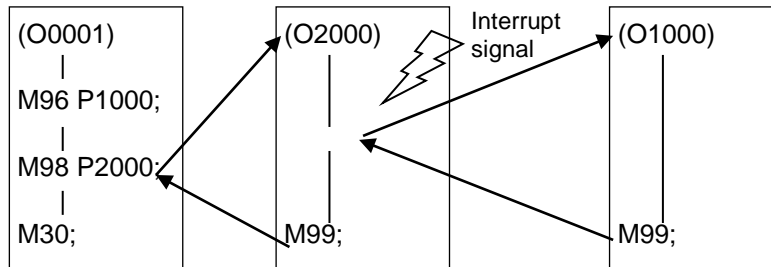
In addition to use of M97, the following operations cancel interruption:

- Operation reset
- Commanding M02 (M30)

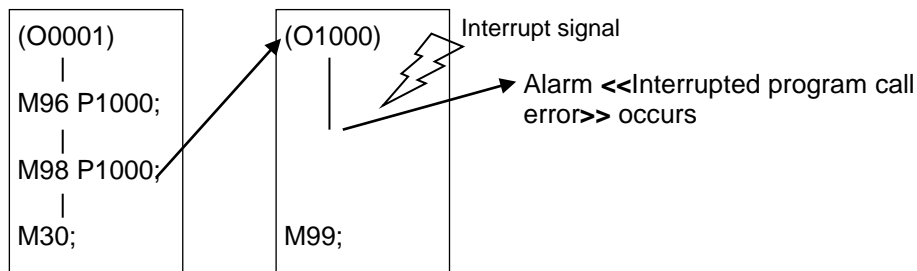
M97 is effectively commanded in an interrupt program or in a program called from an interrupt program.



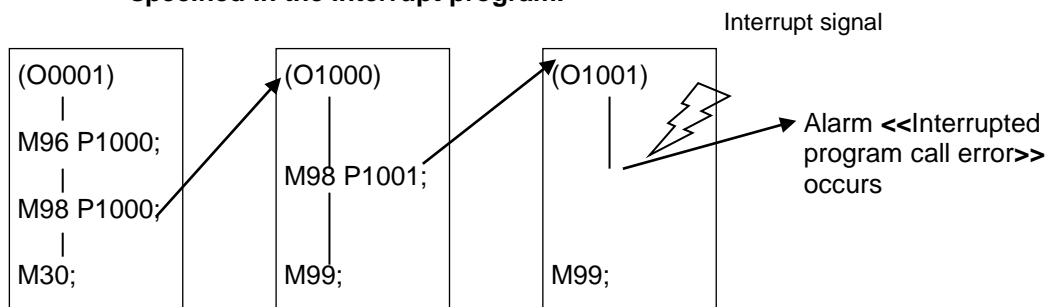
- (Note 1) The alarm <<Invalid command>> occurs when other G / M code is commanded in the blocks of M96 / M97.  
Axes will not move if you command X, Y, and Z axes.
- (Note 2) If P command is absent in an M96 command block, the <<Subprogram number error>> occurs. If a non-existent program number is designated, <<No subprogram>> occurs.
- (Note 3) When an interrupt signal is detected during execution of a subprogram, the interrupt program is called.



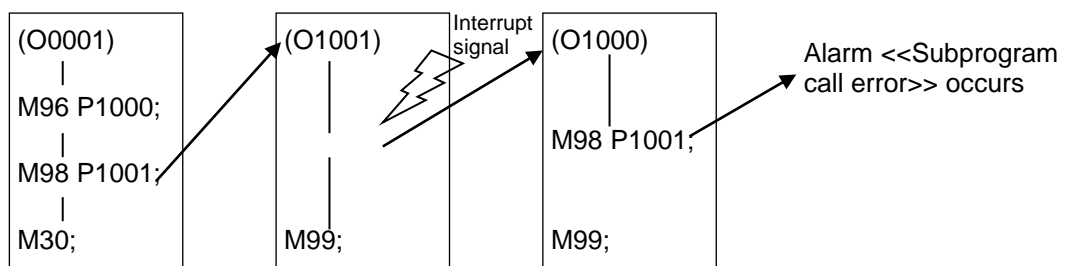
- (Note 4) The alarm <<Interrupted program call error>> occurs at the time of calling an interrupt program when an interrupt signal is detected during execution by G65/G66/M98 of subprogram specified as interrupt program.



- (Note 5) The alarm <<Interrupted program call error>> occurs at the time of calling an interrupt program when an interrupt signal is detected during execution of a subprogram which parent program is the subprogram specified in the interrupt program.



- (Note 6) The alarm <<Subprogram call error>> occurs when calling the parent program from an interrupt program which has been called during execution of a subprogram.





- (Note 7) When the total program size that is loaded (including the interrupt program size) exceeds the size that is set in the user parameter (switch 1) <Program load size>, the machine operates in the extended memory operation mode. In this mode, the operation may stop for a time period equivalent to the time required for loading if the size of the interrupt program is large.
- (Note 8) You may specify the interrupt program number using macro variables. Refer to 8.3 Simple Call Function, Chapter 8 Subprogram Function.

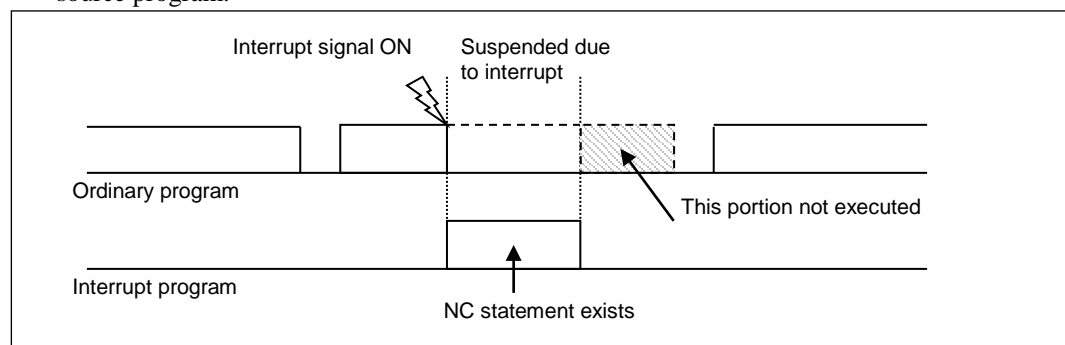
### 6.7.1 Interrupt Type

<Interrupt type macro interrupt system> of <User parameter> is used to set how to interrupt a block currently being executed on detecting an interruption signal.

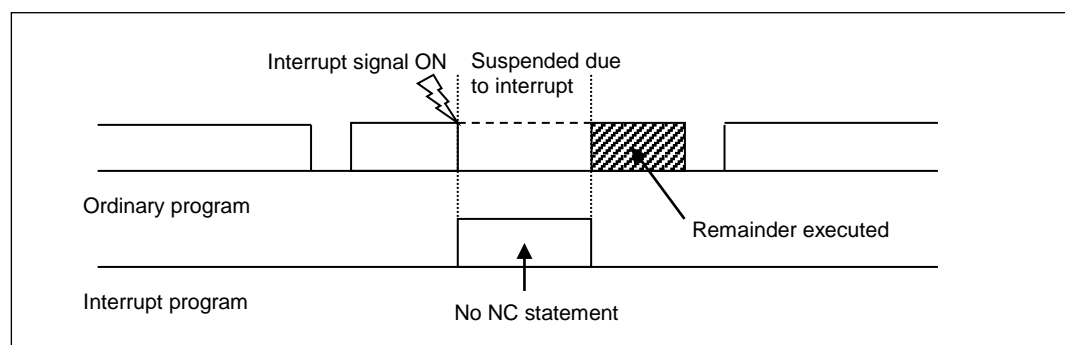
#### 1. Type 1: Interrupt by suspending the execution

The currently being executed block is suspended on detecting an interrupt signal, and the interrupt program starts immediately.

When NC statements are included in the interrupt program or subprogram / macro program called from the interrupt program, the commands included in the interrupted block are lost, and the source program re-starts from the block next to the affected block on returning to the source program.



In the absence of NC statements, the commands in the suspended block continue on returning to the source program.



Refer to “6.7.6.Macro Statement and NC Statement” for the detailed description of NC statement.

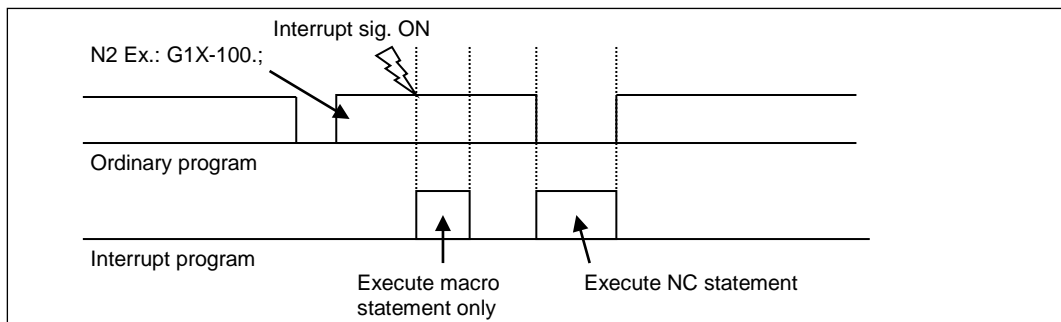
#### 2. Type 2: Interrupt without suspending the execution

The interrupt program is executed on detecting an interrupt signal without suspending the currently executed block.

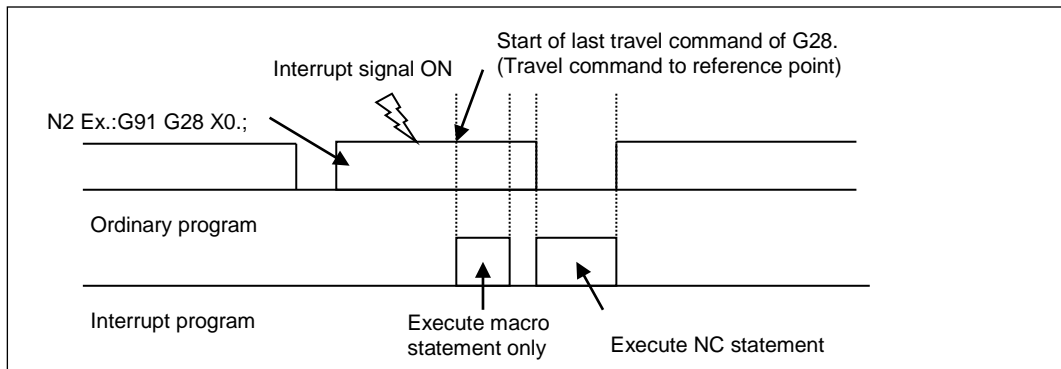
Macro statements up to the first NC statement in the interrupt program or subprogram / macro program called from the interrupt program are processed in parallel with the currently executed block.

Timing of start of processing of macro statements may differ depending on the contents of the currently executed block.

- Currently executed block is a single operation command:  
Macro statement is executed immediately.



- Currently executed block is a multiple operation command:  
Macro statement is executed simultaneously with the last travel command of the cycle operation.



Multiple operation commands are listed below. All other commands are single operation commands.

- Reference point return (G28/G29/G30)
- Canned cycle (G73 to G89, G177 to G189)
- Tool replacement canned cycle (G100/M06)
- Coordinate calculation function (G36 to G39)

For both types of interruption, blocks after the first NC statement are executed after completion of the currently executed block.

Refer to 6.7.6 Macro Statement and NC Statement for the detailed description of NC and macro statements.

**(Note)** Operation of “Type 2: Interrupt without suspending the execution” applies, irrespective of the value of <Interrupt type macro interrupt system>, when the following operations are currently executed:

- Reference point return (G28/G29/G30)
- Canned cycle (G73 to G89, G177 to G189)
- Tool replacement canned cycle (G100/M06)
- Pallet turn
- Coordinate calculation function (G36 to G39)
- Automatic workpiece measurement (G120 to G129)
- Tap torsion direction change (G133/G134)

### 6.7.2 Call Type

There are two types of calling an interrupt program in interrupt macros. One of them is selected in the <Interrupt type macro call system> of <User parameter>.

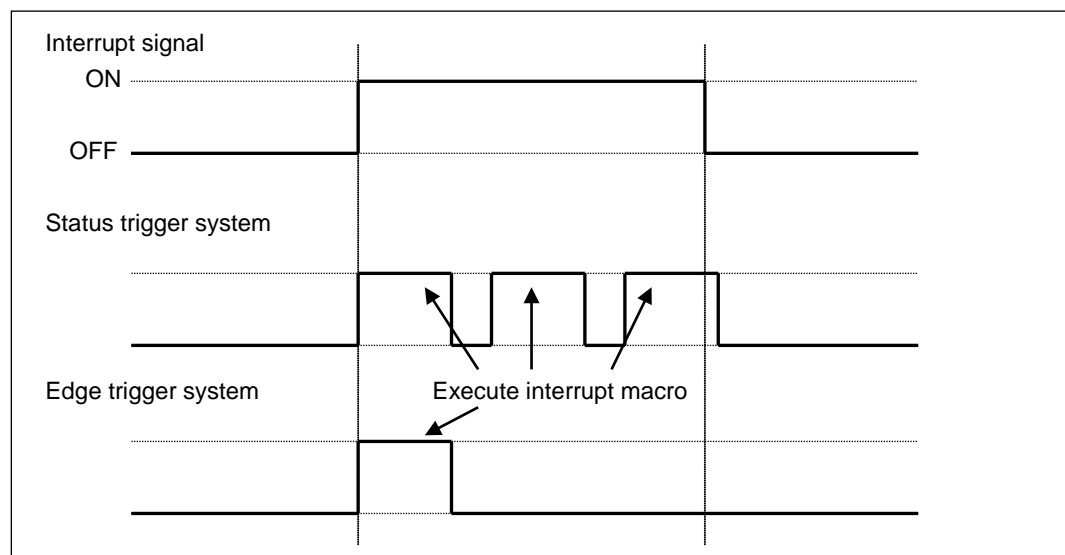
1. Subprogram type interruption  
Interrupt programs are called as subprogram. The level of local variables will not change before and after interruption.
2. Macro type interruption  
Interrupt programs are called as macro program. The level of local variables will change before and after interruption. It is not possible, however, to deliver arguments from the program currently being executed. All local variables immediately after interruption are cleared to <empty.>

In either type of call, the call does not increase multiplicity of subprogram / macro call. Subprogram / macro call conducted in an interrupt program increases respective multiplicity.

### 6.7.3 Acceptance Type

There are two interrupt signal acceptance types. One of them is set with the <Interrupt type macro reception system> of <User parameter>.

1. Type 1: Status trigger system  
Signals are accepted when interrupt signal is ON.  
An interrupt program is executed when the interrupt signal is ON at the time when the interrupt macro becomes valid with M96.  
An interrupt program can be repeatedly executed if you keep the interrupt signal ON steadily.
2. Type 2: Edge trigger system  
Signals are accepted only at the timing of interrupt signal rising from OFF to ON. Interrupt program is not executed even when the interrupt signal is ON at the time when an interrupt macro becomes valid with M96.

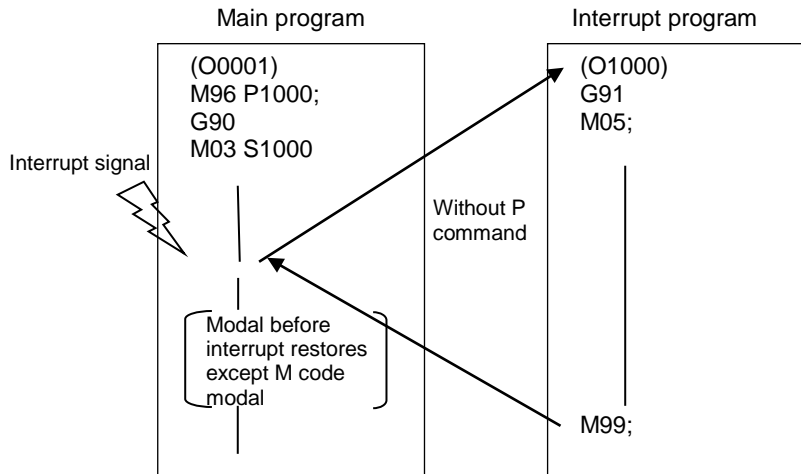


### 6.7.4 Interrupt Macro and Modal Information

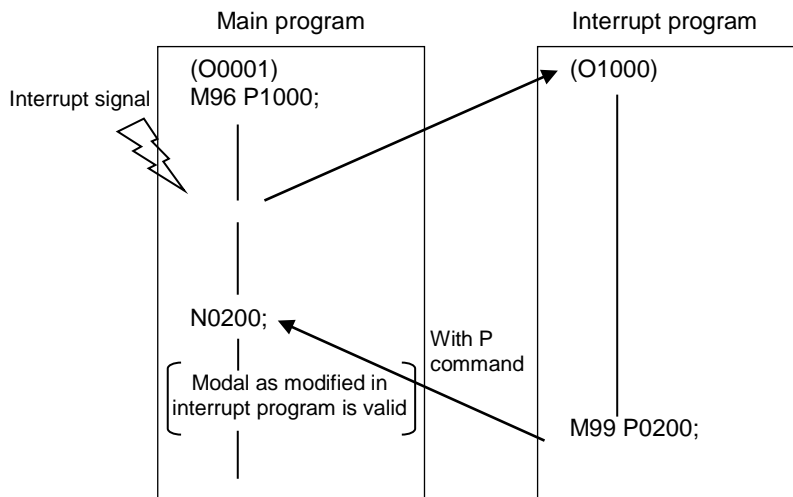
The modal information, changed in the interrupt program, is differently transferred to the source program depending on how the control leaves the interrupt program.

1. Returning with M99 (sequence number not specified)  
Modal information except M code (G/T/H/D/S/F codes) before interruption is valid. Modal information modified in the interrupt program turns invalid.

M code modal information, modified in the interrupt program, continues to be valid as it is modified.

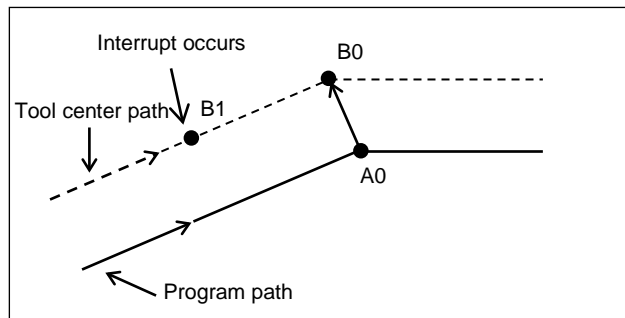


2. Returning with M99PXXXX (with a sequence number specified)  
Modal information, modified in the interrupt program, is valid as it is modified. You can reference the modal information used before interruption using system variables #4401 to #4530.



### 6.7.5 Interrupt Macro and Current Position

The values of macro variables indicating the current position at execution of an interrupt program are shown below.



| Variable No.   | Contents                                       | Conditions of reference                                           | Coordinate values                          |
|----------------|------------------------------------------------|-------------------------------------------------------------------|--------------------------------------------|
| #5001 to #5008 | End point coordinates                          | From interruption to the first NC statement                       | It is undefined. Do not use it.            |
|                |                                                | After start of NC statement that does not include travel commands | Coordinates of B1 position                 |
|                |                                                | After start of NC statement that include travel commands          | Coordinates of end point of travel command |
| #5021 to #5034 | Current position (Machine coordinate system)   |                                                                   | Machine coordinates of B1 position         |
| #5041 to #5054 | Current position (Workpiece coordinate system) |                                                                   | Workpiece coordinates of B1 position       |

Refer to the “6.7.6 Macro Statement and NC Statement” for the detailed description of NC statement.

### 6.7.6 Macro Statement and NC Statement

Blocks satisfying the conditions below are macro statement. Those that do not satisfy them are NC statement.

- Includes calculation command (Refer to “6.3 Calculation function”) (\*)
- Includes control command (Refer to “6.4 Control function”) (\*)
- Includes external output command (Refer to “6.6 External output function”) (\*)
- Includes macro simple call command (G65)
- Includes subprogram call command (M98)
- Includes a return command (M99) from macro program / subprogram

\* Only when <User parameter> <Macro command single stop> is selected to <0: No>.

Blocks that do not conduct Single Block Stop in single operation may be said macro statement and those that do conduct Single Block Stop NC statement.

### 6.7.7 Restrictions

1. The alarm <<Specified G code cannot be used>> occurs when interrupting in the programmable mirror image (G51.1), rotational transformation (G68), or scaling (G51) mode and commanding the same programmable mirror image (G51.1), rotational transformation (G68), or scaling (G51) again in the interrupt program.
2. When <Interrupt macro interrupt type> is set to type 2 and macro program modal call (G66) and interrupt type macro (M96) are modals, if an interrupt signal is detected during a travel axis command, the interrupt type macro (M96) is given priority and called. The registered macro program will not be called in the modal call (G66).
3. Macro modal call (G66) is canceled on calling an interrupt program. G66 must be commanded in the interrupt program if you wish to perform macro modal call in the interrupt program.
4. The interrupt program is executed after finishing search when an interrupt signal is detected during search of sequence number of GOTO instruction, END instruction of WHILE – END, and sequence number of M98Hxxxx and M99Pxxxx.
5. Interrupt signals, which are detected during search of the block specified by program restart (Sequence Search), are ignored.
6. Interrupt signals, which are detected during search of the block specified by program restart (Restart) and during return operation, are ignored.
7. When an interrupt signal is detected during a dwell, the interrupt program is executed after the dwell.
8. When an interrupt signal is detected while waiting for ON/OFF of BCD signal output and M signals of M460, etc., the interrupt program is executed after detecting ON/OFF of the waiting signal.
9. The alarm << High accuracy A (or B) invalid command >> occurs when commanding M96 in the high accuracy mode. The alarm << High accuracy A (or B) invalid modal >> occurs when commanding a high accuracy mode (M260 / M261 / M262 / M265) in the M96 modal.
10. When an NC statement is included in the interrupt program or subprogram / macro program called from an interrupt program, the M99 command block for returning from the interrupt program to the original program will perform single block stop in the case of single operation.
11. When the door opens for safety reasons, the interrupt type macro function does not enable. Refer to “Door Interlock” in the Operation Manual (Operation) for further details on door open definition.
12. The following operations do not apply: mirror image, scaling, rotational transformation, cutter compensation and nose R compensation, when performing axis travel for an interrupt program, and within a sub program / macro program that is called from an interrupt program.
13. It returns to the original path from the second travel operation after the return operation when an interrupt occurs during a cutter compensation operation and nose R compensation operation and when there is an NC statement in the interrupt program, and within a sub program / macro program that is called from an interrupt program.

**(Note) During cutter compensation when an interference check is being carried out for nose R compensation, the interference check starts again at the place where it returns to the original path.**

14. When the item <Interrupt type macro interrupt system> is set to <0: Type 1> (interrupts by cancelling execution), if travel for tool length offset and for tool position compensation is lost due to the interruption, compensation will not be executed until the next Z-axis travel operation, and until the next X-, Y- and Z-axes travel operation.

15. Even if the following keys are pressed: **[MANU]**, **[MDI]** and **[EDIT]** between the time when the interrupt signal is detected until the end when the interrupt program is executed with M99, the alarm <<Interrupt type macro execution is being prepared>> is displayed and the mode cannot be changed.
16. The alarm <<Interrupt type macro post-process being executed>> occurs and the mode will not change if you press **[MANU]**, **[MDI]**, or **[EDIT]** key during the time between completion of interrupt program and return to the original program.
17. When restoring from the interrupt program during extension memory operation/tape operation, the sequence number cannot be specified.  
If an instruction is given, an <<Invalid command>> alarm will occur.
18. If an interrupt occurs during incremental mode (G91) and an NC sentence exists in the interrupt program or a sub-program or macro program which is called from the interrupt program, the first movement command given after the reset will be treated as an incremental command from the coordinates where the reset occurred.
19. When the command M96 is issued inside the following sub program or macro program, the alarm <<Interrupt type macro command not possible>> is triggered. If extension or tape operation is used, the alarm <<Specified M code cannot be used>> is triggered.
  - Macro program that is called using G code (or M code).
  - Sub program / macro program that is called using the command M98, G65 or G66 from a macro program that is called using G code (or M code).
20. When an interrupt signal (UINT) is detected inside the macro program that is called using G code (or M code), it interrupts and executed the program that is specified in the M96 command.
21. The alarm <<Feature coordinate manufacturing mode engaged>> is triggered when an M96 command is issued while in feature coordinate manufacturing mode. In addition, the alarm <<Feature coordinate command error>> is triggered when a feature coordinate setting command (G68.2) is issued during M96 modal.

### 6.7.8 Reference Folder When Interrupt Type Macro is Executed

The folder that is referenced when executing an interrupt type macro is as follows depending on the operation type.

| Operation type            | Reference folder                |
|---------------------------|---------------------------------|
| Internal memory operation | Same folder as the main program |
| Tape operation            | Root folder in the NC memory    |

**(Note 1)** The folder is the same one that is referenced when executing the M98 command.

**(Note 2)** The folders above are referenced as well when an interrupt signal (UINT) is detected inside the macro program that is called using G code (or M code).

(This page was intentionally left blank.)



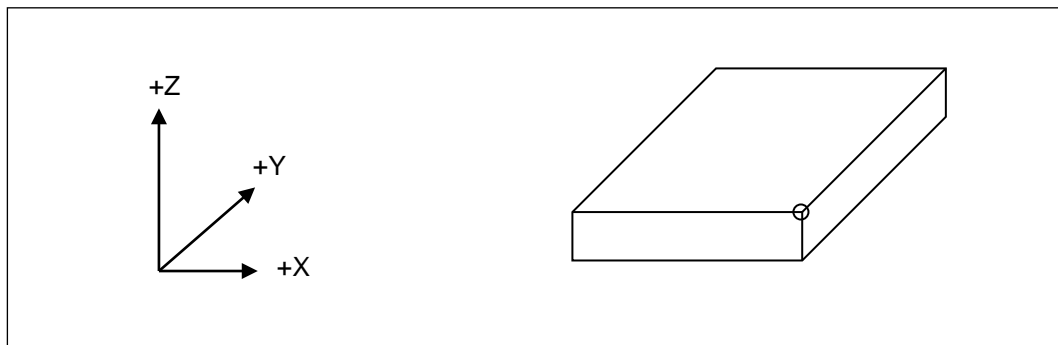
## CHAPTER 7

# AUTOMATIC WORKPIECE MEASUREMENT

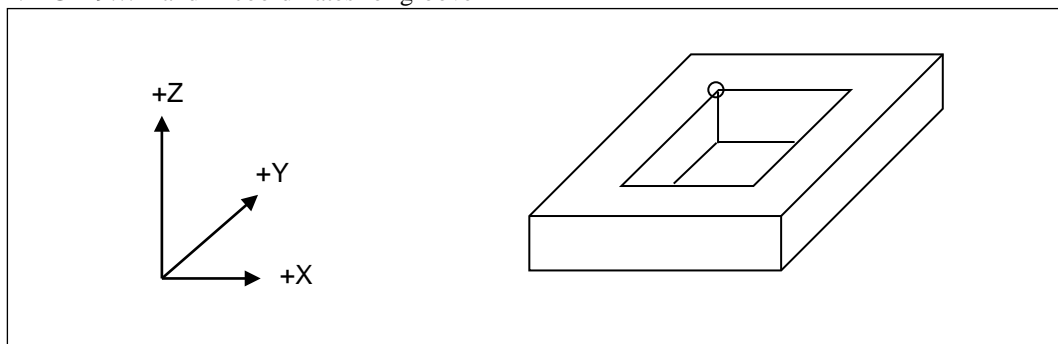
- 7.1 List of Automatic Workpiece Measurement Functions
- 7.2 Before Automatic Workpiece Measurement
- 7.3 Setting Data for Automatic Workpiece Measurement
- 7.4 Command Procedure for Automatic Workpiece Measurement
- 7.5 Measurement Results Processing
- 7.6 Lock Key Operations
- 7.7 Program Restart Operation

## 7.1 List of Automatic Workpiece Measurement Functions

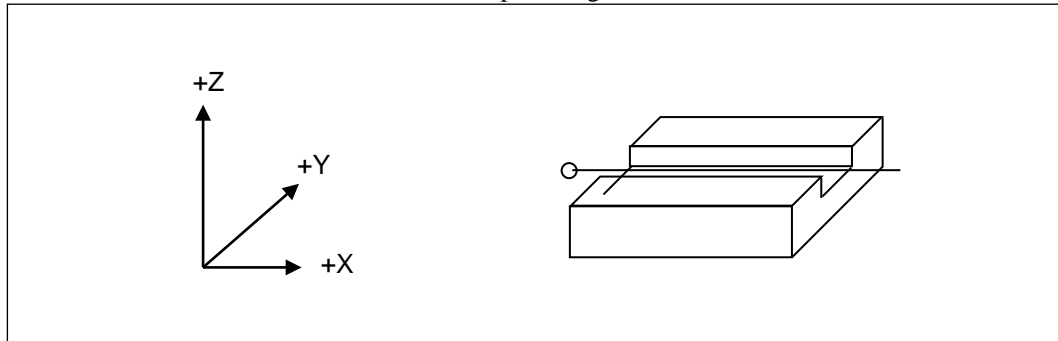
1. G121...X and Y coordinates for corner



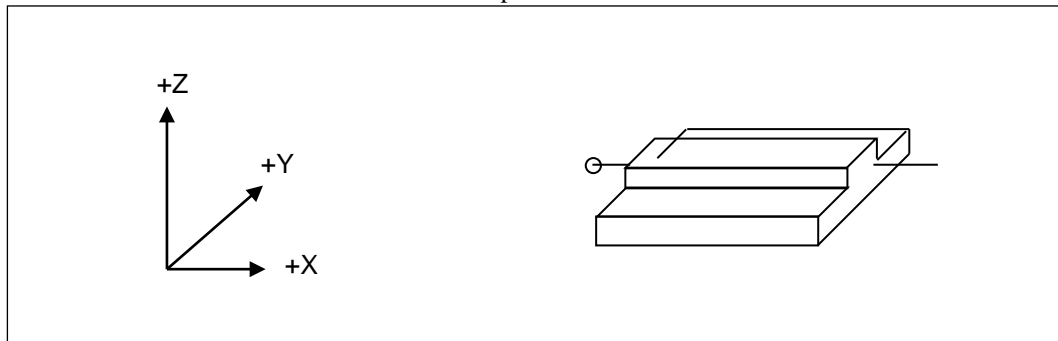
2. G129...X and Y coordinates for groove



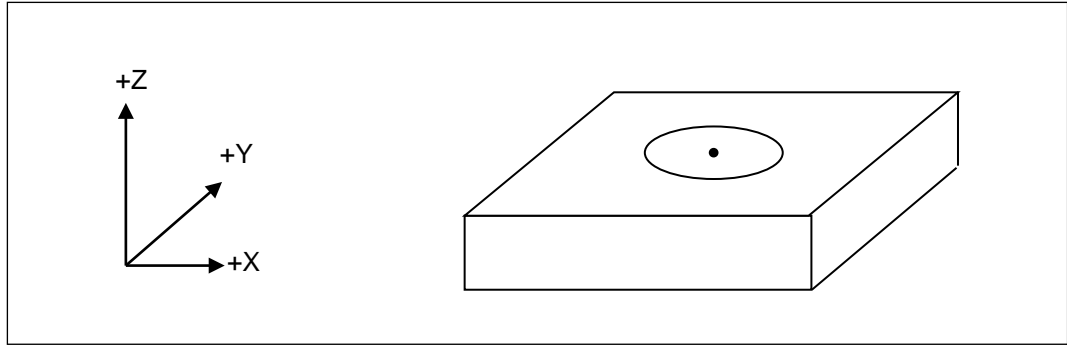
3. G122...X and Y coordinates for center of parallel groove



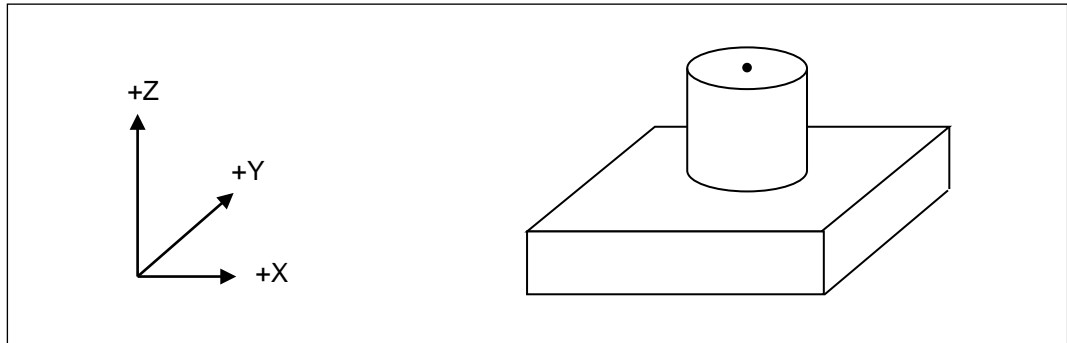
4. G123...X and Y coordinates for center of parallel boss



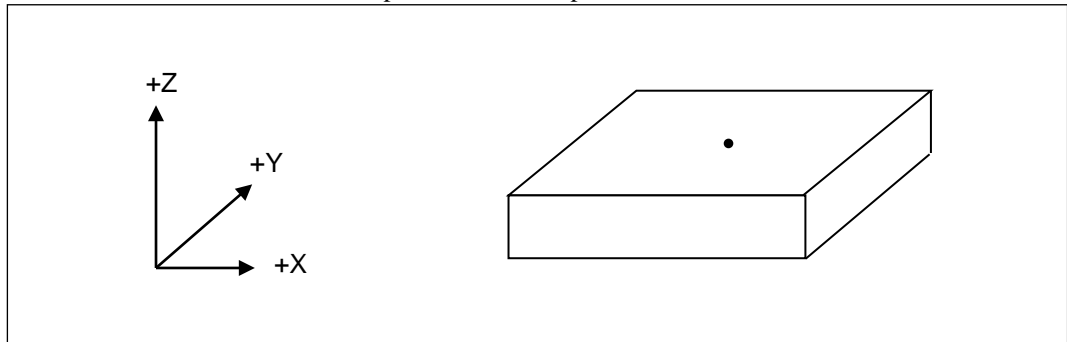
5. G124 and G126...X and Y coordinates for center of hole



6. G125 and G127...X and Y coordinates for center of circular boss



7. G128...Z-axis coordinate on top surface of workpiece



## 7.2 Before Automatic Workpiece Measurement

Set the following item in the user parameter 1 (switch 1).

| Item name                     | Setting range              | Description                                                                                                                                                                                                                                                                                                                                                                                                                           |
|-------------------------------|----------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Probe detection signal method | 0: Method 1<br>1: Method 2 | Set the accuracy of the detection signal that is obtained from the touch probe.<br>Even if the <1: Method 2> increases the travel speed of the probe more than <0: Method 1>, there is less variation in the measurement values. As a result, the speed can be set higher than <Automatic centering speed 2> and <Measurement speed 2>. However, <1: Method 2> cannot be set due to the machine structure, and an alarm is triggered. |

In addition, set the necessary parameters in the user parameters (automatic workpiece measurement).

If the parameters are not correctly set, the probe can become damaged.

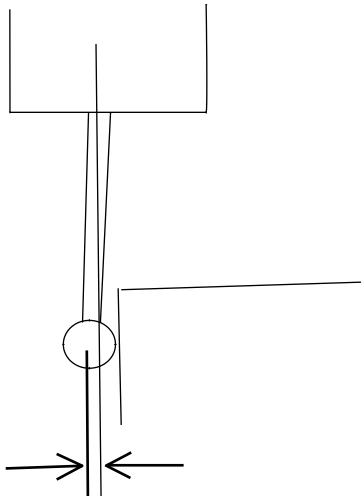
**(Note)** The commands G121 to G129 cannot be issued when using the lathe spindle in the following situations. After switching modals, issue the command.

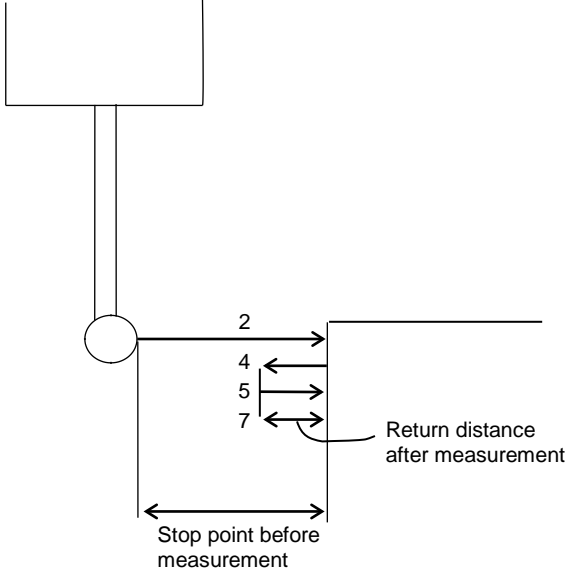
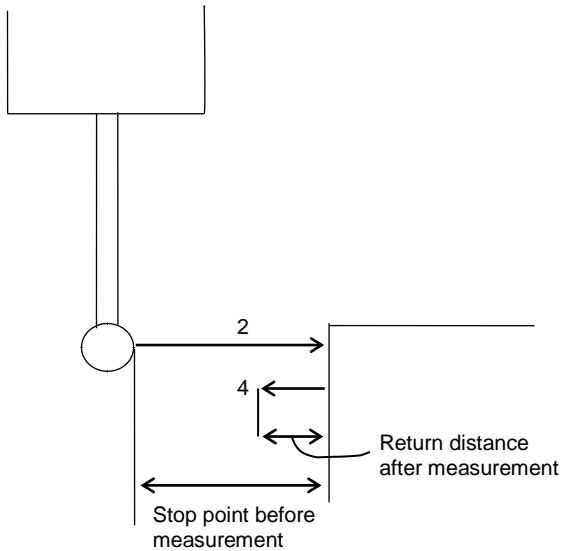
- M142 modal in progress
- G143/G144 modal in progress

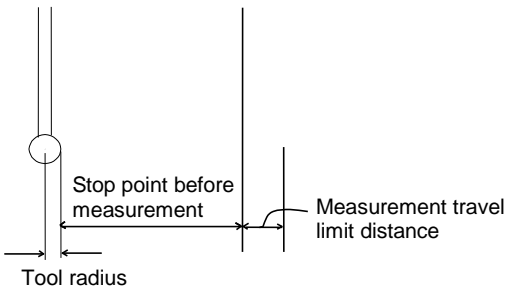
The commands G121 to G129 cannot be issued while in feature coordinate manufacturing mode (G68.2 modal in progress).

## 7.3 Setting Data for Automatic Workpiece Measurement

Description of the user parameters (Automatic workpiece measurement)

| Item                                             | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                           |
|--------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <Probe compensation 1><br><Probe compensation 2> | <p>Set the difference between the center of the stylus tip ball and the center of the spindle when the detection signal turns ON with the touch probe attached to the spindle. The X value of the difference corresponds to &lt;Probe compensation 1&gt; and the Y value corresponds to &lt;Probe compensation 2&gt;. We recommend performing the automatic setting on the automatic centering - &lt;Probe compensation (corner)&gt; screen in MDI mode.</p>  <p>Setting range     -9.999 to 9.999 mm<br/>                         -0.9999 to 0.9999 inch<br/>(When using Type 1 for the minimum unit setting)</p> |
| <Probe compensation 3><br><Probe compensation 4> | <p>Set the difference between the center of the circle, calculated by 3-point measurement (G124 and G125), and the center of the actual circle. The X value of the difference corresponds to &lt;Probe compensation 3&gt; and the Y value corresponds to &lt;Probe compensation 4&gt;. We recommend performing the automatic setting on the automatic centering - &lt;Probe compensation (3-point circle)&gt; screen in MDI mode.</p> <p>Setting range -9.999 to 9.999 mm<br/>                         -0.9999 to 0.9999 inch<br/>(When using Type 1 for the minimum unit setting)</p>                                                                                                                |

| Item                                                                                | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                          |
|-------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <p>&lt;Measurement motion&gt;</p> <p>&lt;0: Type 1&gt;</p> <p>&lt;1: Type 2&gt;</p> | <p>&lt;0: Type 1&gt;</p> <ol style="list-style-type: none"> <li>1. Detection signal OFF check</li> <li>2. Travels at measurement speed 1 in specified axis direction</li> <li>3. Axis travel stops when detection signal turns ON</li> <li>4. Returns to front traveling the return distance</li> <li>5. Travels at measurement speed 2 in specified axis direction</li> <li>6. Axis travel stops when detection signal turns ON</li> <li>7. Returns to position in Step 4</li> <li>8. Detection signal OFF check</li> </ol>  <p>&lt;1: Type 2&gt;</p> <ol style="list-style-type: none"> <li>1. Detection signal OFF check</li> <li>2. Travels at measurement speed 2 in specified axis direction</li> <li>3. Axis travel stops when detection signal turns ON</li> <li>4. Returns to front traveling the return distance</li> <li>5. Detection signal OFF check</li> </ol>  |

| Item                            | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                    |
|---------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <Measurement speed 1>           | <p>Set the first measurement speed for Type 1 measurement motion.</p> <ul style="list-style-type: none"> <li>Relief amount of probe = L (mm)</li> <li>Skip feed time constant 1 = t (msec)</li> <li>Measurement speed 1 = F<sub>1</sub> (mm/min)</li> <li>Control system delay = t<sub>d</sub> (msec) = 4 (msec)</li> <li>Delay unique to probe = t<sub>p</sub> (msec)</li> </ul> $L \geq \frac{(F_1 \times t_d)}{(60 \times 1000)} + \frac{(F_1 \times t / 2)}{(60 \times 1000)} + \frac{(F_1 \times t_p)}{(60 \times 1000)}$ <p style="text-align: center;">↑<br/>Overtravel amount due to delay in control system<br/>↑<br/>Overtravel amount<br/>↑<br/>Overtravel amount due to delay unique to probe</p> <div style="border: 1px solid black; padding: 5px; width: fit-content; margin: 10px auto;"> <math display="block">F_1 &lt; (120000L \div (8+t+2 \times t_p)) \div 1.2</math> <p style="text-align: center;">↑<br/>Safety factor</p> </div> <p>Setting range 1 to 5000 mm/min<br/>0.1 to 196.8 inch/min<br/>(When using Type 1 for the minimum unit setting)</p>                  |
| <Measurement speed 2>           | <p>Set the second measurement speed for Type 1 measurement motion and the measurement speed for Type 2 measurement motion.</p> <ul style="list-style-type: none"> <li>Tolerance in control system = E (μm)</li> <li>Measurement speed 2 = F<sub>2</sub> (mm/min)</li> <li>Control system delay = t<sub>d</sub> (msec)</li> </ul> <p>(When probe detection signal method = 0: Method 1)<br/>t<sub>d</sub> (msec) = 0.300 (msec)<br/>(When probe detection signal method = 1: Method 2)<br/>t<sub>d</sub> (msec) = 0.085 (msec)</p> <ul style="list-style-type: none"> <li>Margin of error unique to probe = e<sub>p</sub> (mm)</li> </ul> $E \geq (F_2 \times t_d) \div (60 \times 1000) + e_p$ <p style="text-align: center;">↓</p> <div style="border: 1px solid black; padding: 5px; width: fit-content; margin: 10px auto;"> <math display="block">F_2 \leq 60000 \times (E - e_p) \div t_d</math> </div> <p>Setting range 1 to 5000 mm/min<br/>0.1 to 196.8 inch/min<br/>(When using Type 1 for the minimum unit setting)</p>                                                              |
| <Stop point before measurement> | <p>Set the distance between the end of the probe at the measurement start point and the predicted value on the workpiece surface.</p> <p>The alarm &lt;&lt;Detection signal off&gt;&gt; is triggered if it travels from this point and skips over the &lt;&lt;Stop point before measurement&gt;&gt; + &lt;&lt;Measurement travel limit distance&gt;&gt;.</p> <p>Setting range 0.000 to 99.999 mm<br/>0.0000 to 9.9999 inch<br/>(When using Type 1 for the minimum unit setting)</p>  <p>The diagram illustrates the relationship between the tool radius, the stop point before measurement, and the measurement travel limit distance. A vertical line represents the tool's path. A circle represents the tool's cross-section. The distance from the tool's center to the stop point is labeled 'Tool radius'. The distance from the stop point to the measurement travel limit distance is labeled 'Measurement travel limit distance'. The stop point is labeled 'Stop point before measurement'.</p> |

| Item                                | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                |
|-------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <Measurement travel limit distance> | Set the overtravel when the measurement skip operation exceeds the predicted value (programmed value).<br>Setting range 0.000 to 99.999 mm<br>0.0000 to 9.9999 inch<br>(When using Type 1 for the minimum unit setting)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                    |
| <Measurement tolerance 1>           | The alarm <<Large error in measured value (1)>> is triggered if the difference between the measurement result and the predicted value (programmed value) exceeds this setting. When 0 is set, the error checking is not performed.<br><br>Setting range 0.000 to 99.999 mm<br>0.0000 to 9.9999 inch<br>(When using Type 1 for the minimum unit setting)                                                                                                                                                                                                                                                                                                                                                                                                    |
| <Measurement tolerance 2>           | The alarm <<Large error in measured value (2)>> is triggered if the difference between the current measurement result and the previous measurement result exceeds this setting. When the following situation applies, the error checking is not performed. <ul style="list-style-type: none"> <li>• When 0 is set</li> <li>• When the first automatic workpiece measurement motion occurs after power is turned ON (1st motion for measurement results 1 through 4)</li> <li>• When there is no measurement result for previous measurement</li> <li>• When the G code for this command is different from the previous command</li> </ul><br>Setting range 0.000 to 99.999 mm<br>0.0000 to 9.9999 inch<br>(When using Type 1 for the minimum unit setting) |
| <Return distance after measurement> | Set the return distance from the position where the measurement probe makes contact with the workpiece. <ul style="list-style-type: none"> <li>• Return distance = <math>L_b</math> (mm)</li> <li>• Skip feed time constant 1 = <math>t</math> (msec)</li> <li>• Measurement speed 2 = <math>F_2</math> (mm/min)</li> </ul><br>$L_b \geq \text{MAX}(1.0, F_2 \times t / 60000)$<br>Setting range 0.000 to 99.999 mm<br>0.0000 to 9.9999 inch<br>(When using Type 1 for the minimum unit setting)                                                                                                                                                                                                                                                           |

**(Note)** The set values, such as the measurement speed and return distance after measurement, vary depending on the probe being used. Therefore, contact the probe manufacturer when deciding on the set values.



The travel speed follows G00/G01 modals when the automatic workpiece measurement is being performed, when traveling to the measurement start point and when returning from the measurement point.

G00: Rapid feedrate

G01: F command value

G02/G03/G102/G103/G202/G203: The alarm <<Arc mode>> is triggered.

G02.2/G03.2: The alarm <<Invalid command for involute interpolation modal>> is triggered.

Make sure that no swarf or chips are stuck to the end of the measurement probe or on the measurement surface. In addition, make sure that there is no disturbance (caused by vibrations from outside of the machine) that adversely affects the machine. We cannot guarantee the measurement accuracy when there is swarf or other factors that can cause inaccuracy.

The mode cannot be changed during the automatic workpiece measurement. Attempting to change the mode will trigger the alarm <<During measurement>>.

To avoid damaging the probe as well, press the **[SINGL]** key for the first measurement in the automatic workpiece measurement operation, and check each motion using the single operation.

Perform the automatic workpiece measurement while the tool length offset is engaged.

The tool length offset is cancelled even if the coordinate is set in the machining coordinates using the G53 command.

## 7.4 Command Procedure for Automatic Workpiece Measurement

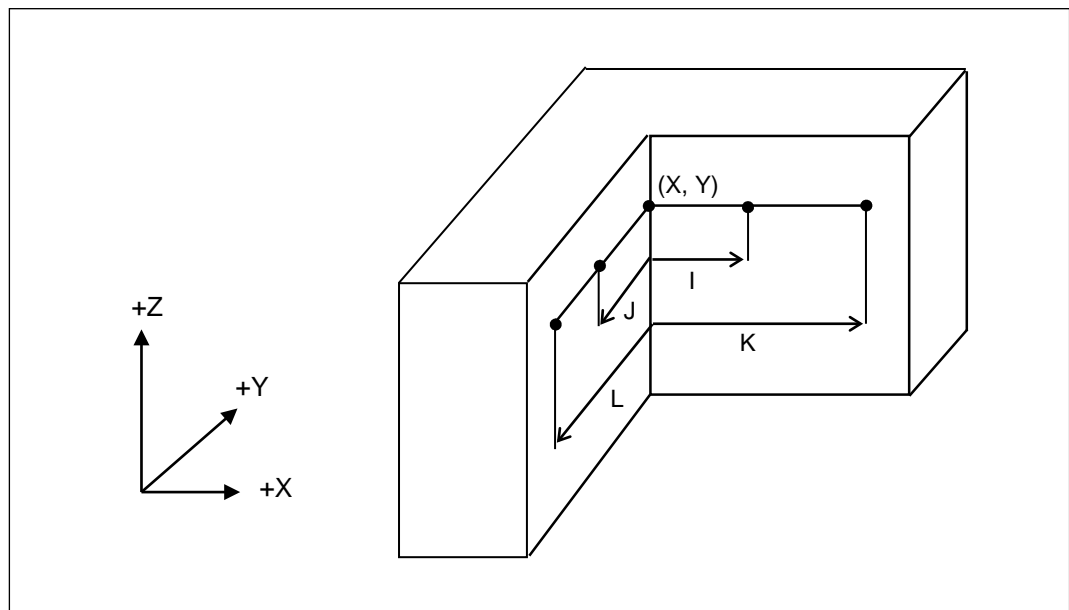
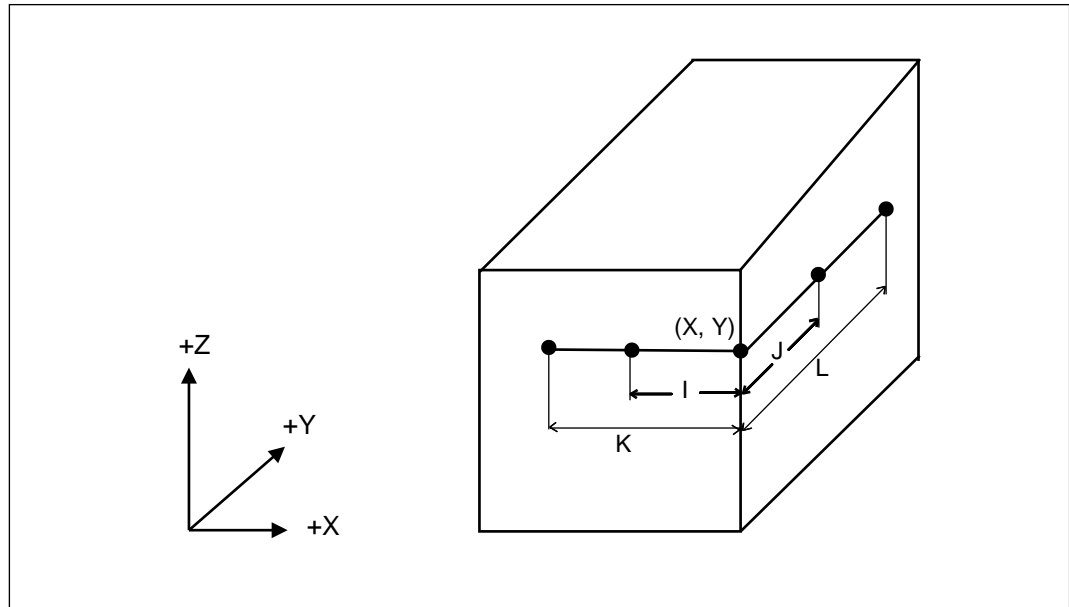
### 7.4.1 Corner

Command format

Boss

Groove

|                                            |
|--------------------------------------------|
| <b>G121</b> X_ Y_ I_ J_ K_ L_ D_ Z_ R_ Q_; |
| <b>G129</b> X_ Y_ I_ J_ K_ L_ D_ Z_ R_ Q_; |



- X and Y : Predicted value for corner
- I and K : X-axis coordinate position when measuring in Y direction. Offset from (X, Y)
- J and L : Y-axis coordinate position when measuring in X direction. Offset from (X, Y)
- D : Cutter compensation No.
- Z : Z coordinate when measuring
- R : When traveling from measurement position to the measurement position, and return height after travel ends
- Q : Register No. stores the measurement result (1 when omitted)

- (Note 1) An alarm is triggered when a J command is not issued or when  $I = 0$  or  $J = 0$ .
- (Note 2) If a workpiece is tilted and a K or L command is issued, the angle of the tilted workpiece (angle used in the rotational transformation) is also calculated.  
The alarm <<Auto workpc. meas. address error>> is triggered when both K and L are specified or when a 0 command is issued.
- (Note 3) Before (G121/G129) is carried out, perform <<Probe comp.(corner)>> for automatic centering and set the compensation for the user parameter <Probe compensation (G121/G129)>.
- (Note 4) The D modal does not change with the D command.

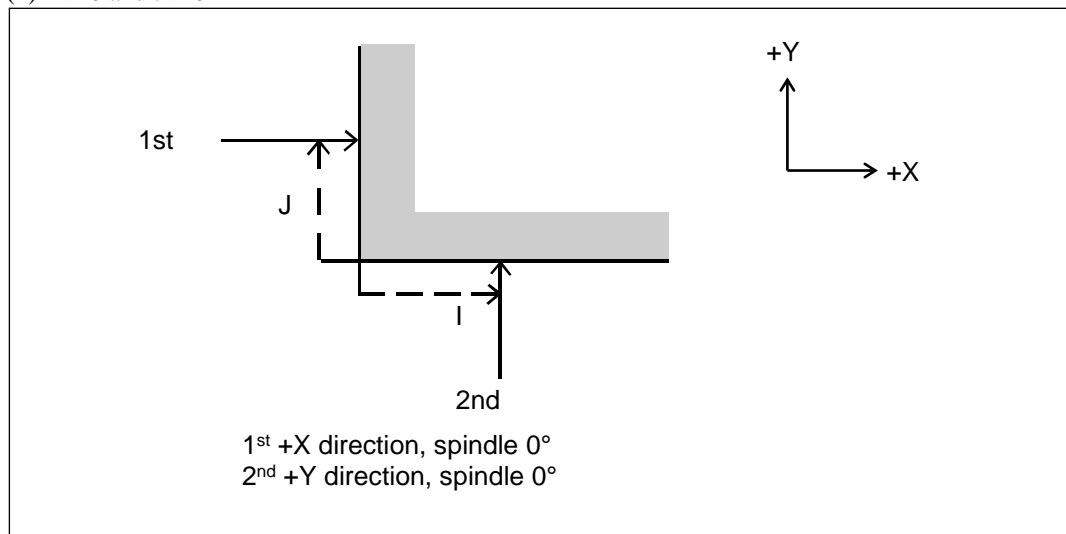
Boss

(Motion)

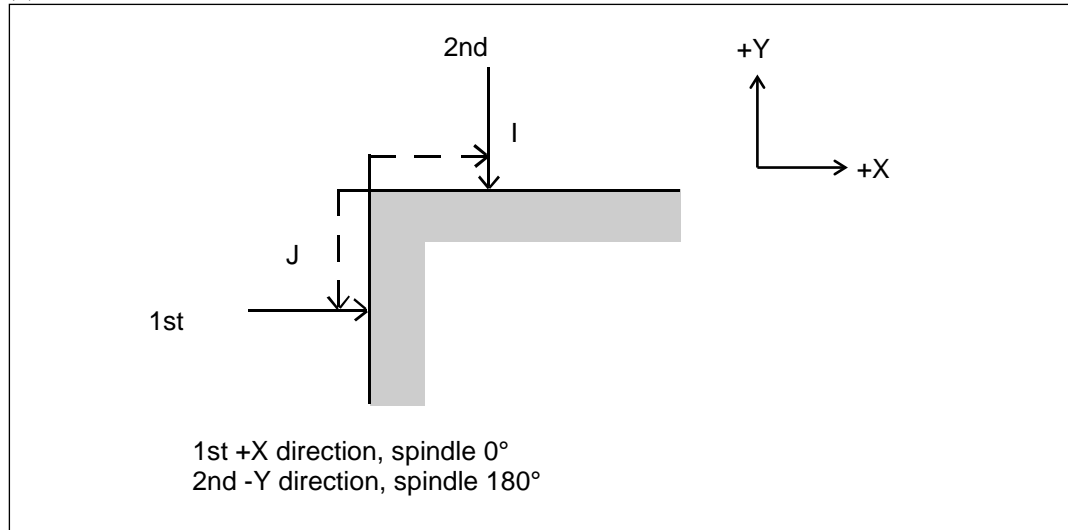
1. Spindle orientation. Travels to the first measurement start position for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement (position J).
4. Measures at position L when there is an L command.
5. Travels to the return height position on the Z-axis.
6. Spindle orientation. Travels to the second measurement start position for the X- and Y-axes.
7. Travels to the measurement height on the Z-axis.
8. Measurement (position I).
9. Measures at position K when there is a K command.
10. Travels to the return height point on the Z-axis.

I and J signs, spindle orientation, and measurement skip travel direction

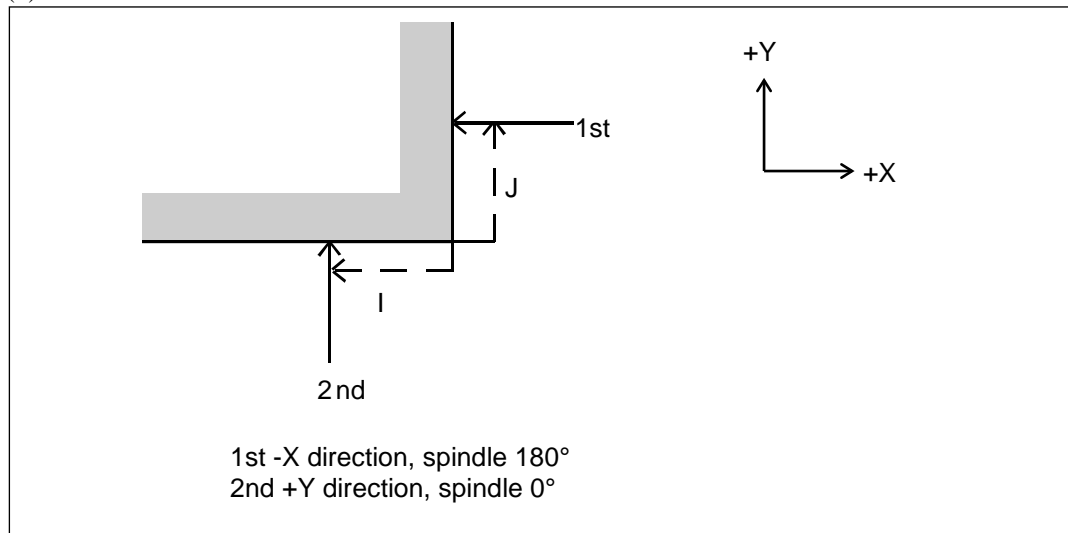
(1)  $I > 0$  and  $J > 0$



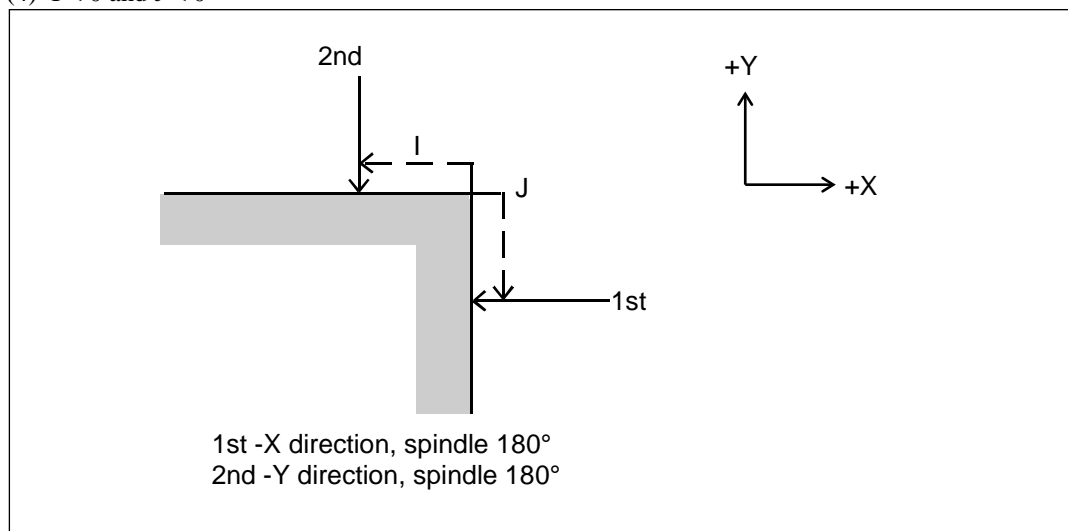
(2)  $I > 0$  and  $J < 0$



(3)  $I < 0$  and  $J > 0$



(4)  $I < 0$  and  $J < 0$

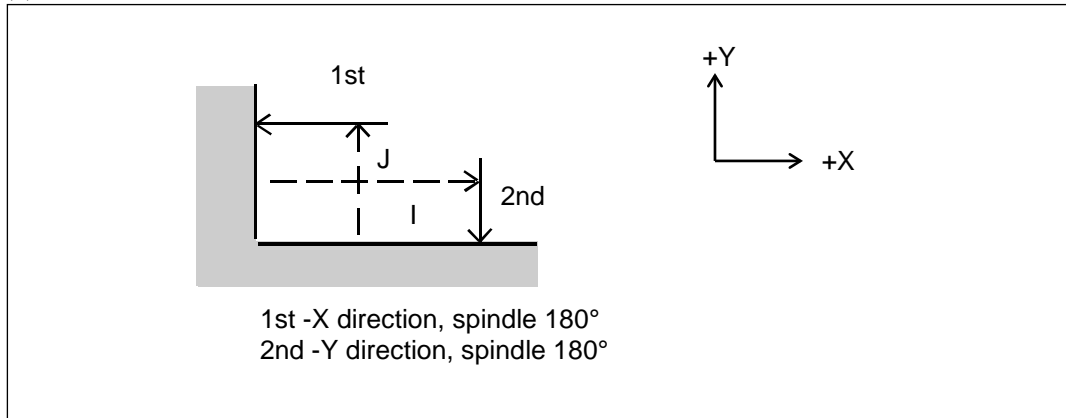


Groove

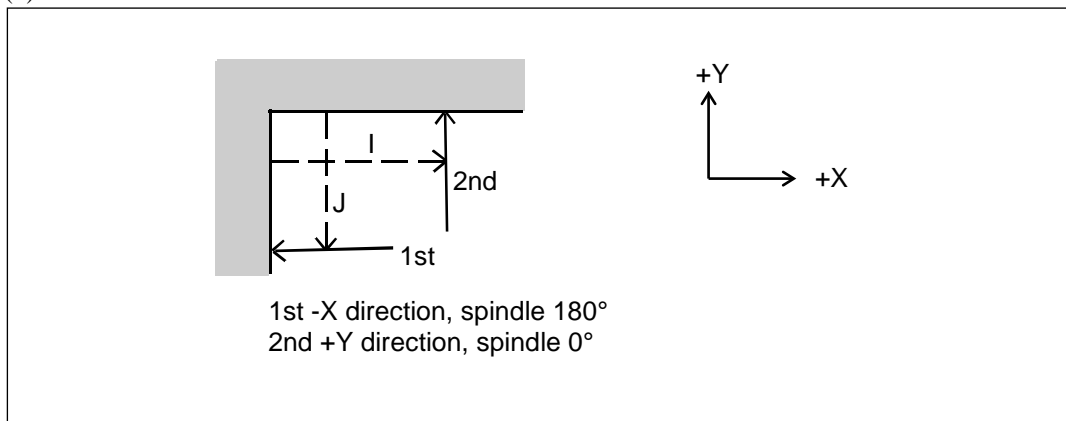
(Motion)

1. Spindle orientation. Travels to the first measurement start position for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement (position J).
4. Measures at position L when there is an L command.
5. Spindle orientation. Travels to the second measurement start position for the X- and Y-axes.
6. Measurement (position I).
7. Measures at position K when there is a K command.
8. Travels to the return height point on the Z-axis.

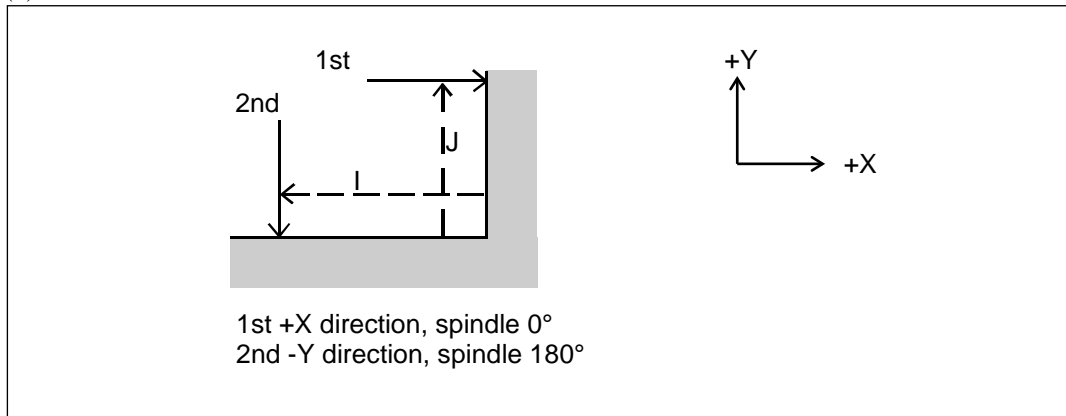
(1)  $I > 0$  and  $J > 0$



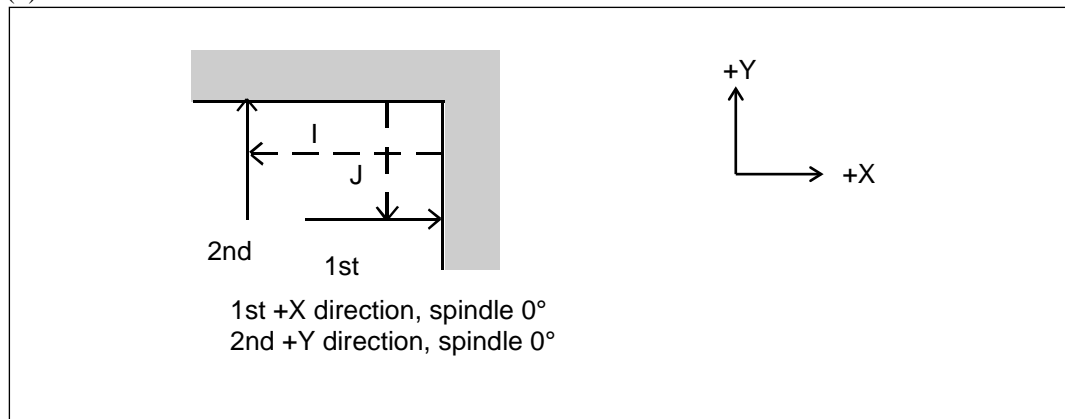
(2)  $I > 0$  and  $J < 0$



(3)  $I < 0$  and  $J > 0$



(4)  $I < 0$  and  $J < 0$



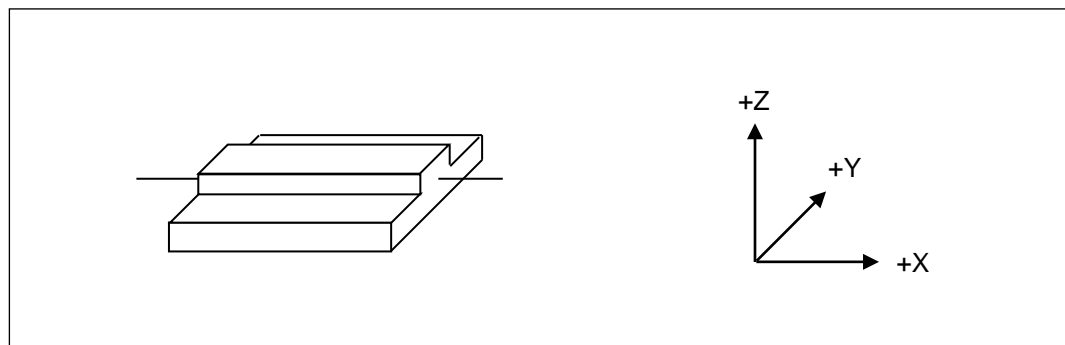
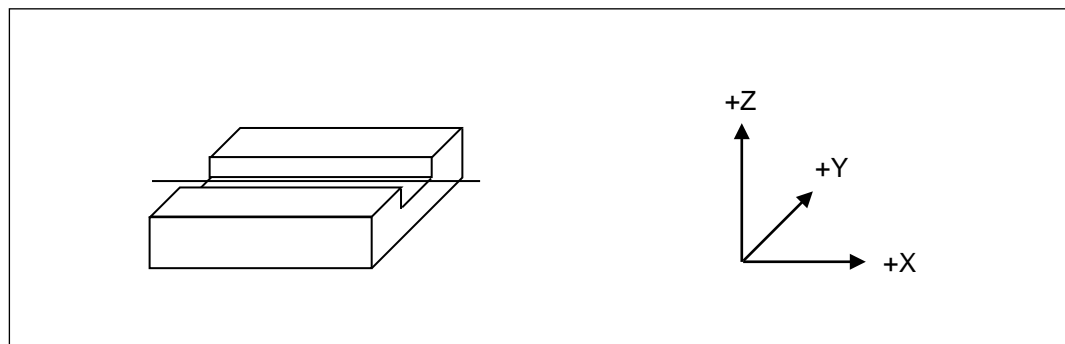
## 7.4.2 Parallel

Command format

Groove

Boss

```
G122 X_ Y_ I_(J_) D_ Z_ R_ Q_;  
G123 X_ Y_ I_(J_) D_ Z_ R_ Q_;
```



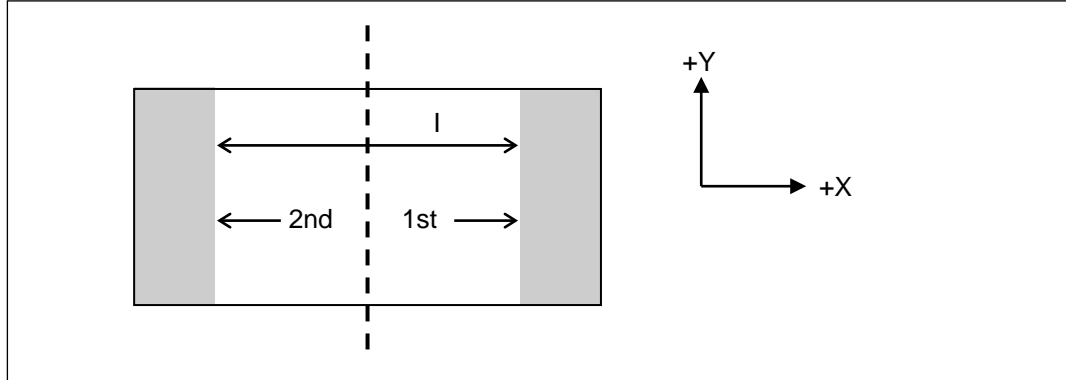
X and Y : Predicted value for groove (boss) center  
I and J : Groove width  
I : Width in X-axis direction  
J : Width in Y-axis direction  
Simultaneous commands for I and J cannot be issued  
D : Cutter compensation No.  
Z : Z coordinate when measuring  
R : Z coordinate for return height position  
Q : Register No. stores the measurement result (1 when omitted)

### Groove

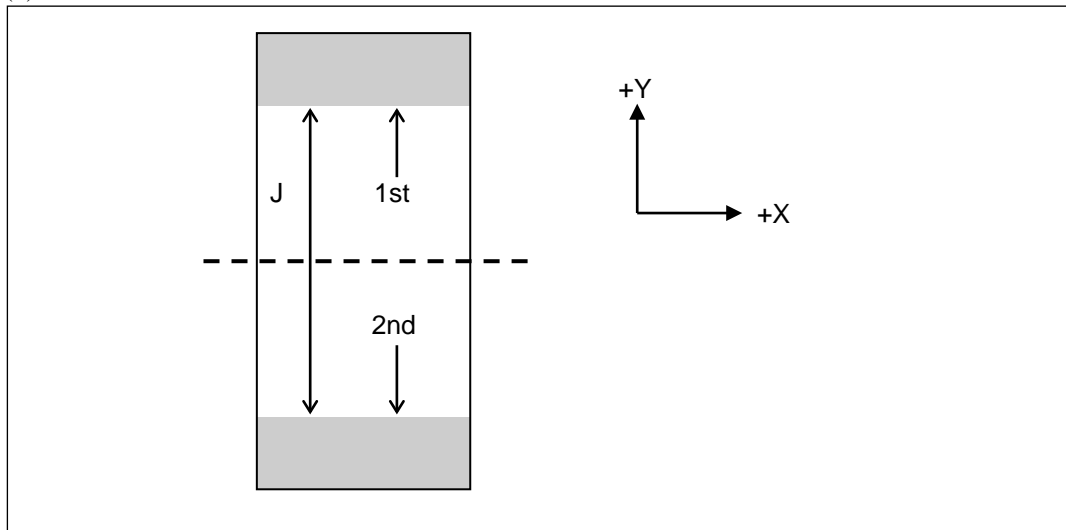
#### (Motion)

1. Spindle orientation  $0^\circ$ . Travels to the first measurement start position for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement.
4. Spindle orientation  $180^\circ$ . Travels to the second measurement start position for the X- and Y-axes.
5. Second measurement.
6. Travels to the return height position on the Z-axis.

#### (1) When command is issued



#### (2) When J command is issued

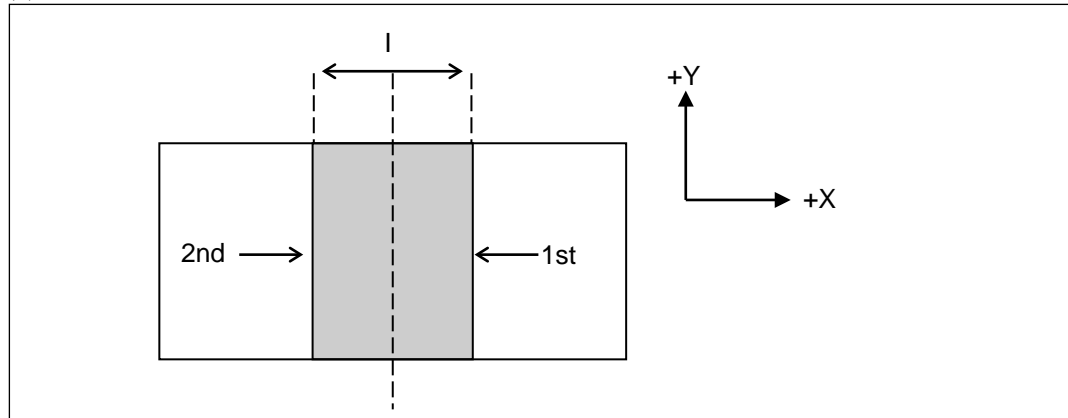


### Boss

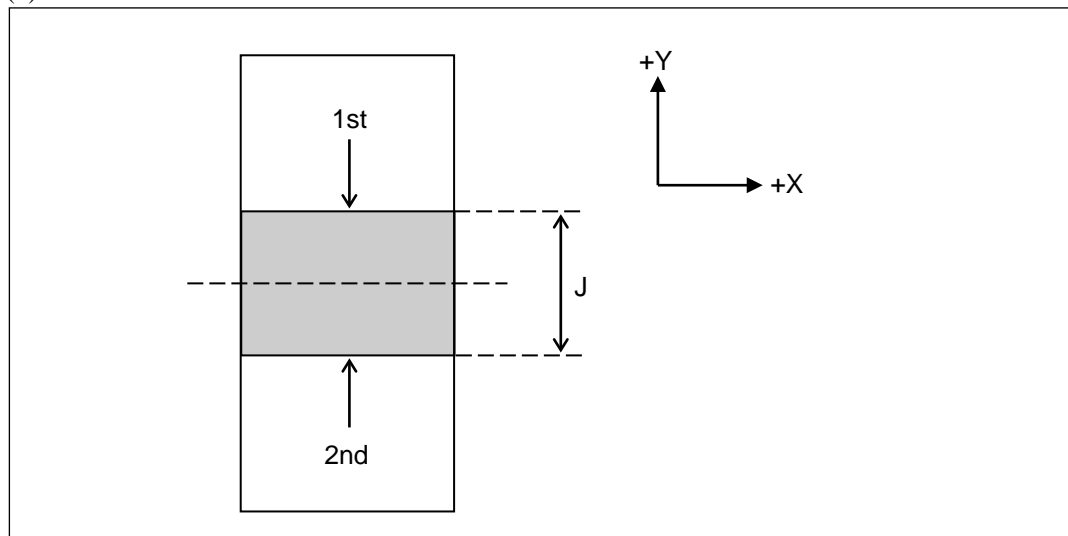
#### (Motion)

1. Spindle orientation  $180^\circ$ . Travels to the first measurement start position for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement.
4. Travels to the return height point on the Z-axis.
5. Spindle orientation  $0^\circ$ . Travels to the second measurement start position for the X- and Y-axes.
6. Travels to the measurement height on the Z-axis.
7. Second measurement.
8. Travels to the return height on the Z-axis.

(3) When I command is issued



(4) When J command is issued



7

## 7.4.3 Circle Center

Command format

3 points are measured to calculate the center of the circle.

Hole  
Boss

```
G124 X_ Y_ I_ D_ Z_ R_ Q_ ;
G125 X_ Y_ I_ D_ Z_ R_ Q_ ;
```

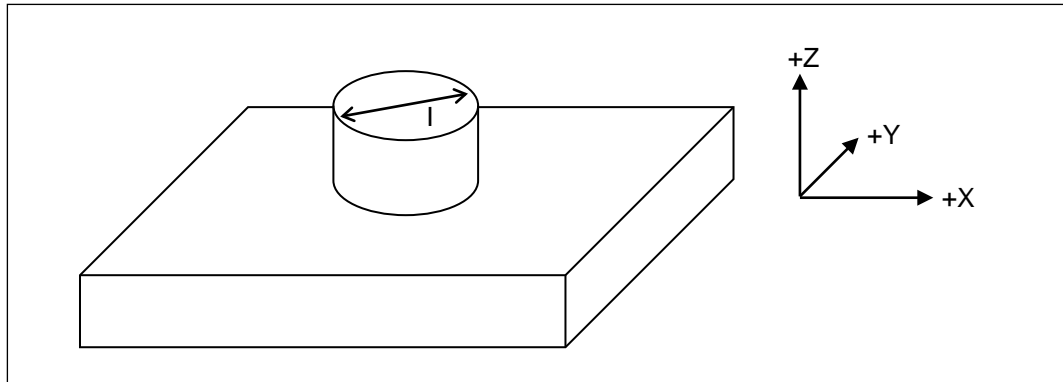
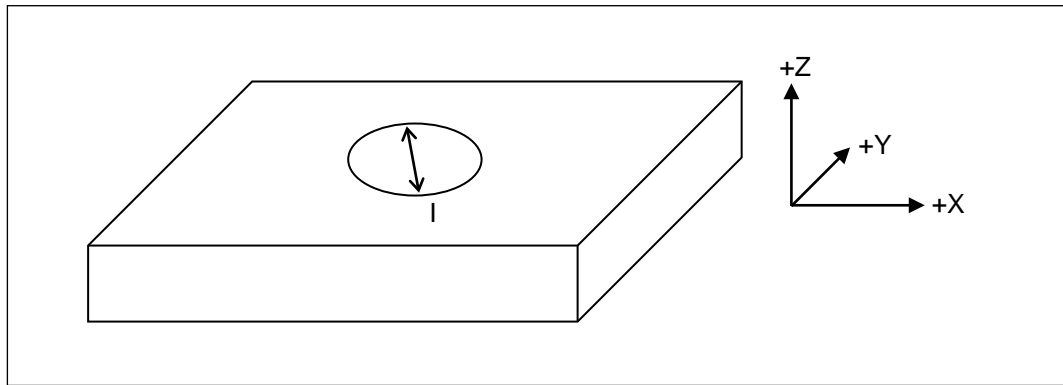
4 points are measured to calculate the center of the circle.

Hole  
Boss

```
G126 X_ Y_ I_ D_ Z_ R_ Q_ ;
G127 X_ Y_ I_ D_ Z_ R_ Q_ ;
```

|           |                                                             |
|-----------|-------------------------------------------------------------|
| X and Y : | Predicted value for hole (boss) center                      |
| I :       | Diameter of circle being measured                           |
| D :       | Cutter compensation No.                                     |
| Z :       | Z coordinate when measuring                                 |
| R :       | Z coordinate for return height position                     |
| Q :       | Register No. stores the measurement result (1 when omitted) |

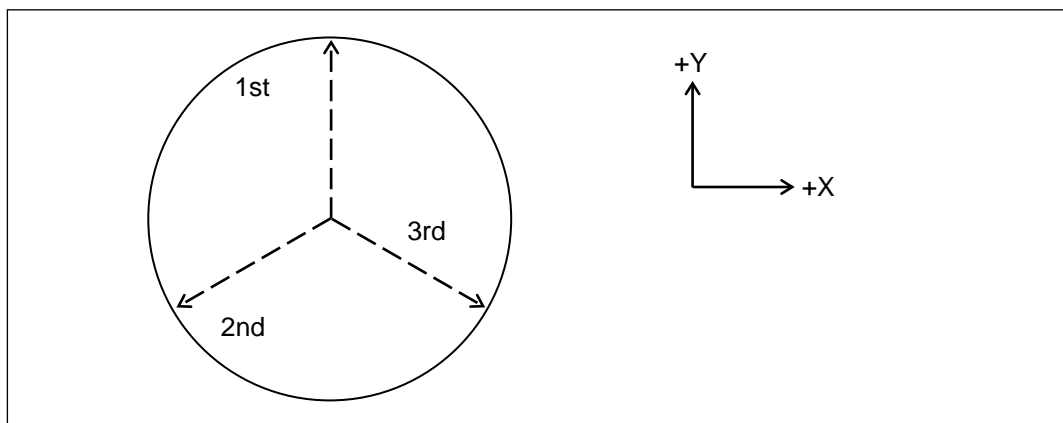




#### Hole and 3-point measurement

##### (Motion)

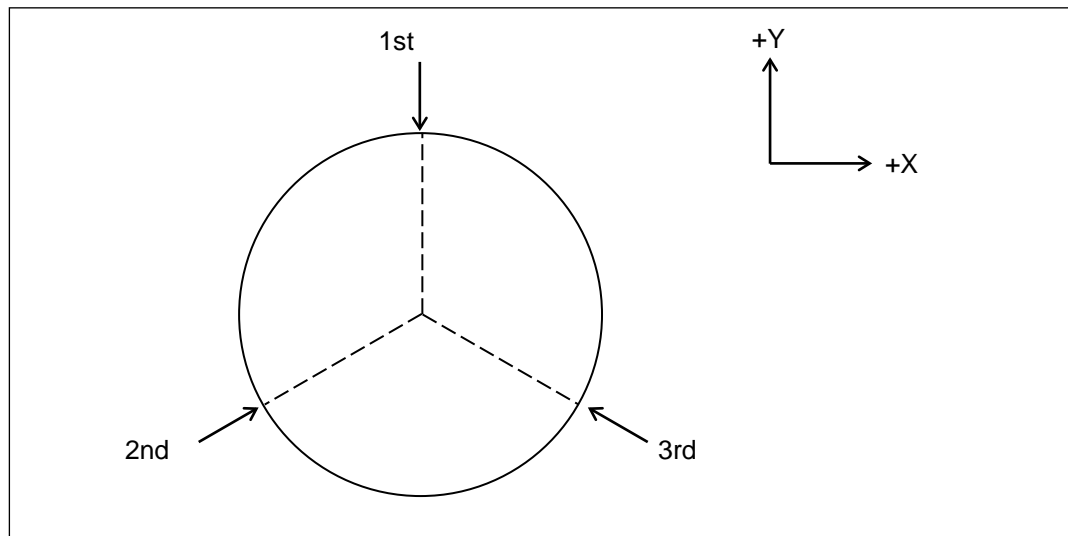
1. Spindle orientation  $0^\circ$ . Travels to the first measurement start position for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement. (Y-axis plus direction)
4. Spindle orientation  $0^\circ$ . Travels to the second measurement start position for the X- and Y-axes.
5. Second measurement. (Direction that forms the  $120^\circ$  angle with the first)
6. Spindle orientation  $0^\circ$ . Travels to the third measurement start position for the X- and Y-axes.
7. Third measurement. (Direction that forms the  $240^\circ$  angle with the first)
8. Travels to the return height point on the Z-axis.



### Boss and 3-point measurement

#### (Motion)

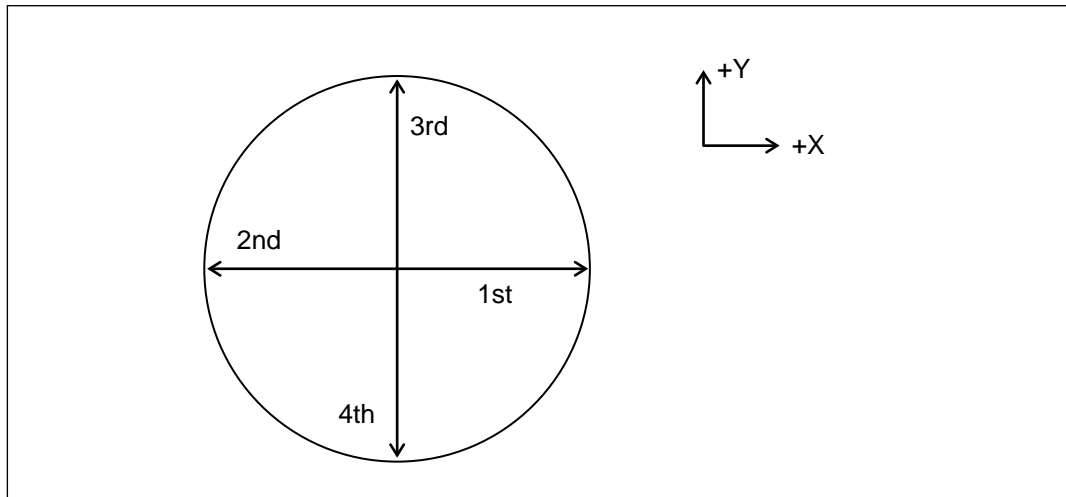
1. Spindle orientation  $180^\circ$ . Travels to the first measurement start value for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement. (Y-axis minus direction)
4. Returns to the return height point on the Z-axis.
5. Spindle orientation  $180^\circ$ . Travels to the second measurement start position for the X- and Y-axes.
6. Travels to the measurement height on the Z-axis.
7. Second measurement. (Direction that forms the  $120^\circ$  angle with the first)
8. Returns to the return height point on the Z-axis.
9. Spindle orientation  $180^\circ$ . Travels to the third measurement start position for the X- and Y-axes.
10. Travels to the measurement height on the Z-axis.
11. Third measurement. (Direction that forms the  $240^\circ$  angle with the first)
12. Returns to the return height point on the Z-axis.



### Hole and 4-point measurement

#### (Motion)

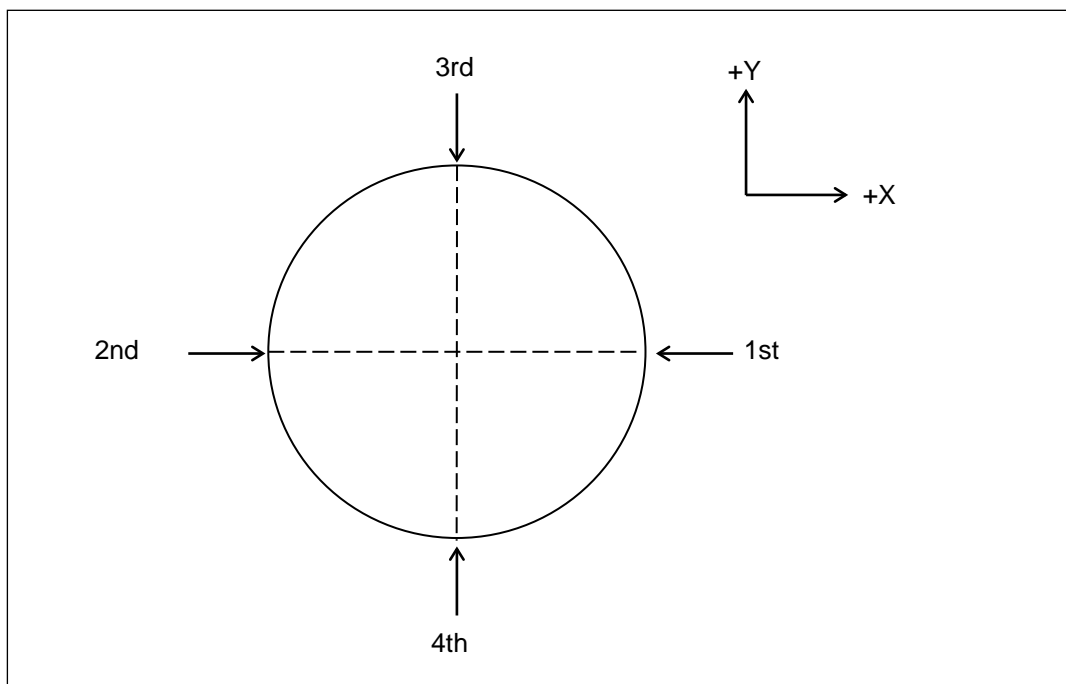
1. Spindle orientation  $0^\circ$ . Travels to the first measurement start position for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement. (X-axis plus direction)
4. Spindle orientation  $180^\circ$ . Travels to the second measurement start position for the X- and Y-axes.
5. Second measurement. (X-axis minus direction)
6. Spindle orientation  $0^\circ$ . Travels to the third measurement start position for the X- and Y-axes.
7. Third measurement. (Y-axis plus direction)
8. Spindle orientation  $180^\circ$ . Travels to the fourth measurement start position for the X- and Y-axes.
9. Fourth measurement. (Y-axis minus direction)
10. Travels to the return height point on the Z-axis.



#### Boss and 4-point measurement

##### (Motion)

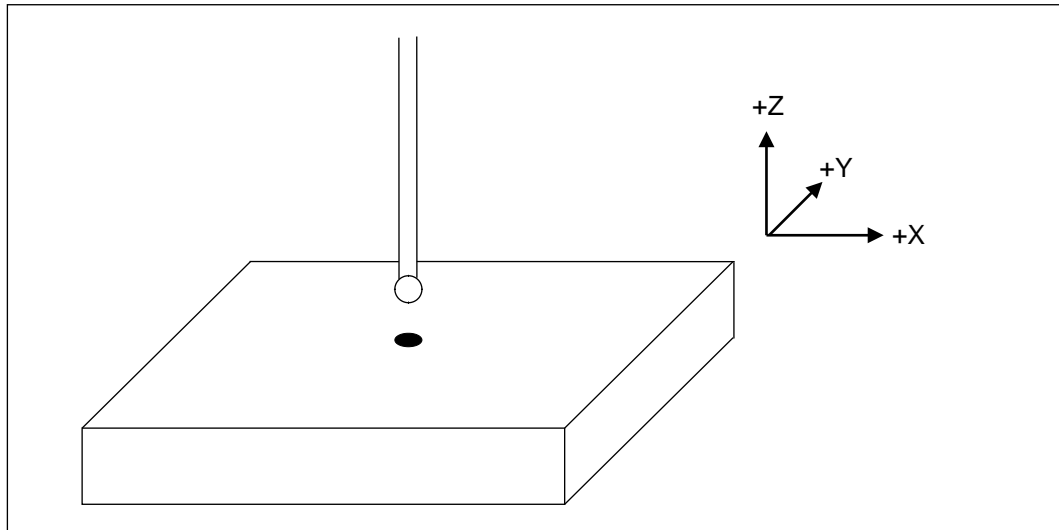
1. Spindle orientation  $180^\circ$ . Travels to the first measurement start value for the X- and Y-axes.
2. Travels to the measurement height on the Z-axis.
3. First measurement. (X-axis minus direction)
4. Returns to the return height point on the Z-axis.
5. Spindle orientation  $0^\circ$ . Travels to the second measurement start position for the X- and Y-axes.
6. Travels to the measurement height on the Z-axis.
7. Second measurement. (X-axis plus direction)
8. Returns to the return height point on the Z-axis.
9. Spindle orientation  $180^\circ$ . Travels to the third measurement start position for the X- and Y-axes.
10. Travels to the measurement height on the Z-axis.
11. Third measurement. (Y-axis minus direction)
12. Returns to the return height point on the Z-axis.
13. Spindle orientation  $0^\circ$ . Travels to the fourth measurement start position for the X- and Y-axes.
14. Travels to the measurement height on the Z-axis.
15. Fourth measurement. (Y-axis plus direction)
16. Returns to the return height point on the Z-axis.



### 7.4.4 Z-axis Height

Command format **G128 X\_ Y\_ Z\_ Q\_;**

X and Y : X and Y coordinates at measurement point  
Z : Predicted value for Z-axis height



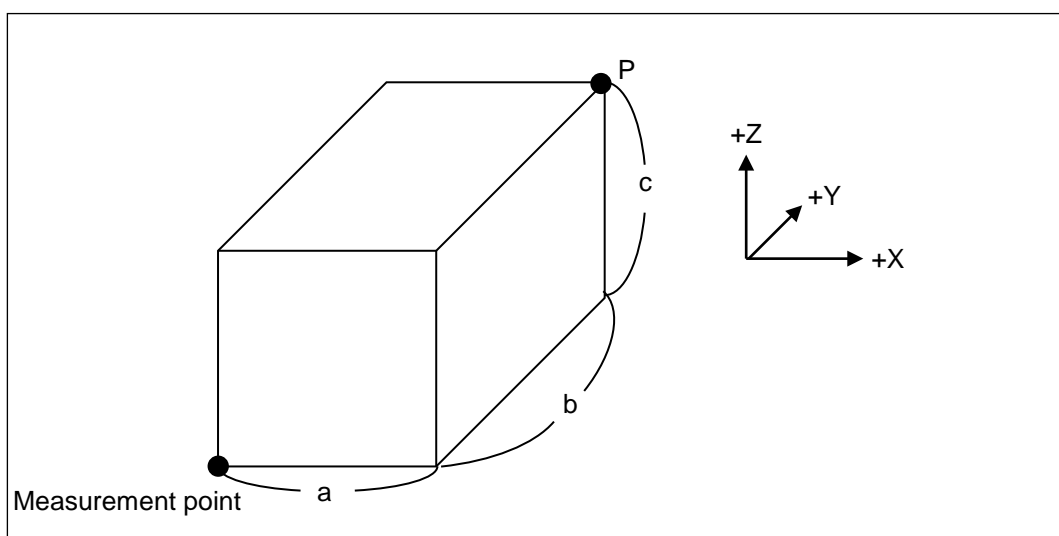
(Motion)

1. Spindle orientation 0°. Travels to the measurement start position for the X- and Y-axes.
2. Travels to the measurement start point on the Z-axis.
3. Executes measurement. (Z-axis minus direction)

### 7.4.5 Positioning to Measurement Position

Command format **G120 X\_ Y\_ Z\_ Q\_;**

X, Y and Z : Incremental amount from measurement position  
Q : Selects whether to use the register with the measurement result saved. 1 is used when it is omitted.



**G120 Xa Yb Zc**

When the above command is issued, it travels to P in the figure.

The alarm <<No measurement data>> is triggered if no measurement data exists.

## 7.5 Measurement Results Processing

### 7.5.1 Display Screen for Measurement Results

Press [4] or move the cursor to <4. Measurement results> on the production monitor menu, and press the [ENT] key to change to the following screen.

| Measurement result 1    |                |                | 2012/06/12 13:52:21 |                |            |
|-------------------------|----------------|----------------|---------------------|----------------|------------|
| Latest                  |                | Previous 1     |                     | Previous 2     |            |
| X                       | -10.000        | X              |                     | X              |            |
| Y                       | -20.000        | Y              |                     | Y              |            |
| Z                       | 600.000        | Z              |                     | Z              |            |
| Rotation                | 10.000         | Rotation       |                     | Rotation       |            |
| Time                    |                | Time           |                     | Time           |            |
| Previous 3              |                | Previous 4     |                     | Previous 5     |            |
| X                       |                | X              |                     | X              |            |
| Y                       |                | Y              |                     | Y              |            |
| Z                       |                | Z              |                     | Z              |            |
| Rotation                |                | Rotation       |                     | Rotation       |            |
| Time                    |                | Time           |                     | Time           |            |
| Previous 6              |                | Previous 7     |                     | Previous 8     |            |
| X                       |                | X              |                     | X              |            |
| Y                       |                | Y              |                     | Y              |            |
| Z                       |                | Z              |                     | Z              |            |
| Rotation                |                | Rotation       |                     | Rotation       |            |
| Time                    |                | Time           |                     | Time           |            |
| X <input type="text"/>  |                |                |                     |                |            |
| Production monitor menu | Meas. result 1 | Meas. result 2 | Meas. result 3      | Meas. result 4 | Delete all |

When continuing to perform measurements, the measurement results that were previously carried out are displayed.

| Measurement result 1    |                     |                | 2012/06/12 15:18:39 |                |                     |
|-------------------------|---------------------|----------------|---------------------|----------------|---------------------|
| Latest                  | G124 + G128         | Previous 1     | G124                | Previous 2     | G122                |
| X                       | -299.462            | X              | -299.462            | X              | -302.004            |
| Y                       | -349.778            | Y              | -349.778            | Y              | -350.000            |
| Z                       | 598.789             | Z              |                     | Z              |                     |
| Rotation                |                     | Rotation       |                     | Rotation       |                     |
| Time                    | 2012/06/12 15:16:49 | Time           | 2012/06/12 15:13:47 | Time           | 2012/06/12 15:11:49 |
| Previous 3              | G121                | Previous 4     |                     | Previous 5     |                     |
| X                       | -300.521            | X              |                     | X              |                     |
| Y                       | -351.234            | Y              |                     | Y              |                     |
| Z                       |                     | Z              |                     | Z              |                     |
| Rotation                | 1.375               | Rotation       |                     | Rotation       |                     |
| Time                    | 2012/06/12 15:11:15 | Time           |                     | Time           |                     |
| Previous 6              |                     | Previous 7     |                     | Previous 8     |                     |
| X                       |                     | X              |                     | X              |                     |
| Y                       |                     | Y              |                     | Y              |                     |
| Z                       |                     | Z              |                     | Z              |                     |
| Rotation                |                     | Rotation       |                     | Rotation       |                     |
| Time                    |                     | Time           |                     | Time           |                     |
| X <input type="text"/>  |                     |                |                     |                |                     |
| Production monitor menu | Meas. result 1      | Meas. result 2 | Meas. result 3      | Meas. result 4 | Delete all          |

## 7.5.2 Apply Measurement Results to Workpiece Coordinates

Command format **G10 L99 Pn X\_ Y\_ Z\_ Q\_;**

Pn : n =  
 1 G54  
 2 G55  
 3 G56  
 4 G57  
 5 G58  
 6 G59

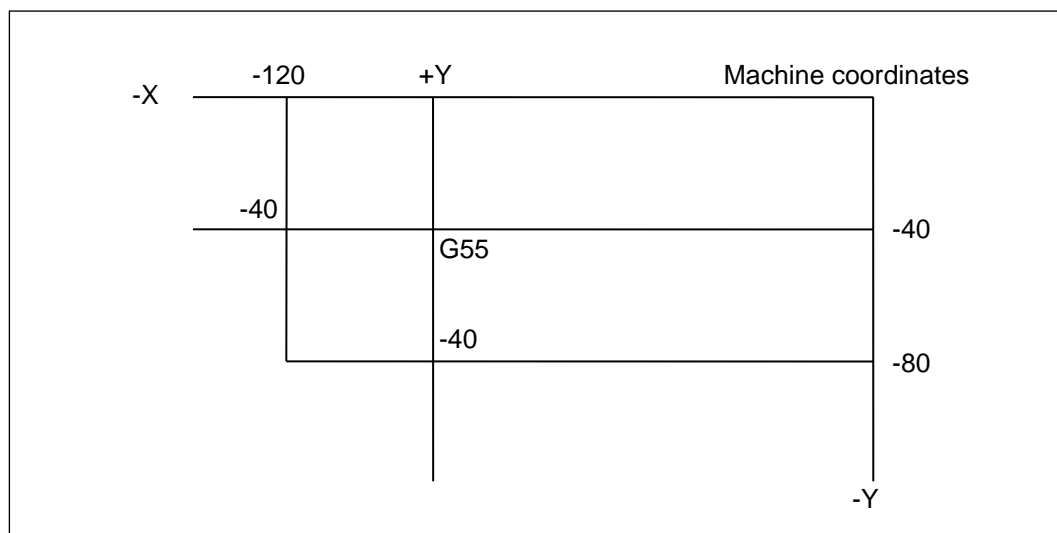
**G10 L98 Pn X\_ Y\_ Z\_ Q\_;**

Pn : n =  
 1 G54.1 P1  
 2 G54.1 P2  
 .  
 .  
 .  
 .  
 48 G54.1 P48

X, Y and Z : Coordinates for measurement position  
 Q : Measurement No. being used

Ex: When the measurement value for Measurement No.2 is (X,Y) = (-120,-80) in the machine coordinates, the following command is issued to change this position to (-40,-40) with G55 using the absolute coordinates.

**G10 L99 P2 X-40. Y-40. Q2;**



The workpiece coordinate data G55 is rewritten as:

X -80.000

Y -40.000

**(Note)** The measurement value is acquired by this function, and the true value varies depending on the delay that is unique to the probe. Therefore, contact the probe manufacturer and adjust it accordingly.

## 7.6 Lock Key Operations

<Dry run>

The axis travels to the start point for each measurement, but the measurement motion is not carried out. The measurement data is also not captured.

If an attempt is made to turn the dry run ON or OFF while the automatic workpiece measurement command is executing, the alarm <<During measurement>> is triggered and the change is not carried out.

<Machine lock>

Axis travel is not carried out. The coordinates on the position screen do not change.

## 7.7 Program Restart Operation

If a command between G121 and G129 exists during program restart, an alarm may be triggered.

(This page was intentionally left blank.)



## CHAPTER 8

# SUB PROGRAM FUNCTION

- 8.1 Overview**
- 8.2 Create Sub Program**
- 8.3 Simple Call Function**
- 8.4 Specify Return Number from Sub Program Function**
- 8.5 Call Specifying Sequence No.**

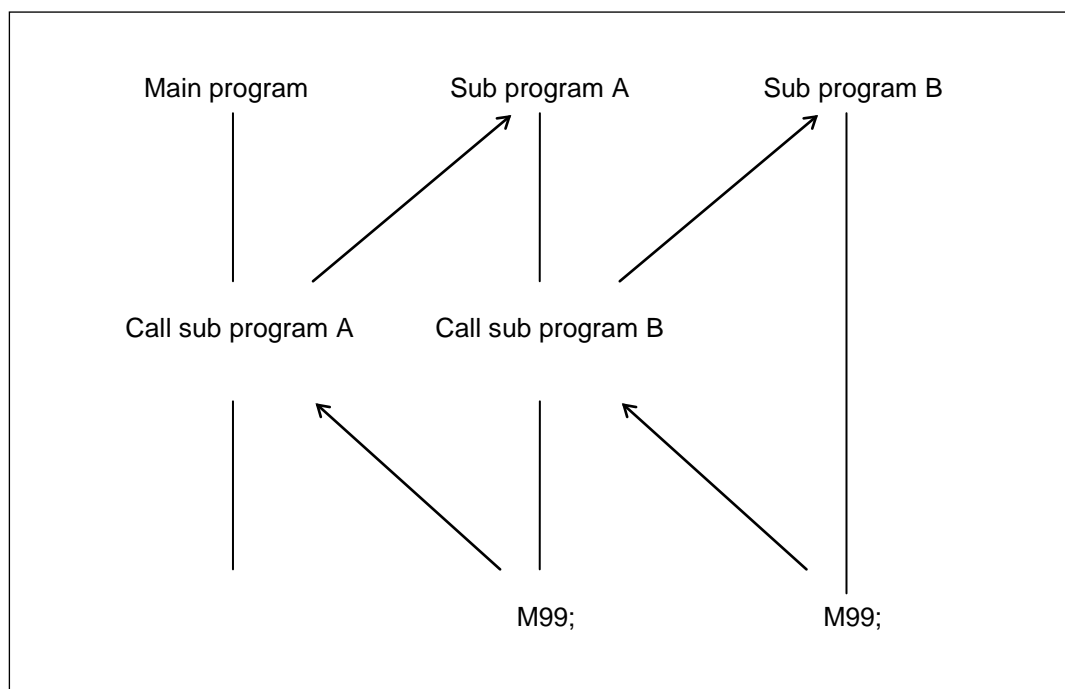
## 8.1 Overview

While programming, if there is a fixed sequence, or a repeated pattern in the program, a sub program can be registered beforehand to execute this. These sub programs are called when needed, which can simplify programming tremendously.

This sub program can be called from memory operation mode. It can also be called in the same way when using extended memory operation.

When using tape operation (with a memory card or general communications device), a sub program registered in the internal memory can be called. However, a sub program registered in an external device (with a memory card or general communications device) cannot be called. In addition, when using tape operation and calling a sub program registered in the internal memory, the communication with the external device stops temporarily while the sub program from the internal memory that was called is executing.

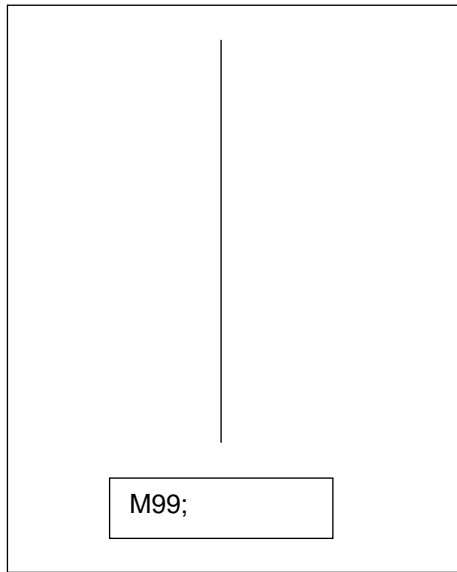
Furthermore, a separate sub program can also be called from a sub program which was called. In this situation, up to 8 sub programs can be called.



A sub program can also be called repeatedly with one call command.

## 8.2 Create Sub Program

A standard sub program is created using a format like the following.



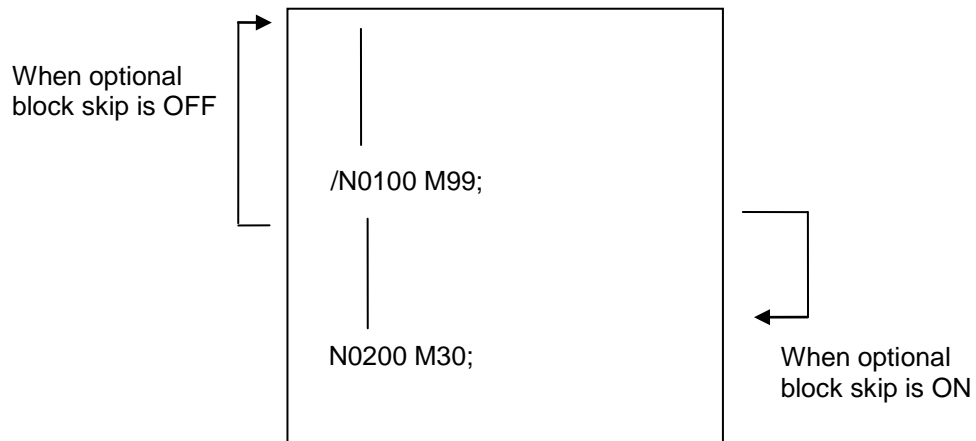
**(Note 1)** Always specify M99 at the last block of the sub program.

**(Note 2)** The alarm <<Invalid command>> is triggered when another G or M code command is issued on the M99 command block.

Special use of M99

When the M99 command is issued in the main program, excluding tape operation, it returns to the beginning of the program and operation starts from the beginning of the program again.

(Excluding the extended memory operation/tape operation)



When programming as described above, the program continues to operate and repeat the content from the beginning of the program until N0100 while the optional block skip is turned OFF. If the optional block skip is turned ON, the program skips the N0100 block and proceeds to execute from the next block.

## 8.3 Simple Call Function

The sub program executes a call from the main program or from another sub program.  
A call can be made from memory operation/extended memory operation as well as from tape operation (with a memory card or general communications device).

Command format **M98 P\_H\_L\_;**

- P : Sub program No. called (current program when omitted)  
H : Sequence No. for sub program called (beginning block when omitted)  
L : Number of repeated calls (9999 times or less) (1 when omitted)

**(Note 1) Another G or M code command cannot be issued on the M98 command block.**

If a G code command is issued, the alarm <<Invalid command>> is triggered.

In addition, if an M code command is issued and the user parameter <Multiple M codes in one block> is set to <0: No>, the alarm <<Same code cannot be used.>> is triggered. If the user parameter <Multiple M codes in one block> is set to <1: Yes>, the alarm <<Simultaneous command unable M code>> is triggered.

**(Note 2) Macro variables can be used in a subprogram number and G/M code macro to call.**

However, when the total size of the program that is loaded does not exceed the program load size, after an M30 (M02) command, all programs that can be called with the macro variables, such as "M98P1" (1<sup>st</sup> program call) or "G200" and "M200" (G/M code number registered in the G/M code macro) must be registered beforehand as "M98P?" (? is the program number), "G?" or "M?" (? is the G code number registered in the G/M code macro).

When a program that is not registered is called, the alarm <<No subprogram (\*\*\*\*)>> is triggered.

**Example 1: Case where a macro variable is used in a sub program number**

When the alternative value for #100 is "1", "5" and "100":

M98P#100; ← Sub program call command using a macro variable

G100T1R150.Z100.;

M30;

M98P1;

M98P5;

M98P100;

} After M30, all alternative "M98P\*" commands are entered as shown on the left

**Example 2: Case where a macro variable is used in G/M code**

When the alternative value for #100 is "400", "500" and "600":

G#100; ← G/M code macro call command using a macro variable

G100T1R150.Z100.;

M30;

G400;

G500;

G600;

} After M30, all alternative G code macros are entered as shown on the left

When the program load size is the size that is selected in the user parameter (switch 1) <Program load size>.

**(Note 3) When the total program size that is loaded (including sub programs) exceeds 2 MB, it operates in the extended memory operation mode.**  
When a macro variable is used in the sub program number that is called, as described in (Note 2), and when the total program size, including the sub programs that are noted after M30, exceeds the program load size, the machine operates in the extended memory operation mode.  
When the program load size is the size that is selected in the user parameter (switch 1) <Program load size>.

**(Note 4) When using extended memory operation and tape operation, the sub program is loaded when it is called. Operation sometimes stops during that time period. (At this time, the startup LED stays lit up.)**

- (Note 5) When using extended memory operation or tape operation, other sub programs that are not executing can be edited during operation. However, the edited content does not apply until that sub program is executed the next time.  
In addition, the alarm <<Editing>> is triggered when an attempt is made to execute that program while it is saved.  
Furthermore, the alarm <<Communicating>> is triggered when an attempt is made to execute that program while data communication is still in progress.
- (Note 6) If M98 is executed while an external device program is being executed during tape operation, etc., the communication with the external device up to that point will stop temporarily until M99 is executed and operation resumes in the external device program, or until the memory operation ends (M30 execution or [RST]). Note that other communication means with the external device is not available during this time period.
- (Note 7) When using memory operation and extended memory operation, a program in the same folder as the main program is called. In addition, when using tape operation, a program in the root directory or root folder is called.
- (Note 8) The axis does not travel even if an X-, Y- or Z-axis command is issued on the M98 command block.
- (Note 9) When a sequence number (H address) and the number of repetitions (L address) are specified during extension memory operation and tape operation, the alarm <<Invalid command>> is triggered.

## 8.4 Specify Return Number from Sub Program Function

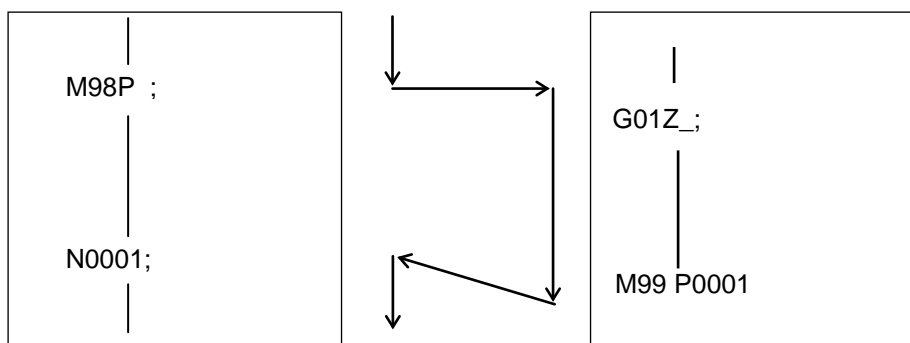
Command format **M99 P<sub>;</sub>**

P : Sequence No.

### 1. Command in sub program

After a command is executed, it returns to the sequence number for the command that was issued in another program. It searches for the sequence number starting from the beginning of the program and returning to the first block that was found. If there is no sequence number for the command, the alarm <<No applicable sequence>> is triggered.

(Program execution order)



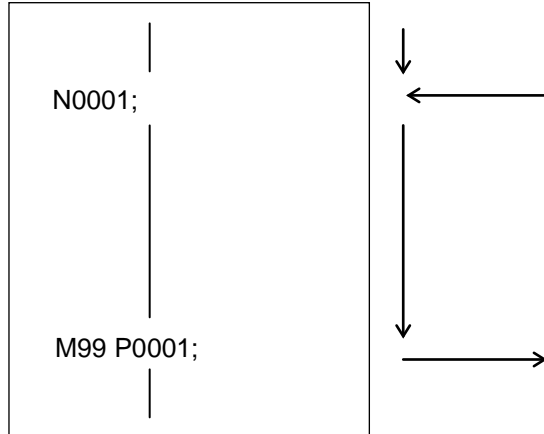
**(Note 1)** The alarm <<Subprogram return error>> is triggered if the number of repetitions in the M98 command is not 1.

**(Note 2)** The return number cannot be specified during extension memory operation and tape operation. Otherwise, the alarm <<Invalid command>> is triggered.

## 2. Main program command

After a command is executed, it jumps to the sequence number for the command that was issued in the main program. It searches for the sequence number starting from the beginning of the program and jumping to the first block that was found. If there is no sequence number for the command, the alarm <<No applicable sequence>> is triggered.

(Program execution order)



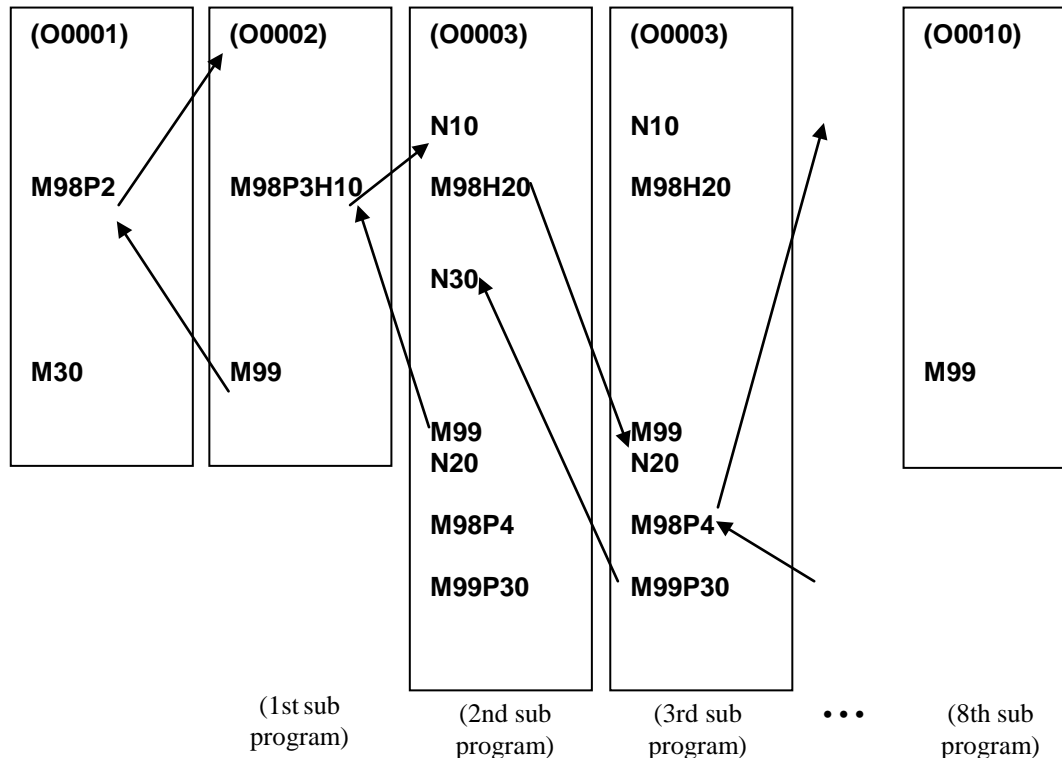
**(Note)** Commands cannot be issued during extension memory operation and tape operation. Otherwise, the alarm <<Invalid command>> is triggered.

## 8.5 Call Specifying Sequence No.

When issuing a command with M98P\_H\_, it can be called and executed from the sequence number specified in the sub program.

Special notes are provided below when programming and using an H address.

(Ex: Program execution order)

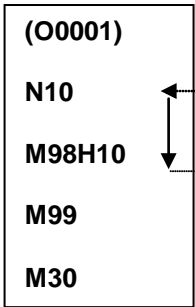


As noted above, up to 8 sub programs can be called.

- (Note 1) When calling with M98H\_, the structure becomes 1 layer (1 sub program) deep even in the same program. (The level does not change in a sub program call with M98 using local variables for the macro.)
- (Note 2) The alarm <<No applicable sequence>> is triggered if there is no sequence number specified in the H address.
- (Note 3) The search for the H address starts from the beginning of the program.
- (Note 4) The alarm <<Subprogram number error>> is triggered when no command has been issued for either the P or H address.
- (Note 5) The alarm <<Subprogram call error>> is triggered when the current program is called by the P address. Specify only using the H address to call the current program.
- (Note 6) The relationship between the main and sub programs for calling the current program using M98H\_ is established with the sequence number. As a result, the alarm <<Subprogram call error>> is triggered if it jumps to N10 before M99 is executed when using M98H10.
- (Note 7) The macro variable command can be used in the P and H address. However, the address is ignored when the macro variable field is "blank".
- (Note 8) The sequence number search for the H address is carried out when the M98 command is issued. Therefore, when the main and sub program are processed together in one program, the cycle time may be delayed before they are processed as one.
- (Note 9) Calls (M98H\_) which specify sequence numbers in the same program cannot be issued during extension memory operation and tape operation. Otherwise, the alarm <<Invalid command>> is triggered.

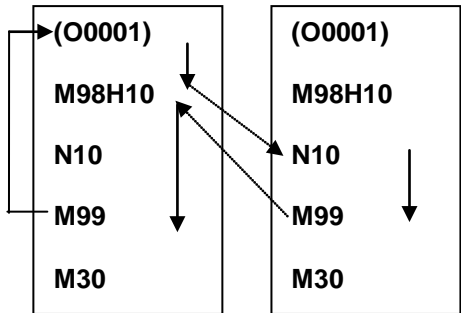


Example 1:



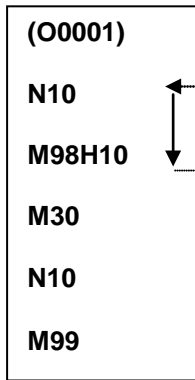
The alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time in a program as shown on the left.

Example 2:



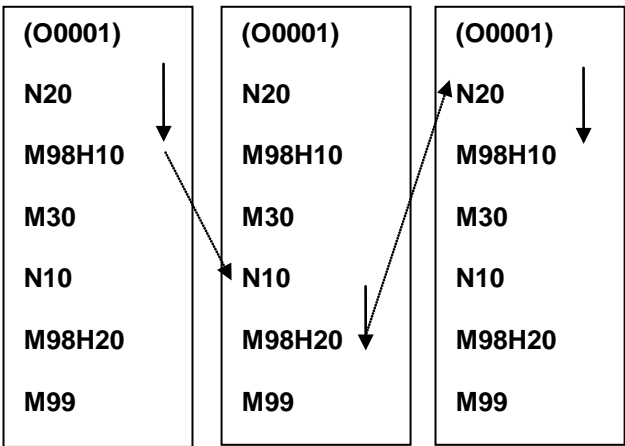
When creating a program as shown on the left, the operation continues because it will not reach M30.  
When creating a sub program in one program, create it after M30 (M02).

Example 3:



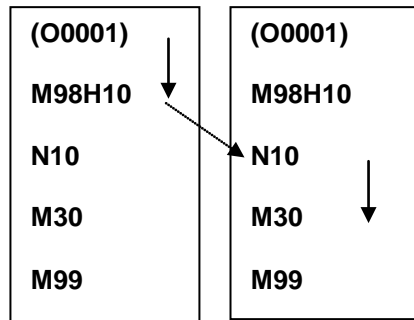
The sequence number search for the H address starts from the beginning of the program. As shown on the left, it jumps close to the block closer to beginning of the program when there are two blocks for “N10”.  
(The alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time.)

Example 4:



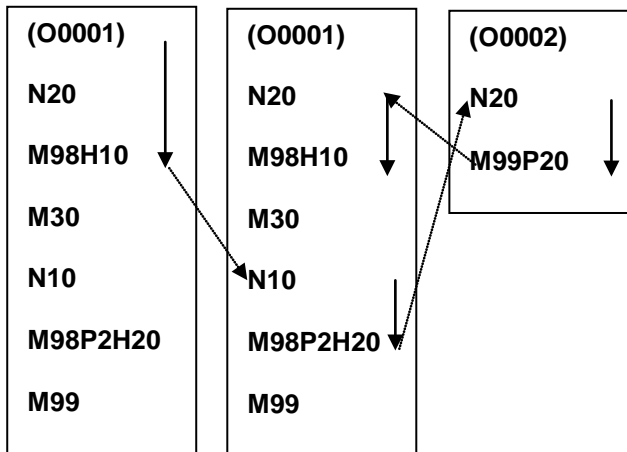
When creating a program as shown on the left, the alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time.

Example 5:



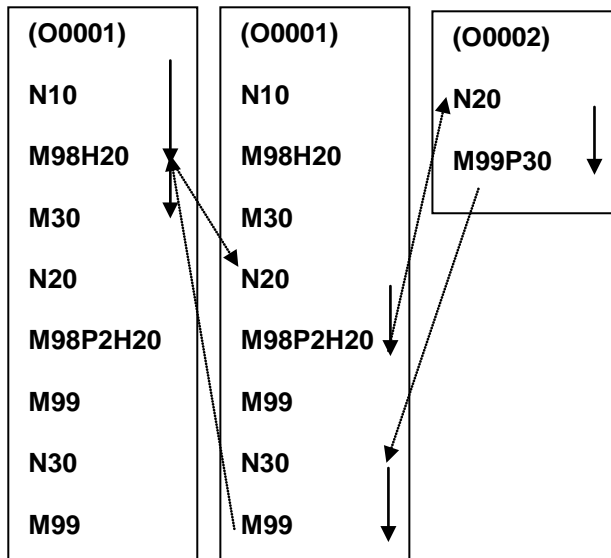
When creating a program as shown on the left, the program ends with M30.

Example 6:



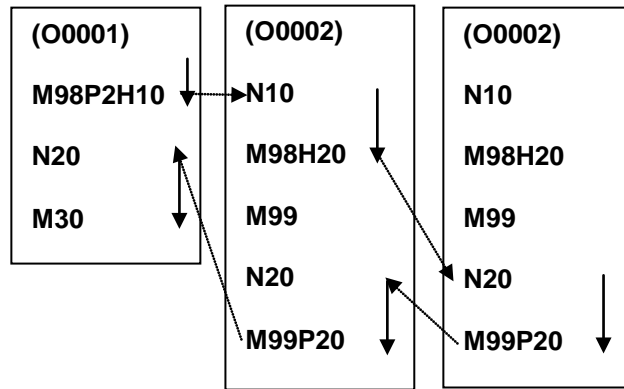
When creating a program as shown on the left, the alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time.

Example 7:



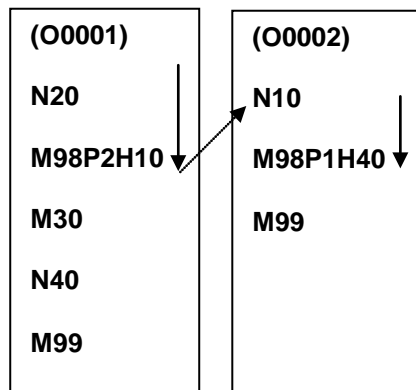
When creating a program as shown on the left and the H20 is executed for the second time in the “M98P2H20” block, an error does not trigger because a different program number is called. This program ends normally with M30.

Example 8:



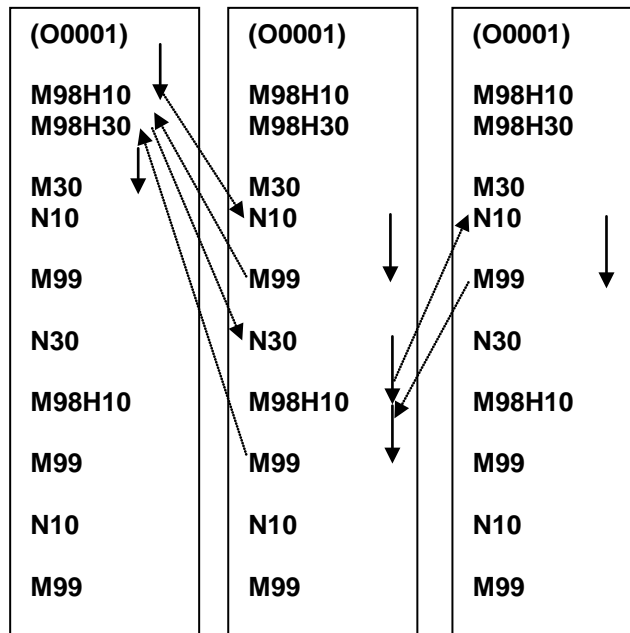
When creating a program as shown on the left, “N20” is called in the “M98H20” block and it is called again in the “M99P20” block, but an error does not trigger. This program ends normally with M30.

Example 9:



When creating a program as shown on the left, the alarm <<Subprogram call error>> is triggered when program 1 is called from program 2.

Example 10:



When creating a program as shown on the left, the “M98H10” block is executed twice. After it is executed for the first time, it resets with M99. As a result, an error does not trigger on the block when it executes the second time. This program ends normally with M30.

(This page was intentionally left blank.)

# CHAPTER 9

## FEED FUNCTION

- 9.1 Feed Function
- 9.2 Automatic F Command at Tool Change

## 9.1 Feed Function

This function issues a command using the numerical values that come after “F” to specify the tool feedrate.

The feedrate per minute (G94), the feedrate per rotation (G95) and the inverse time feed (G93) are available when specifying this command.

### 9.1.1 Feedrate per Minute (G94)

This command uses the numerical values that come after “F” to specify how fast to feed the tool per minute.

Ex: G94;  
G01 Z -100. F1000; (1000.00 mm/min)

### 9.1.2 Feedrate per Rotation (G95)

This command uses the numerical values that come after “F” to specify how fast to feed the tool per spindle rotation.

Ex: M03 S2000;  
G95;  
G01 Z -100. F0.05; (0.0500 mm/rev)

**(Note 1)** The execution speed (actual travel speed on machine) of the feedrate per rotation is calculated using the following formula.

$$\text{Execution speed [mm/min]} = \text{Feedrate [mm/rev]} \times \text{Spindle speed [min}^{-1}] \times (\text{Spindle override [\%]} / 100) \times (\text{Cutting override [\%]} / 100)$$

**(Note 2)** If the spindle rotation is zero when the feedrate per rotation command is executed, the alarm <<Feedrate error>> is triggered.

### 9.1.3 Inverse Time Feed (G93)

The inverse time is the inverse number of the machining time and is indicated by the number following “F” in a command.

After the inverse time feed (G93) command is issued, the inverse time feed is enabled before the command for the feedrate per minute (G94) or the feedrate per rotation (G95) is issued.

Example when using linear interpolation

```
N01 G93;
N02 G91 G01 X -100. F1;(1.00 1/min)
N03 G01 Y -100. F1;(1.00 1/min) Inverse time feed enabled
N04 G01 Z -100. F1;(1.00 1/min)
N05 G94 G01 X -50. F1500; Inverse time feed disabled
```

In the above example, the feedrate for NO2 block is as follows:

$$\begin{aligned} \text{Feedrate (mm/min)} &= \text{Length of segment (mm)} \times \text{F(1/min)} \\ &= 100 \times 1 \\ &= 100 \end{aligned}$$

Example when using circular interpolation

```
N01 G93;
N02 G91 G02 X100. Y-50. I0. J-50 F10; (10.00 1/min) Inverse time feed enabled
N03 G94 G02 X-50. Y-100. I-50. J-50 F1500; Inverse time feed disabled
```

In the above example, the feedrate for NO2 block is as follows:

$$\begin{aligned} \text{Feedrate (mm/min)} &= \text{Radius of arc from start point (mm)} \times \text{F(1/min)} \\ &= 50 \times 10 \\ &= 500 \end{aligned}$$

Example of inverse time calculation

Ex 1: When ending 1 block, the following command calculation is used.

$$\begin{aligned} \text{Inverse time [1/min]} &= 1 \div 1 \text{ block end time [sec]} \div 60[\text{min}] \\ &= 1 \div 60 \div 60 \\ &= 1 \quad (\text{command is issued with F1}) \end{aligned}$$

Ex 2: Travel time when issuing a F0.25 command is as follows.

$$\begin{aligned}\text{Travel time [min]} &= 1 \div \text{Inverse time [1/min]} \\ &= 1 \div 0.25 \\ &= 4.0 \quad (\text{travel time is 4 minutes})\end{aligned}$$

|                                                    | Metric (mm)                                                                                                    | Inch (inch)                                                                                                        |
|----------------------------------------------------|----------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------|
| Linear interpolation (G1)                          | $F = \frac{1}{\text{Time (min)}} = \frac{\text{Feedrate (mm/min)}}{\text{Length of segment (mm)}}$             | $F = \frac{1}{\text{Time (min)}} = \frac{\text{Feedrate (inch/min)}}{\text{Length of segment (inch)}}$             |
| Circular interpolation (G2/G3/G102/G103/G202/G203) | $F = \frac{1}{\text{Time (min)}} = \frac{\text{Feedrate (mm/min)}}{\text{Radius of arc for start point (mm)}}$ | $F = \frac{1}{\text{Time (min)}} = \frac{\text{Feedrate (inch/min)}}{\text{Radius of arc for start point (inch)}}$ |

**(Note 1)** In the inverse time feed, issue an F command each time on a block for the cutting feed, in order to issue a machining time command for the segment.

When there is a block without an F command, the alarm <<Feedrate not specified>> is triggered.

**(Note 2)** An inverse time feed (G93) command cannot be issued during the following modes.

When issuing a command, the alarm <<Specified G code cannot be used>> is triggered.

- Thread cutting command (G33) in progress and thread cutting cycle (G392) in progress
- Canned cycle (apart from G80) in progress
- Constant peripheral speed control (G96) in progress

## 9.1.4 Command Range

Minimum unit setting: Type 1

|                      | Feedrate per minute (G94)   | Feedrate per rotation (G95)  |
|----------------------|-----------------------------|------------------------------|
| Machine unit setting | 0.01 to 999999.99 mm/min    | 0.0001 to 999.9999 mm/rev    |
| 0: Meter             | 0.01 to 999999.99°/min      | 0.0001 to 999.9999°/rev      |
| Machine unit setting | 0.001 to 99999.999 inch/min | 0.00001 to 99.99999 inch/rev |
| 1: Inch              | 0.001 to 99999.999°/min     | 0.00001 to 99.99999°/rev     |

|                      | Inverse Time Feed (G93) |
|----------------------|-------------------------|
| Machine unit setting | 0.001 ~9999.999 1/min   |
| 0: Meter             |                         |
| 1: Inch              |                         |

Minimum unit setting: Type 2

|                      | Feedrate per minute (G94)     | Feedrate per rotation (G95)    |
|----------------------|-------------------------------|--------------------------------|
| Machine unit setting | 0.001 to 999999.999 mm/min    | 0.00001 to 999.99999 mm/rev    |
| 0: Meter             | 0.001 to 999999.999°/min      | 0.00001 to 999.99999°/rev      |
| Machine unit setting | 0.0001 to 99999.9999 inch/min | 0.000001 to 99.999999 inch/rev |
| 1: Inch              | 0.0001 to 99999.9999°/min     | 0.000001 to 99.999999°/rev     |

|                      | Inverse Time Feed (G93) |
|----------------------|-------------------------|
| Machine unit setting | 0.001 ~9999.999 1/min   |
| 0: Meter             |                         |
| 1: Inch              |                         |

The alarm <<Max. speed exceeded>> or <<Feedrate error>> is triggered for the feedrate per minute (G94) and the feedrate per rotation (G95) when an attempt is made to move the axis and the speed exceeds the user parameter (switch 1) setting <Max. actual cutting travel speed>, the machine parameter (system 1) setting <X-(to Z)-axis max. cutting feedrate> or the machine parameter (system 2) setting <4th-(to 8th)-axis max. cutting rotation speed>.

When the feedrate calculated from the inverse time is less than the following limit values, the alarm <<Command data range error>> is triggered.

Minimum limit values for inverse time feed (G93)

|                      | Minimum unit setting: Type 1 | Minimum unit setting: Type 2 |
|----------------------|------------------------------|------------------------------|
| Machine unit setting | 0.01 mm/min                  | 0.001 mm/min                 |
| 0: Meter             | 0.01 °/min                   | 0.001 °/min                  |
| Machine unit setting | 0.001 inch/min               | 0.0001 mm/min                |
| 1: Inch              | 0.001 °/min                  | 0.0001 °/min                 |

The execution speed (actual travel speed on machine) calculated for the feedrate per rotation is rounded off in the command range for the feedrate per minute. However, when the execution speed is less than the minimum in the command range, the execution speed is assigned the minimum value in the command range.

Ex: When the minimum unit setting is Type 1 and when the feedrate is 1.2345 mm/rev and the spindle speed is 4321 min<sup>-1</sup>:  
 Execution speed (mm/min) = 1.2345 mm/rev × 4321 min<sup>-1</sup> = 5334.2745 mm/min  
 Any value below 0.01 mm/min is rounded off and the execution speed becomes 5334.27 mm/min.

### 9.1.5 Switching Between Feedrate per Minute / Feedrate per Rotation / Inverse Time Feed

- (1) When one of the following commands applies, the F code modal turns into a state when the power is ON (not set).
  - When executing a feedrate per rotation command (G95) or an inverse time feed command (G93) during the feedrate per minute (G94).
  - When executing a feedrate per minute command (G94) or an inverse time feed command (G93) during the feedrate per rotation (G95).
- (2) When switching to a feedrate per minute command (G94) or a feedrate per rotation command (G95) during the inverse feed (G93), an F command is required. If there is no F command, the alarm <<Feedrate not specified>> is triggered.

## 9.2 Automatic F Command at Tool Change

When the user parameter <Automatically use F command when changing tool> is set to <1: Yes>, the F command is automatically issued for the <F command value> in the tool data, which automatically corresponds to the tool that is loaded onto the spindle at the tool change (M6 or G100).

Refer to section “5.7 Canned Cycle for Tool Change” for further details.



## CHAPTER 10

# SPINDLE RELATED FUNCTIONS (S FUNCTION)

- 10.1 S Function
- 10.2 M Function (Spindle Control)
- 10.3 M Function (Lathe Spindle Control)
- 10.4 M Function (Spindle Selection)

## 10.1 S Function

### 10.1.1 Spindle Speed Command

A spindle speed ( $\text{min}^{-1}$ ) command is issued.

This command uses the numerical values (under 5 digits) that come after “S” to specify the speed.

- (Note 1) The S command is not erased with NC reset, but it must be set when the power is turned ON.
- (Note 2) The S command must always be issued before the spindle rotation command (M03/M303, M04/M304).
- (Note 3) When the S command is on the same block as axis travel, the S command becomes valid at the same time as the axis travel start.

### 10.1.2 Constant Peripheral Speed Control (G96, G97) (Option)

\* Available when equipped with a lathe function

By issuing a peripheral speed command (m/min, or feet/min) to the G96S address, this function controls the rotation speed of the spindle so that the cutting speed is maintained at a fixed or constant level that corresponds to the machining diameter.

#### Constant peripheral speed control command

Command format

**G96 S\_\_P\_\_;**

- S : Peripheral speed (m/min, or feet/min)
- P : Constant peripheral speed control axis (No omissions. If there is an omission, the alarm <<There is no P address.>> is triggered.)
- P1 : X-axis, P2: Y-axis, P3: Z-axis

#### Constant peripheral speed control cancel command

Command format

**G97 S\_\_;**

- S : Spindle speed ( $\text{min}^{-1}$ )

The G96 modal S command assumes S = 0 (peripheral speed 0) until the spindle rotation command (M03/M303) and the spindle reverse rotation command (M04/M304) is issued.

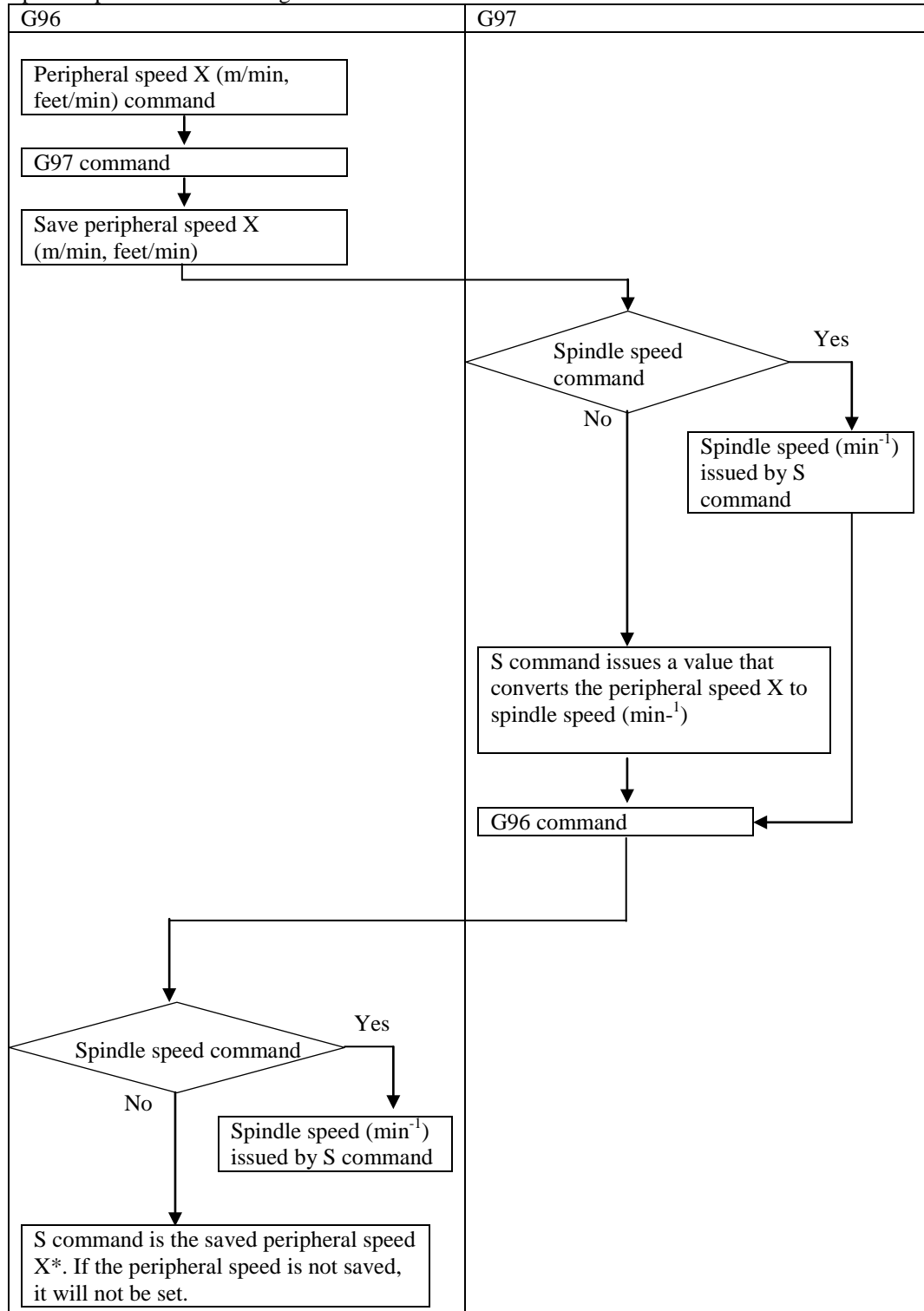
When using the constant peripheral speed control, configure the settings so that the workpiece coordinate zero for the control axis is in the lathe spindle center.

When using the tool length/tool position offset function, the spindle speed is controlled with the tool length/tool position offset amount added to the current machine coordinates.

The constant peripheral speed control for the rapid feed command (G00) and the tool change command (G100/M06) calculates the peripheral speed based on the end position of the constant peripheral speed control axis.

- (Note 1) If a constant peripheral speed control command is issued while using the [SP.CW] key to rotate the spindle, the alarm <<Spindle rotating>> is triggered. Specify the spindle rotation command on the same block.
- (Note 2) Constant peripheral speed control command is not possible while in the inverse time feed (G93) modal. If a command is issued, the alarm <<Command not possible during inverse time feed>> is triggered.

Spindle speed when switching between G96 and G97 modals



\* The peripheral speed will not be set (while power is turned ON) when the spindle is changed using the spindle selection command (M141/M142).

Ex:

M142 G96 P1 S200 ; Peripheral speed 200  
 G97 S1000 ; Peripheral speed 200 is saved  
 M141 G96 P1 ; Peripheral speed is not set (saved peripheral speed is deleted because the spindle is changed)

### 10.1.3 Spindle Speed Clamp (G92) (Option)

\* Available when equipped with a lathe function

The spindle speed is controlled to a value below the command value issued for the S address, or above the command value issued for the Q address.

The spindle speed clamp is only valid when the constant peripheral speed control (G96) is enabled.

Command format

|                     |
|---------------------|
| <b>G92 S__ Q__;</b> |
|---------------------|

|   |   |                                             |
|---|---|---------------------------------------------|
| S | : | Spindle speed ( $\text{min}^{-1}$ )         |
| S | : | Maximum spindle speed ( $\text{min}^{-1}$ ) |
| Q | : | Minimum spindle speed ( $\text{min}^{-1}$ ) |

Always issue the G92 S command when issuing a constant peripheral speed control command (G96). If a G96 command is executed without the other command, an alarm is triggered.

When the power is turned ON, the settings are not configured. In addition, when the command value issued for the Q address is larger than the command value issued for the S address, then the spindle speed is not controlled using the command value issued for the Q address.

### 10.1.4 Automatic S Command at Tool Change

When the User parameter (Switch 1) <Automatically use S command when changing tool> is set to <1: Yes>, the S command is automatically issued for the value set in the tool data, which corresponds to the tool that is loaded onto the spindle at the tool change.

Refer to “Chapter 5 - 5.7 Canned Cycle for Tool Change (Nonstop ATC) (G100)” for further details.

### 10.1.5 Register Maximum Speed

The maximum speed for a tool mounted onto the spindle can be registered in the tool data.

When the User parameter (Switch 1) <Tool data spindle rotation exceeded> is set to <0: Alarm>, if a spindle speed command is issued that exceeds the maximum speed set for the tool, the alarm <<Tool data spindle rotational frequency error>> is triggered. When set to <1: Clamp at max. speed>, the tool is clamped at the maximum spindle speed that is set for the tool.

When not set, the spindle speed is not checked. (However, the alarm <<Spindle speed error>> is triggered when an attempt is made to rotate at a speed that exceeds the Machine parameter (System 1) <Max. spindle speed>.)

**(Note)** When the tool is set in conversation language for the spindle, it is processed as “not set”.

## 10.2 M Function (Spindle Control)

### 10.2.1 Spindle Normal Rotation (M03)

The spindle rotates in a clockwise direction. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

### 10.2.2 Spindle Reverse Rotation (M04)

The spindle rotates in a counterclockwise direction. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

### 10.2.3 Spindle Stop (M05)

The spindle rotation is stopped. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

### 10.2.4 Spindle Orientation (M19)

The spindle is oriented to a 0° position. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

**(Note)** On a machine equipped with the lathe function, the spindle may vibrate slightly when carrying out an orientation operation at the zero degree position while the spindle is fitted with a cutting tool. There is no impact on the machining or measurement operations. However, to stop this slight vibration, perform the orientation operation at another position besides zero degrees, or execute the spindle rotation command (M03/M04).

#### 10.2.4.1 Spindle Orientation to a Given Angle

Command format **M19 R\_;**

R : Spindle angle (-360 to 360°)

The spindle is oriented to the angle that is specified by command R.

When the command angle is a positive value, it turns in a clockwise direction. When the command angle is a negative value, it turns in a counterclockwise direction.

After this operation, the servo stays ON for the spindle.

### 10.2.5 Spindle Orientation (M111)

The spindle is oriented to a 180° position. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

## 10.3 M Function (Lathe Spindle Control)

### 10.3.1 Lathe Spindle Normal Rotation (M303)

The lathe spindle rotates in a clockwise direction when the lathe spindle is selected (M142 modal). The alarm <<Selection of a spindle is abnormal.>> is triggered if a command is issued when the lathe spindle is not selected. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

### 10.3.2 Lathe Spindle Reverse Rotation (M304)

The lathe spindle rotates in a counterclockwise direction when the lathe spindle is selected (M142 modal).

The alarm <<Selection of a spindle is abnormal.>> is triggered if a command is issued when the lathe spindle is not selected. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

### 10.3.3 Lathe Spindle Stop (M305)

The lathe spindle rotation is stopped when the lathe spindle is selected (M142 modal).

The alarm <<Selection of a spindle is abnormal.>> is triggered if a command is issued when the lathe spindle is not selected. When there is an axis travel command on the same block, the axis travel is carried out at the same time.

**(Note)** When there is no lathe spindle, the M303, M304 and M305 commands cannot be issued.

## 10.4 M Function (Spindle Selection)

### 10.4.1 Spindle Selection (M141)

The spindle is selected. When the M141 command is issued while the lathe spindle is rotating, the lathe spindle stops.

### 10.4.2 Lathe Spindle Selection (M142)

The lathe spindle is selected. When the M142 command is issued while the spindle is rotating, the spindle stops.

**(Note 1)** When there is no lathe spindle, the M141 and M142 commands cannot be issued.

**(Note 2)** The M141 and M142 modals are linked with the [L. SP] key. When the [L. SP] key is turned ON in manual mode, it changes to M142 modal. When it is turned OFF, it changes to M141 modal.

## CHAPTER 11

# TOOL RELATED FUNCTIONS (T FUNCTION)

- 11.1 T Function
- 11.2 M Function (Tool Control)

## 11.1 T Function

On an arm type ATC mechanism, a T command turns the magazine to the corresponding pot.

On a turret type ATC mechanism, the tool (magazine) is set to index when the tool change command (G100, M6) is issued.

Command format

|           |
|-----------|
| T _ _ _ ; |
|-----------|

### 11.1.1 When Issuing a Command Using the Tool Number

A tool number command is issued using the number that comes after the “T”. (T1 to T99)

The pot with the corresponding tool is indexed. (Arm type ATC mechanism)

When issuing a command in MDI mode, if the corresponding tool is not mounted, only the modal is updated.

### 11.1.2 When Issuing a Command Using the Pot Number (Magazine Number)

A pot number (magazine number) command is issued using the numerical values (2 digits) that come after the “T1”.

(T101 to T1nn:nn is the maximum value for pot that is mounted)

The pot with the corresponding tool is indexed. (Arm type ATC mechanism)

### 11.1.3 When Issuing a Command Using the Group Number

A tool group number command is issued using the numerical values (2 digits) that come after the “T9”.

(T901 to T930)

The pot with the corresponding tool is indexed. (Arm type ATC mechanism)

The tools are registered to the magazines, and when a command is issued using the group number, the tools are used following the order they is registered in the specified group.

The alarm <<Command data range error>> is triggered when a command is issued in MDI operation mode.



## 11.2 M Function (Tool Control)

### 11.2.1 Tool Change (M06)

It is the same as G100.

When the user parameter (switch 1) <Multiple M codes in one block> is set to <1: Yes>, up to 3 M code commands can be issued simultaneously for G100. However, the limit is up to 3 commands including M06 when issuing an M06 command.

Refer to “Chapter 5 - 5.7 Canned Cycle for Tool Change (Nonstop ATC) (G100)” for details about M codes that can be issued simultaneously and the operation timing.

### 11.2.2 Tool Life Counter (M230 to M231)

After issuing an M231 command, the tool life count for the spindle tool is stopped during operation.

When issuing an M230 command, the count is restarted.

### 11.2.3 Changing ATC Arm Turn Speed (M420 to M423, M432)

When using an arm type ATC mechanism, the ATC arm turn speed can be changed after the M code is executed for the next tool change command. (Note 1)

- M420 : Turn the arm at the maximum speed that corresponds to the tool type being changed.
  - Standard tool turn speed when a standard tool is used (Note 2).
  - Large tool turn speed when a large tool is used (Note 3).
- M421/M432 : Turn the arm at a large tool turning speed regardless of the tool type to be changed. The modal becomes M421 even when an M432 command is issued. (M432 is for compatibility for B00 control. In the C00 control, the same operation is possible with M421, and the operation does not change.)
- M422 : For the standard tool turning speed, the arm turns at a speed that is calculated using the user parameter (switch 1) setting <ATC arm turn speed ratio 1 (M422)>. (Note 4)
- M423 : For the standard tool turning speed, the arm turns at a speed that is calculated using the user parameter (switch 1) setting <ATC arm turn speed ratio 2 (M423)>. (Note 4)

**(Note 1)** This function is disabled on a turret type ATC mechanism. When a command is issued, it follows user parameter (switch 1) <Unregistered M-code>.

**(Note 2)** Set value for machine parameter (system 3) <AT-axis rapid feedrate 1>

**(Note 3)** Set value for machine parameter (system 3) <AT-axis rapid feedrate 2>

**(Note 4)** When all of the following conditions are met, the arm operates using a turning speed for a large tool.

- Tool type being changed is a large tool
- Speed calculated using the ratio exceeds the turning speed for a large tool

Use the following operations to change back to the M420 modal.

- M420 execution
- Program end
- Pressing [RST (Machine lock)] key
- Operation reset
- Tool change (G100/M06)

### 11.2.4 Shutter and Cover Related M Codes (M434, M438, M439)

When using an arm type ATC mechanism, the shutter can be operated after the M code is executed. (Note 1) (Note 2) The code is only valid for 40MG on R650X2.

However, it does not operate when there is no shutter mechanism.

M434: Waits for the shutter to close. (Note 3)

M438: Opens the pot shutter. (Note 2)

M439: Closes the pot shutter. (Note 2)

**(Note 1) This function is disabled on a turret type ATC mechanism.**

**When a command is issued, it follows user parameter (switch 1) <Unregistered M-code>.**

**(Note 2) When a command is issued while the door is open, the alarm <<Door open>> is triggered.**

**(Note 3) The maximum wait time follows the set value in the machine parameter (system 3) <Pot shutter timeout period>.**

### 11.2.5 Specify Magazine Turn Speed (M435, M436, M437)

On arm type ATC models, the magazine turns speed can be changed from the program.

M435: Maximum magazine turn speed

M436: Magazine turn speed 1

M437: Magazine turn speed 2

**(Note) This function is disabled on a turret type ATC mechanism. When a command is issued, it follows user parameter (switch 1) <Unregistered M-code>.**

The following operations return the speed to the maximum speed.

- M435 execution
- Program end
- Reset (Machine lock)
- Operation reset
- Tool change(G100/M06)

The turn speed for M436 and M437 is specified in the user parameter <Magazine turn speed ratio 1(M436)> and <Magazine turn speed ratio 2(M437)>.

**(Note) When the M435, M436, M437 commands are issued on the same block as a T command, the alarm <<Invalid command>> is triggered.**

### 11.2.6 Magazine Turns to the Tool Installation Position (M501 to M599)

On the arm type ATC mechanism, the magazine can be turned so that the pot with the tool (tool number specified by \*\* for M5\*\*) is in the tool installation position.

**(Note 1) This function is disabled on a turret type ATC mechanism. When a command is issued, it follows user parameter (switch 1) <Unregistered M-code>.**

**(Note 2) When an M5xx command is issued on the same block as a T command, the alarm <<Invalid command>> is triggered.**

### 11.2.7 Machining Load Monitor Function (M340 to M343)

This function measures the machining load, monitors it and assesses whether the machining is normal. Refer to “Chapter 11 - 11.13 Machining load monitor function” in the Operation Manual for further details.

M340: Machining load monitor OFF

M341: Machining load monitor ON

M342: Machining load monitor ON (Max. only)

M343: Machining load monitor ON (Min. only)

# CHAPTER 12

## M FUNCTION

- 12.1 Outline of M Function**
- 12.2 M Code List**
- 12.3 M Function (Program Control)**
- 12.4 M Function (Signal Control)**
- 12.5 M Function (Additional Axis Control)**
- 12.6 M Function (Positioning Check Distance)**
- 12.7 M Function (Time Constant Switch)**

## 12.1 Outline of M Function

- The M codes are used for commanding ON/OFF of various solenoids of the machine.
- Command by address M and a following within 3-digit number.
- When the M command is in the same block as that of the axis movement, the motion is divided into following three types.

The M command becomes effective before the axis movement starts.

The M command becomes effective at the same time the axis movement starts.

The M command becomes effective after the axis movement is finished.

**(Note) The modal command is effective until it is cancelled by the next M code or changed. The one-shot command is effective only in the commanded block.**

## 12.2 M Code List

The code with \* is already set when the power is turned on. (Modal status)

For “Multiple commands” and “Multiple command limits”, refer to “12.2.1 Multiple M Code Commands in One Block”.

For multiple M code commands for tool changes, refer to “Chapter5 5.7 Canned cycle for tool change (Non-stop ATC) (G100)”.

| Group | M code       | Contents                                                                          | Timing of axis motions | Modal / One-shot | Multiple commands |
|-------|--------------|-----------------------------------------------------------------------------------|------------------------|------------------|-------------------|
|       | M00          | Program stop                                                                      | After                  | One-shot         | Impossible        |
|       | M01          | Optional stop                                                                     | After                  | One-shot         | Impossible        |
|       | M02          | End of Program                                                                    | -                      | -                | Impossible        |
|       | M30          | End of Program                                                                    |                        |                  |                   |
|       | M03          | Spindle CW                                                                        | Simultaneous           | Modal            | Possible          |
|       | M04          | Spindle CCW                                                                       |                        |                  |                   |
|       | M05*         | Spindle stop                                                                      |                        |                  |                   |
|       | M19          | Spindle orientation                                                               |                        |                  |                   |
|       | M111         | Spindle orientation (180°)                                                        |                        |                  |                   |
|       | M08          | Coolant pump ON                                                                   | Before                 | Modal            | Possible          |
|       | M09*         | Coolant pump OFF                                                                  | After                  |                  |                   |
|       | M06          | Tool change                                                                       | Simultaneous           | One-shot         | Possible          |
|       | M96          | Interruptive macro program                                                        | -                      | Modal            | Impossible        |
|       | M97*         | Cancel interruptive macro program                                                 |                        |                  |                   |
|       | M98          | Sub Program Call                                                                  | -                      | -                | Impossible        |
|       | M99          | Return from Subprogram                                                            | -                      |                  |                   |
|       | M120         | TOUCH signal check (Error when Off.)                                              | After                  | One-shot         | Possible          |
|       | M121         | TOUCH signal check (Error when On.)                                               |                        |                  |                   |
|       | M141*        | Selection of spindle                                                              | Simultaneous           | Modal            | Possible          |
|       | M142         | Selection of turning spindle                                                      |                        |                  |                   |
|       | M159         | Prohibit reading ahead                                                            | -                      | One-shot         | Impossible        |
|       | M200         | Tool breakage detection (with return motion)                                      | Simultaneous           | One-shot         | Possible          |
|       | M201         | Tool breakage detection                                                           | Simultaneous           | One-shot         | Possible          |
|       | M203         | Tool breakage detection                                                           | -                      | One-shot         | Impossible        |
|       | M211         | Workpiece counter 1 set                                                           | Simultaneous           | Modal            | Possible          |
|       | M221*        | Workpiece counter 1 cancel                                                        |                        |                  |                   |
|       | M212         | Workpiece counter 2 set                                                           | Simultaneous           | Modal            | Possible          |
|       | M222*        | Workpiece counter 2 cancel                                                        |                        |                  |                   |
|       | M213         | Workpiece counter 3 set                                                           | Simultaneous           | Modal            | Possible          |
|       | M223*        | Workpiece counter 3 cancel                                                        |                        |                  |                   |
|       | M214         | Workpiece counter 4 set                                                           | Simultaneous           | Modal            | Possible          |
|       | M224*        | Workpiece counter 4 cancel                                                        |                        |                  |                   |
|       | M230*        | Tool life counter set                                                             | Simultaneous           | Modal            | Possible          |
|       | M231         | Tool life counter cancel                                                          |                        |                  |                   |
|       | M238         | To start waveform display measurement                                             | -                      | One-shot         | Impossible        |
|       | M239         | To finish waveform display measurement                                            |                        |                  |                   |
|       | M241 to M249 | Tap time constant 10 ~ 90%                                                        | Simultaneous           | Modal            | Possible (*)      |
|       | M250         | Tap time constant selection                                                       |                        |                  |                   |
|       | M251         |                                                                                   |                        |                  |                   |
|       | M252 to M254 | Tap acceleration kept constant selection<br>High speed / Medium speed / Low speed |                        |                  |                   |
|       | M260 to M267 | High precision mode A On (levels 1 ~ 8)                                           | Simultaneous           | Modal            | Possible          |
|       | M280 to M287 | High precision mode B On (levels 1 ~ 8)                                           | Before                 |                  |                   |
|       | M269*        | High precision mode OFF                                                           | Simultaneous           |                  |                   |
|       | M289         |                                                                                   |                        |                  |                   |
|       | M270*        | Positioning finish check distance setting off                                     | Before                 | Modal            | Possible          |
|       | M271 to M279 | Positioning finish check distance setting                                         |                        |                  |                   |
|       | M290*        | Tool replacement Z axis lower speed 100%                                          | Before                 | Modal            | Possible          |
|       | M291 to M293 | Tool replacement Z axis lower speed 1~3                                           |                        |                  |                   |
|       | M294         | Check tool wash filter                                                            | Before                 | One-shot         | Possible          |
|       | M295         | Diagnosis of tool wash liquid surface sensor failure                              | -                      | One-shot         | Impossible        |
|       | M298         | Machining mode specification                                                      | -                      | Modal            | Impossible        |
|       | M299         | Cancel machining mode specification                                               | -                      |                  |                   |

| Group | M code       | Contents                                                       | Timing of axis motions | Modal / One-shot | Multiple commands |  |  |  |
|-------|--------------|----------------------------------------------------------------|------------------------|------------------|-------------------|--|--|--|
|       | M303         | Forward rotation of turning spindle                            | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M304         | Backward rotation of turning spindle                           |                        |                  |                   |  |  |  |
|       | M305*        | To stop turning spindle                                        |                        |                  |                   |  |  |  |
|       | M320         | Measurement device sensor ON confirmation                      | -                      | One-shot         | Possible          |  |  |  |
|       | M321         | Measurement device sensor OFF confirmation                     |                        |                  |                   |  |  |  |
|       | M322*        | Finishing OFF                                                  | Before                 | Modal            | Possible          |  |  |  |
|       | M323         | Finishing ON                                                   |                        |                  |                   |  |  |  |
|       | M340*        | Machining load monitor OFF                                     | -                      | Modal            | ×                 |  |  |  |
|       | M341         | Machining load monitor ON                                      |                        |                  |                   |  |  |  |
|       | M342         | Machining load monitor ON(Max. only)                           |                        |                  |                   |  |  |  |
|       | M343         | Machining load monitor ON(Min. only)                           |                        |                  |                   |  |  |  |
|       | M350         | Thermal displacement compensation (X)                          | -                      | One-shot         | Impossible        |  |  |  |
|       | M351         | Thermal displacement compensation (Y)                          |                        |                  |                   |  |  |  |
|       | M352         | Thermal displacement compensation (Z)                          |                        |                  |                   |  |  |  |
|       | M353         | Thermal displacement compensation (XYZ)                        |                        |                  |                   |  |  |  |
|       | M355         | Thermal displacement compensation cancel                       |                        |                  |                   |  |  |  |
|       | M360         | Thermal distortion compensation (X-axis ball screw)            | -                      | One-shot         | Impossible        |  |  |  |
|       | M361         | Thermal distortion compensation (X-axis spindle)               |                        |                  |                   |  |  |  |
|       | M362         | Thermal distortion compensation (Y-axis ball screw)            |                        |                  |                   |  |  |  |
|       | M363         | Thermal distortion compensation (Y-axis spindle)               |                        |                  |                   |  |  |  |
|       | M364         | Thermal distortion compensation (Z-axis ball screw)            |                        |                  |                   |  |  |  |
|       | M365         | Thermal distortion compensation (Z-axis spindle)               |                        |                  |                   |  |  |  |
|       | M366         | Thermal distortion compensation (X-, Y- and Z-axes ball screw) |                        |                  |                   |  |  |  |
|       | M367         | Thermal distortion compensation (X-, Y- and Z-axes spindle)    |                        |                  |                   |  |  |  |
|       | M368         | Cancel thermal distortion compensation (Ball screw)            |                        |                  |                   |  |  |  |
|       | M369         | Cancel thermal distortion compensation (Spindle)               |                        |                  |                   |  |  |  |
|       | M380 to M387 | 3D smoothing start                                             | -                      | Modal            | Impossible        |  |  |  |
|       | M389         | 3D smoothing end                                               |                        |                  |                   |  |  |  |
|       | M400         | M400ON (chip shower ON)                                        | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M401*        | M400OFF (chip shower OFF)                                      |                        |                  |                   |  |  |  |
|       | M402         | M402ON                                                         | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M403*        | M402OFF                                                        |                        |                  |                   |  |  |  |
|       | M404         | M404ON                                                         | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M405*        | M404OFF                                                        |                        |                  |                   |  |  |  |
|       | M406         | M406ON                                                         | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M407*        | M406OFF                                                        |                        |                  |                   |  |  |  |
|       | M408         | M408ON                                                         | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M409*        | M408OFF                                                        |                        |                  |                   |  |  |  |
|       | M410         | Index of the pallet 2 to the outside                           | -                      | One-shot         | Possible          |  |  |  |
|       | M411         | Index of the pallet 1 to the outside                           |                        |                  |                   |  |  |  |
|       | M418         | Jig shower ON                                                  | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M419*        | Jig shower OFF                                                 |                        |                  |                   |  |  |  |
|       | M420*        | ATC arm turn speed (maximum speed)                             | Before                 | Modal            | Possible          |  |  |  |
|       | M421         | ATC arm turn speed (large tool speed)                          |                        |                  |                   |  |  |  |
|       | M432         |                                                                |                        |                  |                   |  |  |  |
|       | M422         | ATC arm turn speed 1                                           |                        |                  |                   |  |  |  |
|       | M423         | ATC arm turn speed 2                                           |                        |                  |                   |  |  |  |
|       | M430         | QT-axis (C-axis) unclamp                                       | -                      | Modal            | Possible          |  |  |  |
|       | M431*        | QT-axis (C-axis) clamp                                         |                        |                  |                   |  |  |  |
|       | M434         | Waiting for Pot + ATC arm shutter close                        | -                      | One-shot         | Impossible        |  |  |  |
|       | M435*        | Magazine rotation maximum speed                                | -                      | Modal            | Possible          |  |  |  |
|       | M436         | Magazine rotation speed 1                                      |                        |                  |                   |  |  |  |
|       | M437         | Magazine rotation speed 2                                      |                        |                  |                   |  |  |  |
|       | M438         | Magazine/Pot shutter open                                      | -                      | One-shot         | Impossible        |  |  |  |
|       | M439         | Magazine/Pot shutter close                                     |                        |                  |                   |  |  |  |

| Group | M code             | Contents                                                                                                                                                                           | Timing of axis motions | Modal / One-shot | Multiple commands |
|-------|--------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------|------------------|-------------------|
|       | M440               | Unclamp B axis                                                                                                                                                                     | Simultaneous           | Modal            | Possible          |
|       | M441*              | Clamp B axis                                                                                                                                                                       |                        |                  |                   |
|       | M442               | Unclamp A axis                                                                                                                                                                     | Simultaneous           | Modal            | Possible          |
|       | M443*              | Clamp A axis                                                                                                                                                                       |                        |                  |                   |
|       | M444               | Unclamp C axis                                                                                                                                                                     | Simultaneous           | Modal            | Possible          |
|       | M445*              | Clamp C axis                                                                                                                                                                       |                        |                  |                   |
|       | M450               | To output one-shot (Proceeds to the next block after the signal has turn off)<br><br>To output one-shot (Proceeds to the next block without waiting until the signal has turn off) | Simultaneous           | One-shot         | Possible          |
|       | M451               |                                                                                                                                                                                    |                        |                  |                   |
|       | M455               |                                                                                                                                                                                    |                        |                  |                   |
|       | M456               |                                                                                                                                                                                    |                        |                  |                   |
|       | M460               | Waiting for M460 signal ON                                                                                                                                                         | Simultaneous           | One-shot         | Possible          |
|       | M461               | Waiting for M460 signal OFF                                                                                                                                                        |                        |                  |                   |
|       | M462               | Waiting for M462 signal ON                                                                                                                                                         |                        |                  |                   |
|       | M463               | Waiting for M462 signal OFF                                                                                                                                                        |                        |                  |                   |
|       | M464               | Waiting for M464 signal ON                                                                                                                                                         |                        |                  |                   |
|       | M465               | Waiting for M464 signal OFF                                                                                                                                                        |                        |                  |                   |
|       | M466               | Waiting for M466 signal ON                                                                                                                                                         |                        |                  |                   |
|       | M467               | Waiting for M466 signal OFF                                                                                                                                                        |                        |                  |                   |
|       | M468               | Waiting for M468 signal ON                                                                                                                                                         |                        |                  |                   |
|       | M469               | Waiting for M468 signal OFF                                                                                                                                                        |                        |                  |                   |
|       | M474*              | Coil conveyor automatic mode: Enable                                                                                                                                               | Before                 | Modal            | Possible          |
|       | M475               | Coil conveyor automatic mode: Disable                                                                                                                                              | Before                 | Modal            | Possible          |
|       | M476               | Tool cleaning blowoff valve ON (MDI operation only)                                                                                                                                | -                      | One-shot         | Impossible        |
|       | M477               | Tool cleaning blowoff valve OFF (MDI operation only)                                                                                                                               | -                      | One-shot         | Impossible        |
|       | M478               | Automatic oiling / greasing cycle ON                                                                                                                                               | -                      | One-shot         | Impossible        |
|       | M479               | Liquid removal cycle ON                                                                                                                                                            | Simultaneous           | One-shot         | Impossible        |
|       | M480               | M480 ON                                                                                                                                                                            | Simultaneous           | Modal            | Possible          |
|       | M481*              | M480 OFF                                                                                                                                                                           |                        |                  |                   |
|       | M482               | M482 ON                                                                                                                                                                            | Simultaneous           | Modal            | Possible          |
|       | M483*              | M482 OFF                                                                                                                                                                           |                        |                  |                   |
|       | M484               | M484 ON                                                                                                                                                                            | Simultaneous           | Modal            | Possible          |
|       | M485*              | M484 OFF                                                                                                                                                                           |                        |                  |                   |
|       | M486               | M486 ON                                                                                                                                                                            | Simultaneous           | Modal            | Possible          |
|       | M487*              | M486 OFF                                                                                                                                                                           |                        |                  |                   |
|       | M490               | Coolant through center ON                                                                                                                                                          | Before                 | Modal            | Possible          |
|       | M491               | Coolant through center ON                                                                                                                                                          |                        |                  |                   |
|       | M492               | Coolant through center ON                                                                                                                                                          |                        |                  |                   |
|       | M493               | Coolant through center ON                                                                                                                                                          |                        |                  |                   |
|       | M494               | Coolant through center ON                                                                                                                                                          |                        |                  |                   |
|       | M495*              | Coolant through center OFF                                                                                                                                                         | After                  |                  |                   |
|       | M496               | Back washing cycle ON                                                                                                                                                              | Before                 | One-shot         | Possible          |
|       | M497               | Tool replacement tool washing OFF                                                                                                                                                  | Before                 | One-shot         | Possible          |
|       | M498               | Spindle air blow/tool washing ON                                                                                                                                                   | Simultaneous           | One-shot         | Impossible        |
|       | M499               | Spindle air blow/tool washing OFF                                                                                                                                                  |                        |                  |                   |
|       | M501 to M599       | Magazine rotation for tool setting position                                                                                                                                        | -                      | One-shot         | Impossible        |
|       | M800 to M899       | Signal output for PLC                                                                                                                                                              | Simultaneous           | One-shot         | Possible          |
|       | M900 to M999       | Extend signal output                                                                                                                                                               | Simultaneous           | One-shot         | Possible          |
|       | 2-digit BCD signal | BCD signal output                                                                                                                                                                  | Simultaneous           | One-shot         | Possible          |

(\*) The modal during power startup is specified in the user parameter (switch 1) <Tap accel. setting>.

## 12.2.1 Multiple M Code Commands in One Block

If the <Multiple M codes in one block> in the <User parameter> is set <1: Yes>, maximum tree M codes with “Possible” in “Multiple commands” of “12.2 M Code List” can be commanded together in one block.

Also when multiple M code commands are issued on the same block for axis travel, the operation for each M code is described in “Operation timing for axis travel” found in the “12.2 M Code List.” If multiple commands that operate with the same timing are issued on one block, they are output simultaneously.

If you need to be aware of the output order for the M codes, set it up so that commands are issued and broken up into multiple blocks.

In addition, simultaneously commandable M codes have limits. For combination of M codes, refer to the list below.

If simultaneously uncommandable M codes are commanded in one block, the <<Simultaneous command unable M code>> alarm appears.

|                     | Independent command | M06        | M430/M411  | M430/M431  | M45X       | M46X       | M8XX       | M9XX       | BCD        | Others     |
|---------------------|---------------------|------------|------------|------------|------------|------------|------------|------------|------------|------------|
| Independent command | Impossible          | Impossible | Impossible | Impossible | Impossible | Impossible | Impossible | Impossible | Impossible | Impossible |
| M06                 | -                   | Impossible | Impossible | Impossible | Possible   | Possible   | Possible   | Possible   | Possible   | Possible   |
| M430/M411           | -                   | -          | *5         | *5         | Possible   | Possible   | Possible   | Possible   | Possible   | *1         |
| M430/M431           | -                   | -          | -          | *5         | Possible   | Possible   | Possible   | Possible   | Possible   | Possible   |
| M45X                | -                   | -          | -          | -          | *2         | Possible   | Possible   | Possible   | Possible   | Possible   |
| M46X                | -                   | -          | -          | -          | -          | *3         | Possible   | Possible   | Possible   | Possible   |
| M8XX                | -                   | -          | -          | -          | -          | -          | Impossible | Impossible | Impossible | Possible   |
| M9XX                | -                   | -          | -          | -          | -          | -          | -          | Impossible | Impossible | Possible   |
| BCD                 | -                   | -          | -          | -          | -          | -          | -          | -          | Impossible | Possible   |
| Others              | -                   | -          | -          | -          | -          | -          | -          | -          | -          | *4         |

### 1. Independent commands

The M codes listed in the following table must be independently commanded and cannot be commanded together with other M codes simultaneously.

|      |      |      |      |
|------|------|------|------|
| M00  | M01  | M02  | M30  |
| M96  | M97  | M98  | M99  |
| M203 | M238 | M239 | M350 |
| M351 | M352 | M353 | M355 |
| M360 | M361 | M362 | M363 |
| M364 | M365 | M366 | M367 |
| M368 | M369 | M434 | M438 |
| M439 | M448 | M449 | M478 |
| M479 | M498 | M499 | M159 |
| M5xx | M478 | M298 | M299 |
| M476 | M477 |      |      |

The following M codes are independently commanded only in MDI operation.

|      |      |
|------|------|
| M200 | M201 |
|------|------|

### 2. M06

For M06 (tool change) and simultaneously commandable M codes, refer to “Chapter 5 5.7 Canned cycle for tool change (Non-stop ATC) (G100)”. (\*1)

### 3. Pallet-related M codes (M410, M411, M430, M431)

The pallet-related M codes (M410, M411, M430, M431) can be commanded in one block. (\*5)



4. One-shot output (M45x)  
One-shot output M codes (M450, M451, M455, M456) can be simultaneously commanded if they are different although they are one-shot output M codes. (\*2)  
Ex)  
M450, M451, M455: Commandable because they are all different  
M450, M455, M450: Uncommandable because M450 is duplicated
5. To wait for specific signal ON/OFF (M46x)  
The M codes (M460 to M469) that wait for a specific signal to go ON/OFF are simultaneously commandable if they are commanded for a same signal. (\*3)  
Ex)  
M460, M462, M464: Commandable because they wait for all other signals  
M460, M461, M462; Uncommandable because they are duplicated for M460 signal
6. Signal output to PLC (M8xx)  
Only one of the M codes (M800 to M899) that command to output a signal to PLC can be commanded in one block. In addition, they cannot be simultaneously commanded together with a 2-digit BCD signal output or an expansion signal output (M900 to M999).
7. Expansion signal output (M9xx)  
Only one of the M codes (M900 to M999) that outputs an expansion signal can be commanded in one block. In addition, they cannot be simultaneously commanded together with a 2-digit BCD signal output or a signal output (M800 to M899) to PLC.
8. 2-digit BCD signal output  
Only one of 2-digit BCD signal outputs can be commanded in one block. In addition, they cannot be simultaneously commanded together with a signal output (M800 to M899) to PLC or an expansion signal output (M900 to M999).
9. Others  
With respect to the combination of M codes other than the above, the commands other than the simultaneously commandable M codes like instruction of the same function ON/OFF, instruction of spindle motion, specification of time constant, command to coolant through center and the relatives, etc. can be simultaneously commanded together. (\*4)

M codes that can be a pair are as follows:

|                                        |
|----------------------------------------|
| M03, M04, M05, M19, M111               |
| M303, M304, M305                       |
| M08, M09                               |
| M120, M121                             |
| M141, M142                             |
| M200, M201                             |
| M211, M221                             |
| M212, M222                             |
| M213, M223                             |
| M214, M224                             |
| M230, M231                             |
| M241 to M249, M250, M251, M252 to M254 |
| M260 to M267, M269                     |
| M280 to M287, M289                     |
| M270, M271 to M279                     |
| M290, M291 to M293, M294               |
| M320, M321                             |
| M322, M323                             |
| M340 to M343                           |
| M400, M401                             |
| M402, M403                             |
| M404, M405                             |
| M406, M407                             |
| M408, M409                             |
| M418, M419                             |

|                              |
|------------------------------|
| M420,M421,M422,M423,M432     |
| M435, M436, M437             |
| M440, M441                   |
| M442, M443                   |
| M444, M445                   |
| M430, M431 (QT not equipped) |
| M480, M481                   |
| M482, M483                   |
| M484, M485                   |
| M486, M487                   |
| M490 to M494, M495, M496     |

## 12.3 M Function (Program Control)

### 12.3.1 Program Stop (M00)

The spindle stops after the commanded motions in a block are all finished.

The coolant pump is turned OFF at this time.

Next sequence is started by pressing the START switch.

**(Note)** Issue an M03 or M04 command if spindle rotation is required in the block after M00. Issue a command such as a coolant command when necessary. However, when the user parameter (switch 1) is set to the following, the spindle, lathe spindle and coolant automatically return when executing the following sequence.

- When <Spindle return method for program stop> is set to <0: Method 1>
- When <Lathe spindle return method for program stop> is set to <0:Method 1>
- When <Coolant return method for program stop> is set to <0: Method 1>

### 12.3.2 Optional Stop (M01)

When the [OPT STOP] key is set ON, similar to the M00, the automatic operation is stopped after a block which contains M01 is executed.

### 12.3.3 End of Program (M02, M30)

This code shows the end of program. Executing this command takes the control return to the head of the program. The NC enters the reset status at this time.

**(Note)** If other G or M code is commanded in the M02 or M30 block, an alarm appears. The axis movement is not available even by commanding X, Y or Z address.

### 12.3.4 Workpiece Counter Specification (M211 to M214)

If a code (M211 to M214) is specified to a workpiececounter (1 to 4) and commanded in memory operation, the commanded counter is counted up per counted amount when M02 or M30 is executed.

The counter is cancelled when power is starated, [RST] key is pushed, M02 or M30 is executed, operation is reset, or workpiece counter cancellation (M221-M224) is executed.

Ex) If M211 and M212 are executed in an operation program when the counted number of the counter 1 is 1 and the counted number of the counter 2 is 2, the counter 1 counts up per 1 and the counter 2 counts up per 2 when M02 or M30 is executed.

**(Note)** M211 to M214 can be commanded in MDI operation.

### 12.3.5 Workpiece Counter Cancel (M221 to M224)

To canel a workpiece counter (1-4), command an M code (M221-M224). Then, the commanded counter cancels counting when the M code (M221-M224) is executed in memory operation or MDIoperation.

### 12.3.6 To Prohibit Reading Ahead (M159)

To prohibit reading a program ahead, command M159. Then, the next block is not read ahead until the operation completes.

## 12.4 M Function (Signal Control)

### 12.4.1 2-digit BCD Signal Output

Specify any of the signal outputs 00 to 99 to output a 2-digit value to an external device by a BCD code.

However, the code (M00, M01, M02, M03, M04, M05, M06, M08, M09, M19, M29, M30, M96, M97, M98, M99) are internal use codes, so they are not output outside.

**(Note)** To customers who use a machine model with CNC-A00/B00  
After the control device CNC-C00, M96-M97 became internal use codes. If they are used, change the BCDoutput codes.

### 12.4.2 To Check TOUCH Signal (M120, M121)

When the M120 command is issued, the input signal TOUCH (touch sensor) is checked and if it is turned ON, the operation ends. If it is OFF, the alarm <<Tool breakage error>> is triggered and the TOOL (tool error) is output.

When an M121 command is issued, if the same signal is OFF, the operation ends. If it is ON, the alarm <<Tool breakage error>> is triggered.

### 12.4.3 M Signal Level Outputs (M400 to M409, M480 to M487)

| External input terminal name | ON   | OFF  |
|------------------------------|------|------|
| M400                         | M400 | M401 |
| M402                         | M402 | M403 |
| M404                         | M404 | M405 |
| M406                         | M406 | M407 |
| M408                         | M408 | M409 |
| M480                         | M480 | M481 |
| M482                         | M482 | M483 |
| M484                         | M484 | M485 |
| M486                         | M486 | M487 |

- M401 commands a chip shower to go OFF after the <Chip shower retention time> of the <User parameter> elapses.

### 12.4.4 One-shot Output (M450, M451, M455, M456)

M450 and M451 commands proceed to the next block after output time has passed and the signal has turned off.

M455 and M456 commands proceed to the next block without waiting until the signal turns off. Set the signal output time by the “One-shot signal M450/451/455/456 output time” of the <User parameter>.

### 12.4.5 Waiting Until Response is Given (M460 to M469)

M460, M462, M464, M466, or M468 command waits until M460, M462, M464, M466, or M468 signal turns ON.

M461, M463, M465, M467, or M469 command waits until M460, M462, M464, M466, or M468 signal turns OFF.

The maximum waiting time is set by <External signal reading time limit> of <User parameter>. After this time, the <<Signal output timeout>> alarm appears.

### 12.4.6 Signal Output to PLC (M801 to M899)

Output a BCD code in the lowest two digits to PLC.

Please refer to “PLC System Manual Chapter 6 OM” for details.

### 12.4.7 Expansion Signal Output (M900 to M999)

Output the M900 signal and a BCD code in the lowest two digits.

### 12.4.8 Checking Measurement Instrument Detection Signal (M320, M321)

When the M320 command is issued, the input signal status for TOUCH (touch sensor) is checked when the preceding skip command (G31, G131/132) is executed. If the signal is not ON, the alarm <<Detection signal off>> is triggered. The signal turns OFF after a reset, after switching to machine lock or when the power is turned OFF.

When the M321 command is issued, the input signal TOUCH (touch sensor) is checked and if it is ON, the alarm <<Detection signal on>> is triggered.

## 12.5 M Function (Additional Axis Control)

### 12.5.1 Pallet-related M Codes (M410, M411, M430, M431)

This function is available for QT mounted machine.

When M410 is commanded, the Z-axis is returned to its origin and then the QT-axis is indexed to -180° (pallet 2 is positioned outside).

When M411 is commanded, the Z-axis is returned to its origin and then the QT-axis is indexed to 0° (pallet 1 is positioned outside).

When M430 is commanded, the QT axis is unclamped by force.

When M431 is commanded, the QT axis is clamped by force.

### 12.5.2 C-axis Unclamp/Clamp (M444/M430 and M445/M431)

When the M444 command is issued, the C-axis is unclamped, and thereafter, the C-axis unclamp/clamp is not automatically controlled. When there is an axis travel command on the same block, the axis travel is carried out at the same time. However, when there is a C-axis travel command on the same block, the alarm <<Invalid command>> is triggered.

Example 1: G00 X-50. M444; ← When it travels along the X-axis, the C-axis unclamp is carried out at the same time.

Example 2: G00 C90. M444; ← The alarm <<Invalid command>> is triggered.

**(Note 1) When there is a command on the same block as the axis travel command, the axis travel starts before the unclamp is finished.**

**(Note 2) If the alarm <<\*-axis clamp retry attempt was made>> is triggered by a clamp command (M431, M445), it is possible that there is an impact from the machining load during the unclamp that is right before. We recommend decreasing the machining load or machining while clamped.**

When the M445 command is issued, the C-axis is clamped, and thereafter, the C-axis unclamp/clamp is automatically controlled. When there is an axis travel command on the same block, the axis travel is carried out at the same time. However, when there is a C-axis travel command on the same block, the alarm <<Invalid command>> is triggered.

Example 1: G00 X-50. M445; ← When it travels along the X-axis, the C-axis clamp is carried out at the same time.

Example 2: G00 C90. M445; ← The alarm <<Invalid command>> is triggered.

**(Note 3) When there is a command on the same block as the axis travel command, the axis travel starts before the clamp is finished.**

When C-axis operation is enabled in the <Machine parameter> setting and when the clamp mechanism is set to <1: Type 2> or <2: Type 3>, this command is valid.

When configured to another setting, the alarm\* <<Specified M code cannot be used>> is triggered.

When the C-axis changes from a clamp status to an unclamp status using an M444 command, the C-axis returns to a clamp status if an M02 (M30) command is issued, if the reset key is pressed or if a stop level 5 alarm is triggered. It does not return to clamp status even when the operation reset is performed.

However, when the machine is not equipped with QT, the M430 has the same function as M444, and M431 has the same function as M445.

\* When the user parameter <Unregistered M-code> is set to <0: Error>.

### 12.5.3 B-axis Unclamp/Clamp (M440 and M441)

When the M440 command is issued, the B-axis is unclamped, and thereafter, the B-axis unclamp/clamp is not automatically controlled. When there is an axis travel command on the same block, the axis travel is carried out at the same time. However, when there is a B-axis travel command on the same block, the alarm <<Invalid command>> is triggered.

Example 1: G00 Y-50. M440; ← When it travels along the Y-axis, the B-axis unclamp is carried out at the same time.

Example 2: G00 B90. M440; ← The alarm <<Invalid command>> is triggered.

**(Note 1) When there is a command on the same block as the axis travel command, the axis travel starts before the unclamp is finished.**

When the M441 command is issued, the B-axis is clamped, and thereafter, the B-axis unclamp/clamp is automatically controlled. When there is an axis travel command on the same block, the axis travel is carried out at the same time. However, when there is a B-axis travel command on the same block, the alarm <<Invalid command>> is triggered.

Example 1: G00 Y-50. M441; ← When it travels along the Y-axis, the B-axis clamp is carried out at the same time.

Example 2: G00 B90. M441; ← The alarm <<Invalid command>> is triggered.

**(Note 2) When there is a command on the same block as the axis travel command, the axis travel starts before the clamp is finished.**

When B-axis operation is enabled in the <Machine parameter> setting and when the clamp mechanism is set to <1: Type 2> or <2: Type 3>, this command is valid.

When configured to another setting, the alarm\* <<Specified M code cannot be used>> is triggered.

When the B-axis changes from a clamp status to an unclamp status using an M440 command, the B-axis returns to a clamp status if an M02 (M30) command is issued, if the reset key is pressed or if a stop level 5 alarm is triggered. It does not return to clamp status even when the operation reset is performed.

\* When the user parameter <Unregistered M-code> is set to <0: Error>.

### 12.5.4 A-axis Unclamp/Clamp (M442 and M443)

When the M442 command is issued, the A-axis is unclamped, and thereafter, the A-axis unclamp/clamp is not automatically controlled. When there is an axis travel command on the same block, the axis travel is carried out at the same time. However, when there is an A-axis travel command on the same block, the alarm <<Invalid command>> is triggered.

Example 1: G00 Z-50. M442; ← When it travels along the Z-axis, the A-axis unclamp is carried out at the same time.

Example 2: G00 A90. M442; ← The alarm <<Invalid command>> is triggered.

**(Note 1) When there is a command on the same block as the axis travel command, the axis travel starts before the unclamp is finished.**

When the M443 command is issued, the A-axis is clamped, and thereafter, the A-axis unclamp/clamp is automatically controlled. When there is an axis travel command on the same block, the axis travel is carried out at the same time. However, when there is an A-axis travel command on the same block, the alarm <<Invalid command>> is triggered.

Example 1: G00 Z-50. M443; ← When it travels along the Z-axis, the A-axis clamp is carried out at the same time.

Example 2: G00 A90. M443; ← The alarm <<Invalid command>> is triggered.

**(Note 2) When there is a command on the same block as the axis travel command, the axis travel starts before the clamp is finished.**

When A-axis operation is enabled in the <Machine parameter> setting and when the clamp mechanism is set to <1: Type 2> or <2: Type 3>, this command is valid.

When configured to another setting, the alarm\* <<Specified M code cannot be used>> is triggered.

When the A-axis changes from a clamp status to an unclamp status using an M442 command, the A-axis returns to a clamp status if an M02 (M30) command is issued, if the reset key is pressed or if a stop level 5 alarm is triggered. It does not return to clamp status even when the operation reset is performed.

\* When the user parameter <Unregistered M-code> is set to <0: Error>.

## 12.6 M Function (Positioning Check Distance)

### 12.6.1 Positioning Finished Check Distance (M270 to M279)

M270: Positioning finished check distance, positioning finished angle non-specific.

M271: This makes the value of <M271 positioning end check distance> or <M271 positioning end check angle> of the <User parameter> an imposition width in positioning.

X, Y, Z axes use the value of “positioning end check distance”. A, B, C axes use the value of “positioning end check angle”.

M272 to M279: These work as same as M271.

Return to the positioning end check distance, positioning end check angle non-specific to follow the operation below.

- Execution of M270
- Program ends
- Operation reset

## 12.7 M Function (Time Constant Switch)

### 12.7.1 Tap Time Constant Selection (M241 to M249, M250, M251, M252 to M254)

If M250/M251 is commanded, the Z axis acceleration/deceleration in tapping is carried out in time constant.

If any of M241 to M249 is commanded, the Z axis acceleration/deceleration in tapping is carried out in 10 to 90% of time constant.

However, if the <<Tap time constant too short>> alarm appears, operation works in time constant.

When a command between M252 and M254 is issued, the acceleration and deceleration of the Z-axis during a tapping operation is carried out at a constant acceleration (high, medium and slow speeds).

**(Note 1)** Use the low acceleration speed M253 or M254 if the alarm <<Servo error (overload)>> is triggered during a tapping operation for M252 modal. In addition, use the low acceleration speed M254 if the alarm <<Servo error (overload)>> is triggered during a tapping operation for M253 modal. Issue a dwell or similar command to lower the frequency of the tapping operation when the alarm or error persists.

**(Note 2)** The machine is initialized in the modal that is set in the user parameter (switch 1) <Tap accel. setting> when the power is turned ON, when the [RST] key is pressed, when the M02 or M30 command is executed and when the operation is reset.

**(Note 3)** M253 and M254 cannot be used in some of the models.



## **CHAPTER 13 (1)**

### **HIGH-ACCURACY MODE AIII**

- 1 Outline**
- 2 How to Use**
- 3 Restrictions**
- 4 Detailed Explanations and Adjustments of Parameters**

# 1 Outline

## 1.1 Outline of High-accuracy Mode AIII

The high-accuracy mode AIII is appropriate for contouring or 3-dimensional shape workpiece machining to attain high-speed, high-accuracy, and high-dignity machining. The high-accuracy mode AIII has two functions; 1: To improve shape accuracy and 2: To improve face dignity. It also can change shape accuracy and face dignity in machining by the override values in user parameters.

The parameter default settings have 6 machining levels available, that is, there is one level that focuses on shape accuracy or another that focuses on surface quality. The appropriate machining level can be easily selected depending on the machining needs or requirements. In addition, unique two levels of parameter setting are available.

The machining level can be changed in an NC program.

## 1.2 Functions of High-accuracy Mode AIII

The shape accuracy improving function and the face dignity improving function of the high-accuracy mode AIII are as follows:

### 1. Improving shape accuracy

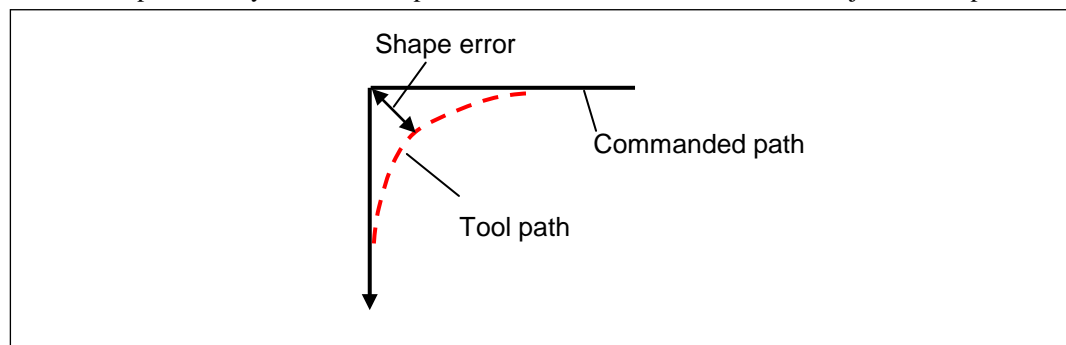
The following 4 deceleration functions are available for improving the shape accuracy.

These functions control the margin of error within a certain value regardless of the shape.

- (1) Automatic corner deceleration function
- (2) Automatic arc deceleration function
- (3) Automatic curve approximation deceleration function
- (4) Fully automatic deceleration function

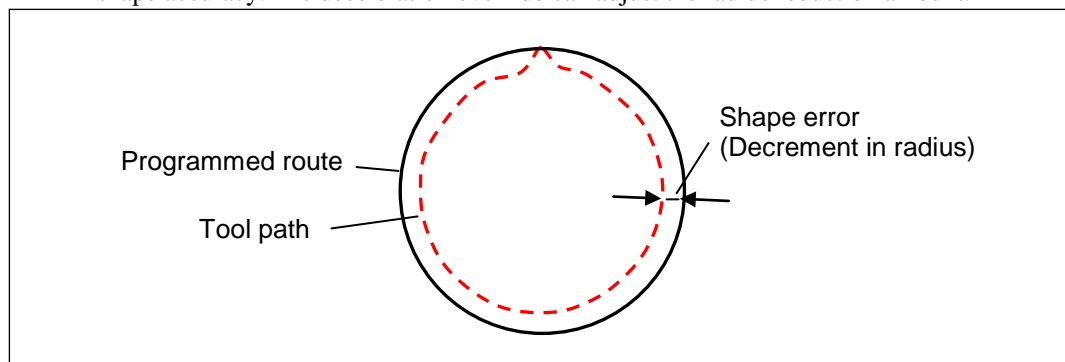
#### (1) Automatic corner decelerating function

This function restricts errors of a command path and tool path at a corner part to improve shape accuracy of the corner part. Corner deceleration override can adjust the shape error.



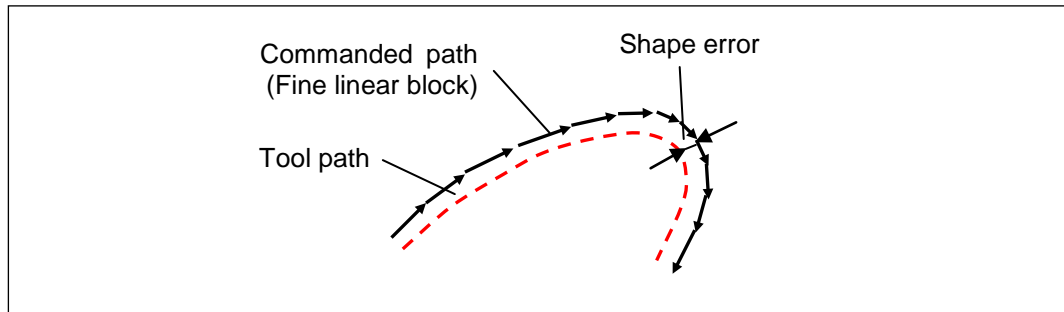
#### (2) Automatic arc decelerating function

This function restricts the radius from becoming small in arc machining to improve arc shape accuracy. Arc deceleration override can adjust the radius reduction amount.



## (3) Automatic curve approximation decelerating function

This function can restrict errors of a curve command path and tool path commanded by a minute linear block to improve shape accuracy of a curved shape and 3-dimensional shape. Curve approximate deceleration override can adjust the shape error.



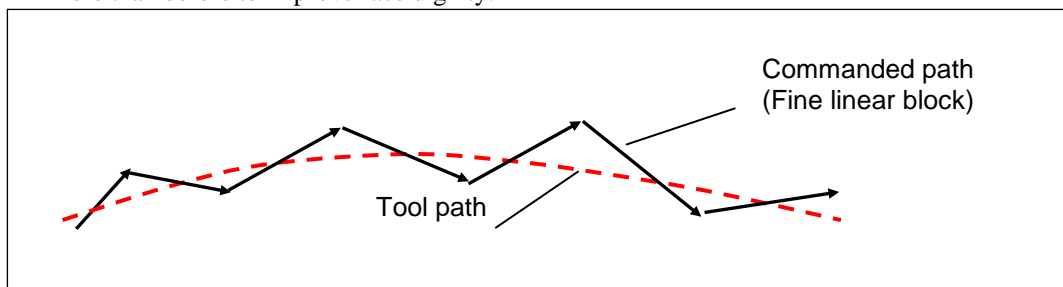
## (4) Fully automatic deceleration function

This function limits all errors in the command paths and tool paths in order to improve the shape accuracy. The user parameter (high accuracy) <Accuracy level> can adjust the margin of error on the shape.

## 2. To improve face dignity

The function to improve face dignity is a smooth function.

The command path created for machining a curved shape or 3-dimensional shape by CAM or the like is commanded linearly, so it becomes polygonal to generate stripes (parallel grooves) or the like and drops face dignity. The following functions can smoothen the machining face more than before to improve face dignity:



The functions are two as follows:

## (1) Smooth path offset function

This function changes a program command path to an approximate curve.

In addition, it switches the function to valid or invalid by a travel distance per block or an angle difference between blocks.

## (2) Smooth override (to adjust cutting time constant)

This function adjusts the smoothness by adjusting the cutting time constant. In addition, this change of cutting time constant can adjust the top speed of cutting used in the high-accuracy AIII.

In addition, the “High-accuracy mode AIII” is represented as “High-accuracy mode A” in the chapters or NC screens after this.

## 2 How to Use

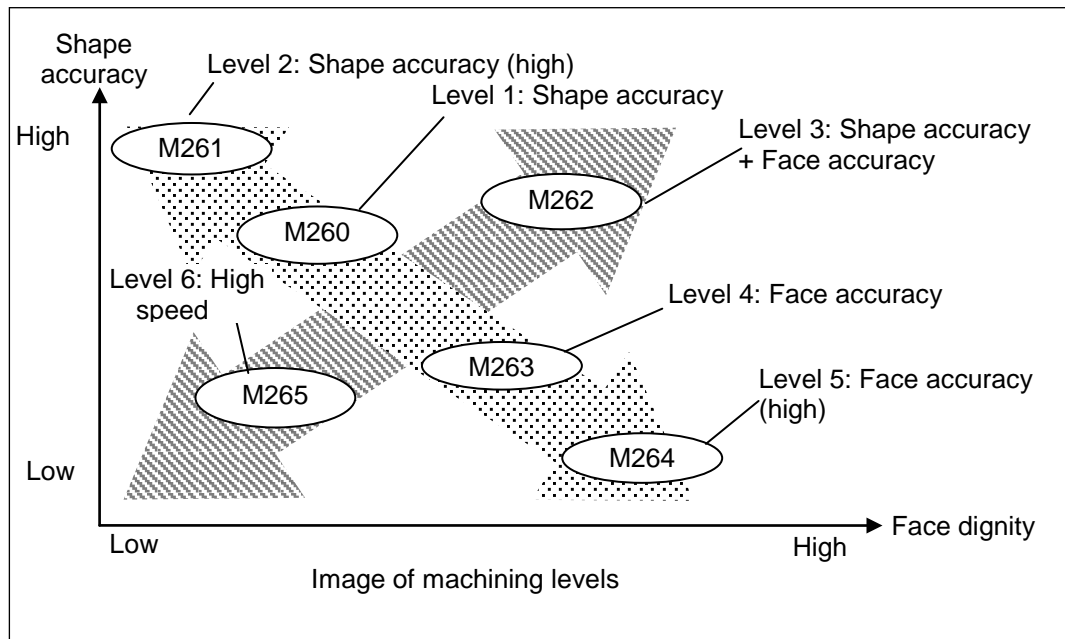
### 2.1 To Select a Machining Level

High accuracy mode A is equipped with a function to improve shape accuracy and another to improve surface quality. There are 6 machining levels available in the default settings, such as one that focuses on shape accuracy and another that focuses on surface quality. Select one from these six machining levels depending on the machining content. (Note, in the default settings, the function that improves the surface quality uses smooth override.)

In addition, there are two more levels other than the above levels that can be defined by users so that the machining levels can be adjusted to the optimum. For detail, refer to the “4 Detailed explanations and adjustments of parameter”.

Machining level default setting

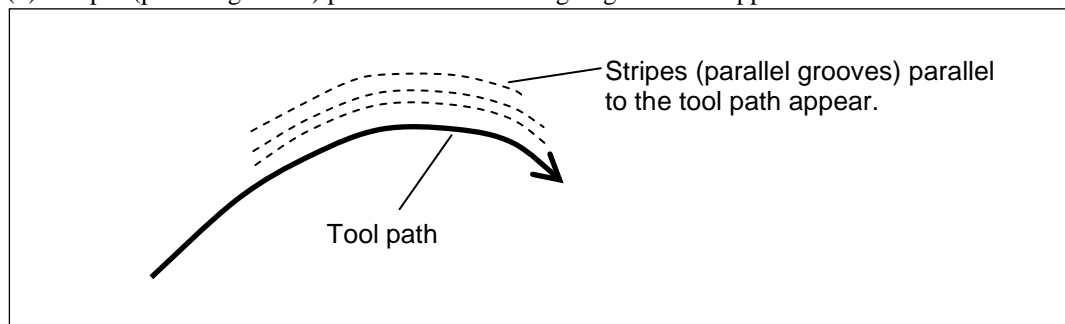
| Machining level | Purpose (Emphasis)            | Contents                                                                                                                                                                                                                                                                                                                   | M code       |
|-----------------|-------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------|
| 1               | Shape accuracy                | The shape accuracy is emphasized.<br>(First recommended for emphasizing shape accuracy)<br><br>The surface quality is equivalent to the level when high accuracy mode is not used. Machining time becomes a little longer because it decelerates in accordance with a shape.                                               | M260         |
| 2               | Shape accuracy (High)         | The shape accuracy is more emphasized than the machining level 1.<br><br>The surface quality is less than when high accuracy mode is not used. Machining time becomes a little longer because it decelerates in accordance with a shape. In addition, the top speed of a command is limited. (F5000 is the default value.) | M261         |
| 3               | Shape accuracy + face dignity | Both the shape accuracy and face dignity are emphasized.<br><br>As the deceleration time become longer, the machining time may become very long.                                                                                                                                                                           | M262         |
| 4               | Face dignity                  | The face dignity is emphasized (first recommended for emphasizing face dignity).<br><br>Use this if smooth face dignity is necessary. However, shape accuracy becomes bad.                                                                                                                                                 | M263         |
| 5               | Face dignity (High)           | The face dignity is emphasized more than the machining level 4.<br><br>Use this if smoother face dignity is necessary. However, shape accuracy becomes worse.                                                                                                                                                              | M264         |
| 6               | High speed                    | High speed is emphasized.<br><br>The shape accuracy is a little higher than when high accuracy mode is not used. The face dignity is in the same level. If you are not sure of this shape accuracy emphasis or the face dignity emphasis, it is recommended to use this machining level.                                   | M265         |
| 7/8             | User definition               | If it is necessary to set other than the above six, set a value of the user parameter referring to the parameter values of other levels.                                                                                                                                                                                   | M266 to M267 |



Points for selection of machining level

1. Select the shape accuracy emphasis or the face dignity emphasis  
Decide which of the shape accuracy or the face dignity is to be emphasized. If the shape accuracy is emphasized, the tool path is faithfully controlled by the command path. If the face dignity is emphasized, it is controlled to improve smoothness of the machining face.
  - Shape accuracy emphasis  
→ Select the machining level 1 (shape accuracy, M260).
  - Face dignity emphasis  
→ Select the machining level 4 (face dignity, M263).
  - If you are not sure which should be emphasized  
→ Select the machining level 1 (shape accuracy, M260) or machining level 6 (high-speed, M265).
2. Decide which of the shape accuracy or face dignity is necessary for the machining.  
Depending on the stripes (parallel grooves) appearing on the machining face, you may know which of the shape accuracy or the face dignity is necessary to be emphasized.

(1) Stripes (parallel grooves) parallel in the tool ongoing direction appear.



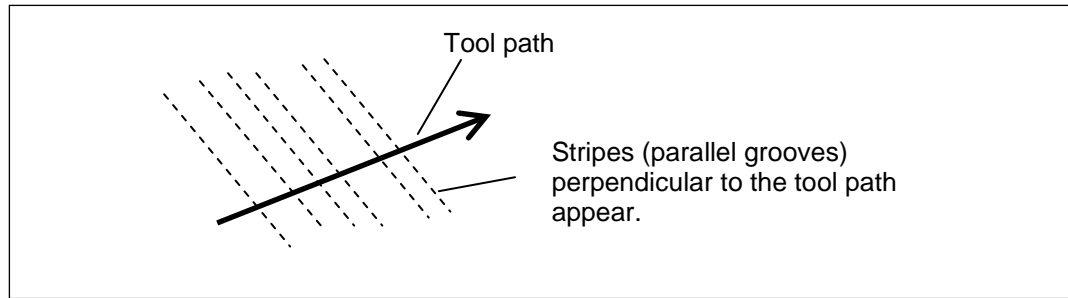
Presumed cause

There is a possibility that the stripes (parallel grooves) may appear due to a shape error. Especially they tend to appear if the commanded speed is too fast or the curvature is too small.

Recommend machining level

Machining level 1 (shape accuracy, M260)

- (2) Stripes (parallel grooves) perpendicular to the tool path in the ongoing direction appear.



### Presumed cause

The minute linear command created by CAM or the like for a curved shape is polygonal, so it tends to generate stripes (parallel grooves). Especially they tend to appear if the speed is too slow or the curvature is too large. They may be none by minimizing the tolerance of CAM.

### Recommend machining level

Machining level 4 (face dignity, M263)

- (3) If you want to emphasize the shape accuracy or the face dignity more  
Use the machining level 2 (shape accuracy (High), M261) or the machining level 5 (face dignity (High), M264). However, the machining level 2 may take more machining time than the machining level 1. The machining level 5 worsens the shape accuracy more than the machining level 4.
- (4) The machining level 3 (shape accuracy + face dignity, M262) may take very long machining time.

## 2.2 How to Use the Program

To use the high-accuracy mode A, use the following M codes:  
M260-M267: High accuracy mode A ON  
M269: High accuracy mode OFF

Example of Use

(Program Example)  
NC program  
G00 X0 Y0 Z0;  
;  
M260;  
G01 X20. Y30. Z50.;  
X40. Y20. Z30.;  
;  
M269;  
;  
M261;  
G01 X20. Y30. Z50.;  
X40. Y20. Z30.;  
;  
M269;  
;  
M262;  
G01 X20. Y30. Z50.;  
X40. Y20. Z30.;  
;  
M269;  
M30;

← High-accuracy mode A (Level 1) ON

} High-accuracy mode A (Level 1) in operation

← High-accuracy mode A (Level 1) OFF

← High-accuracy mode A (Level 2) ON

} High-accuracy mode A (Level 2) in operation

← High-accuracy mode A (Level 2) OFF

← High-accuracy mode A (Level 3) ON

} High-accuracy mode A (Level 3) in operation

← High-accuracy mode OFF

## 2.3 Usable Conditions

In order to use the high-accuracy mode A, it is necessary for the modal status to be as follows:  
In addition, the power is supplied under this condition.

M97: To cancel interruptive macro program  
M299: Cancel machining mode specification

## 2.4 Conditions to be Cancelled

The high-accuracy mode A is set off if the following operations are done whilst the high-accuracy mode A is in action:

- Power is supplied.
- The [RST] key is pressed.
- Memory operation is reset by pressing the [Z.RTN] key or the like in the manual operation mode.
- End of program (M02, M30) is performed.

## 3 Restrictions

### 3.1 Commandable Functions

The functions available using commands while high accuracy mode A is executing are as follows.

1. All M code commands are possible except M96 (interrupt type macro).
2. All the G code commands are available when using the automatic corner deceleration function, the automatic arc deceleration function or the automatic curve approximation deceleration function. G code commands except the involute interpolation (G02.2/G03.2) command are available on the fully automatic deceleration function.

**(Note 1)** When a function command is accidentally issued even though it is not available, the alarm <<High accuracy A invalid command>> is triggered.

**(Note 2)** The high-accuracy mode A is temporarily cancelled by the following commands (modes). When the commands (modes) finish, the high-accuracy mode A becomes valid again.

- M code commands
- Blocks which have no movement amounts
- Blocks which have no cutting feed movement
- Every rotary feed
- Fixed cycle
- Thread cutting
- Thread cutting cycle
- Complex thread cutting cycle
- Exact stop mode
- Skip feed (G31, G131 and G132)
- Measurement related G code commands (G120 to G129)
- Inverse time feed

**(Note 3)** The smooth path offset function in the high-accuracy mode A is temporarily cancelled by the following commands (modes). When the commands (modes) finish, the high-accuracy mode A becomes valid again.

- M code commands
- Blocks which have no movement amounts
- Blocks which have no cutting feed movement
- Every rotary feed
- Fixed cycle
- Thread cutting
- Thread cutting cycle
- Complex thread cutting cycle
- Circular interpolation
- Involute interpolation
- Exact stop mode
- Tool diameter offset mode
- Scaling
- Mirror image
- Coordinate rotational function
- Feature coordinate manufacturing mode
- Inverse time feed



## 3.2 Additional Axis Movement Commands

If an additional axis movement is commanded while the high-accuracy mode A is in action, the high-accuracy mode A is temporarily cancelled only while the command is in action. In addition, the commands that move the feed axes and additional axis simultaneously by the cutting feed (G01, G02, G03) are invalid.

## 3.3 Imposition Checked by High-accuracy Mode A Command

If any of the following conditions is satisfied, an imposition is checked by an M code command in the high-accuracy mode A:

- M code commands (M260-M269) alone in the high-accuracy mode A
- Commands changed to the high-accuracy mode A (M260-M267) from a modal state other than the high-accuracy mode A
- When a value of the <Cutting feed time constant selection> or <Smooth override> changes

Example) M260G1X-10.Y-10.

G1X-20.Y-20.

M261G1X-30.Y-30...(1)

G1X-40Y-40

The program like above checks the imposition of the block (1) before operation if the value of the User parameter (Switch 1) <Cutting feed time constant selection> or <Smooth override> of M260 or M261 differs.

## 3.4 Feedrate When Commanding of High-accuracy Mode A

The feedrate when commanding the high-accuracy mode A is limited by the machine parameter (high-accuracy) <Reference feedrate A>.

In addition, the limited speed can be adjusted by setting the user parameter (high-accuracy) <Smooth override>. (For details, refer to “ 4.1 (4) Smooth override function (To adjust a cutting time constant)”).

However, if a speed faster than the machine parameter (high accuracy) <Max. feedrate A> is commanded, the <<Feedrate error>> appears.

The machine parameter (system 1) <X- to Z-axis max. cutting feedrates > and the user parameter (switch 1) <Max. actual cutting travel speed> are not used in the high-accuracy mode A.

## 3.5 Notes in Motions by Smooth Path Offset Function

- Operates only when the user parameter (high accuracy) <Smooth path offset function> is set to <1: Valid>.
- The block stop position in a single operation differs from the program command position.
- Mode change is unavailable.
- Dry run is unchangable. If changed, the operator message <<The spline function is valid.>> appears.
- A block that is cutting with the smooth path offset function cannot be specified as a block for resuming or restarting the program. If it is specified, the alarm <<Program restart error>> is triggered.
- If the user parameter (high accuracy) <Smooth path offset cancel angle> is set too large, the alarm <<Curve speed error>> may be triggered.

## 4 Detailed Explanations and Adjustments of Parameters

### 4.1 Detailed Explanations

We are going to explain the following functions provided in the high-accuracy mode A:

- (1) Automatic corner decelerating function
- (2) Automatic arc decelerating function
- (3) Automatic curve approximation decelerating function
- (4) Fully automatic deceleration function
- (5) Smooth override (Cutting time constant adjusting method)
- (6) Smooth path offset function

- (1) Automatic corner decelerating function

#### Function

If a tool approaches a corner in corner machining, the actual tool path moves away from the command path a little by little. It is an error. In addition, the faster the feedrate is at a corner, the larger the error becomes.

This function automatically decelerates only the feedrate near the corner according to the setting value of <Corner deceleration override> so that the shape accuracy is kept constant with respect to any corner specified in the NC program.

#### Setting

The setting area of the <Corner deceleration override> is up to “0% or 10%-9999%”. The smaller the setting value is, the more strongly the deceleration function works to minimize the error and makes the shape accuracy better. In addition, if you input “0%”, the automatic corner deceleration function goes off.

#### Guide amount of shape error

The shape error at a corner becomes a value in proportion to the <Corner deceleration override>. For example, if the error amount when the <Corner deceleration override> is set at 100% is 100, it becomes 200 when set at 200%, and it becomes 50 when at 50%.

- (2) Automatic arc decelerating function

#### Function

For machining of arc interpolation, the actual tool path has an error in the command in the radius direction to minimize the arc radius. In addition, the faster the feedrate is, the bigger the error tends to become.

This function automatically decelerates the feedrate of circular cutting according to the setting values of <Arc deceleration override> so that the shape accuracy in the radial direction is kept constant with respect to any arc specified in the NC program.

#### Setting

The setting area of the <Arc deceleration override> is up to “0% or 10%-9999%” (Default value “100%”). The smaller the setting value is, the more the deceleration function works strongly to minimize the error and the shape accuracy becomes better. In addition, if you input “0%”, the automatic arc deceleration function goes off.

#### Guide amount of shape error

The shape error at an arc becomes a value in proportion to the <Arc deceleration override>. For example, if the error amount when the <Arc deceleration override> is set at 100% is 100, it becomes 200 when set at 200%, and it becomes 50 when set at 50%.

## (3) Automatic curve approximation decelerating function

Function

This function automatically decelerates the feedrate at the curve approximate block part according to the setting value of the <Curve approximate deceleration override> so that the shape accuracy is kept fixed for any curve approximate block (where the curve is represented by the group of the minute linear blocks) commanded by an NC program.

Setting

The setting area of the <Curve approximate deceleration override> is up to "0% or 10%-9999%" (Default value is "100%"). The smaller the setting value is, the more strongly the deceleration function works to minimize the error and the shape accuracy becomes better. In addition, if you input "0%", the automatic curve approximate deceleration function goes off.

Guide amount of shape error

The shape error at a curve becomes a value in proportion to the <Curve approximate deceleration override>. For example, if the error amount when the <Curve approximate deceleration override> is set at 100% is 100, it becomes 200 when set at 200%, and it becomes 50 when set at 50%.

## (4) Fully automatic deceleration function

Function

This function automatically decelerates the feedrate in order to limit the margin of error from the command path within the value set in the <Accuracy level> on corner, arc and curve approximation blocks that are specified in the NC program.

Set value

The setting range for the <Accuracy level> is from <0 (Invalid)> to 5.000 mm (when the user parameter (switch 1) <Machine unit system> is set to <1: Inch>, then the range is from <0 (Invalid)> to 0.1999 inch) (Default value: <0 (Invalid)>).

When the parameter is set to <0 (Invalid)>, the functions: (1) automatic corner deceleration function, (2) automatic arc deceleration function and (3) automatic curve approximation deceleration function are enabled. When it is configured to another setting, (1) automatic corner deceleration function, (2) automatic arc deceleration function and (3) automatic curve approximation deceleration function are disabled and fully automatic deceleration function is enabled.

Approximation for margin of error on shape

The set value for the <Accuracy level> is an approximation for the margin of error.

## (5) Smooth override function (To adjust a cutting time constant)

Function

This function is to improve the face dignity. The command path created by CAM or the like in machining a curved shape or 3-dimensional shape becomes a linearly commanded continual polygon to have stripes (parallel grooves) or drops the face dignity. This function makes the machining face smoother than before and improves the face dignity.

The <Smooth override> can adjust the face dignity.

Setting

The setting area of the <Smooth override> is up to "10%-9999%" (Default value is "100%"). If it becomes bigger than 100, smoother face dignity is attained. If it becomes smaller than 100, the shape accuracy becomes better.

Guide amount of shape error

Shape errors at a corner, arc, and curve vary depending on the settings of the <Smooth override>.

The shape error at a corner becomes a value in proportion to the square root of the setting value. For example, if the error amount when the setting value is set at 100% is 100, it becomes 140 when set at 200%, and it becomes 70 when set at 50%.

The shape error at an arc or curve becomes a value in proportion to the setting value. For example, if the error amount when the setting value is set at 100% is 100, it becomes 200 when set at 200% and it becomes 50 when set at 50%.

To adjust the limit value of cutting feedrate

The cutting feedrate that can be commanded in a program is limited as follows:

- If the <Smooth override> is 100% or more, the limit value becomes double of the machine parameter (high accuracy) “Reference feedrate A”  $\times \sqrt{(\text{Smooth override value}/100)}$ . (The amount after the decimal point is rounded off.)
- If the <Smooth override> is less than 100%, the limit value becomes double of the machine parameter (high accuracy) “Reference feedrate A”  $\times (\text{Smooth override value}/100)$ . (The amount after the decimal point is rounded off.)

If the smooth override value is set at 100% or more, the cutting feedrate can be raised. In addition, if the limit value exceeds the machine parameter (high accuracy) <Max. feedrate A>, it is clamped by the <Max. feedrate A>.

Example) Limit value of cutting feedrate for the reference feedrate A: 10000 mm/min, Max. feedrate A: 20000 mm/min

|                  |      |                |
|------------------|------|----------------|
| Smooth override: | 100% | : 10000 mm/min |
|                  | 400% | : 20000 mm/min |
|                  | 50%  | : 5000 mm/min  |

### (6) Smooth path offset function

#### Function

This function improves the face dignity as same as the smooth override function.

It makes a program, which creates a curved shape from minute blocks, create a curve line from multiple program command points, approximate the path, and perform curve interpolation.

This function changes the program command to a curve line, so the automatic deceleration is minimized compared with the smooth override.

The <Smooth path offset level> can adjust the face dignity.

#### Setting

The setting area of the <Smooth path offset level> is “1-5” (The default value is “1”). If this value is made bigger, the path is offset by more command points to have much smoother path. However, if it is made big, the path may error.

(If the user parameter (high accuracy) <Smooth path offset function> is “Invalid”, this setting value becomes invalid.)



(High accuracy mode A)

| Parameter name                            | Contents                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                          | Setting range  |
|-------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------|
| Corner deceleration override              | This is used together with the automatic corner deceleration function.<br>When "100" is set, automatic corner deceleration is performed based on the machine's unique deceleration rate. When a value larger than "100" is set, the deceleration rate is smaller, making machining time shorter. When a value smaller than "100" is set, the deceleration rate is larger, resulting in more accurate machining. When "0" is set, automatic corner deceleration is not performed.<br>When the <Accuracy level> is set to another value besides "0", this parameter is disabled.                                    | 0, 10~9999 (%) |
| Arc deceleration override                 | This is used together with the automatic arc deceleration function.<br>When "100" is set, automatic arc deceleration is performed based on the machine's unique deceleration rate. When a value larger than "100" is set, the deceleration rate is smaller, making machining time shorter. When a value smaller than "100" is set, the deceleration rate is larger, resulting in more accurate machining. When "0" is set, automatic arc deceleration is not performed.<br>When the <Accuracy level> is set to another value besides "0", this parameter is disabled.                                             | 0, 10~9999 (%) |
| Curve approximation deceleration override | This is used together with the automatic curve approximate deceleration function.<br>When "100" is set, automatic curve approximation deceleration is performed based on the machine's unique deceleration rate. When a value larger than "100" is set, the deceleration rate is smaller, making machining time shorter. When a value smaller than "100" is set, the deceleration rate is larger, resulting in more accurate machining. When "0" is set, automatic curve approximation deceleration is not performed.<br>When the <Accuracy level> is set to another value besides 0, this parameter is disabled. | 0, 10~9999 (%) |
| Smooth path offset level                  | This is used when the <Smooth path offset function> (explained later) is set <Valid>. The larger it is set, the smoother it becomes.                                                                                                                                                                                                                                                                                                                                                                                                                                                                              | 1~5            |
| Smooth override                           | When "100" is set, the cutting feed time constant selected by the <Cutting feed time constant selection> is used.<br>When a value larger than "100" is set, a smoother surface is attained, or when a value smaller than "100" is set, the accuracy becomes higher.                                                                                                                                                                                                                                                                                                                                               | 10~9999 (%)    |
| Cutting feed time constant selection      | The cutting feed time constant of the high-accuracy mode A is selected.<br><0: Group 1><br>Use <High accuracy A cutting feed time constant 1-3 (Group 1)> of the machine parameter (high accuracy).<br><1: Group 2><br>Use <High accuracy A cutting feed time constant 1-3 (Group 2)> of the machine parameter (high accuracy).<br><2: Group 3><br>Use <High accuracy A cutting feed time constant 1-3 (Group 3)> of the machine parameter (high accuracy).                                                                                                                                                       | 0~2            |
| Minute block deletion distance            | If the travel distance of a block is less than the parameter value, the block is deleted.<br>If the <Smooth path offset function> is <Valid>, it is performed at a command point after path approximation.                                                                                                                                                                                                                                                                                                                                                                                                        | 0~2.000 (mm)   |
| Accuracy level                            | Used with the fully automatic deceleration function.<br>When a value other than 0 is set, automatic deceleration is carried out so that the margin of error from the command path is less than the set value. In this situation, the following are disabled: <Corner deceleration override>, <Arc deceleration override> and <Curve approximation deceleration override>.<br>When set to 0, the fully automatic deceleration function is disabled and the following are enabled: <Corner deceleration override>, <Arc deceleration override> and <Curve approximation deceleration override>.                     | 0~5.000 (mm)   |

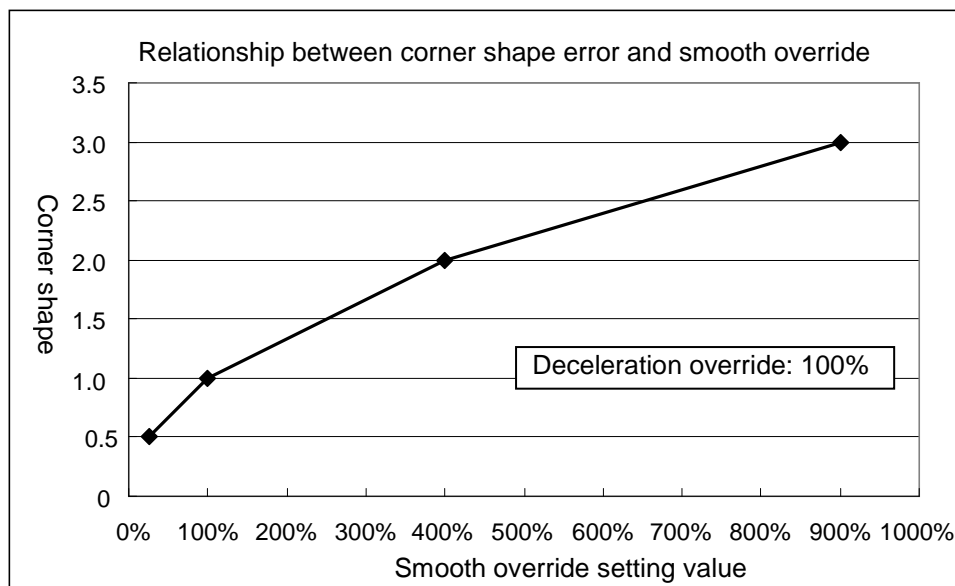
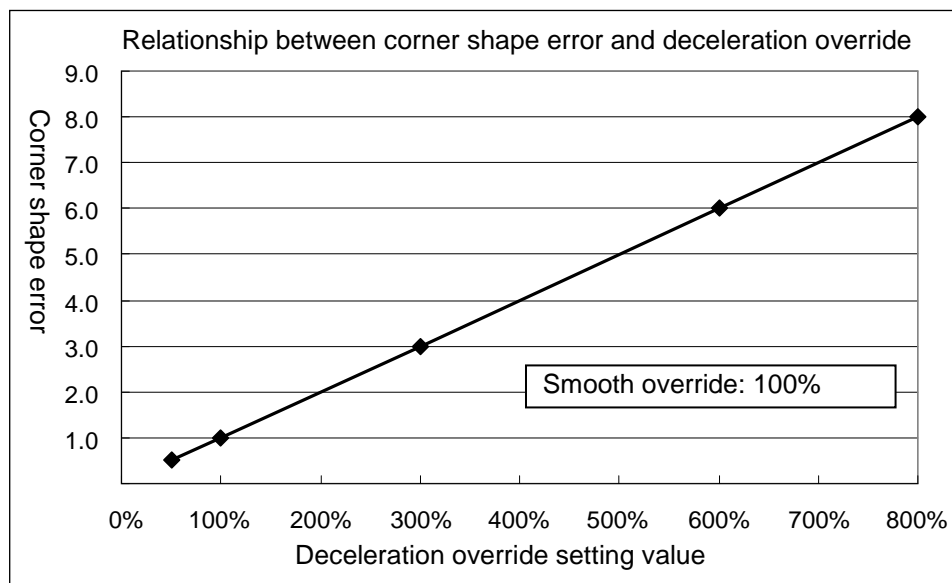
(High-accuracy common)

| Parameter name                                             | Contents                                                                                                                                                                                                                                                                                                                                                                                                                                                                           | Setting range    |
|------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------|
| Smooth path offset function                                | The <Valid/invalid> of the <Smooth path offset function> is set.                                                                                                                                                                                                                                                                                                                                                                                                                   | 0~1              |
| Smooth path offset cancel distance                         | This is used when the <Smooth path offset function> is set <Valid>.<br>If the travel distance of one block is longer than the set value, the smooth path offset is cancelled.<br>If the set value is too large, it may deviate significantly from the program path. Be careful when changing the value.                                                                                                                                                                            | 0~99.999<br>(mm) |
| Smooth path offset cancel angle                            | This is used when the <Smooth path offset function> is set <Valid>.<br>If the angle difference between blocks is larger than the set value, the smooth path offset is cancelled.<br>If the set value is too large, it may deviate significantly from the program path. Be careful when changing the value.                                                                                                                                                                         | 0~120.000<br>(°) |
| Smooth path offset angle determination invalid distance    | This is used when the <Smooth path offset function> is set <Valid>.<br>The smooth path offset is not cancelled even if the block where the smooth path offset is cancelled for decision of the <Smooth path offset cancel angle> is less than the value set for a travel distance of one block.                                                                                                                                                                                    | 0~9.999<br>(mm)  |
| Smooth path offset reference length                        | Used when <Smooth path offset function> is set to <Valid>.<br>If the travel distance of one block is longer than the set value, the path is offset using the set value.                                                                                                                                                                                                                                                                                                            | 0~9.999<br>(mm)  |
| Minute block deletion when smooth path offset is cancelled | Used when the <Smooth path offset function> is set to <Valid>.<br>When this parameter is set to <1: Enabled>, depending on the settings for <Smooth path offset cancel distance> and the <Smooth path offset cancel angle>, the <Minute block deletion distance> applies the deletion even if the smooth path offset is cancelled for the block.<br><b>(Note) Update the software version to W*10.*** or later in order to change the parameter setting to &lt;1: Enabled&gt;.</b> | 0~1              |
| Accuracy level method                                      | Set the deceleration method when the accuracy level is specified in high accuracy mode A.                                                                                                                                                                                                                                                                                                                                                                                          | 0~1              |

## 4.3 Relationship Between Parameters and Shape Errors

### 1. Corner shape error

The relationship between a shape error and an override when the shape error is 1.0 where the deceleration override is 100% and the smooth override is 100% is as follows:



#### Reference

Corner shape error =  $E * D * S$

E : Error when the corner deceleration override is 100% and the smooth override is 100%  
(Guide amount: 100-120  $\mu\text{m}$ )

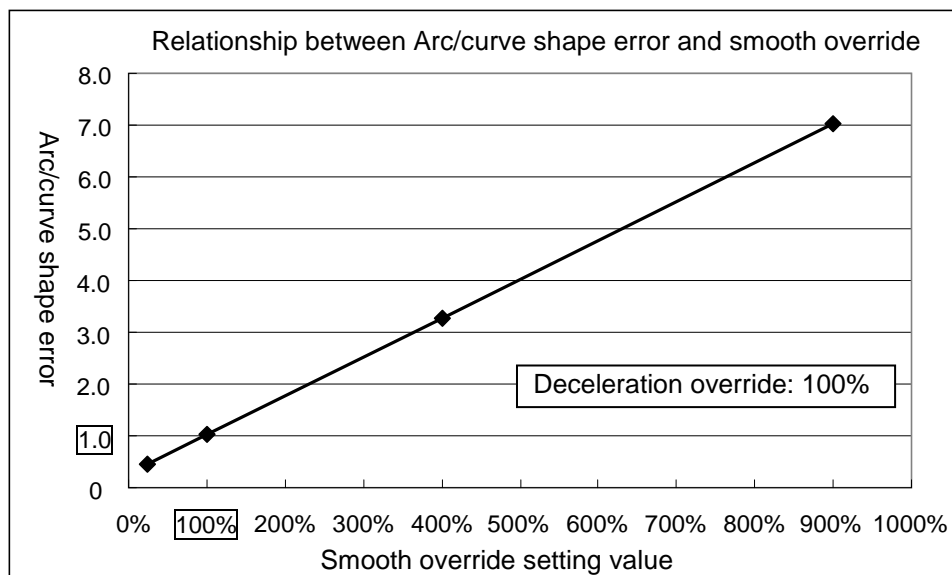
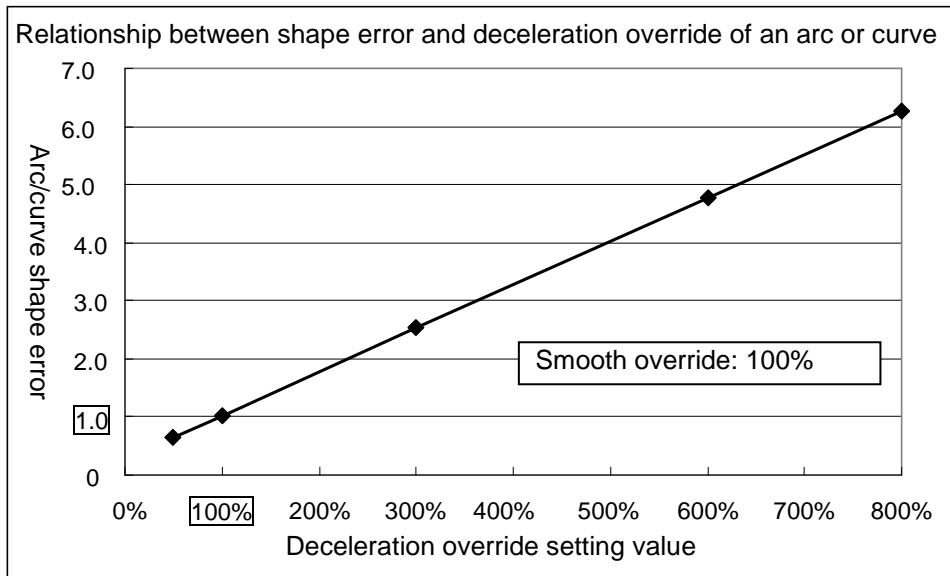
D : Corner deceleration override/100

S : Smooth override/100



## 2. Arc/curved shape error

The relationship between a shape error and an override when the shape error is 1.0 where the deceleration override is 100% and the smooth override is 100% is as follows:



## Reference

$$\text{Arc/curved shape error} = E * D * S$$

E : Error when the arc (approximate curve) deceleration override is 100% and the smooth override is 100% (Guide amount is 10-20  $\mu\text{m}$ )

D : Arc (approximate curve) deceleration override/100

S : Smooth override/100

## 4.4 Adjusting Parameters

The M code can be changed to set up the appropriate machining depending on the machining conditions. However, the parameters can also be adjusted to further optimize the machining. There are 2 types of parameters that can be adjusted: the parameter for adjusting shape accuracy and the smooth override. There are 4 parameters that can adjust the shape accuracy: Corner deceleration override, arc deceleration override, curve approximation deceleration override and accuracy level. A description of each parameter function is noted below.

|               | Parameter for adjusting shape accuracy                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                              | Smooth override                                                                                                                                                                                                                                                                                                                                  |
|---------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Description   | Parameter for improving shape accuracy.<br>It reduces the speed depending on the shape in order to improve the shape accuracy.<br>It impacts the shape accuracy and the machining time.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             | Override for improving the surface quality.<br>It reduces the unevenness in the command path in order to improve the surface quality.<br>It impacts the surface quality and the shape accuracy.                                                                                                                                                  |
| Setting range | Deceleration override: 0, 10 to 9999 (%)<br>Accuracy level: 0 to 5.000 (mm)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                         | 10 to 9999 (%)                                                                                                                                                                                                                                                                                                                                   |
| Description   | Deceleration override:<br>When the <Accuracy level> is set to another value besides 0, this parameter is disabled.<br>When it is set to 0, deceleration override is disabled.<br>When it is set to a small value, the shape accuracy improves but the machining time becomes longer.<br>When it is set to a large value, the machining time becomes shorter but the shape accuracy deteriorates.<br>Accuracy level:<br>When it is set to 0, the fully automatic deceleration override is disabled and the deceleration override is enabled.<br>When it is set to a small value, the shape accuracy improves but the machining time becomes longer.<br>When it is set to a large value, the machining time becomes shorter but the shape accuracy deteriorates.<br>How much the machining time increases or decreases varies depending on the model. | The surface quality is the same as when setting 100 as the normal mode. When it is set to a small value, the shape accuracy improves but the surface quality deteriorates. In addition, the maximum speed for the cutting feed is limited.<br>When it is set to a large value, the surface quality improves but the shape accuracy deteriorates. |

## **CHAPTER 13 (2)**

### **HIGH-ACCURACY MODE B**

- 1 Outline**
- 2 How to Use**
- 3 Restrictions**
- 4 Explanations on Parameters**

# 1 Outline

The high-accuracy mode B can perform better shape accuracy machining compared with the high-accuracy mode A by looking ahead a program and accelerating/decelerating the speed before interpolation.

The high-accuracy mode B has the following functions as same as the high-accuracy mode A:

- Automatic arc decelerating function
- Automatic curve approximation decelerating function
- Smooth override function
- Smooth path offset function

Note, the following specifications vary based on whether the machine is equipped with the option or not.

| Option                      | Not equipped: High accuracy mode BI | Equipped: High accuracy mode BII                                          |
|-----------------------------|-------------------------------------|---------------------------------------------------------------------------|
| No. of look ahead blocks    | 30 or 40 (*1)                       | Machine parameter (high accuracy)<br>No.8087 <No. of look-ahead blocks B> |
| Smooth path offset function | Not available                       | Available                                                                 |

\*1 The number of look ahead blocks varies depending on the machine model.

| Machine model                                                                                                               | No. of look ahead blocks |
|-----------------------------------------------------------------------------------------------------------------------------|--------------------------|
| 300X1/S500X1/S700X1/S1000X1/R450X1/R650X1/M140X1<br>S500Z1/S700Z1/R450Z1                                                    | 30                       |
| S300X1N/S500X1N/S700X1N/S1000X1N/R450X1N/R650X1N<br>M140X2/F600X1<br>S300X2/S500X2/S700X2<br>R450X2/R650X2<br>M200X3/M300X3 | 40                       |

In addition, when the machine parameter <No. of look-ahead blocks B> is less than 30 or 40, that set value is valid.

## 2 How to Use

### 2.1 To Select a Machining Level

High accuracy mode B is equipped with a function to improve shape accuracy (arc deceleration and curve approximation deceleration) and another to improve surface quality. The default setting is configured to the optimum machining level for shape accuracy and surface quality.

In addition to the level mentioned above, there are 7 separate user defined levels, and the user can adjust the settings to the most appropriate machining level. Refer to the “Detailed description of 4 functions and parameter adjustment” in “Chapter 13 (1) High Accuracy Mode A III” for further details. For details, refer to “Chapter13 (1) 4. Detailed explanations and adjustments of parameters”.

### 2.2 How to Use Programs

The following M codes are used for the high-accuracy mode B:

M280-M287 : To activate the high-accuracy mode B  
M289 : To deactivate the high-accuracy mode A/B

Example of Use

|                     |                                          |
|---------------------|------------------------------------------|
| Program example:    |                                          |
| NC program          |                                          |
| G00 X0 Y0 Z0;       |                                          |
| ;                   |                                          |
| M285;               | ← To activate the high-accuracy mode B   |
| G01 X20. Y30. Z50.; |                                          |
| X40. Y20. Z30.;     | } The high-accuracy mode B is in action. |
| ;                   |                                          |
| M289;               | ← To deactivate the high-accuracy mode B |
| M30;                |                                          |

### 2.3 Usable Conditions

To use the high-accuracy mode B, the modal status must be as follows:

**M97: To cancel interruptive macro program**  
**M299: Cancel machining mode specification**

In addition, this condition is applied when power is supplied.

**(Note)** If the high-accuracy mode B is used when the above modal status is not applied, the alarm <<High-accuracy B unable modal>> appears.

### 2.4 Conditions to be Cancelled

The high-accuracy mode B goes off if the following operations are done in the high-accuracy mode B:

- Power is supplied.
- The [RST] key is pressed.
- Memory operation is reset by pressing the [Z.RTN] key in the manual operation mode.
- End of program (M02, M30)

# 3 Restrictions

## 3.1 Commandable Functions

The functions available using commands while high accuracy mode B is executing are as follows.

1. All M code commands are possible except M96 (interrupt type macro).
2. All G code commands are possible.

**(Note 1)** If an uncommandable function is commanded, the alarm <<High-accuracy B unable command>> appears.

**(Note 2)** The high-accuracy mode B is temporarily cancelled when the following commands (modes) are issued:

When these commands (modes) finish, the high-accuracy mode B becomes valid again.

- M-code command
- S-code command
- T-code command
- Blocks that have no travel amount
- Blocks that have no cutting feed
- Every rotary feed
- Fixed cycle
- Thread cutting
- Thread cutting cycle
- Complex thread cutting cycle
- Exact stop mode
- Skip feed (G31, G131 and G132)
- Measurement related G code commands (G120 to G129)
- Inverse time feed

**(Note 3)** The smooth path offset function of the high-accuracy mode B is temporarily cancelled when the following commands (modes) are issued. When these commands (modes) finish, the high-accuracy mode B becomes valid again.

- M-code command
- S-code command
- T-code command
- Blocks that have no travel amount
- Blocks that have no cutting feed
- Blocks with an additional axis travel command
- Every rotary feed
- Fixed cycle
- Thread cutting
- Thread cutting cycle
- Complex thread cutting cycle
- Arc interpolation
- Involute interpolation
- Exact stop mode
- Diameter offset mode
- Scaling
- Mirror image
- Rotation of coordinates
- Feature coordinate manufacturing mode
- Inverse time feed

## 3.2 Temporary Stop of Operation

If an operation is tried to be stopped by the **[FEED HOLD]** switch in the high-accuracy mode B, it may not stop in the commanded block because it decelerates smoothly in the allowable acceleration.

## 3.3 Single Operation

As reasoned in “3.3 Temporarily stop of operation”, even if you try to perform a single operation in the high-accuracy mode B and press the **[SINGL]** key, it may not stop in the commanded block.

## 3.4 Cutting Override

The cutting override in the high-accuracy mode B is valid. However, if an override value exceeding 100% is commanded, the override is not performed in the connection part with blocks but becomes 100%.

## 3.5 Dry Run

It is unable to switch ON/OFF a dry run in the high-accuracy mode B. If the **[DRY]** key is pressed, the operator message “During high-accuracy B” appears.

## 3.6 Feedrates When Commanding the High-accuracy Mode B

If you command a speed faster than the machine parameter (high accuracy) <Maximum feedrate B>, the message << Feedrate error>> appears.

The high-accuracy mode B cannot use the machine parameter (system 1) <X- to Z-axes max. cutting feedrates > and the user parameter (switch 1) <Max. actual cutting travel speed>.

## 3.7 Notes in Smooth Path Offset Function

- The smooth path offset function works only when the user parameter (high accuracy) <Smooth path offset function> is “Valid”.
- The block stop position in single operation differs from the program command position.
- Mode change is unavailable.
- Dry run change is unavailable.
- No block during cutting by the smooth path offset function can be specified in a program restart block. If specified, the message <<Program restart error>> appears.
- If the <Smooth path offset cancel angle> set in the user parameters (high precision) is too large, a <<Curve speed error>> may occur.

## 3.8 Additional Axis Travel Command

When a travel command is issued on the additional axis in high accuracy mode B, make sure to check the setting beforehand for <High accuracy B additional axes> in the machine parameter (high accuracy).

When set to <1: Valid>, the additional axis travel operates in high accuracy mode B.

When set to <0: Invalid>, the additional axis travel does not operate in high accuracy mode B.

- If a travel command is issued only for the additional axis, then high accuracy mode B is temporarily canceled only during that command.
- The feed axis and the additional axis cannot move at the same time during the cutting feed.

When issuing a travel command on the additional axis in high accuracy mode B, contact the index manufacturer and set the appropriate value in the machine parameter before using it.

## 4 Explanations on Parameters

(High-accuracy mode B)

| Parameter name                            | Contents                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                 | Setting range  |
|-------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------|
| Corner deceleration override              | Used for automatic corner deceleration function.<br>Automatic corner deceleration is performed at 100 based on the machine's unique deceleration rate. When a value larger than 100 is set, the deceleration rate is smaller, making machining time shorter. When a value smaller than 100 is set, the deceleration rate is larger, resulting in more accurate machining. In addition, when 0 is set, automatic corner deceleration is not performed.<br>This parameter is disabled when the smooth override type is set to <1: Type 2>. | 0, 10-9999 (%) |
| Arc deceleration override                 | Used for automatic arc deceleration.<br>When 100 is set, automatic arc deceleration is performed based on the machine's unique deceleration rate. When a value larger than 100 is set, the deceleration rate is smaller, making machining time shorter. When a value smaller than 100 is set, the deceleration rate is larger, resulting in more accurate machining. In addition, when 0 is set, automatic arc deceleration is not performed.                                                                                            | 0, 10-9999 (%) |
| Curve approximation deceleration override | Used for automatic curve approximation deceleration.<br>When 100 is set, automatic curve approximation deceleration is performed based on the machine's unique deceleration rate. When a value larger than 100 is set, the deceleration rate is smaller, making machining time shorter. When a value smaller than 100 is set, the deceleration rate is larger, resulting in more accurate machining. In addition, when 0 is set, automatic curve approximation deceleration is not performed.                                            | 0, 10-9999 (%) |
| Smooth path offset level                  | Used when <Smooth path offset function> (described later) is set to <Valid>.<br>The larger the value is, the smoother the path becomes.                                                                                                                                                                                                                                                                                                                                                                                                  | 1-5            |
| Smooth override                           | Used machine parameter (high accuracy) <High accuracy B cutting feed time constant> at a value of 100.<br>If the value is larger than 100, the path becomes smoother.                                                                                                                                                                                                                                                                                                                                                                    | 100-9999 (%)   |
| Minute block deletion distance            | If the travel distance of one block is less than the parameter value, the block is deleted.<br>If <Smooth path offset function> is set to <Valid>, it is executed at a programmed point after path approximation.                                                                                                                                                                                                                                                                                                                        | 0-2.000 (mm)   |
| Smooth override type                      | Change the smooth override type.<br><0: Type 1><br>Even if the smooth override setting is changed, the speed difference that is permitted between blocks does not change.<br><1: Type 2><br>The speed difference that is permitted between blocks is equal to: machine parameter (high accuracy) "speed difference" × "smooth override value"/100.<br>When type 2 is selected, increasing the set value for the smooth override makes it more difficult to decelerate at corners. Therefore, the machining time can be reduced.          | 0-1            |



(High-accuracy common)

| Parameter name                                             | Contents                                                                                                                                                                                                                                                                                                                                                                                                                                                                           | Setting range    |
|------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------|
| Smooth path offset function                                | The <Valid/invalid> of the <Smooth path offset function> is set.                                                                                                                                                                                                                                                                                                                                                                                                                   | 0-1              |
| Smooth path offset cancel distance                         | This is used when the <Smooth path offset function> is set <Valid>.<br>If the travel distance of one block is longer than the set value, the smooth path offset is cancelled.<br>If the set value is too large, it may deviate significantly from the program path. Be careful when changing the value.                                                                                                                                                                            | 0-99.999<br>(mm) |
| Smooth path offset cancel angle                            | This is used when the <Smooth path offset function> is set <Valid>.<br>If the angle difference between blocks is larger than the set value, the smooth path offset is cancelled.<br>If the set value is too large, it may deviate significantly from the program path. Be careful when changing the value.                                                                                                                                                                         | 0-120.000<br>(°) |
| Smooth path offset angle determination invalid distance    | This is used when the <Smooth path offset function> is set <Valid>.<br>The smooth path offset is not cancelled even if the block where the smooth path offset is cancelled for decision of the <Smooth path offset cancel angle> is less than the value set for a travel distance of one block.                                                                                                                                                                                    | 0-9.999<br>(mm)  |
| Smooth path offset reference length                        | Use when <Smooth path offset function> is set to <Valid>.<br>If the travel distance of one block is longer than the set value, the path is offset using the set value.                                                                                                                                                                                                                                                                                                             | 0-9.999<br>(mm)  |
| Minute block deletion when smooth path offset is cancelled | Used when the <Smooth path offset function> is set to <Valid>.<br>When this parameter is set to <1: Enabled>, depending on the settings for <Smooth path offset cancel distance> and the <Smooth path offset cancel angle>, the <Minute block deletion distance> applies the deletion even if the smooth path offset is cancelled for the block.<br><b>(Note) Update the software version to W*10.*** or later in order to change the parameter setting to &lt;1: Enabled&gt;.</b> | 0~1              |
| Accuracy level method                                      | Set the deceleration method when the accuracy level is specified in high accuracy mode A.                                                                                                                                                                                                                                                                                                                                                                                          | 0~1              |

(This page was intentionally left blank.)

## **CHAPTER 13 (3)**

### **QUICK SETTING FOR HIGH ACCURACY MODE**

- 1 Overview**
- 2 High Accuracy (Adjustment) Function**
- 3 High Accuracy (Accuracy Specification) Function**
- 4 Mode Selection Using Program Command**
- 5 Check Current Mode**

# 1 Overview

The quick setting function for high accuracy mode streamlines the steps required for adjusting parameters in order to access and use high accuracy mode quickly.

There are two functions available: the “High accuracy (adjustment) function” for selecting the desired function from the graph on the screen and “High accuracy (accuracy specification) function” for directly specifying the accuracy level.

Use the desired mode to support the machining content.

This function is available on the following machine models.

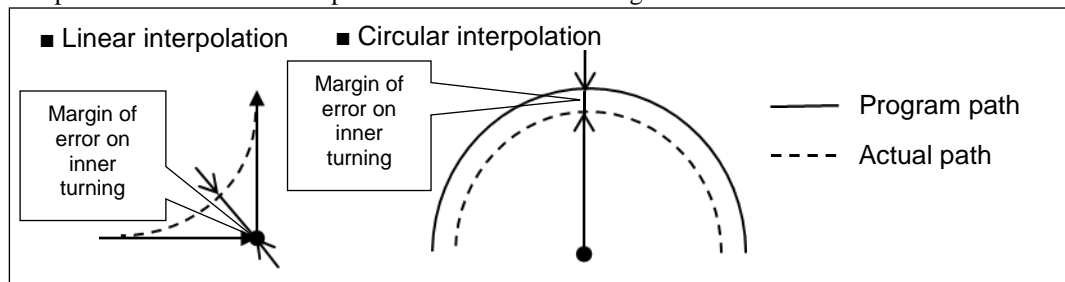
|       |                                                                                                                                                                                    |
|-------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Model | S300X2/S500X2/S700X2<br>R450X2/R650X2<br>M200X3/M300X3<br>The function can also be used on the following models if the software version is W*0*.08.010 or later.<br>S500X1/S500X1N |
|-------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|

## 2 High Accuracy (Adjustment) Function

The user can use the high accuracy (adjustment) function to browse from 6 available machining modes and select the one that best fits the machining content, for example, the user can select an option that focuses on machining accuracy or on the smoothness of the machining surface.

| Machining mode | Description                                                                                                  |
|----------------|--------------------------------------------------------------------------------------------------------------|
| Standard       | Standard setting that balances accuracy and smoothness                                                       |
| Rough          | Setting with shorter machining time                                                                          |
| Medium rough   | Setting between rough and finishing                                                                          |
| Medium rough S | Setting focused on machining surface quality                                                                 |
| Finishing      | Setting focused on true accuracy for program command                                                         |
| Finishing S    | Setting focused on machining surface quality and true accuracy of program commands, machining time is longer |

The shape accuracy refers to the margin of error on the inner turning that occurs during linear interpolation and circular interpolation as shown in the diagram below.



The user can add up to 3 machining modes that can be customized in addition to the 6 modes noted above.

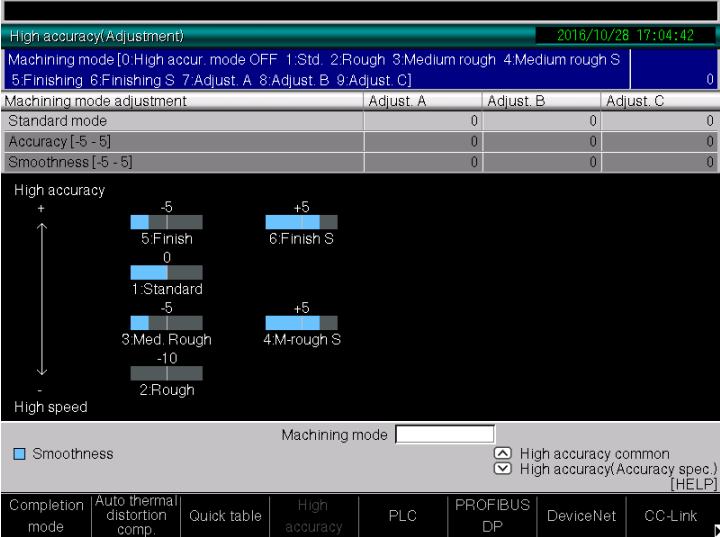
| Machining mode | Description                      |
|----------------|----------------------------------|
| Adjustment A   | Machining mode A set by customer |
| Adjustment B   | Machining mode B set by customer |
| Adjustment C   | Machining mode C set by customer |

## 2.1 Usage

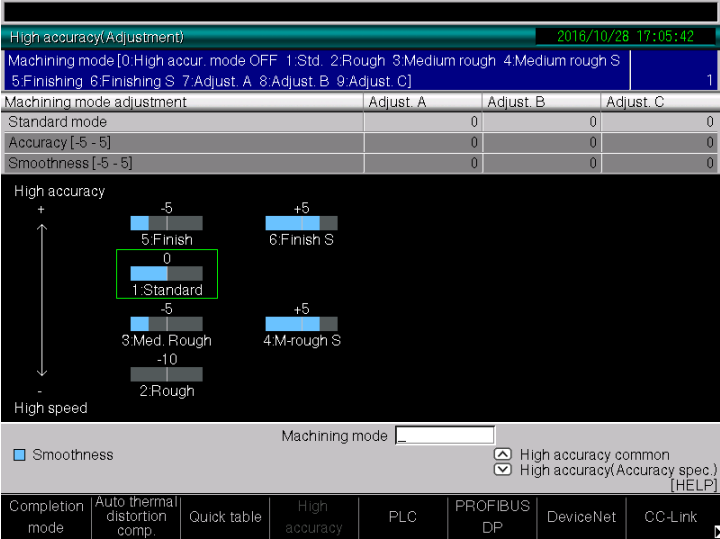
This section provides a description on using the high accuracy (adjustment) function. There are 6 basic machining modes for the high accuracy (adjustment) function, and there are an additional 2 modes (machining mode adjustment) that the user can customize based on the basic modes.

### 2.1.1 Machining mode

1. Open the <High accuracy (Adjustment)> screen in the user parameter (high accuracy).



2. Enter the desired machining mode number for <Machining mode>, and press the [ENT] key. The machining mode that is set is displayed in the green box on the graph on the bottom of the screen.  
e.g.) When <1. Std > is set for the <Machining mode>



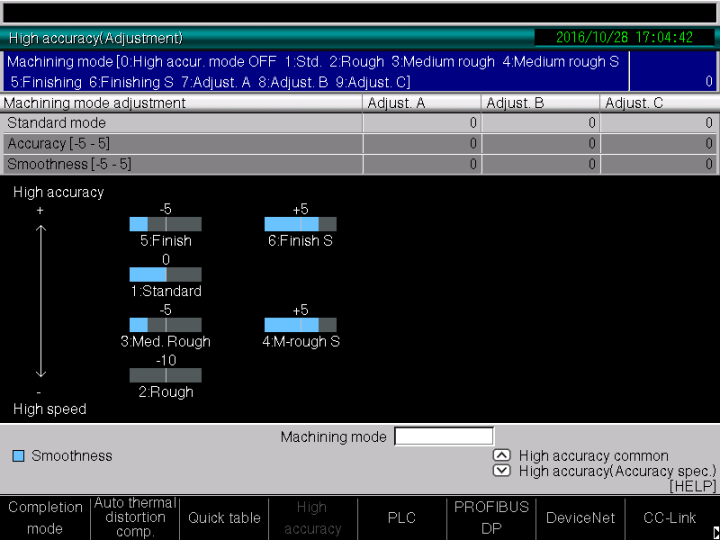
3. Press the [F0] (Completion mode) key to complete and exit the editing. After pressing the [RST] key, the setting appears.

2.1.2 Machining mode adjustment

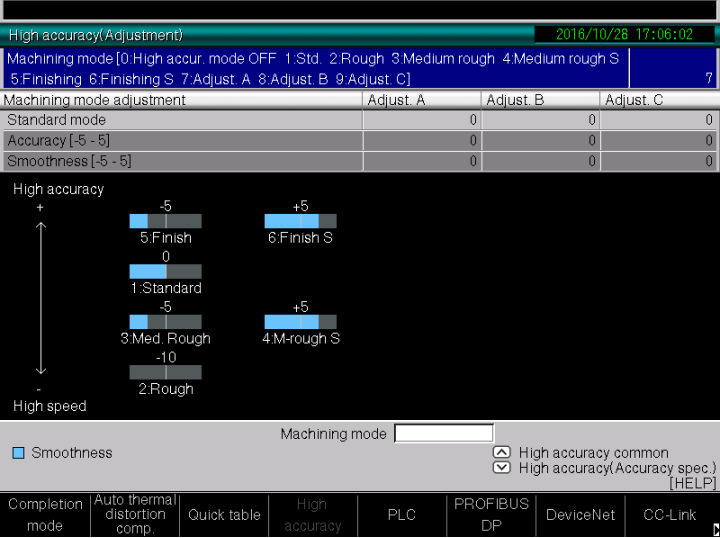
After setting <7. Adjust. A>, <8. Adjust. B> and <9. Adjust. C> for the machining mode, the user can add the customized machining modes, which are based on the 6 basic modes already available, i.e. <1. Std.> and <2: Rough>.

The following procedure is a description when adding <7. Adjust. A>.  
The procedures for adding <8. Adjust. B> and <9. Adjust. C> are the same.

- 1. Use [F0] to open the <High accuracy (Adjustment)> screen in the user parameter (high accuracy).

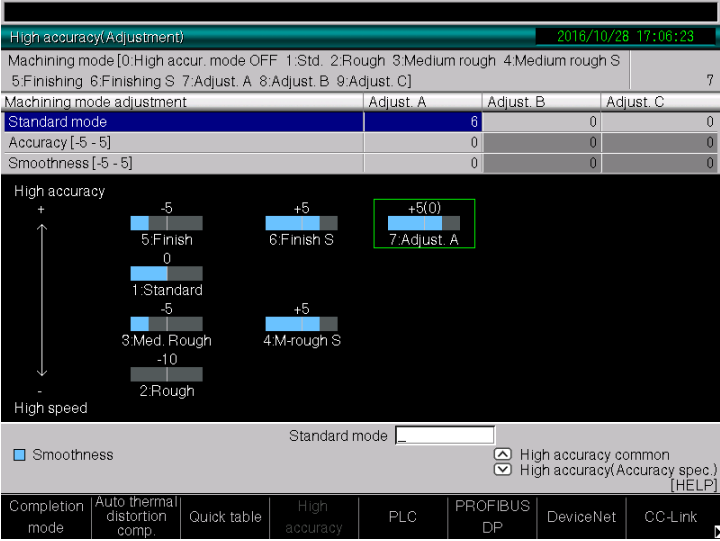


- 2. Enter <7. Adjust. A> for the <Machining mode>, and press the [ENT] key.



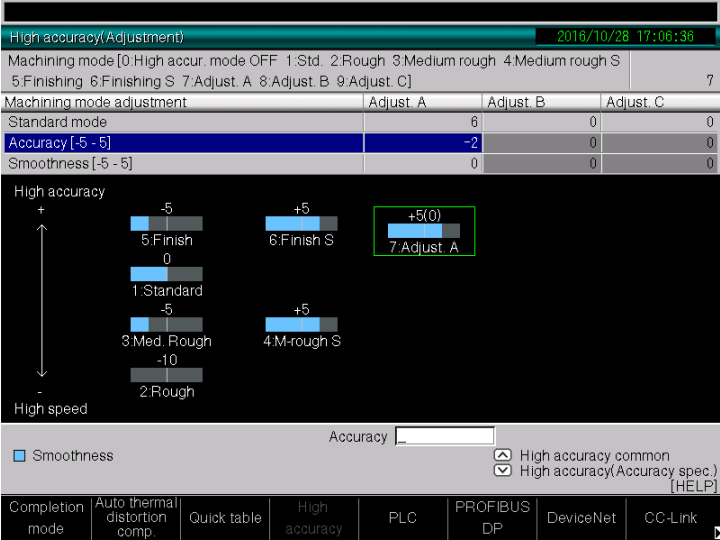
3. Enter the mode number to set the <Standard mode>, and press the [ENT] key. <7. Adjust. A> is displayed inside a box on the graph on the bottom of the screen. The fields <Accuracy> and <Smoothness> will no longer be grayed out.

e.g.) When <6. Finishing S> is set for the <Standard mode>



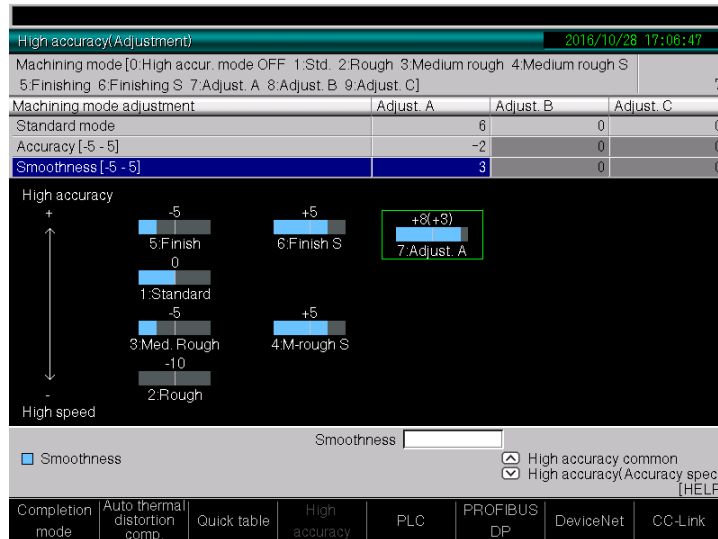
4. Set the accuracy level for the standard mode that is set in <Accuracy>. The accuracy refers to the vertical axis on the graph on the bottom of the screen. The higher the value is, the more accurate (increase) the machining becomes. The smaller the value is, the faster (decrease) the machining becomes. The accuracy can be adjusted to 5 levels in the positive and negative directions for the standard mode.
- If the <Accuracy> is changed, the display on the axis changes only for the increment set for the standard mode (<Adjust. A>) on the bottom of the screen.

e.g.) When <6. Finishing S> is set for the <Standard mode>, and <-2> is set for <Accuracy>



5. Set the smoothness level on the machining surface for the standard mode that is set in <Smoothness>. The smoothness level is displayed in the light blue gauge. The higher the value is, the smoother the machining surface becomes.
- The smoothness level can be adjusted to 5 levels in the positive and negative directions for the standard mode (When <2. Rough> is set for the standard mode, the <Smoothness> becomes +5.) The adjusted value is displayed in the parentheses above the gauge.

e.g.) When <6. Finishing S> is set for the <Standard mode>, and <-2> is set for <Accuracy> and <3> is set for smoothness



6. Press the [F0] (Completion mode) key to complete and exit the editing. After pressing the [RST] key, the setting appears.

**(Note)** The position and the gauge on the graph from <7. Adjust. A> to <9. Adjust. C> show the relationship with the standard mode. When comparing with a mode that is not the standard mode, the position on the graph and the relationship with the gauge may be different from the actual machining results.



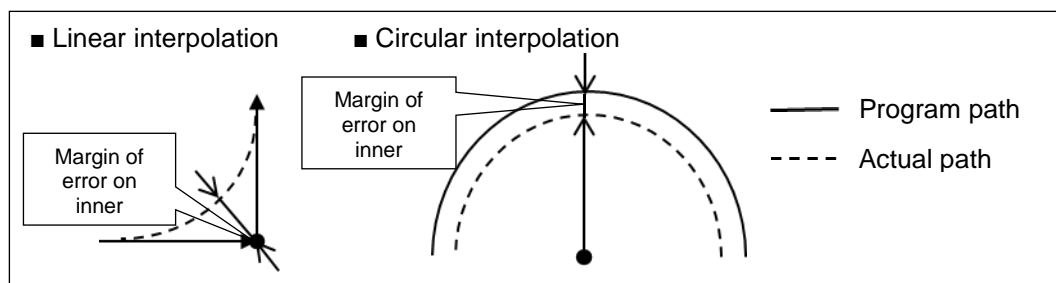
## 2.2 Restrictions

- Depending on the machining mode, the alarm <<The high-accuracy B is being executed>> may trigger and switching the dry run may be unavailable during cutting travel.
- Depending on the machining mode, if cutting travel is carried out in single operation mode, the operation message <<Command interrupted in high-accuracy mode B>> may appear.
- Depending on the machining mode, the feed axis and the additional axis cannot be moved at the same time.
- M96 (interrupt type macro) cannot be used.
- When using a feedrate faster than 20,000 mm/min and the machining mode is changed, the alarm <<Feedrate error>> may trigger.
- When equipped with the option: high accuracy B/smooth path offset, the cycle time may become shorter and the machining surface may change slightly.
- The following restrictions apply when the smooth path offset function is enabled. Refer to “Chapter 13 (1) High accuracy mode A” and “Chapter 13 (2) High accuracy mode B” for further details on the smooth path offset function.
  1. Block stop position in single operation mode is different from the program command position.  
The mode cannot be changed.
  2. The dry run cannot be changed.
  3. A block that is cutting with the smooth path offset function cannot be specified as a block for resuming or restarting the program. If it is specified, the alarm <<Program restart error>> is triggered.
  4. If the user parameter (high accuracy) <Smooth path offset cancel angle> is set too large, the alarm <<Curve speed error>> may be triggered.
  5. The user parameter (high accuracy) <Minute block deletion distance> does not apply to blocks where the smooth path offset function has been cancelled.

**(Note)** When a function command is accidentally issued even though it is not available in the selected machining mode, the alarm <<Invalid command in machining mode>> is triggered.

### 3 High Accuracy (Accuracy Specification) Function

When using the user parameter (high accuracy) <Accuracy spec. mode>, the machining automatically decelerates so that if a margin of error on inner turning, between the actual path and the programmed path, occurs when performing linear or circular interpolation, the difference becomes less than the value specified in the <Accuracy level>, as shown in the diagram below.



There are 3 accuracy specification modes, and the default factory settings are as follows. The set value is a guide and cannot always be guaranteed to fall within the tolerance.

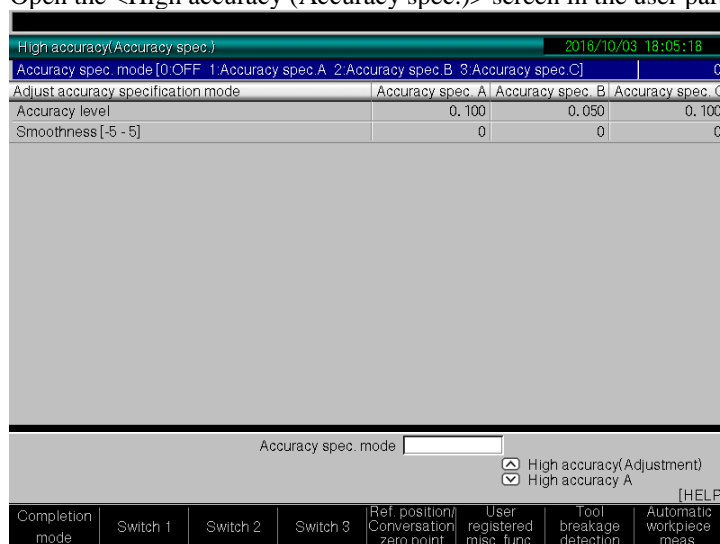
| Accuracy specification mode | Description  |
|-----------------------------|--------------|
| Accuracy spec. A            | Rough        |
| Accuracy spec. B            | Medium rough |
| Accuracy spec. C            | Finishing    |

#### 3.1 Usage

The following description is a setting procedure for <1. Accuracy spec. A>.

The setting procedures for <2. Accuracy spec. B> and <3. Accuracy spec. C> are the same.

1. Open the <High accuracy (Accuracy spec.)> screen in the user parameter (high accuracy).



- Enter <1. Accuracy spec. A> for the <Accuracy spec. mode>, and press the [ENT] key. The <Accuracy spec. A> is the mode for rough machining in the default factory settings. To adjust the accuracy and machining time, change the accuracy level and the smoothness in steps (3) and (4).

| High accuracy (Accuracy spec.)                                                       |       | 2016/10/03 18:05:48 |                  |
|--------------------------------------------------------------------------------------|-------|---------------------|------------------|
| Accuracy spec. mode [0:OFF 1:Accuracy spec. A 2:Accuracy spec. B 3:Accuracy spec. C] |       | 1                   |                  |
| Adjust accuracy specification mode                                                   |       | Accuracy spec. A    | Accuracy spec. B |
| Accuracy level                                                                       | 0.100 | 0.050               | 0.100            |
| Smoothness [-5 - 5]                                                                  | 0     | 0                   | 0                |

Accuracy spec. mode

☐ High accuracy (Adjustment)  
☒ High accuracy A

[HELP]

|                 |          |          |          |                                       |                             |                         |                           |
|-----------------|----------|----------|----------|---------------------------------------|-----------------------------|-------------------------|---------------------------|
| Completion mode | Switch 1 | Switch 2 | Switch 3 | Ref. position/Conversation zero point | User registered misc. func. | Tool breakage detection | Automatic workpiece meas. |
|-----------------|----------|----------|----------|---------------------------------------|-----------------------------|-------------------------|---------------------------|

- Enter the target accuracy in units (setting units depends on the user parameter (switch 1) <Machine unit system>) of mm for the <Accuracy level>, and press the [ENT] key.
- Set the smoothness level in <Smoothness>. The higher the value is, the smoother the machining surface becomes.
- Press the [F0] (Completion mode) key to complete and exit the editing. After pressing the [RST] key, the setting appears.

## 3.2 Restrictions

- The feed axis and the additional axis cannot be moved at the same time while in accuracy specification mode.
- When equipped with the option: high accuracy B/smooth path offset, the cycle time may become shorter and the machining surface may change slightly.
- M96 (interrupt type macro) cannot be used.
- The following restrictions apply when the smooth path offset function is enabled. Refer to “Chapter 13 (1) High accuracy mode A” and “Chapter 13 (2) High accuracy mode B” for further details on the smooth path offset function.
  - Block stop position in single operation mode is different from the program command position.
  - The mode cannot be changed.
  - The dry run cannot be changed.
  - A block that is cutting with the smooth path offset function cannot be specified as a block for resuming or restarting the program. If it is specified, the alarm <<Program restart error>> is triggered.
  - If the user parameter (high accuracy) <Smooth path offset cancel angle> is set too large, the alarm <<Curve speed error>> may be triggered.
  - The user parameter (high accuracy) <Minute block deletion distance> does not apply to blocks where the smooth path offset function has been cancelled.

**(Note)** When a function command is accidentally issued even though it is not available during accuracy specification mode, the alarm <<Invalid command in accuracy specification mode>> is triggered.

## 4 Mode Selection Using Program Command

Issue an M298 command in memory operation to set the machining mode or the accuracy specification mode.

Command format **M298 L**

Issue a machining mode or an accuracy specification mode command by using the numerical values that come after the “L” (Refer to the table below).

L0 (OFF) is the setting to not use the high accuracy mode. The same is true when the user parameter (high accuracy) <Machining mode> and <Accuracy spec. mode> are both <0>.

| L0  | OFF              |
|-----|------------------|
| L1  | Standard         |
| L2  | Rough            |
| L3  | Medium rough     |
| L4  | Medium rough S   |
| L5  | Finishing        |
| L6  | Finishing S      |
| L7  | Adjustment A     |
| L8  | Adjustment B     |
| L9  | Adjustment C     |
| L21 | Accuracy spec. A |
| L22 | Accuracy spec. B |
| L23 | Accuracy spec. C |

**(Note)** If a command is issued when high accuracy mode A/B is ON, the alarm <<Unavailable modal for machining mode specification>> is triggered and operation stops.

Issuing an M299 command can cancel the machining mode that was initiated by the command M298.

Command format **M299**

The M298 command is canceled regardless of the following operations.

- If the power is turned ON
- If the [RST] key is pressed
- If an operation resets the memory operation such as pressing the [Z.RTN] key in manual operation mode
- If end of program (M02, M30) is executed

The following is an example of a program when <1. Std.> is set for <Machining mode> in the user parameter (high accuracy).

(Program example)

G00 X0 Y0 Z0;

;

G01 X20. Y30. Z50.;

X40. Y20. Z30.;

;

M260;

G01 X20. Y30. Z50.;

X40. Y20. Z30.;

M269;

;

M298 L5;

G01 X20. Y30. Z50.;

X40. Y20. Z30.;

M299;

;

G01 X20. Y30. Z50.;

X40. Y20. Z30.;

;

M30;

← <Std. >machining mode

← High accuracy mode A (Level 1) ON

← <Finishing> machining mode ON

← <Finishing> machining mode OFF

← <Std.> machining mode

## 5 Check Current Mode

The current status of the high accuracy mode can be checked on the <Modal info 2> screen.

The screenshot shows the <Modal info 2> screen with the following sections:

- Modal info 2**: 2016/10/03 18:24:20
- Program(1000)**
- Positioning end check distance(M270)**: X 0.300, Y 0.300, Z 0.300, A 0.300, A 0.300, A 0.300, B 0.300, C 0.300
- Rotational transformation**: Center X, Center Y, Center Z, Angle
- Mirror image**: Reference X, Reference Y, Reference Z
- Tool offset amount**: X, Y, Z
- High accuracy**: Standard
- Scaling**: Center, Scaling
- Tool wear offset amount**: X, Y, Z
- Spindle speed clamp**: S, Q
- Fixd outer ax ctrl**
- Macro variables**
- Modal 1**: ☒ Modal 1, ☒ Soft limit
- [HELP]**
- Current position**: Modal 1, Modal 2, Soft limit, Enlargement, Workpiece coordinate zero, Select add. axes/change display

| Status           | Description                                                                  |
|------------------|------------------------------------------------------------------------------|
| OFF              | High accuracy mode is not be used.                                           |
| Standard         | The machine is operating in <Standard> machine mode.                         |
| Rough            | The machine is operating in <Rough> machine mode.                            |
| Medium rough     | The machine is operating in <Medium rough> machine mode.                     |
| Medium rough S   | The machine is operating in <Medium rough S> machine mode.                   |
| Finishing        | The machine is operating in <Finishing> machine mode.                        |
| Finishing S      | The machine is operating in <Finishing S> machine mode.                      |
| Adjustment A     | The machine is operating in <Adjust. A> machine mode.                        |
| Adjustment B     | The machine is operating in <Adjust. B> machine mode.                        |
| Adjustment C     | The machine is operating in <Adjust. C> machine mode.                        |
| Accuracy spec. A | The machine is operating in accuracy specification mode: <Accuracy spec. A>. |
| Accuracy spec. B | The machine is operating in accuracy specification mode: <Accuracy spec. B>. |
| Accuracy spec. C | The machine is operating in accuracy specification mode: <Accuracy spec. C>. |
| M260 to M267     | The machine is operating in high accuracy mode A.                            |
| M280 to M287     | The machine is operating in high accuracy mode B.                            |

---

(This page was intentionally left blank.)

---

©2012-2020 BROTHER INDUSTRIES, LTD.  
All Rights Reserved.  
Printed in Japan.  
This is the original instructions.

693716001  
2002(16)