



**W1000Xd1 Series / S300Xd1 Series / S500Xd1 Series
S700Xd1 Series / R450Xd1 Series / R650Xd1 Series
U500Xd1 Series / M200Xd1 Series**

COMPACT MACHINING CENTER SPEEDIO

**CNC-D00
PROGRAMMING MANUAL
(NC)**

(For Installers & Machine Setup Specialists)

Please read this manual carefully before operating the machine.
Be careful not to lose the manuals, and keep them handy at all times.

PROGRAMMING MANUAL

(NC)

Read this manual before performing work.

The logo consists of the word "brother" in a lowercase, bold, sans-serif font. The letters are black and have a slightly rounded appearance.

————— Introduction ————

Thank you for purchasing the SPEEDIO (hereafter referred to as “machine”) made by Brother.

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

This manual gives a description of NC programming.

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

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.

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.

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.

Any trade names and product names of companies appearing on Brother products, related documents and any other materials are all trademarks or registered trademarks of those respective companies.

———— 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

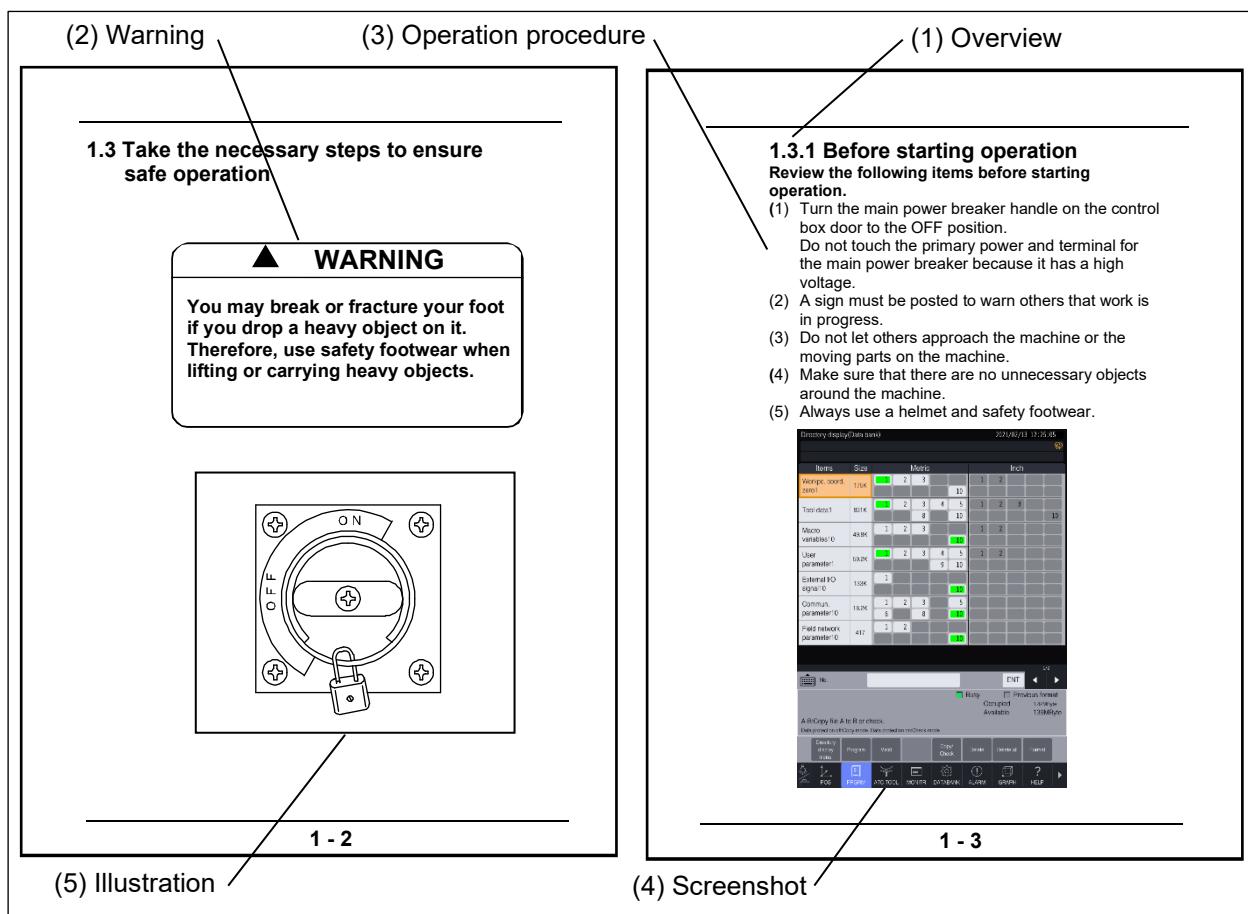
(2-3) Potential damage

(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.



In this manual, the following symbols are used to differentiate between keys, switches, text displayed on screens and alarm messages.

[] : Keys

[] : Switches

< > : Text displayed on screens

<< >> : Alarm messages

(This page was intentionally left blank.)

CHAPTER 1 PROGRAM STRUCTURE -----	1-1
1.1 Program Types and Structure -----	1-2
1.2 Block Structure -----	1-2
1.3 Word Structure-----	1-2
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
CHAPTER 2 COORDINATE COMMANDS -----	2-1
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
2.4 Table Coordinate System -----	2-3
CHAPTER 3 PREPARATION FUNCTION -----	3-1
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 Speed command for standard circle on rotation axis -----	3-9
3.3.1.2 Chamfering to Desired Angle and Cornering R -----	3-11
3.3.2 Circular Interpolation / Helical Thread Cutting Interpolation-----	3-14
3.3.2.1 Circular Interpolation (G02, G03) -----	3-14
3.3.2.2 XZ Circular Interpolation (G102, G103)-----	3-15
3.3.2.3 YZ Circular Interpolation (G202, G203)-----	3-15
3.3.2.4 Precautions for Circular Interpolation -----	3-16
3.3.2.5 Helical Thread Cutting Interpolation -----	3-19
3.3.2.6 Spiral Interpolation-----	3-20
3.3.2.7 Conical interpolation-----	3-22
3.3.2.8 Cutter Compensation for Spiral and Conical Interpolation -----	3-25
3.3.2.9 Involute interpolation-----	3-26
3.3.3 Circle Cutting (G12, G13)-----	3-27
3.4 Dwell (G04) -----	3-29
3.5 Exact Stop Check (G09, G61, G64) -----	3-30
3.6 Programmable Data Input (G10)-----	3-31
3.6.1 Entering Workpiece Coordinate Zero -----	3-31
3.6.2 Entering Tool Length / Cutter Compensation Data-----	3-31
3.6.3 Tool Wear Compensation-----	3-32
3.6.4 Tool Offset Data Input -----	3-32
3.6.5 Tool Offset Wear Data Input -----	3-33
3.6.6 Data Input of Extended Workpiece Coordinate -----	3-33
3.6.7 Workpiece Coordinate System Using the Results of Measurements -----	3-34
3.6.8 Data Input of Extended Workpiece Coordinate Zero Based on Measurement Results -----	3-34
3.6.9 Data Input for Reference Rotary Fixture Offset -----	3-35
3.6.10 Tool Information Input -----	3-35
3.7 Programmable Parameter Input (G10) -----	3-36
3.7.1 Usage -----	3-36
3.7.2 Usage Conditions -----	3-37
3.7.3 Parameter Setting -----	3-37
3.7.3.1 User parameters -----	3-37
3.7.4 Usage Examples-----	3-37
3.8 Coordinate System (G17 to 19, G52 to 59, G54.1, G92, G68.2) -----	3-38
3.8.1 Plane Selection (G17, G18, G19)-----	3-38

Contents

3.8.2 Machine Coordinate System Selection (G53) -----	3-38
3.8.3 Workpiece Coordinate System Selection (G54 to G59) -----	3-39
3.8.4 Extended Workpiece Coordinate System Selection (G54.1) -----	3-39
3.8.5 Workpiece Coordinate System Setting (G92) -----	3-40
3.8.6 Local Coordinate System Function (G52) -----	3-42
3.8.7 Rotary Fixture Offset Function (G54.2)-----	3-43
3.8.7.1 Overview -----	3-43
3.8.7.2 Command Format -----	3-44
3.8.8 Feature Coordinate System (G68.2) -----	3-45
3.8.8.1 Overview -----	3-45
3.8.8.2 Command format-----	3-47
3.8.8.3 Restrictions-----	3-51
3.8.9 Involute Interpolation Function -----	3-52
3.8.9.1 Overview -----	3-52
3.8.9.2 Command Format -----	3-52
3.8.9.3 Margin of Error for End Point on Involute Curve-----	3-55
3.8.9.4 Involute Interpolation Command During Cutter Compensation -----	3-56
3.8.9.5 Involute Interpolation Command in High Accuracy Mode -----	3-57
3.8.9.6 Restrictions-----	3-57
3.9 Soft Limit-----	3-58
3.9.1 Stroke-----	3-58
3.9.2 Stroke Limit -----	3-58
3.9.3 Programmable Stroke Limit (G22) -----	3-59
3.10 Reference Position (G28 to G30) -----	3-60
3.10.1 Return to the Reference Point (G28)-----	3-60
3.10.2 Return from the Reference Point (G29) -----	3-61
3.10.3 Return to the 2nd to 6th Reference Point (G30)-----	3-61
3.11 Skip Function (G31, G131/G132)-----	3-62
3.11.1 Before using the skip function-----	3-62
3.11.2 Skip Function (G31, G131/G132)-----	3-62
3.11.3 Multiple Skip Function (G31, G131/132)-----	3-63
3.11.4 Continuous Skip Function (G31)-----	3-64
3.12 Scaling (G50/G51) -----	3-65
3.13 Programmable Mirror Image (G50.1/51.1) -----	3-69
3.14 Rotational Transformation Function (G68/69, 168)-----	3-71
3.14.1 Coordinate Rotation (G68/69) -----	3-71
3.14.2 Coordinate Rotation Using Measured Results (G168) -----	3-72
3.15 Absolute Command and Incremental Command (G90/91) -----	3-73
3.16 Change of Tap Twisting Direction (G133/134)-----	3-75
3.17 G code Priority -----	3-76
3.18 Programmable Data Input (High Accuracy) (G210)-----	3-83
3.18.1 High Accuracy Mode A Parameter Changes -----	3-83
3.18.2 High Accuracy Mode B Parameter Changes-----	3-84
3.18.3 Temporary Parameter Change with TCP Control-----	3-85
3.19 Thread Cutting-----	3-86
3.19.1 Single Start Lead Thread Cutting (G33) -----	3-86
3.19.2 Thread Cutting Cycle (G392)-----	3-88
3.19.2.1 Thread Cutting Cycle for Straight Screws-----	3-88
3.19.2.2 Thread Cutting Cycle for Taper Screws-----	3-90
3.19.3 Thread Cutting in Complex Thread Cutting Cycle (G376) -----	3-92
3.19.3.1 Infeed -----	3-93
3.19.3.2 Detailed Description of Operation-----	3-94
3.19.4 Thread Runout -----	3-97
3.19.5 Program Example -----	3-98
3.19.5.1 Machining and Thread Cutting Straight Screws-----	3-98
3.19.5.2 Machining and Thread Cutting Taper Screws -----	3-99
3.20 Lathe Machining Infeed Direction -----	3-100

CHAPTER 4 PREPARATION FUNCTION (COMPENSATION FUNCTION) -----	4-1
4.1 Cutter Compensation (G40, G41 and G42)-----	4-2
4.1.1 Cutter Compensation Function-----	4-2
4.1.1.1 Cutter Wear Offset-----	4-3
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 -----	4-13
4.1.5.1 Inner Side Cutting ($180^\circ \leq \theta$)-----	4-13
4.1.5.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)-----	4-14
4.1.5.3 Outer Side (Acute Angle Cutting) ($\theta < 90^\circ$)-----	4-15
4.1.6 G40 Individual Command -----	4-16
4.1.7 Compensation Direction Change in Offset Mode-----	4-17
4.1.8 Offset Direction Change in Offset Mode -----	4-18
4.1.8.1 When There is an Intersection-----	4-18
4.1.8.2 When There is No Intersection -----	4-19
4.1.8.3 When an Arc Laps Around in a Circle -----	4-20
4.1.8.4 Arc angle check when offset direction changes-----	4-21
4.1.9 G Code Command for Cutter Compensation in Offset Mode-----	4-22
4.1.10 Special Notes for Cutter Compensation -----	4-23
4.1.11 Override Function Related to Cutter Compensation -----	4-29
4.1.11.1 Automatic Corner Override -----	4-29
4.1.11.2 Inner Arc Override -----	4-30
4.1.12 Arc Angle Check During Inner Side Cutting -----	4-31
4.1.13 Interference Check for Cutter Compensation / Nose R Compensation-----	4-32
4.1.13.1 Interference detection method -----	4-32
4.1.13.2 Operation when interference is detected-----	4-34
4.1.13.3 Restrictions and special notes-----	4-34
4.2 Tool Length Offset (G43, G44 and G49) -----	4-35
4.2.1 Tool Length Offset Function-----	4-35
4.2.2 Tool Length Wear Offset -----	4-36
4.2.3 Z-axis Travel with Tool Length Offset Command-----	4-36
4.3 Nose R Compensation (G141 and G142 - Option) -----	4-38
4.3.1 Command Format-----	4-39
4.3.1.1 Nose R Wear Offset-----	4-40
4.3.2 Virtual Teeth-----	4-41
4.3.3 Cancel Mode-----	4-41
4.3.4 Startup-----	4-42
4.3.4.1 Inner Side Cutting ($180^\circ \leq \theta$)-----	4-44
4.3.4.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)-----	4-45
4.3.4.3 Outer Side (Acute Angle Cutting) ($\theta < 90^\circ$)-----	4-46
4.3.5 Offset Mode -----	4-47
4.3.5.1 Inner Side Cutting ($180^\circ \leq \theta$)-----	4-47
4.3.5.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)-----	4-49
4.3.5.3 Outer Side (Acute Angle Cutting) ($\theta < 90^\circ$)-----	4-50
4.3.5.4 Exceptional Cases-----	4-51
4.3.6 Offset Cancel -----	4-52
4.3.6.1 Inner Side Cutting ($180^\circ \leq \theta$)-----	4-52
4.3.6.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)-----	4-53
4.3.6.3 Outer Side (Acute Angle Cutting) ($\theta < 90^\circ$)-----	4-54
4.3.7 G40 Individual Command -----	4-55
4.3.8 Compensation Direction Change in Offset Mode-----	4-56
4.3.9 Offset Direction Change in Offset Mode -----	4-56

Contents

4.3.9.1 When There is an Intersection -----	4-56
4.3.9.2 When There is No Intersection -----	4-57
4.3.10 G Code Command for Nose R Compensation in Offset Mode -----	4-59
4.3.11 Special Notes for Nose R Compensation-----	4-60
4.3.12 Override Function Related to Nose R Compensation-----	4-65
4.3.12.1 Automatic Corner Override -----	4-65
4.3.12.2 Inner Arc Override -----	4-66
4.3.13 Arc Angle Check During Inner Side Cutting -----	4-67
4.4 Tool Position Compensation (G143, G144 and G49 - Option) -----	4-68
4.4.1 Tool Position Compensation Function -----	4-68
4.4.2 Axis Travel with Tool Position Compensation-----	4-69

CHAPTER 5 PREPARATION FUNCTION (CANNED CYCLE)--- 5-1

5.1 Outline -----	5-2
5.2 List of Canned Cycle function -----	5-2
5.3 Basic Operation of Canned Cycle -----	5-3
5.4 General Rules of Canned Cycle -----	5-4
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
5.5 Details of Canned Cycle -----	5-8
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) -----	5-13
5.5.6 Drilling Cycle (G81, G82)-----	5-15
5.5.7 Peck Drilling Cycle (G83)-----	5-16
5.5.8 Tapping Cycle (G84)-----	5-17
5.5.9 Boring Cycle (G85, G89)-----	5-18
5.5.10 Boring Cycle (G86)-----	5-19
5.5.11 Back Boring Cycle (G87) -----	5-20
5.5.12 End Milling/Tapping Cycle (G177)-----	5-21
5.5.13 End Milling/Tapping Cycle (G178)-----	5-22
5.5.14 Double Drilling Cycle (G181, G182)-----	5-23
5.5.15 Double Boring Cycle (G185, G189)-----	5-24
5.5.16 Double Boring Cycle (G186)-----	5-25
5.5.17 Deep Hole Tapping Cycle (Synchro Mode) (G277) -----	5-26
5.5.18 Reverse Deep Hole Tapping Cycle (Synchro Mode) (G278)-----	5-27
5.5.19 Reducing Step of Canned Cycle-----	5-29
5.5.19.1 High speed peck drilling cycle (G73) (reducing step) -----	5-29
5.5.19.2 Peck drilling cycle (G83) (reducing step) -----	5-30
5.5.19.3 Tapping cycle (synchro mode) (G77) (reducing step) -----	5-31
5.5.19.4 Reverse tapping cycle (synchro mode) (G78) (reducing step)-----	5-32
5.5.19.5 Deep hole tapping cycle (synchro mode) (G277) (reducing step)-----	5-34
5.5.19.6 Reverse deep hole tapping cycle (synchro mode) (G278) (reducing step) -----	5-36
5.5.19.7 2nd and after cutting depth in G73, G83, G173, and G183 -----	5-37
5.5.19.8 2nd and after cutting depth in G77, G78, G273, and G283 -----	5-37
5.5.20 Canned Cycle Cancel (G80) -----	5-38
5.5.21 General Precautions for Canned Cycle -----	5-38
5.6 One-shot Canned Cycle-----	5-39
5.6.1 High Speed Peck Drilling Cycle (G173)-----	5-39
5.6.1.1 High Speed Peck Drilling Cycle (G173) (Reducing Step) -----	5-40
5.6.2 Peck Drilling Cycle (G183)-----	5-41
5.6.2.1 Peck Drilling Cycle (G183) (Reducing Step) -----	5-42
5.7 Canned Cycle for Tool Change (Non-stop ATC) (G100) -----	5-43
5.7.1 W1000Xd1/S300Xd1/S500Xd1/S700Xd1/U500Xd1/R450Xd1/R650Xd1 -----	5-43

5.7.2 M200Xd1 -----	5-45
5.7.2.1 Operation when spindle is selected (M141 modal) -----	5-45
5.7.2.2 Operation when lathe spindle is selected (M142 modal)-----	5-45
5.7.2.3 Notes -----	5-46
5.7.3 R650Xd1 40MG -----	5-47
5.7.3.1 Tool change operation -----	5-49
5.7.3.2 Next tool preparation operation -----	5-50
5.7.4 Simultaneously Commandable M Codes -----	5-50
5.7.5 Automatic Command of Tool Data in Tool Change-----	5-53
5.8 Coordinate Calculation Function -----	5-54
5.8.1 Outline-----	5-54
5.8.2 Coordinate calculation-----	5-54
5.8.3 Coordinates Calculation Parameters-----	5-54
5.8.4 Description of Coordinate Calculation Function-----	5-55
5.8.4.1 Bolt Hole Circle (G36) -----	5-55
5.8.4.2 Line (Angle) (G37) -----	5-56
5.8.4.3 Line (X, Y) (G38) -----	5-57
5.8.4.4 Grid (G39) -----	5-58
5.8.5 Examples of Application-----	5-59
CHAPTER 6 MACRO -----	6-1
6.1 What is Macro? -----	6-2
6.2 Variables Function-----	6-4
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 -----	6-9
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-----	6-12
6.2.6.12 Workpiece Counter-----	6-12
6.2.6.13 Result of Auto Workpiece Measurement-----	6-13
6.2.6.14 Rotary Fixture Offset -----	6-13
6.2.6.15 Machining load monitor -----	6-13
6.2.6.16 Rotation axis / Tilt axis -----	6-14
6.3 Calculation Function -----	6-15
6.3.1 Types of Calculation-----	6-15
6.3.2 Precedence of Calculation -----	6-15
6.3.3 Precautions for Calculation -----	6-16
6.4 Control Function -----	6-17
6.4.1 GOTO Instruction (Unconditional Branching)-----	6-17
6.4.2 IF Instruction (Conditional Branching) -----	6-18
6.4.3 WHILE Instruction (Repetition) -----	6-19
6.4.4 Precautions for control function -----	6-20
6.5 Call Function -----	6-23
6.5.1 Simple Call Function -----	6-23
6.5.2 Modal Call Function-----	6-24
6.5.3 G Code Macro Call -----	6-25
6.5.4 M Code Macro Call -----	6-26
6.5.5 Macro Call Arguments -----	6-28
6.5.5.1 Argument for simple/modal call function -----	6-28

Contents

6.5.5.2 Argument for G code macro call -----	6-29
6.5.5.3 Argument for M code macro call -----	6-30
6.5.5.4 Special notes on macro call arguments -----	6-31
6.5.6 Difference Between G65 and M98/M198 -----	6-31
6.5.7 Multiple Call-----	6-32
6.6 External Output Function -----	6-33
6.6.1 POPEN -----	6-33
6.6.2 BPRNT -----	6-34
6.6.3 DPRNT -----	6-35
6.6.4 PCLOS -----	6-36
6.6.5 External Output to Memory Card -----	6-36
6.6.6 External Output to FTP(S) Server -----	6-37
6.6.7 Precautions for External Output Command -----	6-38
6.7 Interrupt Macro (Option) -----	6-39
6.7.1 Interrupt Type -----	6-41
6.7.2 Call Type -----	6-43
6.7.3 Acceptance Type -----	6-43
6.7.4 Interrupt Macro and Modal Information -----	6-44
6.7.5 Interrupt Macro and Current Position -----	6-45
6.7.6 Macro Statement and NC Statement-----	6-45
6.7.7 Restrictions -----	6-46
6.7.8 Reference Folder When Interrupt Type Macro is Executed -----	6-47
CHAPTER 7 AUTOMATIC WORKPIECE MEASUREMENT -----	7-1
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 -----	7-11
7.4.1 Corner -----	7-11
7.4.2 Parallel -----	7-15
7.4.3 Circle Center-----	7-18
7.4.4 Workpiece Top Surface -----	7-22
7.4.5 Positioning to Measurement Position -----	7-23
7.5 Measurement Results Processing-----	7-24
7.5.1 Display Screen for Measurement Results -----	7-24
7.5.2 Apply Measurement Results to Workpiece Coordinates-----	7-25
7.6 Lock Key Operations -----	7-26
7.7 Program Restart Operation -----	7-26
CHAPTER 8 SUB PROGRAM FUNCTION -----	8-1
8.1 Overview -----	8-2
8.2 Create Sub Program -----	8-3
8.3 Simple Call Function-----	8-4
8.4 External Sub Program Call Function -----	8-6
8.5 Specify Return Number from Sub Program Function-----	8-7
8.6 Call Specifying Sequence No. -----	8-8
CHAPTER 9 FEED FUNCTION -----	9-1
9.1 Feed Function -----	9-2
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
9.2 Automatic F Command at Tool Change -----	9-4

CHAPTER 10 SPINDLE RELATED FUNCTIONS**(S FUNCTION) ----- 10-1****10.1 S Function ----- 10-2**

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

10.2 M Function (Spindle Control) ----- 10-5

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

10.3 M Function (Lathe Spindle Control) ----- 10-6

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

10.4 M Function (Spindle Selection) ----- 10-6

10.4.1	Spindle Selection (M141)-----	10-6
10.4.2	Lathe Spindle Selection (M142)-----	10-6

CHAPTER 11 TOOL RELATED FUNCTIONS (T FUNCTION)---11-1**11.1 T Function----- 11-2**

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

11.2 M Function (Tool Control) ----- 11-3

11.2.1	Tool Change (M06)-----	11-3
11.2.2	Tool Life Counter (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

CHAPTER 12 M FUNCTION ----- 12-1**12.1 Outline of M Function ----- 12-2****12.2 M Code List ----- 12-2**

12.2.1	Multiple M Code Commands in One Block-----	12-5
--------	--	------

12.3 M Function (Program Control) ----- 12-8

12.3.1	Program Stop (M00)-----	12-8
12.3.2	Optional Stop (M01)-----	12-8
12.3.3	End of Program (M02, M30)-----	12-8
12.3.4	Workpiece Counter Specification (M211 to M214)-----	12-8
12.3.5	Workpiece Counter Cancel (M221 to M224)-----	12-8
12.3.6	To Prohibit Reading Ahead (M159)-----	12-8
12.3.7	Time Measurement (M258 and M259)-----	12-9
12.3.8	Z-axis Perimeter Mode (M300 and M301)-----	12-9

12.4 M Function (Signal Control) ----- 12-11

12.4.1	2-digit BCD Signal Output-----	12-11
12.4.2	Tool Breakage Error Check (M120 and M121)-----	12-11
12.4.3	Checking Measurement Instrument Detection Signal ON (M320)-----	12-11
12.4.4	Checking Measurement Instrument Detection Signal OFF (M321, M324 to M327)-----	12-11
12.4.5	M Signal Level Outputs (M400 to M409, M480 to M487)-----	12-12
12.4.6	One-shot Output (M450, M451, M455, M456)-----	12-12
12.4.7	Waiting Until Response is Given (M460 to M469)-----	12-12
12.4.8	Signal Output to PLC (M801 to M899)-----	12-12

Contents

12.4.9 Expansion Signal Output (M900 to M999) -----	12-12
12.5 M Function (Additional Axis Control) -----	12-13
12.5.1 Pallet-related M Codes (M410, M411, M430, M431)-----	12-13
12.5.2 C-axis Unclamp/Clamp (M444/M430 and M445/M431)-----	12-13
12.5.3 B-axis Unclamp/Clamp (M440 and M441)-----	12-14
12.5.4 A-axis Unclamp/Clamp (M442 and M443)-----	12-15
12.6 M Function (In-position Check Distance) -----	12-16
12.6.1 Change In-position Check Distance (M270 to M279)-----	12-16
12.7 M Function (Time Constant Switch) -----	12-17
12.7.1 Tap Time Constant Selection (M241 to M249, M250, M251, M252 to M254)-----	12-17
12.8 M Function (Brake Load Test) -----	12-17
12.8.1 Execute Brake Load Test (M470 and M471)-----	12-17

CHAPTER 13 HIGH-ACCURACY MODE ----- 13-1

13.1 HIGH-ACCURACY MODE AIII ----- 13-1-1

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

13.2 HIGH-ACCURACY MODE B ----- 13-2-1

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 Path Offset Function -----	13-2-5
3.8 Additional Axis Travel Command -----	13-2-5
4 Explanations on Parameters -----	13-2-6

CHAPTER 14 5 AXES MACHINING FUNCTION ----- 14-1

14.1 Interpolation Using 5 Axes Simultaneously ----- 14-2

14.1.1 Overview -----	14-2
14.1.2 Linear Interpolation (G01) -----	14-2
14.1.3 Helical Thread Cutting Interpolation (G02) -----	14-4
14.1.4 Positioning (G00) -----	14-5
14.1.5 TCP Control (G43.4/G43.5) -----	14-6
14.1.6 Restrictions-----	14-6

14.2 TCP Control -----	14-7
14.2.1 Overview -----	14-7
14.2.1.1 Introduction -----	14-7
14.2.1.2 Prerequisites -----	14-7
14.2.1.3 Machine configuration -----	14-8
14.2.2 Program Commands -----	14-9
14.2.2.1 TCP control ON command -----	14-9
14.2.2.2 Cancel command (G49) -----	14-12
14.2.2.3 Command conditions-----	14-12
14.2.2.4 Command functions-----	14-13
14.2.2.5 Feedrate -----	14-14
14.2.2.6 Positioning-----	14-15
14.2.2.7 Interpolation method on additional axis -----	14-15
14.2.3 Coordinate System -----	14-16
14.2.3.1 Programming coordinate system -----	14-16
14.2.3.2 Fixed position in table coordinate system -----	14-16
14.2.3.3 Operation example (additional axis at ON command is 0 degrees)-----	14-17
14.2.3.4 Operation example (additional axis at ON command is not 0 degrees) -----	14-18
14.2.4 Startup Operation-----	14-19
14.2.4.1 No travel command -----	14-19
14.2.4.2 Travel command-----	14-20
14.2.4.3 Simultaneous Commands with Tool Change -----	14-21
14.2.5 Cancel operation-----	14-23
14.2.5.1 No travel command -----	14-23
14.2.5.2 Travel command-----	14-23
14.2.5.3 Cancelled by other commands -----	14-24
14.2.6 Machining Parameter -----	14-24
14.2.6.1 Overview of acceleration/deceleration control -----	14-24
14.2.6.2 Stop command -----	14-25
14.2.6.3 Override functions-----	14-25
14.2.6.4 Machining level change (M280 to M287)-----	14-26
14.2.6.5 Program usage -----	14-26
14.2.6.6 Parameter adjustment screen-----	14-28
14.2.6.7 Programmable data input (G210)-----	14-28
14.2.7 Other -----	14-29
14.2.7.1 Operation pause-----	14-29
14.2.7.2 Emergency stop-----	14-29
14.2.7.3 Single operation -----	14-29
14.2.7.4 Cutting override -----	14-29
14.2.7.5 Rapid traverse override -----	14-29
14.2.7.6 MDI operation -----	14-29
14.2.7.7 Manual operation -----	14-29
14.2.7.8 Dry run-----	14-31
14.2.7.9 Software limit-----	14-33
14.2.7.10 Skip function -----	14-34
14.2.7.11 Graphic function -----	14-34
14.2.7.12 Smooth path offset function -----	14-34

Contents

(This page was intentionally left blank.)

CHAPTER 1	PROGRAM STRUCTURE	1
CHAPTER 2	COORDINATE COMMANDS	2
CHAPTER 3	PREPARATION FUNCTION	3
CHAPTER 4	PREPARATION FUNCTION (COMPENSATION FUNCTION)	4
CHAPTER 5	PREPARATION FUNCTION (CANNED CYCLE)	5
CHAPTER 6	MACRO	6
CHAPTER 7	AUTOMATIC WORKPIECE MEASUREMENT	7
CHAPTER 8	SUB PROGRAM FUNCTION	8
CHAPTER 9	FEED FUNCTION	9
CHAPTER 10	SPINDLE RELATED FUNCTIONS (S FUNCTION)	10
CHAPTER 11	TOOL RELATED FUNCTIONS (T FUNCTION)	11
CHAPTER 12	M FUNCTION	12
CHAPTER 13	HIGH-ACCURACY MODE	13
CHAPTER 14	5 AXES MACHINING FUNCTION	14

(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

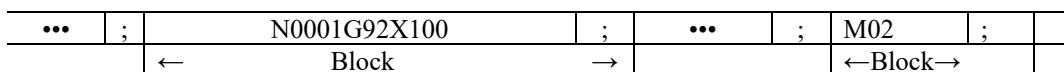
N0010 G91X10
:
:
:
M99;

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.)



(NOTE 1) The end of block code in ISO code is "LF" OA (hexadecimal).

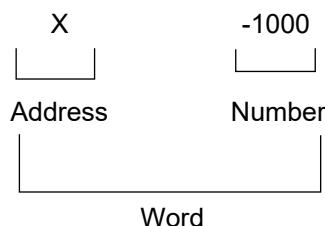
In EIA code, it is "CR" 80 (hexadecimal).

(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: programming) for the coordinate values and travel amount.

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

<Program unit>: 0 (Base)

Specifying program	Command axis	Actual amount (meters)	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 (meters)	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 (meters)	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

Select the <Program feed rate unit> in user parameter (switch 1: programming) for the feedrate.

<Program feed rate unit>: 0 (Basic)

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

<Program feed rate unit>: 1 (Minimum)

Specifying program	Command axis	Actual amount (meters)	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 (meters)	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)

- Usable digits in address number commands (coordinate values and travel commands)
- <Program unit>: 0(Basic)

Minimum unit setting	Commands with/without decimal points	Command axis	Usable digits in commands	
			meters	inches
Type 1 (Micron)	Commands with decimal points (Ex: "1.")	Feed axis	999999.999	99999.9999
		Rotation axis	999999.999	999999.999
	Commands without decimal points (Ex: "1")	Feed axis	999999	99999
		Rotation axis	999999	999999
Type 2 (Submicron)	Commands with decimal points (Ex: "1.")	Feed axis	999999.9999	99999.99999
		Rotation axis	999999.9999	999999.9999
	Commands without decimal points (Ex: "1")	Feed axis	999999	99999
		Rotation axis	999999	999999

<Program unit>:1 (Minimum)

Minimum unit setting	Commands with/without decimal points	Command axis	Usable digits in commands	
			meters	inches
Type 1 (Micron)	Commands with decimal points (Ex: "1.")	Feed axis	999999.999	99999.9999
		Rotation axis	999999.999	999999.999
	Commands without decimal points (Ex: "1")	Feed axis	999999999	999999999
		Rotation axis	999999999	999999999
Type 2 (Submicron)	Commands with decimal points (Ex: "1.")	Feed axis	999999.9999	99999.99999
		Rotation axis	999999.9999	999999.9999
	Commands without decimal points (Ex: "1")	Feed axis	9999999999	9999999999
		Rotation axis	9999999999	9999999999

(NOTE) If the value includes a place value lower than the usable digits in the minimum unit setting, then that part is rounded off in the command.

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 code (.....) Comment Control in 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**
- 2.4 Table Coordinate System**

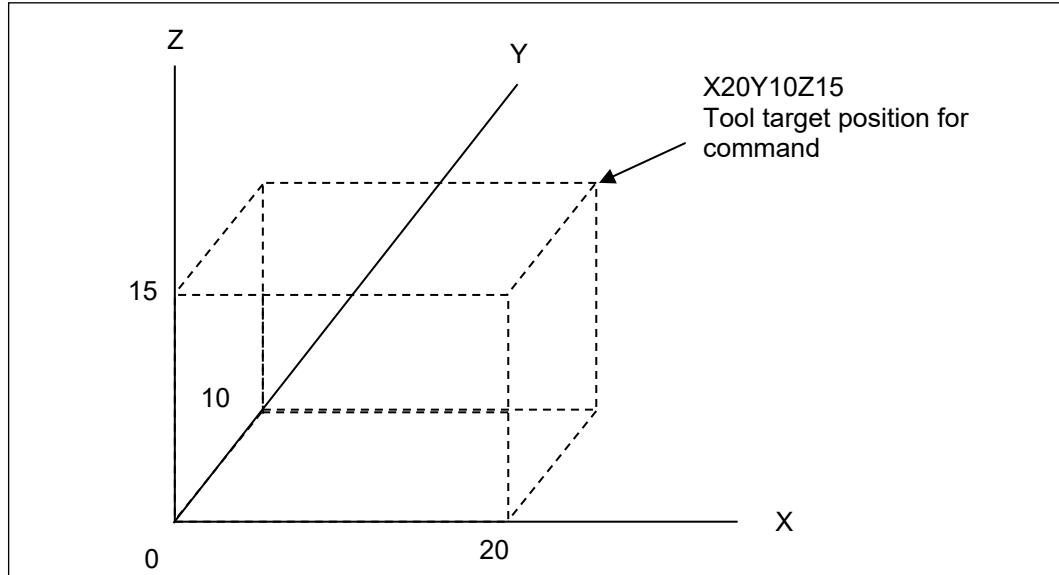
2.1 Coordinate System and Coordinate Commands

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
3. Table coordinate system

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

2



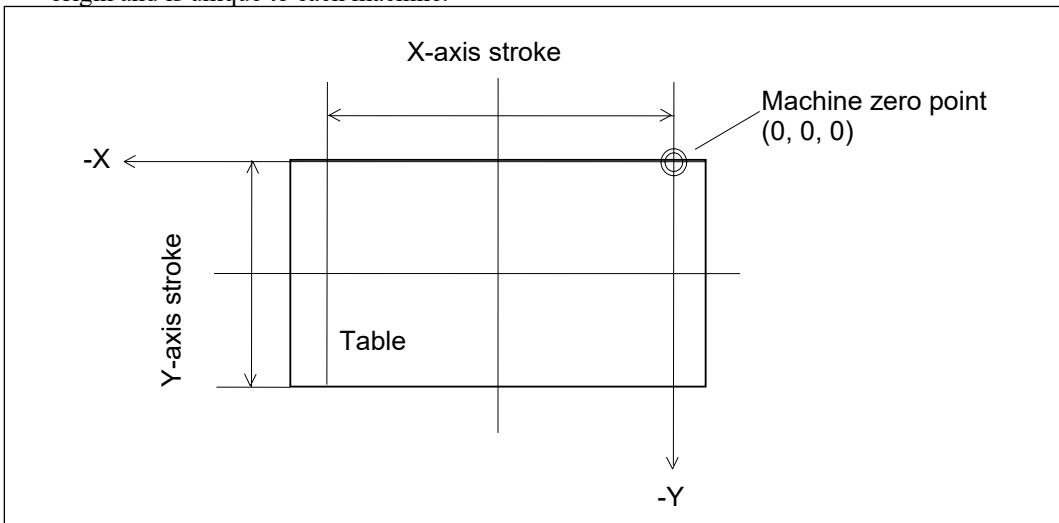
2.2 Machine Zero Point and Machine Coordinates

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.

2.4 Table Coordinate System

The table coordinate system refers to a coordinate system that is fixed for the table, so that as the table rotates, the zero position in the coordinate system and the direction of X, Y and Z change. Refer to “14.2 TCP control” for further details.

(This page was intentionally left blank.)

CHAPTER 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 Programmable Parameter Input (G10)
- 3.8 Coordinate System
(G17 to 19, G52 to 59, G54.1, G92, G68.2)
- 3.9 Soft Limit
- 3.10 Reference Position (G28 to 30)
- 3.11 Skip Function (G31, G131/G132)
- 3.12 Scaling (G50/G51)
- 3.13 Programmable Mirror Image (G50.1/51.1)
- 3.14 Rotational Transformation Function (G68/69, G168)
- 3.15 Absolute Command and Incremental Command
(G90/91)
- 3.16 Change of Tap Twisting Direction (G133/134)
- 3.17 G code Priority
- 3.18 Programmable Data Input (High Accuracy) (G210)
- 3.19 Thread Cutting
- 3.20 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.

3

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	One-shot
	G392	Thread cutting cycle	
	G04	Dwell	
	G09	Exact stop check	
	G10	Programmable data input/Parameter input mode start	
	G10L52	Programmable parameter input mode start	Modal(NOTE 2) (NOTE 3)
	G11	Programmable parameter input mode end	
	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 nd to 6 th 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 -	
	G43.4	TCP control command (ABC command format)	
	G43.5	TCP control command (IJK command format)	
	G143	Tool position offset +	
	G144	Tool position offset -	
	G49*	Tool length/position offset cancel	
	G50*	Scaling cancel	Modal
	G51	Scaling	

Group	G code	Description	Modal/One-shot
	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	
	G55	Working coordinate system selection 2	Modal
	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	
	G64*	Cutting mode	Modal
	G65	Macro call	
	G66	Macro modal call	One-shot
	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)	
	G74	Canned cycle (Reverse tapping cycle)	Modal
	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

Chapter 3 Preparation Function

3

Group	G code	Description	Modal/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
	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 workpiece top surface	
	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 1) Specify the modal during power startup in the user parameter (switch 1: canned cycle) <Lathe machining infeed direction when power is turned ON>.

(NOTE 2) Modal information cannot be read in the macro.

(NOTE 3) Even if the user parameters (switch 3: modal display setting): <G code for modal display 1> to <G code for modal display 16> are not set to 0 (zero), the G code modal does not display on the <Modal info 1> and <MDI operation> screens.

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 (switch 1: program) <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.

3

<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.

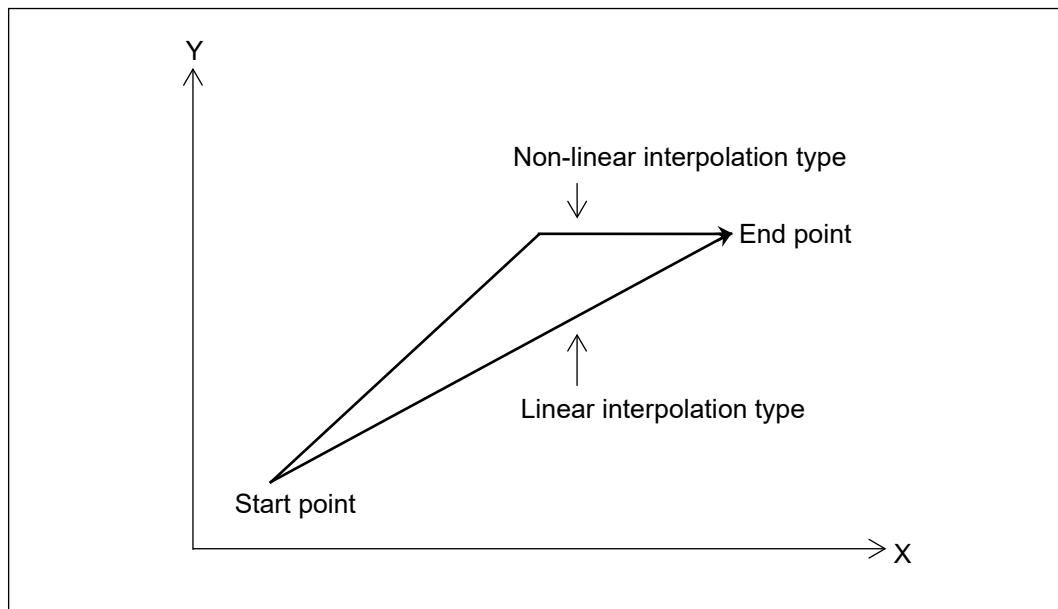
Be careful of collisions because if the margin of error on the path for positioning (including the additional axis) is greater than the positioning only on the linear axis, then the positioning changes depending on the feedrate and the load.

<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 (switch 1: program) <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.



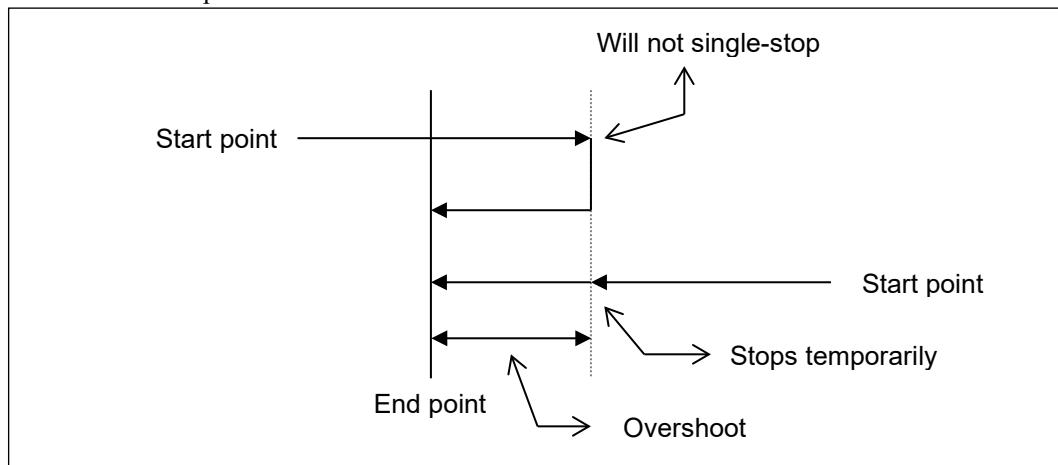
(NOTE 1) The in-position check refers to checking whether or not the position detected on the machine has reached a range within the target position (end point). However, when there are successive Z-axis only operations, such as G0Z_ → G0Z_, the in-position check may not be carried out.

- (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 (switch 1: program) <Positioning method>.
- (NOTE 5) While the Z-axis perimeter mode is on (M300), the Z-axis perimeter operation is carried out when there are consecutive operations for the Z-axis up positioning and the X-axis or Y-axis positioning operations. Refer to “Chapter 12 M function” for further details on the Z-axis perimeter operation.

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.

Set the overrun travel amount with the user parameter (switch 2: single direction positioning excess travel amount) <X-axis (to 8th-axis) single direction positioning excess travel amount>.

- (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>.
- (NOTE 6) When a command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

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

(NOTICE) If a door is opened when the door interlock mode is set to automatic mode (or when the door interlock mode is set to setting mode and the [ENABLE] switch is OFF), then the rotation axis servo turns OFF. As a result, the axis may shift out of place due to the load from the workpiece, etc. Thereafter, if the shift amount when closing the door is smaller than the machine parameter (system 2: QT-axis) <Return angle with servo controller ON> (or if it is smaller than the machine parameter (system 2: additional axis) <Return angle with servo controller ON> (5th to 8th-axes)), 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>. In addition, the workpiece will not be machined at the correct position if the machining is carried out during this alarm.

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

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

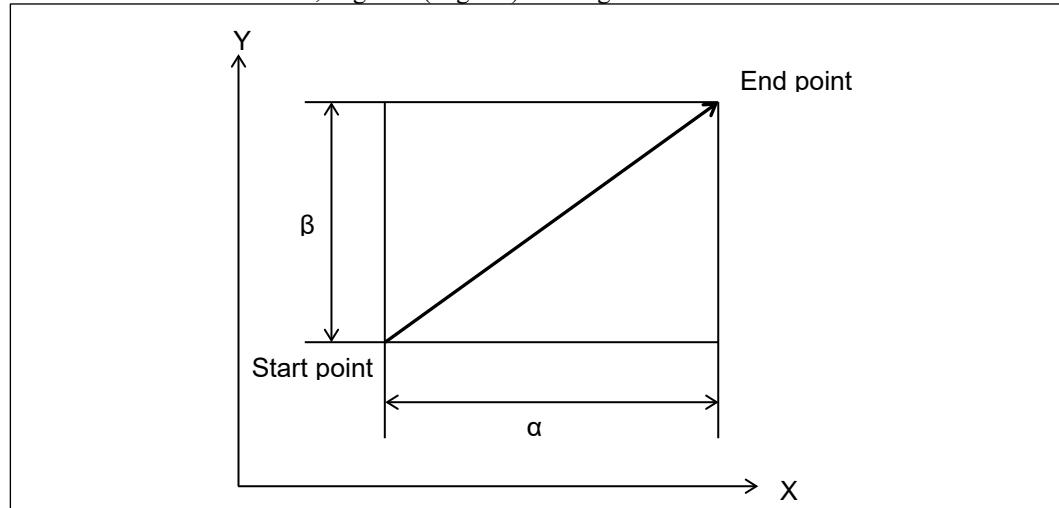
```
G01 X_ Y_ Z_ A_ F_;  
G01 X_ Y_ Z_ B_ F_;  
G01 X_ Y_ Z_ C_ F_;
```

Axis command is effective for up to 3 linear axes plus one additional axis. If a command for 2 additional axes is issued at the same time, the alarm <<Interpolation command using 5 axes simultaneously is not possible>> is triggered.

3

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.

When the command axis is the X-, Y- or Z-axis, mm/min (mm/rev) is recognized for the feedrate. When it is an additional axis, deg/min (deg/rev) is recognized for the feedrate.



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

When "G01 G91 X α Y β Z γ Ff;" is programmed:

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

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

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

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

Linear interpolation using a linear axis and rotation axis is as follows.

When the user parameter (switch 1: programming) <Rotation axis speed command type> is set to <Type 2>, the speed of each axis changes depending on the user parameter (switch 1: programming) <Standard circle radius> setting. Refer to section “3.3.1.1 Speed command for standard circle on rotation axis” for further details.

When "G01 G91 X α Y β Z γ B δ Ff" is programmed

Time required for distributing on B-axis

$$T = \frac{L}{f}$$

Feed rate along B axis

$$F_b = \frac{\delta}{T}$$

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

$$F_z = \frac{\gamma}{L} \cdot f$$

The L value is calculated based on the user parameter (switch 1: programming) <Rotation axis speed command type> settings below.

- <Type 1> $L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \delta^2}$
- <Type 2> $L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + (\delta \times \gamma \times \pi / 180)^2}$
r: User parameter (switch 1: programming) <Standard circle radius>

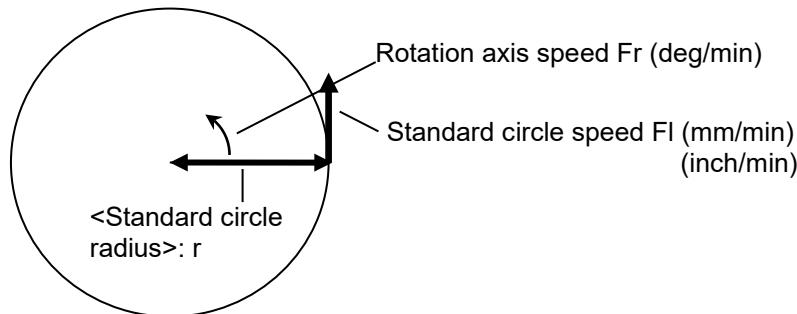
(NOTE) When the <Rotation axis speed command type> is set to <Type 2> and the <Standard circle radius> is set to 0, the alarm <<User param. setting error (switch 1)>> is triggered.

3.3.1.1 Speed command for standard circle on rotation axis

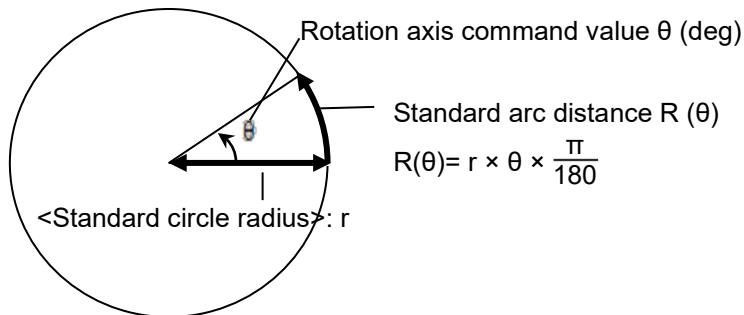
When the user parameter (switch 1: programming) <Rotation axis speed command type> is set to <Type 2>, the command value for the rotation axis is the perimeter Fl speed based on the user parameter (switch 1: programming) <Standard circle radius>. The circle is a standard circle and the peripheral speed is referred to as the standard circle speed.

When a linear interpolation command is issued on the rotation axis, the relationship between the F command value (= standard circle speed Fl) and the actual command speed Fr (deg/min) for the rotation axis is expressed in the following formula.

$$Fr = Fl \times 180 / (\pi \times r)$$



In addition, when there is a linear interpolation on a linear axis and rotation axis, the speed is applied as follows. The travel command value θ deg for the rotation axis is converted to the arc length on a standard circle (referred to as the standard arc distance). 1 mm on the standard arc distance = 1 mm on the linear axis when the machine unit system is set to meters. 1 inch on the standard arc distance = 1 inch on the linear axis when the machine unit system is set to inches. These ratios are applied and used for the speed.



Ex.1: Rotation axis only command

An example is given for each machine unit system. <Standard circle radius> is set to 20 mm or 0.7874 inches.

When machine unit system is in meters:

Standard circle distance (mm) = $R(30 \text{ deg}) = 20 \text{ mm} \times 30 \times \pi / 180 \approx 10.47 \text{ mm}$

G94 G91 G1 A30. F2540

A-axis feedrate (deg/min)

= Standard circle speed (mm/min) × Command value (deg) ÷ L (mm/min)

= $2540 \text{ (mm/min)} \times 30 \text{ (deg)} \div \sqrt{(\text{Standard circle distance}} (\approx 10.47 \text{ mm})^2)$

$\approx 7276 \text{ (deg/min)}$

When machine unit system is in inches:

G94 G91 G1 A30. F100

Standard circle distance (inch) = $R(30 \text{ deg}) = 0.7874 \text{ inches} \times 30 \times \pi / 180 \approx 0.412 \text{ inches}$

A-axis feedrate (deg/min)

= Standard circle speed (inch/min) × Command value (deg) ÷ L (inch/min)

= $100 \text{ (inch/min)} \times 30 \text{ (deg)} \div \sqrt{(\text{Standard circle distance}} (\approx 0.412 \text{ inches})^2)$

$\approx 7276 \text{ (deg/min)}$

Ex.2: Linear axis + Rotation axis command

When machine unit system is in meters:

G91 G1 X40 A30. F1000

X-axis feedrate (mm/min) = $1000 \times 40 \div (\sqrt{40^2 + 10.47^2}) \approx 967 \text{ (mm/min)}$

A-axis feedrate (deg/min) = $1000 \times 30 \div (\sqrt{(40^2 + 10.47^2)}) \approx 725 \text{ (deg/min)}$

When machine unit system is in inches:

G91 G1 X1.575 A30. F196.85

X-axis feedrate (inch/min) = $196.85 \times 1.575 \div (\sqrt{(1.575^2 + 0.412^2)}) \approx 190 \text{ (inch/min)}$

A-axis feedrate (deg/min) = $196.85 \times 30 \div (\sqrt{(1.575^2 + 0.412^2)}) \approx 3628 \text{ (deg/min)}$

Ex.3: Relationship between <Standard circle radius> and feedrate

When the user parameter (switch 1: programming) <Standard circle radius> value is made greater, the additional axis feedrate for the F command value becomes smaller. Similarly, when the radius is made smaller, the additional axis feedrate becomes greater.

In addition, when the value is set to 57.295 ($=180 \div \pi$), regardless of the machine unit system setting, a speed equivalent to <Type 1> applies.

If the values are changed for <Rotation axis speed command type> and <Standard circle radius> in the program from example 2, the following speed applies.

When machine unit system is in meters:

G91 G1 X40 A30. F1000

	<Type 1>	<Type 2>	
<Standard circle radius>	-	20 mm	57.295 mm
X-axis speed (mm/min)	800	967	800
A-axis speed (deg/min)	600	725	600

When machine unit system is in inches:

G91 G1 X1.575 A30. F196.85

	<Type 1>	<Type 2>	
<Standard circle radius>	-	0.7874 inches	57.295 inches
X-axis speed (inch/min)	10	190	10
A-axis speed (deg/min)	197	3628	197

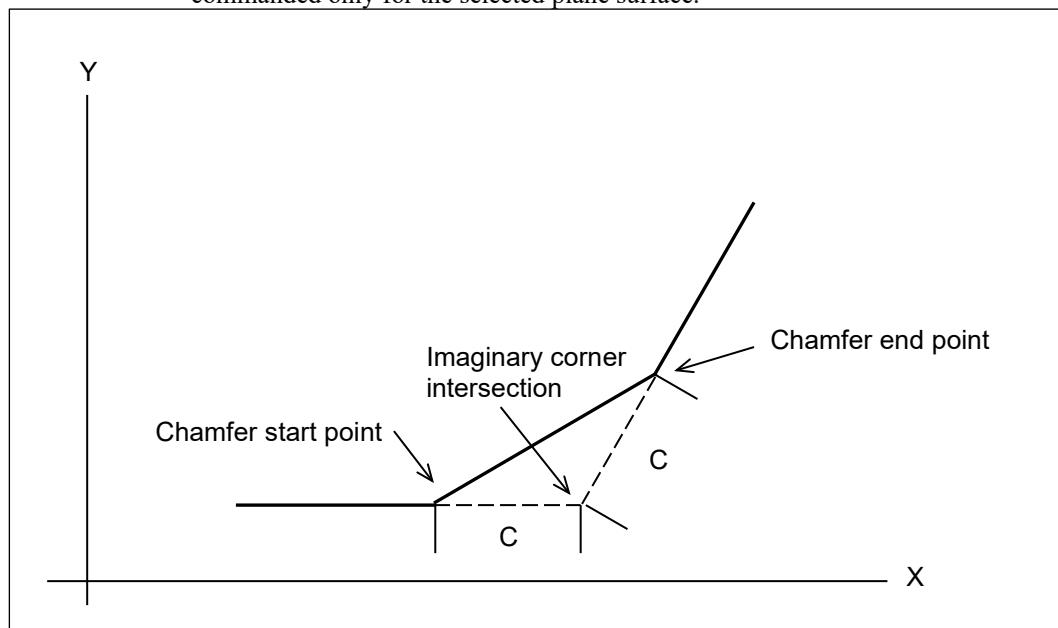
3.3.1.2 Chamfering to Desired Angle and Cornering R

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.



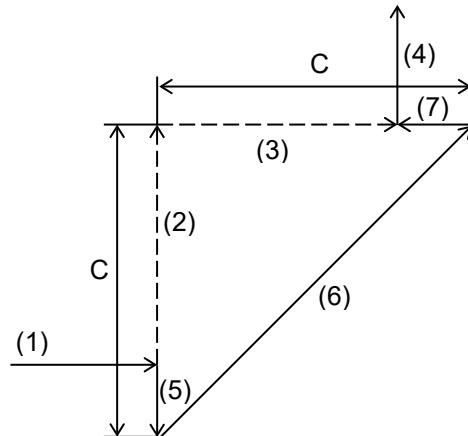
(NOTE 1) The corner chamfering command block and subsequent block must contain the interpolation command (G01-G03). When there is no interpolation command, or when a travel command is not possible on the next block, the alarm <>Angle chamfering/corner R command error>> is triggered.

Chapter 3 Preparation Function

- (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.
- (NOTE 5) A chamfering command is not possible while under TCP control.

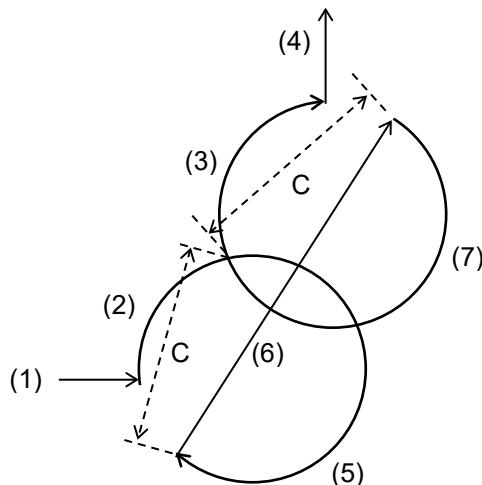
Example.1: Linear cutting

3



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



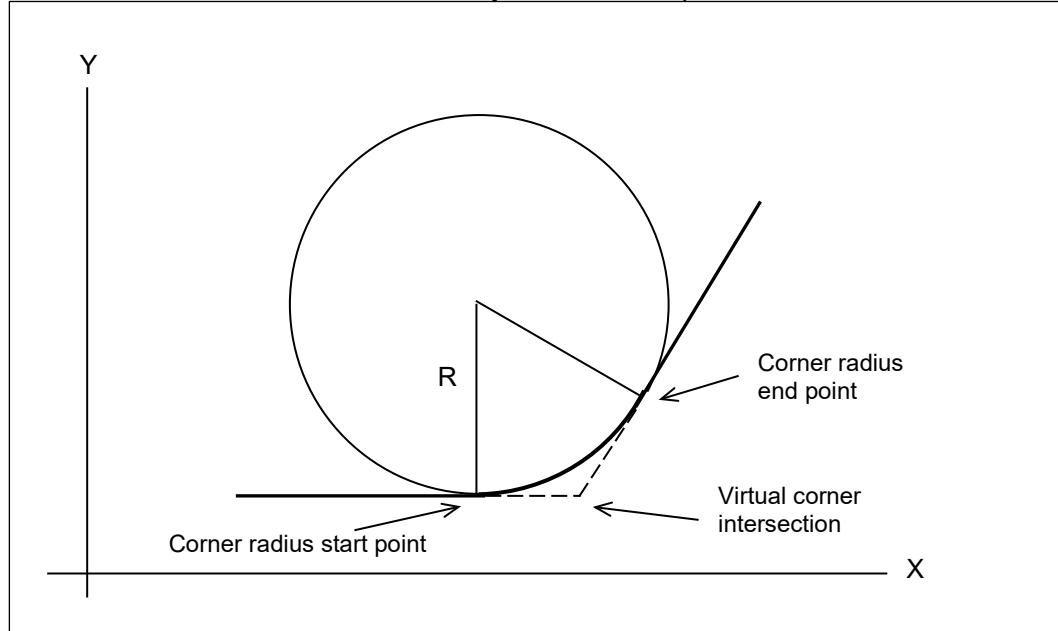
When the programmed path is (1)→(2)→(3)→(4) and the block (2) is C-specified, the machine operates through (1)→(5)→(6)→(7)→(4).

Cornering R Command format

G01 X_ Y_, R_;

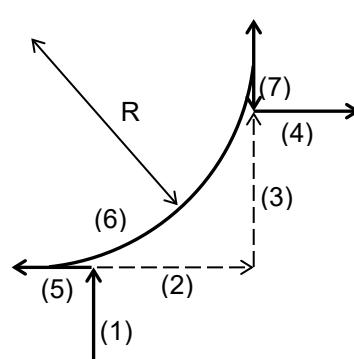
R : Radius of cornering

This can be commanded only for the selected plane surface.



- (NOTE 1) Corner rounding command block and the next block must be a corner interpolation command (G01 to G03). When there is no interpolation command, or when a travel command is not possible on the next block , the alarm <<Angle chamfering/corner R command error>> is triggered.
- (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 coner 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.
- (NOTE 6) A corner R command is not possible while under TCP control.

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_ $\left(\begin{array}{c} I_J \\ R \end{array} \right)$ F_;

G17 G03 X_ Y_ $\left(\begin{array}{c} I_J \\ R \end{array} \right)$ F_;

ZX plane:

G18 G02 Z_ X_ $\left(\begin{array}{c} K_I \\ R \end{array} \right)$ F_;

G18 G03 Z_ X_ $\left(\begin{array}{c} K_I \\ R \end{array} \right)$ F_;

YZ plane:

G19 G02 Y_ Z_ $\left(\begin{array}{c} J_K \\ R \end{array} \right)$ F_;

G19 G03 Y_ Z_ $\left(\begin{array}{c} J_K \\ R \end{array} \right)$ F_;

3

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.	
		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

G102	X_Z_	I_K_	F_;
G103		R_	

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.
Distance between start point and arc center		Z	Distance from the start point to the end point in the Z direction.
		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

G202	Y_Z_	J_K_	F_;
G203		R_	

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.
Distance between start point and arc center		Z	Distance from the start point to the end point in the Z 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 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).

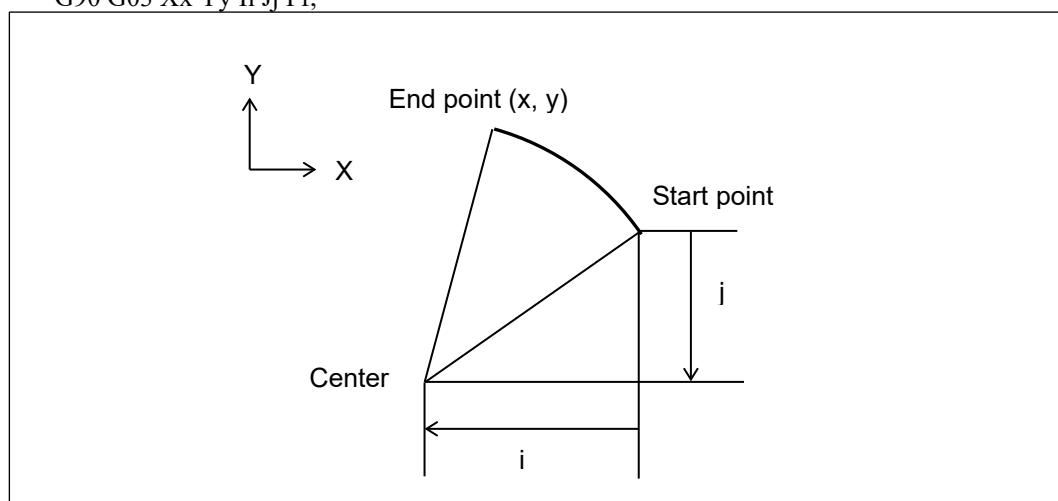
3.3.2.4 Precautions for Circular Interpolation

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.

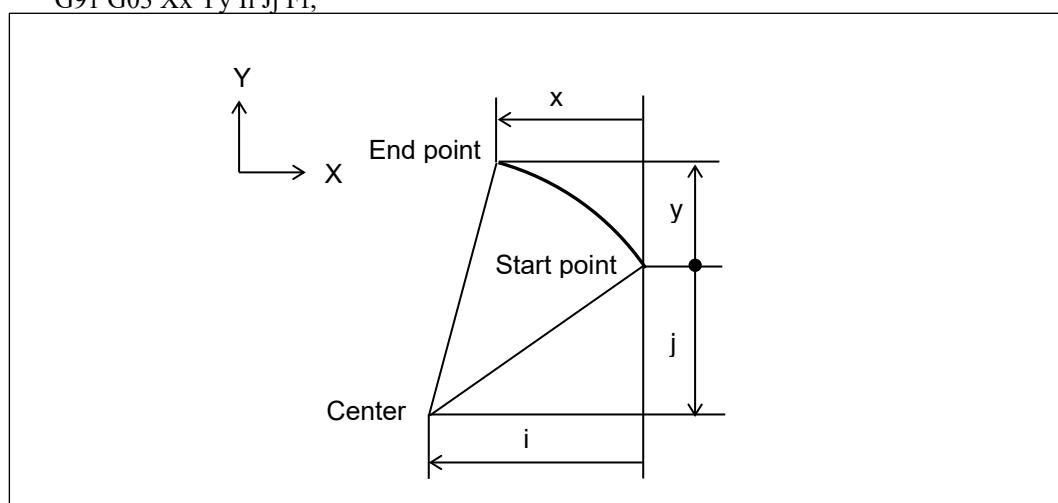
Center of the arc is specified by I, J, and K with respect to X, Y, and Z axis.

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;

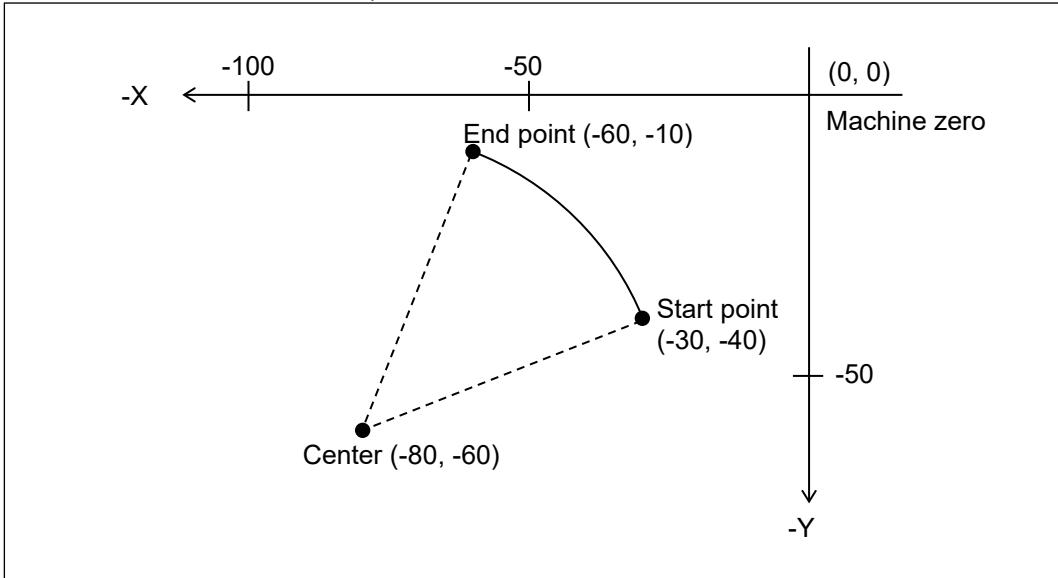


Absolute command

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

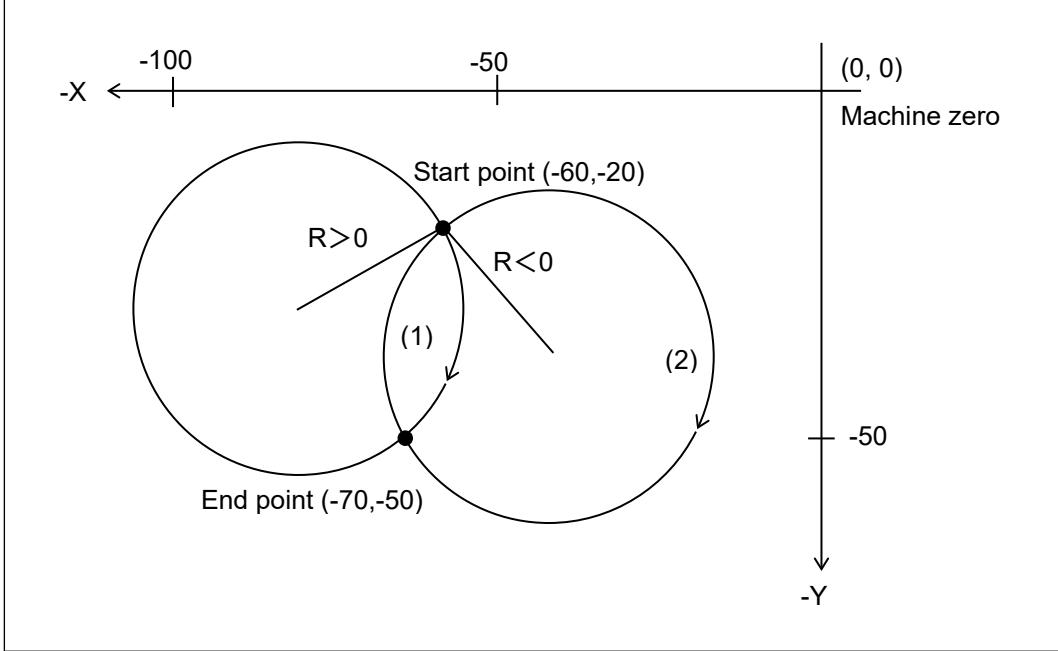
Incremental command

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.

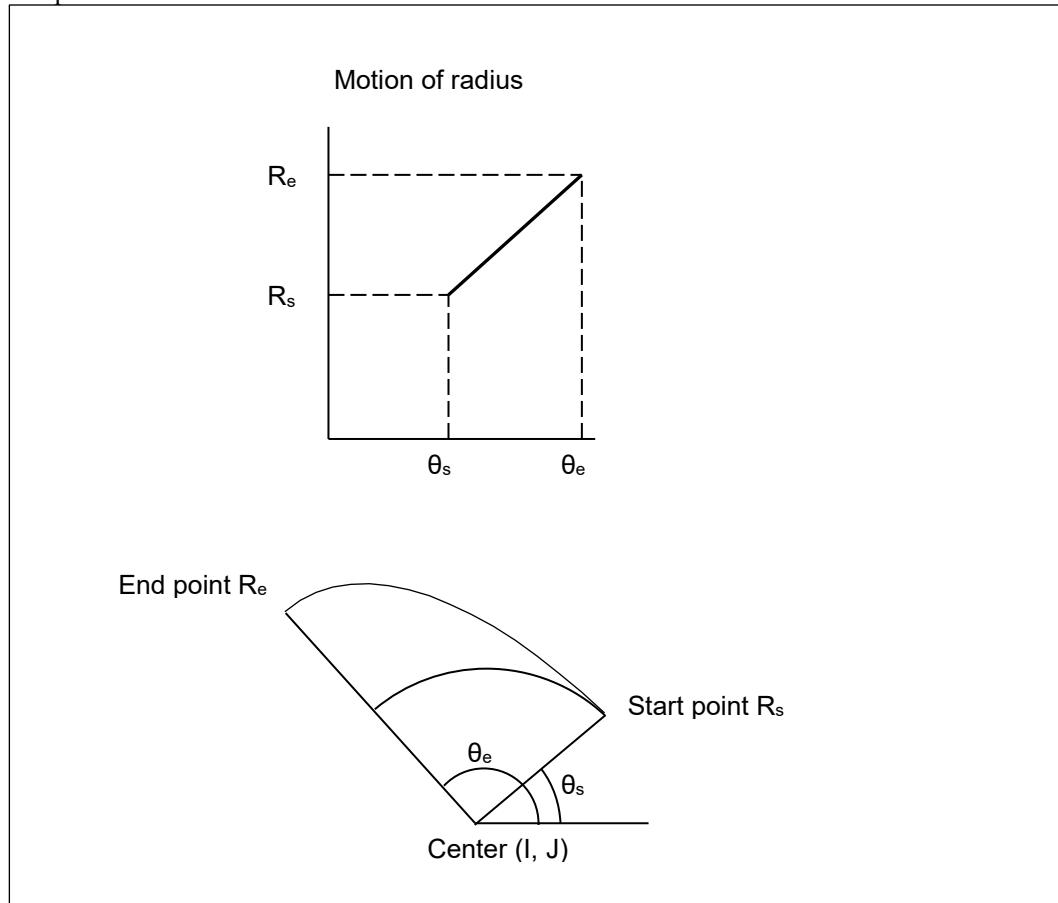
1. G02 X-70. Y-50. R25. F1000;
2. G02 X-70. Y-50. R-25. F1000;



- 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) If a command is issued with the center at I, J or K, then the arc is 360° (entire perimeter of circle).
 - (2) The alarm <<Arc Command Error>> occurs when the center is defined by R.
- The R command cannot be issued at the same time as the I, J and K commands. If the commands are issued at the same time, the alarm <<Arc calculation error>> is triggered.

Chapter 3 Preparation Function

- 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.
- A TCP control ON (G43.4/G43.5) command cannot be issued while in arc mode. Otherwise, the alarm <<Arc command error>> is triggered.
- A circular interpolation command cannot be issued while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

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

XY plane:
G17 (G02) X_ Y_ Z_ (I_ J_) (R_) (A_ B_ C_) F_;
ZX plane:
G18 (G02) Z_ X_ Y_ (K_ I_) (R_) (A_ B_ C_) F_;
YZ plane:
G19 (G02) Y_ Z_ X_ (J_ K_) (R_) (A_ B_ C_) 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..

When F is greater than the machine parameter (system 1: X-, Y- and Z-axes) <Maximum cutting travel speed> (X-, Y- and Z-axes) or greater than the rapid feedrate, the alarm <<Feedrate error>> is triggered.

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

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

F : Command speed (Selected plane axis)
 R : Radius
 θ : 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),
 When the angle equals 360° and the travel distance equals 2 mm:

$$F_z = (180*2*500) / (\pi*10*360) \\ \approx 15.9 \text{ (mm/min)}$$

If the calculated feedrate for the axis outside of the selected plane is: greater than the machine parameter (system 1: X-, Y- and Z-axes) <Maximum cutting travel speed> or the <Rapid feedrate> (X-, Y- or Z-axes), or greater than the machine parameter (system 2: additional axis) <Maximum cutting rotation speed> (5th- to 8th-axes) or <Rapid feedrate> (5th- to 8th-axes), then the slowest speed is applied and that minimum speed on both the selected plane axis and the axis outside of the selected plane are fixed and used.

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

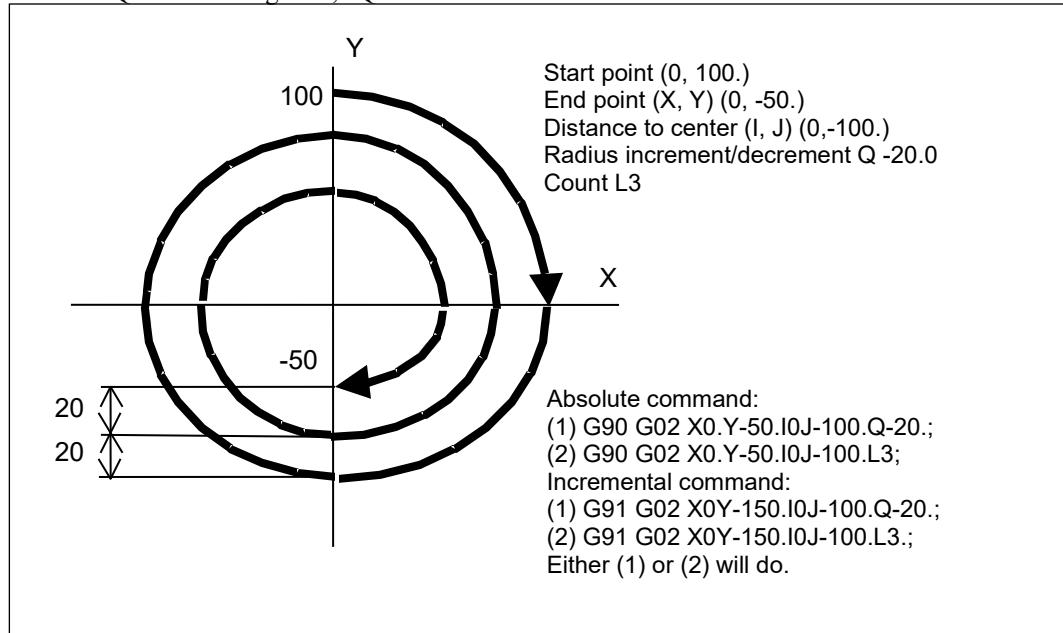
XY plane:
`{ G17 } G02 X_ Y_ I_ J_ Q_ L_ F_;`
`{ G17 } G03 X_ Y_ I_ J_ Q_ L_ F_;`

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

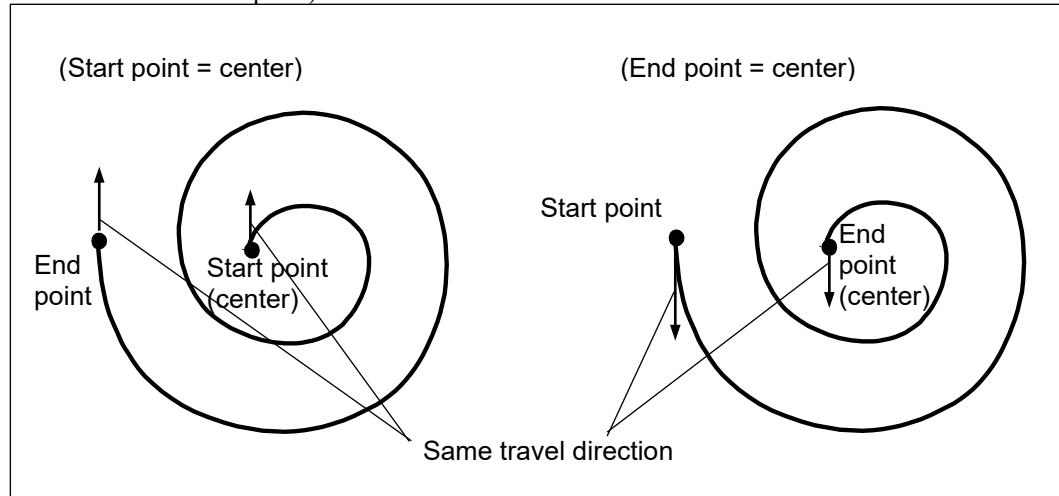
YZ plane:
`{ G19 } G02 Y_ Z_ J_ K_ Q_ L_ F_;`
`{ G19 } G03 Y_ Z_ J_ K_ Q_ L_ F_;`

- 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.
- (NOTE 3) The alarm <<Cutter compensation too large>> is triggered when the compensated tool path crosses or touches the center of the spiral.
- (NOTE 4) If the spiral end point required due to a radius increase/decrease per rotation does not match the end point of the program and that difference exceeds the user parameter (switch 1: program) <Arc radius error limit>, the alarm <<Arc radius error has exceeded the limit.>> is triggered.
- (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. When a command is issued, the alarm <<Address where command is not possible>> is triggered.
- (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) The alarm <<Arc Command Error>> occurs when a radius is specified by R parameters.
- (NOTE 9) The alarm <<Arc Command Error>> occurs when radius increment/decrement is zero.
- (NOTE 10) 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 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 circule 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_;

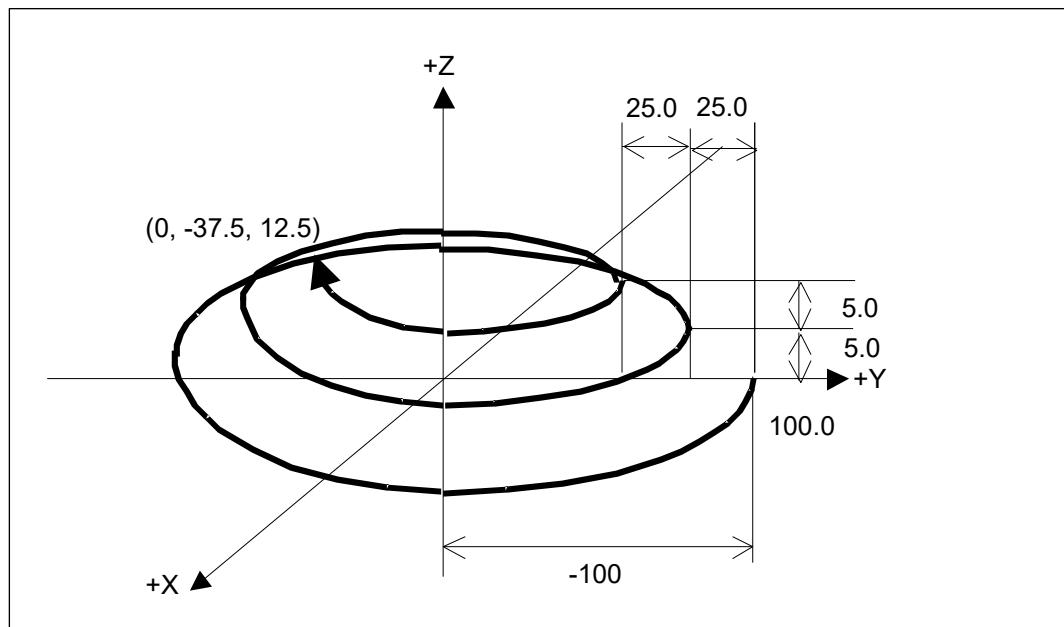
3

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.*
F	:	Cutting speed (Selected plane axis)

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

- * 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. $\left[\begin{array}{c} K5 \\ Q-25 \\ L3 \end{array} \right]$ F300.;

Incremental command

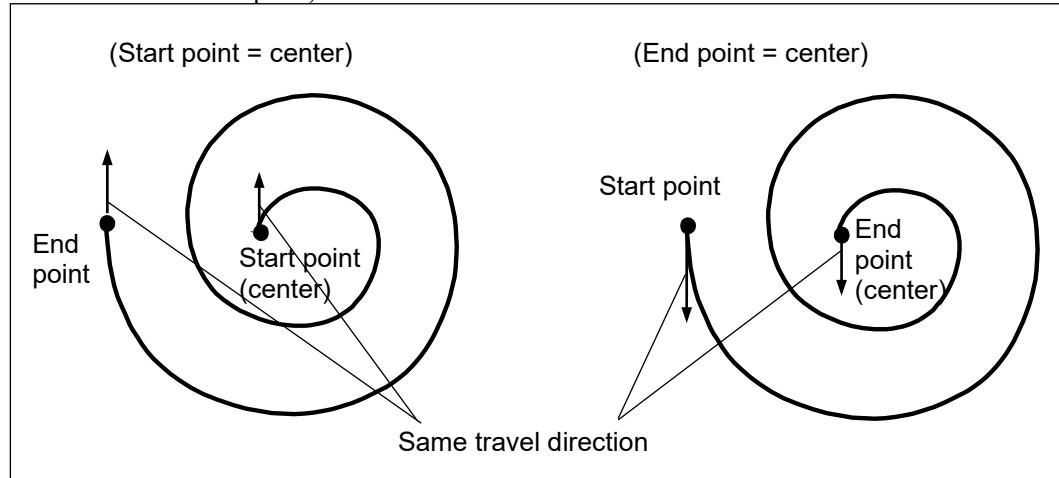
G91 G02 X0. Y-137.5 Z12.5 I0. J-100. $\left[\begin{array}{c} K5 \\ Q-25 \\ L3 \end{array} \right]$ F300.;

- (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.
- (NOTE 3) The alarm <<Cutter compensation too large>> is triggered when the compensated tool path crosses or touches the center of the conical.
- (NOTE 4) If the conical end point required due to a radius increase/decrease per rotation does not match the end point of the program and that difference exceeds the user parameter (switch 1: program) <Arc radius error limit>, the alarm <<Arc radius error has exceeded the limit.>> is triggered.
- (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. When a command is issued, the alarm <<Address where command is not possible>> is triggered.
- (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.

Chapter 3 Preparation Function

- (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 circule 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).

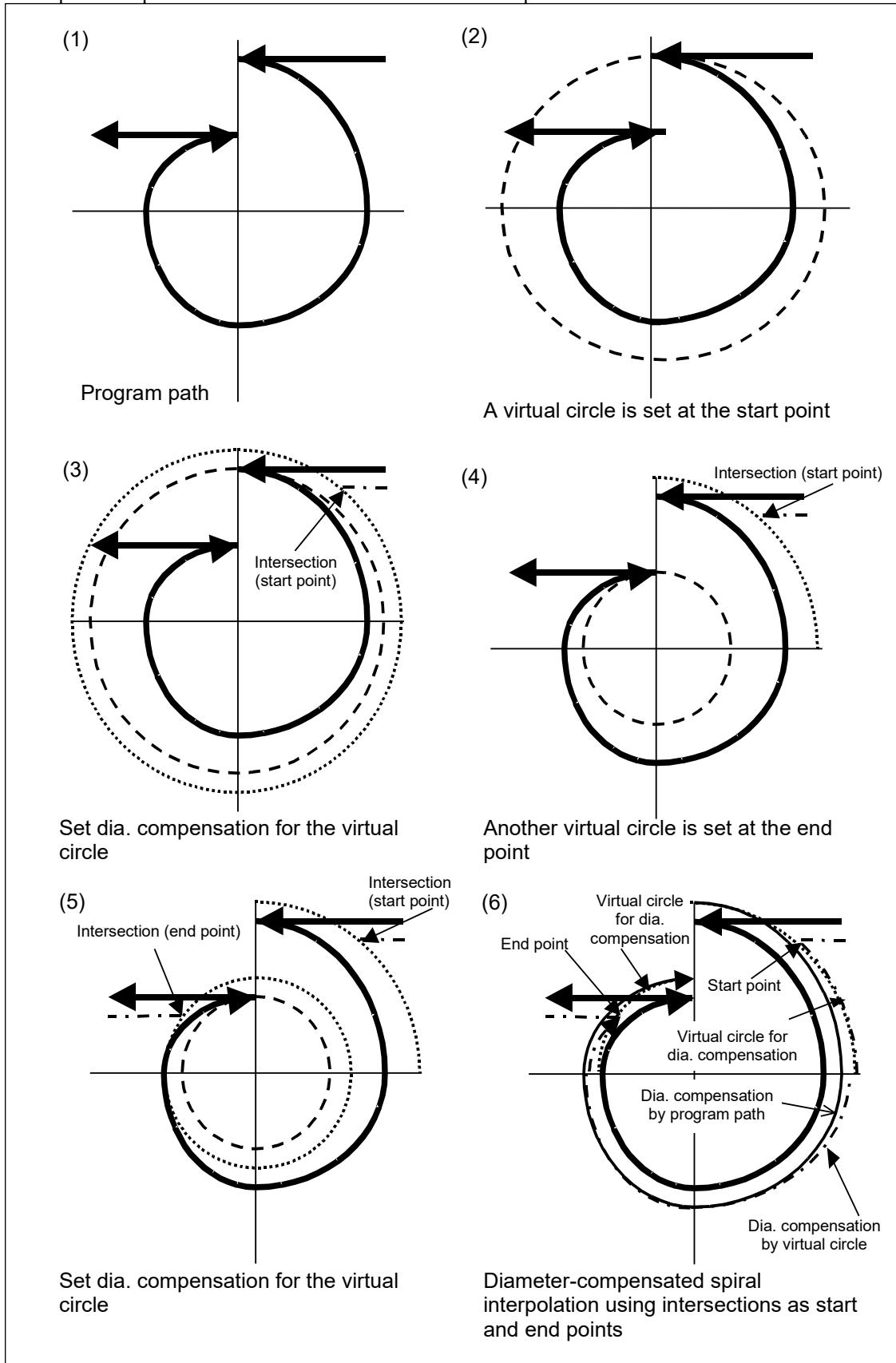
3



- (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_;

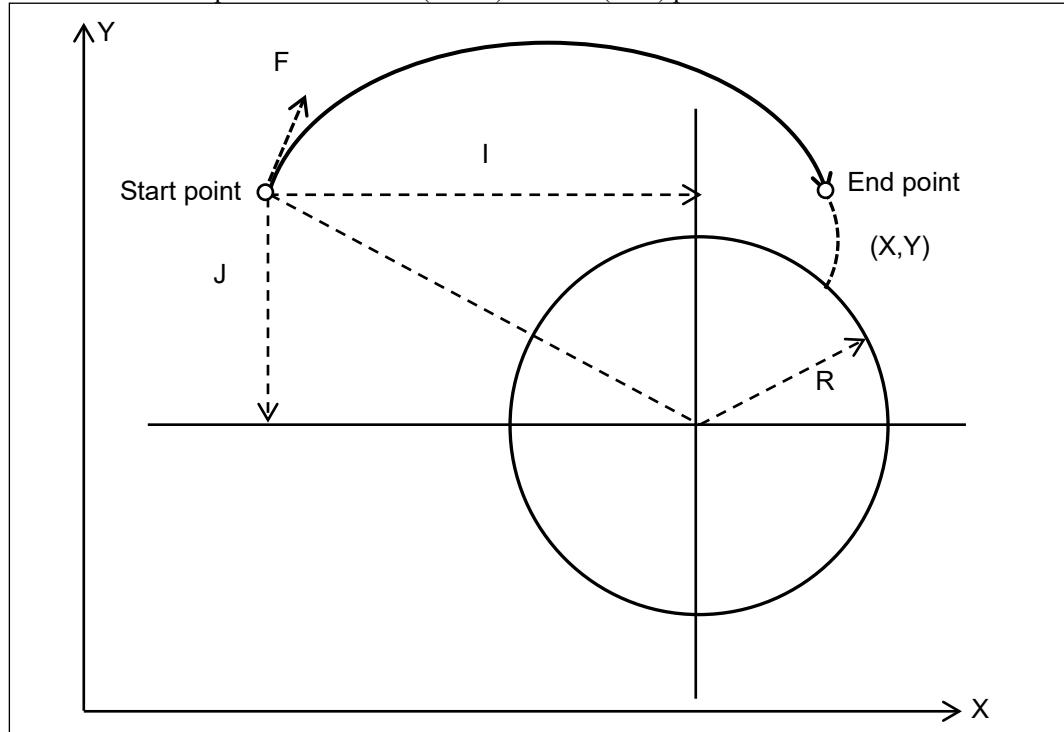
For Y-Z plane:

{G19} G02.2 Y_ Z_ X_ J_ K_ R_ F_;
{G19} G03.2 Y_ Z_ X_ J_ K_ R_ F_;

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 9 (6) Involute interpolation function” in the Operation Manual II 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.

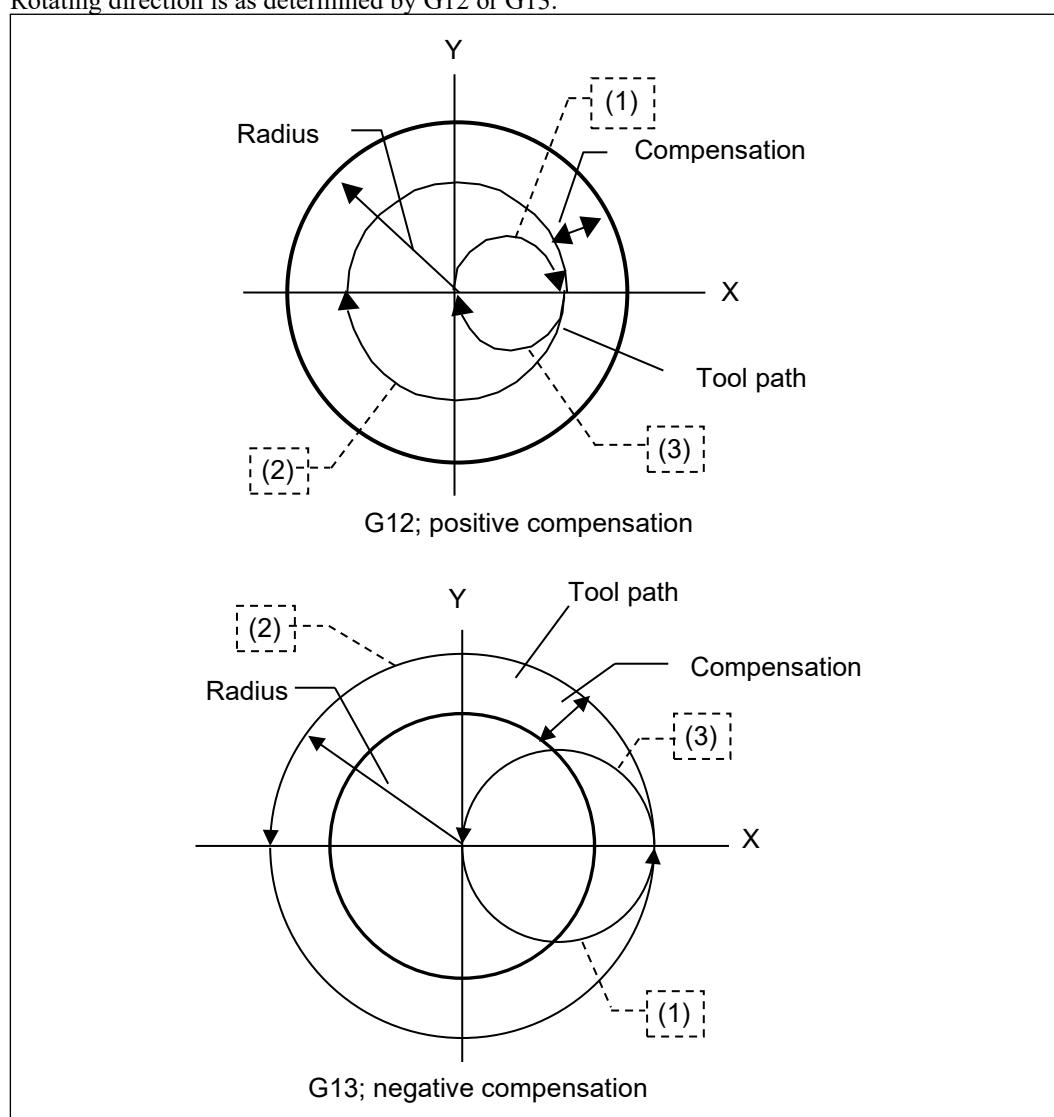
F : Cutting speed

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.



Chapter 3 Preparation Function

- (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) If a cutter compensation control command (G40, G41 and G42) (in startup and cancel mode) and circular cutting (G12 and G13) are specified at the same time, the alarm <<Simultaneous specified code cannot be used.>> is triggered.
- (NOTE 4) Corner CR may not be set in the circle cutting command and immediately preceding block command. If set, the alarm <<Address where command is not possible>> or <<Specified G code cannot be used.>> is triggered.
- (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.
- (NOTE 11) A circular cutting command cannot be issued while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

3.4 Dwell (G04)

After the in-position check at the end of the previous block, the operation proceeds to the next block either when the specified time has elapsed, or when the measuring instrument detection signal (that is enabled) turns ON. The enabled measuring instrument detection signal is determined by the value in the user parameter (switch 1: programming) <Measurement setting 1 (2/3/4)> that corresponds to the Q address. However, the parameter <Measurement setting (Q omitted)> is used when the Q address is omitted.

Command format **G04 P_ Q_;**

or **G04 X_ Q_;**

P, X : Dwell time (sec.)

Q : 1 to 4 (omission is possible)

3

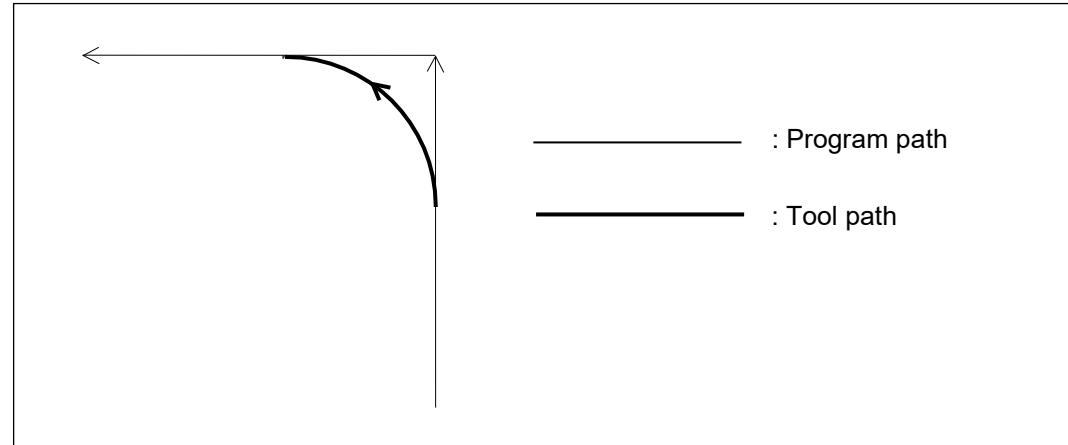
The measurement settings used with the Q addresses are as follows.

Q address command	Compatible set value
G04 Q1	<Measurement setting 1>
G04 Q2	<Measurement setting 2>
G04 Q3	<Measurement setting 3>
G04 Q4	<Measurement setting 4>
G04 (when Q is omitted)	<Measurement setting (Q omitted)>

(NOTE) When there is a command without a decimal point for the dwell time, the unit for that time is in seconds if the user parameter (switch 1: programming) <Program unit> is set to <0:Base>. The unit for the dwell time is in milliseconds if the user parameter is set to <1:Minimum>.

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.



3

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.
- (NOTE 3) The workpiece coordinate zero cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <>TCP under control<> 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, 201 to 299)
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 1) 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.
- (NOTE 2) The tool length offset data cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <>TCP under control<> is triggered.

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, 201 to 299)
R : Compensation amount

3

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 1) 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.
- (NOTE 2) The tool length wear offset cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

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 to 99, 201 to 299)
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 to 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 1) 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.
- (NOTE 2) The tool length offset (Z) cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

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 to 99, 201 to 299)
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 1) 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.
- (NOTE 2) The tool length wear offset (Z) cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

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 to 300
X, Y, Z	:	Workpiece origin offset

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).

- (NOTE 1) 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.
- (NOTE 2) The workpiece coordinate zero cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

3.6.7 Workpiece Coordinate System Using the Results of Measurements

Command format

G10 L99 Pn X_ Y_ Z_ Q_;

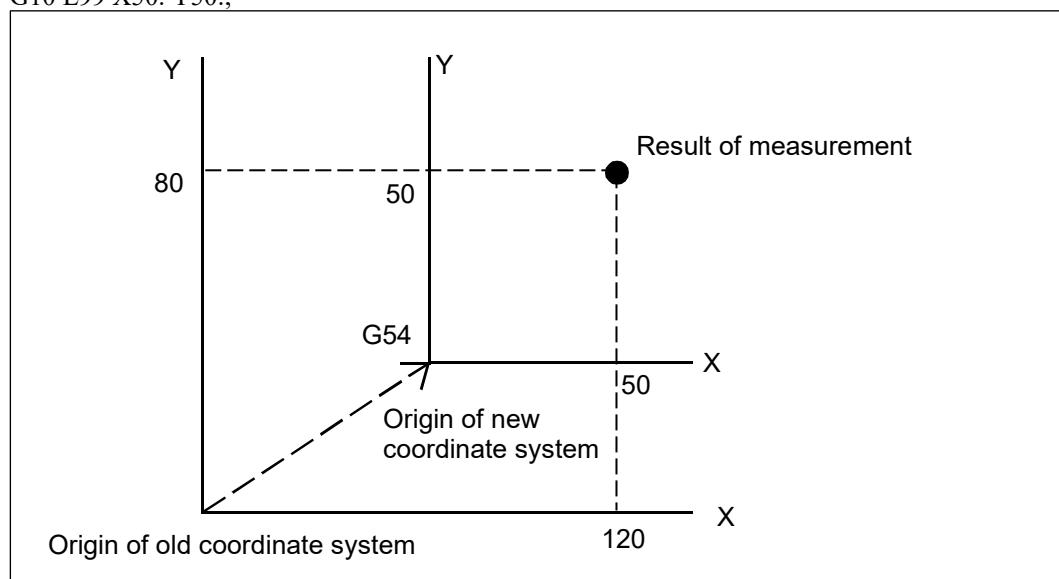
n = 1 : G54
 n = 2 : G55
 n = 3 : G56
 n = 4 : G57
 n = 5 : G58
 n = 6 : G59
 Q : Measurement result number (1 to 4)

3

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 1) 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.
- (NOTE 2) The workpiece coordinate zero cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> 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 300)
 Q : Measurement result number (1 to 4)

- (NOTE 1) 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.
- (NOTE 2) The workpiece coordinate zero cannot be changed while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> 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, 201 to 299)
Q	:	Life type
	1.	Not counted
	2	Time (minutes)
	3	Drilling (holes)
	4	Program (cycles)
	5	Time (sec.)
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 Programmable Parameter Input (G10)

3.7.1 Usage

Parameter values can be changed in a program.

Refer to “3.7.3 Parameter setting” for details on parameters that can be changed, those parameter numbers and the ranges for the set values.

Command format

G10 L52;	Parameter input mode start
P_D_;	Numerical value parameter setting
G11;	Parameter input mode end

P_ : Parameter number

D_ : Parameter set value

3

Parameter input mode start (G10 L52) and parameter input mode end (G11) cannot be combined with other commands in one block. Parameter input mode start (G10 L52) and parameter setting (P_D_) also cannot be combined in a block.

If an attempt is made to set a value that is out of range, the alarm <>Parameter setting error>> is triggered.

- (NOTE 1) A parameter can be changed even when data protection is enabled.
- (NOTE 2) It is prohibited to change the operation mode while a parameter input mode block (G10 L52 to G11) is running.
- (NOTE 3) While a parameter input mode block (G10 L52 to G11) is running, the operator message <>Programmable parameter input in progress>> appears.
- (NOTE 4) The value range of each parameter is checked on each block, and the consistency of the values between parameters is checked with G11. If there is no problem with either check, then the values are applied in G11.
- (NOTE 5) When executing both data bank editing and parameter input mode (G10 L52 to G11) at the same time, the values are reflected in the parameter after the execution operation.
- (NOTE 6) If operation ends without issuing the parameter input mode end (G11) command in MDI operation, then the operator message <>Programmable parameter input was interrupted.>> appears and the value trying to be set to the parameter will not be applied.
- (NOTE 7) If an attempt is made to overwrite a parameter while G68.2 modal (feature coordinate setting) is in progress or while G43.4/G43.5 modal (TCP control) is in progress, the alarm <>Programmable parameter setting error>> is triggered and the value trying to be set to the parameter will not be applied.
- (NOTE 8) When executing a program that uses parameter input mode during drawing before operation or tool path simulation, the value trying to be set to the parameter will not be applied.
- (NOTE 9) After the parameter input mode start (G10 L52) command is issued, if another G code command is issued thereafter that is not a parameter input mode end (G11) command, then the alarm <>Specified G code cannot be used.>> is triggered.
- (NOTE 10) When a parameter input mode end (G11) command is issued on another block that is not the parameter input mode block (G10 L52 to G11), then the alarm <>Specified G code cannot be used.>> is triggered.
- (NOTE 11) When an M code command is issued on a parameter input mode block (G10 L52 to G11), then the alarm <>Specified M code cannot be used.>> is triggered.

3.7.2 Usage Conditions

When executing programmable parameter input, the following modal status is required. This modal status applies when the power is turned ON.

Code	Function
M97	Interrupt type macro cancel

3.7.3 Parameter Setting

The parameter numbers for parameters that can be set and the ranges for the set values are noted.

3.7.3.1 User parameters

The following items are parameters that can be set.

Rotation axis/tilt axis setting

Parameter number	Item
130009	Rotation center X coordinate offset for tilt axis 1
130010	Rotation center Y coordinate offset for tilt axis 1
130011	Rotation center Z coordinate offset for tilt axis 1
130012	Rotation center X coordinate offset for rotation axis 1
130013	Rotation center Y coordinate offset for rotation axis 1
130014	Rotation center Z coordinate offset for rotation axis 1
130015	Rotation center X coordinate offset for tilt axis 2
130016	Rotation center Y coordinate offset for tilt axis 2
130017	Rotation center Z coordinate offset for tilt axis 2
130018	Rotation center X coordinate offset for rotation axis 2
130019	Rotation center Y coordinate offset for rotation axis 2
130020	Rotation center Z coordinate offset for rotation axis 2

Refer to “1.5 User parameters” in the Data Bank & Alarm Manual for details about the setting range.

3.7.4 Usage Examples

Ex: 0.5 mm is set to the user parameter (rotation axis/tilt axis setting) <Rotation center X coordinate offset for tilt axis 1> (Parameter No.130009)

G10 L52;	Parameter input mode start
P130009 D0.5;	Numerical value parameter (w/decimal) setting
or	
P130009 D500;	Numerical value parameter (no decimal) setting (NOTE)
G11;	Parameter input mode end

(NOTE) When the user parameter (switch 1: programming) <Program unit> is set to <1:Minimum>, and the minimum unit setting is type 1 (micron)

3.8 Coordinate System (G17 to 19, G52 to 59, G54.1, G92, G68.2)

3.8.1 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

3

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.

3.8.2 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.

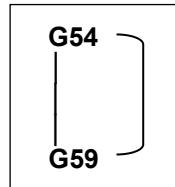
(NOTE 1) A G53 command is ignored when specified in the incremental mode (G91).

(NOTE 2) When a command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

3.8.3 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.

(NOTE) When a command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

3.8.4 Extended Workpiece Coordinate System Selection (G54.1)

Command format

G54.1 Pn;

- | | |
|------|--|
| Pn : | Code specified in extended workpiece coordinate system |
| n : | 1 to 300 |

The above command allows you to select 300 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.

(NOTE) When a command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

3.8.5 Workpiece Coordinate System Setting (G92)

Change of workpiece zero position can be commanded as follows:

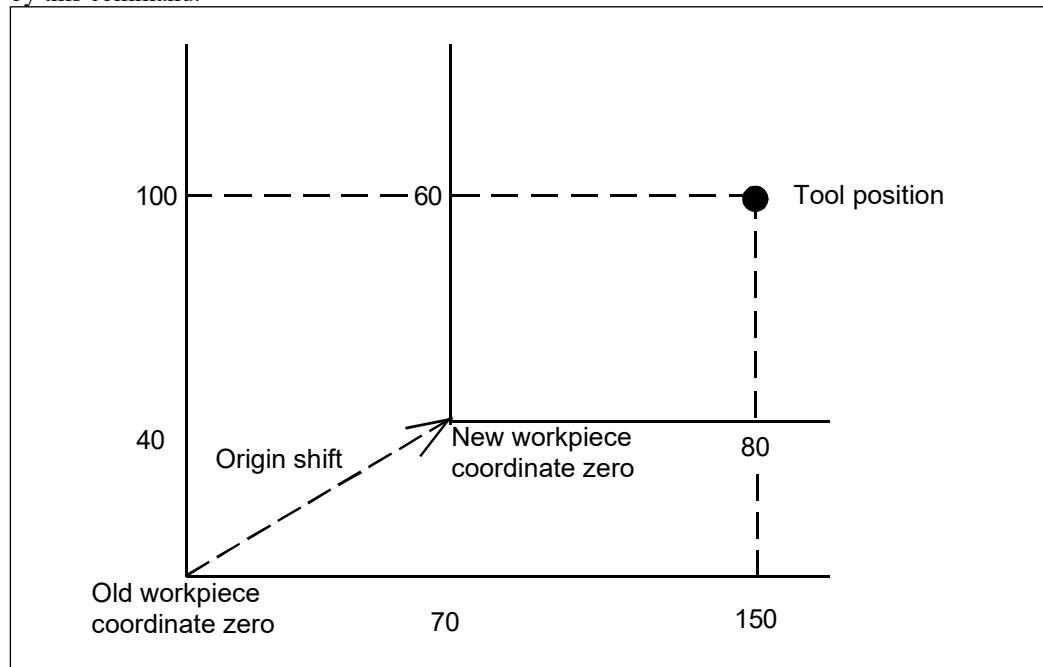
Command format

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.

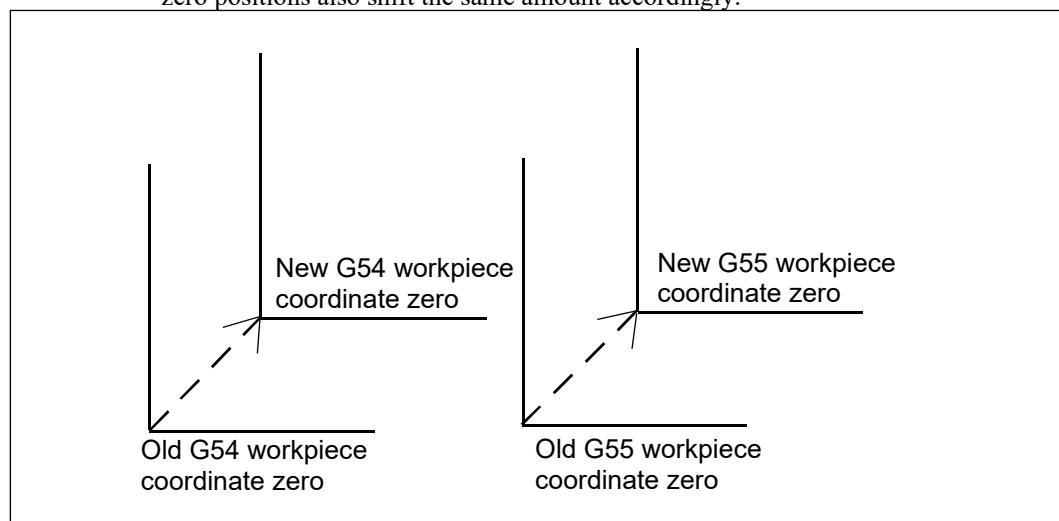
3



(NOTE 1) The commanded coordinate values are always absolute regardless of G90 and G91.

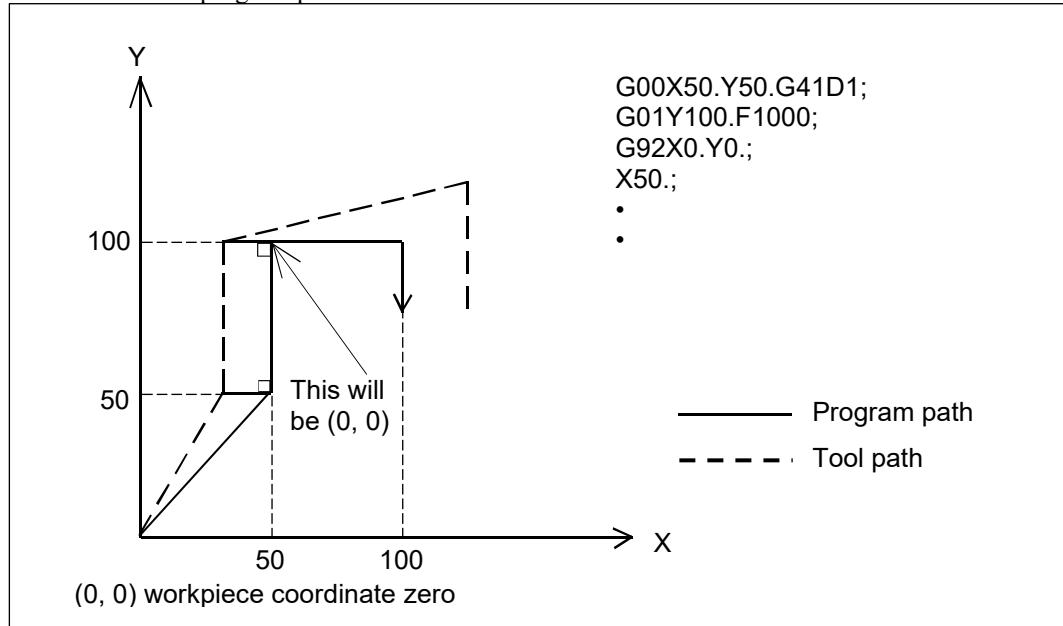
(NOTE 2) The working coordinate values of the not commanded axes do not change.

(NOTE 3) The current working zero position shifts when G92 is executed, and other working zero positions also shift the same amount accordingly.

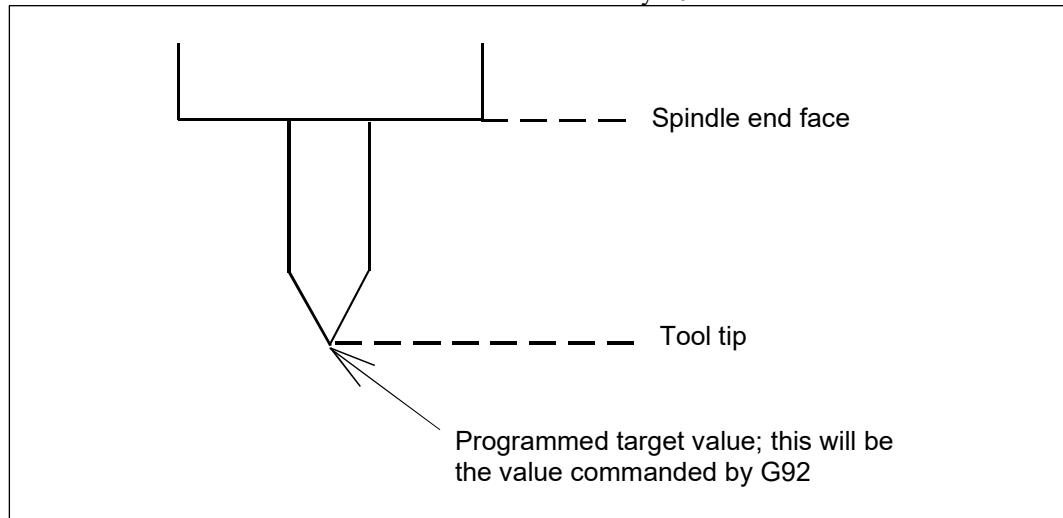


In the above figures, G92 is commanded in the coordinate system of G54. When the working zero position of G54 shifts, the other working zero positions of G55 through G59 also shift the same amount as G54.

- (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.

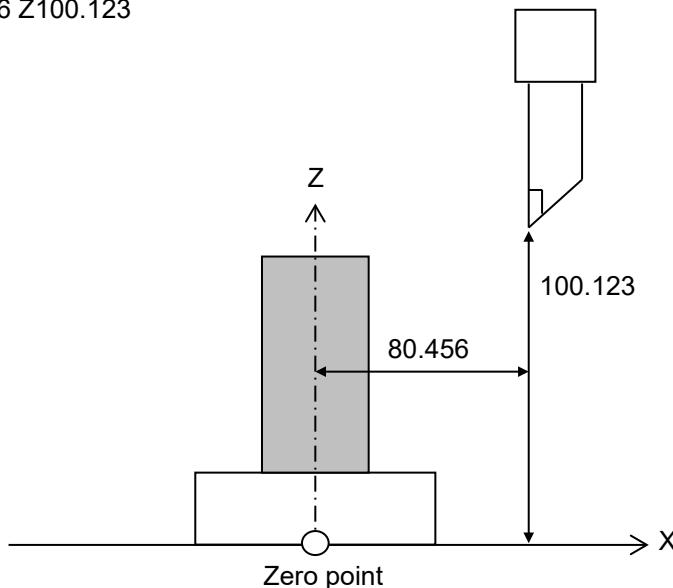


- (NOTE 5) When G92 is commanded during tool length compensation, a new workpiece coordinate system will be created where the programmed target value of Z axis coincides with the coordinates commanded by G92.



- (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.

G92 X80.456 Z100.123



3

- (NOTE 7) When the additional axis is commanded and the optional additional axis is not installed, an alarm will occur.
 (NOTE 8) When a command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

3.8.6 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

- (NOTE) When a TCP control (G43.4/G43.5) command is issued and the shift amount is set to an axis in G52, the alarm <<TCP control command error>> is triggered. In addition, when a G52 command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

3.8.7 Rotary Fixture Offset Function (G54.2)

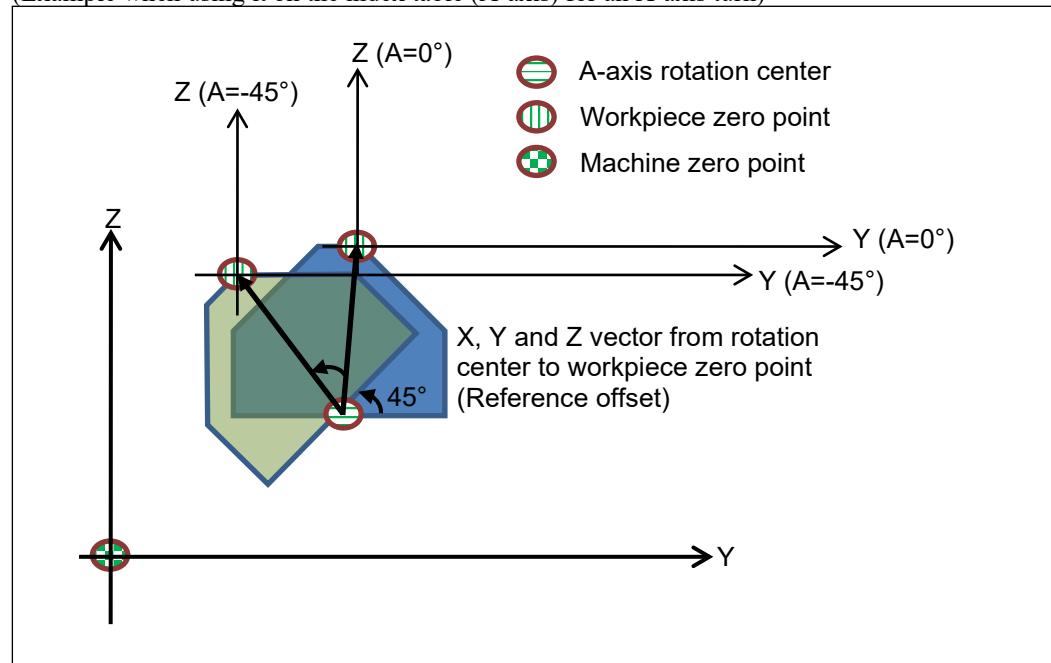
3.8.7.1 Overview

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)



3.8.7.2 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.

3

- (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: <<Mirror image mode>>)
 - When the G43.4/G43.5 modal is operating (alarm: <<TCP under control>>)
- (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
 - G43.4/G43.5
- (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 coordinate 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
```

3.8.8 Feature Coordinate System (G68.2)

3.8.8.1 Overview

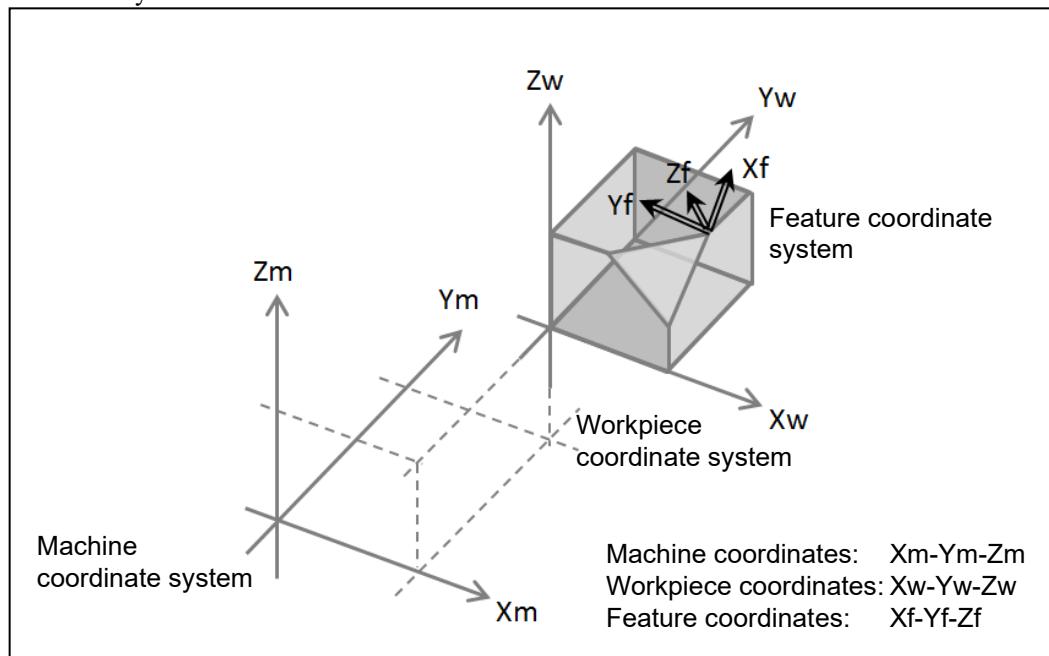
When machining on a tilt plane, set a new coordinate system on this plane to issue position commands in order to make the program simpler.

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

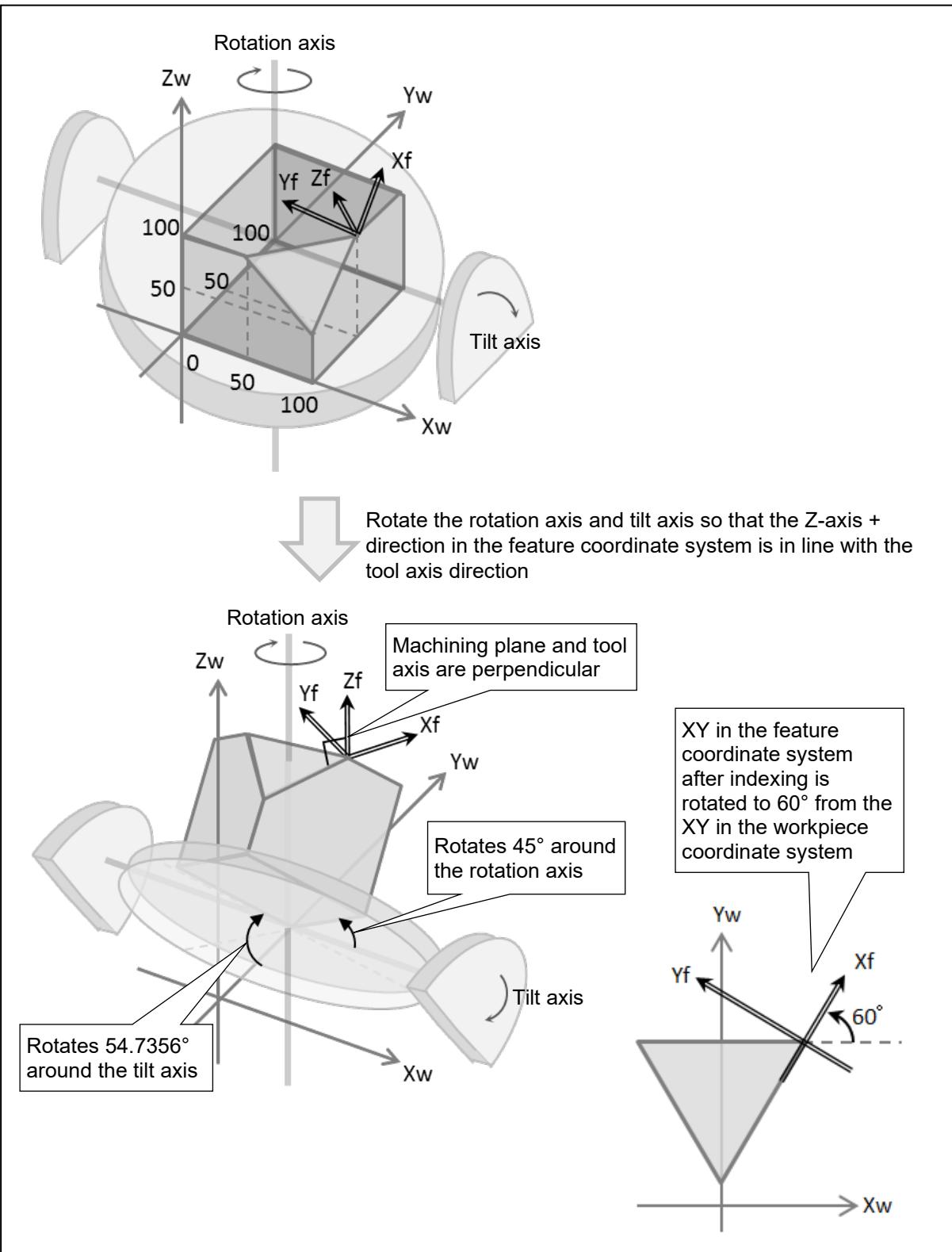
When the feature coordinate system is set, this is called feature coordinate manufacturing mode. The commands are controlled as commands in the feature coordinate system until the feature coordinate manufacturing mode is cancelled.

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



3



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.

3.8.8.2 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 XxYyZzI α J β K γ ;

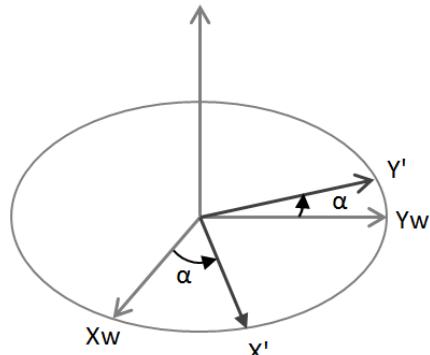
x, y, z: Feature coordinate zero point

α, β, γ : Euler angles

Specify within the range: -360.000 to 360.000.

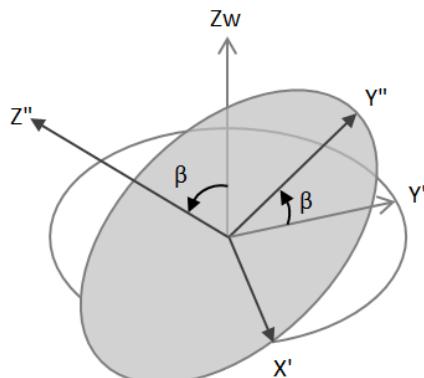
When one of the following is omitted: X, Y, Z, I, J or K, the command is processed as a 0 value. The alarm <<Feature coordinate command error>> is triggered when I, J and K are all omitted.

Zw

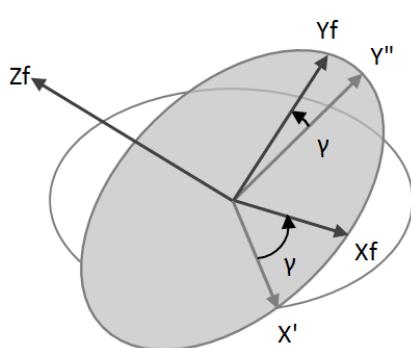


The Euler angles refer to a set of angles (α, β and γ) that rotates around each axis, in order to acquire the feature coordinate system (X_f, Y_f and Z_f) from (X_w, Y_w and Z_w) in the workpiece coordinate system.

1. Rotates (X_w, Y_w and Z_w) for the angle α around the Z_w -axis forming (X', Y' and Z_w).



2. Rotates (X', Y' and Z_w) for the angle β around the X' -axis forming (X', Y'' and Z'').



3. Rotates (X', Y'' and Z'') for the angle γ around the Z'' -axis forming (X_f, Y_f and Z_f).

(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) If the feature coordinate setting command is issued in one of the following situations, then the alarm <>Rotation axis/Tilt axis parameter setting error<> is triggered.
 - When the <Tilt axis *> or <Rotation axis *> in the user parameter (rotation axis/tilt axis setting) is set to an axis where the machine parameter (system 2: additional axis) <Optional *-axis> was set to <No>.
 - When the user parameter (rotation axis/tilt axis setting) <Forward direction for the coordinate system on tilt axis 1(2)> is not set to <1:Z->Y>, when it is not set to <3:X->Z>, or when <Forward direction for the coordinate system on rotation axis 1 (2)> is not set to <5:Y->X>.
- (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).
- (viii) When a command is issued during TCP control (G43.4/G43.5), the alarm <>TCP under control<> is triggered.

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 G/M code command is issued on the same block, the alarm <<Simultaneous specified code cannot be used.>> 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.

3

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: stroke) <*-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.

Chapter 3 Preparation Function

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)

3

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.

3.8.8.3 Restrictions

1. Simultaneous use with other functions
 - 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
 - TCP control (G43.4/G43.5) command is not possible while in feature coordinate manufacturing mode. Otherwise, the alarm <>Feature coordinate manufacturing mode engaged>> is triggered.

Refer to “3.17 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.

3.8.9 Involute Interpolation Function

3.8.9.1 Overview

Introduction

This function enables the user to machine along an involute curve.

By using the involute interpolation, the user does not need to approximate minute or fine straight lines and arcs for an involute curve but can machine the curve with good accuracy.

In addition, the involute interpolation override function and high accuracy mode can be used to machine with even better accuracy.

Usage Conditions

The involute interpolation function option is required to enable and use this function.
This function can only be used in NC language mode.

3

3.8.9.2 Command Format

Command format

For X-Y plane:

{G17} G02.2 X_Y_I_J_R_F_;
{G17} G03.2 X_Y_I_J_R_F_;

For Z-X plane:

{G18} G02.2 Z_X_K_I_R_F_;
{G18} G03.2 Z_X_K_I_R_F_;

For Y-Z plane:

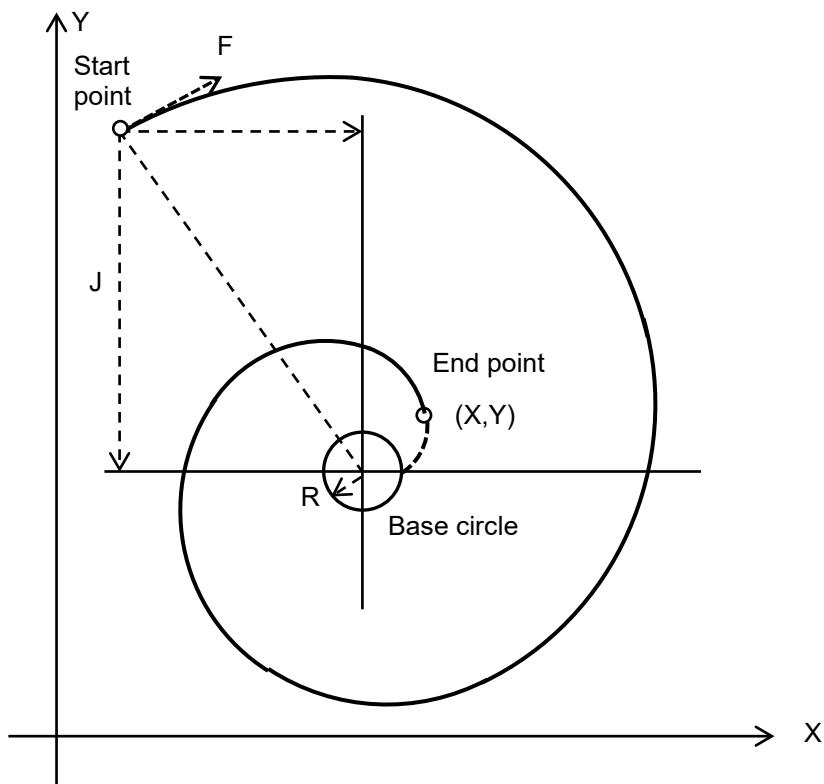
{G19} G02.2 Y_Z_J_K_R_F_;
{G19} G03.2 Y_Z_J_K_R_F_;

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		I	Distance in X-axis direction between the start and center points	
		J	Distance in Y-axis direction between the start and center points	
		K	Distance in Z-axis direction between the start and center points	
Base circle radius		R	Radius of base circle	
Feedrate		F	Speed along tangent line of involute curve	

On an XY plane, clockwise and counterclockwise refer to the rotation directions when looking at the negative direction from the positive direction on the Z-axis.

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



Start Point and End Point

The start point on an involute curve is the current position.

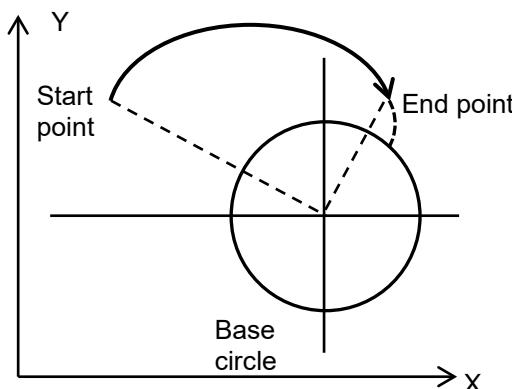
The end point on an involute curve follows the command that is based on the address X, Y and Z. When an end point command is not carried out, or when an axis that is not on the selected plane is used in the command, then the alarm <<Involute interpolation command error>> is triggered.

When the start point or end point on an involute curve is on the inside of the base circle, the alarm <<Involute interpolation command error>> is triggered because an involute curve is not required. The same alarm is triggered when the start and end points are inside the base circle after being corrected or compensated due to cutter compensation or nose R compensation.

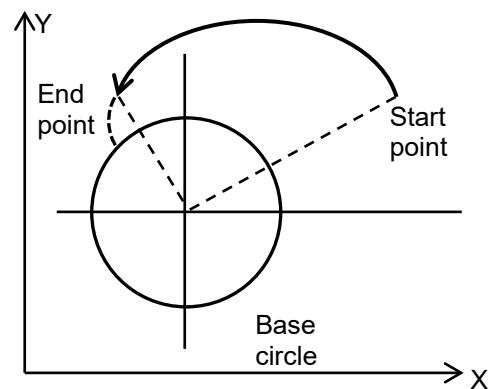
Depending on the relative distance between the center of the base circle, the start point and the end point, the involute curve will either curve toward the base circle or away from it.

The involute curve will curve toward the base circle when the end point is closer to the center of the base circle than the start point.

When using G02.2



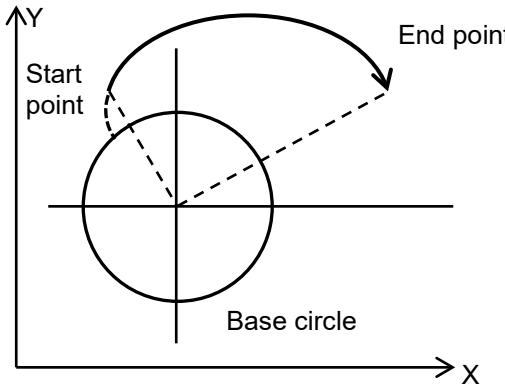
When using G03.2



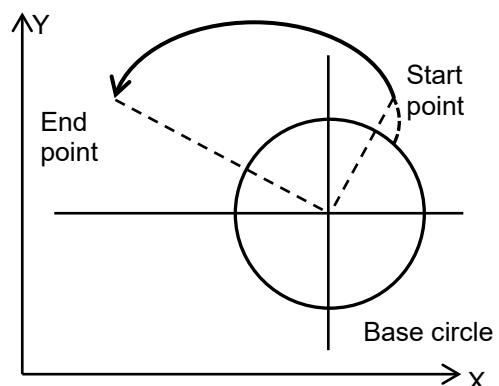
Chapter 3 Preparation Function

The involute curve will curve away from the base circle when the end point is further away from the center of the base circle than the start point.

When using G02.2



When using G03.2



3

When the start point and end point are equidistant from the center of the base circle, the alarm <>Involute interpolation command error>> is triggered because it cannot decide on a direction for the involute curve.

Base Circle Center Coordinate

I, J and K are used to issue a command for the center coordinate of the base circle. The command value is always an increment from the start point regardless of G90/G91. The alarm <>Involute interpolation command error>> is triggered in the following situations.

- When there is no I, J or K command
- When the command values for I, J and K are all zero (0)
- When a command was issued for an axis that is not inside the selected plane

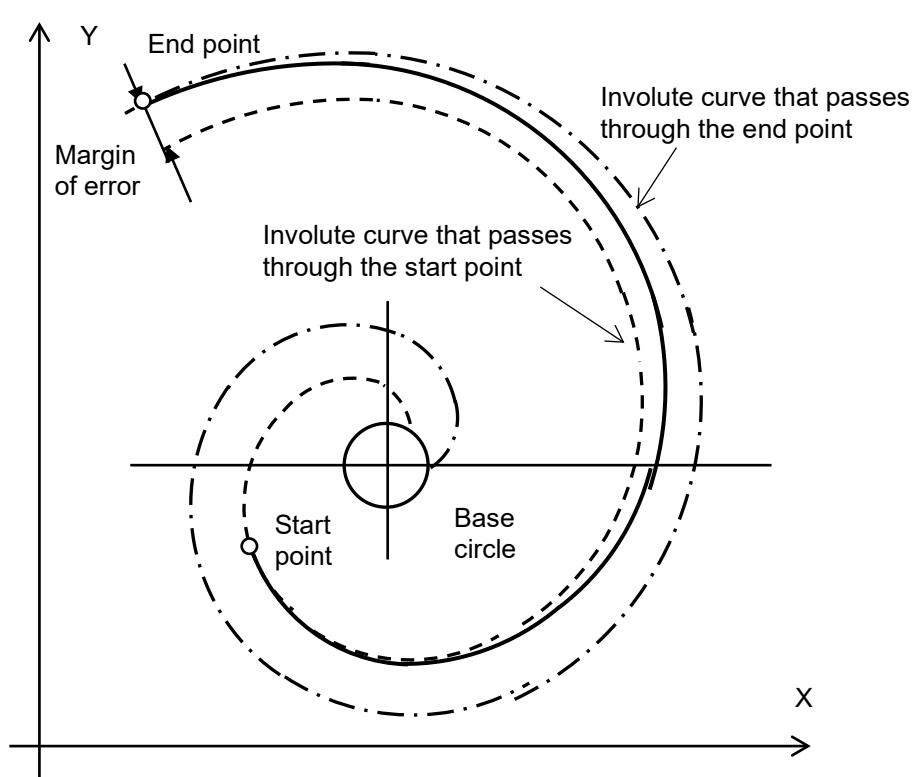
Base Circle Radius

The radius of the base circle uses a positive value in the R address to issue a command. The alarm <>Involute interpolation command error>> is triggered when the command value is less than zero (0).

3.8.9.3 Margin of Error for End Point on Involute Curve

When there is no end point for a command on the involute curve that passes through the start point, the following operations apply.

- When the margin of error, between the involute curve that passes through the start point and the end position of the involute curve that passes through the end point, exceeds the user parameter (switch 1: compensation function) <Allowable margin of error for involute interpolation>, then the curve leads toward the command end point.
- When the margin of error, between the involute curve that passes through the start point and the end position of the involute curve that passes through the end point, exceeds the user parameter (switch 1: compensation function) <Involute interpolation error limit>, then the alarm <<Involute interpolation error limit exceeded>> is triggered.



3.8.9.4 Involute Interpolation Command During Cutter Compensation

An involute interpolation command can be issued during offset mode for cutter compensation (G41/G42) and nose R compensation (G141/G142). The involute interpolation during cutter compensation uses an imaginary circle for the start point and end point and carries out cutter compensation for that imaginary circle. Then, this function interpolates on the involute compensation curve relative to the results of the cutter compensation.

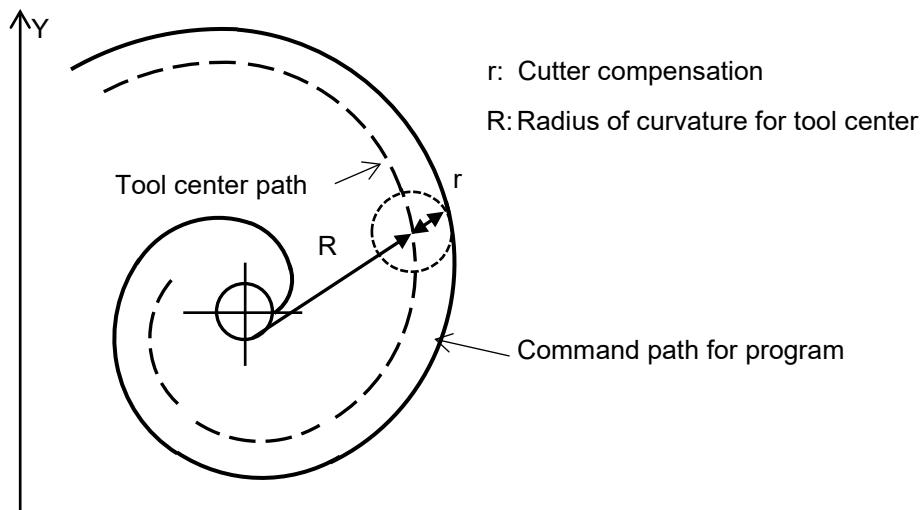
- (NOTE 1) When an involute interpolation command is issued in startup and cancel mode, the alarm <<Cutter compensation error>> is triggered.
- (NOTE 2) When the next block for the involute interpolation is a cutter compensation cancel command, the position that makes up a vertical vector to the end point of the involute interpolation becomes the end point.

3

Involute Interpolation Override

The radius of curvature is not fixed for an involute curve. Therefore, even if the speed for the tool center is constant or fixed, the speed on the cutting plane will not be constant and changes depending on the radius of curvature. In particular, the radius of curvature is small close to the base circle, and therefore that change becomes bigger.

The involute interpolation override function adjusts the actual feedrate to a speed that matches the radius of curvature, so that the speed on the cutting plane becomes the command value.



Cutter compensation	Override value	Actual feedrate
Inside workpiece	$R / (R + r)$	Command speed × Override value
Outside of workpiece	$R / (R - r)$	

- (NOTE 1) When cutting the inside of the workpiece in cutter compensation mode, the override value may become extremely small as it gets closer to the base circle. The lower limit can be set in the user parameter (switch 1: compensation function) <Involute interpolation override limit>. If the calculated override value is less than the <Involute interpolation override limit>, the override value is calculated using the <Involute interpolation override limit> value.

$$\text{Actual feedrate} = \text{Command speed} \times \frac{\text{Involute interpolation override limit}}{100}$$

- (NOTE 2) When cutting the outside of the workpiece in cutter compensation mode, the actual feedrate is greater than the command speed. When the feedrate for each axis exceeds the following parameters, the speed is clamped at the following values.
- User parameter (switch 1: programming) <Max. actual cutting travel speed (linear axis / rotation axis)>
 - Machine parameter (system 1: X-, Y- and Z-axes) <Maximum cutting travel speed> (X- to Z-axis)
 - Machine parameter (high accuracy: X-, Y- and Z-axes) <Max. feedrate A>
 - Machine parameter (high accuracy: X-, Y- and Z-axes) <Max. feedrate B> (X- to Z-axes)
- (NOTE 3) When the user parameter (switch 1: compensation function) <Involute interpolation override limit> is zero (0), the involute interpolation override function is disabled. The override value is fixed at 100% and it always operates at the command speed.

3.8.9.5 Inolute Interpolation Command in High Accuracy Mode

An involute interpolation command is possible in high accuracy mode (high accuracy A / high accuracy B / quick setting for high accuracy). The user can machine with even better accuracy by using the <Automatic arc deceleration function> in high accuracy mode.

- (NOTE 1) An involute interpolation command is not possible with the fully automatic deceleration function in high accuracy mode A, nor is it possible in accuracy specification mode in the quick setting for high accuracy. When there is a command, the alarm <<High accuracy A invalid command>> and <<Invalid command in accuracy specification mode>> are triggered.
- (NOTE 2) Smooth path offset function is cancelled when there is an involute interpolation command.

3.8.9.6 Restrictions

- A command is only possible when the start and end points for the involute interpolation command are less than 100 rotations from the starting point of the involute curve. When the command is more than 100 rotations, the alarm <<Inolute interpolation command error>> is triggered.
- When the corner CR is specified in the preceding block, the alarm <<Specified G code cannot be used.>> is triggered.
- The corner CR cannot be specified for the involute interpolation. When a command is issued, the alarm <<Address where command is not possible>> is triggered.
- An involute interpolation command is not possible when the mirror image is enabled (G51.1). When a command is issued, the alarm <<Mirror image mode>> is triggered.
- An involute interpolation command is not possible when the scaling is enabled (G51). When a command is issued, the alarm <<Scaling>> is triggered.
- After the feature coordinate setting, an involute interpolation command cannot be issued before the feature coordinate index. When a command is issued, the alarm <<Feature coordinate manufacturing mode engaged>> is triggered.
- An involute 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.
- An involute interpolation command cannot be issued when the TCP control (G43.4/G43.5) modal is in use. Otherwise, the alarm <<TCP under control>> is triggered.
- When the following command is issued during the involute interpolation modal, the alarm <<Invalid command for involute interpolation modal>> is triggered.
 - Tool length offset (G43 and G44), tool position compensation (G143 and G144) and TCP control (G43.4 and G43.5)
 - Coordinate calculation function (G36 to G39)
 - Scaling (G51) and mirror image (G51.1)
 - Automatic workpiece measurement (G121 to G129)
 - Inverse time feed (G93)

3.9 Soft Limit

Tool operation range is set by software.

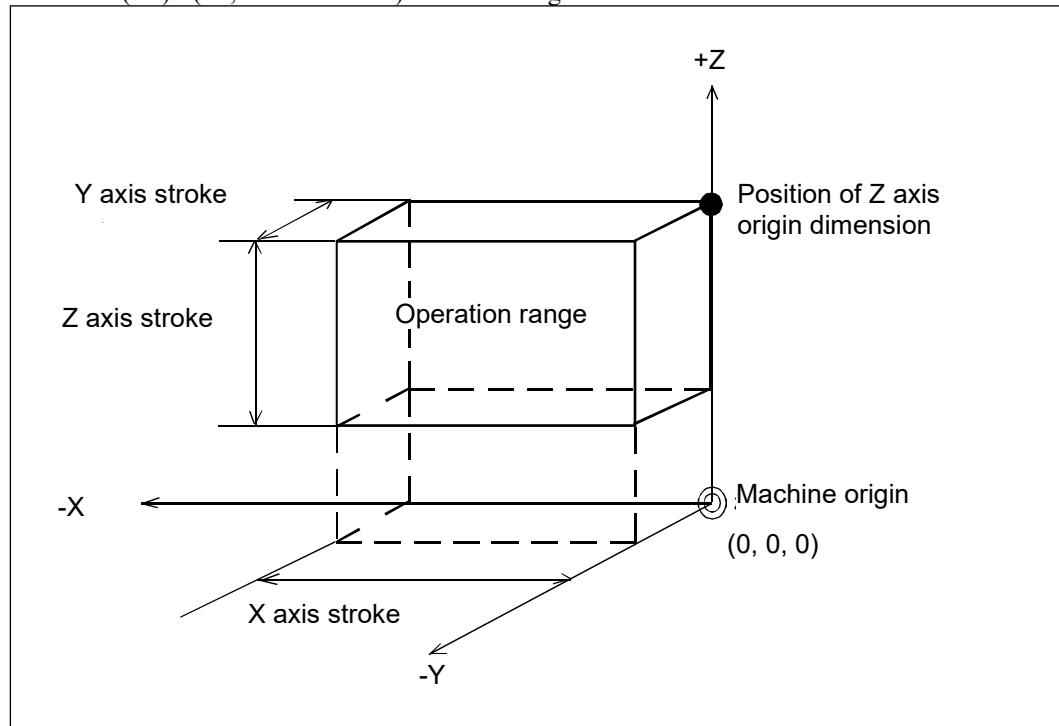
Tool operation range is specified by one of the three methods below.

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

3.9.1 Stroke

The maximum machine stroke is set using the machine parameter (system 1: X-, Y- and Z-axes) <Stroke 1 (+/-)> (X-, Y- and Z-axes). Do not change it.

3



(NOTE) The distance to the Z-axis zero point is set in the machine parameter (system 1: X-, Y- and Z-axes) <Distance to zero point> (Z-axis).

3.9.2 Stroke Limit

The operation range for X-, Y- and Z-axes is set in the user parameter (switch 2: stroke) <Stroke limit 1 (+/-)> (X/Y/Z-axes).

3.9.3 Programmable Stroke Limit (G22)

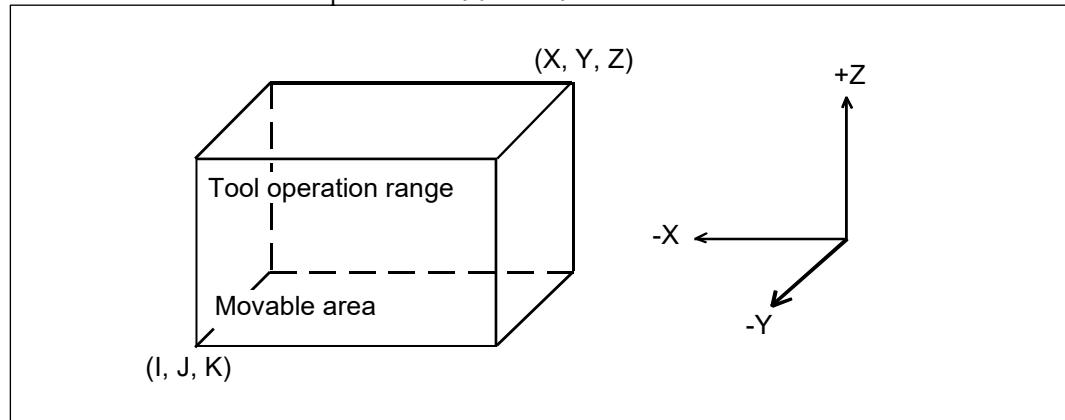
Tool operation range is commanded by program.

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.

3.10 Reference Position (G28 to G30)

3.10.1 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.

3

- (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) The reference position is set in the user parameter (switch 3) <Reference position *-axis> (X- to Z- / 5- to 8-axes).
- (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.
Refer to “4.2 Tool length offset (G43, G44 and G49)” and “4.4 Tool position compensation (G143, G144 and G49 - Option)” for further details.
- (NOTE 8) While the Z-axis perimeter mode is on (M300), the Z-axis perimeter operation is carried out when there are consecutive operations for the Z-axis up positioning and the X-axis or Y-axis positioning operations. Refer to “Chapter 12 M function” for further details on the Z-axis perimeter operation.
- (NOTE 9) When a command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

3.10.2 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.
- (NOTE 6) While the Z-axis perimeter mode is on (M300), the Z-axis perimeter operation is carried out when there are consecutive operations for the Z-axis up positioning and the X-axis or Y-axis positioning operations. Refer to “Chapter 12 M function” for further details on the Z-axis perimeter operation.
- (NOTE 7) When a command is issued during TCP control (G43.4/G43.5), the alarm <>TCP under control>> is triggered.

3

3.10.3 Return to the 2nd to 6th Reference Point (G30)

Command format

G30 P_ X_ Y_ Z_ A_ B_ C_;

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

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

- (NOTE 1) The 2nd to the 6th reference positions are set in the user parameter (switch 3) <2nd (to 6th) reference position *-axis> (X- to Z- / 5- to 8-axes).
- (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.
- (NOTE 4) While the Z-axis perimeter mode is on (M300), the Z-axis perimeter operation is carried out when there are consecutive operations for the Z-axis up positioning and the X-axis or Y-axis positioning operations. Refer to “Chapter 12 M function” for further details on the Z-axis perimeter operation.(NOTE 5) When a command is issued during TCP control (G43.4/G43.5), the alarm <>TCP under control>> is triggered.

3.11 Skip Function (G31, G131/G132)

3.11.1 Before using the skip function

The user parameters (switch 1: programming) <Measurement setting 1 (2/3/4)> and <Measurement setting 90> are used with the skip function, the multiple skip function and the continuous skip function. The detector or detecting instrument that is enabled for each function is set in the user parameter.

The user parameters that correspond to each function are as follows.

Refer to “1.5 User parameters” in the Data Bank & Alarm Manual for further details on each measurement setting.

3

		Compatible set value
Skip function	G31/131/132	<Measurement setting 1>
Multiple skip function	G31/131/132 P1	<Measurement setting 1>
	G31/131/132 P2	<Measurement setting 2>
	G31/131/132 P3	<Measurement setting 3>
	G31/131/132 P4	<Measurement setting 4>
Continuous skip function	G31/131/132 P90	<Measurement setting 90>

3.11.2 Skip Function (G31, G131/G132)

Linear travel is carried out using the command feedrate until the end point of the current position or until the measuring instrument detection signal (that is enabled) turns ON. The enabled measuring instrument detection signal uses the set value in the user parameter (switch 1: programming) <Measurement setting 1>.

Command format

```
G31 X_Y_Z_A_B_C_F_;  
G131 X_Y_Z_A_B_C_F_;  
G132 X_Y_Z_A_B_C_F_;
```

Simultaneous axis commands that can be used and the feedrate F setting are the same as in linear interpolation (G01).

For G131, the alarm <<Detection signal off>> is triggered if travel is carried out to the end point without the measuring instrument detection signal (that is enabled) turning ON. For G31, G132, an alarm does not occur.

When using G31, after the valid measuring instrument detection signal turns ON, axis travel decelerates and then stops. When using G131/132, after the signal turns ON, axis travel decelerates and stops. Then, it returns back to the same coordinate where the signal turned ON.

The coordinate values where the detection signal turns ON are stored in the macro system variables (#5061 to #5068, #5071 to #5074).

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.

- (NOTICE 1) The appropriate feedrate varies depending on the probe being used. Therefore, contact the probe manufacturer when deciding on the rate.
- (NOTICE 2) 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.
- (NOTICE 3) 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 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 measuring instrument detection signal is already ON, the motion is not carried out.

- (NOTE 5) While in the inverse time feed (G93) modal, the alarm <>Command not possible during inverse time feed>> is triggered.
- (NOTE 6) Only X-, Y- and Z-axes commands are possible while under TCP control. If an additional axis command is issued, the alarm <>Address where command is not possible>> is triggered.

3.11.3 Multiple Skip Function (G31, G131/132)

Linear travel is carried out using the command feedrate until the end point of the current position or until the measuring instrument detection signal (that is enabled) turns ON. The enabled measuring instrument detection signal uses the set value in the user parameter (switch 1: programming) <Measurement setting 1 (2/3/4)> that corresponds to the P address.

Command format

```
G31 P_X_Y_Z_A_B_C_F;
G131 P_X_Y_Z_A_B_C_F;
G132 P_X_Y_Z_A_B_C_F;
```

P : 1~4

Refer to “3.11.1 Before using the skip function” for details related to the P address value and the <Measurement setting 1 (2/3/4)>.

The motions (excluding conditions when the signal turns ON) are the same as the skip function (G31, G131/132).

- (NOTICE 1) The appropriate feedrate varies depending on the probe being used. Therefore, contact the probe manufacturer when deciding on the rate.
- (NOTICE 2) The “Coordinate when the measuring instrument 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.
- (NOTICE 3) 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 1) The alarm <>Compensating diameter>> is triggered when cutter compensation mode is enabled.
- (NOTE 2) The axes do not travel during a dry run.
- (NOTE 3) The axes travel up to the end point when the machine lock is engaged.
- (NOTE 4) When the measuring instrument detection signal is already ON, the motion is not carried out.
- (NOTE 5) While in the inverse time feed (G93) modal, the alarm <>Command not possible during inverse time feed>> is triggered.
- (NOTE 6) Only X-, Y- and Z-axes commands are possible while under TCP control. If an additional axis command is issued, the alarm <>Address where command is not possible>> is triggered.

3.11.4 Continuous Skip Function (G31)

The tool moves linearly (linear interpolation) at the specified feedrate from the current position to the target position. During this time, when the enabled measuring instrument detection signal is turned on, the coordinate at that time is saved as a macro system variable (#5061 to #5068, and #5071 to #5074).

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.

The enabled measuring instrument detection signal uses the set value in the user parameter (switch 1: programming) <Measurement setting 90>.

Command format

G31 P90	X_F_;
G31 P90	Y_F_;
G31 P90	Z_F_;

3

- (NOTICE 1) The appropriate feedrate varies depending on the probe being used. Therefore, contact the probe manufacturer when deciding on the rate.
- (NOTICE 2) 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.
- (NOTICE 3) 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 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) While in the inverse time feed (G93) modal, the alarm <<Command not possible during inverse time feed>> is triggered.
- (NOTE 4) Only X-, Y- and Z-axes commands are possible while under TCP control. If an additional axis command is issued, the alarm <<Address where command is not possible>> is triggered.

3.12 Scaling (G50/G51)

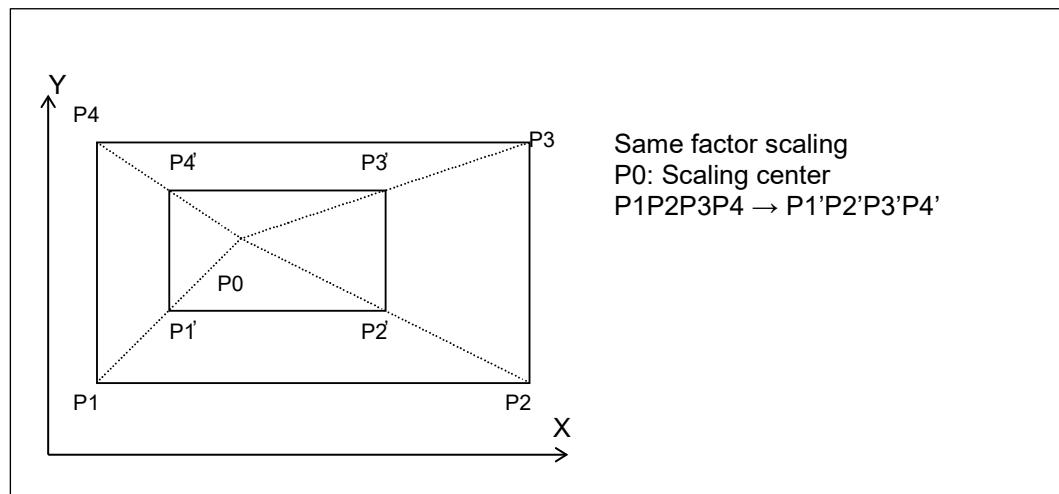
The programmed shape can be enlarged or reduced by the desired scaling factor. Scaling is possible using the same ratio for all axes or a different ratio for each axis.

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

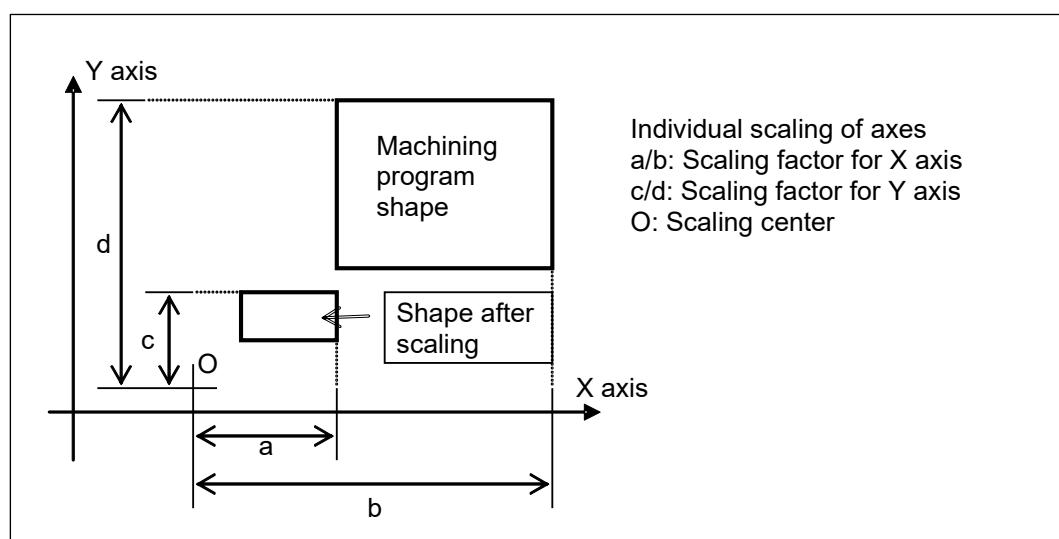


Scaling using a different ratio for each axis

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 ±1 to ±999999.
Accordingly, the valid scaling range is ±0.001 to ±999.999 or ±0.00001 to ±9.99999.

Precautions for use of scaling function:

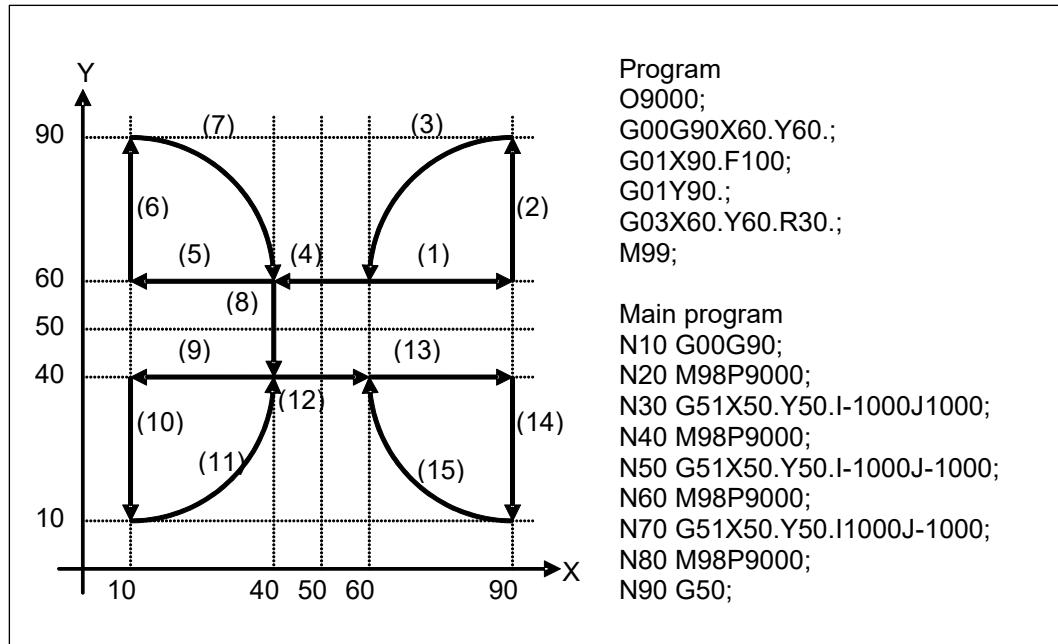
- (NOTE 1) Issue a scaling command independently on the G51 block. If another G/M code is issued at the same time, the alarm <>Simultaneous specified code cannot be used.>> is triggered.
- (NOTE 2) When scaling center coordinates (XYZ) is omitted, tool position at G51 command is used as the center coordinates.
- (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.;	=	G90 G00 X0.Y100.;
G51 X0.Y0.Z0.I2000J1000;		
G02 X100.Y0.R100.F500;		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.;	=	G90 G00 X0.Y100.;
G51 X0.Y0.I2000J1000;		
G02 100.Y0.I0.J-100.F500;		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), 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
 - TCP control (G43.4/G43.5)
 2. If a scaling command is issued on an axis where the user parameter (switch 1: programming) <X/Y/Z-axis scaling> is set to <0:Invalid>, then the alarm <>Scaling address error>> is triggered.
 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 <>Feature coordinate manufacturing mode engaged>> is triggered when a scaling command is issued while in feature coordinate manufacturing mode (G68.2 modal in progress).
 5. When scaling is specified in MDI operation, the alarm <>Specified G code cannot be used>> is triggered.
 6. When a scaling command is issued during TCP control (G43.4/G43.5), the alarm <>TCP under control>> is triggered.
- (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.

3.13 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.

3

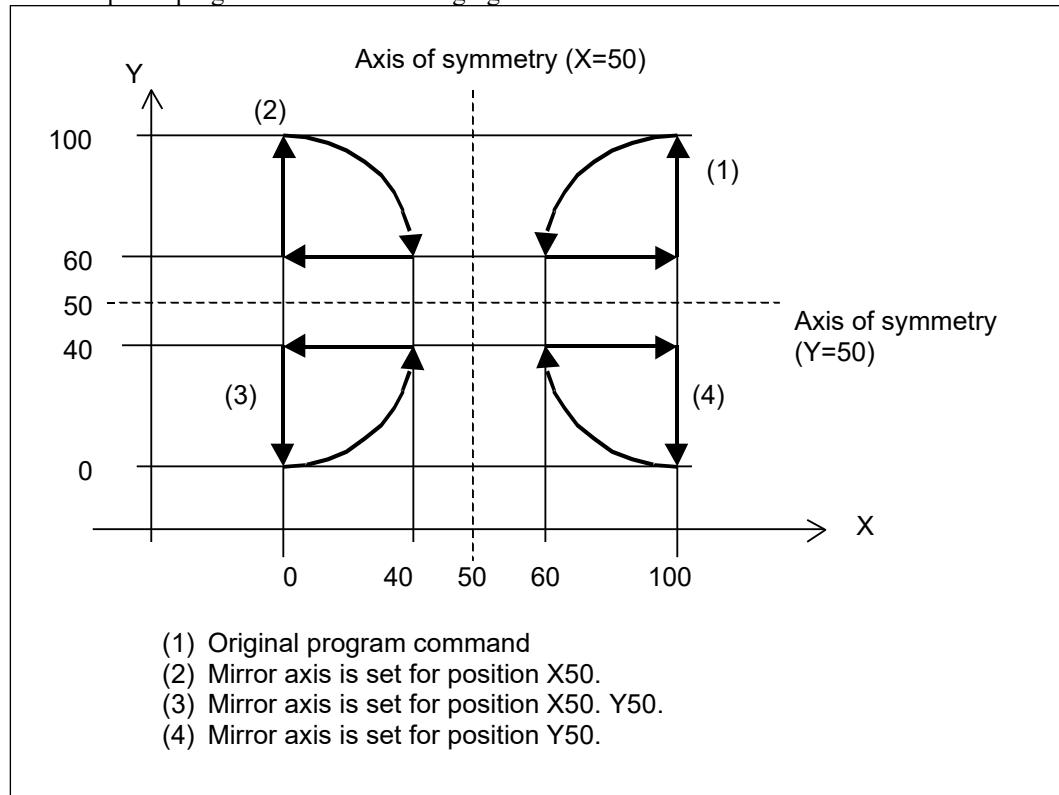
Using G50.1 command is valid while setting a mirror image.

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



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
 - TCP control (G43.4/G43.5)
 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 <>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).
 4. When a mirror command is issued in MDI operation, the alarm <>Specified G code cannot be used>> is triggered.
 5. When a mirror command is issued during TCP control (G43.4/G43.5), the alarm <>TCP under control>> is triggered.
- (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.

3.14 Rotational Transformation Function (G68/69, 168)

3.14.1 Coordinate Rotation (G68/69)

The shape specified in the program is rotated.

Coordinate rotation function

Command format

G17 G18 G19	G68 α_β_R_;
-------------------	-------------

Coordinate rotation function cancel

Command format

G69;

α, β : Rotation center coordinates

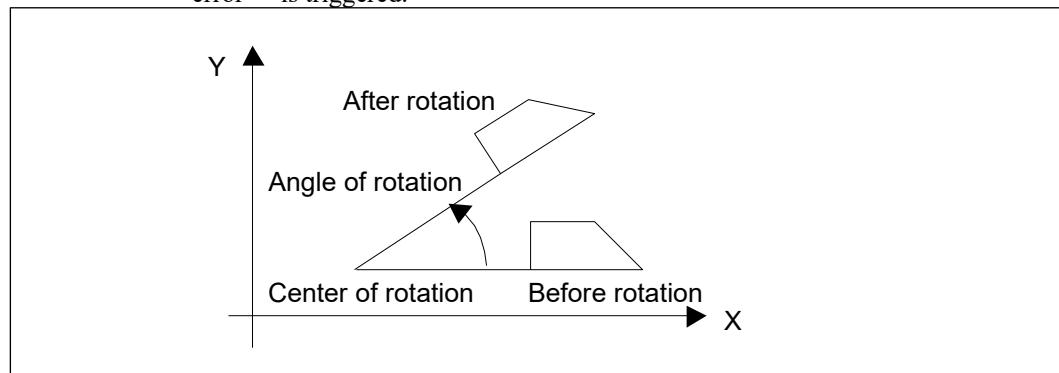
Recognize coordinates consistently that commanded absolute value.

When omit it, position G69 to G68 is a center.

R : Rotation angle (based on CCW)

Selectable between -360.000 and 360.000.

Command "R" cannot be omitted. When omitted, the alarm <<Command address error>> is triggered.



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	α	β
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 α 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
 - When a XZ circular interpolation (G102 and G103) command is issued
 - When a YZ circular interpolation (G202 and G203) command is issued
 - TCP control (G43.4/G43.5)
 2. If the axis specified for the rotational transformation center and plane selection do not match, the alarm <<Plane selection error>> is triggered.
 3. 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).
 4. If a rotational transformation command is carried out during MDI operation, the alarm <<Specified G code cannot be used>> is triggered.
 5. When a rotational transformation is carried out during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.
- (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.

3.14.2 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.

3.15 Absolute Command and Incremental Command (G90/91)

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.

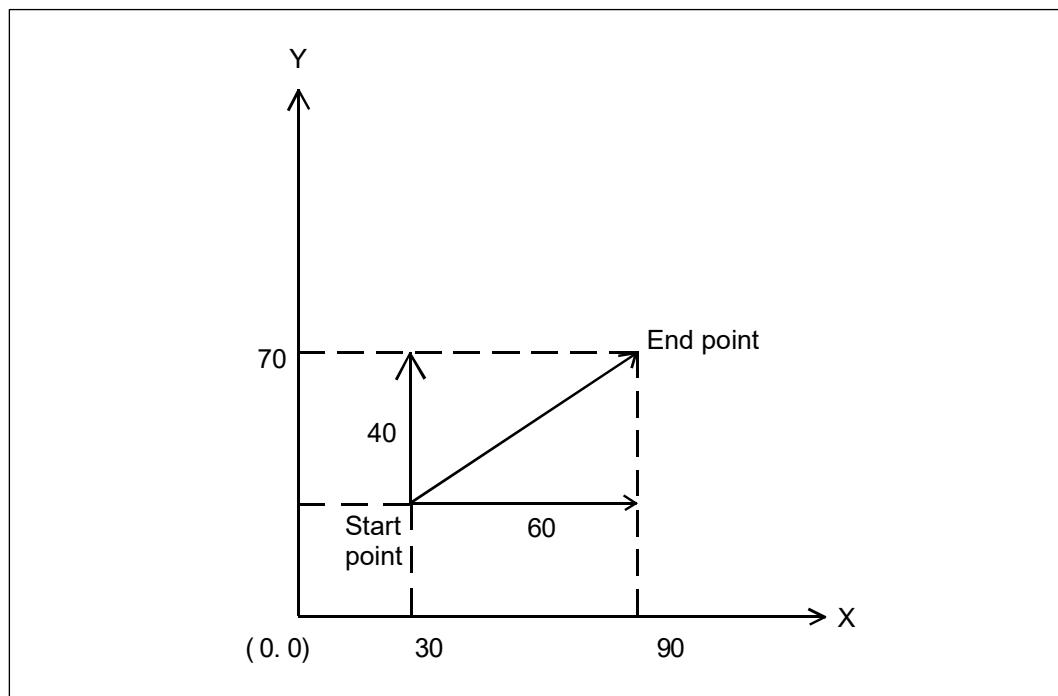
- Absolute command

G90 X90 Y70;

- Incremental command

G91 X60 Y40;

3



3. Additional axis command

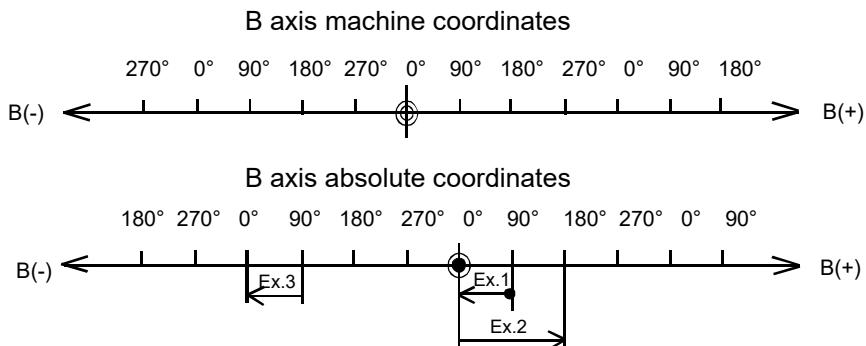
- (1) Absolute command (Ex: 5th-axis)

- The machine rotates to the specified angle when the user parameter (switch 2: stroke) <5th-axis stroke control> is set to <1:Yes>.
- The machine uses the shortest route to rotate to the specified angle when the user parameter (switch 2: stroke) <5th-axis stroke control> is set to <0>No>. When the turning angle is the same in either positive or negative direction (e.g., 180°), the axis rotates toward positive.
- When the user parameter (switch 2: stroke) <5th-axis stroke control> is set to <0>No>, the command value is rounded to less than 360° if the command angle is greater than 360°.

Chapter 3 Preparation Function

When not using the 5th-axis stroke control:

When the machine parameter (system 2: additional axis) <Address> (5th-axis) is set to <1:B>:



3

- 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.

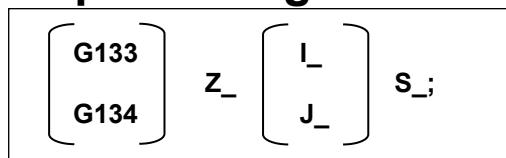
- (○) B axis machine origin
- (●) B axis machining origin (set to 90 deg in this example)
- B axis current position (angle) before travel

(2) Incremental command (Ex: 5th-axis)

The machine rotates only to the specified angle regardless if the user parameter (switch 2: stroke) <5th-axis stroke control> is set to <0:No> or to <1:Yes>. However, when the user parameter (switch 2: stroke) <5th-axis stroke control> is set to <1:Yes>, the alarm “stroke exceeded” or “limit exceeded” may trigger due to the restrictions of the stroke or stroke limit.

3.16 Change of Tap Twisting Direction (G133/134)

Command format



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

3

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.

When the user parameter (switch 2: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the infeed operation. However, if the spindle override is greater than 100%, the motion is carried out at 100%.

When the user parameter (switch 2: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.

- (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.

3.17 G code Priority

- 3
- 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)

(NOTE) G10L52/G11 are noted in the one shot field.

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

Front \ Back	G0 G1	G2 G3	G2.2 G3.2	G102 G103 G202 G203	G33 G392	G17	G18 G19	G22	G23	G40	G41 G42 G14 1 G14 2	G43 G44	G143 G144	G49	G43. 4	G43. 5	G50	G51	G50.1	G51.1	G54	G54.1 P	G54.2	G61	G66	G67	G68	G168	G69	G68.2	G73 G83	G80	G90	G93	G94 G95	G96	G97	G98 G99	G32 1 G32 2 G32 3
G0, G1	3	3	3	3	3	1	1	5	1	1	1	1	1	1	1	1	1	2	2	5	5	1	1	1	2	2	5	5	1	5	5	1	1	1	1	1	1		
G2, G3	3	3	3	3	3	1	1	5	1	12	14	2	2	2	2	2	2	2	2	5	5	1	1	1	2	2	5	5	1	5	5	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	2	2	2	2	5	2	1	1	1	1	2	2	2	2	1	2	5	1	1	2	1	1	1	
G102, G103 G202, G203	3	3	3	3	3	1	2	5	1	12	2	2	2	2	2	2	2	2	2	5	2	1	1	1	2	2	2	2	2	1	2	5	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	2	2	2	2	1	1	2	1	2	2	2	2	1	2	19	1	1	1	1	1		
G17	1	1	1	1	1	3	3	2	1	13	2	1	1	1	1	1	1	1	2	2	2	1	1	1	2	2	9	9	1	1	1	1	1	1	1				
G18, G19	1	1	1	2	1	3	3	2	1	13	2	1	1	1	1	1	1	2	2	2	1	1	1	1	2	2	9	2	1	2	2	1	1	1	1	1			
G22	5	5	5	5	2	2	2	3	3	2	2	2	2	2	2	2	2	2	2	2	1	1	1	1	2	2	2	2	1	2	2	1	1	1	1	1			
G23	1	1	1	1	1	1	1	3	3	2	2	2	2	2	2	2	2	2	2	2	1	1	1	1	2	2	2	2	1	2	2	1	1	1	1	1			
G40	1	12	12	12	2	13	13	2	2	3	3	1	1	1	2	2	2	2	2	2	1	1	1	1	2	2	2	2	1	2	2	1	1	1	1	1			
G41, G42 G141, G142	1	14	14	2	2	2	2	2	2	3	3	1	1	1	1	2	2	2	2	2	1	1	1	1	2	2	2	2	2	1	1	1	1	1	1	1			
G43, G44	1	2	2	2	2	1	1	2	2	1	1	3	3	3	3	3	2	2	2	2	1	1	1	1	2	2	2	2	1	1	1	1	1	1	1				
G143, G144	1	2	2	2	2	1	1	2	2	1	1	3	3	3	3	3	2	2	2	2	1	1	1	1	2	2	2	2	2	1	1	1	1	1	1				
G49	1	2	2	2	2	1	1	2	2	1	1	3	3	3	3	3	2	2	2	2	1	1	1	1	2	2	2	2	1	2	18	1	1	1	1	1			
G43.4	1	2	2	2	2	1	1	2	1	2	2	3	3	3	3	3	2	2	2	2	1	2	2	1	2	2	2	2	1	1	1	2	1/2	2	2	1			
G43.5	1	2	2	2	2	1	1	2	1	2	2	3	3	3	3	3	2	2	2	2	1	2	2	1	2	2	2	2	1	1	1	2	1/2	2	2	1			
G50	2	2	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			
G51	2	2	2	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			
G50.1	5	5	5	5	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	1	1	1	1	1	1			
G51.1	5	5	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	3	3	2	2	2	2	2	2	2	2	1	1	1	1	1	1	1				
G54	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	2	3	3	2	1	2	2	2	1	1	1	1	1	2	1	1			
G54.1P	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	2	3	3	2	1	2	2	2	1	1	1	1	1	2	1	1			
G54.2P	1	2	2	2	2	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	2	2	3	1	2	2	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	1	1	2	2	2	1	1	1	1	3	2	2	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	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	2	2	2	2	2	2	2	2	2	2	2	2	2	2			
G68	5	5	2	2	2	2	9	9	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	3	3	2	2	1	1	1	1	1	1			
G168	5	5	2	2	2	2	9	2	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	3	3	2	2	1	1	1	1	1	1			
G69	1	1	1	1	1	1	1	1	1	1	2	2	2	2	2	2	2	2	2	1	1	1	1	2	2	3	3	1	1	1	1	1	1	1	1	1			
G68.2	5	5	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	3	3	2	1	1	1	1	2	1	1			
G73, G83	6	6	6	6	2	1	2	2	2	2	2	1	2	18	2	2	2	2	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	1	1	1	2	2	2	2	1	1	1	1	2	2	1	1	1	1	1	3	3	1	1	1			
G90	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1			
G94, G95	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	1	1	1	1	1	2	2	1	1	1	1	2	1	1	1	1			
G93	1	1	1	1	1	19	1	1	1	1	1	1	1	1	1	1	1/2	1/2	2	2	1	1	1	1	2	2	1	1	1	1	1	3	3	1	1	1			
G96	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	2	1	1	2	2	2	1	1	2	2	1	1	2	1	3	3	1			
G97	1	1	1	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	1	3	3	1	1	1			
G98, G99	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	1	1	1	1	2	2	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	1	1	2	2	2	2	1	1	1	1	2	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	G10 L52	G11	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	2	2	4	4	4	4	4	4	1	20	4	2	4	4	4	4	4	4	5	2	5
G2, G3	4	1	4	2	2	4	4	4	4	2	4	1	2	4	2	4	4	4	4	4	4	5	2	5
G2.2, G3.2	4	1	4	2	2	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	2	2	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	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G17	2	1	2	2	2	2	2	2	2	1	2	1	2	1	2	2	1	2	1	2	1	1	2	1
G18, G19	2	1	2	2	2	2	2	2	2	2	2	1	2	1	2	2	1	2	2	2	2	2	2	1
G22	2	1	2	2	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	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	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	2	2	1	20	2	2	2	8	2	2	2	2	2	2	2
G43, G44	2	1	2	2	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	2	2	1	1	1	2	2	17	20	1	2	2	1	1	1	2	1	2	2	2
G49	2	1	2	2	2	1	1	1	1	1	2	1	20	1	2	2	1	1	1	18	1	2	18	2
G43.4	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2
G43.5	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	2	2	2	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	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	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	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	2	2
G54	1	1	1	2	2	1	1	1	1	2	1	4	2	1	2	1	1	1	1	1	1	1	2	1
G54.1P	2	1	2	2	2	1	2	2	2	2	1	4	2	1	2	1	1	1	1	1	1	1	2	2
G54.2P	2	1	2	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	2	2	2
G61	1	1	1	2	2	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	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	2	2
G68	2	1	2	2	2	1	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G168	2	1	2	2	2	1	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G69	1	1	1	2	2	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	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	2	2	1	2	1	2	2	2	2	2	2	2	2	3	2
G80	1	1	1	2	2	1	4	4	4	4	4	1	20	4	2	4	4	4	4	4	3	2	1	
G90	1	1	1	2	2	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	2	1
G93	1	1	1	2	2	1	1	1	2	2	1	1	1	20	1	2	1	1	1	2	2	2	2	2
G94, G95	1	1	1	2	2	1	1	1	1	1	1	1	20	1	1	2	2	1	1	1	1	1	1	19
G96	2	1	2	2	2	1	2	2	2	2	1	1	2	1	2	1	1	1	2	2	2	2	2	2
G97	1	1	2	2	2	1	1	1	1	2	1	1	20	1	2	2	1	1	1	2	2	2	2	2
G98, G99	1	1	1	2	2	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	1	1
G321, G322, G323	1	1	1	2	2	1	1	1	1	1	1	1	1	20	1	2	1	1	1	1	1	1	1	1

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

Back \ Front	G0 G1	G2 G3	G2.2 G3.2	G10 2 G10 3 G20 2 G20 3	G33 G39 2	G17	G18 G19	G22	G23	G40	G41 G42 G14 1 G14 2	G43 G44	G14 3 G14 4	G49	G43 .4	G43 .5	G50	G51	G50.1	G51.1	G54	G54.1	G54.2	G61	G66	G67	G68	G16 8	G69	G68.2	G73	G80	G90	G93	G94 G95	G96	G97	G98 G99	G32 1 G32 2 G32 3
G4	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	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					
G10	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	2	1	1	1	1	2	1	1				
G10L5 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					
G11	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					
G12	4	4	2	4	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	1	1	1	1	2	1	2	1					
G28	4	4	4	4	2	2	2	2	2	2	2	1	1	1	2	2	2	2	2	1	2	2	1	2	2	2	2	1	2	2	4	1	1	1					
G30	4	4	4	4	2	2	2	2	2	2	1	1	1	2	2	2	2	2	2	1	2	2	1	2	2	2	2	1	2	4	1	1	1						
G31	4	4	4	4	2	2	2	2	2	2	1	1	1	2	2	2	2	2	1	2	2	1	2	2	2	2	1	2	2	4	1	1	1						
G36	4	2	2	2	2	1	2	2	2	2	1	2	1	2	2	2	2	2	1	2	2	1	2	2	2	1	2	2	4	1	1	2	1						
G52	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	2	1	2	2	4	1	1	1	1						
G53	1	1	1	1	1	1	1	1	1	1	17	17	1	2	2	2	2	2	2	4	4	2	1	2	2	1	1	1	1	1	1	1	1						
G53.1	20	2	2	2	2	2	2	2	20	2	20	20	20	2	2	2	2	2	20	2	2	20	2	2	2	2	2	2	20	20	20	20	20						
G60	4	4	4	4	2	1	1	2	2	2	1	1	1	2	2	2	2	2	1	1	1	1	2	2	2	1	2	1	4	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					
G92	4	4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	2	1	2	2	4	1	1	1	2	1					
G100	4	4	4	4	2	2	2	2	2	1	8	1	1	1	1	1	2	2	2	1	1	1	1	2	2	2	2	1	2	2	4	1	1	1					
G120	4	4	4	4	2	2	2	2	2	2	1	1	1	2	2	2	2	2	1	1	1	2	2	2	2	1	2	2	4	1	1	1	1						
G121	4	2	2	2	2	1	2	2	2	2	1	2	18	2	2	2	2	2	1	1	2	1	2	2	2	1	2	2	4	1	2	1	2	1					
G131	4	4	4	4	2	2	2	2	2	2	1	1	1	2	2	2	2	2	1	1	2	1	2	2	2	1	2	2	4	1	2	1	2	1					
G133	4	4	4	4	2	1	2	2	2	2	2	2	2	2	2	2	2	2	1	1	2	1	2	2	2	1	2	2	4	1	2	1	2	1					
G173	6	6	6	6	2	1	2	2	2	2	1	2	18	2	2	2	2	2	1	2	2	1	2	2	2	1	2	3	3	1	2	1	2	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					
G376	6	6	6	6	2	1	1	2	2	2	2	2	2	2	2	2	2	2	2	1	2	2	1	2	2	2	1	2	1	1	2	19	2	2	1				

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

	G4	G9	G10	G10 L52	G11	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	2	2	1	2	2	2	2	2	2	2	2	2	2	2	
G9	1	3	1	2	2	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	2	2	1	2	2	2	2	2	2	2	2	2	2	2	2
G10L52	2	2	2	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G11	2	2	2	2	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G12	2	1	2	2	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	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	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	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	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	2	2	3	1	2	2	2	2	2	2	2	2	2	2	2	2
G53	1	1	1	2	2	1	1	1	1	1	3	2	1	2	1	1	1	1	1	1	1	1	2	1
G53.1	2	20	2	2	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	2	2	2	1	2	1	2	3	2	2	2	2	2	2	2	1	2
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
G92	2	1	2	2	2	2	2	2	2	2	2	1	2	2	2	2	3	2	2	2	2	2	2	2
G100	2	1	2	2	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	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	2	2	1	2	2	2	2	2	2	3	2	2	2	2	2
G131	2	1	2	2	2	2	2	2	2	3	2	2	1	2	2	2	2	2	2	3	2	2	2	2
G133	2	1	2	2	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	2	2	1	2	1	2	2	2	2	2	2	3	2	2	2
G210	2	2	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	2	2	1	2	2	2	2	2	2	2	2	2	2	2	3

■ When a command is given during modal call

Modal Command	G0 G1	G2 G3	G2.2 G3.2	G33 G39 2	G17	G18 G19	G22	G23	G40	G41 G42 G14 1 G14 2	G43 G44 3 G14 4	G49	G43. 4	G43. 5	G50	G51	G50. 1	G51. 1	G54	G54. 1	G54.2 P0	G54.2 P1	G61	G66	G67	G68 G16 8	G69	G68.2 Before G53	G68. 2 After G53. 1	G73 G83	G74 G84	G80	G90	G93	G94 G95	G96	G97	G98 G99	G32 1 G32 2 G32 3
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	1	1	1	1	1						
G2 G3	3	3	3	3	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1	2	1	6	6	1	1	1	1					
G2.2 G3.2	3	3	3	3	1	1	1	1	1	1	1	1	1	2	2	1	2	1	2	1	1	1	1	1	1	1	2	1	6	6	1	1	2	1	1	1			
G102 G103 G202 G203	3	3	3	3	1	2	1	1	1	2	1	1	2	2	1	2	1	2	1	1	1	1	1	2	1	2	2	6	6	1	1	1	1	1	1				
G33 G392	3	3	3	3	1	1	1	1	1	2	1	1	1	2	2	1	2	1	2	1	1	1	1	1	1	2	1	2	2	6	6	1	1	2	19	1	1	1	
G17	1	1	1	1	3	3	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1		
G18 G19	1	1	1	1	3	3	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	2	2	2	2	1	1	1	1	1		
G22	1	1	1	1	1	1	3	3	1	1	1	1	1	2	2	1	2	1	2	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	1	1	1	2	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			
G41 G42 G141 G142	1	1	1	2	1	1	1	1	3	15	1	1	1	2	2	1	1	1	1	1	1	1	1	1	2	1	1	1	2	1	1	1	1	1	1	1			
G43 G44	1	20	20	2	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	20	20	2	1	1	1	1	1	16	16	3	2	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1			
G49	1	1	1	2	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			
G43.4	1	2	2	2	1	1	1	1	1	2	1	2	1	2	2	1	2	1	2	1	1	1	1	1	1	2	1	2	2	2	2	1	1	2	1	1			
G43.5	1	2	2	2	1	1	1	1	1	2	1	2	1	2	2	1	2	1	2	1	1	1	1	1	2	1	2	2	2	2	1	1	2	1	2	1			
G50	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	3	3	1	1	1	1	1	1	1	1	2	1	2	2	2	1	1	1	1	1	1			
G51	1	1	1	2	2	1	1	1	1	1	1	1	1	1	2	2	3	3	1	1	1	1	1	1	1	2	1	2	2	2	1	1	1	1	1	1			
G50.1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	3	3	1	1	1	1	1	1	2	1	2	2	2	1	1	1	1	1	1		
G51.1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	2	2	3	3	1	1	1	1	1	1	1	2	1	2	2	2	1	1	1	1	1	1			
G54	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	1	2	3	3	1	1	1	1	1	1	2	2	2	1	1	1	1	1	1	1			
G54.1P	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	2	1	2	3	3	1	1	1	1	1	1	2	2	2	1	1	1	1	1	1	1			
G54.2P0	1	1	1	2	1	1	1	1	1	1	1	1	1	2	2	2	1	2	1	1	3	3	1	1	1	2	1	1	1	1	1	1	1	1	1	1			
G54.2P1	1	1	1	1	2	1	1	1	1	1	1	1	1	2	2	2	1	2	1	1	3	3	1	1	1	2	1	2	2	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	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	1	1	1	2	3	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	1	1	1	3	3	1	1	1	1	1	1	1	1	1			
G68	1	1	1	1	2	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	2	1	1	1	3	3	2	2	1	1	1	1			
G168	1	1	1	1	2	1	2	1	1	1	1	1	1	2	2	2	1	1	1	1	1	1	1	1	2	1	1	1	3	3	2	2	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	1	1	1	1	1	1	1	1	1	1	1	1			
G68.2	1	1	1	1	2	1	1	1	2	1	1	2	1	2	2	1	2	1	2	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1			

Modal Command \	G0 G1	G2 G3	G2.2 G3.2	G33 G39	G17	G18 G19	G22	G23	G40	G41 G42 G14 1 G14 2	G43 G44	G14 3 G14 4	G49	G43. 4	G43. 5	G50	G51	G50. 1	G51. 1	G54	G54. 1	G54.2 P0	G54.2 P1	G61	G66	G67	G68 G16 8	G69	G68.2 Before G53.	G68. 2 After G53. 1	G73 G83	G74 G84	G80	G90	G93	G94 G95	G96	G97	G98 G99	G32 1 G32 2 G32 3									
G73 G83	1	1	1	2	1	2	1	1	1	2	1	2	1	2	2	1	1	1	1	10	1	1	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	2	2	1	1	1	1	10	1	1	1	1	1	1	1	1	1	1	1	2	1	3	3	3	1	2	1	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	1	1	1	1	1	1	3	3	3	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	1	1	1	1	1	1	1	1	1	1	1	1							
G93	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	3	2	1	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	1	1	1	1	3	1	1	1	1	1	1							
G96	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	1	2	1	3	3	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	1	1	1	3	3	1	1	1	1								
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	1	1	1	1	1	1	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	1	1	1	3	1	1	1	1					
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	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	1	1	1	1	1	1	1	1	1			
G10	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	11	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			
G10L52	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	1	1	1	1	1	1	1	1	1			
G11	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	1	1	1	1	1	1	1	1	1			
G12	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	2	1	1	1	1	1	1	1	1	1			
G28	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		
G30	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		
G31	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	1	1	1	1	1	1	1	1	2	1	1	1	2	2	1	1	1	2	1	1	1	1	1	1	1			
G36	1	2	2	2	1	2	1	1	2	1	2	1	2	2	1	2	1	2	1	2	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1		
G52	1	1	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	2	2	1	1	1	1	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	1	1	2	2	1	2	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		
G53.1	1	1	1	2	1	2	1	1	2	1	2	2	1	2	1	2	1	2	1	2	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		
G60	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	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		
G65	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	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	1	1	2	1	2	1	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		
G100	1	1	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	1	1	1	1	1	1	1	1		
G120	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		
G121	1	2	1	2	1	2	1	1	2	1	2	1	1	1	1	2	1	2	1	2	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	
G131	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	2	1	2	1	1	1	1	1	1	1	2	1	1	1	2	2	1	1	1	2	1	1	1	1	1	1	1	1	1		
G133	1	1	1	1	2	1	1	1	2	1	2	2	1	1	1	1	1	1	1	1	10	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	2	1	1	1	1	1	1	1		
G173	1	1	1	1	2	1	1	1	2	1	1	1	1	1	1	1	1	1	1	10	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	
G210	1	1	1	1	1	1	1	1	1	1	1	1	1	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	1	1	1	1	1	1	1		
G376	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	2	2	1	1	2	1	1	1	1	1	1	1	1	2	1	2	2	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1

3.18 Programmable Data Input (High Accuracy) (G210)

3.18.1 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 “1.5 User parameters” in the Data Bank & Alarm Manual for further details.

3.18.2 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)

3

- 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 “1.5 User parameters” in the Data Bank & Alarm Manual for further details.

3.18.3 Temporary Parameter Change with TCP Control

Command format

G210 L35 Pn I_ U_ V_ Z_ B_ C_ Q_ E_;

n = 1 : TCP control (M280)
 n = 2 : TCP control (M281)
 n = 3 : TCP control (M282)
 n = 4 : TCP control (M283)
 n = 5 : TCP control (M284)
 n = 6 : TCP control (M285)
 n = 7 : TCP control (M286)
 n = 8 : TCP control (M287)

I : Corner deceleration override (X-, Y- and Z-axes) (%)
 U : Corner deceleration override (tilt axis) (%)
 V : Corner deceleration override (rotation axis) (%)
 Z : Acceleration override (X-, Y- and Z-axes) (%)
 B : Acceleration override (tilt axis) (%)
 C : Acceleration override (rotation axis) (%)
 Q : Smooth override (all common axes) (%)
 E : Maximum rotation speed override (%)

- (NOTE 1) The set parameters can be cancelled using M02, 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 omitted parameter does not change from the current value.
- (NOTE 4) The range from 10 to 100 (%) can be set to the E numerical value.
- (NOTE 5) The range that can be set to the numerical value for IUVZBCQ is the same as the corresponding user parameter. Refer to “1.5 User parameters” in the Data Bank & Alarm Manual for further details.

3.19 Thread Cutting

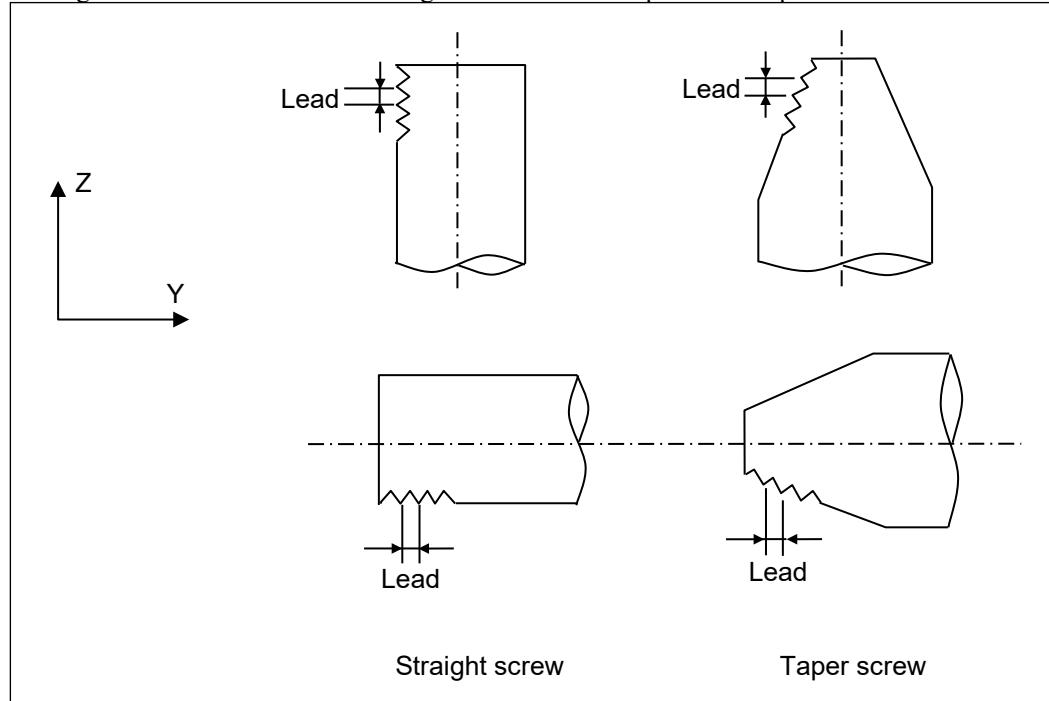
The thread cutting function is available when equipped with the lathe function. A thread cutting command can be issued when using the feed rate per rotation (G95) modal. If issuing a command using the feed rate per minute (G94) modal, the alarm <<Thread cutting command not possible in feed rate per minute>> 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.

(NOTE) When a thread cutting command (G33/G392/376) is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered. In addition, when a TCP control command is issued during a thread cutting modal (G33/G392), the alarm <<TCP control command not possible>> is triggered.

3

3.19.1 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 $\begin{pmatrix} X \\ Y \\ Z \end{pmatrix}$ F_ Q_

When cutting a taper screw

G33 $\begin{pmatrix} X \\ Y \\ X \\ Z \\ Y \\ Z \end{pmatrix}$ 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)

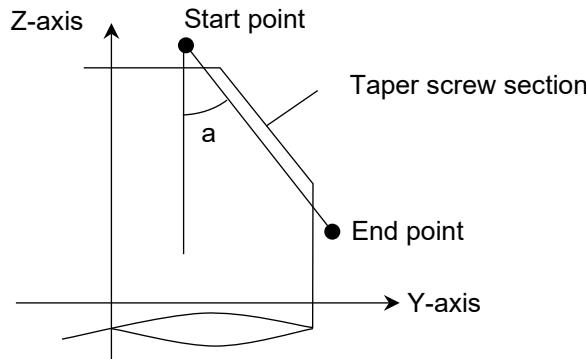
Q : Start angle for thread cutting (0.000° to 360.000°)

If this is omitted, the default is 0 degrees.

The lathe spindle machine zero point is the start angle for thread cutting: 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.

- (NOTICE) 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 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) When the user parameter <Thread cutting override> is set to <0: Disable>, the spindle override and the feedrate override are disabled, and the setting is fixed at 100%. When the user parameter <Thread cutting override> is set to <1: Spindle override>, the spindle override is applied to the spindle speed and cutting feedrate. However, in this situation, if the spindle override is greater than 100%, the motion is carried out at 100%.
- (NOTE 4) 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.
- (NOTE 5) The cutting speed during thread cutting, while the [DRY] key is OFF, is not restricted in the PLC signals (BDY164 to BDY165).

3.19.2 Thread Cutting Cycle (G392)

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

3.19.2.1 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)

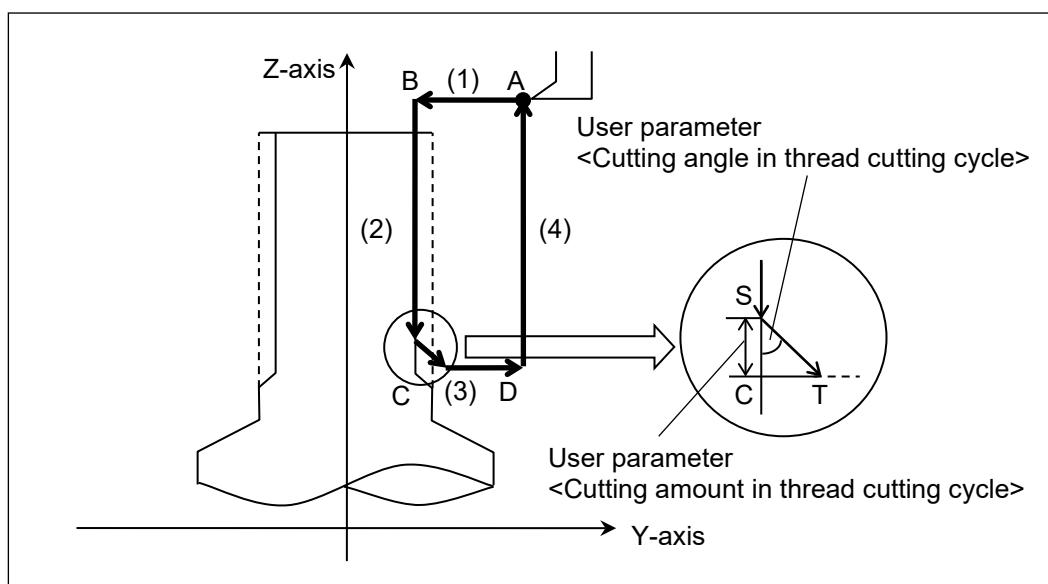
F : Screw lead (mm) (inch)

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

If this is omitted, the default is 0 degrees.

The lathe spindle machine zero point is the start angle for thread cutting: 0 degrees.

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 "3.20 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^{-1})

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.

- (NOTICE) 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.
- (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 the motion in (2), if the user parameter <Thread cutting override> is set to <0: Disable>, the spindle override and the feedrate override are disabled, and the setting is fixed at 100%. When the user parameter <Thread cutting override> is set to <1: Spindle override>, the spindle override is applied to the spindle speed and cutting feedrate. However, in this situation, if the spindle override is greater than 100%, the motion is carried out 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.
 - Operation 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) When the Z-axis perimeter mode is on (M300), the rapid feed operation for (4) applies to Z-axis perimeter operation. Refer to “Chapter 12 M function” for further details on the Z-axis perimeter operation.
- (NOTE 9) The cutting feedrate for (2), while the [DRY] key is OFF, is not restricted in the PLC signals (BDY164 to BDY165).

3.19.2.2 Thread Cutting Cycle for Taper Screws

Command format

G392	$\left[\begin{array}{l} X_Y_ \\ X_Z_ \\ Y_Z_ \end{array} \right]$	R_F_Q_
-------------	--	---------------

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

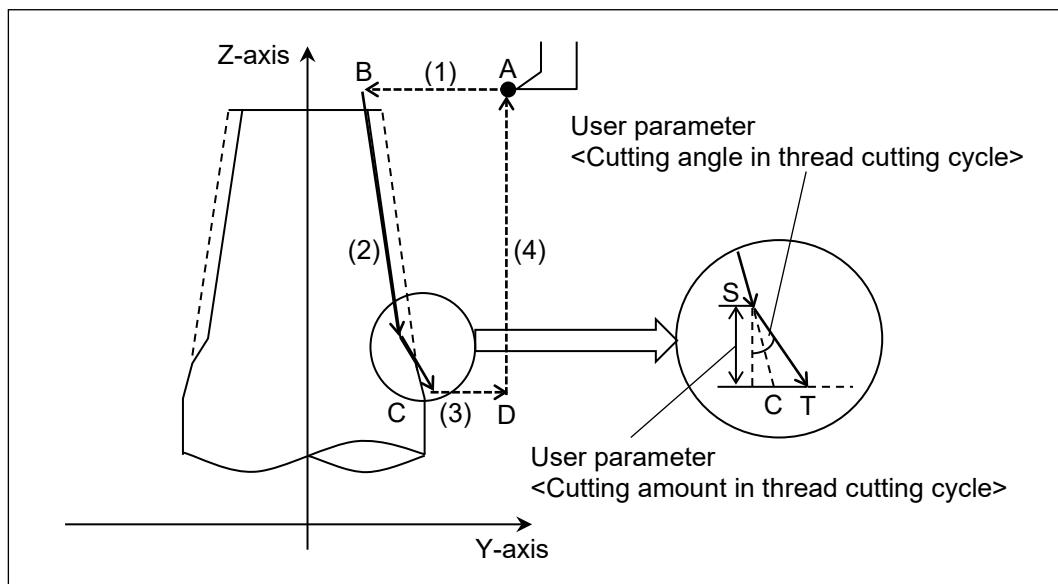
R : Taper amount (mm) (inch)

F : Screw lead (mm) (inch)

Q : Start angle for thread cutting (0.000° to 360.000°)
If this is omitted, the default is 0 degrees.

The lathe spindle machine zero point is the start angle for thread cutting: 0 degrees.

3



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 “3.20 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).

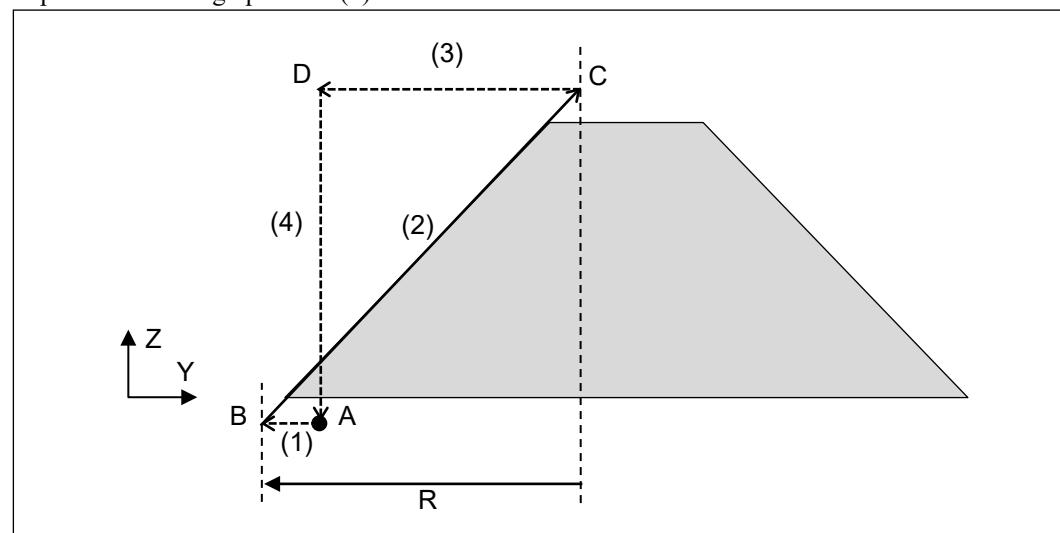
Operations (3) to (4) are the same as the thread cutting cycle for straight screws.

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
+	<p>1</p>	<p>2</p>
-	<p>3</p>	<p>4</p>

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.

3.19.3 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

G376	$\begin{bmatrix} X_Y_ \\ X_Z_ \\ Y_Z_ \end{bmatrix}$	R_P_Q_F_D_E_I_J_K_L_
------	--	----------------------

3

- 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°)
 If this is omitted, the default is 0 degrees.
 The lathe spindle machine zero point is the start angle for thread cutting: 0 degrees.

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>

3.19.3.1 Infeed

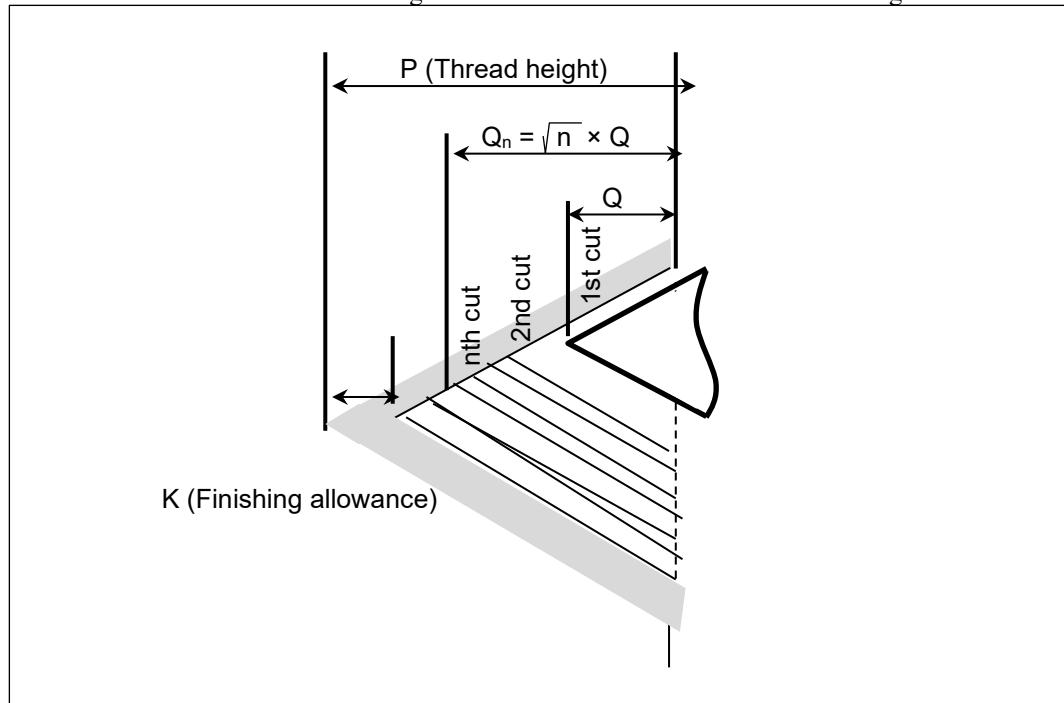
When a command for the complex thread cutting cycle is issued, the infeed is carried out for a “fixed cutting amount and single edge cutting” until the finishing operation. The infeed amount Q_n for nth cut is as follows:

$$Q_n = \sqrt{n} \times Q$$

Q: 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.

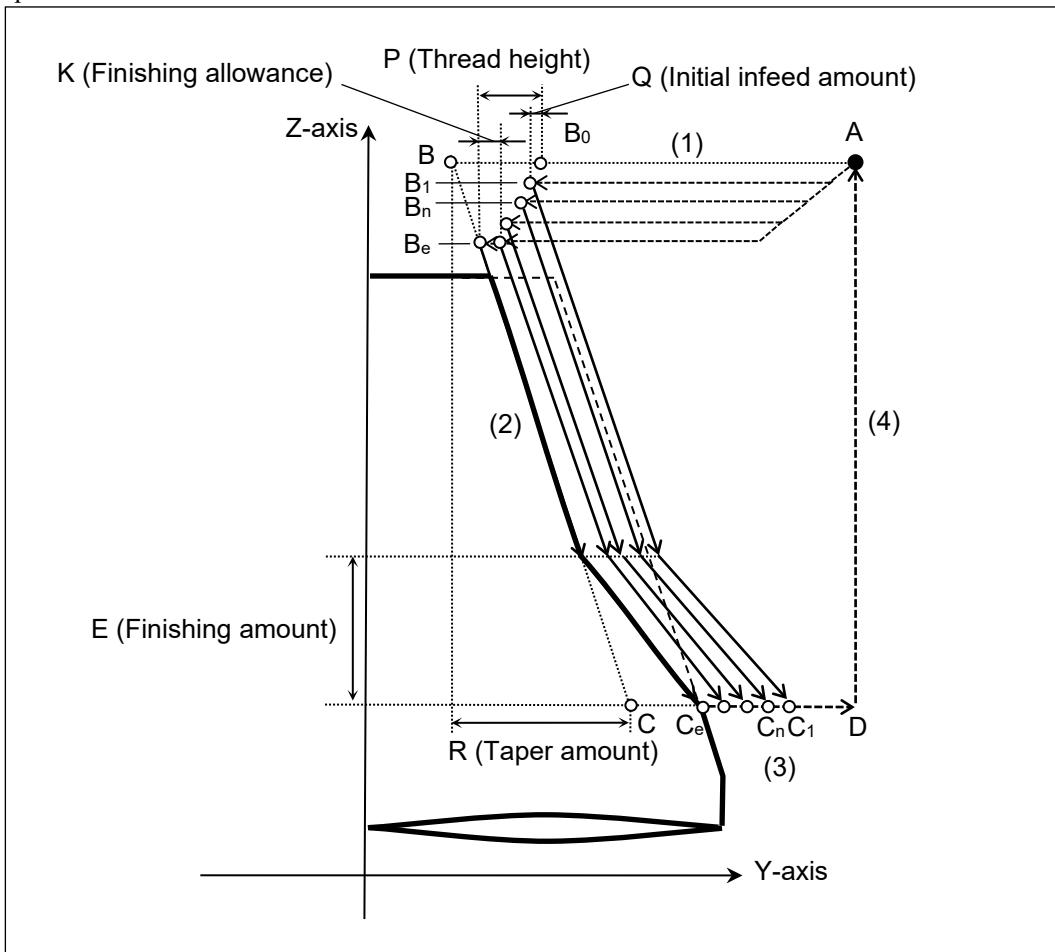
In addition, if the infeed overlaps the finishing section (gray area below), then the infeed amount becomes the difference when finishing allowance is subtracted from the thread height.



3.19.3.2 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.

3



Rough cutting

- (1) It travels using rapid feed from start point A to start point B_n for thread cutting on the n^{th} cut.

The infeed amount is calculated based on the formula noted in “3.19.3.1 Infeed” and it travels in the cutting direction only for the distance that is calculated from the following formula.

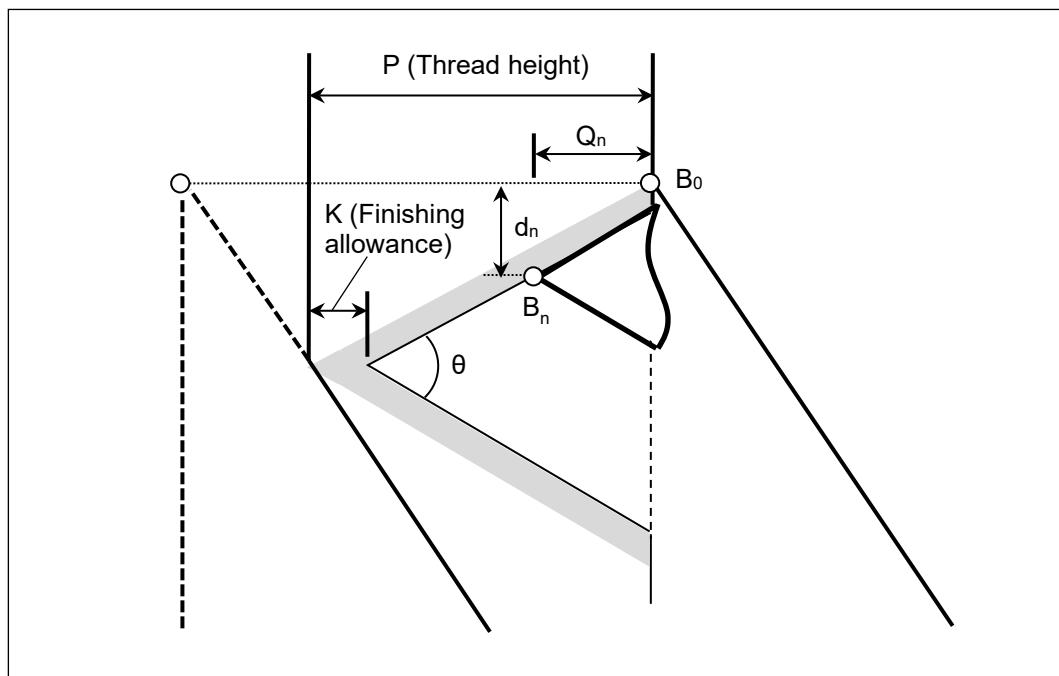
$$d_n = (Q_n + K) \times \tan(\theta / 2)$$

d_n : Distance that is traveled in the cutting direction from the start point for thread cutting on the n^{th} cut (mm)(inch)

Q_n : Infeed amount for nth cut (mm) (inch)

K : Finishing allowance (mm) (inch)

θ : Thread angle ($^{\circ}$)



- (2) It travels using the cutting feed from start point B_n for thread cutting on the n^{th} cut until the end point C_n 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_n 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_e for finishing. It travels in the cutting direction only for the distance d_e that is calculated from the following formula.

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

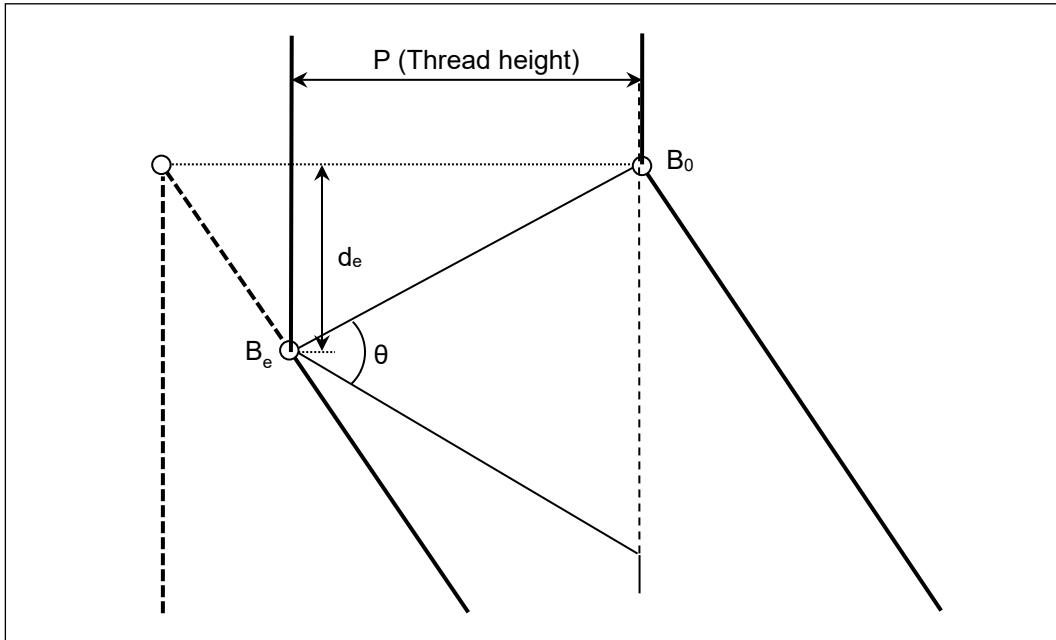
d_e : Distance that is traveled in the cutting direction from the start point for finishing (mm)
(inch)

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

θ : Thread angle ($^{\circ}$)

When d_e 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.

3



- (2) It travels using the cutting feed from start point B_e for finishing until the end point C_e 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_e 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.

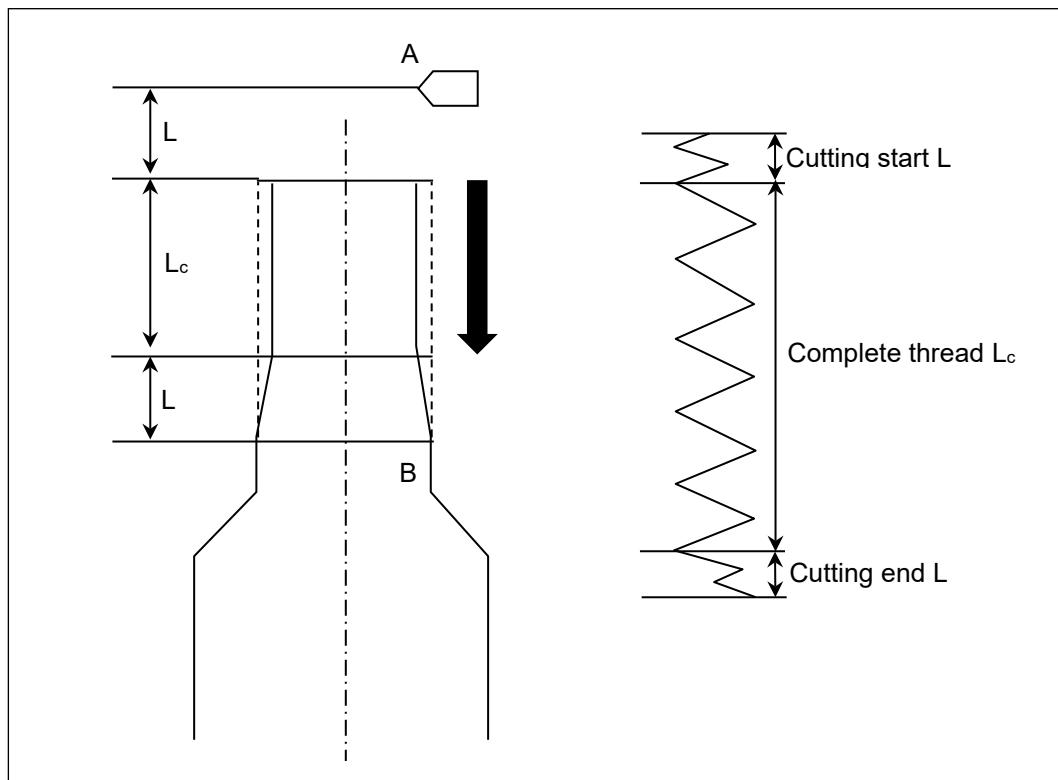
The special notes for the operations are the same as the thread cutting cycle (G392).

3.19.4 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

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

P : Lead (mm)

Model	K
M200Xd1	0.047

3.19.5 Program Example

This is an example of a program when thread cutting on a machine model equipped with the lathe function.

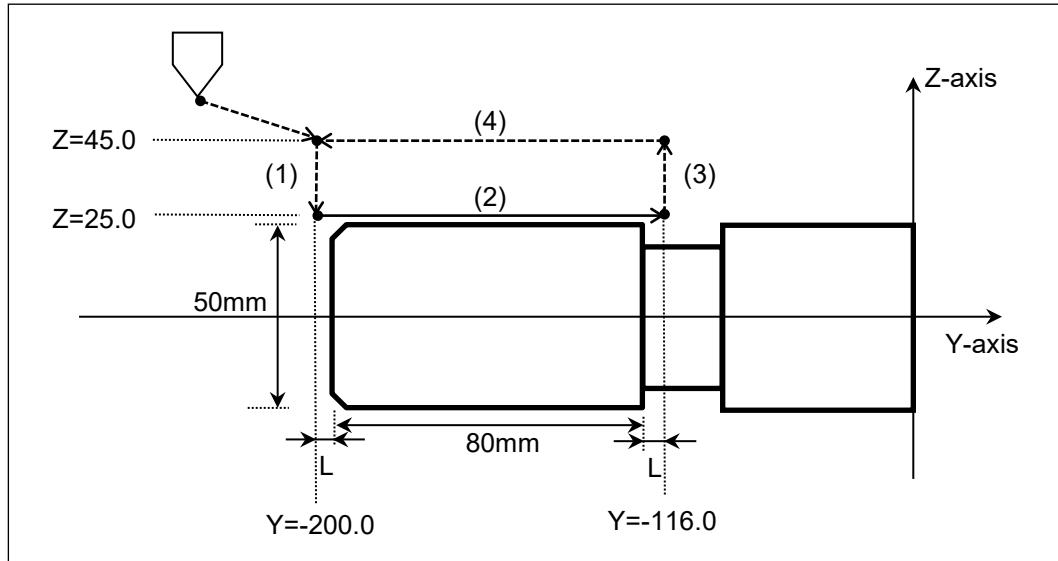
3.19.5.1 Machining and Thread Cutting Straight Screws

Screw type	Right screw
Screw diameter	50 mm
Lead	2 mm
Spindle speed	1000 min ⁻¹

Thread runout $0.047 \times (1000 / 60) \times 2 = 1.566$ (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 2 mm.

3



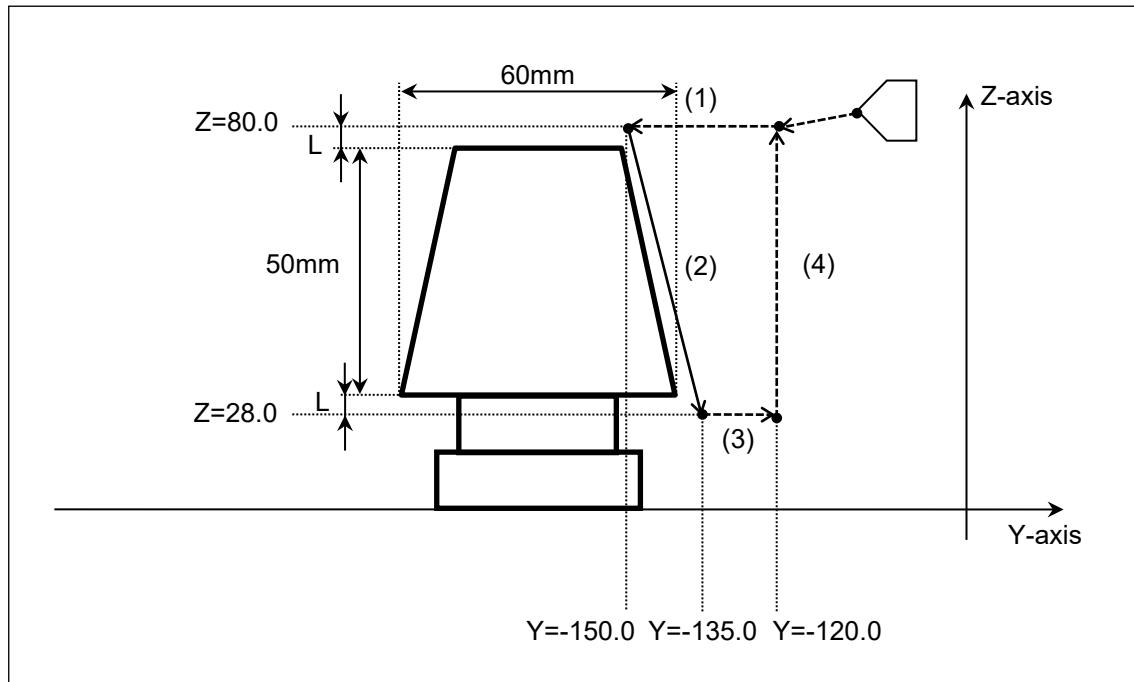
When made using G33	When made using G392	When made using G376
M303 S1000	M303 S1000	M303 S1000
G00 Y-200. Z45.	G00 Y-200. Z45.	G00 Y-200. Z45.
Z25.	(1)	G392 G322 Y-116. Z25. F2. M323
G33 Y-116. F2.	(2)	G376 G322 Y-116. Z23.5 P2. Q0.5 F2. M323
G00 Z45.	(3)	
Y-200.	(4)	
Z24.5	Z24.5	
G33 Y-116.		
G00 Z45.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

3.19.5.2 Machining and Thread Cutting Taper Screws

Screw type	Right screw
Screw diameter	60 mm
Lead	2 mm
Spindle speed	500 min ⁻¹

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.



3

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

3.20 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

3

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

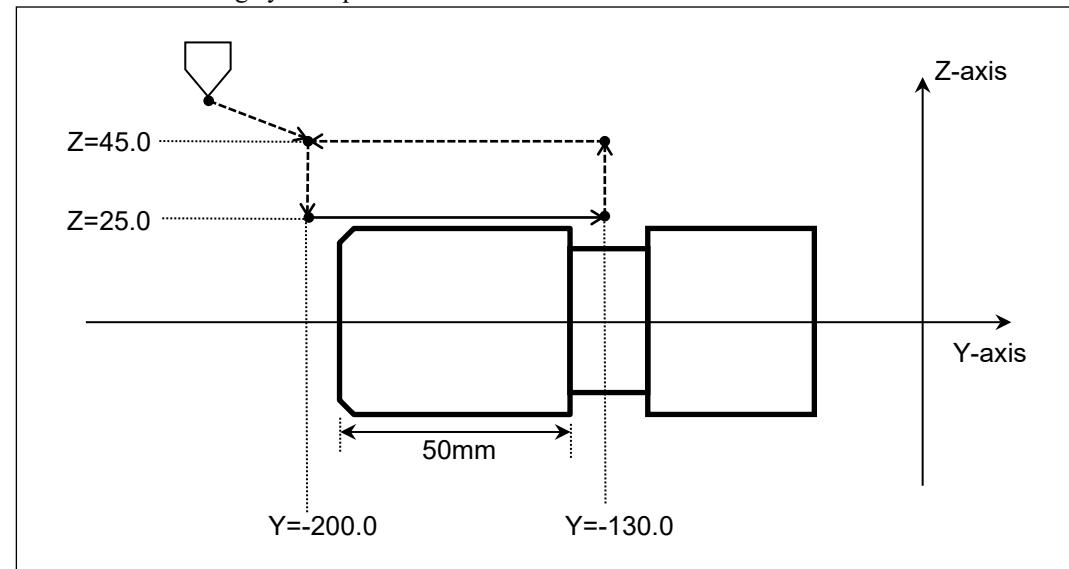
At this time, the travel direction is set by the command axis address and the G code modal (Refer to table below). If that setting is invalid, the alarm <<Thread cutting command error>> is triggered at the time of the command.

Command axis address	G code modal for infeed direction	Travel direction
X_Y_-	G321	X-axis
	G322	Y-axis
	G323	<<Thread cutting command error>>
X_Z_-	G321	X-axis
	G322	<<Thread cutting command error>>
	G323	Z-axis
Y_Z_-	G321	<<Thread cutting command error>>
	G322	Y-axis
	G323	Z-axis

The modal can be specified during power startup in the user parameter (switch 1: canned cycle) <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



3

Program	Description
M303 S1000	Lathe spindle rotates clockwise at 1000 min ⁻¹
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 FUNCTION (COMPENSATION FUNCTION)

- 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

$(G41 \\ G42) Dn;$

G code and D code used for cutter compensation

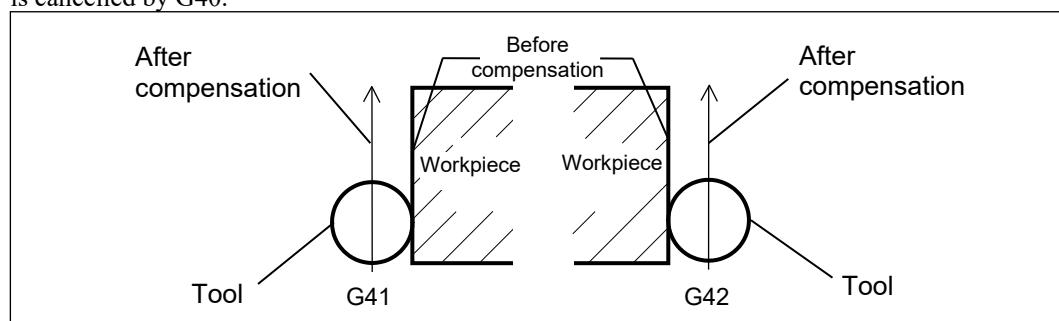
G40: Cutter compensation cancels (This mode is used when the power is turned ON.)

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)

4

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, 201 to 299), 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).
- (NOTE 5) When a cutter compensation command (G41 and G42) is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered. In addition, when a TCP control command is issued while in cutter compensation mode, the alarm <<TCP control command not possible>> is triggered.

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.

$$\text{Cutter offset} = \text{Cutter compensation} + \text{Cutter wear offset}$$

(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.

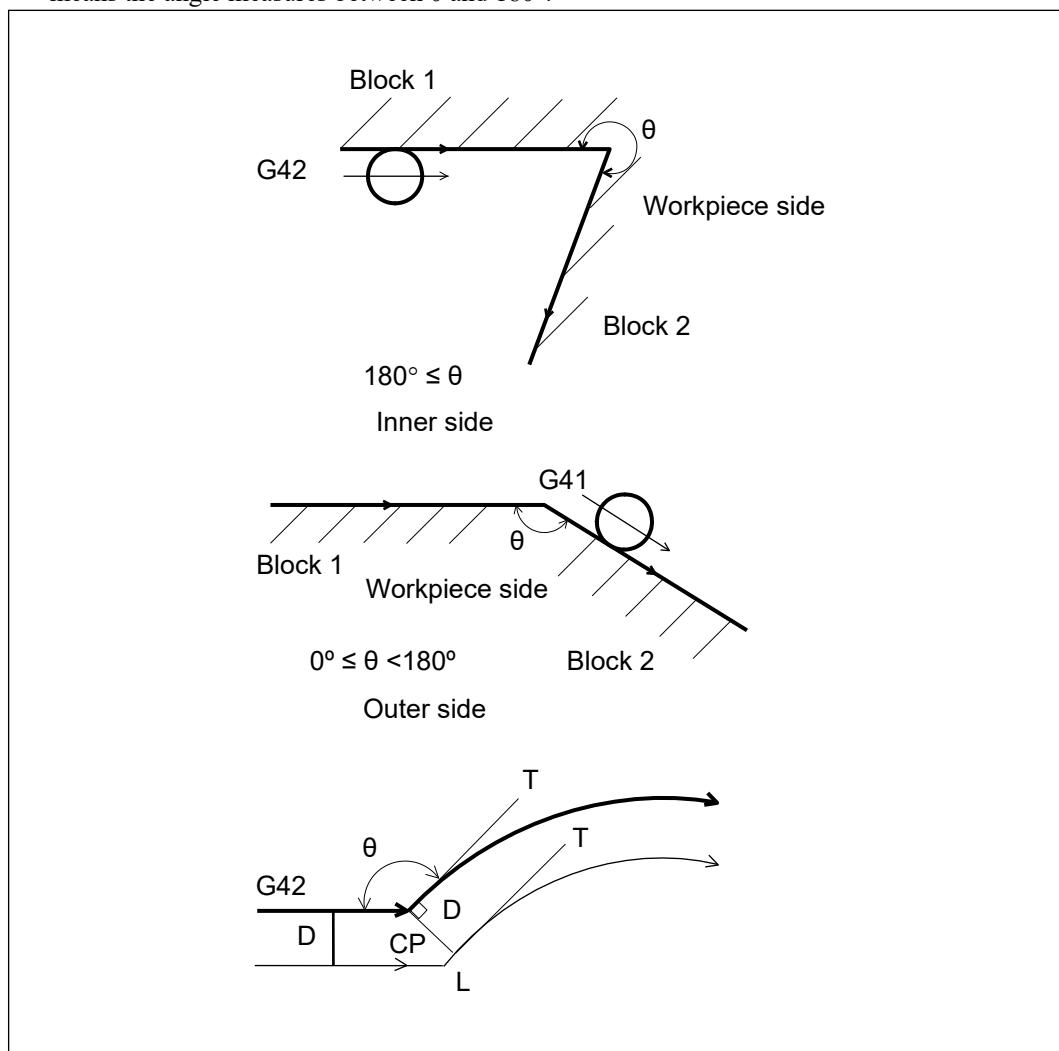
Normally, the tool path that is programmed matches the center path of the actual tool.

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

4

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° on the workpiece side. Outer side means the angle measures between 0 and 180° .



2. Explanation of symbols in diagram

<u> </u>	: Program path
<u> </u>	: Tool center path
<u> </u>	: Auxiliary line
L	: Straight line
C	: Arc
D	: Cutter compensation
θ	: 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.

4

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

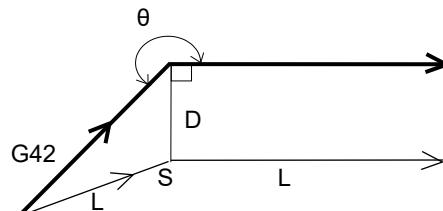
(NOTE 1) The alarm <<Cutter compensation error>> is triggered when an arc command or an involute interpolation is issued.

(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.

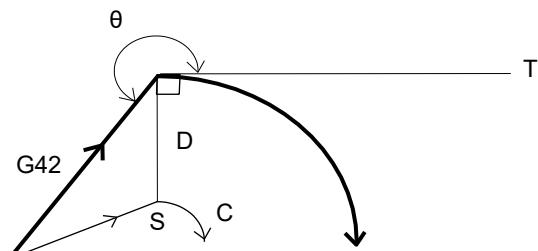
(NOTE 3) There are two setting types for the startup and cancel operations: <0: Type 1 (shortcut)> and <1: Type 2 (detour)>. Use the user parameter (switch 1: compensation function) <Start up/cancel> to set one of the types.

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

Straight line to straight line

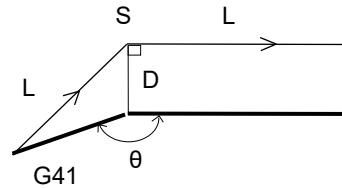


Straight line to arc

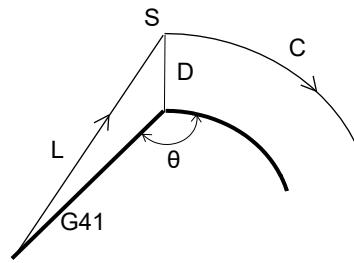


4.1.3.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)

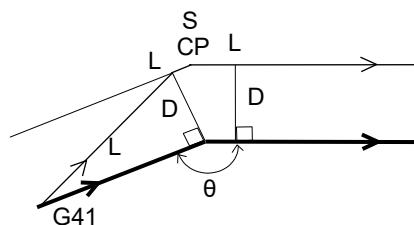
1. Type 1: Straight line to straight line



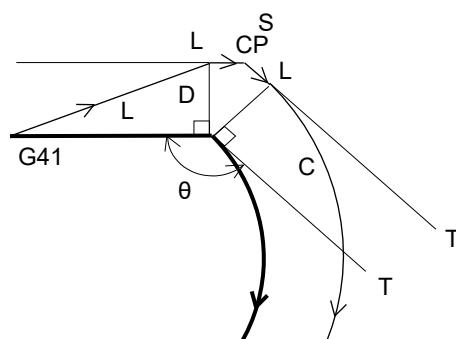
Type 1: Straight line to arc



2. Type 2: Straight line to straight line

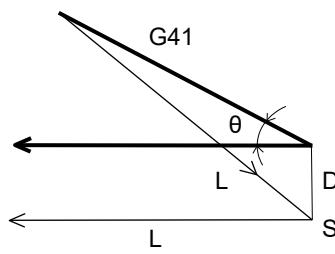


Type 2: Straight line to arc

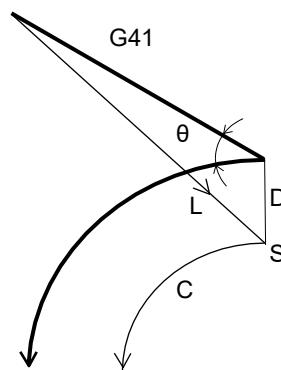


4.1.3.3 Outer Side (Acute Angle Cutting) ($\theta < 90^\circ$)

1. Type 1: Straight line to straight line

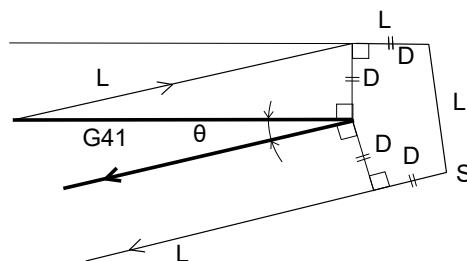


Type 1: Straight line to arc

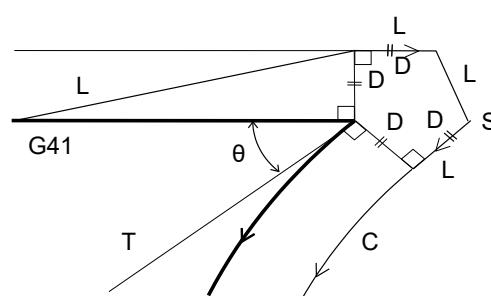


4

2. Type 2: Straight line to straight line



Type 2: Straight line to arc



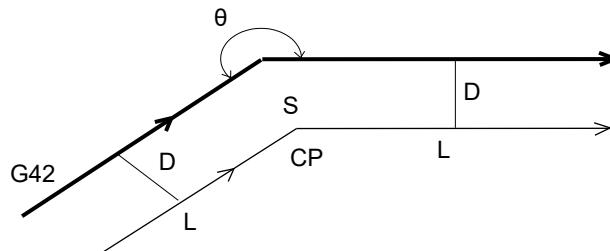
(NOTE) When $\theta \leq 1^\circ$, the setting <0: Type 1 (shortcut)> is used or enabled, even if <1: Type 2 (detour)> is specified for the user parameter (switch 1: compensation function) <Start up/cancel>.

4.1.4 Offset Mode

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

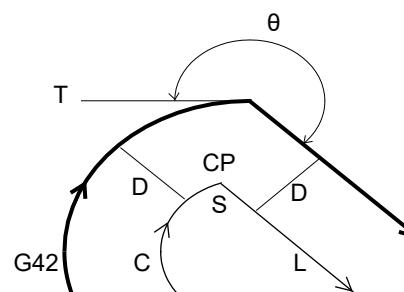
4.1.4.1 Inner Side Cutting ($180^\circ \leq \theta$)

Straight line to straight line

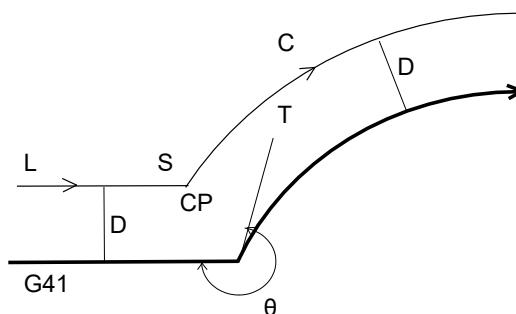


4

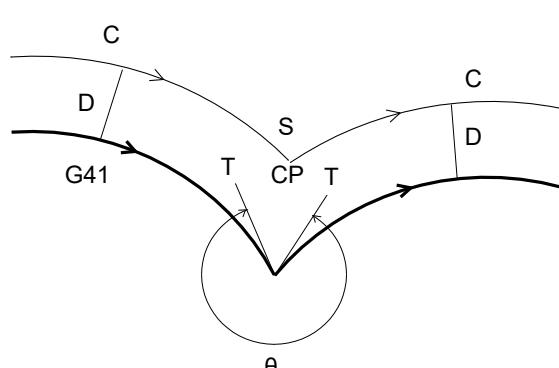
Arc to straight line



Straight line to arc



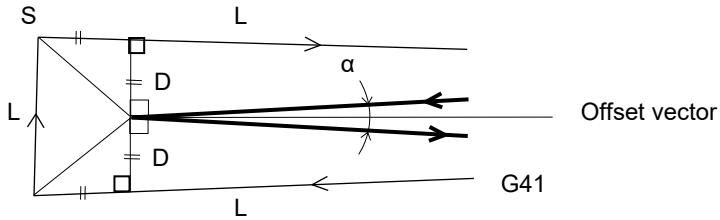
Arc to arc



Chapter 4 Preparation Function (Compensation Function)

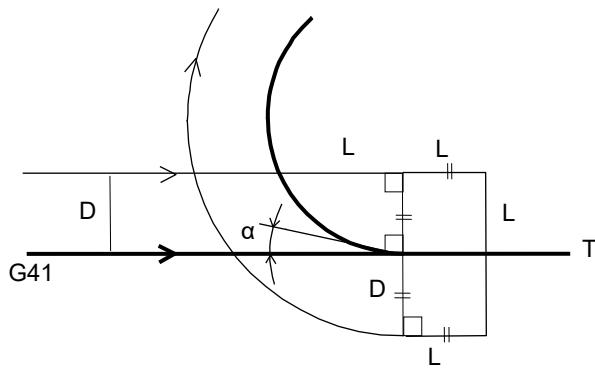
(NOTE 1) When turning on the inner side of a narrow angle ($\alpha < 1^\circ$), and the offset vector is abnormally large.

Straight line to straight line



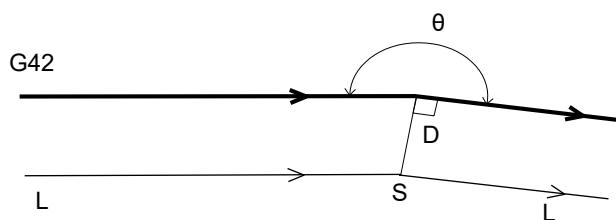
4

Straight line to arc



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

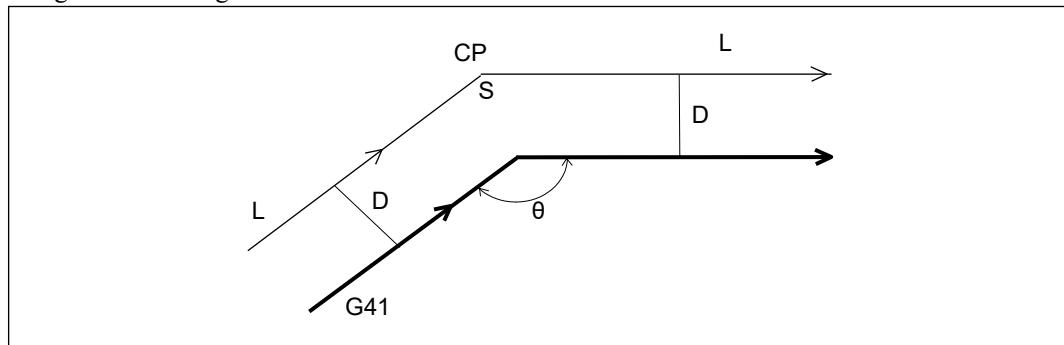
Straight line to straight line



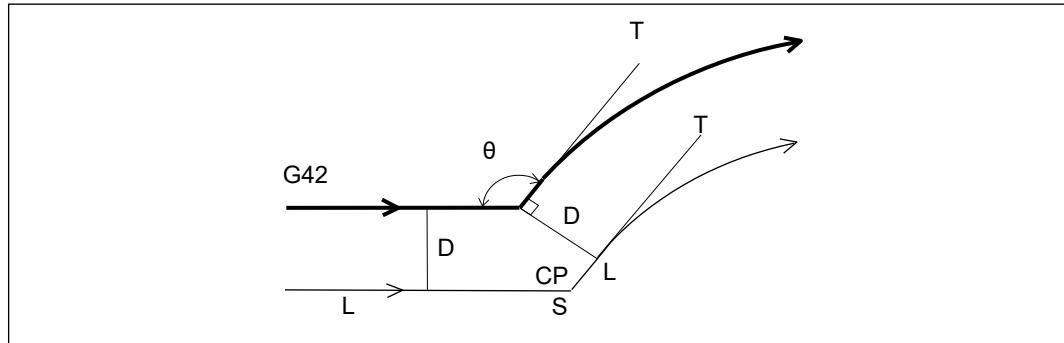
The processing is the same for: arc → straight line, straight line → arc and arc → arc.

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

Straight line to straight line

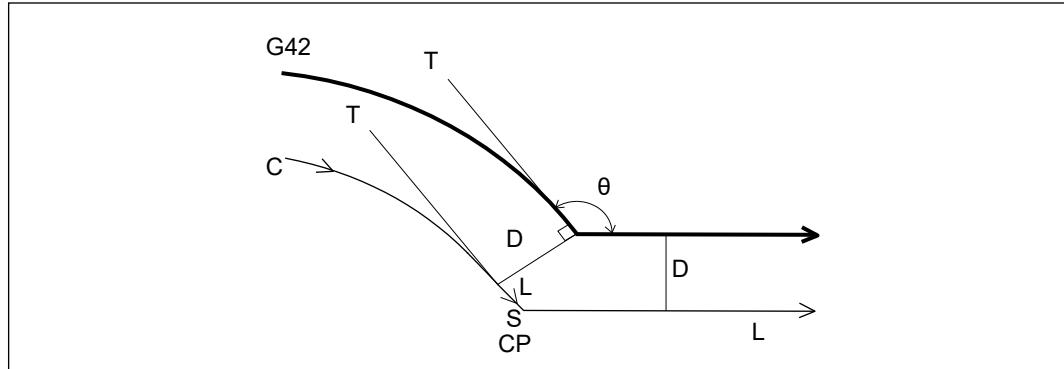


Straight line to arc

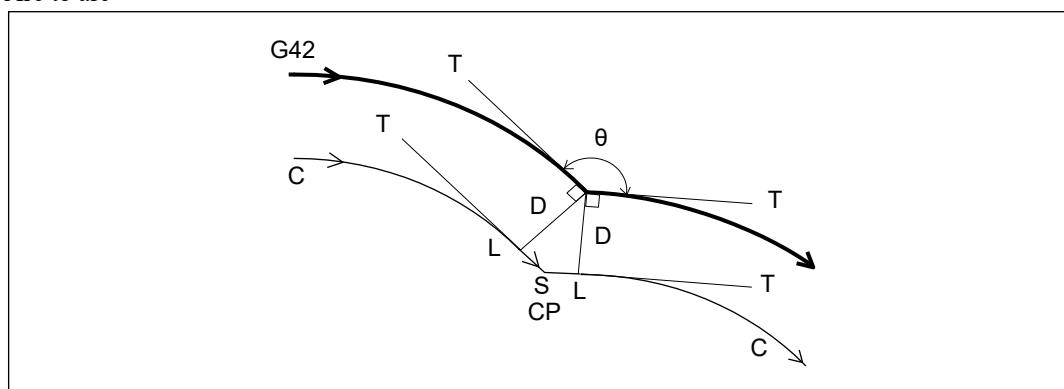


4

Arc to straight line



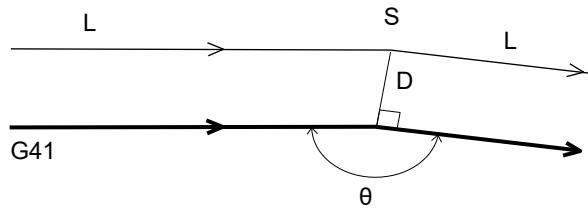
Arc to arc



Chapter 4 Preparation Function (Compensation Function)

(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

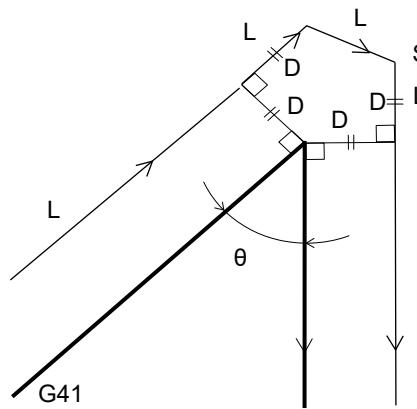


The processing is the same for: arc → straight line, straight line → arc and arc → arc.

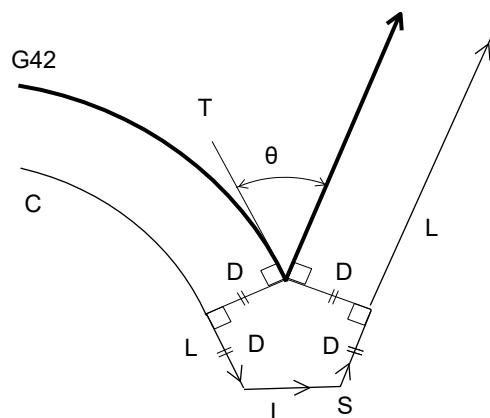
4

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

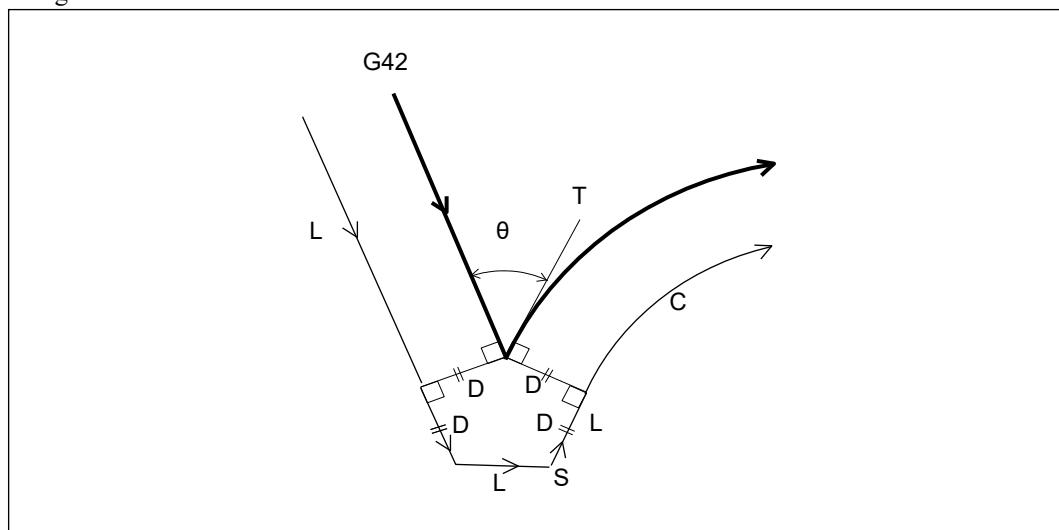
Straight line to straight line



Arc to straight line

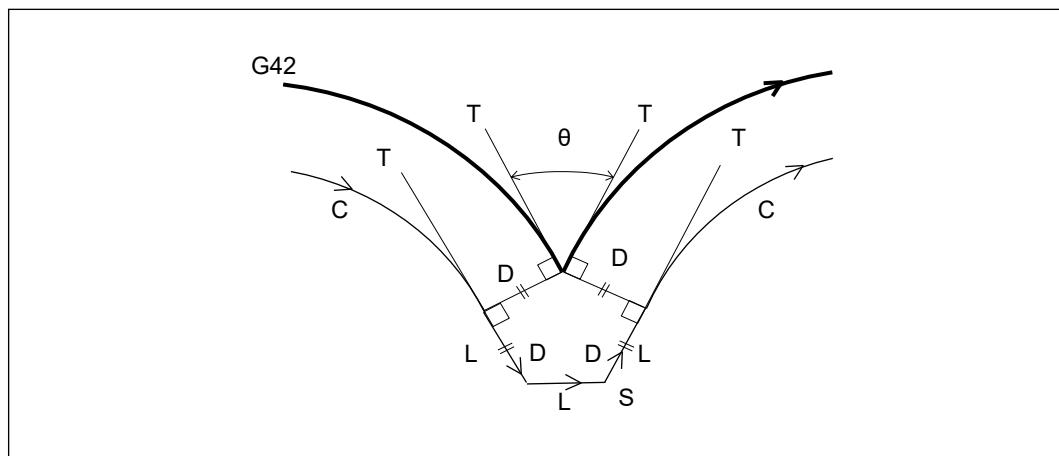


Straight line to arc



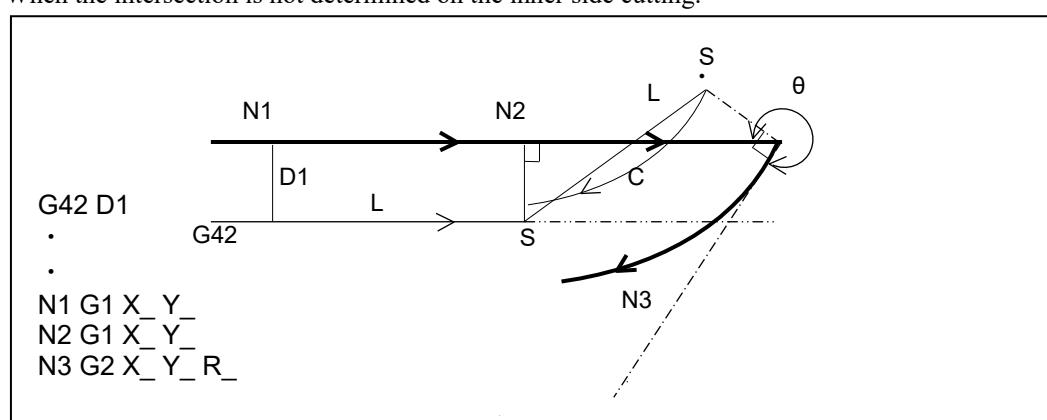
4

Arc to arc



4.1.4.4 Exceptional Cases

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



When moving to the inner side, the path offset by N2 and N3 does not intersect, so there is no intersection CP.

When the user parameter (switch 1: compensation function) <Error detection when there is no intersection during inner diameter compensation> is set to <1: Enable>, the alarm <<No intersection>> is triggered and travel stops at the end point for the previous block (N1).

When the user parameter (switch 1: compensation function) <Error detection when there is no intersection during inner diameter compensation> is set to <0: Disable>, the path shown in the figure above applies.

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.

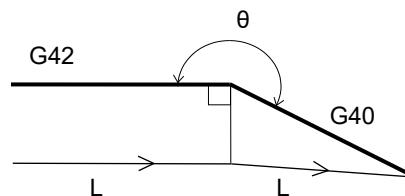
(NOTE 1) When an arc, involute interpolation or thread cutting command is issued, an alarm is triggered.

(NOTE 2) There are two setting types for the startup and cancel operations: <0: Type 1 (shortcut)> and <1: Type 2 (detour)>. Use the user parameter (switch 1: compensation function) <Start up/cancel> to set one of the types.

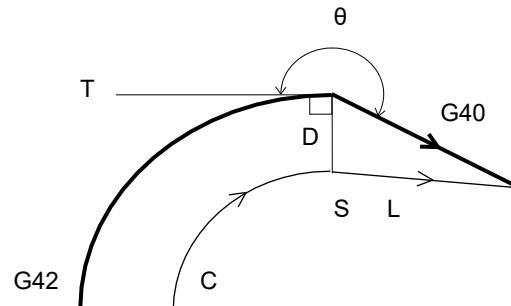
4

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

Straight line to straight line

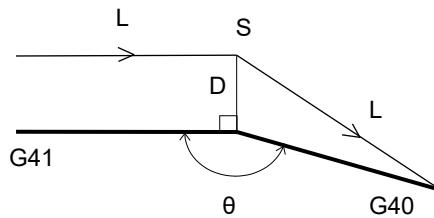


Arc to straight line

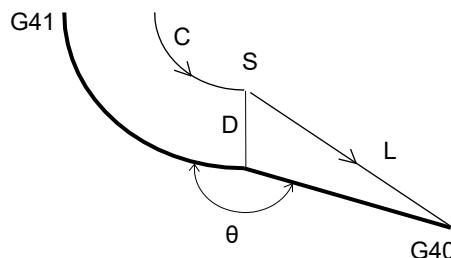


4.1.5.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)

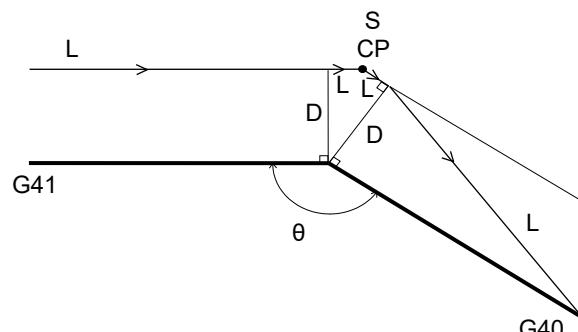
1. Type 1: Straight line to straight line



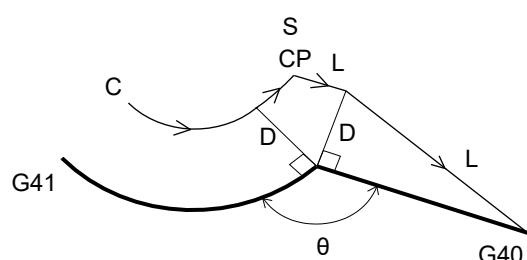
Type 1: Arc to straight line



2. Type 2: Straight line to straight line



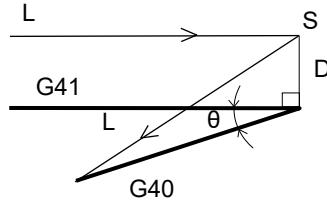
Type 2: Arc to straight line



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

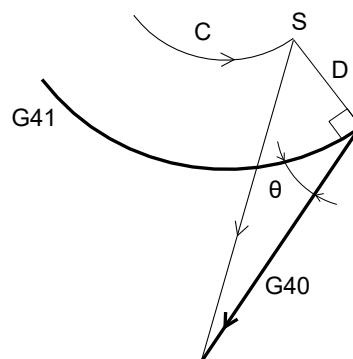
4.1.5.3 Outer Side (Acute Angle Cutting) ($\theta < 90^\circ$)

1. Type 1: Straight line to straight line

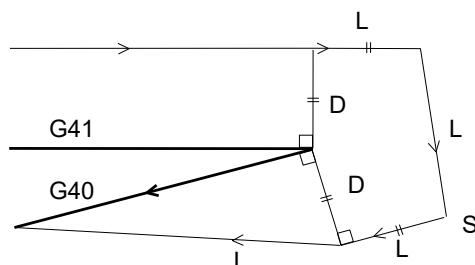


4

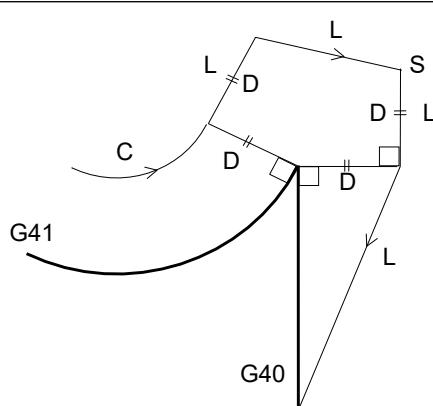
Type 1: Arc to straight line



2. Type 2: Straight line to straight line



Type 2: Arc to straight line

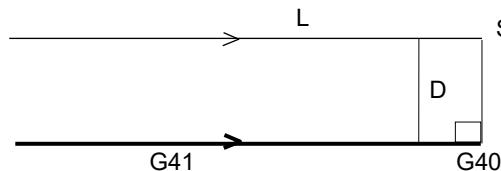


4.1.6 G40 Individual Command

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

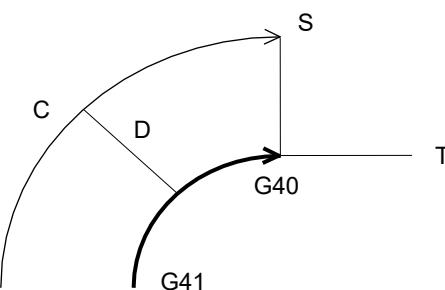
```
G41 X_Y_D_;  
G40;
```



4

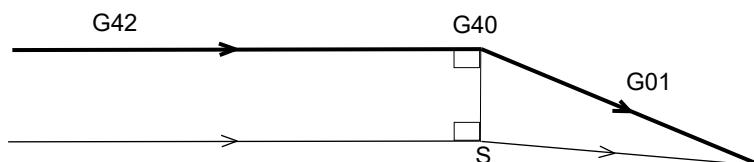
Arc to straight line

```
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				

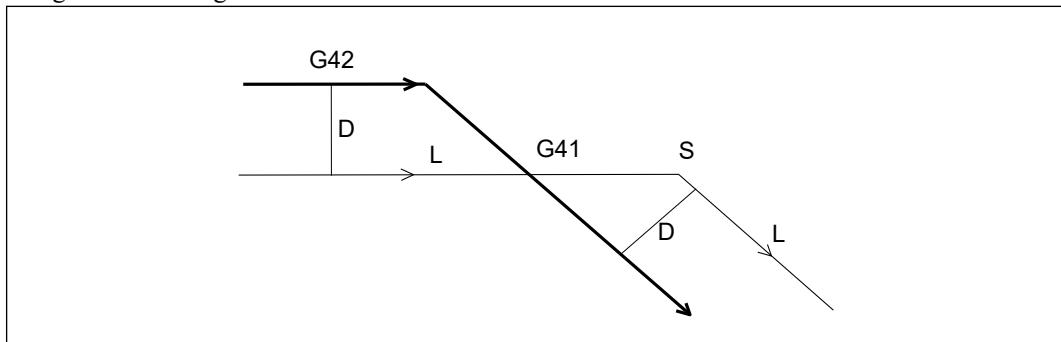
4

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.

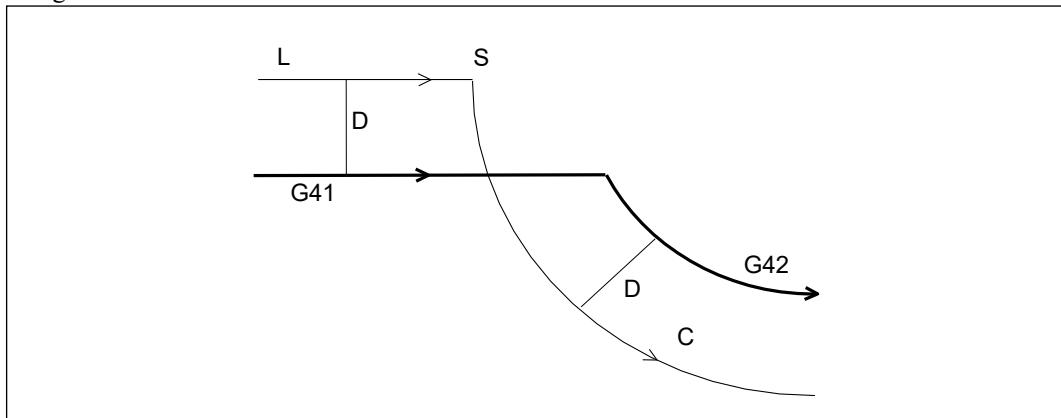
4.1.8 Offset Direction Change in Offset Mode

4.1.8.1 When There is an Intersection

Straight line to straight line

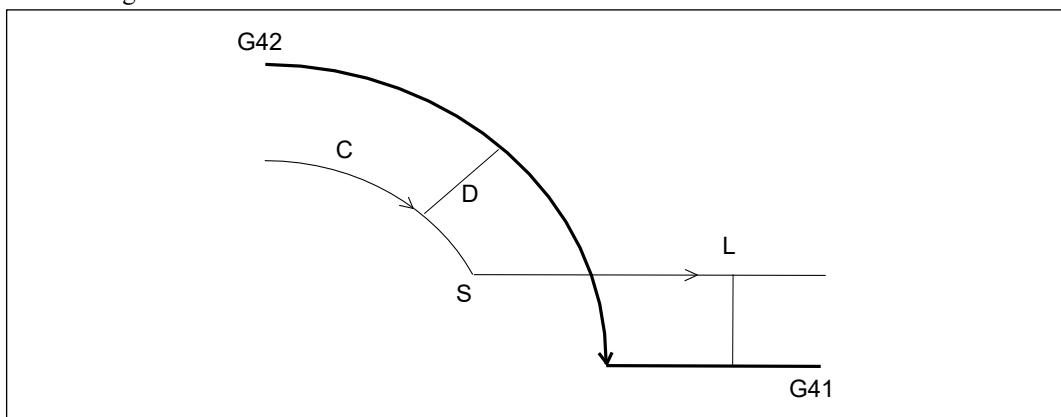


Straight line to arc

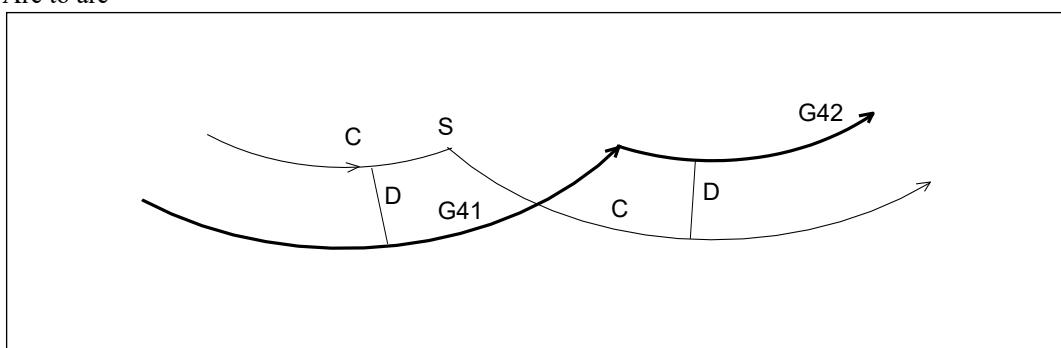


4

Arc to straight line

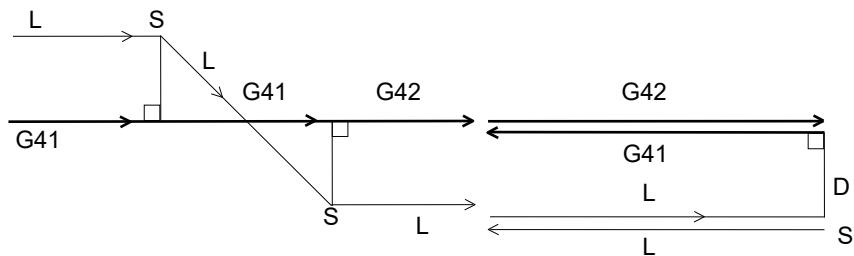


Arc to arc



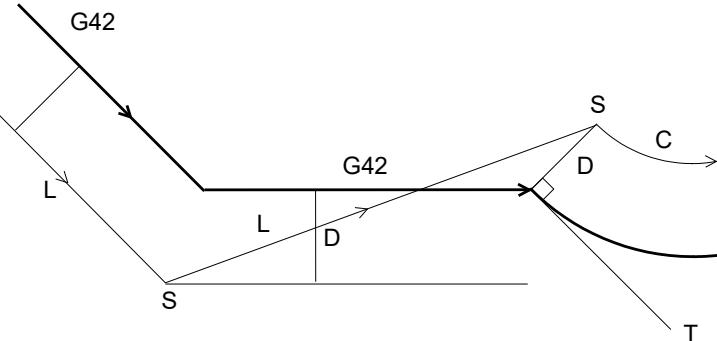
4.1.8.2 When There is No Intersection

Straight line to straight line

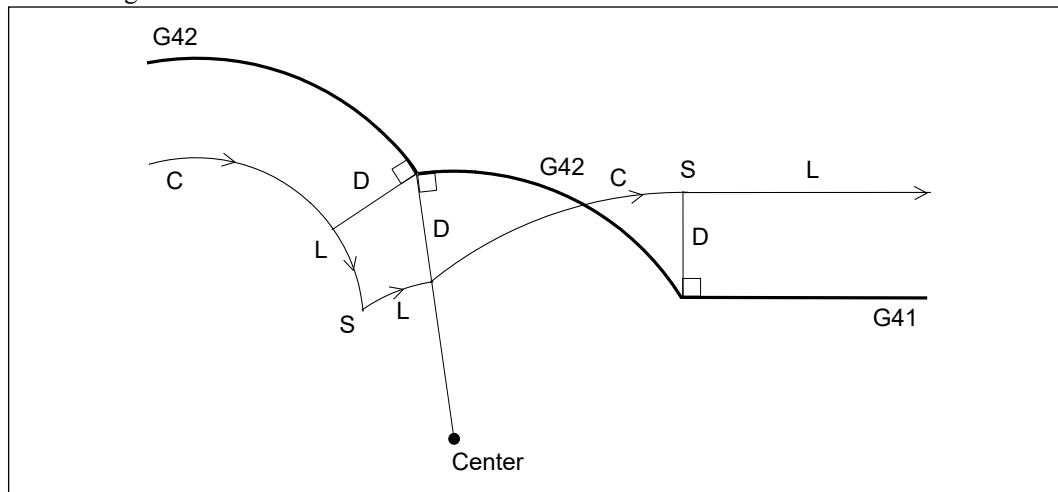


Straight line to arc

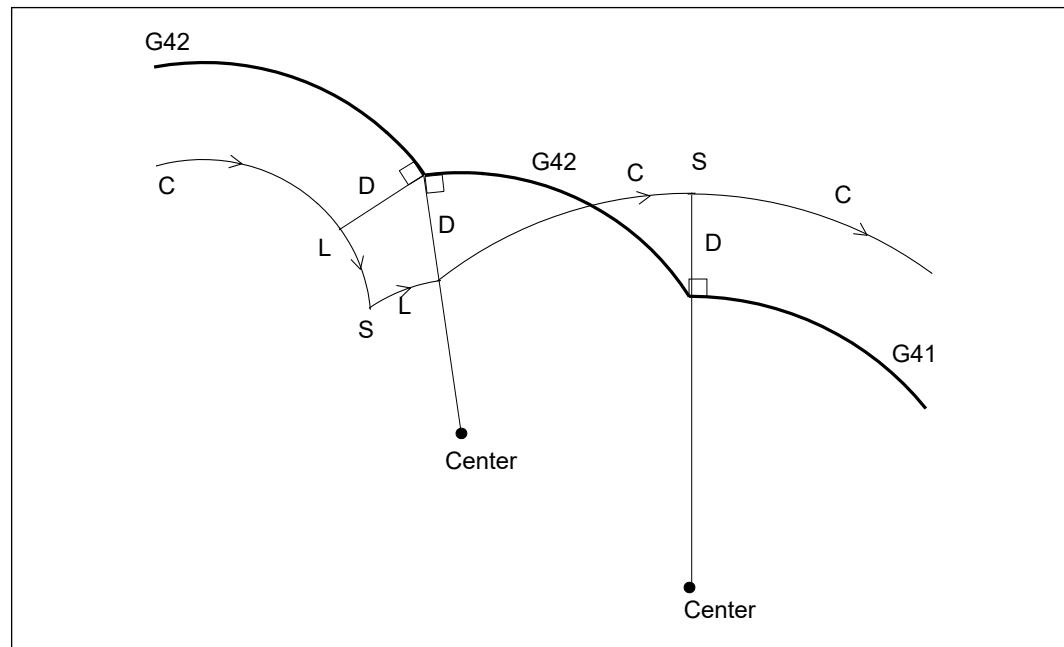
4



Arc to straight line

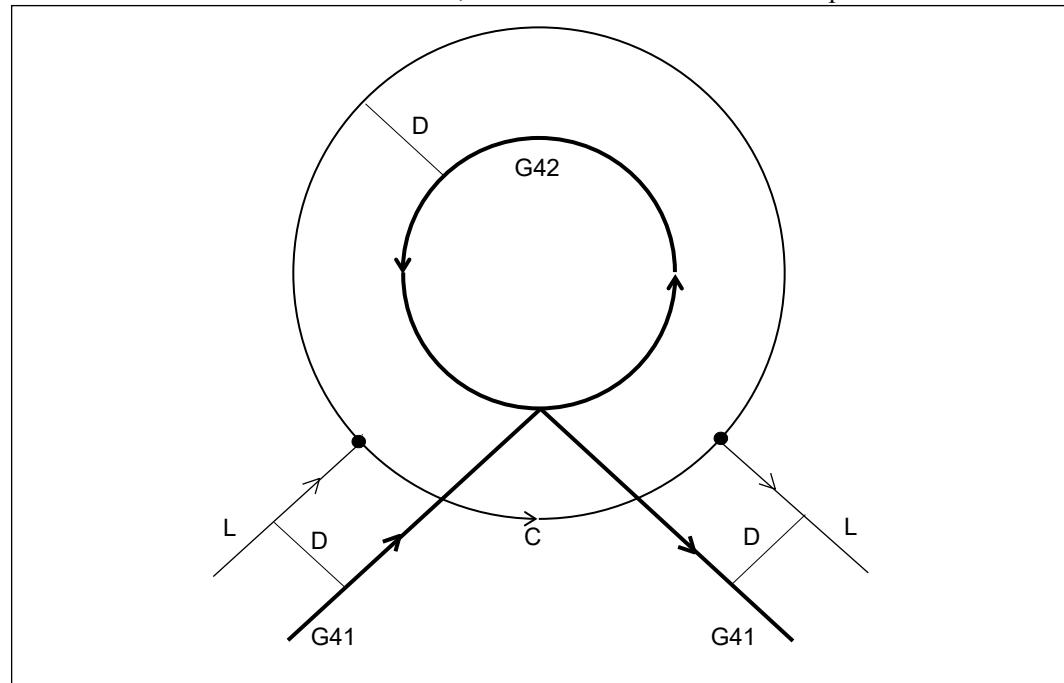


Arc to arc



4.1.8.3 When an Arc Laps Around in a Circle

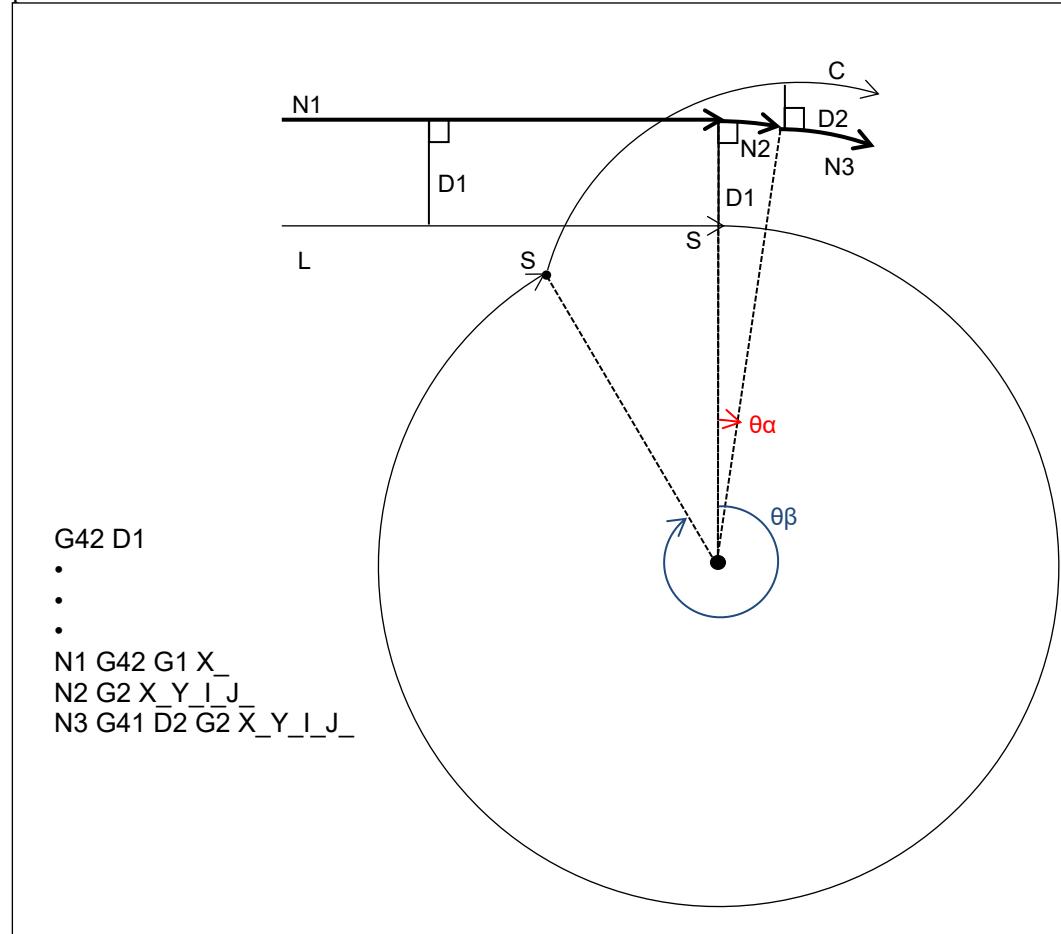
If the direction of the compensation is changed and the arc laps around in a circle, a short arc is executed as shown below. In this situation, use commands to divide the arc up.



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.

4

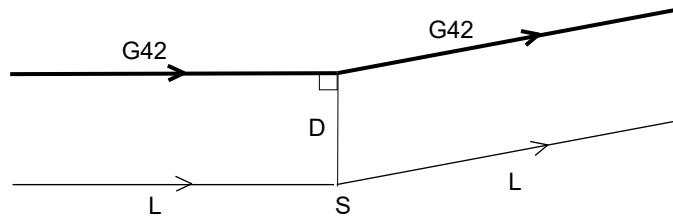


This function checks the difference between the arc angle ($\theta\alpha$) at the start and end points in the N2 program path and the arc angle ($\theta\beta$) at the start and end points in tool center path after being offset. If $\theta\beta$ is greater than $\theta\alpha$ by 180° or more, then an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

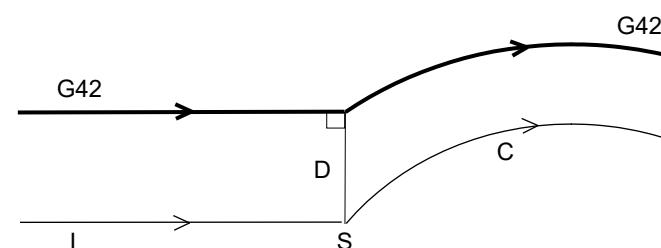
- (NOTE) This function carries out the check at the end point after 3 blocks of travel. If one of the situations below applies while the offset mode is enabled, this check function may not work properly because the tool center path start and end points change after being offset.
- When there is a command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

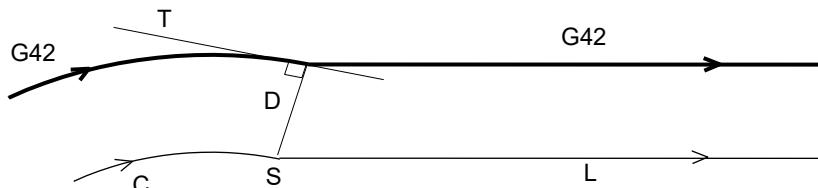


Straight line to arc

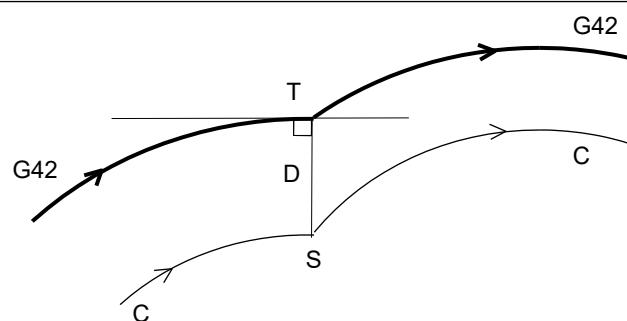


4

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.

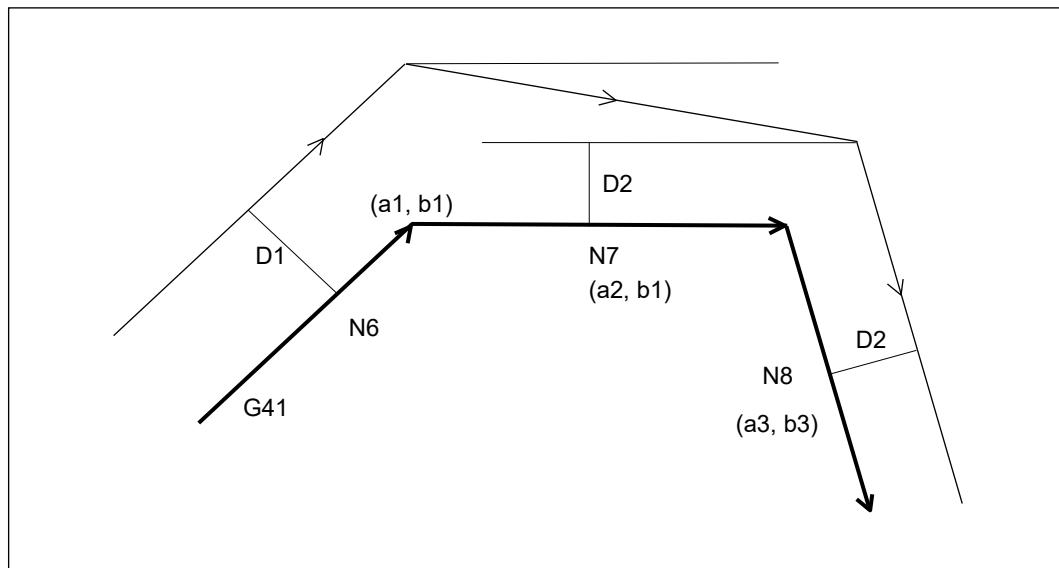
N1 G41 X_a Y_b D1;

N6 X_{a1} Y_{b1};

N7 X_{a2} D2; Offset changed

N8 X_{a3} Y_{b3};

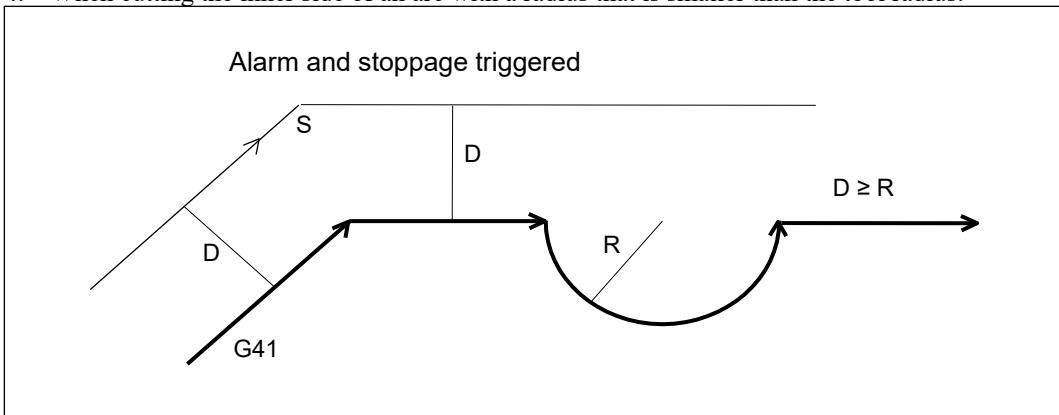
4



3. Current position display

The current position display shows the tool center position.

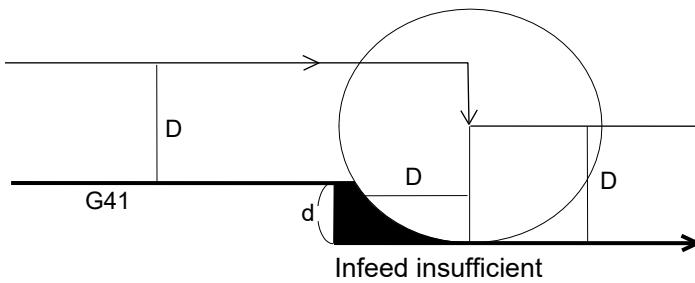
4. When cutting the inner side of an arc with a radius that is smaller than the tool radius.



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

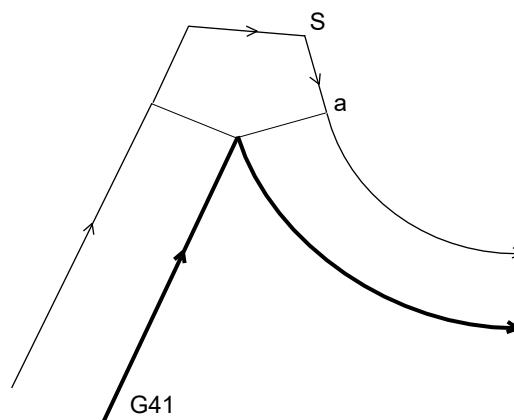
This occurs when machining a step that is smaller than the tool diameter.



6. Corner travel

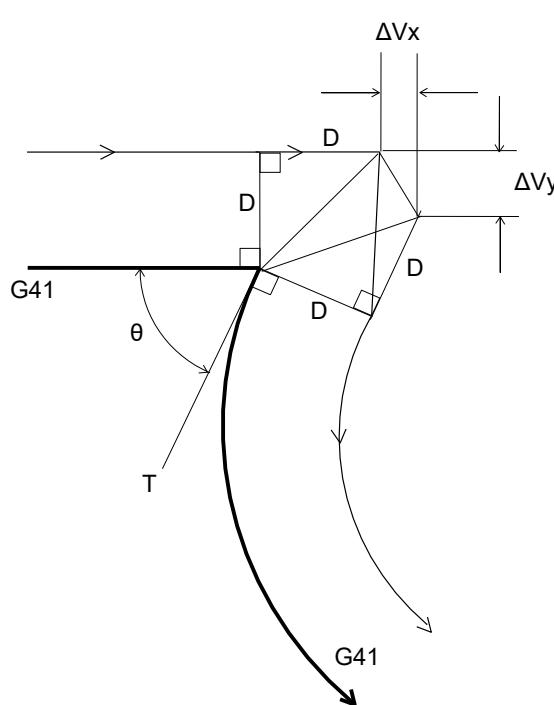
When cutting the outer side, it can turn on a corner with many angles. The travel mode and feedrate when turning the corner up to point *a* in the diagram below are based on the command that is issued for the current block.

4



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 ΔV is set in the user parameter (switch 1: compensation function) <Corner travel limit>.

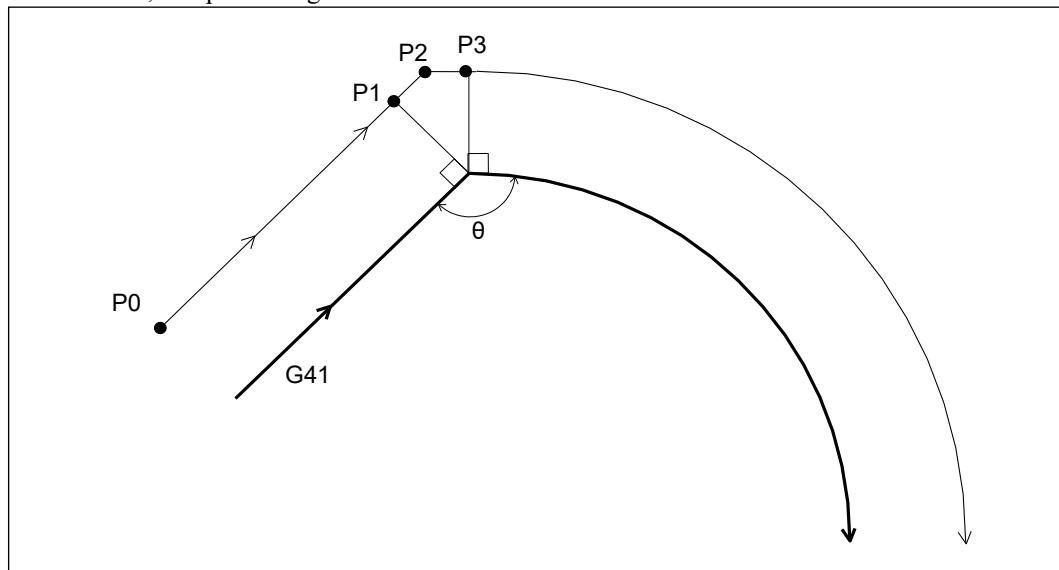


Chapter 4 Preparation Function (Compensation Function)

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.

4



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

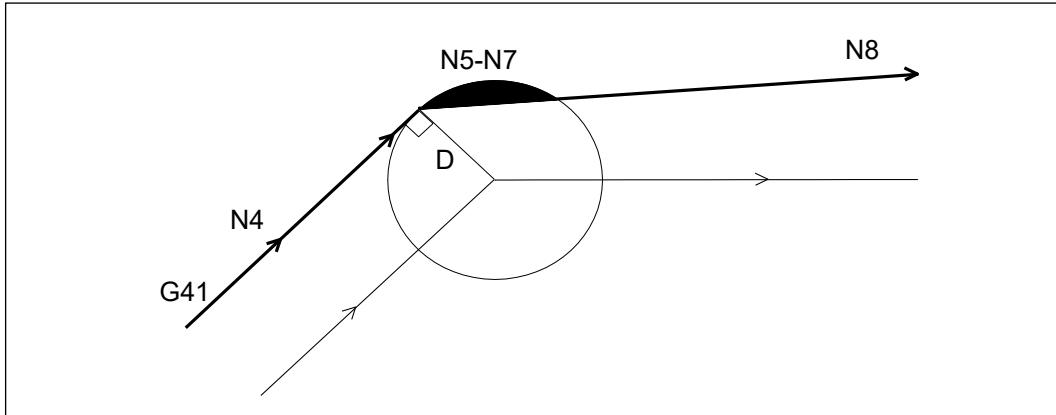
P2-P3 Travel in an arc (extremely small)

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.

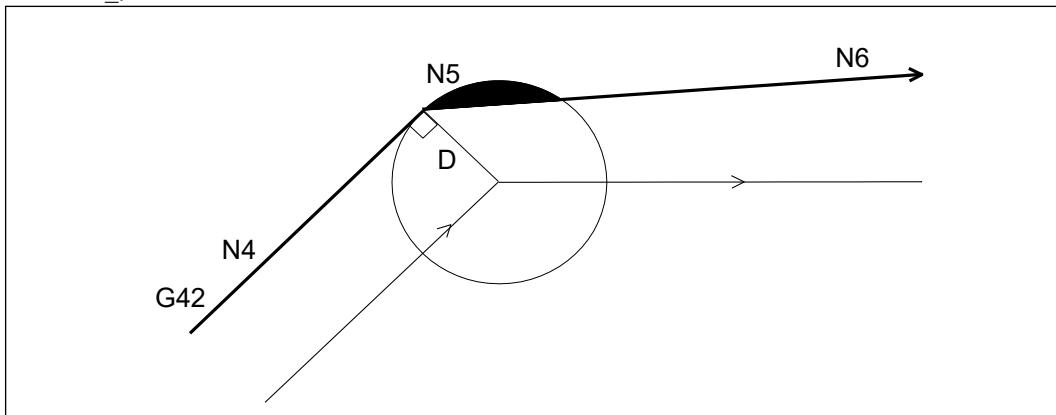
```
N4 X_Y_;
N5 Z_;
N6 F_;
N7 Z_;
N8 X_;
```



4

(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 ≠ 0).

Chapter 4 Preparation Function (Compensation Function)

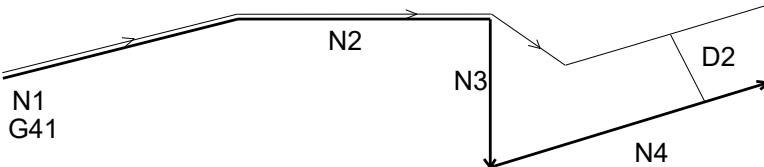
8. Tool movement when the offset for cutter compensation is 0

- (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 ≠ 0.

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



4

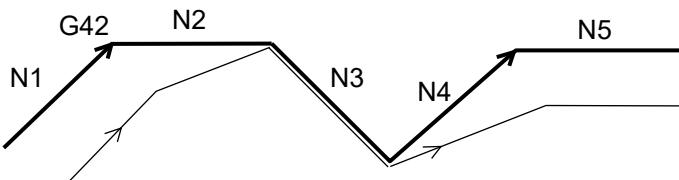
- (2) Offset mode enabled

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 ≠ 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
G277~G278	: Positioning to measurement position
G120	: Automatic measurement
G121~G129	: Skip feed
G31, G131, G132	: Change tap twist direction
G133, G134	: Pallet index
M410, M411	: Arc with 0 start point or 0 end point radius
G2, G3	: XZ circular interpolation
G102, G103	: YZ circular interpolation
G202, G203	: YZ circular interpolation

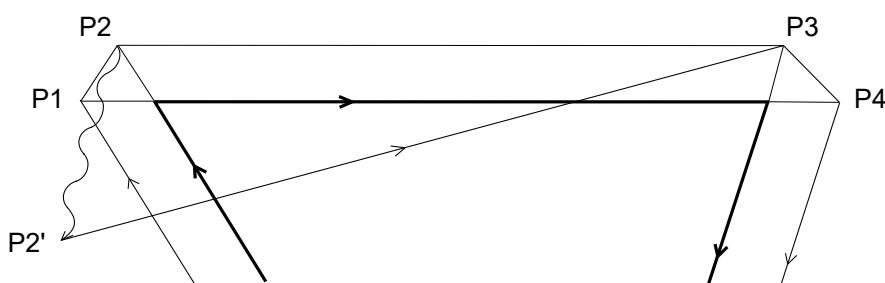
10. Input command from MDI operation

The alarm <<Specified G code cannot be used>> is triggered when there is an input related to tool 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

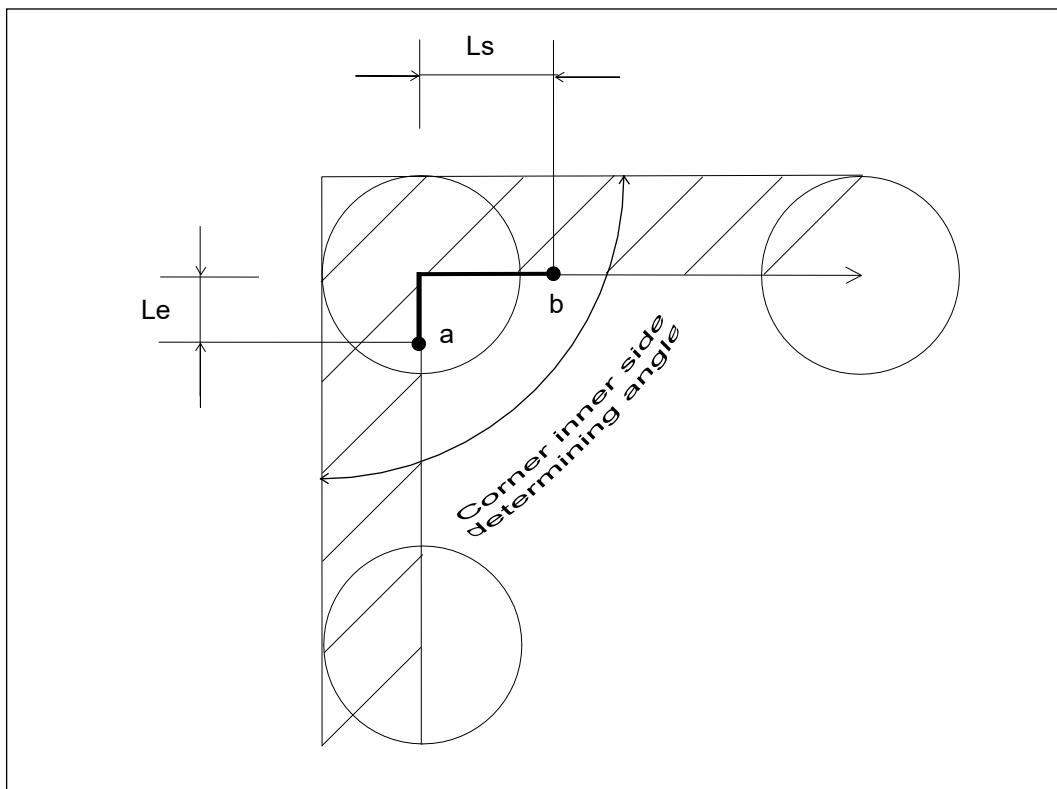
When both the block before and after the corner of the inner side meet the following conditions while in offset mode, the override function is automatically enabled in order to reduce the load on the tool.

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 (switch 1: compensation function) <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 (switch 1: compensation function) settings.

- (1) Automatic corner override (length 1) : Corner end point deceleration distance Le
- (2) Automatic corner override (length 2) : Corner start point deceleration distance Ls
- (3) Automatic corner override (ratio) : Deceleration ratio (%) Y
- (4) Automatic corner override (angle) : Corner inner side determining angle θ

4



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

$$\text{Actual feedrate} = \text{Command speed} \times \frac{\text{Deceleration ratio}}{100}$$

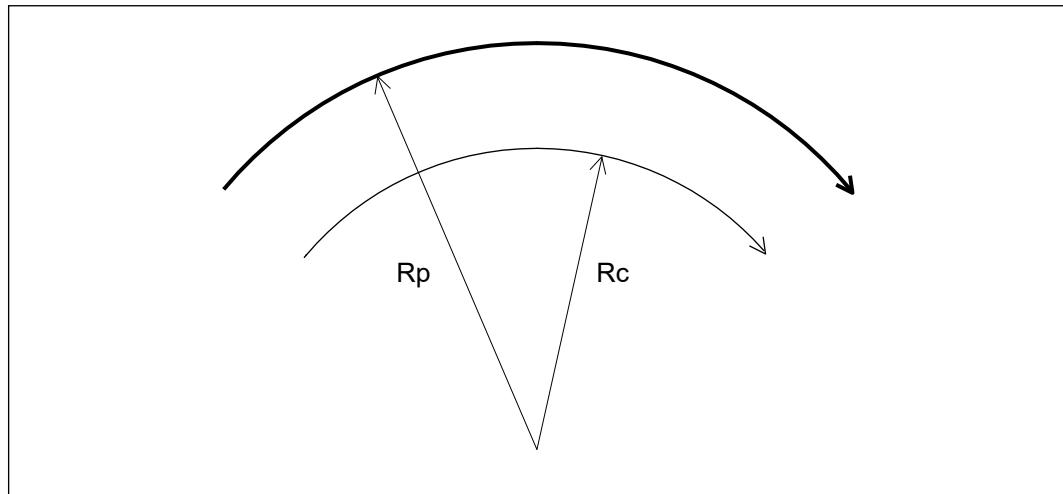
4.1.11.2 Inner Arc Override

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

$$\text{Actual feedrate} = \text{Command speed} \times \frac{R_c}{R_p}$$

R_p: Program radius

R_c: Tool center path radius



4

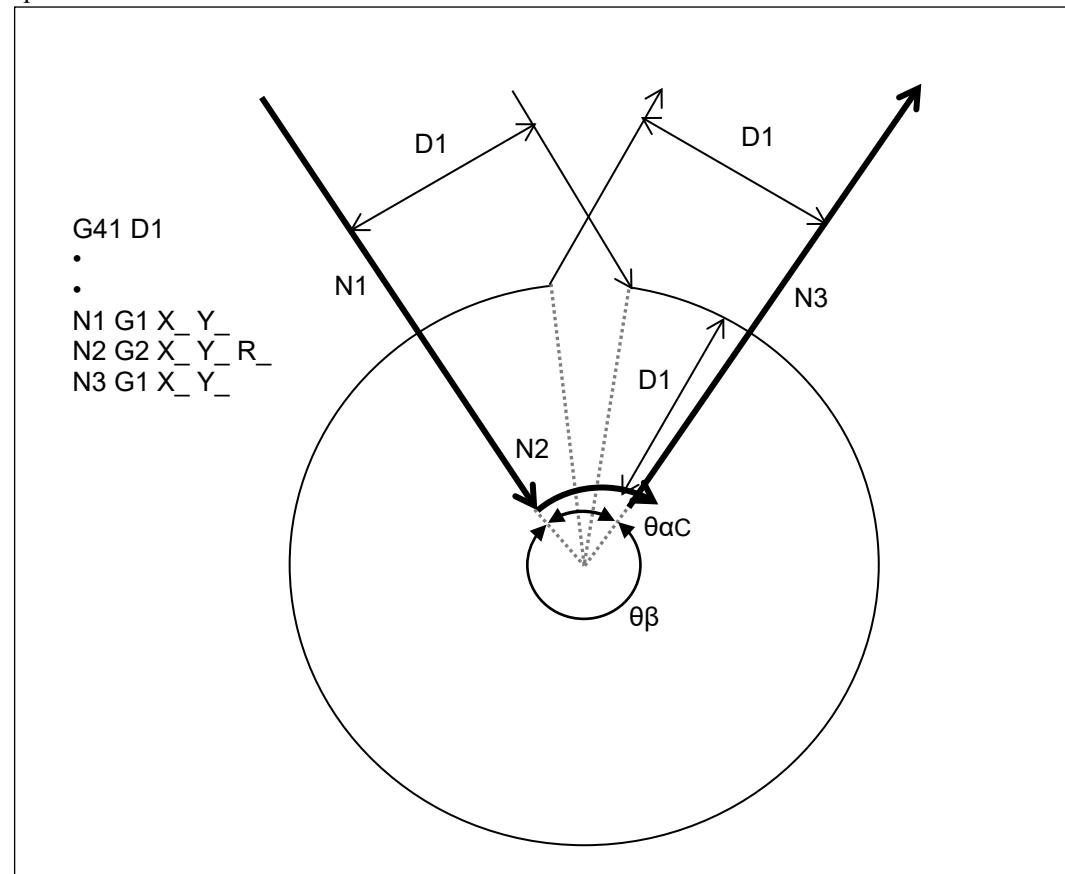
- (NOTE 1) When $\frac{R_c}{R_p}$ is less than the <Inner arc override limit> that is set in the user parameter (switch 1: compensation function), that parameter value is multiplied as an alternative to $\frac{R_c}{R_p}$.

$$\text{Actual feedrate} = \text{Command speed} \times \frac{\text{Inner arc override limit}}{100}$$

- (NOTE 2) Refer to “Chapter 9 (6) Involute interpolation function” in the Operation Manual II 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.



4

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°, 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

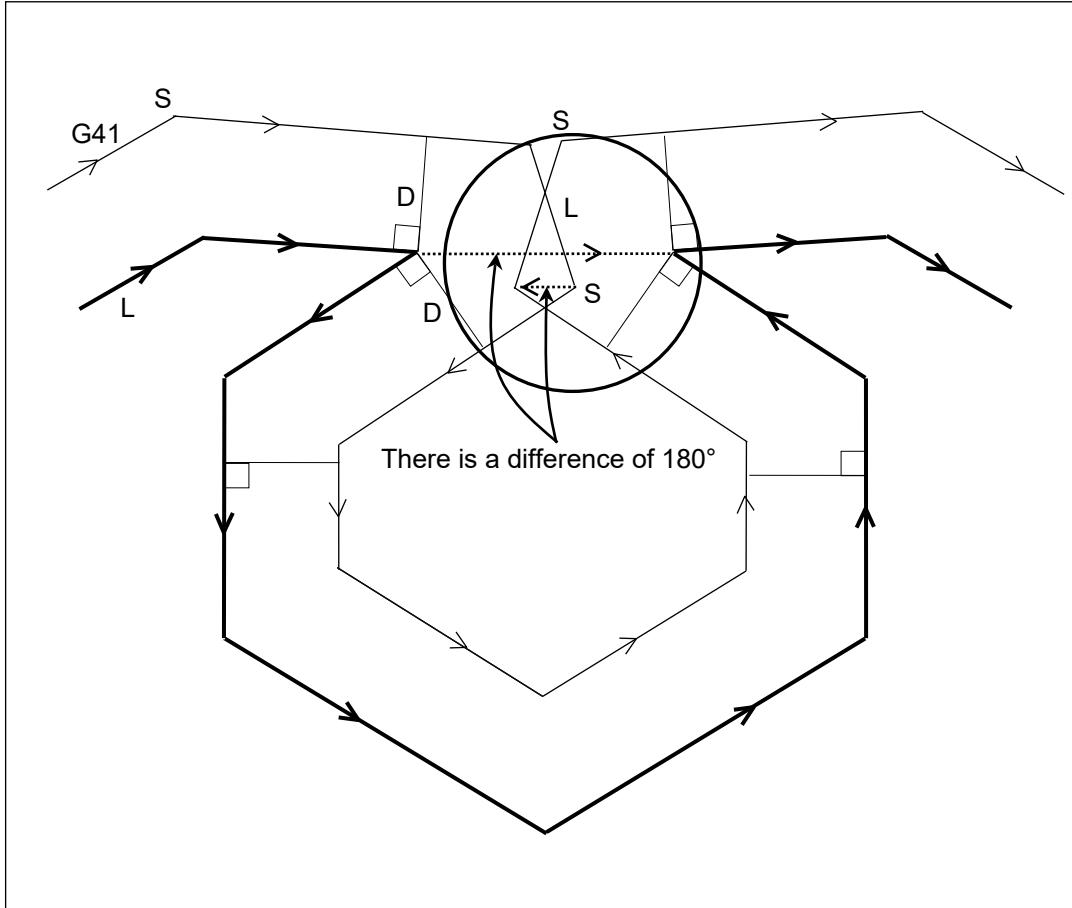
The cutter compensation function checks between the offset vectors that occur at the end point of the block for the program path.

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° and less than 270° .

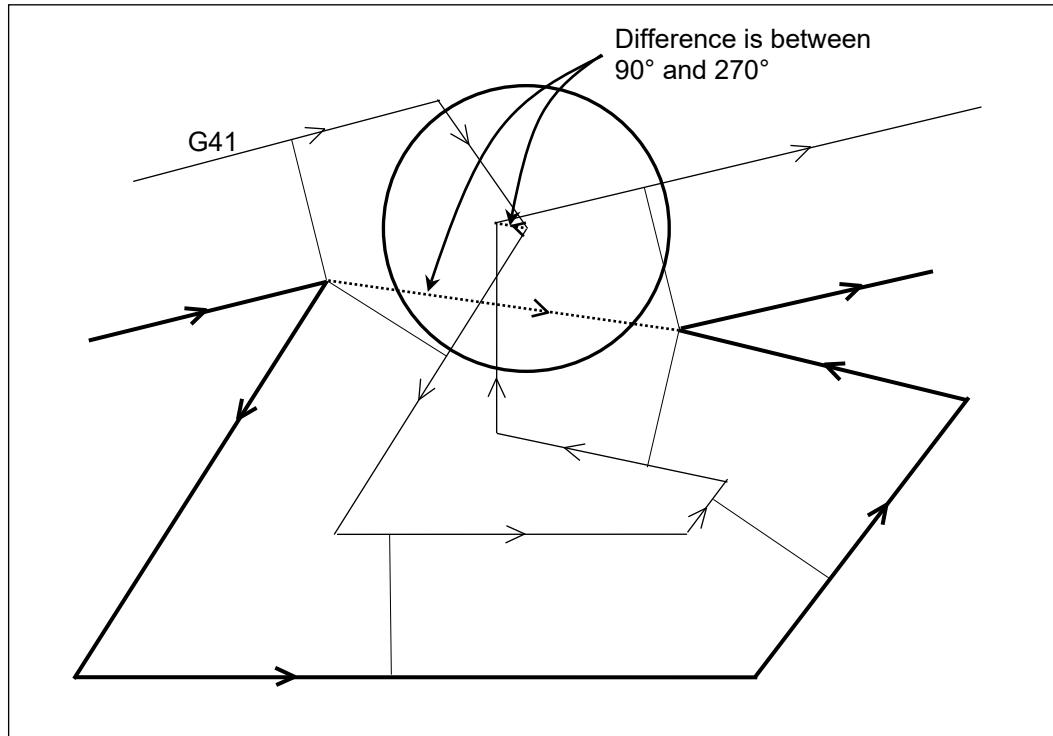
When there is a block without travelling included, the interference check range becomes smaller.

Example when interference is detected



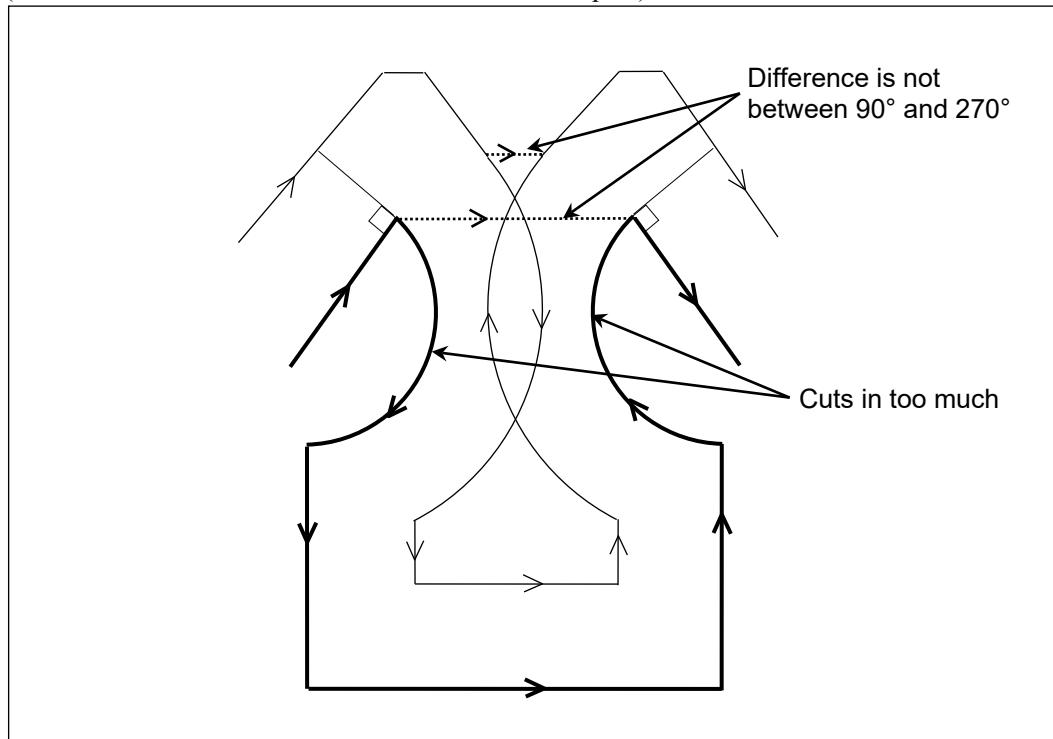
Chapter 4 Preparation Function (Compensation Function)

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)



4

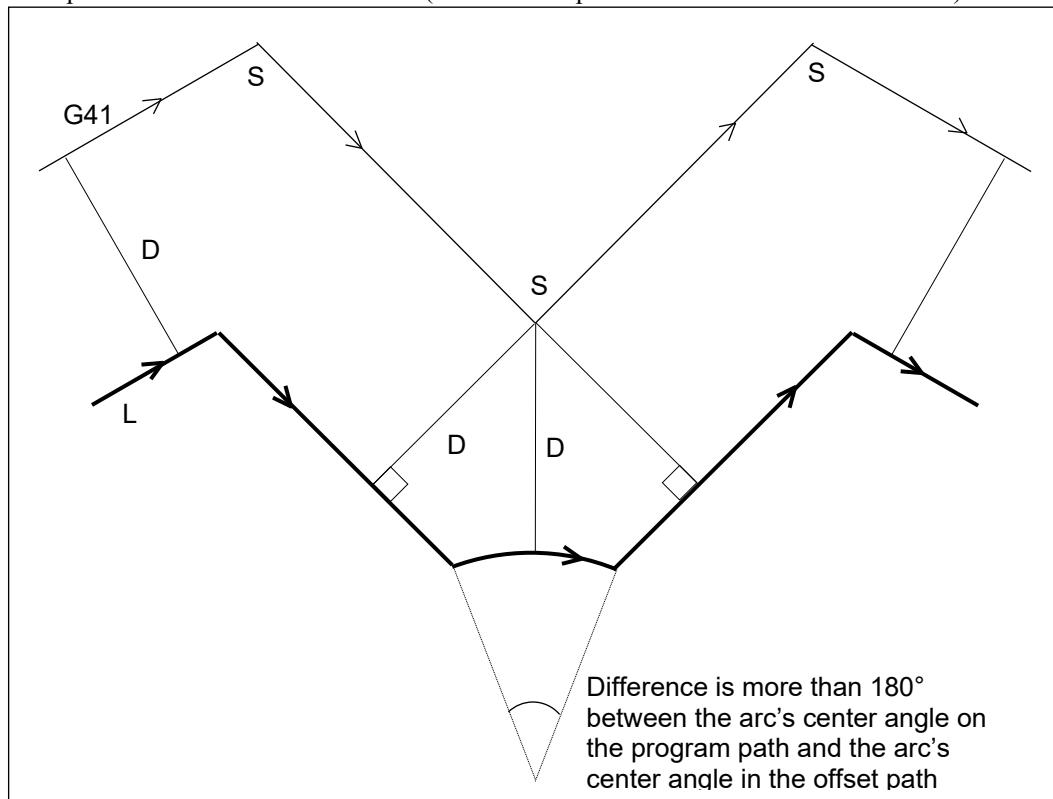
Example when interference is not detected but there is actually interference
(Interference is not detected in the middle of the offset path)



2. Arc angle check

When there is an arc, this check detects interference if the difference is more than 180° 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

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 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, 201 to 299), 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.

4

2. Tool length offset (-)

Command format

G44 Hn;

Hn : Tool number (n = 0 to 99, 201 to 299), 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.
- (NOTE 8) When a TCP control (G43.4/G43.5) command is issued during tool length offset, the tool length offset is cancelled and the TCP control turns ON. In addition, when a tool length offset (G43/G44) command is issued while under TCP control, then the TCP control is cancelled and the tool length offset is enabled. Refer to the “14.2.5.3 Cancelled by other commands” for further details.

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 (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change>	0: Type 1		1: Type 2
User parameter (switch 1: compensation function) <Error check when traveling during tool length/tool position offset cancel>	0: Check	1: 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 (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change>	0: Type 1		1: Type 2
User parameter (switch 1: compensation function) <Error check when traveling during tool length/tool position offset cancel>	0: Check	1: 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.

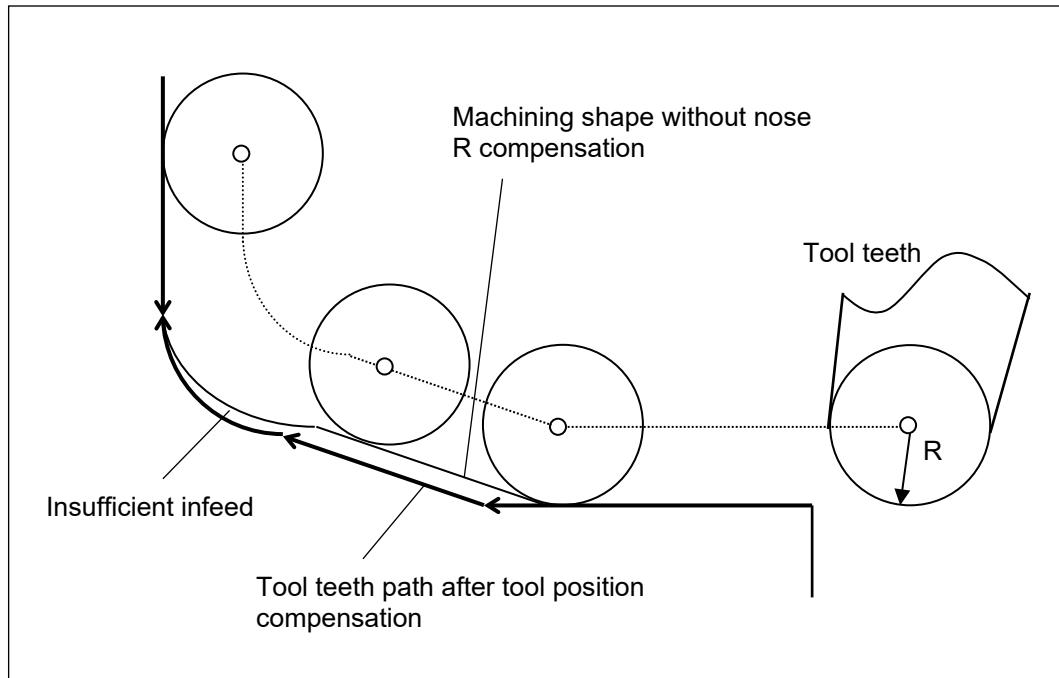
Ex:

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

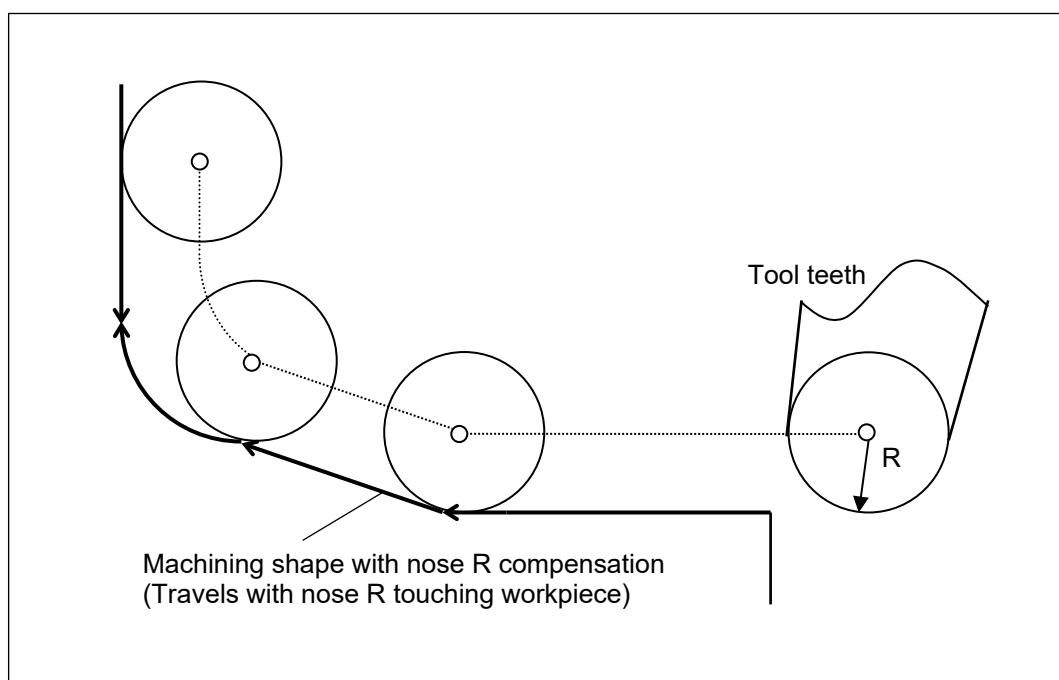
4.3 Nose R Compensation (G141 and G142 - Option)

* 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



4.3.1 Command Format

Command format

G141
G142
Dn;

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, 201 to 299), 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.

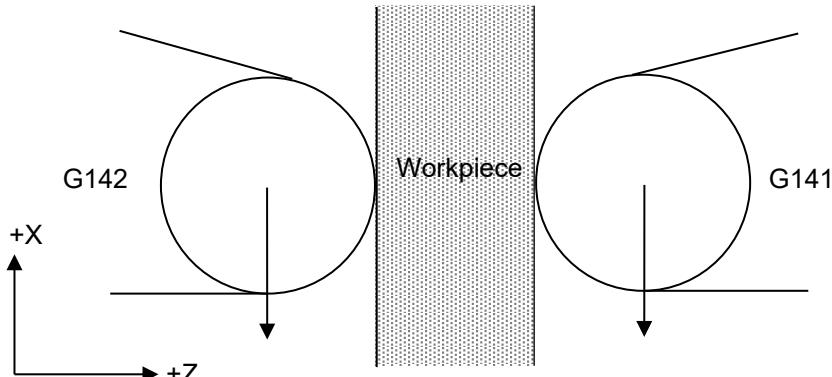
If either the command G141 or G142 is issued, the nose R compensation mode is enabled. This mode is cancelled by G40.

4

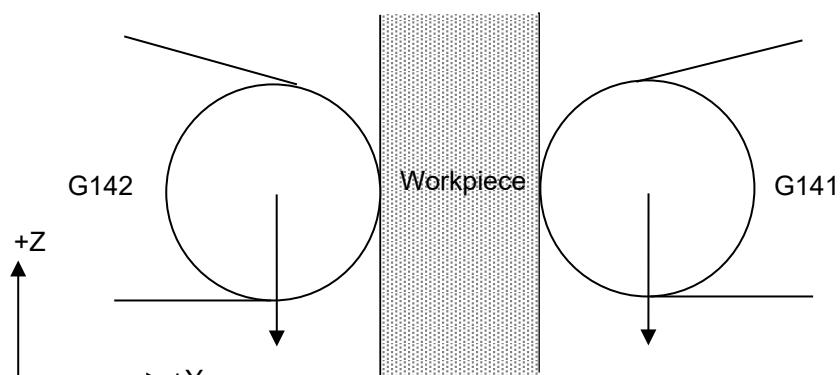
(NOTE) When a nose R compensation command (G141 and G142) is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered.

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

G18 modal

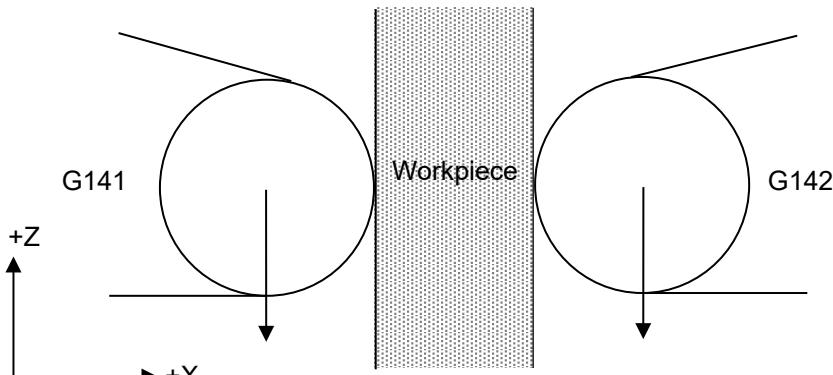


G19 modal



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

G18 modal



4

- (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

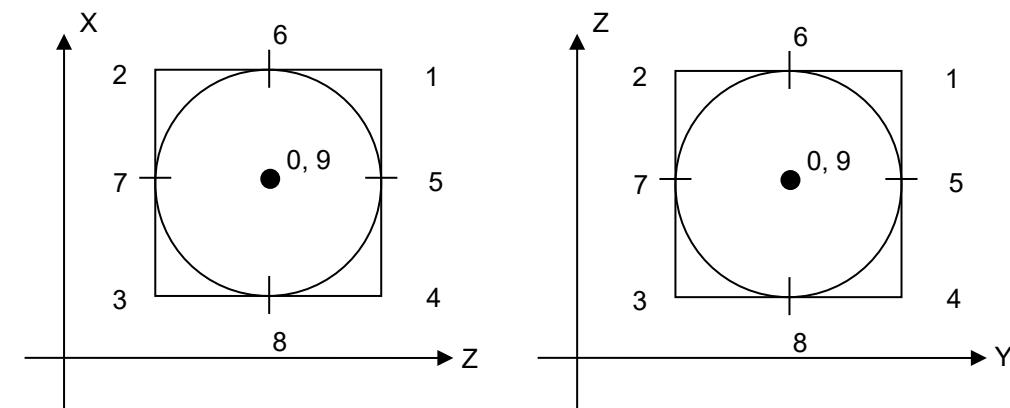
When a G141 or G142 command is issued in the program, the nose R wear offset is added to the nose R compensation on the tool number for the command. The nose R wear offset can be set on the tool list screen.

$$\text{Nose R offset} = \text{Nose R compensation} + \text{Nose R wear offset}$$

- (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.

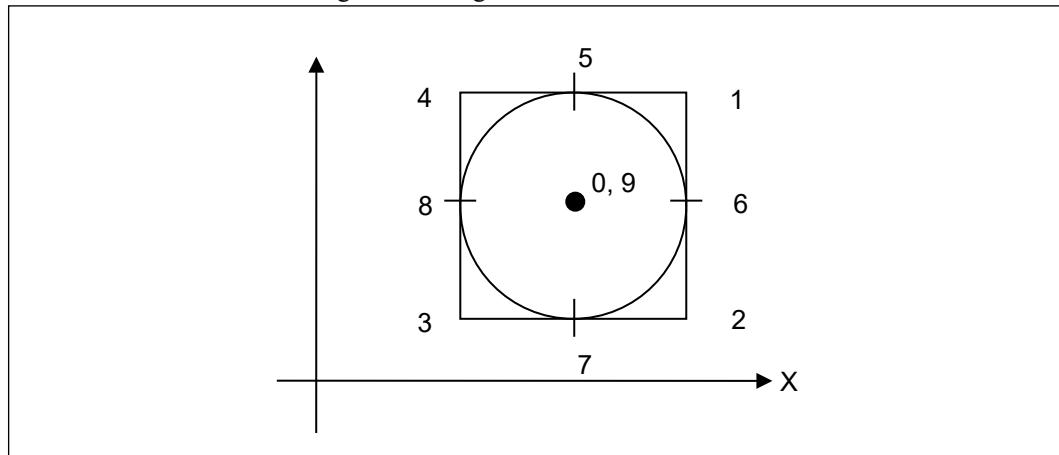
4.3.2 Virtual Teeth

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.



4

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.

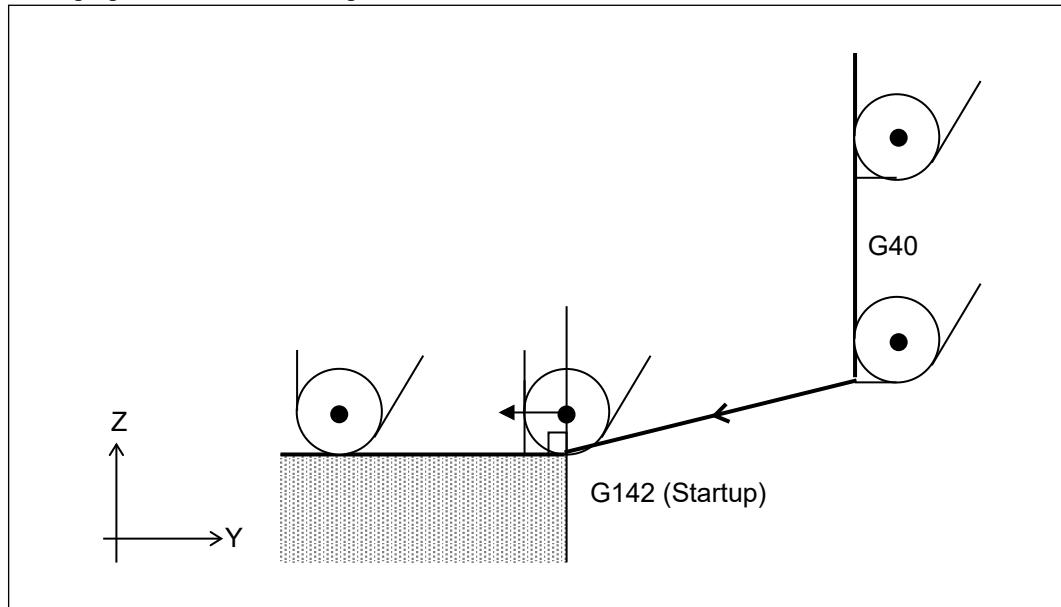
Normally, the tool path that is programmed matches the path of the virtual teeth.

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.



4

(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 <Start up/cancel> to set one of the types.

Chapter 4 Preparation Function (Compensation Function)

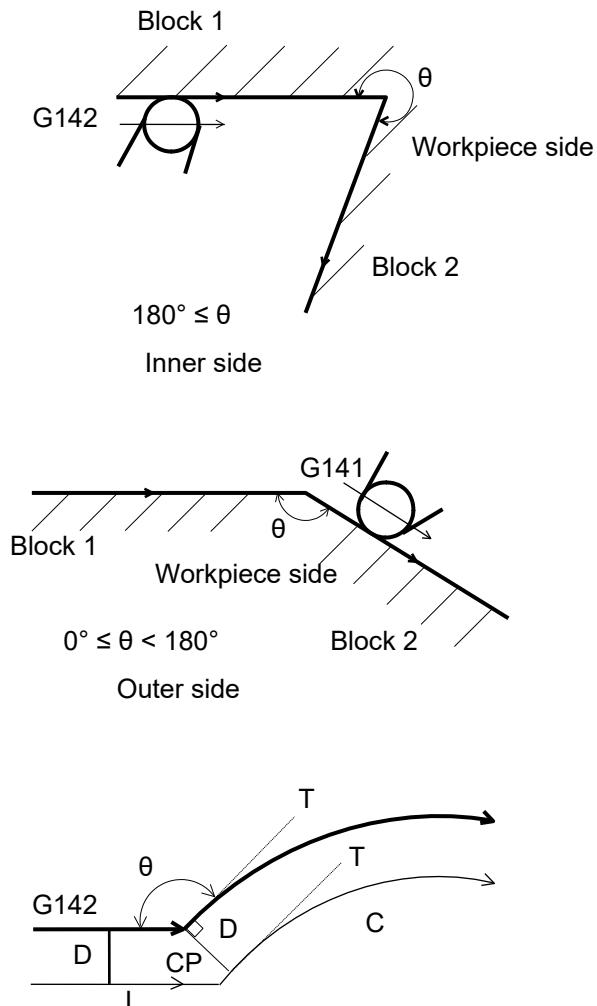
The following explanation applies to the Y- and Z-planes. When considering the X- and Z-planes if looking from the front of the machine, the compensation direction is reversed (G141 → G142, G142 → G141).

<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° on the workpiece side. Outer side means the angle measures between 0 and 180° .

4

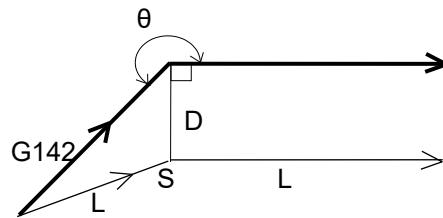


2. Explanation of symbols in diagram

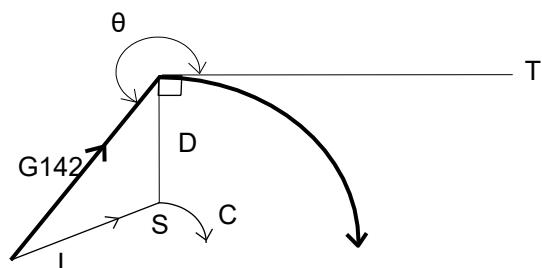
—	:	Program path
— — —	:	Nose R center path
- - -	:	Auxiliary line
L	:	Straight line
C	:	Arc
D	:	Nose R radius
θ	:	Angle on workpiece side
T	:	Arc tangent line
CP	:	Intersection
S	:	Single block stop point

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

Straight line to straight line

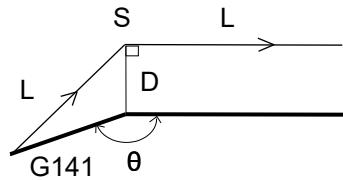


Straight line to arc



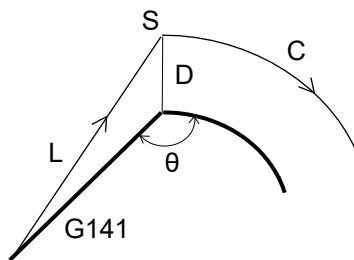
4.3.4.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)

- Type 1: Straight line to straight line

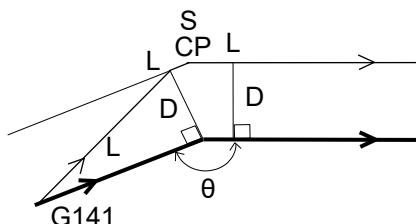


4

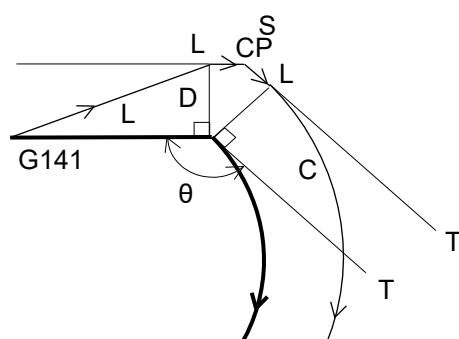
- Type 1: Straight line to arc



- Type 2: Straight line to straight line

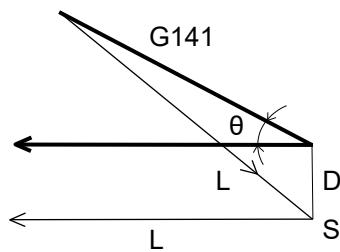


- Type 2: Straight line to arc

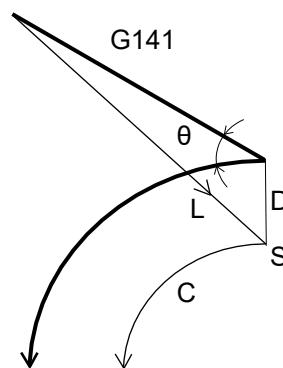


4.3.4.3 Outer Side (Acute Angle Cutting) ($\theta < 90^\circ$)

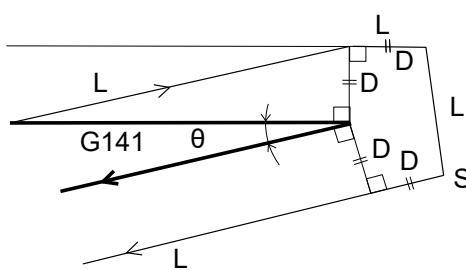
1. Type 1: Straight line to straight line



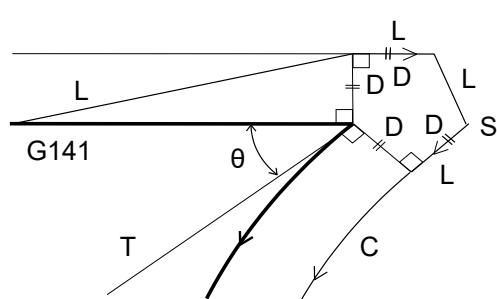
Type 1: Straight line to arc



2. Type 2: Straight line to straight line



Type 2: Straight line to arc



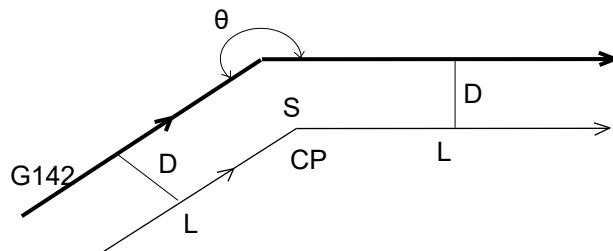
(NOTE) When $\theta \leq 1^\circ$, the setting <0: Type 1 (shortcut)> is used or enabled, even if <1: Type 2 (detour)> is specified for the user parameter <Start up/cancel>.

4.3.5 Offset Mode

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

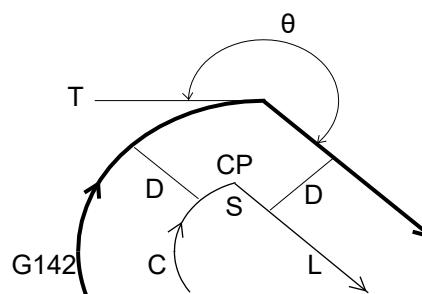
4.3.5.1 Inner Side Cutting ($180^\circ \leq \theta$)

Straight line to straight line

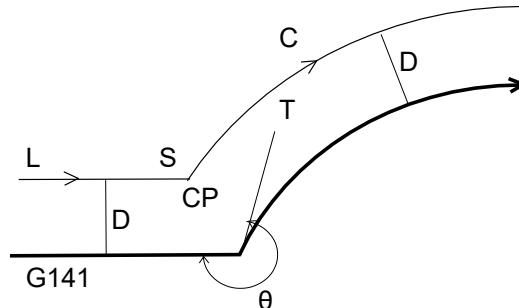


4

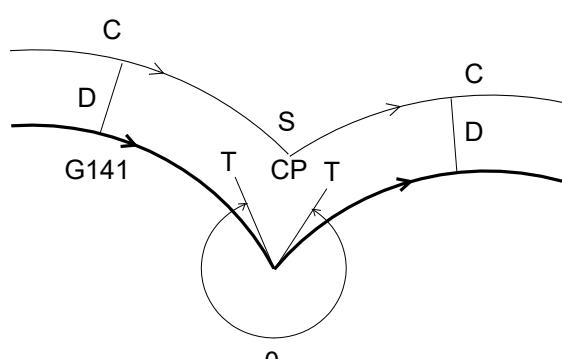
Arc to straight line



Straight line to arc

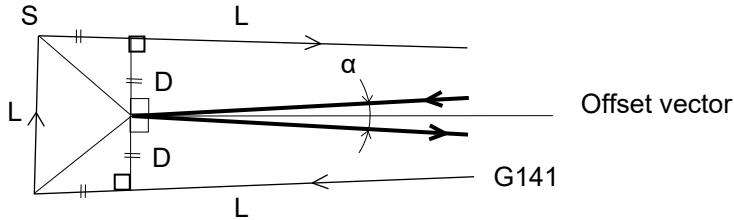


Arc to arc



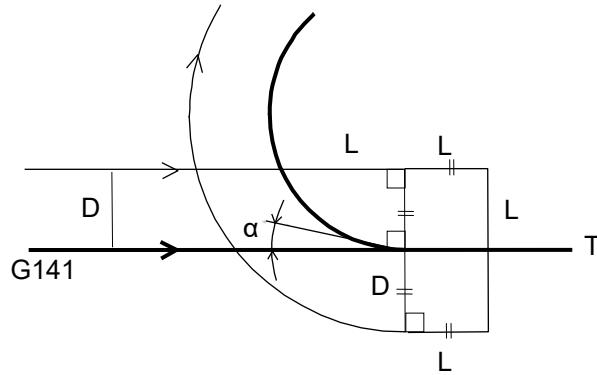
(NOTE 1) When turning on the inner side of a narrow angle ($\alpha < 1^\circ$), and the offset vector is abnormally large.

Straight line to straight line



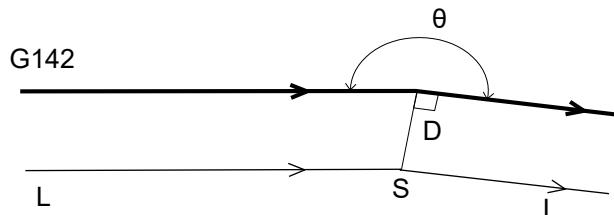
Straight line to arc

4



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

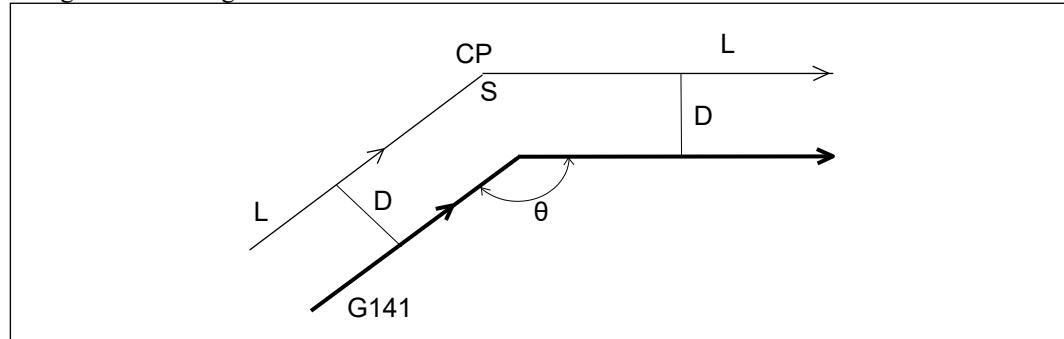
Straight line to straight line



The processing is the same for: arc → straight line, straight line → arc and arc → arc.

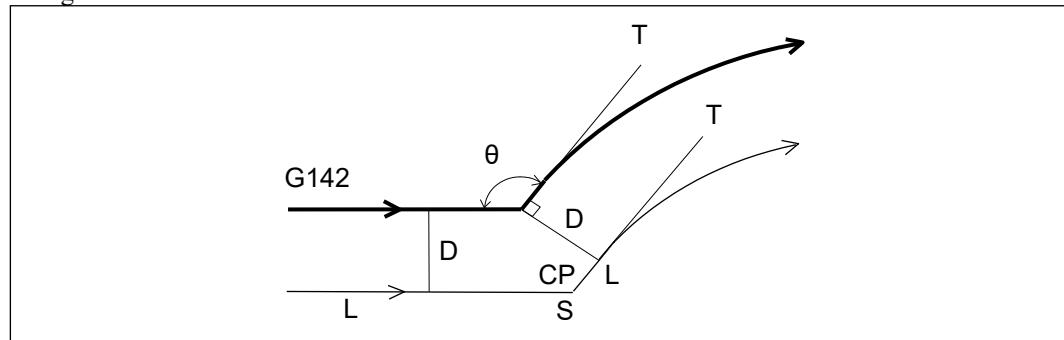
4.3.5.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)

Straight line to straight line

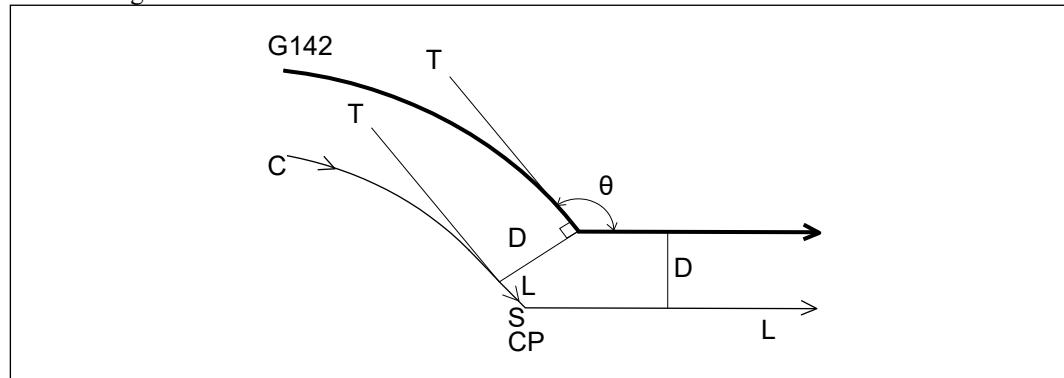


Straight line to arc

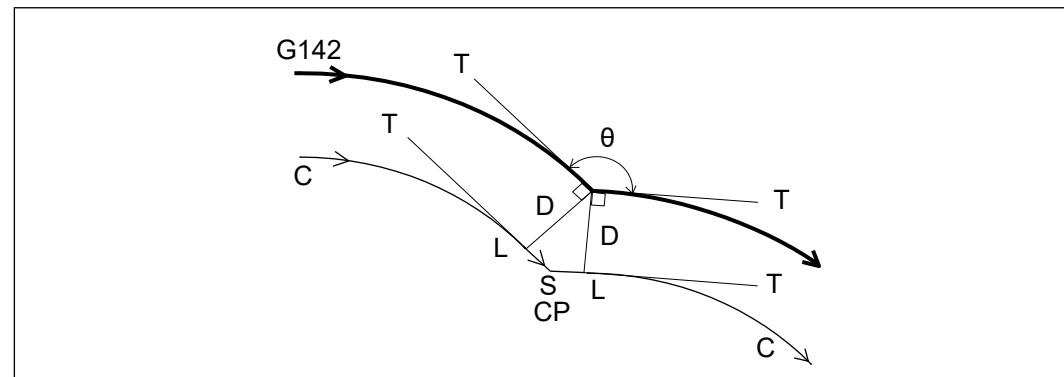
4



Arc to straight line

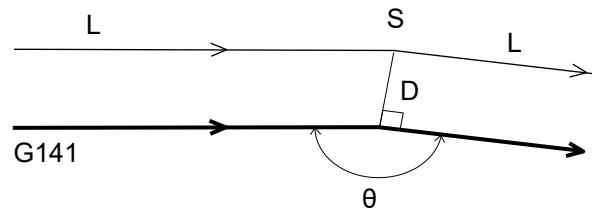


Arc to arc



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

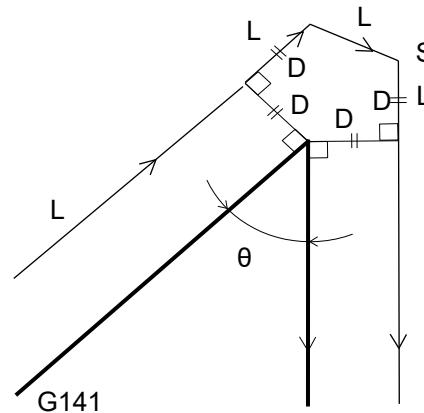
Straight line to straight line



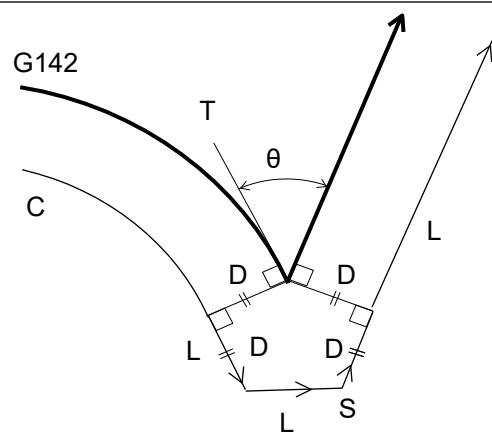
The processing is the same for: arc → straight line, straight line → arc and arc → arc.

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

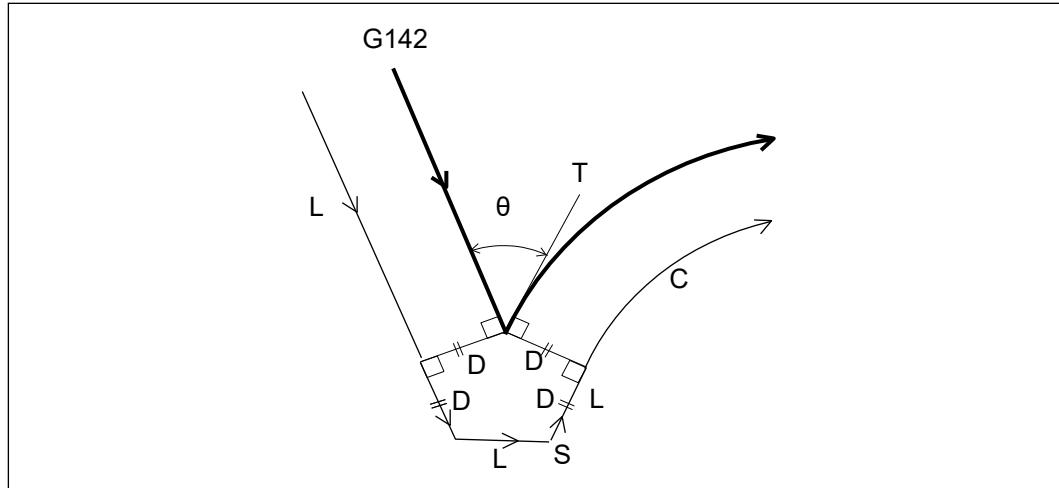
Straight line to straight line



Arc to straight line

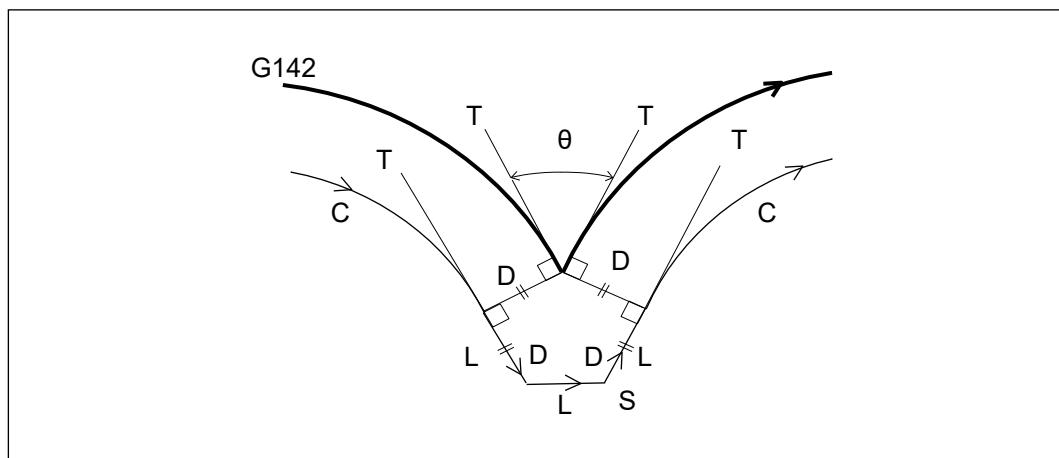


Straight line to arc



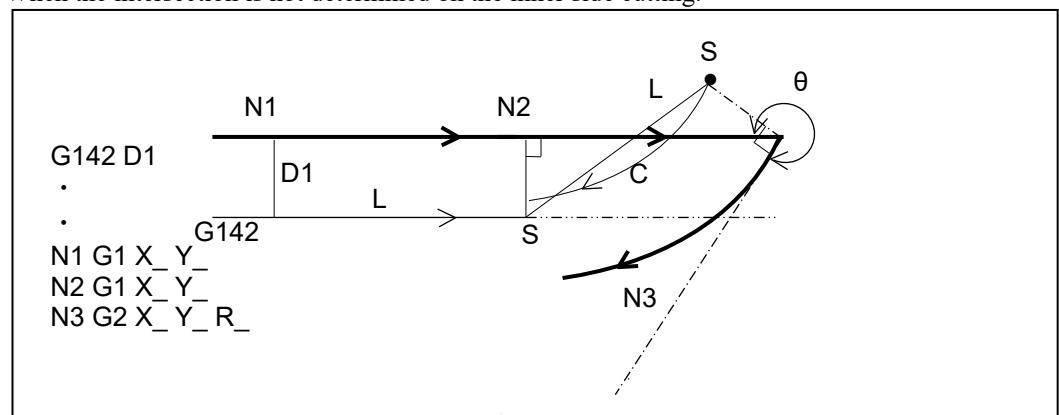
4

Arc to arc



4.3.5.4 Exceptional Cases

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



When moving to the inner side, the path offset by N2 and N3 does not intersect, so there is no intersection CP.

When the user parameter (switch 1: compensation function) <Error detection when there is no intersection during inner diameter compensation> is set to <1: Enable>, the alarm <>No intersection<> is triggered and travel stops at the end point for the previous block (N1).

When the user parameter (switch 1: compensation function) <Error detection when there is no intersection during inner diameter compensation> is set to <0: Disable>, the path shown in the figure above applies.

4.3.6 Offset Cancel

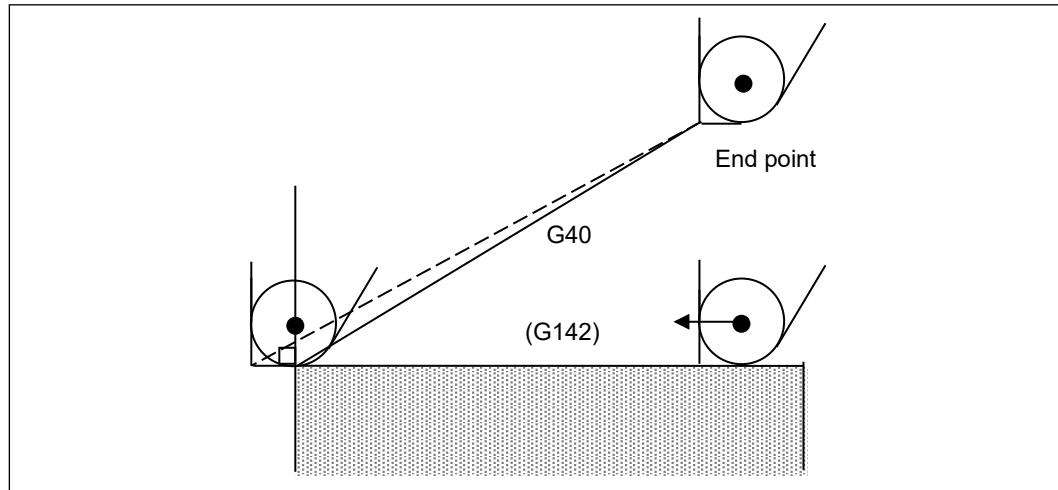
Cancel mode is enabled for the control when a command that meets all the conditions below is executed for offset mode. Offset cancel refers to the travel operation in this situation.

- 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.



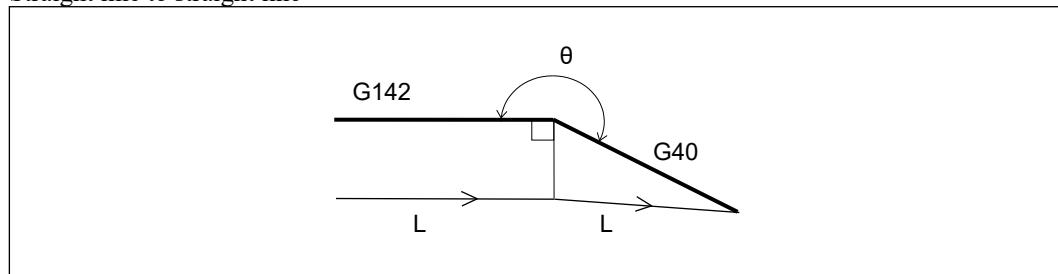
4

(NOTE 1) When an arc and thread cutting command are issued, an error is triggered.

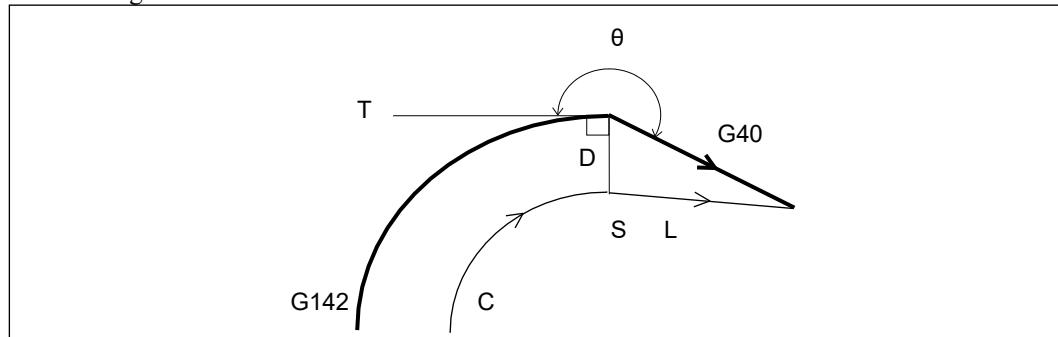
(NOTE 2) There are two setting types for the startup and cancel operations: <Type 1 (shortcut)> and <Type 2 (detour)>. Use the user parameter <Start up/cancel> to set one of the types.

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

Straight line to straight line

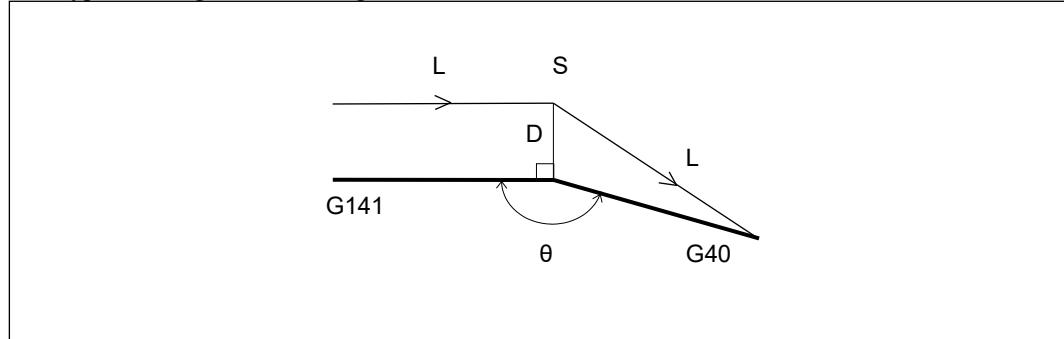


Arc to straight line



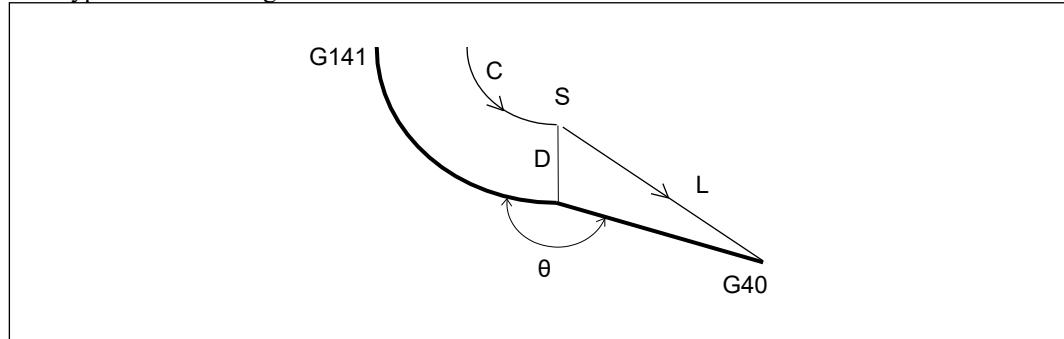
4.3.6.2 Outer Side (Obtuse Angle Cutting) ($90^\circ \leq \theta < 180^\circ$)

1. Type 1: Straight line to straight line

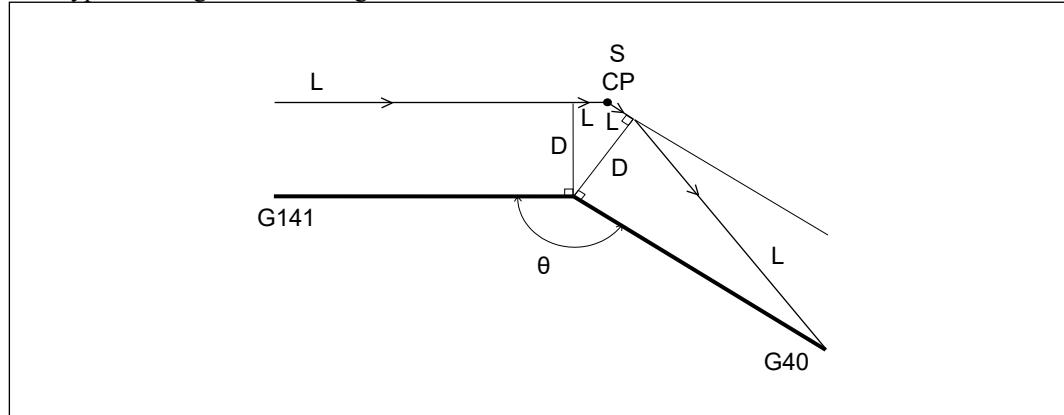


Type 1: Arc to straight line

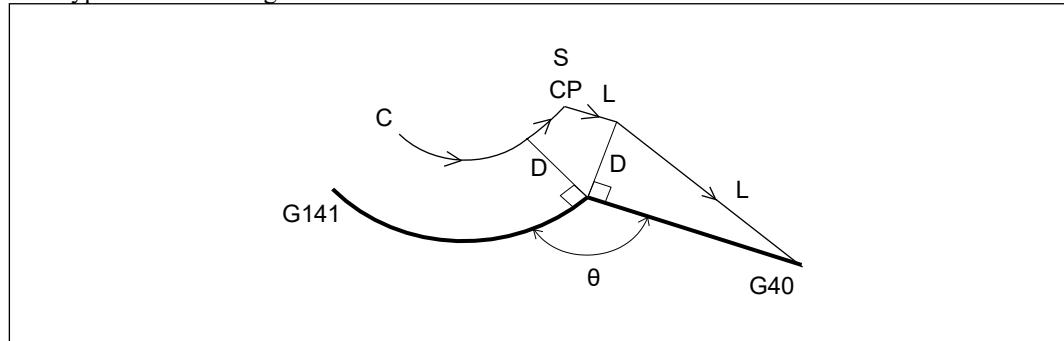
4



2. Type 2: Straight line to straight line



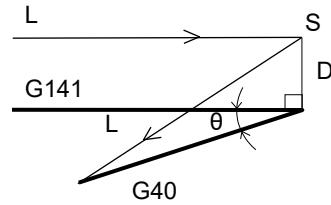
Type 2: Arc to straight line



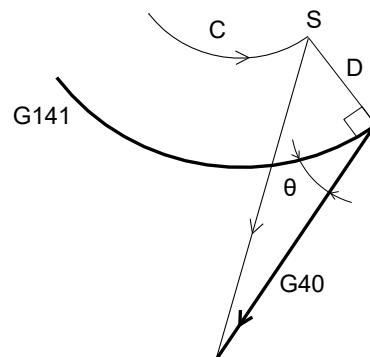
(NOTE) When $179^\circ \leq \theta < 180^\circ$, the setting <0: Type 1 (shortcut)> is used or enabled, even if <1: Type 2 (detour)> is specified for the user parameter <Start up/cancel>.

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

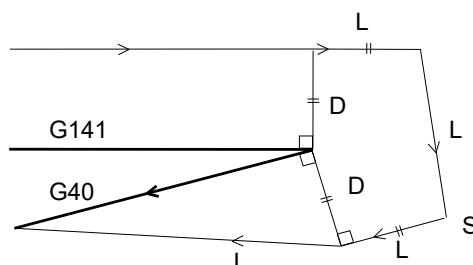
1. Type 1: Straight line to straight line



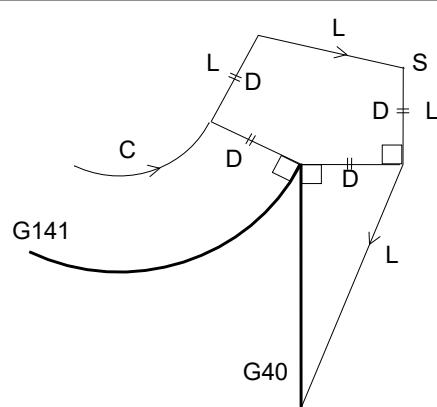
Type 1: Arc to straight line



2. Type 2: Straight line to straight line



Type 2: Arc to straight line

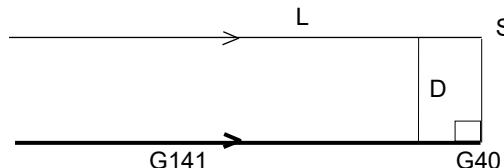


4.3.7 G40 Individual Command

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

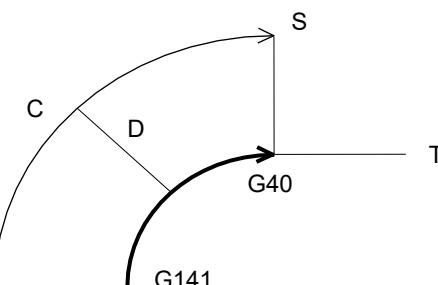
```
G141 Y_Z_D_;  
G40;
```



4

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.

G code	Compensation sign +	-
G141	Left side offset	Right side offset
G142	Right side offset	Left side offset

Execution conditions

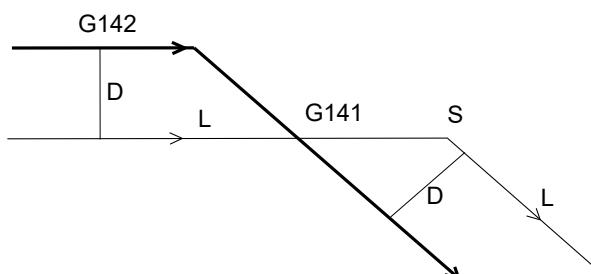
-	Command	Straight line to straight line	Straight line to arc	Arc to straight line	Arc to arc		
G141	G141	Executes (The location that is offset by the nose R radius at a position perpendicular to the end point of the previous block becomes the stop point.)					
G142	G142						
G141	G142	Executes		Executes			
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.

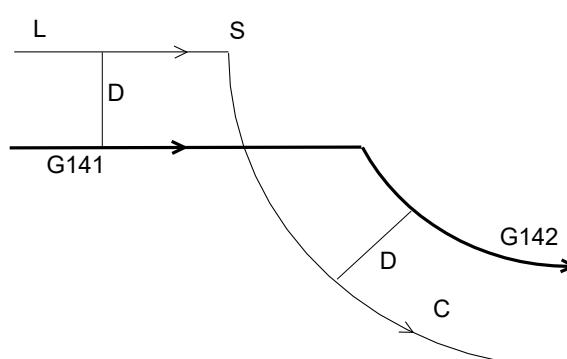
4.3.9 Offset Direction Change in Offset Mode

4.3.9.1 When There is an Intersection

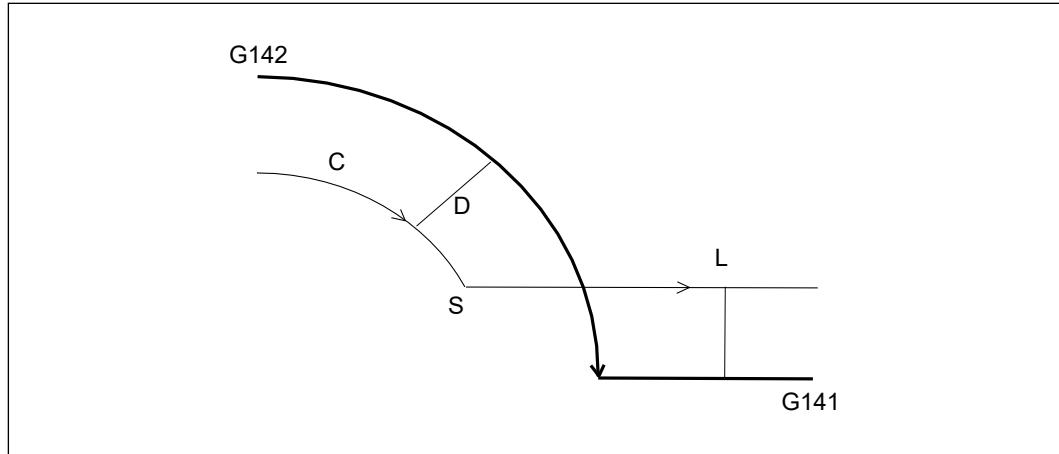
Straight line to straight line



Straight line to arc

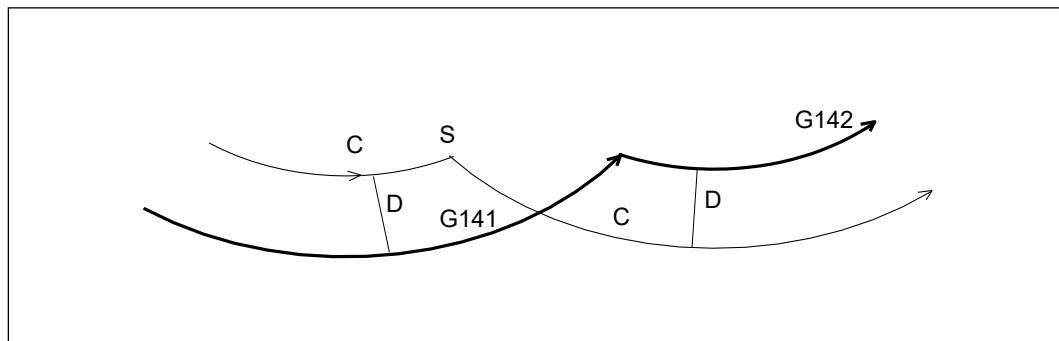


Arc to straight line



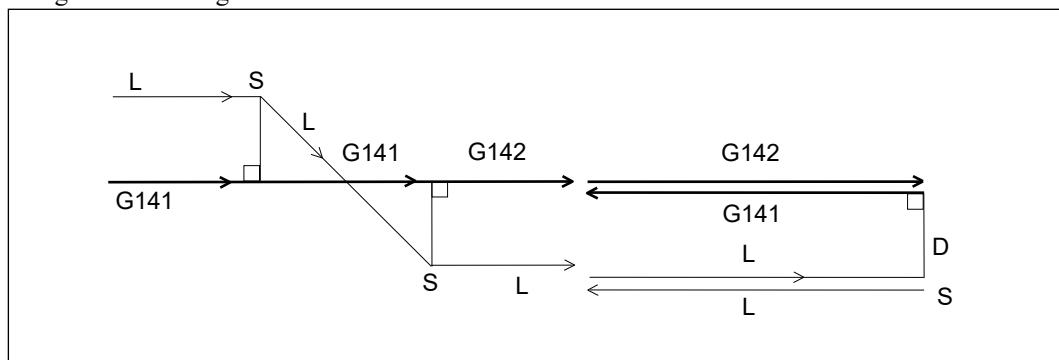
4

Arc to arc

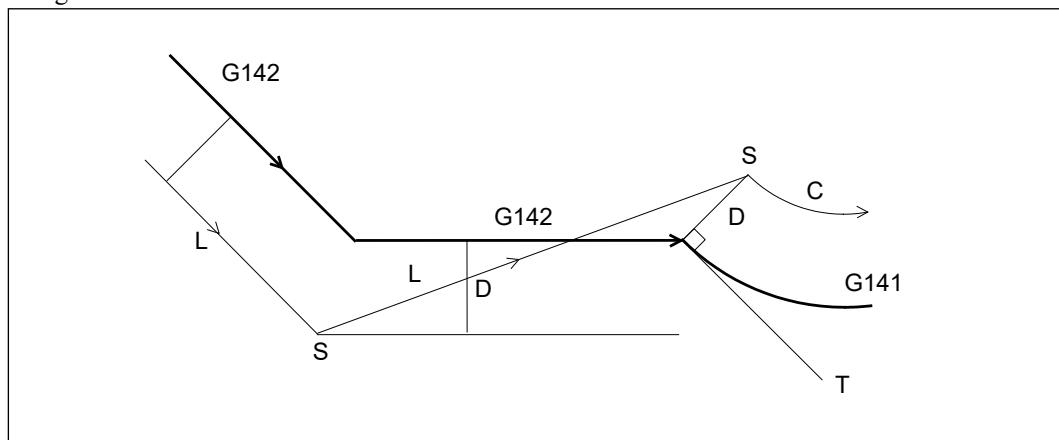


4.3.9.2 When There is No Intersection

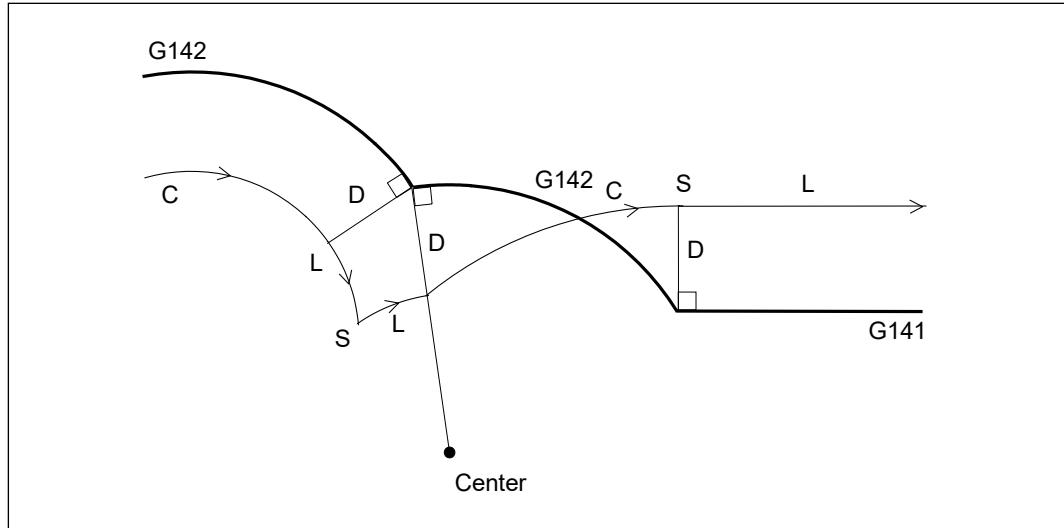
Straight line to straight line



Straight line to arc

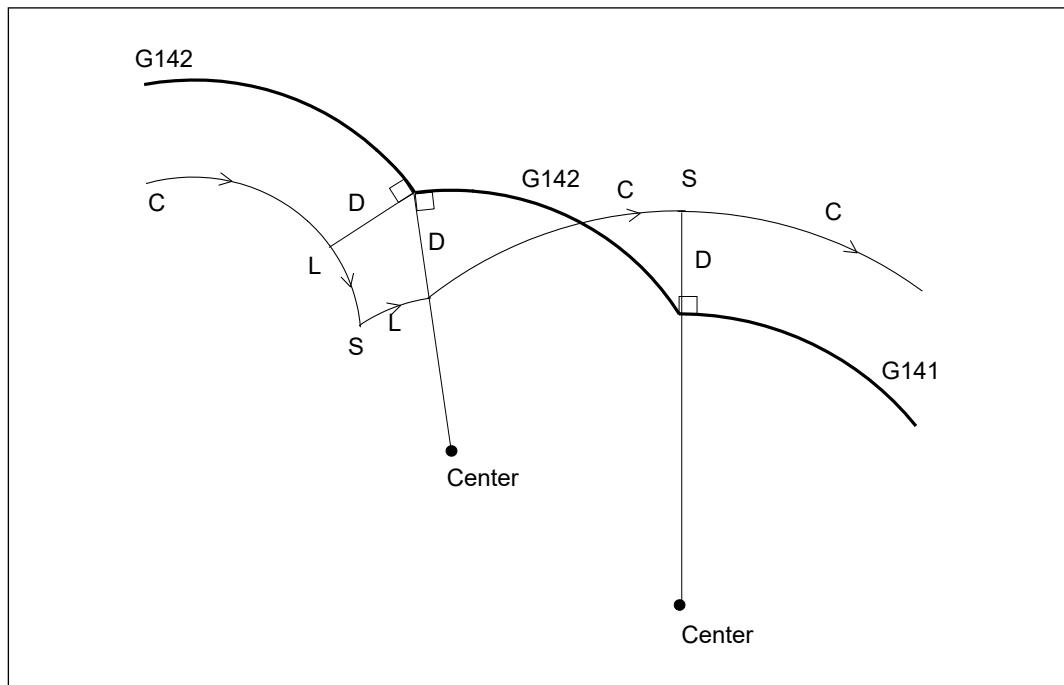


Arc to straight line



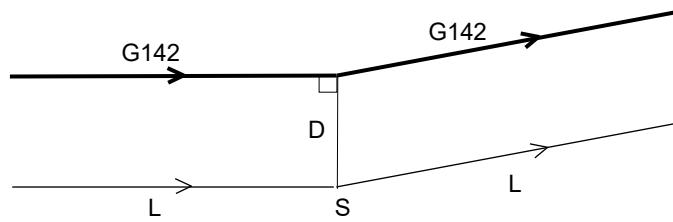
4

Arc to arc



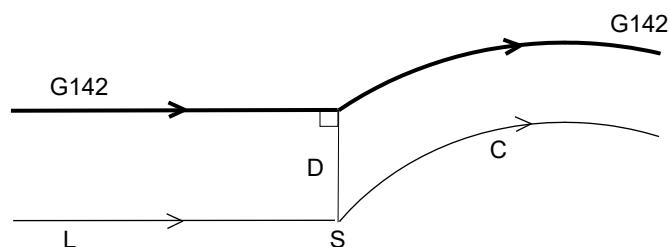
4.3.10 G Code Command for Nose R Compensation in Offset Mode

Straight line to straight line

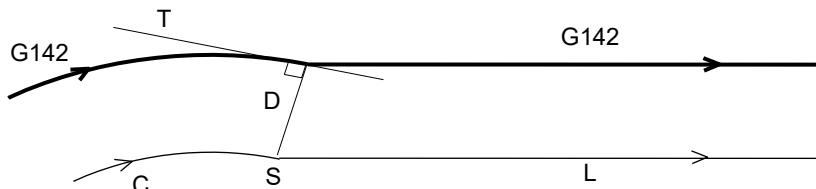


4

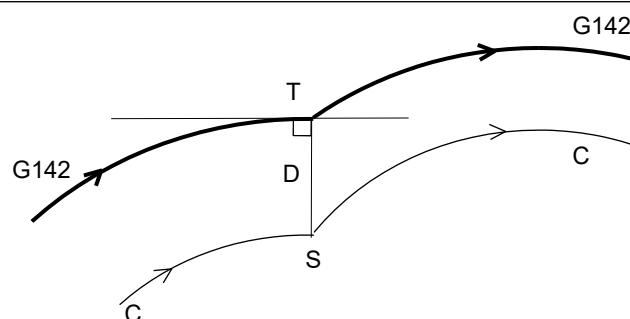
Straight line to arc



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

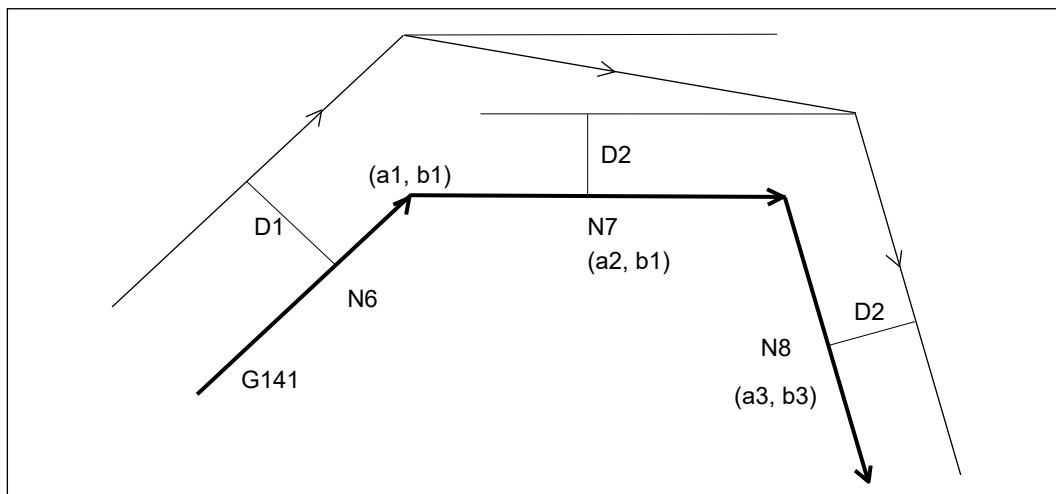
When the nose R compensation is changed while in offset mode, that compensation change applies from the compensation for the end point of that block.

N1 G141 Y_Z_D1;

N6 Ya1 Zb1;

N7 Ya2 D2; Compensation changed

N8 Ya3 Zb3;

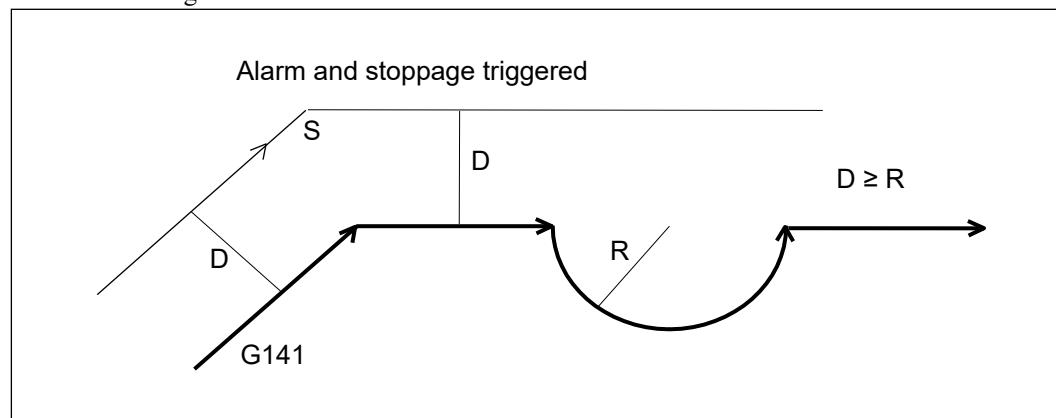


4

3. Current position display

The current position display shows the virtual teeth position.

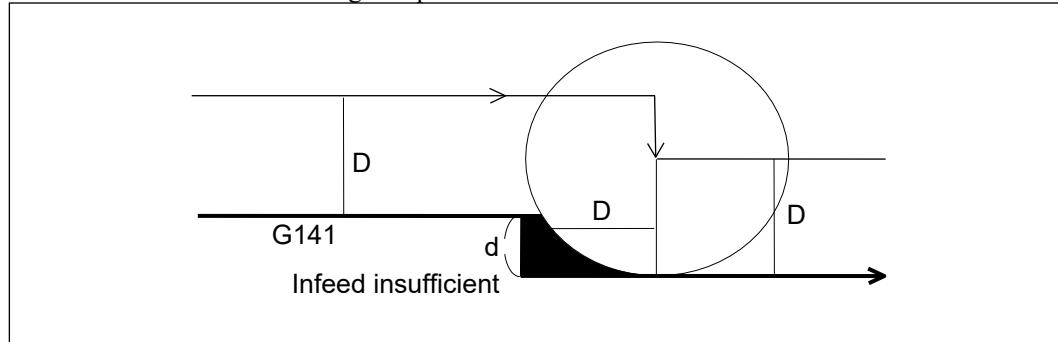
4. When cutting the inner side of an arc with a radius that is smaller than the nose R radius.



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

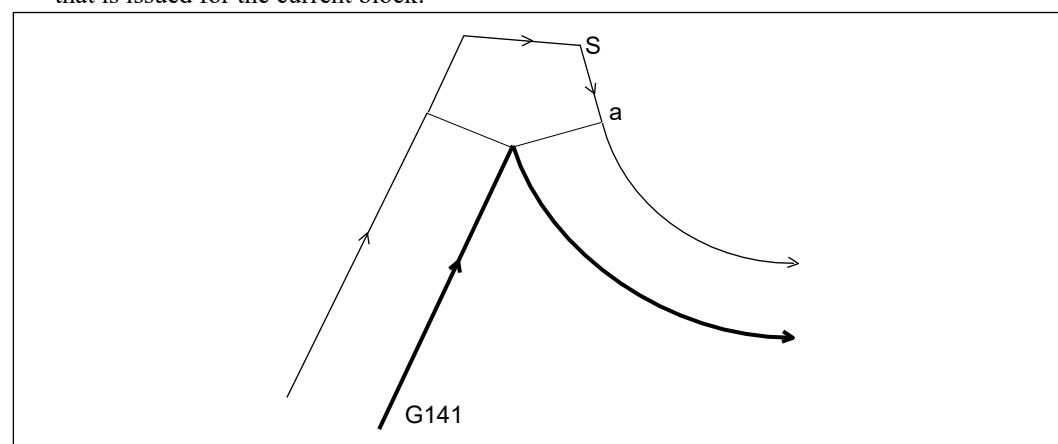
This occurs when machining a step that is smaller than nose R.



6. Corner travel

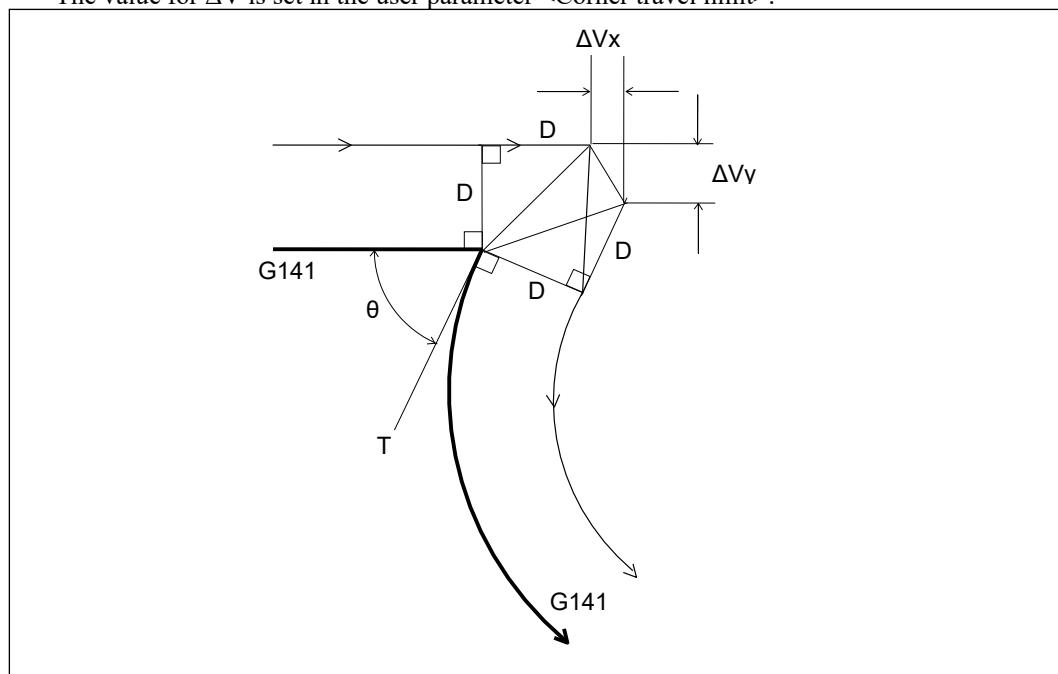
When cutting the outer side, it can turn on a corner with many angles. The travel mode and feedrate when turning the corner up to point a in the diagram below are based on the command that is issued for the current block.

4



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 ΔV is set in the user parameter <Corner travel limit>.

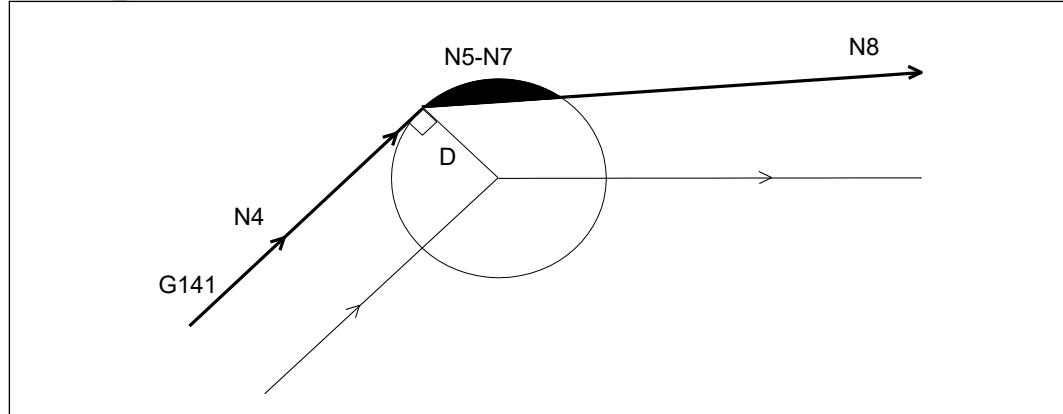


As a result, extremely small travel operations for a corner can be kept to a minimum.

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.

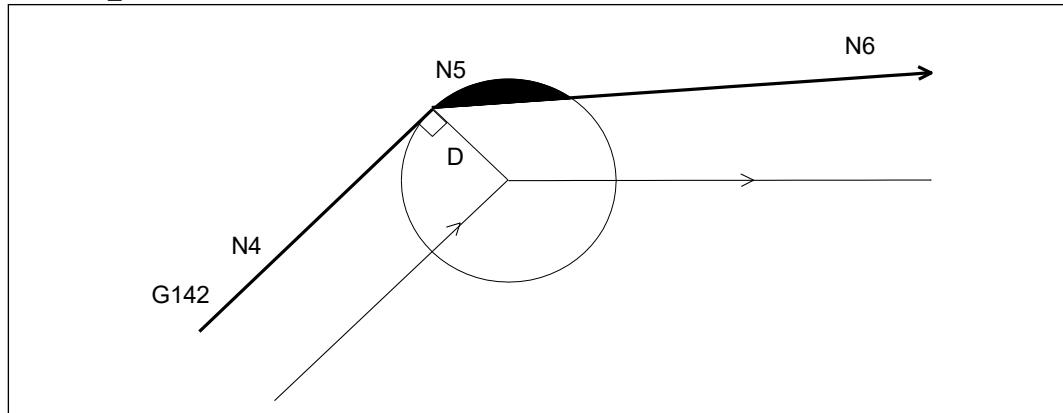
```
N4 Y_Z_;  
N5 X_;  
N6 F_;  
N7 X_;  
N8 Y_;
```



4

(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 ≠ 0).

Chapter 4 Preparation Function (Compensation Function)

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 ≠ 0.

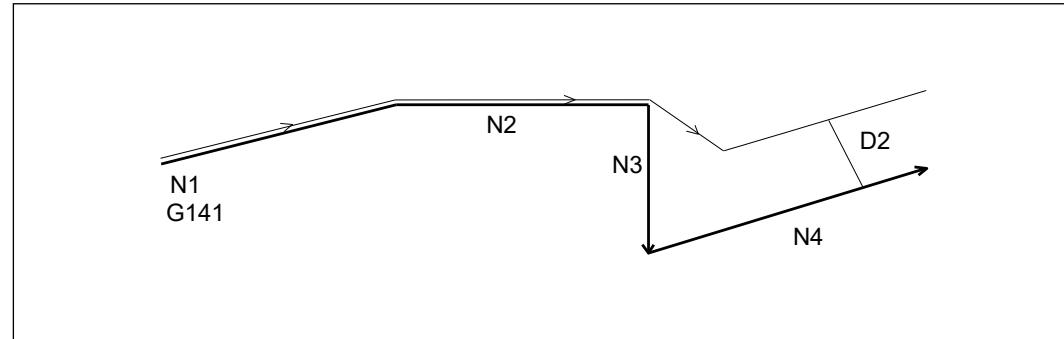
N1 G141 Y_Z_D1; (D1=0)

N2 Y_;

N3 Z_D2; (D2≠0)

N4 Y_Z_;

4



- (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 ≠ 0.

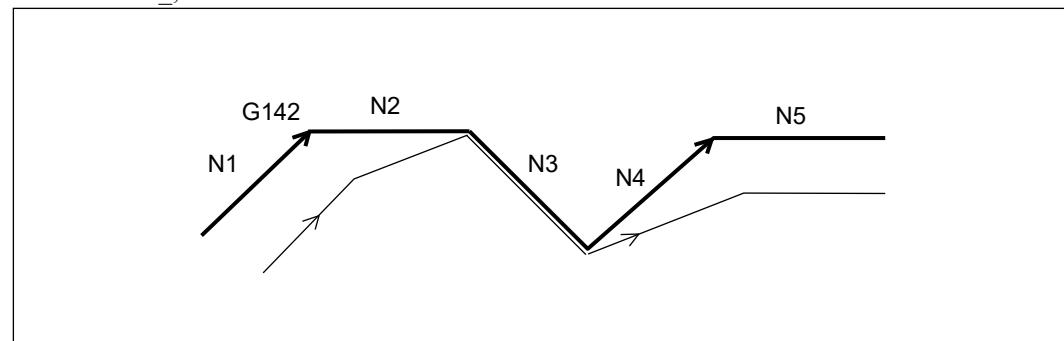
N1 Y_Z_;

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>>

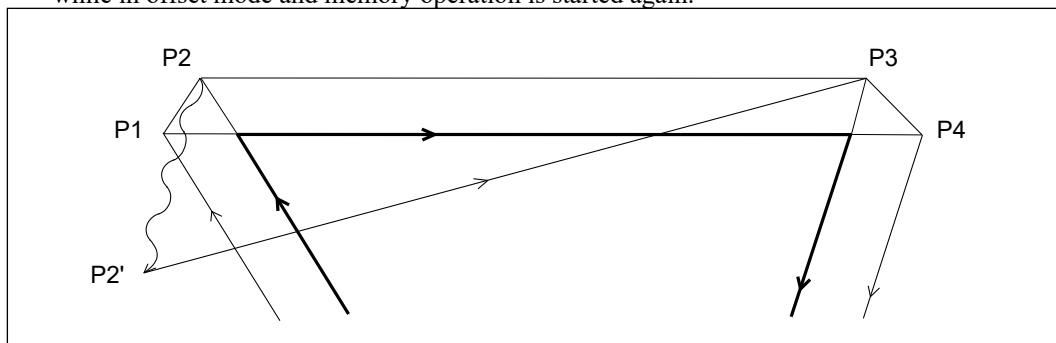
G17~G19	: Plane selection
G28	: Reference position return
G29	: Return from reference position
G30	: No. 2 to 6 reference position return
G33,G376,G392	: Thread cutting
G36~G39	: Coordinate calculation
G60	: Single direction positioning
G66	: Macro program modal call
G68.2	: Feature coordinate setting
G73~G89, G173~G189, G277~G278	: 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
G102, G103	: XZ circular interpolation
G202, G203	: YZ circular interpolation
G43.4, G43.5	: TCP control

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

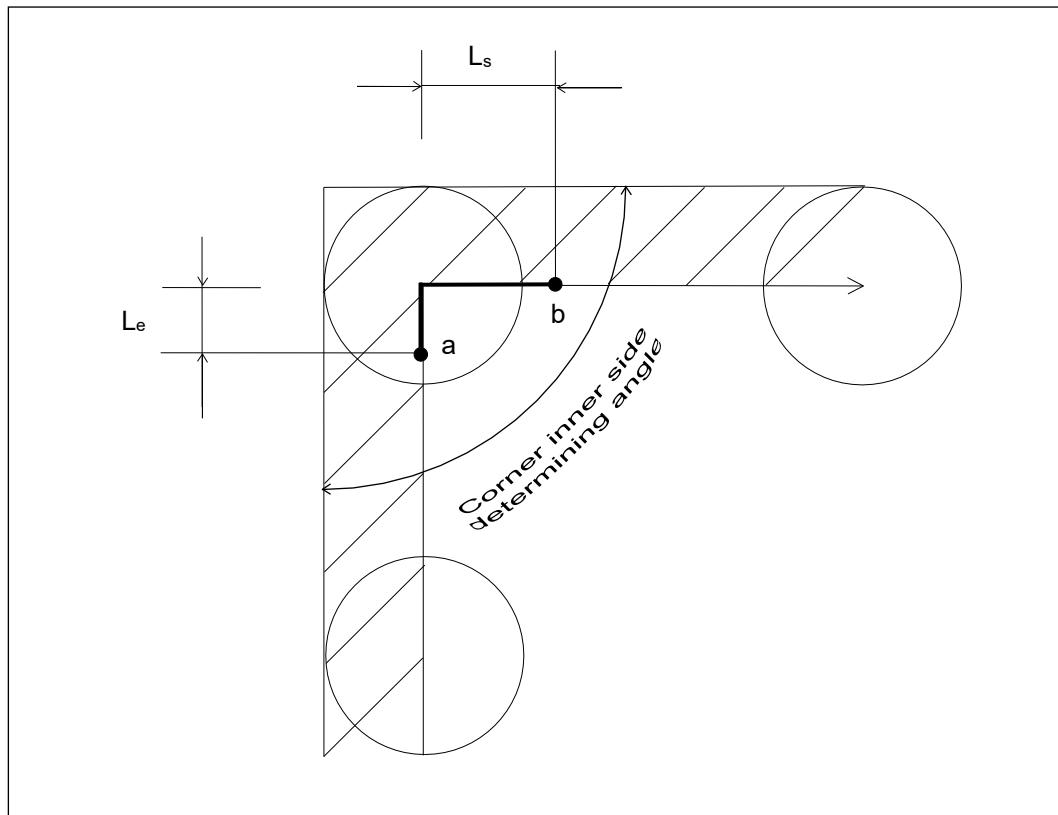
When both the block before and after the corner of the inner side meet the following conditions while in offset mode, the override function is automatically enabled in order to reduce the load on the tool.

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 | θ |

4



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

$$\text{Actual feedrate} = \text{Command speed} \times \frac{\text{Deceleration ratio}}{100}$$

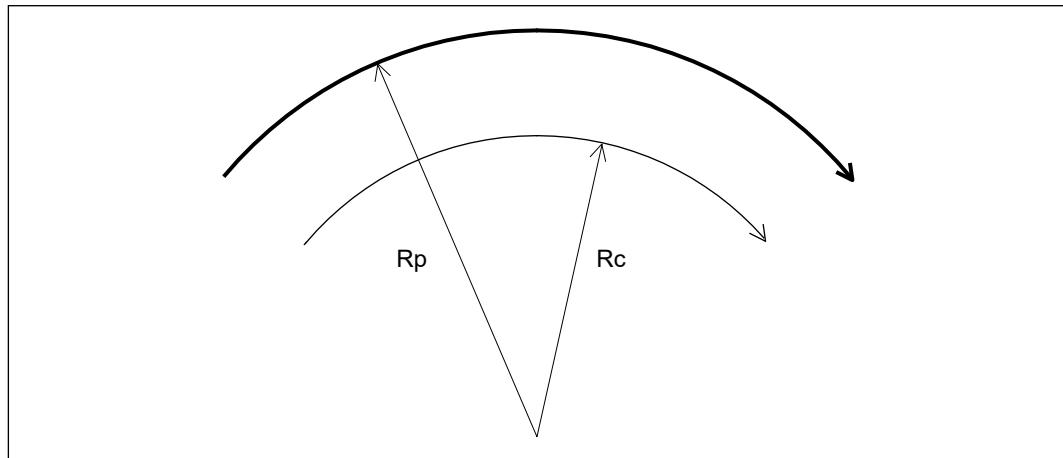
4.3.12.2 Inner Arc Override

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

$$\text{Actual feedrate} = \text{Command speed} \times \frac{R_c}{R_p}$$

R_p: Program radius

R_c: Tool center path radius



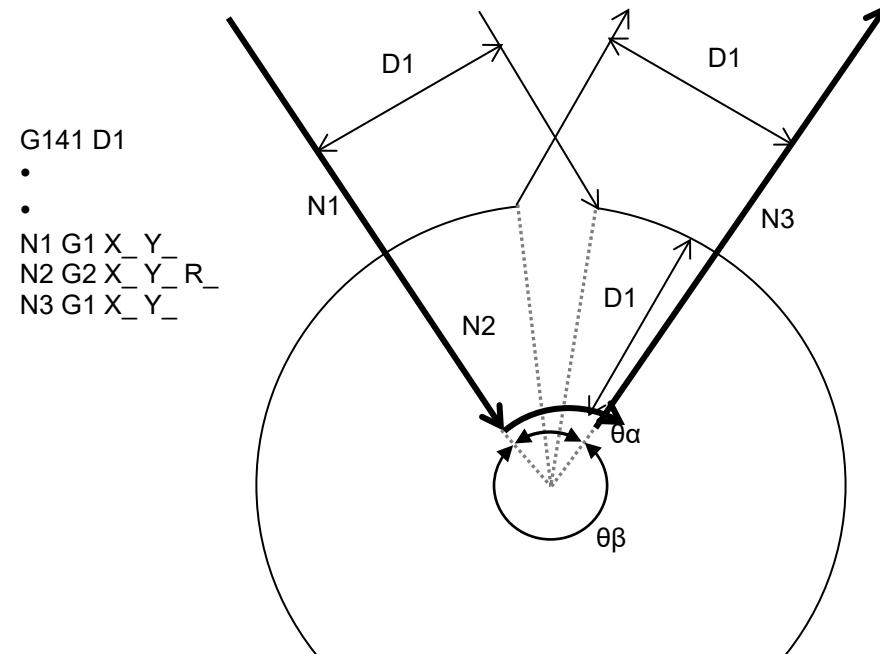
4

(NOTE) When $\frac{R_c}{R_p}$ is less than the <Inner arc override limit> that is set in the user parameter, that parameter value is multiplied as an alternative to $\frac{R_c}{R_p}$.

$$\text{Actual feedrate} = \text{Command speed} \times \frac{\text{Inner arc override limit}}{100}$$

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.



4

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°, 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.

4

1. Tool position compensation (+)

Command format

G143 Hn;

Hn : Tool number (n = 0 to 99, 201 to 299), or group number (n = 901 to 930)

2. Tool position compensation (-)

Command format

G144 Hn;

Hn : Tool number (n = 0 to 99, 201 to 299), 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 “X-, Y- and Z-axis travel for 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.

- (NOTE 8) When a tool position compensation command (G143/G144) is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered. In addition, when a TCP control command is issued while in tool position compensation mode, the alarm <<TCP control command not possible>> is triggered.

4.4.2 Axis Travel with Tool Position Compensation

- 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 (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change>	0: Type 1		1: Type 2
User parameter (switch 1: compensation function) <Error check when traveling during tool length/tool position offset cancel>	0: Check	1: 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> : <0: Type 1> <Error check when traveling during tool length/tool position offset cancel> : <0: Yes>		
	X		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> : <0: Type 1> <Error check when traveling during tool length/tool position offset cancel> : <1: No>		
	X		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> : <1: Type 2>*		
	X		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 <0: Yes> or <1: No>.

- (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 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.;<br>...;<br>G0 X-100. Y-100.;<br>...;<br>G0 Z300.; | { ← Temporarily cancels the tool position compensation<br>{ ← X- and Y-axes tool position compensation resumes<br>{ ← 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

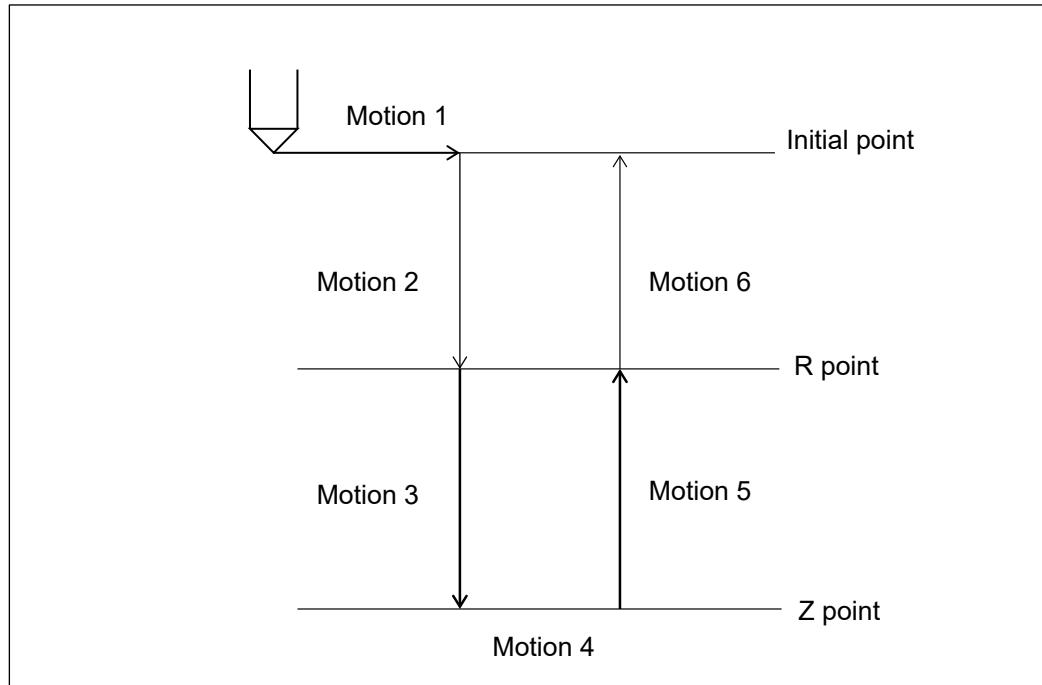
Functions of canned cycle

| 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



5

In a single block operation, the control stops at the end of motions 1, 2, and 6, respectively.

(NOTE 1) 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 [FEED HOLD] switch, selecting the manual mode, etc.), the control stops at the end of motion 5. This also applies when you press the [RST] key in motion 3, 4, or 5.

(NOTE 2) When the Z-axis perimeter mode is on (M300), motion 1, motion 5 (rapid feed only) and motion 6 apply to Z-axis perimeter operation. However, it does not operate right after the tapping cycle (G74, G77, G78, G84, G177, G178, G277 and G278) recovery operation. Refer to “Chapter 12 M function” for further details on the Z-axis perimeter operation.

## 5.4 General Rules of Canned Cycle

### 5.4.1 Canned Cycle Operation Commands

(I) Data format

$$\left\{ \begin{array}{ll} G90 & \text{Absolute} \\ G91 & \text{Incremental} \end{array} \right.$$

(II) Return level

$$\left\{ \begin{array}{ll} G98 & \text{Return to initial point level} \\ G99 & \text{Return to R point level} \end{array} \right.$$

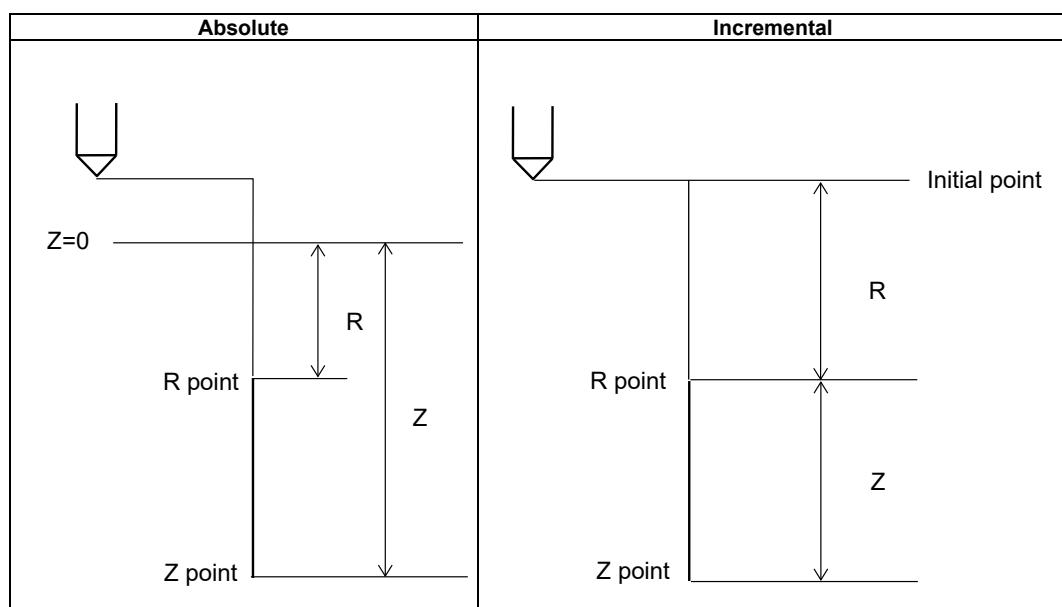
(III) Hole machining mode

$$\left\{ \begin{array}{l} G73, G74 \\ G76 \sim G78 \\ G81 \sim G87 \\ G89 \\ G177, G178 \\ G181, G182 \\ G185, G186 \\ G189 \\ G277, G278 \end{array} \right\}$$

Please refer to “5.2 List of canned cycle function”

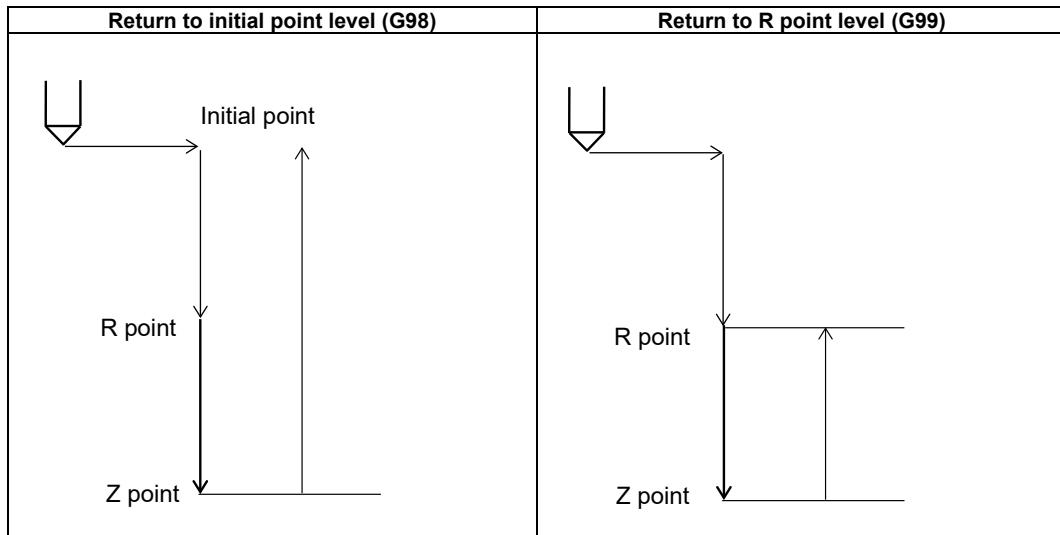
5

### 5.4.2 Data in Absolute and Incremental Mode



### 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.



5

- (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

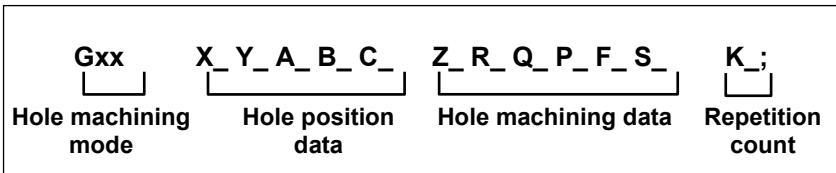
Canned cycle operations are performed under the following conditions:

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).
- (NOTE 4) A canned cycle command cannot be issued while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

### 5.4.5 Canned Cycle Machining Data

Command  
format



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

5

(NOTE 1) 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

(NOTE 2) When the TCP control command (G43.4/G43.5) is issued during a canned cycle, the alarm <<TCP control command not possible>> is triggered.

### 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.

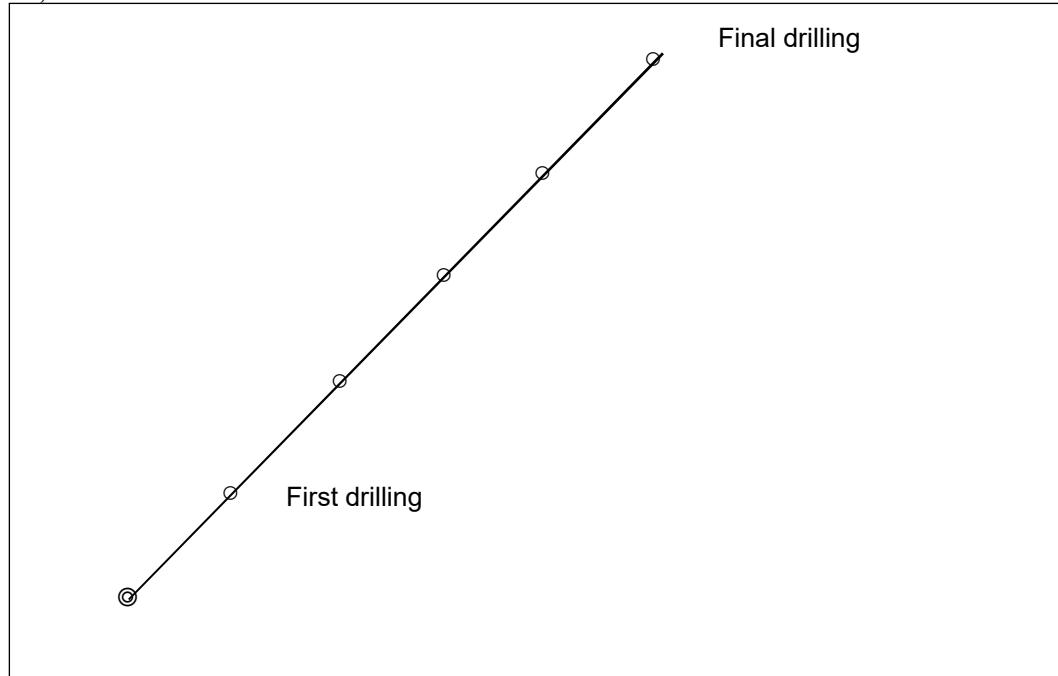
K is assumed to be 1 if a value is not specified.

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)



5

G81X\_Y\_Z\_R\_K5F\_ ; (G91 mode)

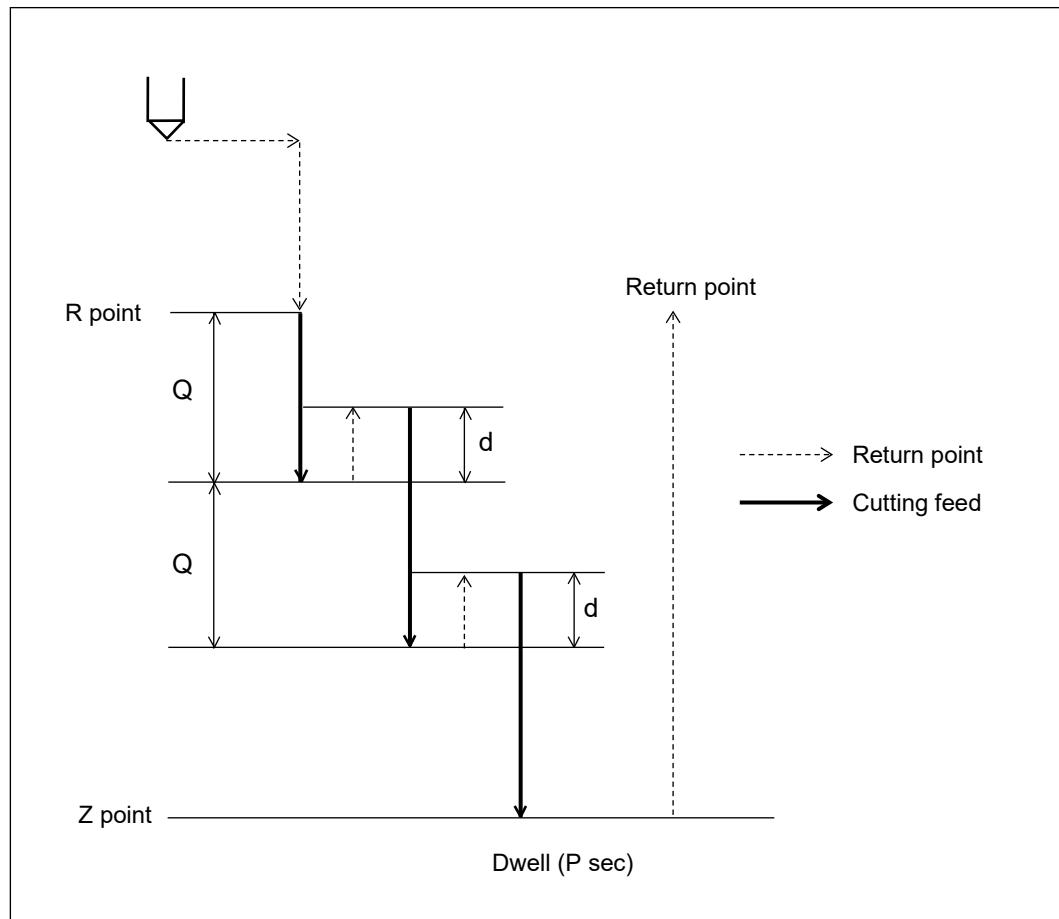
## 5.5 Details of Canned Cycle

### 5.5.1 High Speed Peck Drilling Cycle (G73)

Command format

G73 X\_ Y\_ Z\_ R\_ P\_ Q\_ F\_;

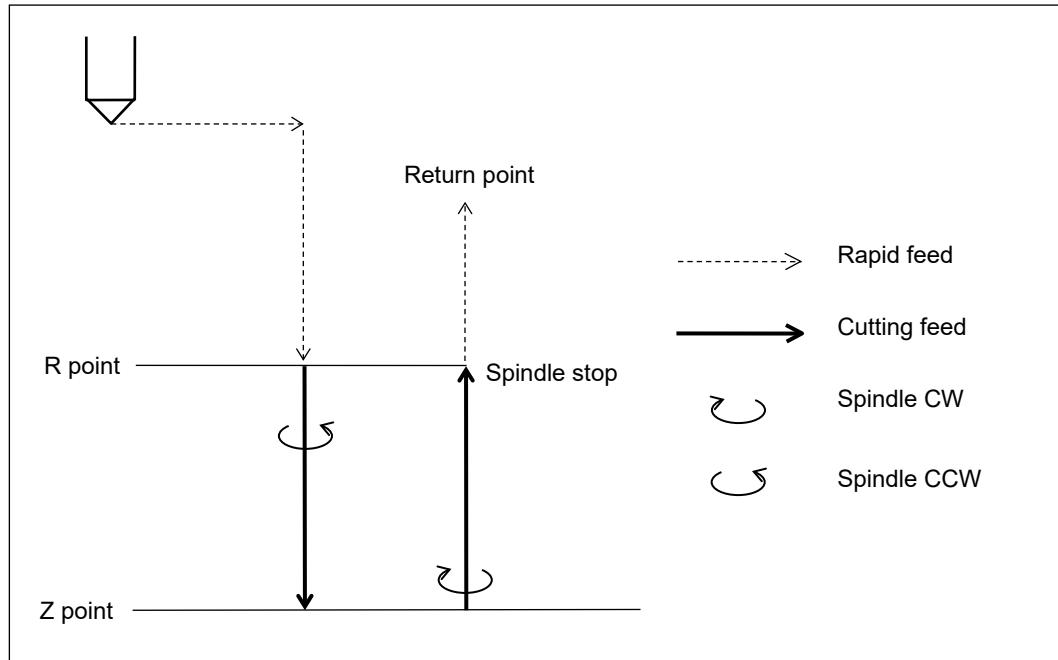
5



- The relief amount  $d$  is set in the user parameter (switch 1: canned cycle) <G73 relief amount>.
- 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\_;**

5

Spindle stops at Z point; dwells for P sec; and rotates clockwise.

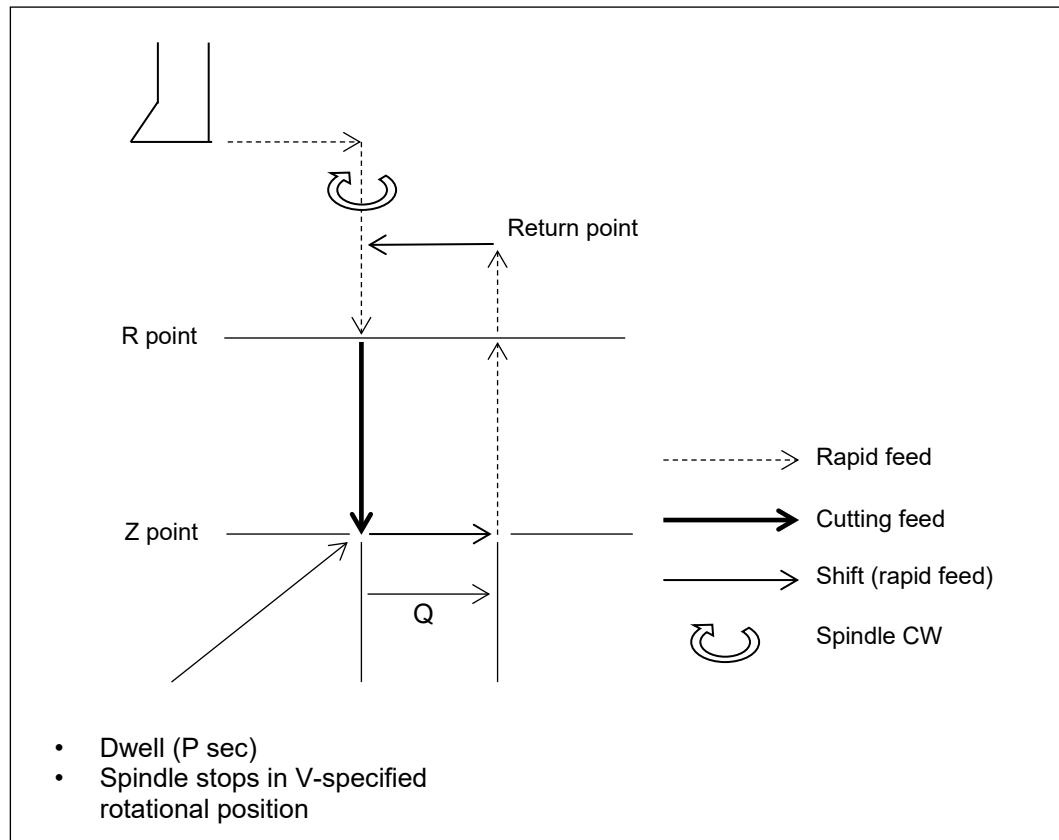
- The tool stops on finally reaching the R point when a temporary stop is commanded at any moment en route from R to Z point and further back to the R point.
- For feed per minute (G94), thread pitch = cutting feed rate ÷ spindle speed.
- For feed per revolution (G95), thread pitch = cutting feed rate.
- The alarm <>Spindle speed error>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <>The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- 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\_;

5

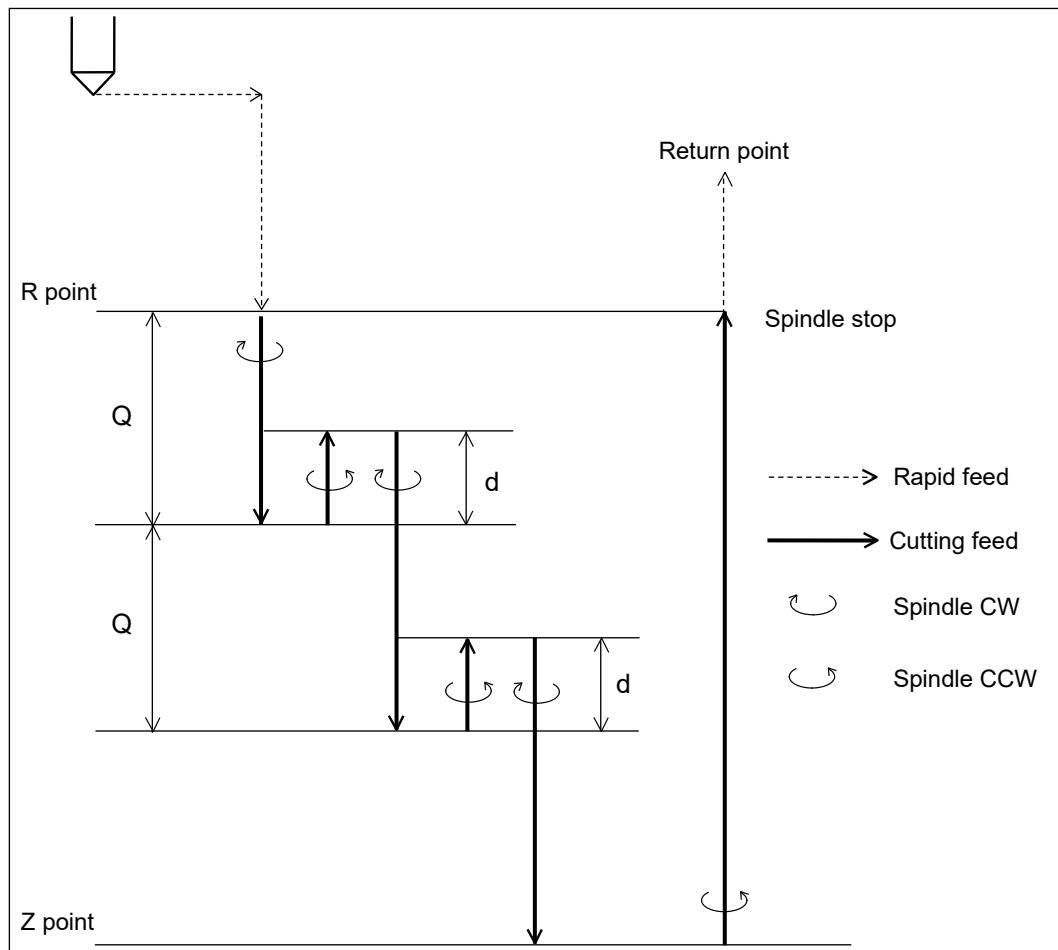


- The sign is ignored even when a negative value command is issued for the amount of shift for Q.
- Set the shift direction ahead of time to either +X, -X, +Y or -Y in the user parameter (switch 1: canned cycle) <G76, G87 shift direction>.
- Shift direction is selectable from only 4 directions of +X, -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

**G77 X\_ Y\_ Z\_ R\_  $\begin{pmatrix} I_- \\ J_- \end{pmatrix}$  Q\_ S\_;**



5

- The relief amount  $d$  is set in the user parameter (switch 1: canned cycle) <G77, G78 relief amount>.
- 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>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the user parameter (switch 1: canned cycle) <Tapping cycle return speed> is set to <1: Max. speed>, it uses the machine parameter (system 1: common) <Max. tapping speed> to return.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

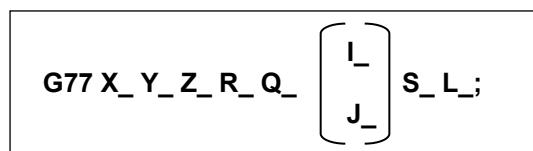
## Chapter 5 Preparation Function (Canned Cycle)

- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- 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



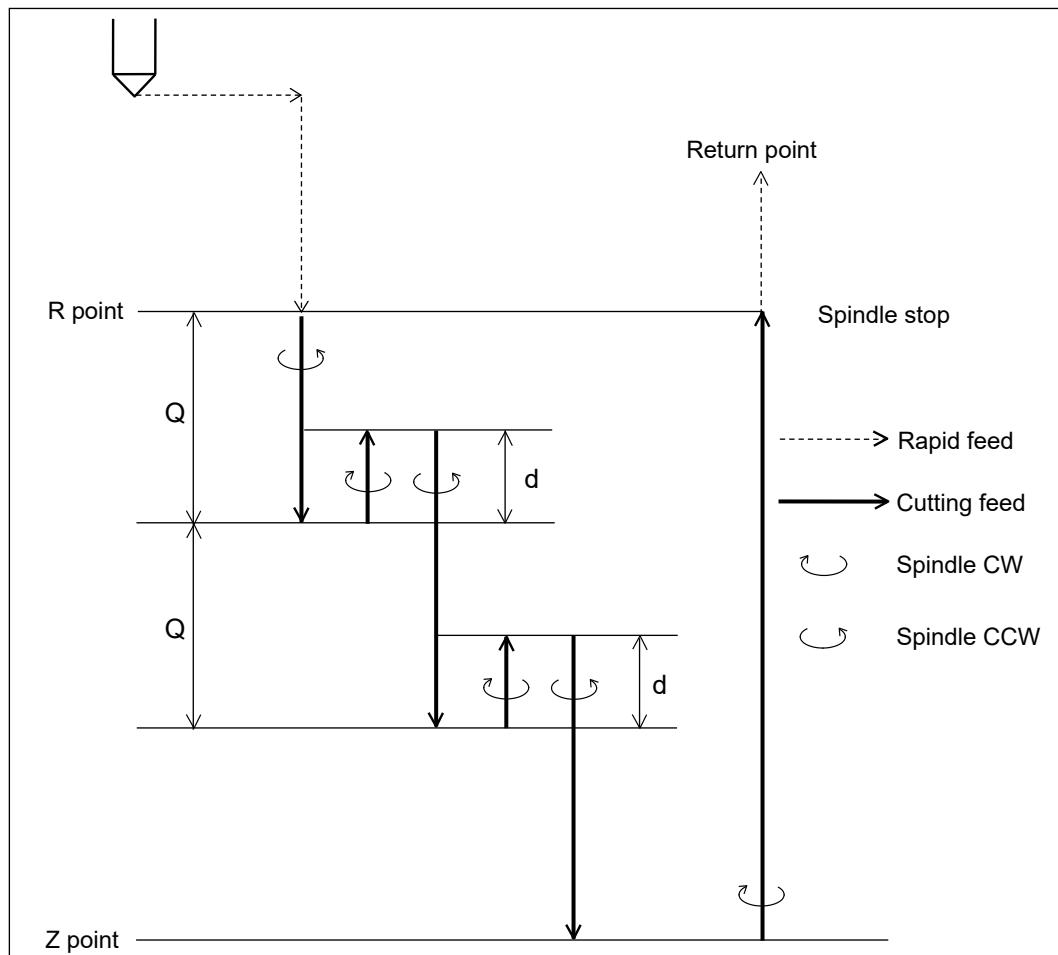
5

- 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>> is triggered when the command value for the L address is larger than the machine parameter (system 1: common) <Max. tapping speed>.
- The tool moves according to the address S value when address L value is smaller than it.
- Even when the user parameter (switch 1: canned cycle) <Tapping cycle return speed> is set to <1: Max. speed>, the L address is given priority.
- When the user parameter (switch 1: common) <Tap override> is set to another setting that is not <0: Disable>, the operation uses the speed value that incorporates the feedrate override or spindle override in the command value for the L address.

### 5.5.5 Reverse Tapping Cycle (Synchro Mode) (G78)

Command format

**G78 X\_ Y\_ Z\_ R\_ [I\_ J\_]** Q\_ S\_;



5

- The relief amount  $d$  is set in the user parameter (switch 1: canned cycle) <G77, G78 relief amount>.
- 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>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the user parameter (switch 1: canned cycle) <Tapping cycle return speed> is set to <1: Max. speed>, it uses the machine parameter (system 1: common) <Max. tapping speed> to return.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

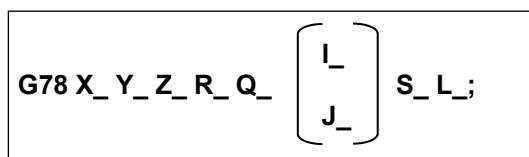
## Chapter 5 Preparation Function (Canned Cycle)

- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- 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

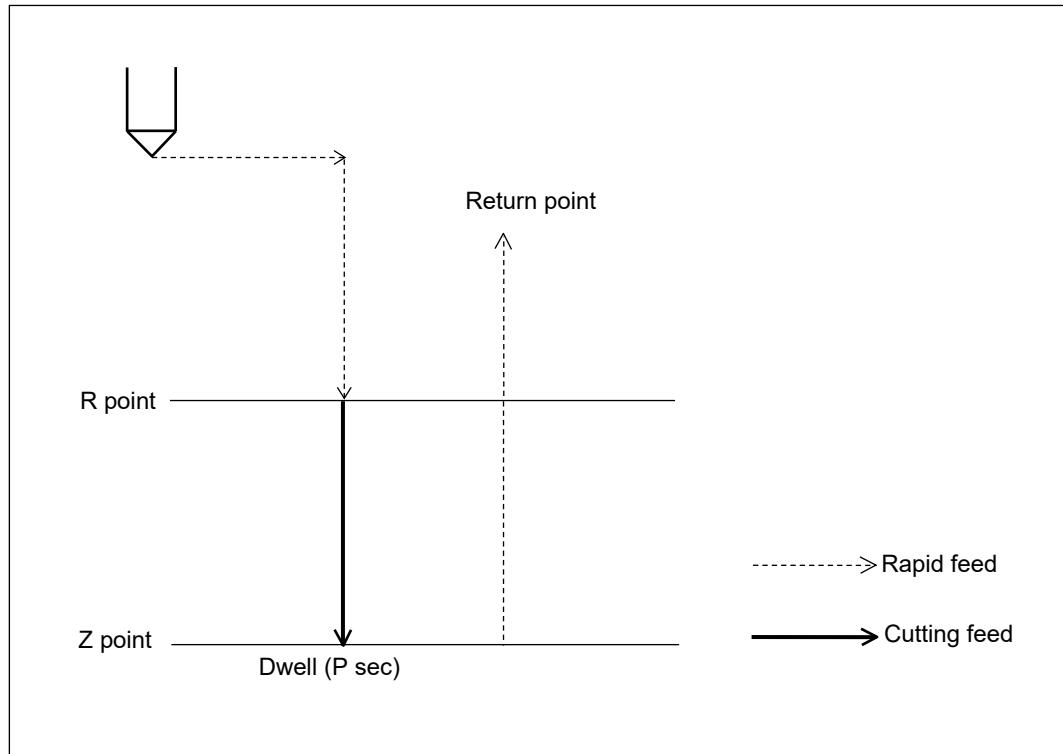
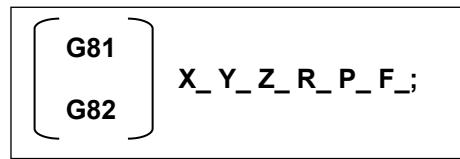


5

- 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>> is triggered when the command value for the L address is larger than the machine parameter (system 1: common) <Max. tapping speed>.
- The tool moves according to the address S value when address L value is smaller than it.
- Even when the user parameter (switch 1: canned cycle) <Tapping cycle return speed> is set to <1: Max. speed>, the L address is given priority.
- When the user parameter (switch 1: common) <Tap override> is set to another setting that is not <0: Disable>, the operation uses the speed value that incorporates the feedrate override or spindle override in the command value for the L address.

### 5.5.6 Drilling Cycle (G81, G82)

Command format



5

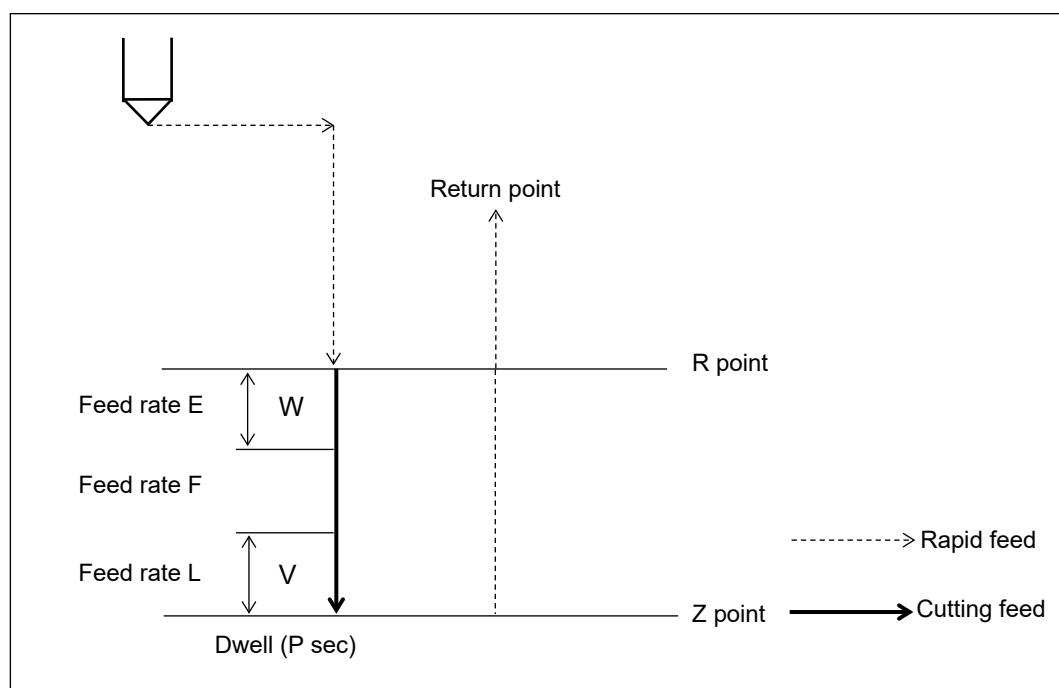
#### High speed cycle

Feed rate at start and end of cutting in drilling cycle (G81 or G82) is varied.

Command format



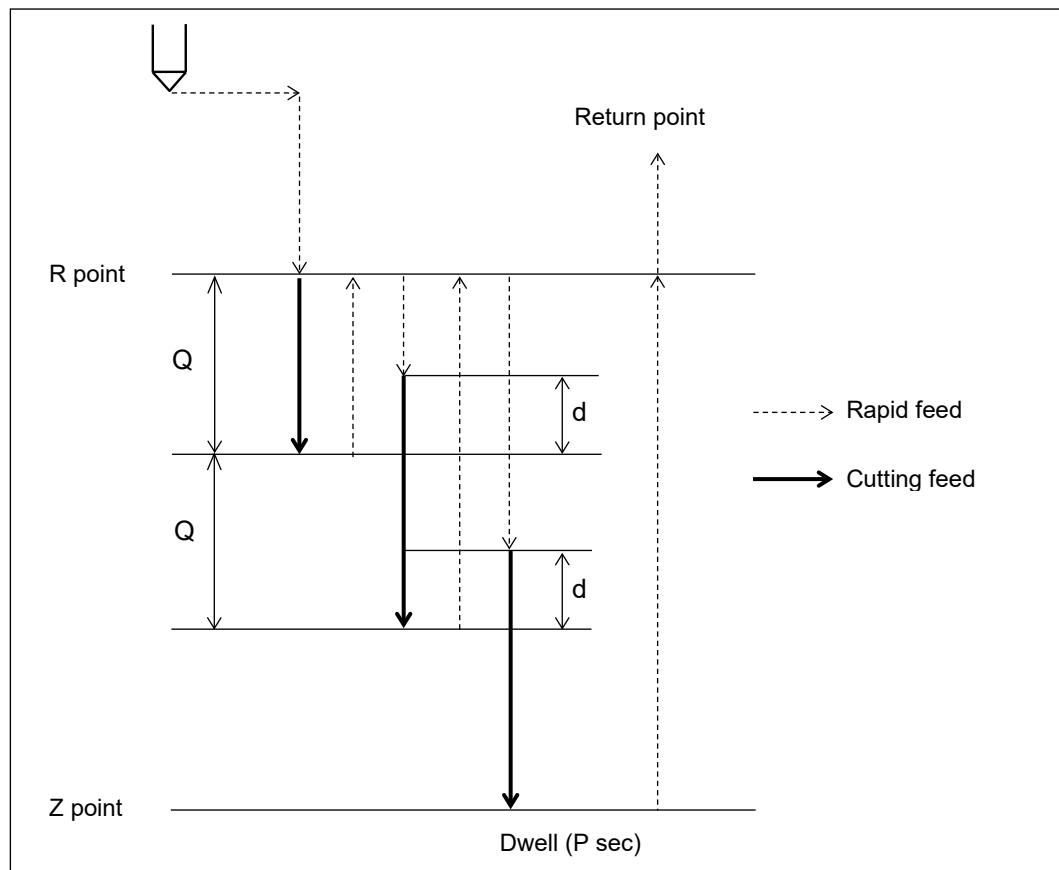
- W : Feed rate switching point  
Distance from R point irrespective of absolute (G90) or incremental (G91) mode.
- V : Feed rate switching point  
Distance from Z point irrespective of absolute (G90) or incremental (G91) mode.
- E : Feed rate for range W from R point
- L : Feed rate for range V from Z point



### 5.5.7 Peck Drilling Cycle (G83)

Command format

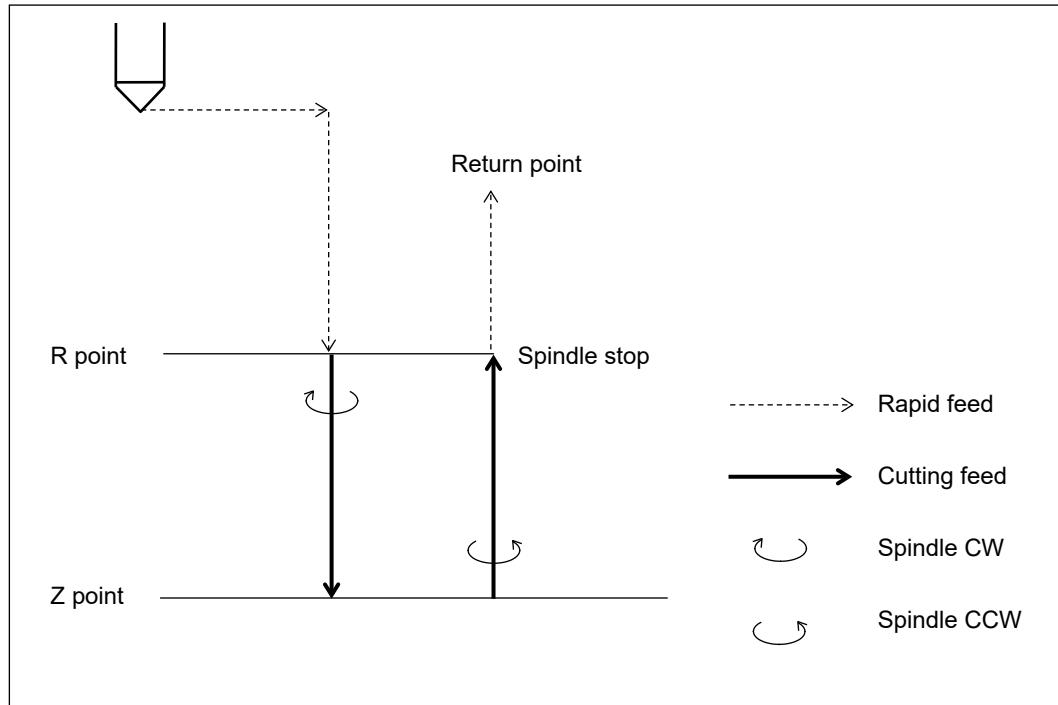
**G83 X\_ Y\_ Z\_ R\_ P\_ Q\_ F\_;**



- The cutting start position  $d$  is set in the user parameter (switch 1: canned cycle) <G83 cutting start position>.
- 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\_;**

5

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>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.

When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

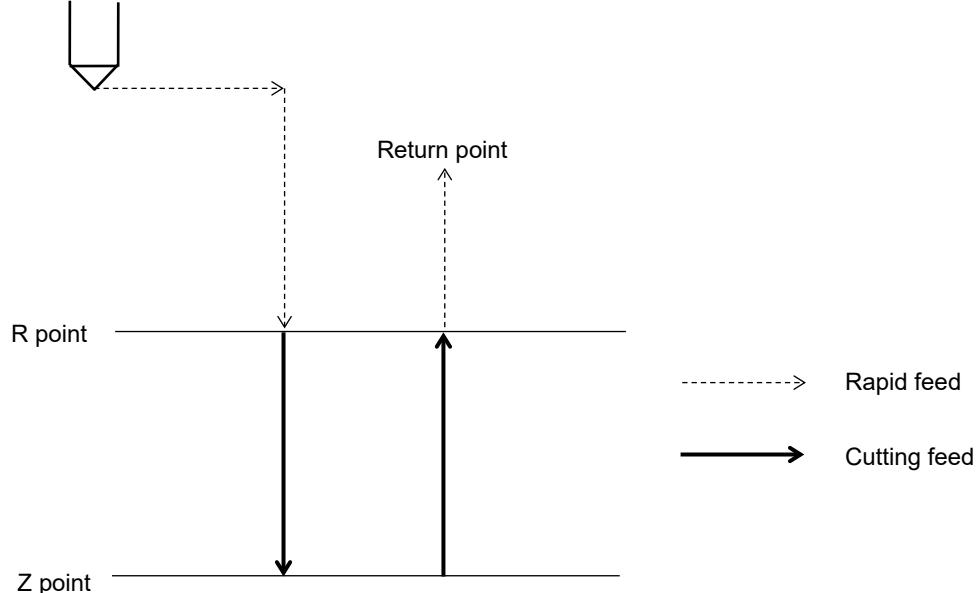
- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- 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



5



#### High speed cycle

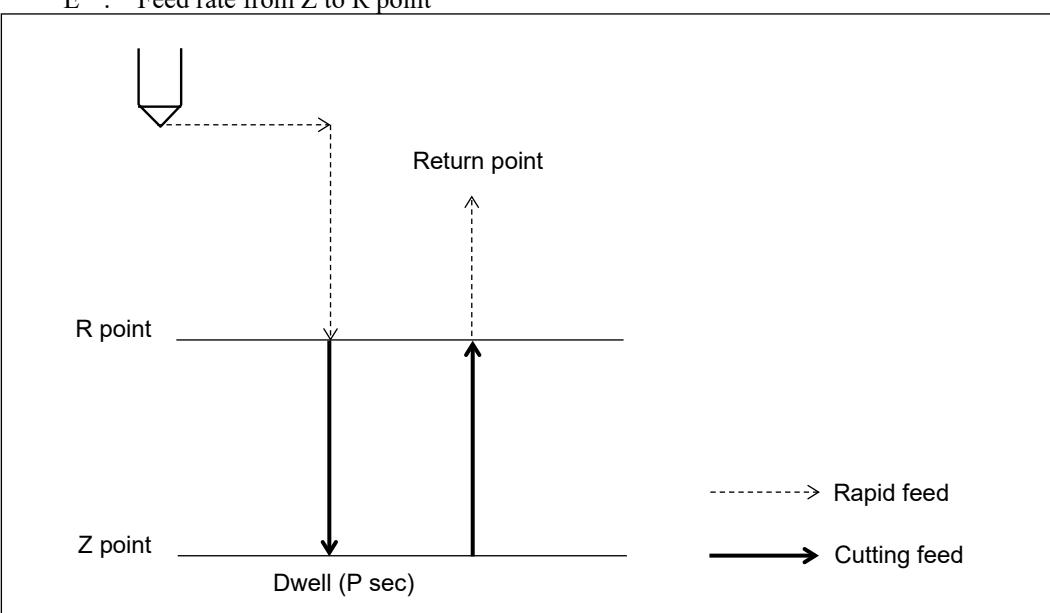
Feed rate at return of boring cycle (G85 or G89) is varied.

Command format



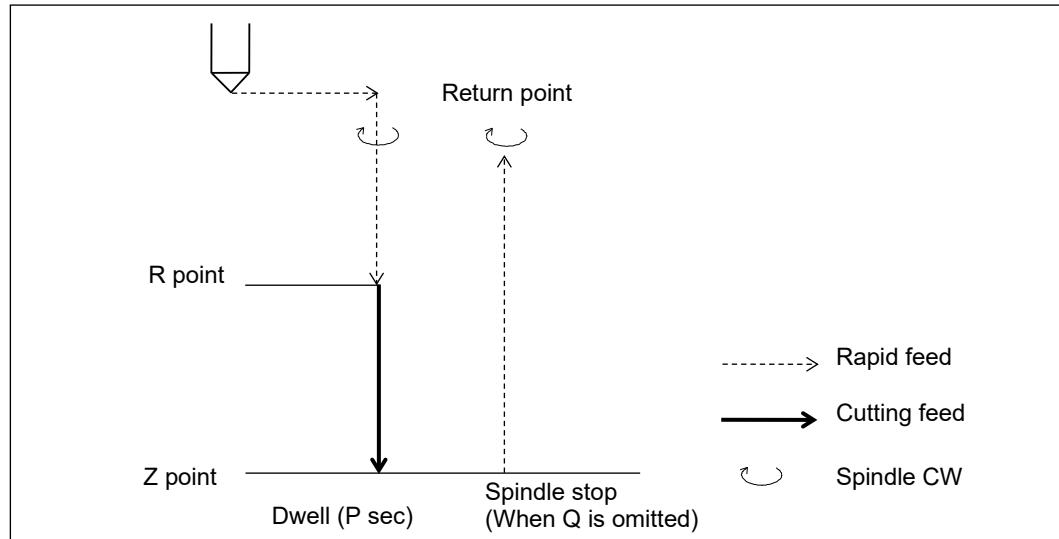
F : Feed rate from R to Z point

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\_;**

5

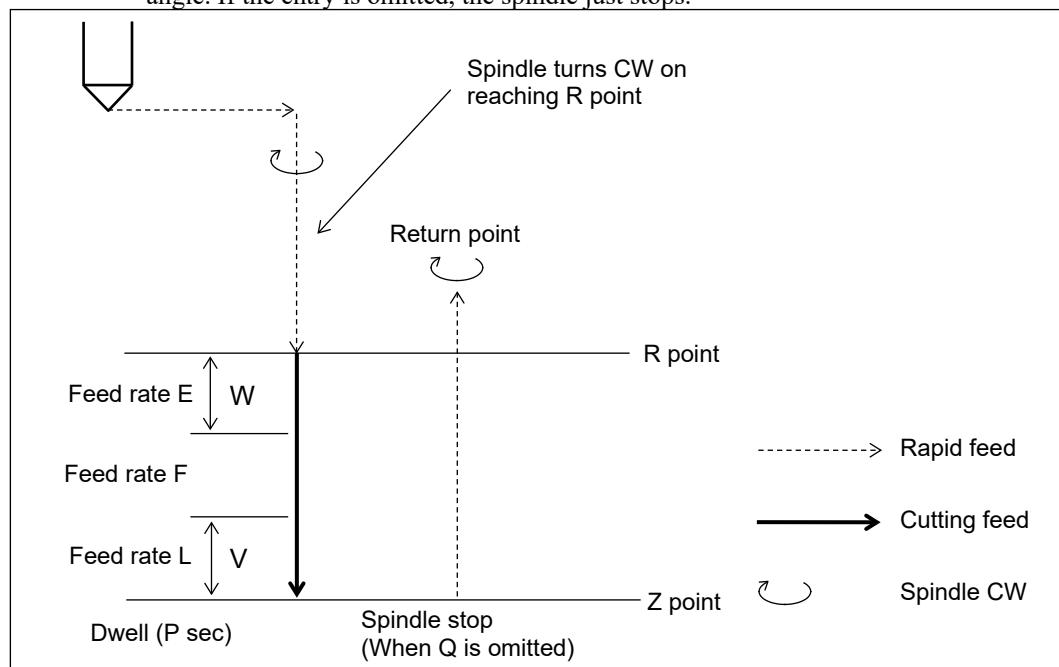
High speed cycle

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\_;**

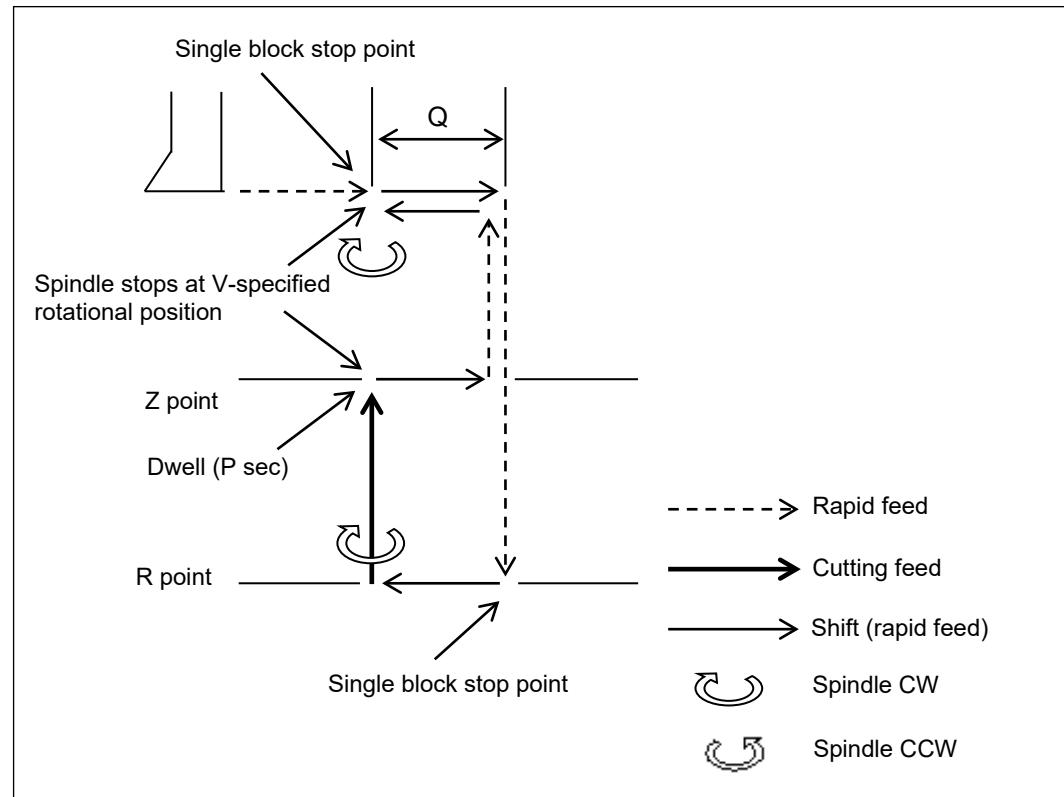
- W** : Feed rate switching point  
Distance from R point irrespective of absolute (G90) or incremental (G91) mode.
- E** : Feed rate for range W from R point
- V** : Feed rate switching point  
Distance from Z point irrespective of absolute (G90) or incremental (G91) mode.
- L** : Feed rate for range 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.
- Set the shift direction to either +X, -X, +Y or -Y in the user parameter (switch 1: canned cycle) <G76, G87 shift direction>.
- 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.

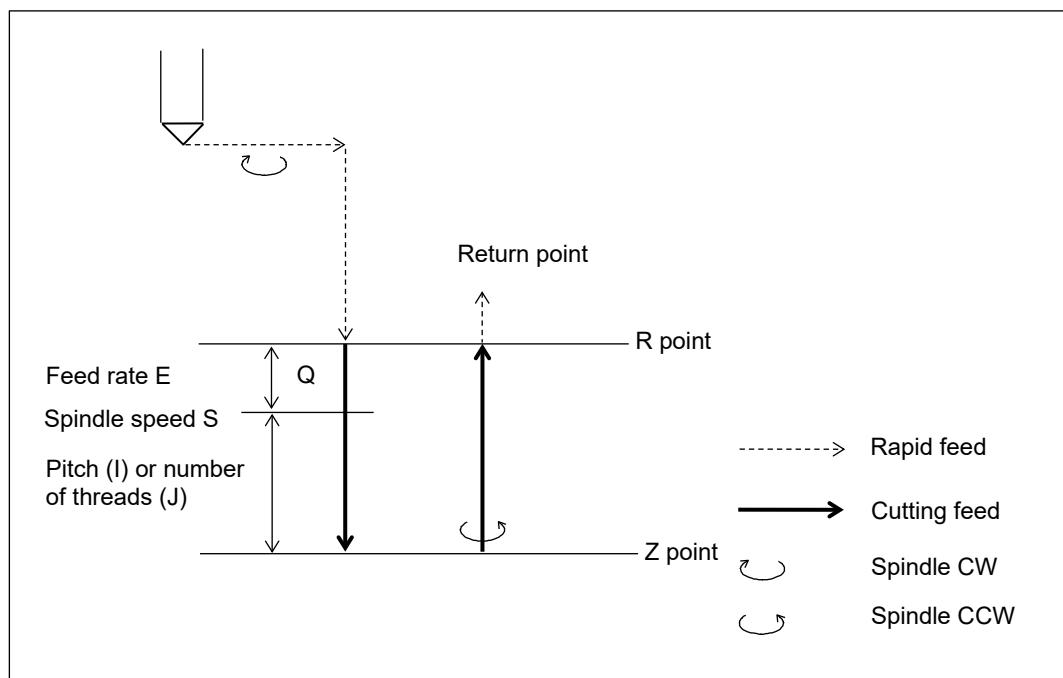
### 5.5.12 End Milling/Tapping Cycle (G177)

Command format

**G177 X\_ Y\_ Z\_ R\_ [L\_ J\_] 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

5



- The alarm <>Spindle speed error>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <>The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error 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.

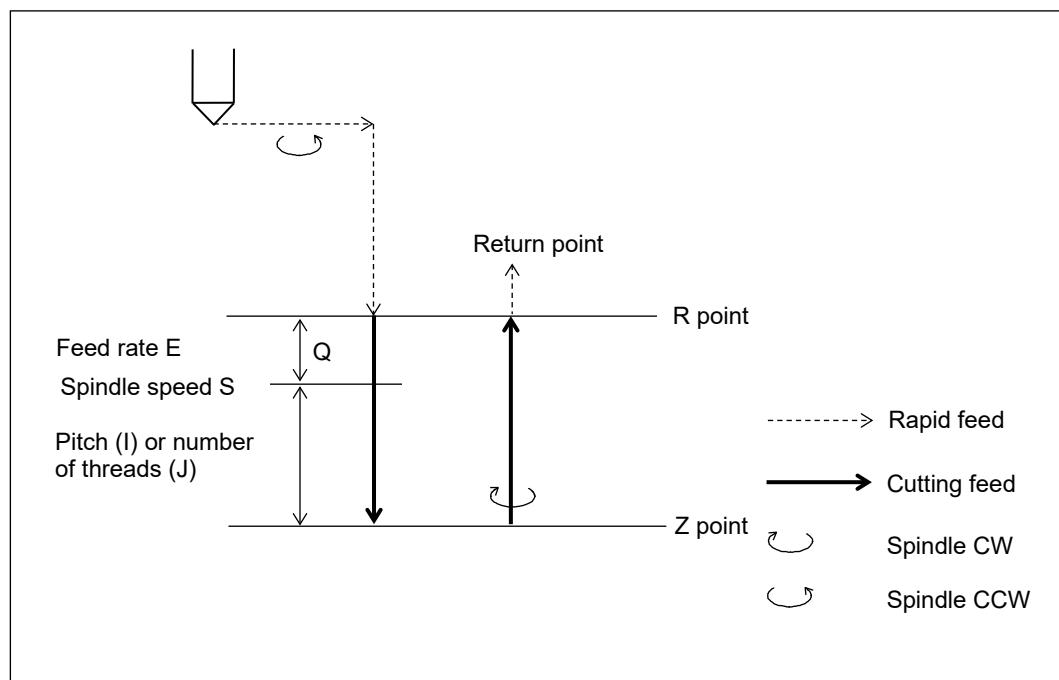
### 5.5.13 End Milling/Tapping Cycle (G178)

Command format

**G178 X\_ Y\_ Z\_ R\_ [L\_ J\_] 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

5



- The alarm <<Spindle speed error>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error 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.

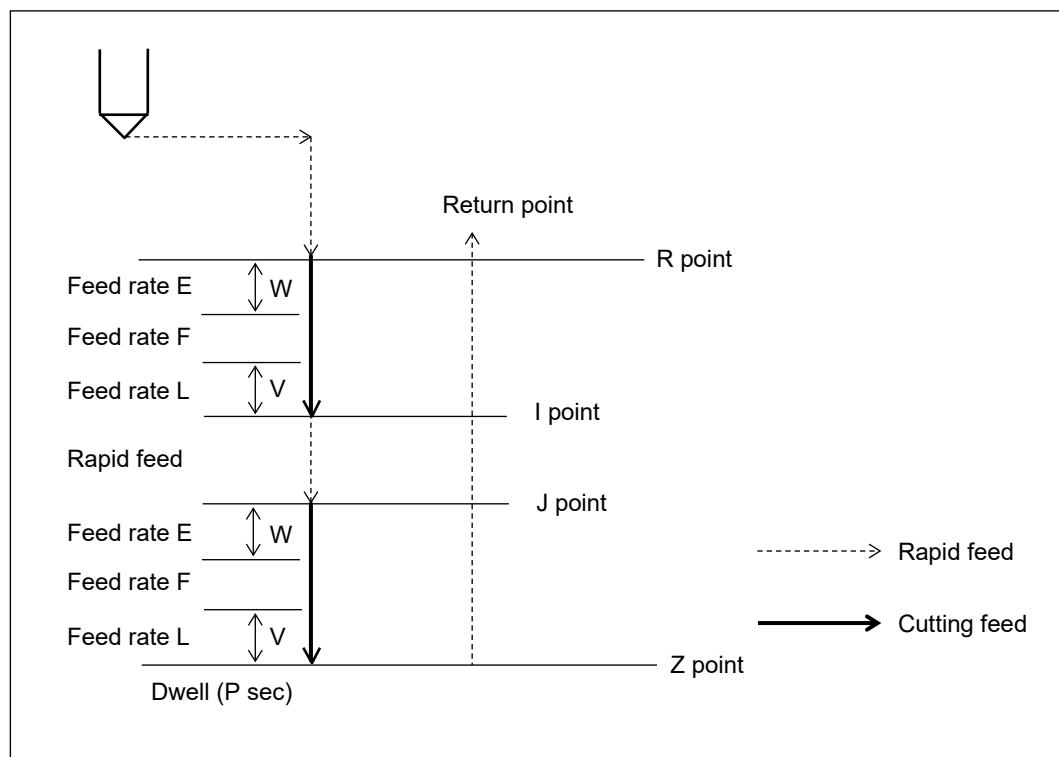
### 5.5.14 Double Drilling Cycle (G181, G182)

Command format

**G181**  
**G182** X\_ Y\_ Z\_ R\_ I\_ J\_ P\_ W\_ V\_ F\_ E\_ L\_ ;

- 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
- E : Feed rate for range W
- L : Feed rate for range V

5



### 5.5.15 Double Boring Cycle (G185, G189)

Command format

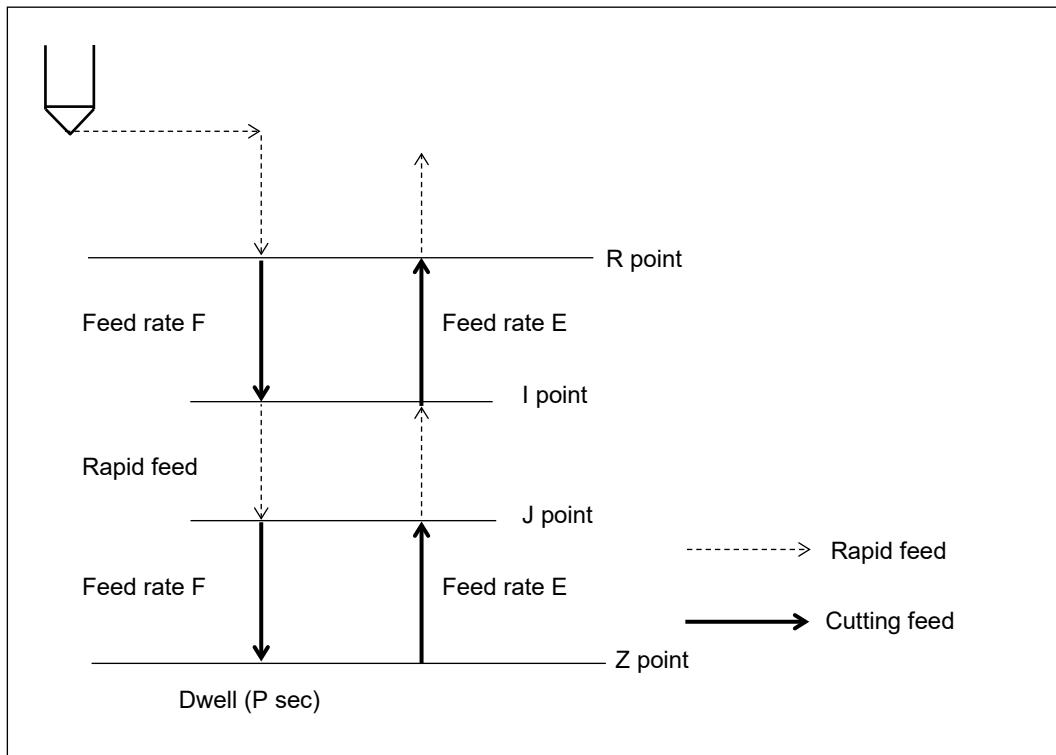
```

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



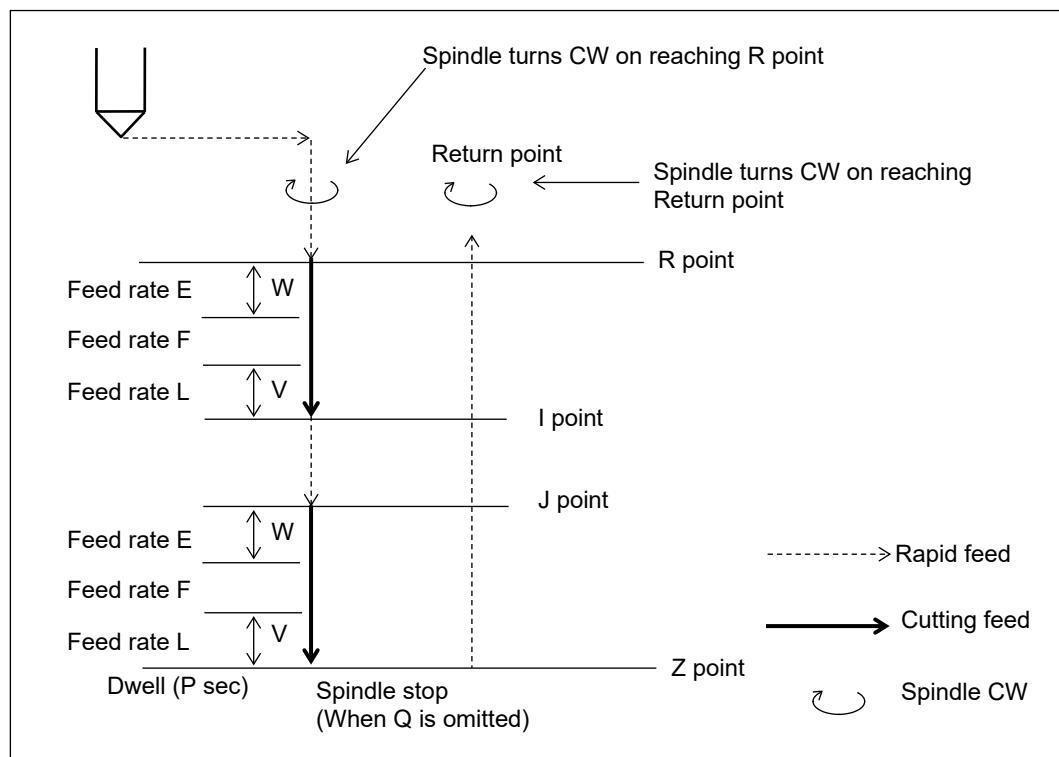
### 5.5.16 Double Boring Cycle (G186)

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 rotatational 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

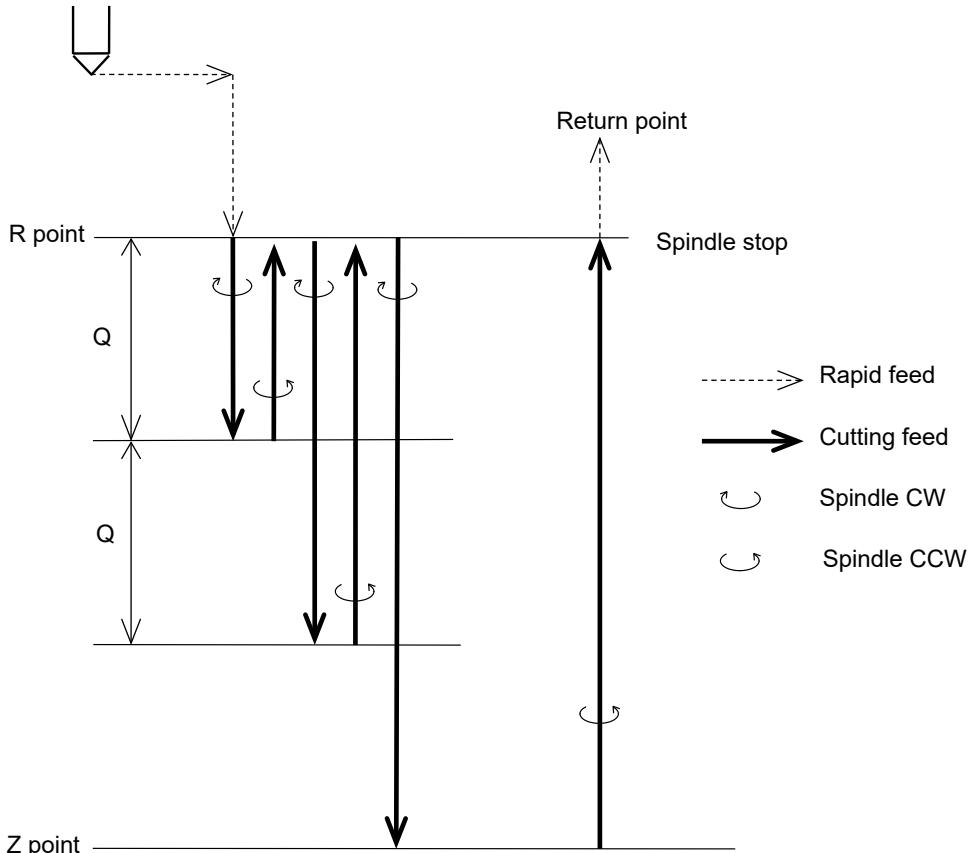
5



### 5.5.17 Deep Hole Tapping Cycle (Synchro Mode) (G277)

Command format

**G277 X\_ Y\_ Z\_ R\_ [I\_ J\_ ] Q\_ S\_;**



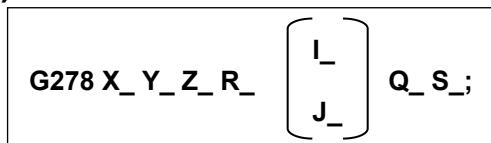
5

- 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 is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the user parameter (switch 1: canned cycle) <Tapping cycle return speed> is set to <1: Max. speed>, it uses the machine parameter (system 1: common) <Max. tapping speed> to return.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered. When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

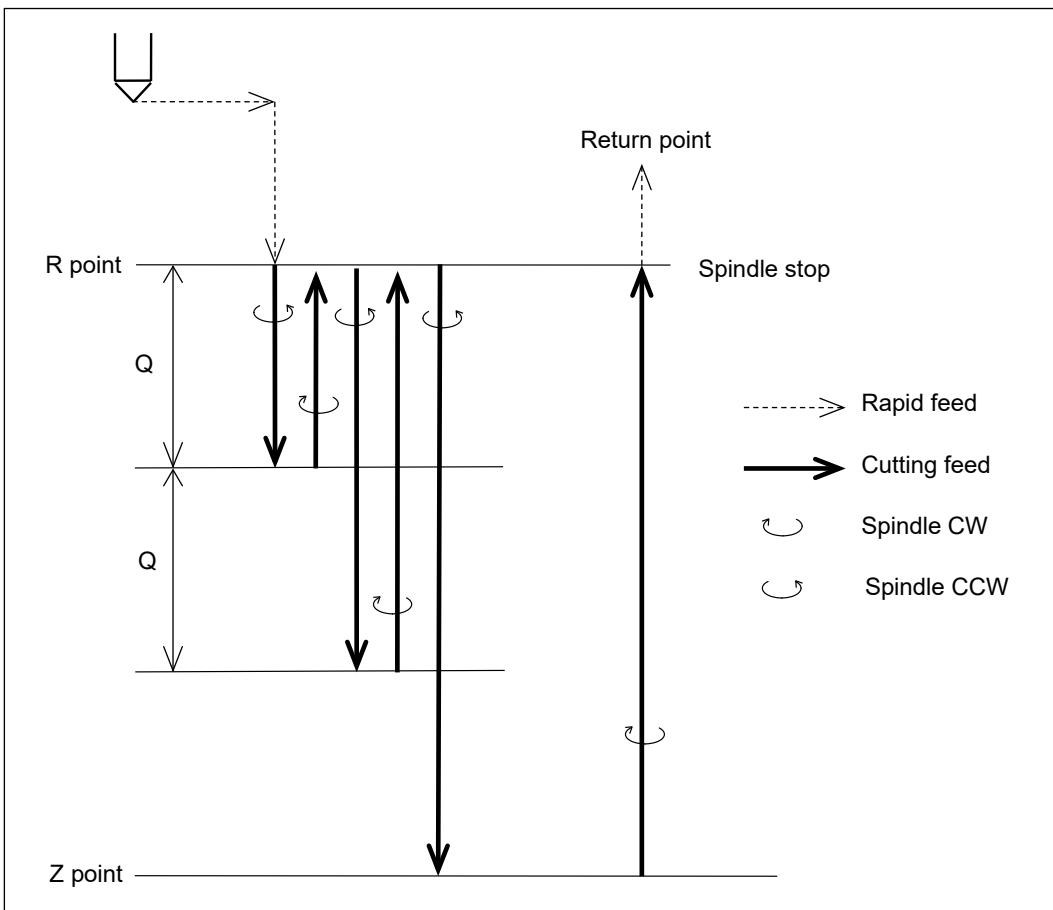
- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- 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



5



- 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 is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.

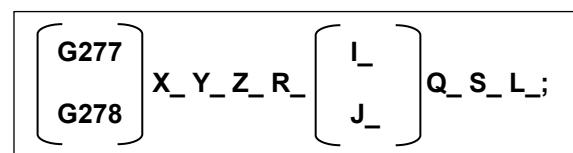
- When the user parameter (switch 1: canned cycle) <Tapping cycle return speed> is set to <1: Max. speed>, it uses the machine parameter (system 1: common) <Max. tapping speed> to return.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- 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

### High speed tap return

Spindle speed at return of synchro tap (G277, G278) is varied.

Commad format



- Address L commands spindle speed at returnm.
- 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 is triggered when the command value for the L address is larger than the machine parameter (system 1: common) <Max. tapping speed>.
- The tool moves according to the address S value when address L value is smaller than it.
- Even when the user parameter (switch 1: canned cycle) <Tapping cycle return speed> is set to <1: Max. speed>, the L address is given priority.
- When the user parameter (switch 1: common) <Tap override> is set to another setting that is not <0: Disable>, the operation uses the speed value that incorporates the feedrate override or spindle override in the command value for the L address.

## 5.5.19 Reducing Step of Canned Cycle

Cutting depth is gradually reduced in canned cycles of G73, G77, G78, G83, G173, and G183.

### 5.5.19.1 High speed peck drilling cycle (G73) (reducing step)

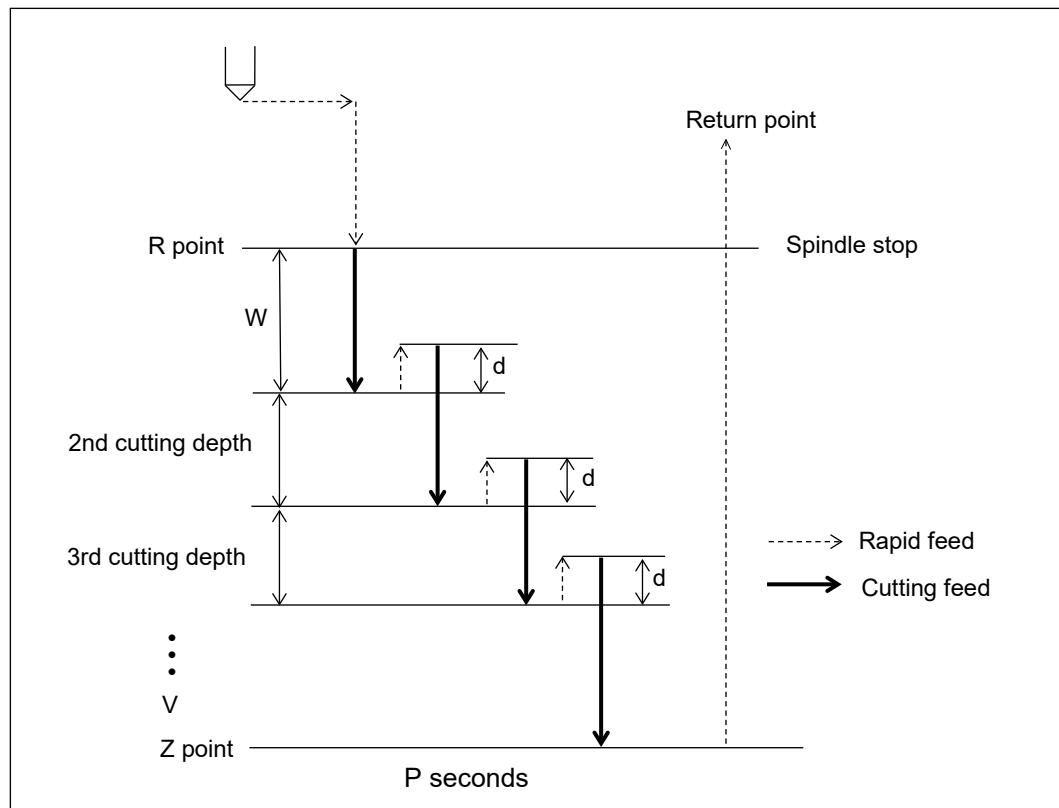
Command format

**G73 X\_ Y\_ Z\_ R\_ P\_ W\_ V\_ F\_;**

W : 1st cutting feed

V : Minimum cutting feed

5



- The relief amount d is set in the user parameter (switch 1: canned cycle) <G73 relief amount>.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

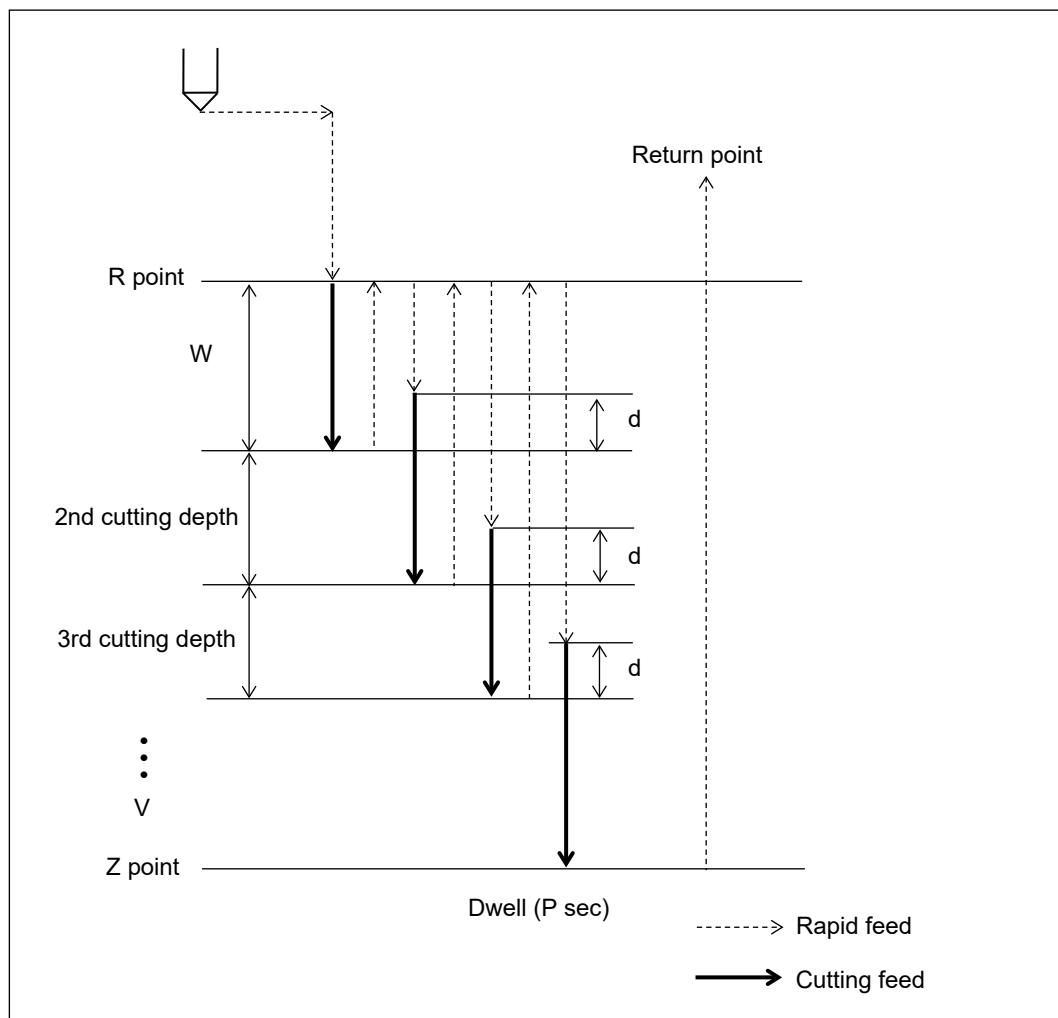
### 5.5.19.2 Peck drilling cycle (G83) (reducing step)

Command format

G83 X\_ Y\_ Z\_ R\_ P\_ W\_ V\_ F\_;

W : 1st cutting feed  
 V : Minimum cutting feed

5



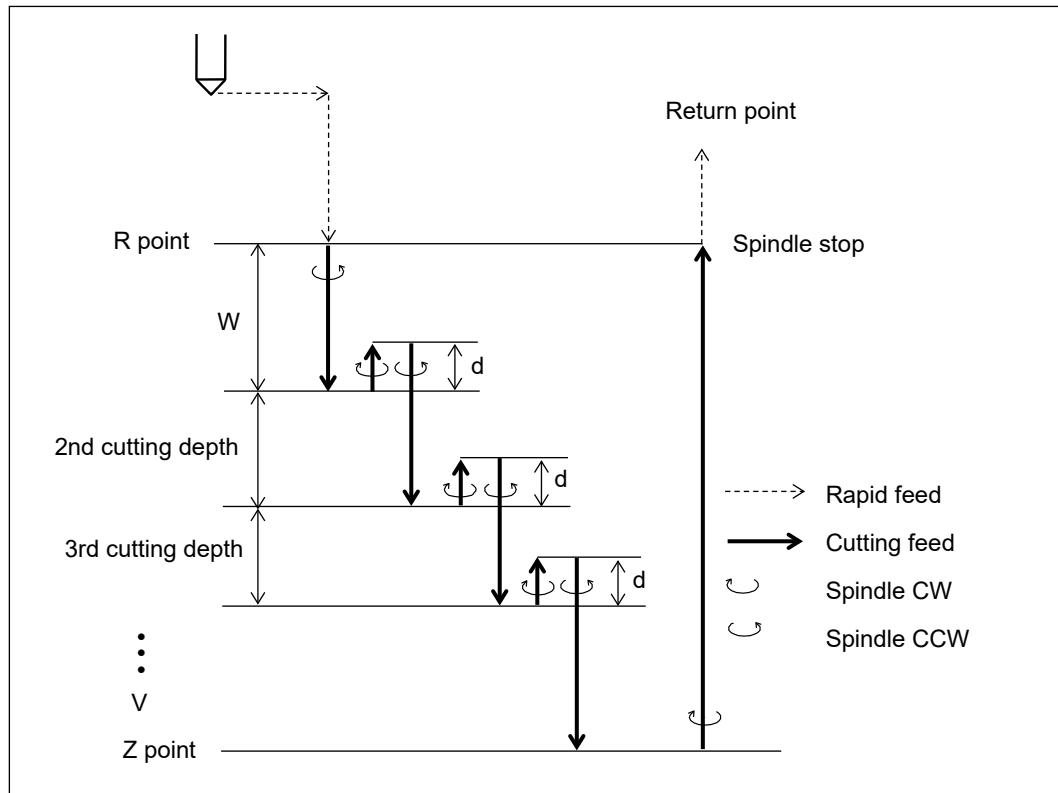
- The cutting start position d is set in the user parameter (switch 1: canned cycle) <G83 cutting start position>.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

### 5.5.19.3 Tapping cycle (synchro mode) (G77) (reducing step)

Command format

**G77 X\_ Y\_ Z\_ R\_**  $\left[ \begin{array}{c} L \\ J \end{array} \right]$  W\_ V\_ S\_;

W : 1st cutting feed  
V : Minimum cutting feed



5

- The relief amount d is set in the user parameter (switch 1: canned cycle) <G77, G78 relief amount>.
- 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>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <>The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- When the screw pitch is less than the <Minimum tapping pitch> in the machine parameter (system 1: common), the alarm <>Pitch data error>> is triggered.

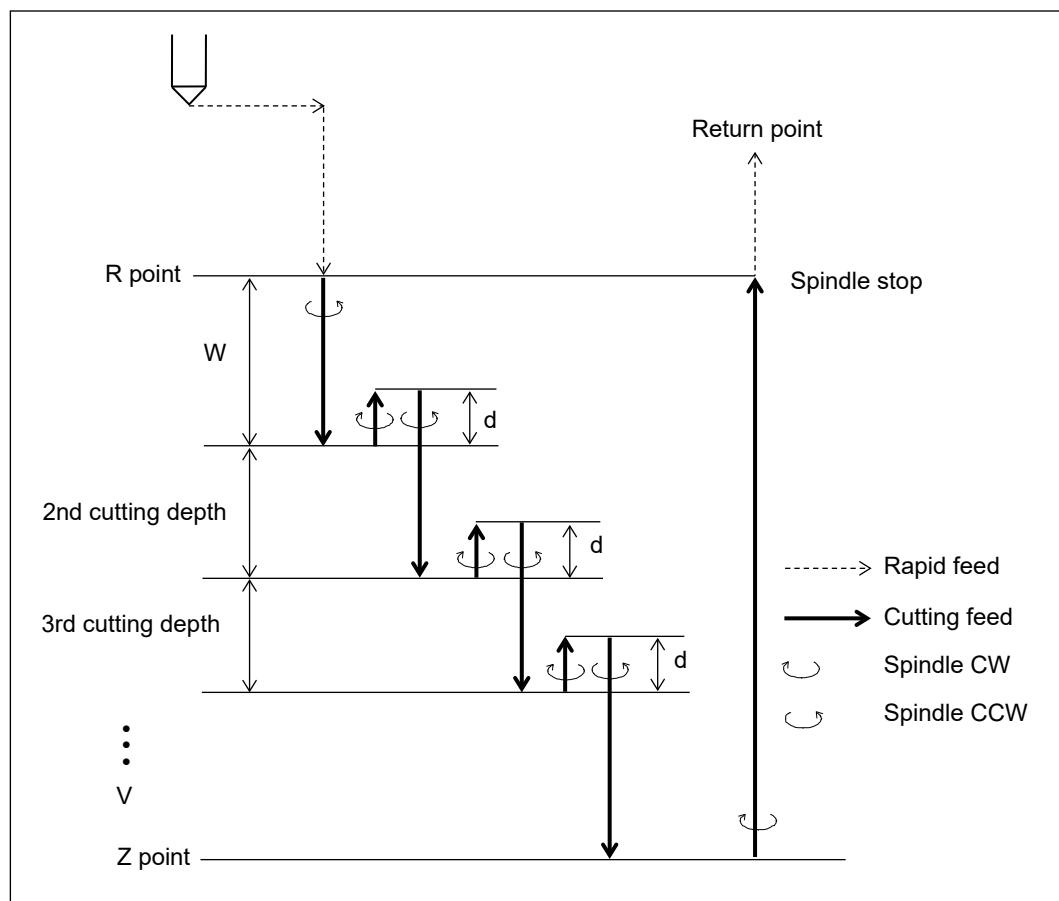
#### 5.5.19.4 Reverse tapping cycle (synchro mode) (G78) (reducing step)

Command format

**G78 X\_ Y\_ Z\_ R\_ [ I\_- J\_- ] W\_ V\_ S\_;**

5

W : 1st cutting feed  
V : Minimum cutting feed



- The relief amount d is set in the user parameter (switch 1: canned cycle) <G77, G78 relief amount>.
- 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>> is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <>The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- When the screw pitch is less than the <Minimum tapping pitch> in the machine parameter (system 1: common), the alarm <>Pitch data error>> is triggered.

### 5.5.19.5 Deep hole tapping cycle (synchro mode) (G277) (reducing step)

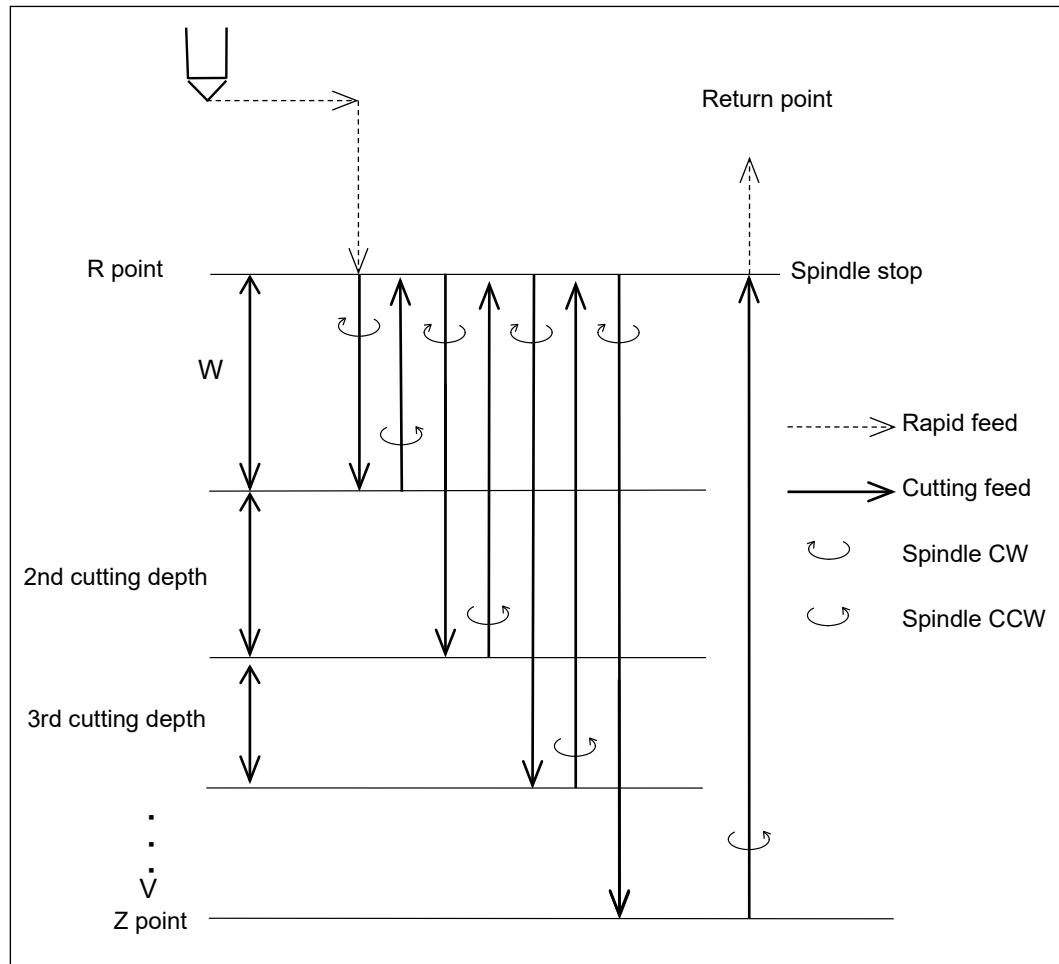
Command format

|                         |           |                  |
|-------------------------|-----------|------------------|
| <b>G277 X_ Y_ Z_ R_</b> | <b>L_</b> | <b>W_ V_ S_;</b> |
|                         | <b>J_</b> |                  |

W : 1st cutting feed

V : Minimum cutting depth

5



- 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 is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.
- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <>The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.

- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the spindle override is greater than 100%, the motion is carried out at 100%.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- When the screw pitch is less than the <Minimum tapping pitch> in the machine parameter (system 1: common), the alarm <>Pitch data error.>> is triggered.

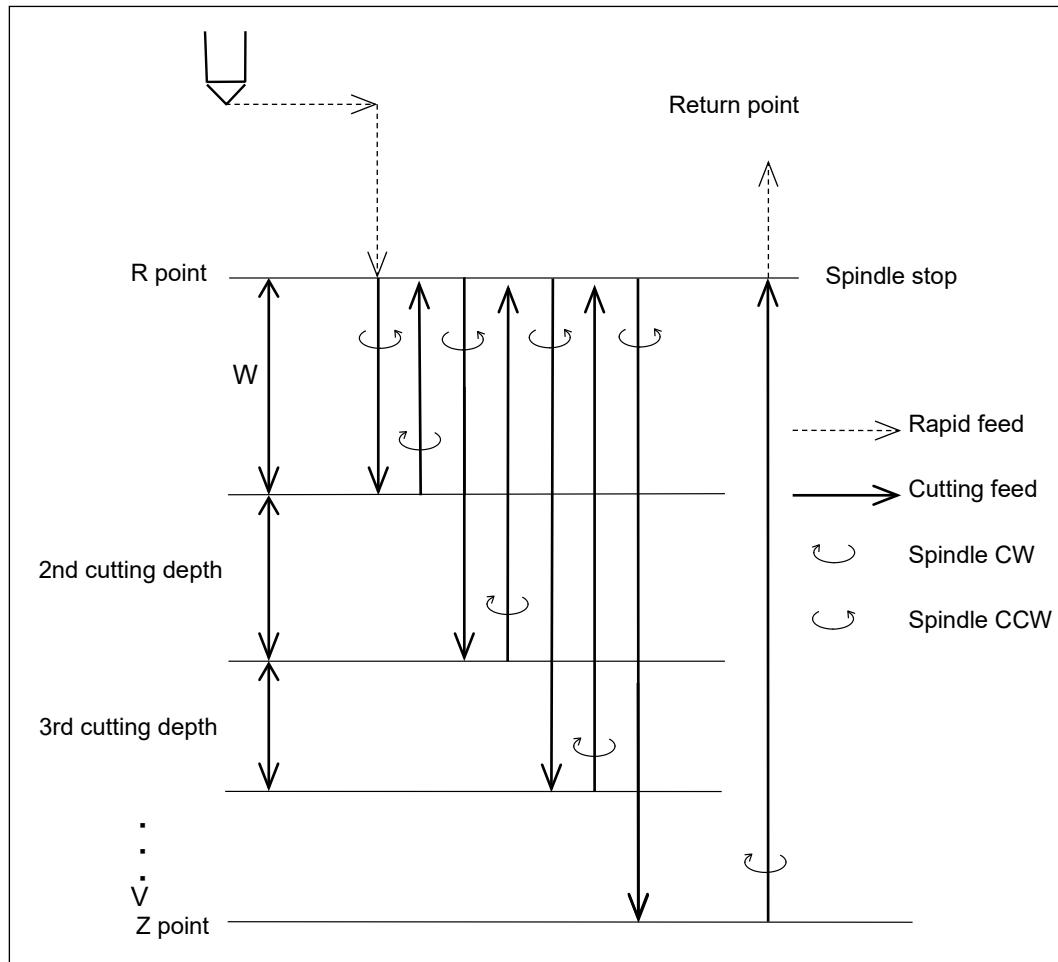
### 5.5.19.6 Reverse deep hole tapping cycle (synchro mode) (G278) (reducing step)

Command format

```
G278 X_ Y_ Z_ R_ [I_ J_] W_ V_ S_;
```

W : 1st cutting feed  
V : Minimum cutting feed

5



- 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 is triggered when S exceeds the machine parameter (system 1: common) <Max. tapping speed>.

- When the synchronized error for the Z-axis and spindle exceeds the machine parameter (system 1: common) <Synchronized tapping error limit> during tapping, the alarm <<The tapping synchronous error is too big.>> is triggered.  
When the user parameter (switch 1: canned cycle) <Tapping sync. error 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: canned cycle) <Tapping sync. error too big - alarm stop level> is set to <1: Level 5> and this alarm is triggered, it immediately stops.
- When the user parameter (switch 1: common) <Tap override> is set to <1: Spindle override>, the spindle override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out.
- When the user parameter (switch 1: common) <Tap override> is set to <2: Feedrate override>, the feedrate override is applied to the cutting feedrate and spindle speed during the tapping infeed and when pulling out. However, if the cutting feedrate override is greater than 100%, the motion is carried out at 100%.
- When the screw pitch is less than the <Minimum tapping pitch> in the machine parameter (system 1: common), the alarm <<Pitch data error>> is triggered.

### **5.5.19.7 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)

**5**

|        |       |       |       |       |      |     |       |       |
|--------|-------|-------|-------|-------|------|-----|-------|-------|
| 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.

### **5.5.19.8 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

**G80;**

(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

### 5.5.21 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. If an M00 or M01 command is issued on the same block as the canned cycle command, the spindle and coolant stop after X-axis and Y-axis positioning operations.  
When the user parameter (switch 1: operation) is set to the following, machine operation recovers automatically.
  - When <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>
 If the parameter is not set, issue a command in manual operation or MDI operation when necessary, because it will not carry out the automatic return operation at the next startup.
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.  
If a command is issued, the alarm <>Simultaneous specified code cannot be used.>> is triggered.
6. A canned cycle 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.
7. While the Z-axis perimeter mode is on (M300), the Z-axis perimeter operation is carried out when there are consecutive operations for the return height (rapid feed only) and the X-axis or Y-axis positioning operations. However, it does not operate right after the tapping cycle (G74, G77, G78, G84, G177, G178, G277 and G278) recovery operation. Refer to “Chapter 12 M function” for further details on the Z-axis perimeter operation.

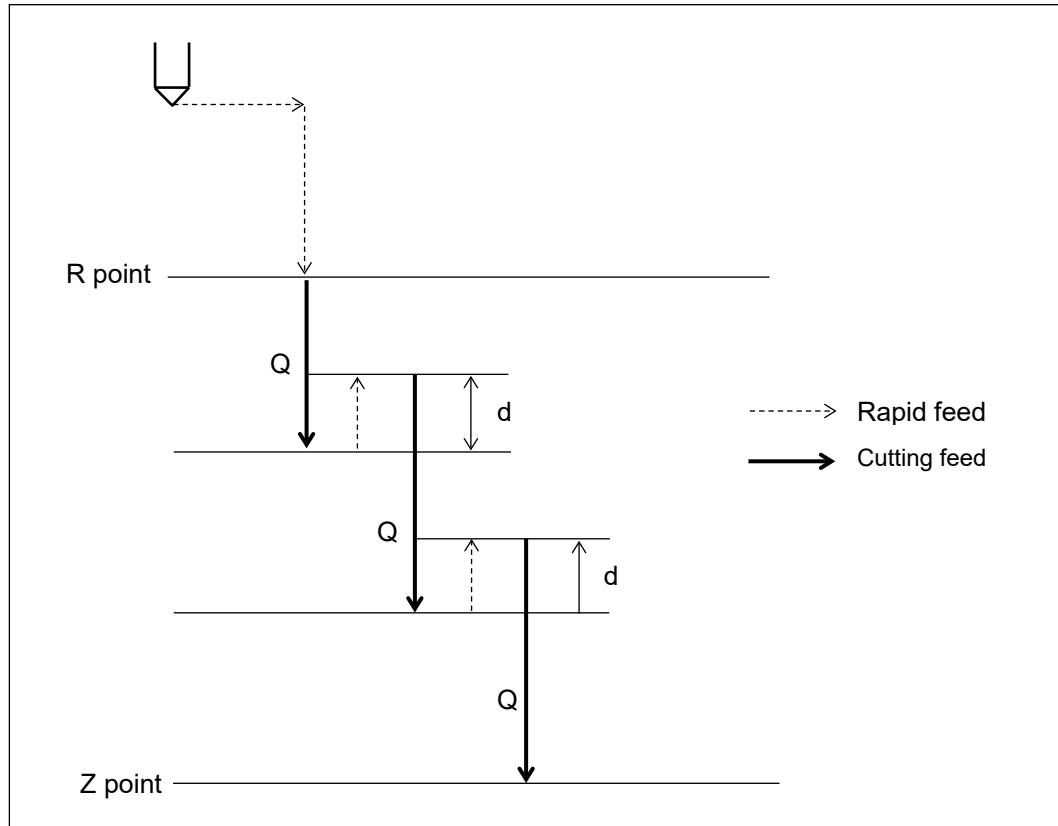
## 5.6 One-shot Canned Cycle

### 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)

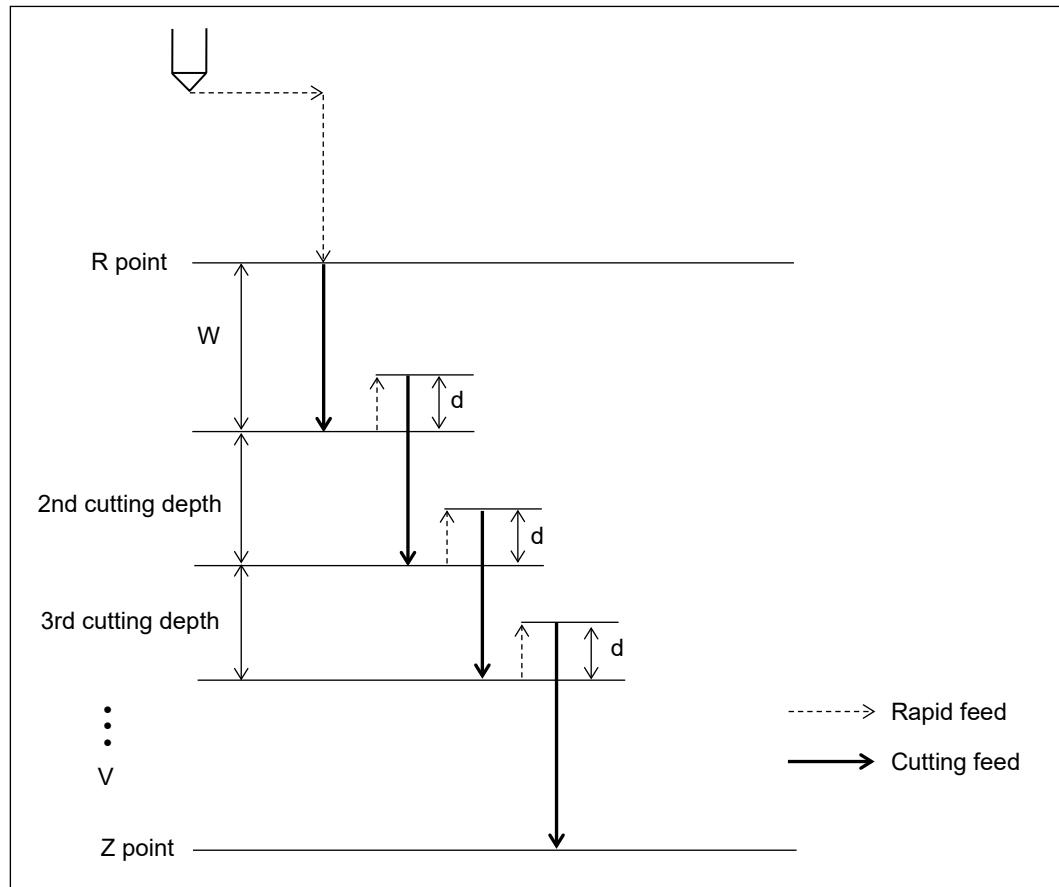
Reducing step is available which reduces the cutting depth gradually.  
For the 2nd and after cutting depth, refer to 5.5.19 Reducing step of canned cycle.

Command format

G173 X\_ Y\_ Z\_ R\_ W\_ V\_ F\_;

W : 1st cutting feed  
V : Minimum cutting feed

5



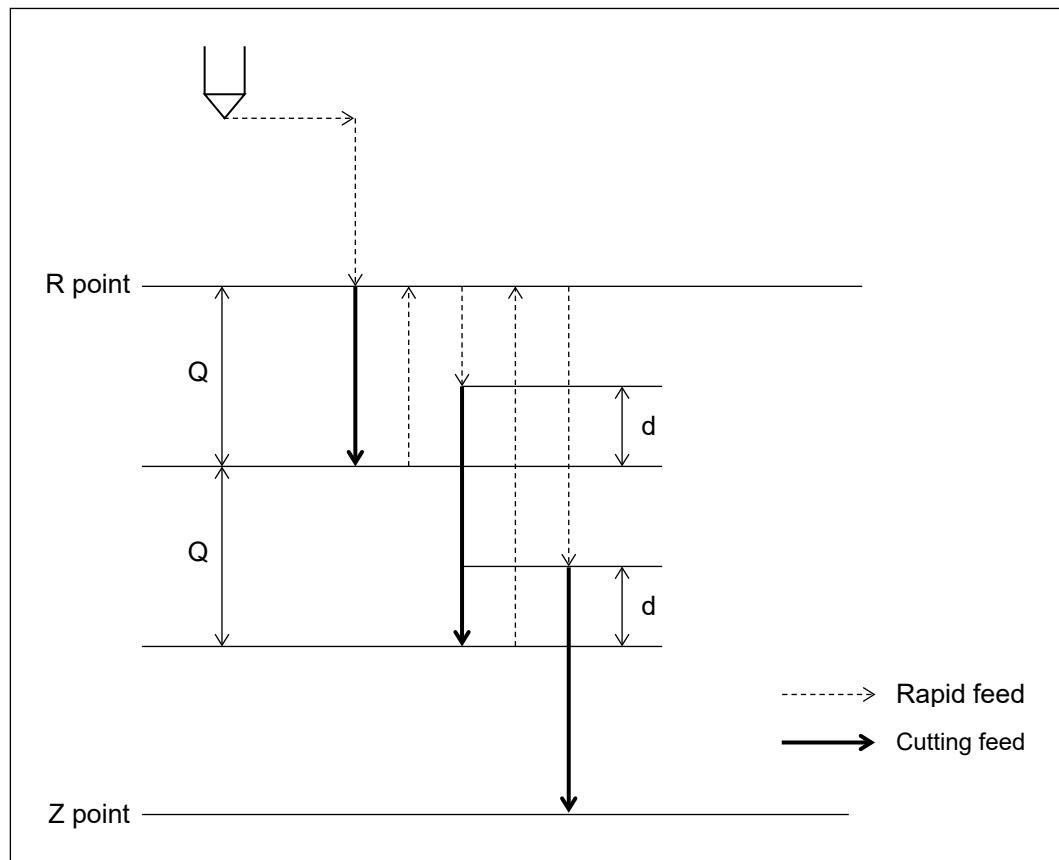
- The relief amount d is set in the user parameter (switch 1: canned cycle) <G73 relief amount>.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

## 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.



5

- 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.

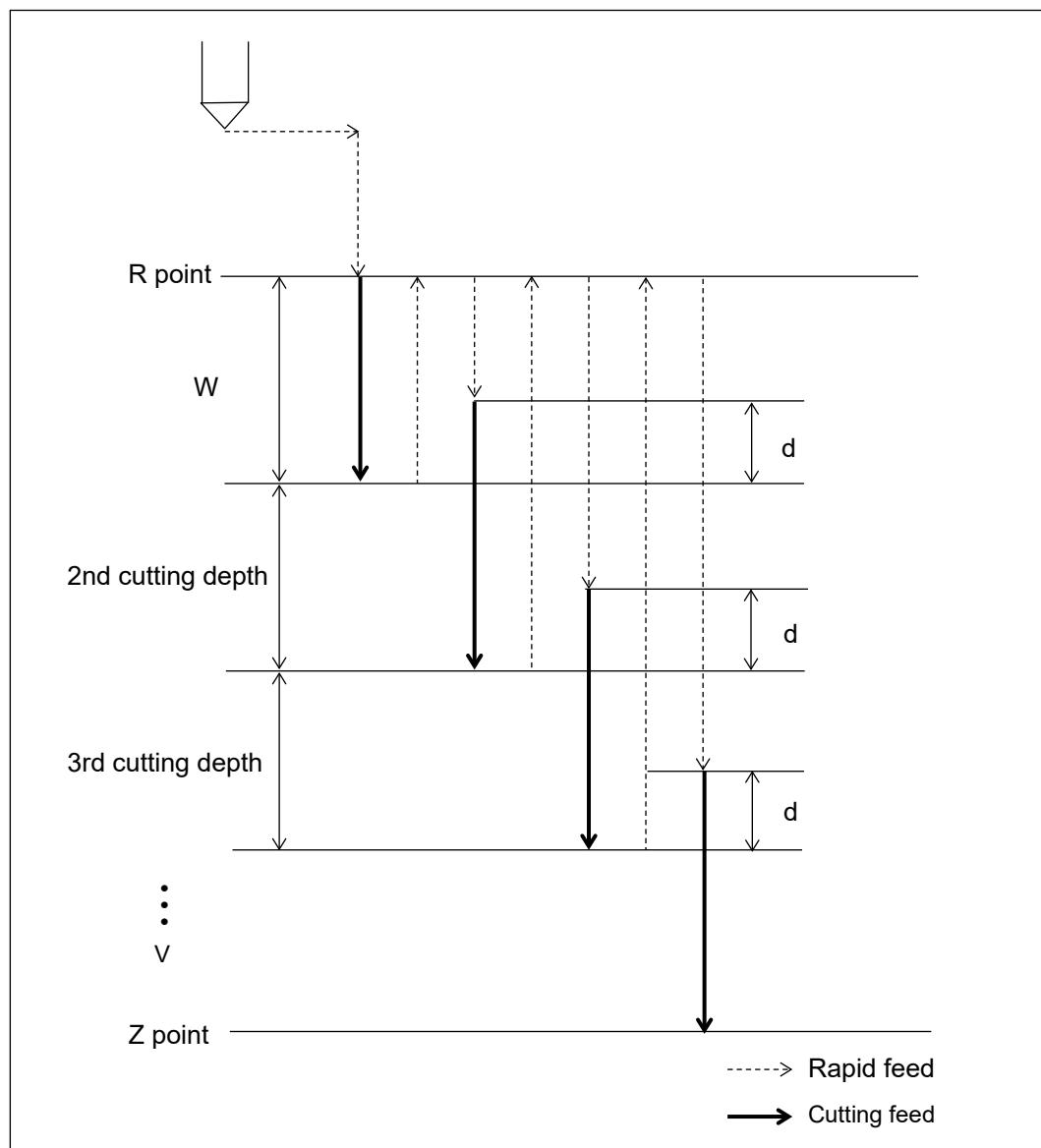
Command format

**G183 X\_ Y\_ Z\_ R\_ W\_ V\_ F\_;**

W : 1st cutting feed

V : Minimum cutting feed

5

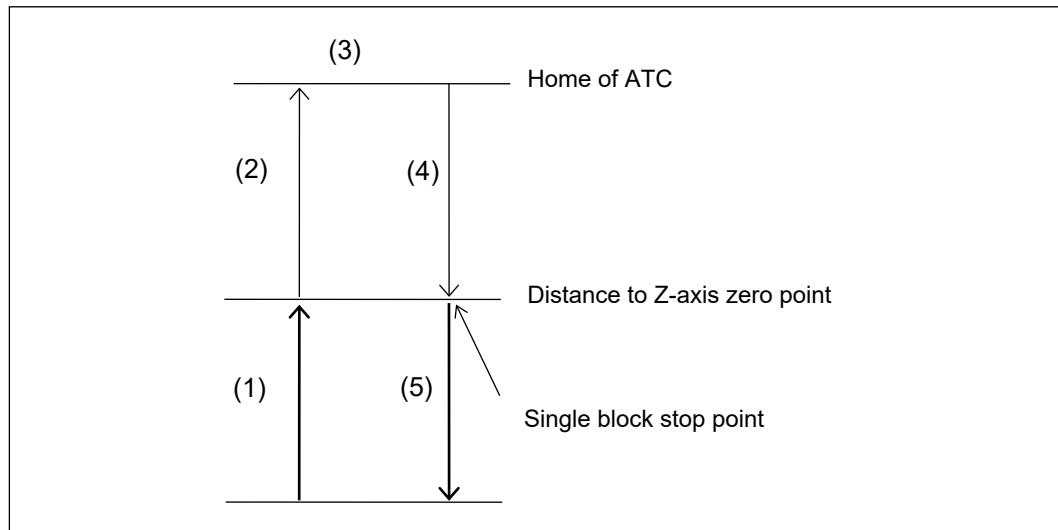


- The cutting start position is set in the user parameter (switch 1: canned cycle) <G83 cutting start position>.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

## 5.7 Canned Cycle for Tool Change (Non-stop ATC) (G100)

### 5.7.1 W1000Xd1/S300Xd1/S500Xd1/S700Xd1/U500Xd1/R450Xd1/R650Xd1

Command format

**G100 T\_X\_Y\_Z\_R\_A\_B\_C\_L\_;**

5

- T : Tool number (1 to 99, 201 to 299), or pot number (magazine number) (101 to 199), or group number (901 to 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 : Specifies the tool number, pot number (magazine number) and group number after L.  
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.

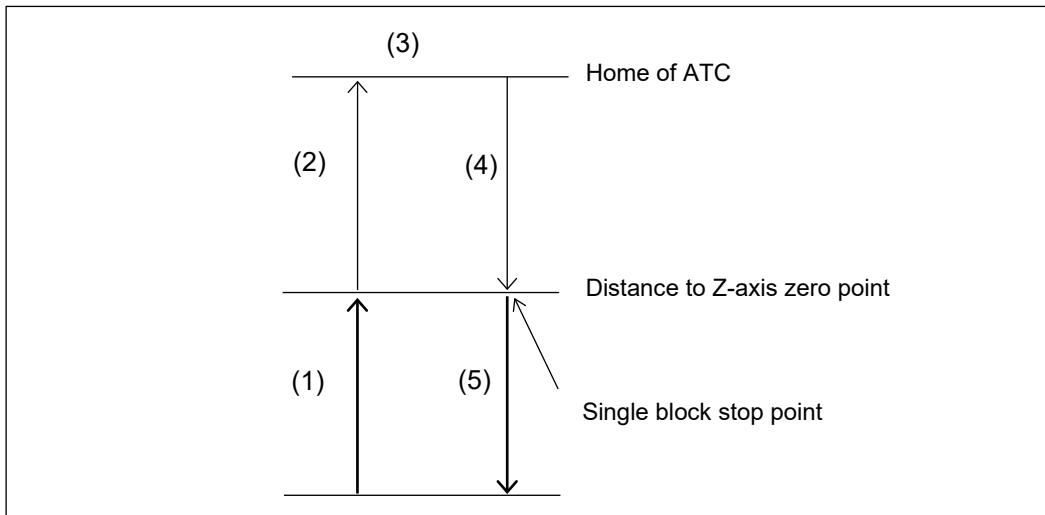
- (NOTE 1) During cycle operation, (1) and (2), and also (4) and (5) are linked together in their movement during machining mode.
- (NOTE 2) The machine stops after operation (4) when the [RST] key or the [FEED HOLD] switch is pressed between (2) and (4). However, the X-, Y-, A-, B- and C-axis travel 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 carried out in the travel for the selected plane axis in (2). Regardless of the user parameter (switch 1: compensation function) <Start up/cancel>, the startup operation follows the Type 1 (shortcut) 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 <<Simultaneous specified 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 user parameter (switch 1: ATC/Magazine) <ATC simultaneous operation start position> is not used.
- (NOTE 11) The user parameter (switch 1: ATC/Magazine) <ATC reference tool length> 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.
- (NOTE 14) The TCP control (G43.4/G43.5) command is cancelled when the G100 command is issued. G100 and TCP control commands can be issued on the same block. Refer to “14.2 TCP control” for further details.

## 5.7.2 M200Xd1

Command format

```
G100 G100 T_ X_ Y_ Z_ R_ A_ B_ C_ L_;
```



5

- T : Tool number (1 to 99, 201 to 299), or pot number (magazine number) (101 to 199), or group number (901 to 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 : Specifies the tool number, pot number (magazine number) and group number after L.  
The number specified by L is the T modal after G100.

### 5.7.2.1 Operation when spindle is selected (M141 modal)

- Travels the distance to Z-axis zero position and performs the spindle orientation at the same time.  
(Performs the C-axis unclamp operation at the same time as well.)
- 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.
- The magazine turns and indexes the tool specified in T.
- Travels to the Z-axis zero position.
- Travels to the position that is specified when the Z-axis command is issued.  
The operations are carried out at the same time when the spindle command (M03 group) is issued.

### 5.7.2.2 Operation when lathe spindle is selected (M142 modal)

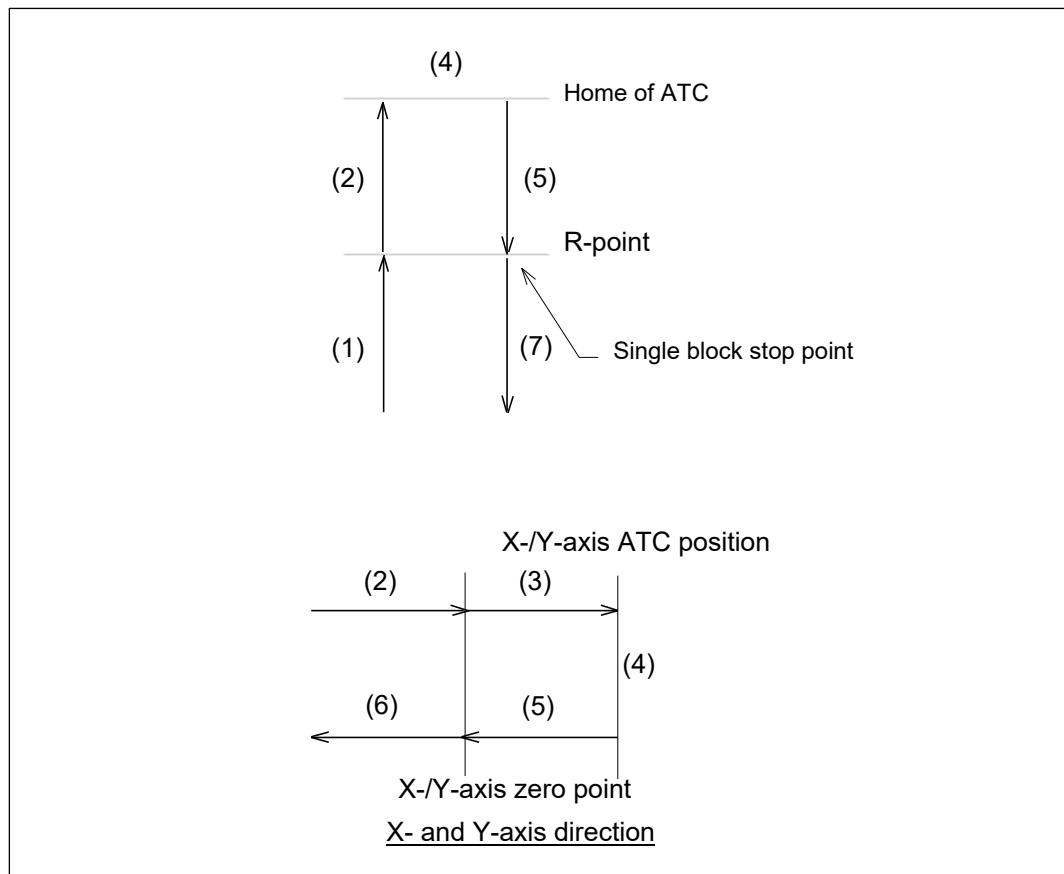
- Travels the distance to Z-axis zero position and performs the spindle orientation at the same time.  
Performs the lathe spindle operation at the same time when the lathe spindle command (M303 to M305) is issued.
- Travels to the ATC zero point.  
Travels along the corresponding axes at the same time when X-, Y-, A- and B -axes are specified.
- The magazine turns and indexes the tool specified in T.
- Travels to the Z-axis zero position.
- Travels to the position that is specified when the Z-axis command is issued.

### 5.7.2.3 Notes

- (NOTE 1) During cycle operation, travel connecting (1) and (2) as well as (4) and (5) occurs in cutting mode.
- (NOTE 2) When the [RST] key or the [FEED HOLD] switch is pressed between operations (2) and (4), it stops after operation (4). However, the X-, Y-, A- and B-axes 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 the T code command has to be issued once before the G100 command is issued.
- (NOTE 6) The cutter / nose R compensation is cancelled when the G100 command is issued. In addition, the tool length / tool position offset (Z-axis) is cancelled from (1).
- (NOTE 7) When cutter / nose R compensation commands on the G100 block and travel commands on the selected plane axis are issued, the cutter / nose R compensation startup operation is performed in the travel for the selected plane axis in (2). Regardless of the user parameter (switch 1: compensation function) <Start up/Cancel>, the startup operation follows 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 “5.7.4 M code for simultaneous commands” (described later) are valid for the G100 block. The alarm <<Simultaneous specified code cannot be used. >> is triggered when an unlisted or invalid M code command is issued.
- (NOTE 10) The alarm <<No \*-axis option>> is triggered if A-axis, B-axis and C-axis commands are issued when there is no A-axis, B-axis and C-axis options.
- (NOTE 11) The user parameter (switch 1: ATC/Magazine) <ATC simultaneous operation start position> is not used.
- (NOTE 12) The user parameter (switch 1: ATC/Magazine) <ATC reference tool length> is not used. Even if the G100 command is issued when the tool length / tool position offset is cancelled (G49), (1) travels to the Z-axis zero position (distance to Z-axis zero point).
- (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 G100 and G53.1 commands are issued on the same block in a 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.
- (NOTE 15) The spindle selection command (or the lathe spindle selection command) and the tool change command can be performed on the same block. Refer to “8.2.3 Lathe spindle function” in Operation Manual I for further details.
- (NOTE 16) The TCP control (G43.4/G43.5) command is cancelled when the G100 command is issued. G100 and TCP control commands can be issued on the same block. Refer to “14.2 TCP control” for further details.

### 5.7.3 R650Xd1 40MG

Command format

**G100 T\_X\_Y\_Z\_R\_A\_B\_C\_L\_;**

- T : Tool number (1 to 99, 201 to 299), or pot number (magazine number) (101 to 199), or group number (901 to 930).
- A, B, C : Target values when moving A-, B- and C-axes at the same time as the tool change operation.  
Travel is rapid feed.
- X, 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/magazine <ATC simultaneous operation start position>) is used as the R command value for operation.
- L : Specifies the tool number, pot number (magazine 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-axes travel to the command value.
- (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 a 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.  
The mode cannot be changed between operations (1) and (6).
- (NOTE 6) 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 7) The cutter compensation is cancelled when the G100 command is issued.  
In addition, the tool length offset is canceled from (2).
- (NOTE 8) 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: compensation function) <Start up/cancel> value, the startup operation follows the Type 1 (shortcut) setting.
- (NOTE 9) The Z-axis compensation is enabled from operation (5) when the tool length offset command is issued on the G100 block.
- (NOTE 10) When the G100 command is issued while the tool length offset is cancelled (G49), the user parameter (switch 1: ATC/magazine) setting <ATC reference tool length> compensates for the tool length in operation (1).
- (NOTE 11) When there is no tool length offset command in the G100 block, the user parameter (switch 1: ATC/magazine) setting <ATC reference tool length> compensates for the tool length or position in operation (5).
- (NOTE 12) Only the M code commands listed in the “M code for simultaneous commands” (described later) are valid for the G100 block. The alarm <<Simultaneous specified code cannot be used. >> is triggered when an unlisted or invalid M code command is issued.
- (NOTE 13) 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 14) When the Z-axis travel falls for operation (1), the Z-axis does not travel in operation (1).
- (NOTE 15) 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 16) 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 17) 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.
- (NOTE 18) 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) When a G100 command is issued, the machining load monitor (M340) is disabled in the machining load monitor function.

- (NOTE 20) 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 21) 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: common) <R-point lower limit for pallet loading during tool change>, the alarm <<Pallet turn restricted range error>> is triggered.

### 5.7.3.1 Tool change operation

- 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 one <Standard tool> and another one, 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 one <Large tool> and another one, the arm turns at a speed that is set by the machine parameter (system 3: common) <Rapid feedrate AT-axis 2>.

- When changing from <Large tool> to <Standard tool> or from <Standard tool> to <Large 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: common) <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: common) <Rapid feedrate AT-axis 1>.

- (NOTE 1) When changing from a <Large tool> to <Standard tool>, if there is no empty pot for the <Large tool>, the alarm <<No empty pot>> is triggered.
- (NOTE 2) When changing from a <Standard tool> to <Large tool>, if there is no empty pot for the <Standard tool>, the alarm <<No empty pot>> is triggered.

### 5.7.3.2 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 tool> 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 tool> to <Large tool>, if there is no empty pot for the <Standard tool>, the alarm <<No empty pot>> is triggered.

5

### 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.

The alarm <<Simultaneous specified code cannot be used.>> is triggered if an M code command not listed below is issued.

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                    |
| M258         | Production monitor – Time measurement start |
| M259         | Production monitor – Time measurement end   |
| M290         | Tool replacement Z axis lower speed 100%    |
| M291 to M293 | Tool replacement Z axis lower speeds 1 to 3 |
| M300         | Z-axis perimeter mode on                    |
| M301         | Z-axis perimeter mode off                   |
| 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                              |

| M code           | Contents                                                                                   |
|------------------|--------------------------------------------------------------------------------------------|
| M407             | M406 signal OFF                                                                            |
| 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                                                                       |
| M423             | ATC arm turn speed 2                                                                       |
| 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<br>(Proceeds to the next block after the signal has turned off)     |
| M451             | One shot signal output<br>(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                                                                            |
| M494             | Coolant through center ON                                                                  |
| M495             | Coolant through center OFF                                                                 |
| M497             | Tool replacement tool washing ON                                                           |
| M498             | Spindle air blow / Tool wash ON                                                            |
| M499             | Spindle air blow / Tool wash OFF                                                           |
| M801 to M899     | Signal output * for PLC                                                                    |
| M900 to M999     | Extend signal output                                                                       |
| 2-bit BCD signal | BCD signal output                                                                          |

## Chapter 5 Preparation Function (Canned Cycle)

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.

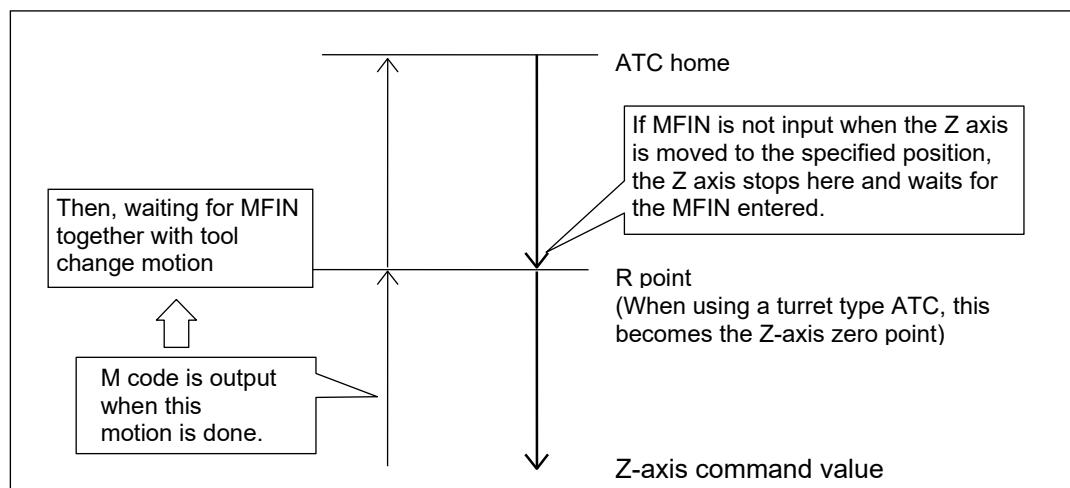
- (1) M03 (Spindle forward rotation)
- (2) M04 (Spindle backward rotation)
- (3) M05 (Spindle stop)
- (4) M19 (Spindle orientation)
- (5) M111 (Spindle orientation to 180°)
- (6) M08 (Coolant pump ON)
- (7) M494 (Coolant through center ON)

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)

5

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.



When the user parameter (switch 1: program) <Multiple M codes in one block> is set to <1: Yes>, up to 3 M code commands can be issued on 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(switch 1: canned cycle) | 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

5

Command format

|                 |                       |
|-----------------|-----------------------|
| G36<br>:<br>G39 | X_ Y_ I_ J_ K_ P_ Q_; |
|-----------------|-----------------------|

X, Y : Datum point coordinates

I, J, K, P, Q : Coordinates calculation parameters

1. 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.
2. 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 1) 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.

(NOTE 2) A coordinate calculation function command cannot be issued while under TCP control (G43.4/G43.5). Otherwise, the alarm <>TCP under control>> is triggered.

## 5.8.4 Description of Coordinate Calculation Function

### 5.8.4.1 Bolt Hole Circle (G36)

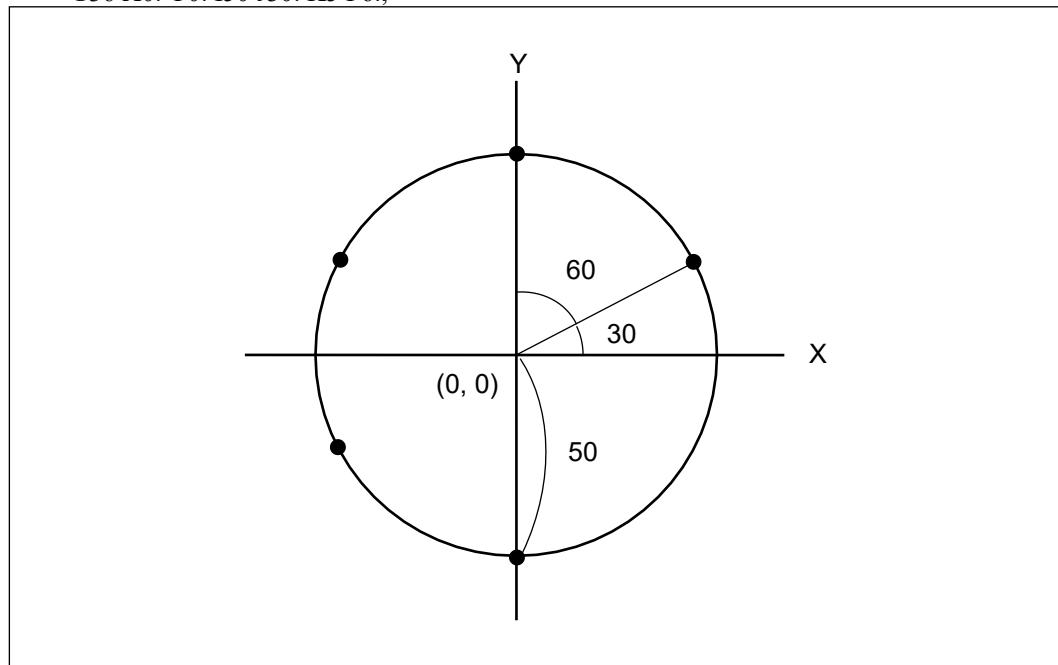
The commanded coordinate value is the center of the arc and the partitioned coordinate values are found for the discretionary points on the circumference as a starting point, etc.

Command format

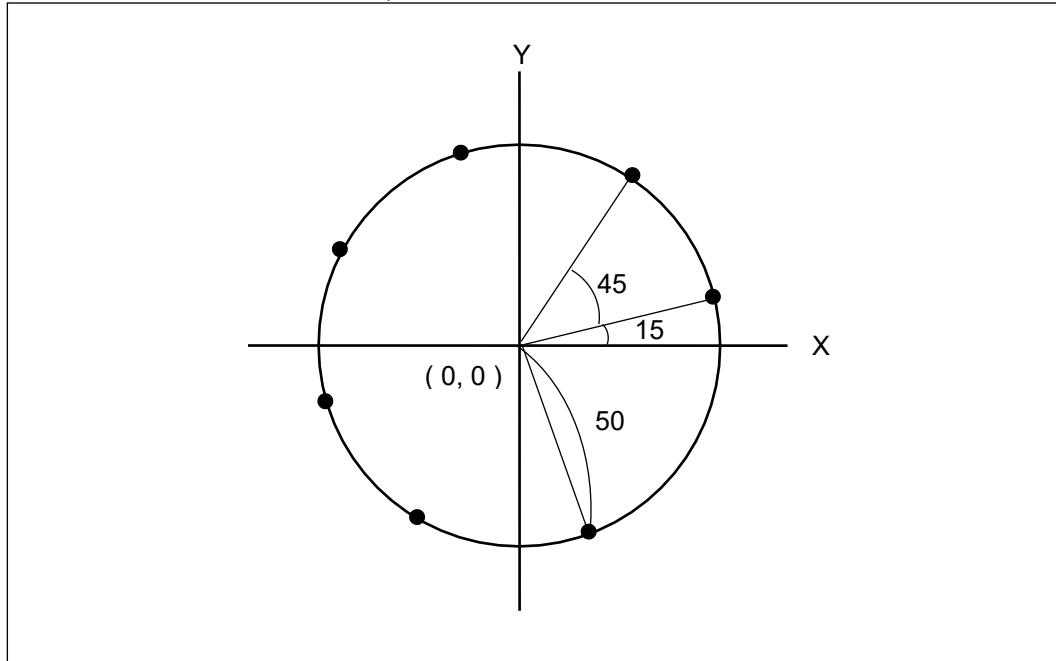
**G36 X\_ Y\_ I\_ J\_ K\_ P\_;**

- 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) When using P6 where P is the number of intervals (360 degrees / 60 degrees = 6 intervals)  
G36 X0. Y0. I50 J30. K5 P6.;



Ex) When using P8 where P is the number of intervals ( $360 \text{ degrees} / 45 \text{ degrees} = 8 \text{ intervals}$ )  
 G36 X0. Y0. I50. J15. K7 P8.;



**5**

(NOTE) Coordinate values are measured counterclockwise from the start point.

#### **5.8.4.2 Line (Angle) (G37)**

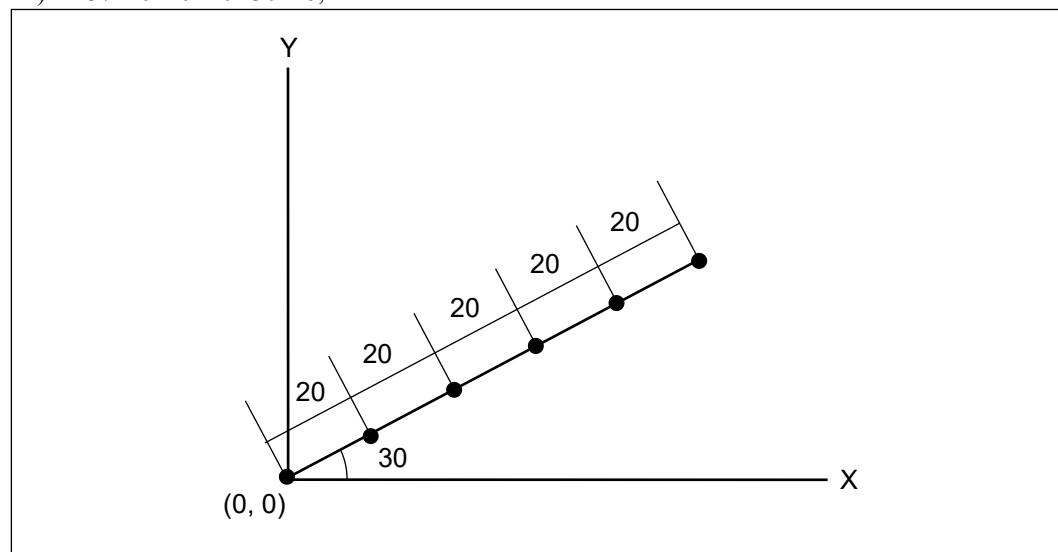
The commanded coordinate value is assumed as the datum point, and the coordinate value is found that lines up on the straight line in the angle ( $\theta^\circ$ ) direction for the X axis.

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)

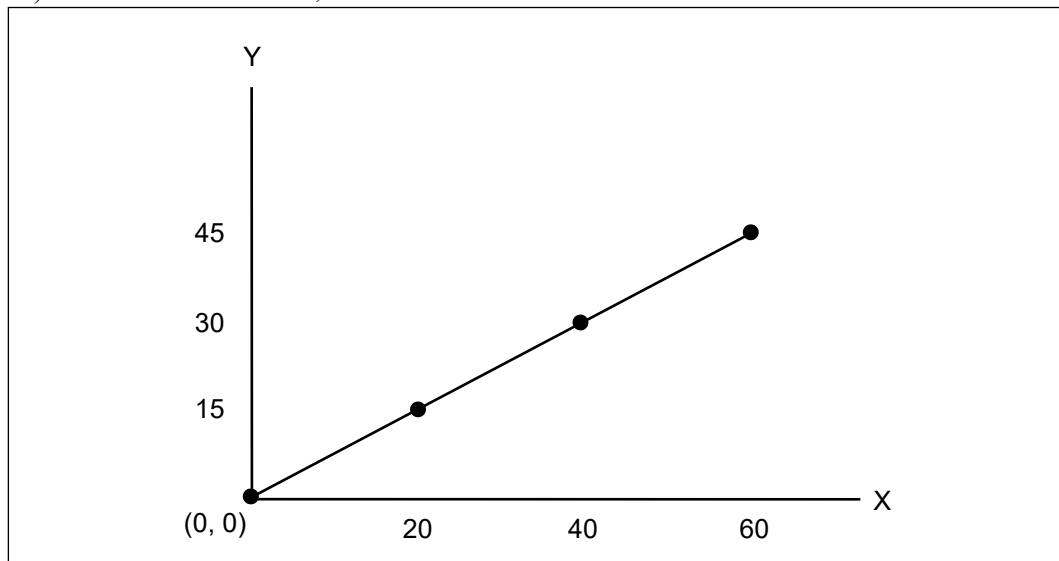
The commanded coordinate value is assumed to be a datum point, and the coordinate values are found that have a plus for both the X direction and the Y direction respectively.

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;



5

(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.

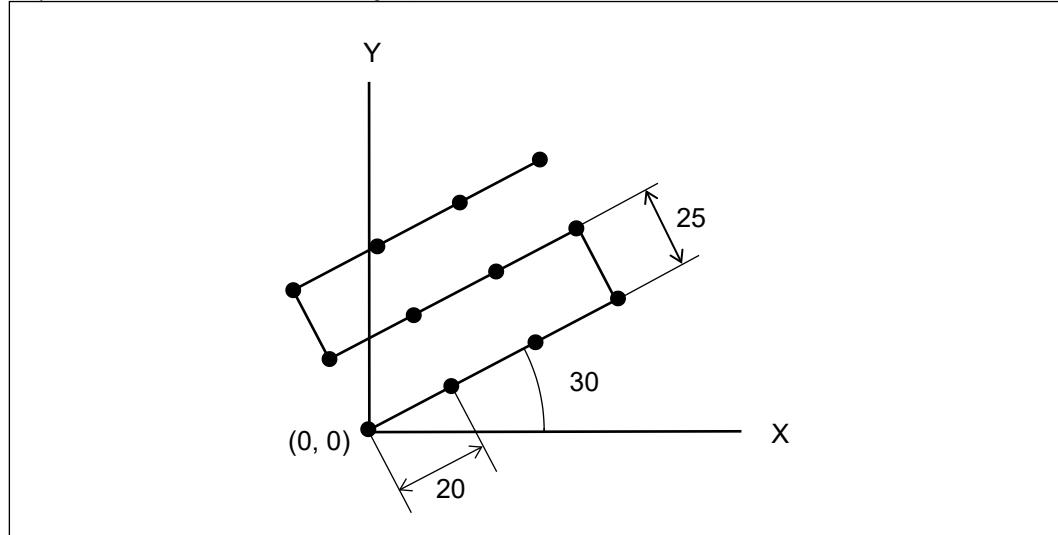
Command format

**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;

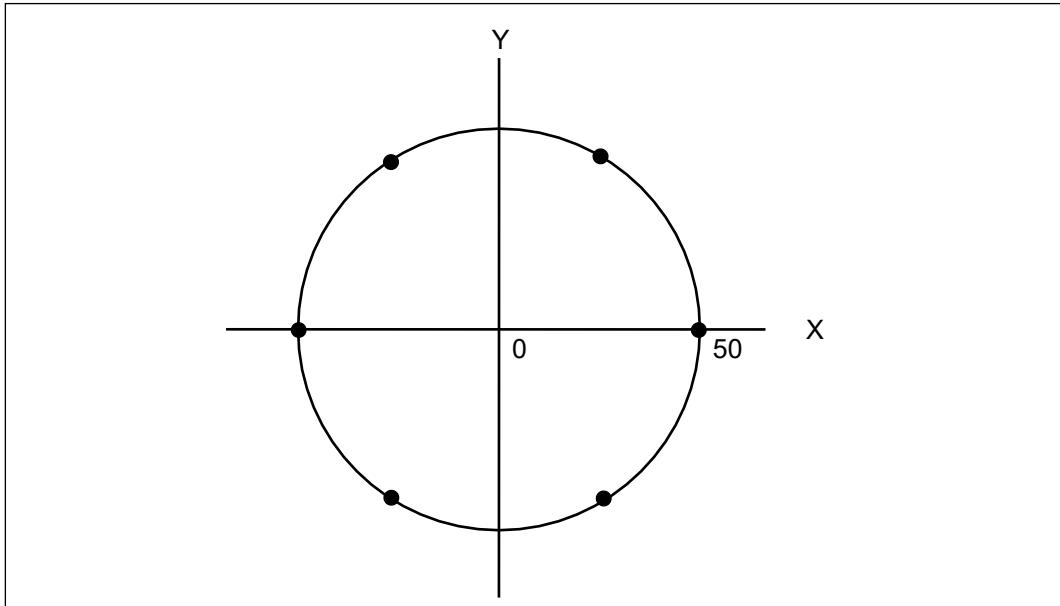
5



(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



5

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.

(This page was intentionally left blank.)

# 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

## 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;
```

6

Ex. 2) Arc machining by specifying the center, radius, and angle.

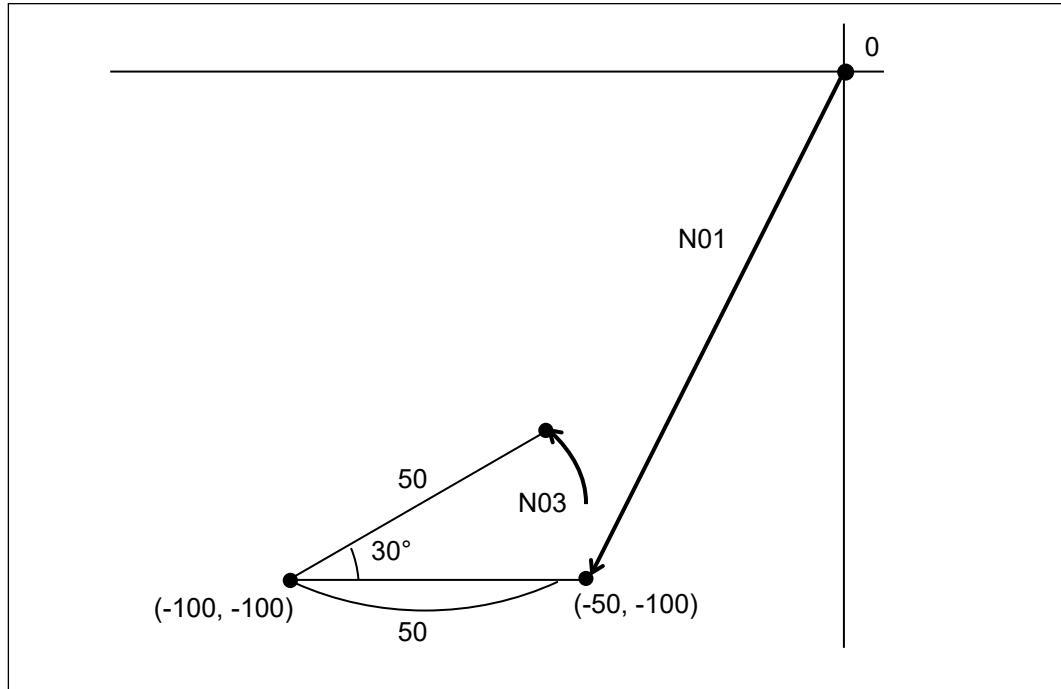
|                        |                        |                      |                       |
|------------------------|------------------------|----------------------|-----------------------|
| X: center X            | Y: center Y            | R: radius            | Z: Cutting position Z |
| W: 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 agree 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

### 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>$-1.0 \times 10^{99}$ to $-1.0 \times 10^{-99}$ , 0, $1.0 \times 10^{-99}$ to $1.0 \times 10^{99}$<br>(NOTE) All digits are not necessarily displayed on the screen but actual variables can be represented in the above range.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                         |
| #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>$-1.0 \times 10^{99}$ to $-1.0 \times 10^{-99}$ , 0, $1.0 \times 10^{-99}$ to $1.0 \times 10^{99}$<br>(NOTE) All digits are not necessarily displayed on the screen but actual variables can be represented in the above range.<br>Initialization is also carried out when the setting is changed in the user parameter (switch 1)<br><Conversation/NC language change>.                                                                                                                                                                                                                                                                                                                                                                                                                                 |
| #500 to #999 |                  | Used in all different macro programs commonly.<br>The data is retained when power is turned off.<br>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.<br>(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: -99999.9999 to 99999.9999 (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

## 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 |
| (G54.1P1)                        | #14001 to #14006 | R/W |
| (G54.1P2)                        | #14021 to #14026 | R/W |
| •                                | •                | R/W |
| •                                | •                | R/W |
| (G54.1P300)                      | #19981 to #19986 | R/W |

### 6.2.6.3 Tool Data

Tool compensation/life data are read and written.

|                                     |                                                                |                                                    |
|-------------------------------------|----------------------------------------------------------------|----------------------------------------------------|
| Tool length offset                  | #11001 to #11099(T01 to T99)<br>#11201 to #11299(T201 to T299) | R/W                                                |
| T length wear offset                | #10001 to #10099(T01 to T99)<br>#10201 to #10299(T201 to T299) | R/W                                                |
| Cutter compensation                 | #13001 to #13099(T01 to T99)<br>#13201 to #13299(T201 to T299) | R/W                                                |
| Tool diameter wear offset           | #12001 to #12099(T01 to T99)<br>#12201 to #12299(T201 to T299) | R/W                                                |
| Tool position offset (X)            | #25001 to #25099(T01 to T99)<br>#25201 to #25299(T201 to T299) | R/W, Available when equipped with a lathe function |
| Tool position wear offset data (X)  | #20001 to #20099(T01 to T99)<br>#20201 to #20299(T201 to T299) | R/W, Available when equipped with a lathe function |
| Tool length offset (Z)              | #26001 to #26099(T01 to T99)<br>#26201 to #26299(T201 to T299) | R/W, Available when equipped with a lathe function |
| T length wear offset (Z)            | #21001 to #21099(T01 to T99)<br>#21201 to #21299(T201 to T299) | R/W, Available when equipped with a lathe function |
| Tool diameter / nose R compensation | #27001 to #27099(T01 to T99)<br>#27201 to #27299(T201 to T299) | R/W, Available when equipped with a lathe function |
| Tool diameter / nose R wear offset  | #22001 to #22099(T01 to T99)<br>#22201 to #22299(T201 to T299) | R/W, Available when equipped with a lathe function |
| Tool position offset (Y)            | #29001 to #29099(T01 to T99)<br>#29201 to #29299(T201 to T299) | R/W, Available when equipped with a lathe function |
| Tool position wear offset data (Y)  | #24001 to #24099(T01 to T99)<br>#24201 to #24299(T201 to T299) | R/W, Available when equipped with a lathe function |
| Virtual teeth direction             | #23001 to #23099(T01 to T99)<br>#23201 to #23299(T201 to T299) | R/W, Available when equipped with a lathe function |

|                                      |                                                                                              |                                                                                                           |
|--------------------------------------|----------------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------|
| Tool life unit                       | #5501 to #5599(T01 to T99)<br>#31001 to #31099(T01 to T99)<br>#31201 to #31299(T201 to T299) | R/W<br>1: Is not counted<br>2: Time (min.)<br>3: Drilling (holes)<br>4: Program (times)<br>5: Time (sec.) |
| Initial tool life / End of tool life | #5601 to #5699(T01 to T99)<br>#32001 to #32099(T01 to T99)<br>#32201 to #32299(T201 to T299) | R/W                                                                                                       |
| Life warning                         | #5701 to #5799(T01 to T99)<br>#33001 to #33099(T01 to T99)<br>#33201 to #33299(T201 to T299) | R/W                                                                                                       |
| Tool life                            | #5801 to #5899(T01 to T99)<br>#38001 to #38099(T01 to T99)<br>#38201 to #38299(T201 to T299) | 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.

6

(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) When an attempt is made to assign the date (#3011) as a common variable (#500 to #999), the alarm <<Macro variable error>> is triggered.

- (NOTE 2) If the user parameter (switch 1: system) <Machine unit system> is set to <1:Inch>, when an attempt is made to assign the time (#3012) as a common variable (#500 to #999), the alarm <<Macro variable error>> is triggered.

### 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><b>[Override]</b><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. The M signal output time (when set to <Do not wait>) is set in the user parameter (switch 1: programming) <External signal output time when MFIN is invalid>.
- 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 the feedrate override is disabled, the override is fixed at 100% regardless of the feedrate override setting on the operations box.
- 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, G43.4, G43.5, 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                                                                                           |

| Variable No. | Contents                                                                                                                                    |
|--------------|---------------------------------------------------------------------------------------------------------------------------------------------|
| #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.  
When the user parameter (switch 1: programming) <Multiple M codes in one block> is set to <Yes> and multiple commands have been issued on one block, the M code number noted at the end of the block applies.

6

#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                                                                                     |
| #4422        | G50.1, G51.1                                                                                       |
| #4507        | D code                                                                                             |
| #4509        | F code                                                                                             |
| #4511        | H code                                                                                             |
| #4513        | M code                                                                                             |
| #4514        | Sequence No.                                                                                       |

| Variable No. | Contents                                                                                                                                    |
|--------------|---------------------------------------------------------------------------------------------------------------------------------------------|
| #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 or table coordinate system<br>(NOTE 1) | 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 or table coordinate system<br>(NOTE 1) | Follows the user parameter (switch 1: common) <System variable format (current position)>. <Spindle end position> → Included <TCP> → Not included | No                   |
| #5061~#5068<br>#5071~#5074     | Skip coordinates                   | Workpiece coordinate system or table coordinate system<br>(NOTE 1) | Follows the user parameter (switch 1: common) <System variable format (skip coordinate)>. <Spindle end position> → Included <TCP> → Not 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                                          | Follows the user parameter (switch 1: common) <System variable format (skip coordinate)>. <Spindle end position> → Included <TCP> → Not included  | Possible             |
| #30000                         | Spindle feedback position          |                                                                    |                                                                                                                                                   | No                   |

(NOTE 1) When the TCP control is OFF, the workpiece coordinate system is usually used. When under TCP control (G43.4/G43.5 modal), the coordinate system for the end point coordinate on X-, Y- and Z-axes (#5001 to #5003), for the current position (#5041 to #5043) and for skip coordinate (#5061 to #5063) follows the set value in the user parameter (5 axes machining: common) <Programming coordinate system>. Either the <Wrkpc. coord. sys.> or the <Table coordinate system> is used.

#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.  
#30000 reads the spindle feedback position. The range is 0° to 359.999°.

(NOTE) After the power is turned ON, the value is undefined until the spindle rotation is carried out.

Reading during travel is not possible for the current position, the tool length offset, the servo deviation and the spindle feedback position. However, this means that the value is not guaranteed to be accurate because the numerical value is set at the time when the look-ahead is carried out.

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.

|       |         |   |
|-------|---------|---|
| #3700 | Spindle | R |
| #3701 | Pot 1   | R |
| #3702 | Pot 2   | R |
| •     | •       | R |
| •     | •       | R |
| #3750 | Pot 50  | R |

0 : Cap designation  
1 to 99, 201 to 299 : 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),<br>G code 2 (NOTE 3) | R |
| #3611 to #3618 | Result of measurement 2 latest                                                                                    | R |
| #3821 to #3628 | 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 1                                           | 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)                                                         |   |
| #37000        | Most recent machining load for parameter No.201 (peak value)       | R |
| #37001        | Most recent machining load for parameter No.201 (average value)    | R |
| #37002~#37009 | (Reserved)                                                         |   |
| •             | •                                                                  | • |
| •             | •                                                                  | • |

|               |                                                                 |   |
|---------------|-----------------------------------------------------------------|---|
| #37980        | Most recent machining load for parameter No.299 (peak value)    | R |
| #37981        | Most recent machining load for parameter No.299 (average value) | R |
| #37982~#37999 | (Reserved)                                                      |   |

### 6.2.6.16 Rotation axis / Tilt axis

The set values are read for the user parameters and machine parameters used in the feature coordinate setting function and TCP control.

User parameter (rotation axis/tilt axis setting)

|               |                                                                |   |
|---------------|----------------------------------------------------------------|---|
| #39001        | Tilt axis 1                                                    | R |
| #39002        | Rotation axis 1                                                | R |
| #39003        | Tilt axis 2                                                    | R |
| #39004        | Rotation axis 2                                                | R |
| #39005        | Forward direction for the coordinate system on tilt axis 1     | R |
| #39006        | Forward direction for the coordinate system on rotation axis 1 | R |
| #39007        | Forward direction for the coordinate system on tilt axis 2     | R |
| #39008        | Forward direction for the coordinate system on rotation axis 2 | R |
| #39009        | Rotation center X coordinate offset for tilt axis 1            | R |
| #39010        | Rotation center Y coordinate offset for tilt axis 1            | R |
| #39011        | Rotation center Z coordinate offset for tilt axis 1            | R |
| #39012        | Rotation center X coordinate offset for rotation axis 1        | R |
| #39013        | Rotation center Y coordinate offset for rotation axis 1        | R |
| #39014        | Rotation center Z coordinate offset for rotation axis 1        | R |
| #39015        | Rotation center X coordinate offset for tilt axis 2            | R |
| #39016        | Rotation center Y coordinate offset for tilt axis 2            | R |
| #39017        | Rotation center Z coordinate offset for tilt axis 2            | R |
| #39018        | Rotation center X coordinate offset for rotation axis 2        | R |
| #39019        | Rotation center Y coordinate offset for rotation axis 2        | R |
| #39020        | Rotation center Z coordinate offset for rotation axis 2        | R |
| #39021~#39028 | (Reserved)                                                     |   |

6

Machine parameter (System 2: common)

|        |                                                  |   |
|--------|--------------------------------------------------|---|
| #39029 | Rotation center X coordinate for tilt axis 1     | R |
| #39030 | Rotation center Y coordinate for tilt axis 1     | R |
| #39031 | Rotation center Z coordinate for tilt axis 1     | R |
| #39032 | Rotation center X coordinate for rotation axis 1 | R |
| #39033 | Rotation center Y coordinate for rotation axis 1 | R |
| #39034 | Rotation center Z coordinate for rotation axis 1 | R |
| #39035 | Rotation center X coordinate for tilt axis 2     | R |
| #39036 | Rotation center Y coordinate for tilt axis 2     | R |
| #39037 | Rotation center Z coordinate for tilt axis 2     | R |
| #39038 | Rotation center X coordinate for rotation axis 2 | R |
| #39039 | Rotation center Y coordinate for rotation axis 2 | R |
| #39040 | Rotation center Z coordinate for rotation axis 2 | R |

## 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

Precidence 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.  
Ex.) For #1 = 12; #1 is 12.000.
- (NOTE 2) Angle calculation  
The unit of functions SIN, COS, TAN, and ATAN is degrees.  
Ex.) 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.  
Ex.) 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:  
-999999999999 to -0.00000000001,  
0, and  
+0.00000000001 to +999999999999  
Maximum number of digits available for a constant is 12 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.

When the sequence number *n* is out of range, the alarm <>Macro command error>> is triggered. When there is no corresponding sequence number, the alarm <>No applicable sequence>> is triggered.

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 1) If a command is issued in extended memory operation and tape operation, only searching within a range for approximately 60 KB is possible.
- (NOTE 2) If a GOTO command is issued on the same block as the block with the sequence number that was skipped using M98H/M99P while in extended memory operation or tape operation, then the alarm <>Macro command error>> is triggered. In addition, when a macro statement is issued on the same block as the sequence number block and only a macro statement is issued on the next block until a GOTO command, then the same alarm is also triggered.

## 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 or equal to #j    |

(NOTE 1) Conditional equation is placed within the brackets [ ].

(NOTE 2) The range of values used in a conditional equation is given below.

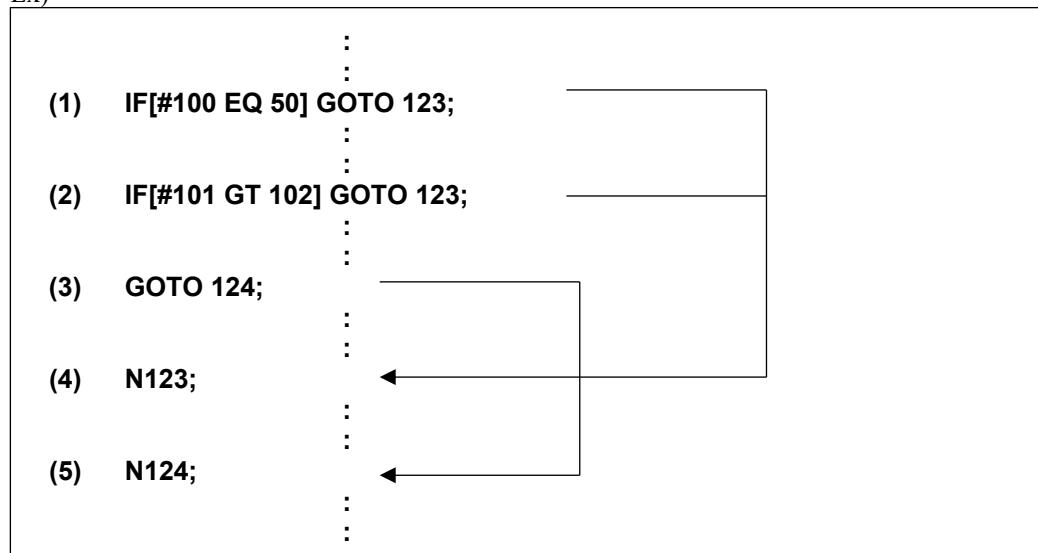
Metric: -9223372036854775.808 to 9223372036854775.807

Inch: -922337203685477.5808 to 922337203685477.5807

If an out of range numerical value is used, the alarm <>Macro variable error>> is triggered.

6

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≤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.

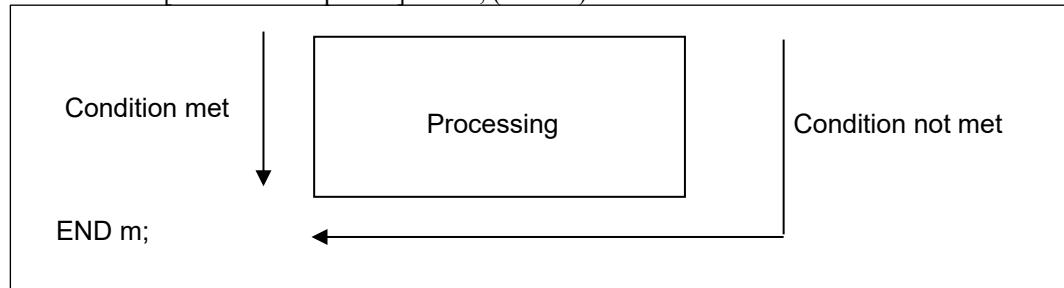
Programs between DO and END are executed as long as the conditional equation is met.

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

If an out of range numerical value is used, the alarm <<Macro variable error>> is triggered.

(NOTE 2) If a command is issued in extended memory operation and tape operation, the alarm <<Macro execution 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.

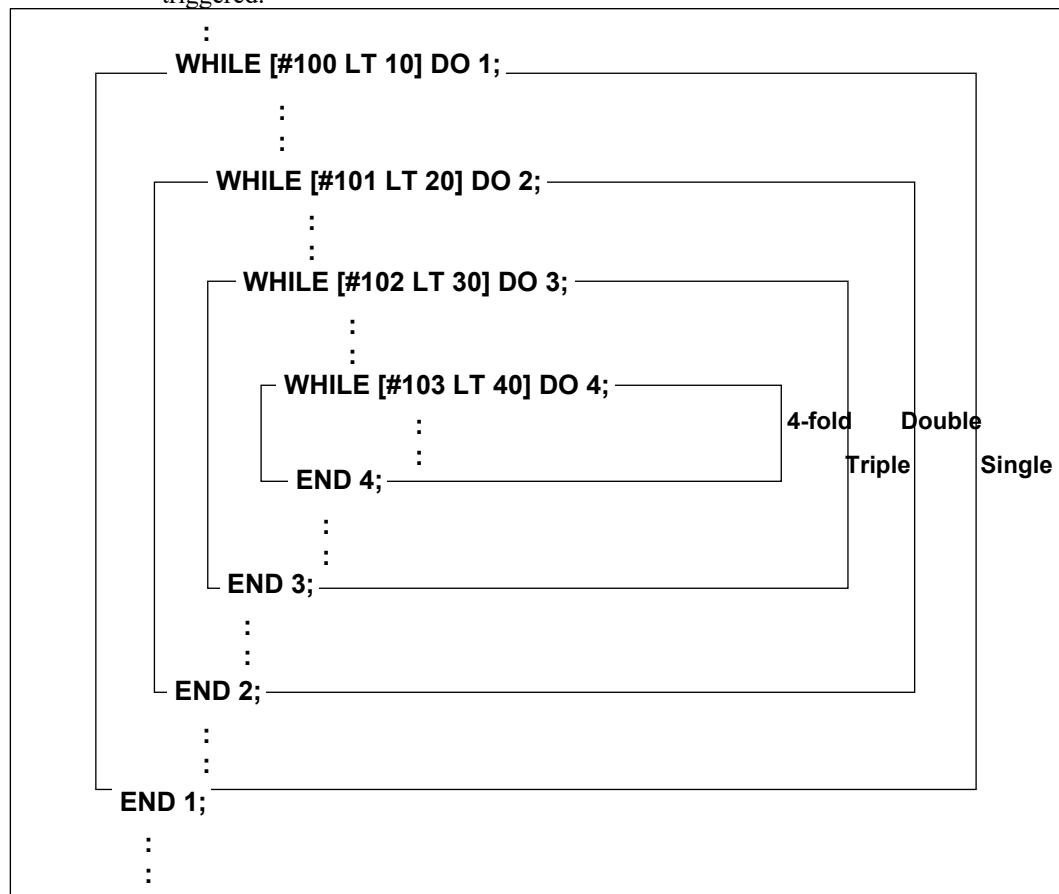
```
:
WHILE [#100 LT 10] DO 1; _____
:
:
END 1; _____
:
:
WHILE [#101 EQ 50] DO 1; _____
:
:
END 1; _____
:
```

6

(NOTE 2) You may not cross DOm and ENDm in WHILE instructions.  
The alarm <<Macro Command Error>> occurs.

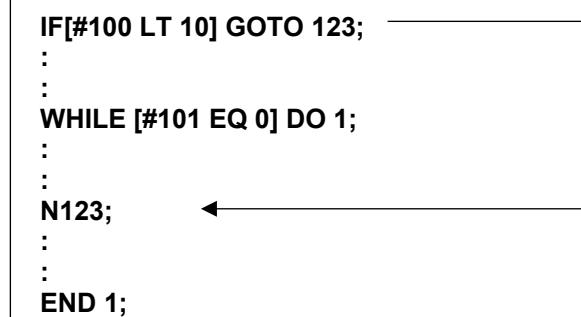
```
:
WHILE [#100 LT 10] DO 1; _____
:
:
WHILE [#101 EQ 50] DO 2; _____
:
:
END 1; _____
:
:
END 2; _____
:
```

(NOTE 3) Multiplicity of DOs in WHILE instructions is up to 4-fold.  
When the nesting has exceeded 4, the alarm <<Macro execution error>> is triggered.



6

(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

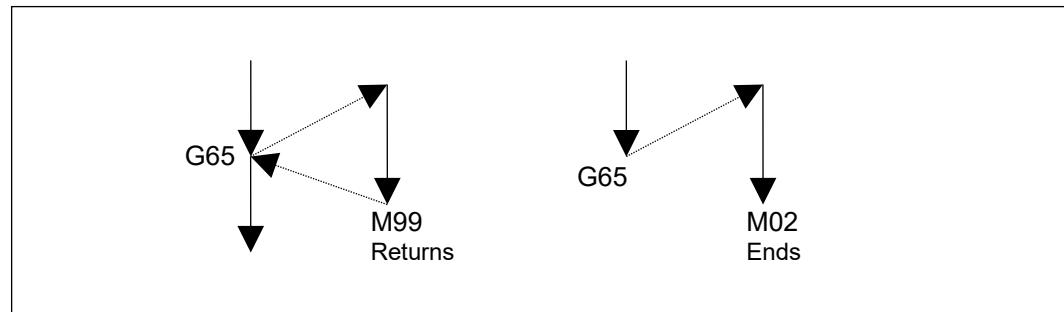
This function uses G65 or G66 and G code or M code that is registered in the data bank <G/M code macro> to call and execute a separate program. More specifically it could be called a macro calling function or a macro program, which is a program that is called.

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 and MDI operation mode.

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



6

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.

There is a sub program calling (M98, M198) for a function similar to macro program calling (G65, G66), and the differences between these two are described later in detail.  
These call functions are described below.

### 6.5.1 Simple Call Function

There are two methods to call a program: using a program number and using a program name.

Command format

|                                       |                         |
|---------------------------------------|-------------------------|
| <b>G65 P_ L_ (argument);</b>          | ... Program number call |
| <b>G65 &lt;***&gt; L_ (argument);</b> | ... Program name call   |

- P : Call macro program number, or
- <\*\*\*> : Call macro program name (\*\* refers to the program name)
- L : Number of repeats of call (up to 9999)  
A single call is determined if it is omitted  
A command cannot be issued when carrying out tape operation (general communications device).
- (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.

There are two methods to call a program: using a program number and using a program name.

Command format

|                                        |                         |
|----------------------------------------|-------------------------|
| <b>G66 P_ L_ (argument);</b>           | ... Program number call |
| <b>G65 &lt;****&gt; L_ (argument);</b> | ... Program name call   |

- P : Call macro program number, or
- <\*\*\*\*> : Call macro program name (\*\*\*\* refers to the program name)
- L : Number of repeats of call (up to 9999)
  - A single call is determined if it is omitted (see description of G65)
  - A command cannot be issued when carrying out tape operation (general communications device).
- (Argument) : Data to be sent to the macro; may be omitted.

Cancel is commanded as follows:

Command format

|             |
|-------------|
| <b>G67;</b> |
|-------------|

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).

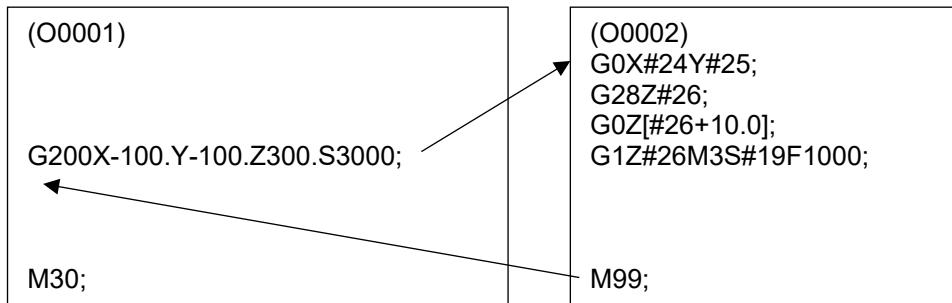
6

- (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<br>after M30 command. |
| M98P5;             |                                                                   |
| M98P100;           |                                                                   |
- (NOTE 5) G/M code macro command cannot be used on the same block. If used, the subsequent command is processed as an argument, or the alarm <<Macro program call address error>> is triggered.
- (NOTE 6) When a command is issued for P\_ and <\*\*\*\*> simultaneously, the data is overwritten after the command.

### 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, 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).  
A command cannot be issued when carrying out tape operation (general communications device).
- (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.

6

<G/M code macro (Common parameter)>

- <Selected folder 1 for G code macro program>
- <Specified folder 1 for G code macro program>
- <Selected folder 2 for G code macro program>
- <Specified folder 2 for G code macro program>
- <Selected folder 3 for G code macro program>
- <Specified folder 3 for G code macro program>
- <Macro program call control type>

<G/M code macro (G code macro)>

- <G code number>
- <Program>
- <Call method>

There are 3 folders that can be specified for calling a macro program. The description below shows how to specify a folder. The default setting is configured to the root folder (root directory).

|                                                                                                                      |                                                                    |                                                                   |
|----------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------|-------------------------------------------------------------------|
| Macro program call folder                                                                                            | Selection: <Selected folder * for G code macro program> (*:1 to 3) | Setting: <Specified folder * for G code macro program> (*:1 to 3) |
| Same folder as the main program<br>(NOTE) This folder is the root folder when using MDI operation or tape operation. | <1: Current folder>                                                | Grayed out and disabled.                                          |
| Specified folder                                                                                                     | <0: Specified folder>                                              | Folder name                                                       |
| Root directory or root folder                                                                                        | <0: Specified folder>                                              | Blank field                                                       |

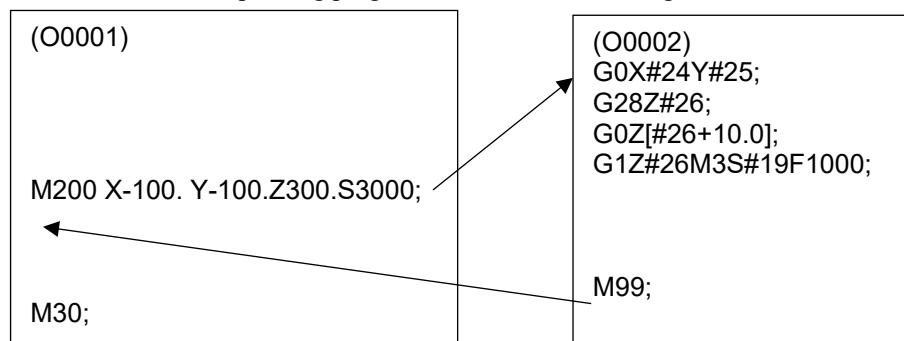
When a program is called, a search is conducted starting from <Selected folder 1 for G code macro program> - <Specified folder 1 for G code macro program> and continues up to the <Selected folder 3 for G code macro program> - <Specified folder 3 for G code macro program>. The first program that is found in the search is called.

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 specified in “8.3 Simple call function”. 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 operatin 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.If used, the subsequent command is processed as an argument, or the alarm <<Macro program call address error>> 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 specified in “8.3 Simple call function”.

#### 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, used for macro calling, ahead of time in the parameters.



Command format

**Mxxx L\_ (argument);**

- |            |                                                                                                                                                                                                                      |
|------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| xxx        | : M code (discussed later) registered in the parameter<br>However, 0, 1, 2, 30 and 98 to 99 cannot be used.                                                                                                          |
| L          | : Number of repeated calls (9999 times or less)<br>If omitted, it is processed as 1 time (Refer to G65 description).<br>A command cannot be issued when carrying out tape operation (general communications device). |
| (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)>

- <Selected folder 1 for M code macro program>
- <Specified folder 1 for M code macro program>
- <Selected folder 2 for M code macro program>
- <Specified folder 2 for M code macro program>
- <Selected folder 3 for M code macro program>
- <Specified folder 3 for M code macro program>
- <Macro program call control type>

<G/M code macro (M code macro)>

- <M code number>
- <Program>

There are 3 folders that can be specified for calling a macro program. The description below shows how to specify a folder. The default setting is configured to the root folder (root directory). When the current folder is selected, the folder is the same as the main program.

| Macro program call folder                                                                                                | Selection: <Selected folder * for M code macro program> (*:1 to 3) | Setting: <Specified folder * for M code macro program> (*:1 to 3) |
|--------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------|-------------------------------------------------------------------|
| Same folder as the main program<br><br>(NOTE) This folder is the root folder when using MDI operation or tape operation. | <1: Current folder>                                                | Grayed out and disabled.                                          |
| Specified folder                                                                                                         | <0: Specified folder>                                              | Folder name                                                       |
| Root directory or root folder                                                                                            | <0: Specified folder>                                              | Blank field                                                       |

When a program is called, a search is conducted starting from <Selected folder 1 for M code macro program> - <Specified folder 1 for M code macro program> and continues up to the <Selected folder 3 for M code macro program> - <Specified folder 3 for M code macro program>. The first program that is found in the search is called.

When an M98, G65 or G66 command is issued inside the macro program that is called using a M code, the program is called from a folder that is specified in “8.3 Simple call function”. 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. If used, the subsequent command is processed as an argument, or the alarm <<Macro program call address error>> is triggered.
- (NOTE 6) When there is a G66 command inside the macro program that is called using a 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 specified in “8.3 Simple call function”.

## 6.5.5 Macro Call Arguments

Arguments must be specified if it is necessary to send local variables to macro programs.  
There are 2 methods for specifying arguments.

### 6.5.5.1 Argument for simple/modal call function

Method 1:

Arguments are specified with all addresses except G and O.

| 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                                | -/#12(NOTE 1)      |
| M                                | #13                |
| N                                | #14(NOTE 2)        |
| O                                | -                  |
| P                                | #16                |
| Q                                | #17                |
| R                                | #18                |
| S                                | #19                |
| T                                | #20                |
| U                                | #21                |
| V                                | #22                |
| W                                | #23                |
| X                                | #24                |
| Y                                | #25                |
| Z                                | #26                |

6

- (NOTE 1) When the user parameter <Macro call method (Address L)> is set to <0: Method 1>, the number of repeats is used to call the macro for the L address. When set to <1: Method 2>, an argument is used to call the macro.
- (NOTE 2) A sequence number is used to specify the N address at the header of the block, and an argument is used to specify the address in all other places.
- (NOTE 3) For the address P, when a Pnnnn (nnnn refers to the program number) command is issued, the program number is assigned as #16. When <\*\*\*\*> (\*\*\*\* refers to the character string for a given file name) command is issued, #16 is defined as unregistered. However, if a command is issued when the character string for a given file name is a four-digit number that starts from “O” (such as <O0123>), then the numerical value without the “O” is assigned as #16.

## 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                |

### 6.5.5.2 Argument for G code macro call

The same content as in “6.5.5.1 Argument for simple/modal call function” applies.

### 6.5.5.3 Argument for M code macro call

Method 1:

An argument can be specified in all addresses except O

| Address for specifying arguments | Variables in macro |
|----------------------------------|--------------------|
| A                                | #1                 |
| B                                | #2                 |
| C                                | #3                 |
| D                                | #7                 |
| E                                | #8                 |
| F                                | #9                 |
| G                                | #28 to #32(NOTE 1) |
| H                                | #11                |
| I                                | #4                 |
| J                                | #5                 |
| K                                | #6                 |
| L                                | -/#12(NOTE 2)      |
| M                                | #13, #27(NOTE 3)   |
| N                                | #14(NOTE 4)        |
| O                                | -                  |
| P                                | #16                |
| Q                                | #17                |
| R                                | #18                |
| S                                | #19                |
| T                                | #20                |
| U                                | #21                |
| V                                | #22                |
| W                                | #23                |
| X                                | #24                |
| Y                                | #25                |
| Z                                | #26                |

- (NOTE 1) Variables are saved in the arguments in the specified order. A maximum of five arguments can be used. If that maximum is exceeded, an alarm is triggered. However, when specifying multiple G codes in the same G code group, any G code value that is specified subsequently is overwritten and the number of arguments is processed as one.  
Ex: M06 is registered as an M code macro call  
M06 G00 G04 G01  
(1) #28 ← 0.0 (G00)  
(2) #29 ← 4.0 (G04)  
(3) #28 ← 1.0 (G01: G00 is already used in the same group and is overwritten as G01)
- (NOTE 2) When the user parameter <Macro call method (Address L)> is set to <0: Method 1>, the number of repeats is used to call the macro for the L address. When set to <1: Method 2>, an argument is used to call the macro.
- (NOTE 3) The address M in the M code used for calling is stored for #27. The last M address (apart from the M code used for calling) is stored for #13.  
Ex: M07 is registered as an M code macro call  
M06 M07  
(1) #27 ← 7.0 (M07)  
(2) #13 ← 6.0 (M06 is used as an M code macro argument)  
In addition, when the user parameter <Multiple M codes in one block> is set to <0: No>, there is one M address that can be specified for an M code that is not used for calling. When the user parameter <Multiple M codes in one block> is set to <1: Yes>, there are three M addresses that can be specified for an M code that is not used for calling. If the number of specified M addresses is exceeded, the alarm <<M code macro call address error>> is triggered.

- (NOTE 4) A sequence number is used to specify the N address at the header of the block, and an argument is used to specify the address in all other places.
- (NOTE 5) When an integer is noted for the argument value, the value that is actually registered and stored as an argument is affected by the setting of the user parameter <Program unit>.

Method 2:

The same content as method 2 in “6.5.5.1 Argument for simple/modal call function” applies.

#### 6.5.5.4 Special notes on macro call arguments

1. An address that is not required does not need to be specified.
2. The value is left blank for any local variable that corresponds to an unspecified address.
3. The method changes back and forth between method 1 and 2 depending on the address being used. When both methods are being used, the value specified afterward becomes valid.

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 (Numerical value specified in D is invalid)
- (7) #8 ← 7.0 (Numerical value specified in E is invalid)
- (8) #9 ← 8.0
- (9) #9 ← 9.0 (Numerical value specified the second time in K is invalid)

6

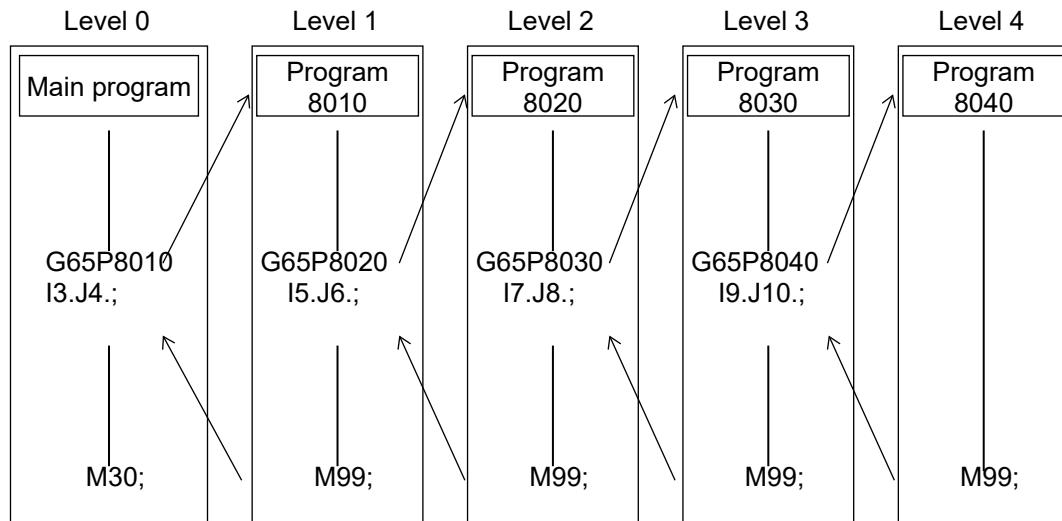
#### 6.5.6 Difference Between G65 and M98/M198

1. G65 can specify arguments but M98/M198 cannot.
2. G65 can have local variables for respective levels of its multiplicity but M98/M198 cannot.
3. Multiplicity of call of G65 is up to 8-fold including call by M98/M198, 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

| Local variables for level 0                                                    | Local variables for level 1                                                            | Local variables for level 2                                                            | Local variables for level 3                                                            | Local variables for level 4                                                             |
|--------------------------------------------------------------------------------|----------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------|
| #1<br>.<br>.<br>#4<br>#5<br>.<br>.<br>#9<br>#11<br>#13<br>#17<br>.<br>.<br>#33 | #1<br>.<br>.<br>#4 3.0<br>#5 4.0<br>.<br>.<br>#9<br>#11<br>#13<br>#17<br>.<br>.<br>#33 | #1<br>.<br>.<br>#4 5.0<br>#5 6.0<br>.<br>.<br>#9<br>#11<br>#13<br>#17<br>.<br>.<br>#33 | #1<br>.<br>.<br>#4 7.0<br>#5 8.0<br>.<br>.<br>#9<br>#11<br>#13<br>#17<br>.<br>.<br>#33 | #1<br>.<br>.<br>#4 9.0<br>#5 10.0<br>.<br>.<br>#9<br>#11<br>#13<br>#17<br>.<br>.<br>#33 |

Common variables: Read and written from all levels

|                        |
|------------------------|
| #100~#199<br>#500~#999 |
|------------------------|

## 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(S) 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 macro variable 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 including external output command

```
G90;
:
POOPEN;
BPRNT[#100[3]];
DPRNT[#2[63]];
PCLOS;
:
M30;
```

### 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

**POOPEN;**

6

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(S) server>

This connects to the FTP(S) 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.

6

### 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 |    |    |    |    |    |    |    |

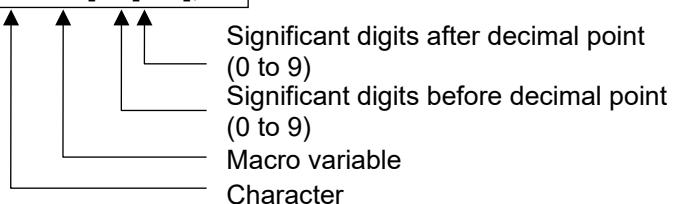
\* SP: space code

### 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>

|                                                                                                        |
|--------------------------------------------------------------------------------------------------------|
| <b>D8 A0 2D A0 B1 B2 33 2E B4 35 36 30 59 A0 A0 B2 33 2E B4 36 5A A0 A0 A0 30 A0 2D 2E B4 36 8D 0A</b> |
| 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>

|                                                                                            |
|--------------------------------------------------------------------------------------------|
| <b>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</b> |
| 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(S) server>

The connection to the FTP(S) 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.

6

Output folder : Root folder  
 Output file name : yyyyymmdd\_\*\*.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 “yyyyymmdd\_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(S) 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: | SSSSS <sub>yyyy</sub> mmdd_****.log<br>SSSSS: Character string specified in the communication parameter <External output – FTP output name><br>yyyy: Year<br>mm: Month<br>dd: Day<br>****: 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(S) server for the first time, execute and check the operation at least 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 “SSSSS<sub>yyyy</sub>mmdd\_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(S) 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.

- (NOTE) Do not change the external output file attributes that exist in the FTP server to reading only. Otherwise, operation stops.

### 6.6.7 Precautions for External Output Command

1. External output commands are used in memory operation (including extended memory operation) of NC language. The alarm <>Macro execution error>> is triggered when an external output command is made in MDI operation mode.
2. Data is output also in dry run and machine lock.
3. The mode cannot be changed when executing an external output command.
4. At program restart, external output commands specified before the restart position are also executed.
5. Variables that are empty are regarded as 0.
6. Up to 10 macro variables may be specified in the output data of one block.
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]];**

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(S) server> and a PCLOS command is processed, the connection to the FTP(S) 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(S) 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

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.
10. When connected to a <General COMM device>, a <Memory card> or an <FTP(S) 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.

There are two methods to call a program: using a program number and using a program name.

Command format

|                          |                         |
|--------------------------|-------------------------|
| <b>M96 P_;</b>           | ... Program number call |
| <b>M96 &lt;****&gt;;</b> | ... Program name call   |

P : Interrupt program number, or  
 <\*\*\*\*> : Interrupt program name (\*\* refers to the program name)

Use the command below to cancel an interruption.

Command format

|             |
|-------------|
| <b>M97;</b> |
|-------------|

The following conditions are required to use interrupt macros:

- Memory operation
- 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.

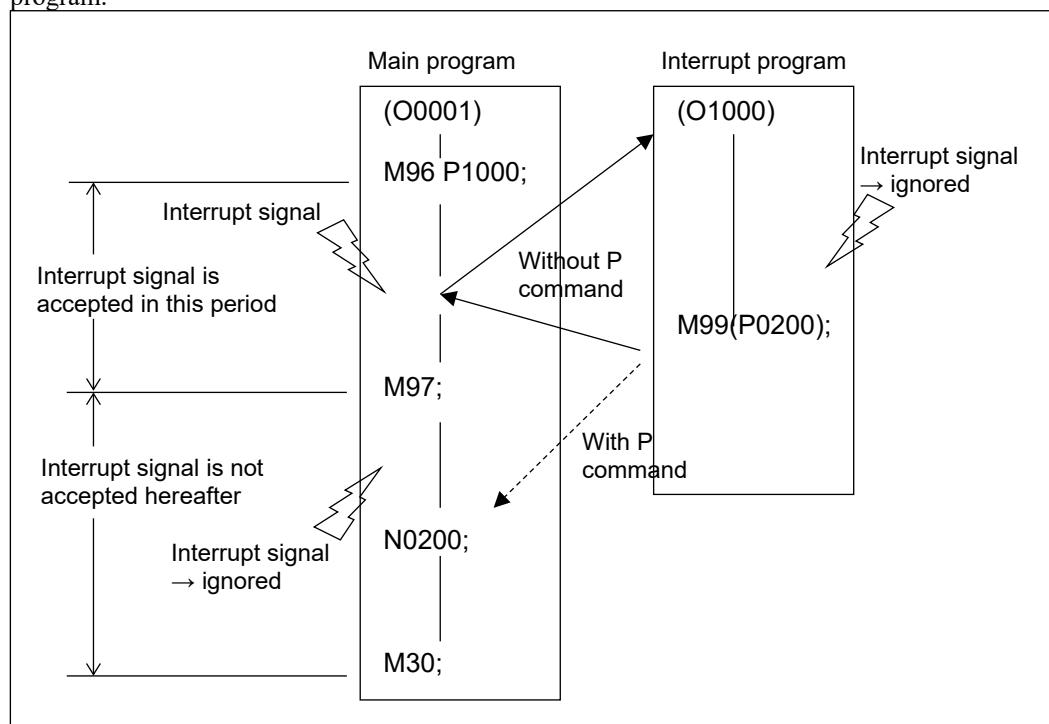
Insert M99 command to return from an interrupt macro to the main program. The sequence number in the main program to return to can be specified using P address.

6

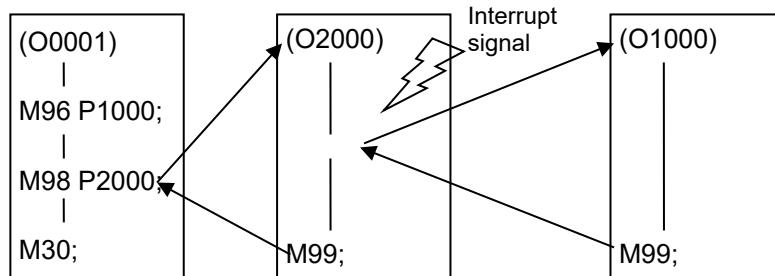
In addition to use of M97, the following operations cancel interruption:

- Operation reset
- Commanding M02 (M30)

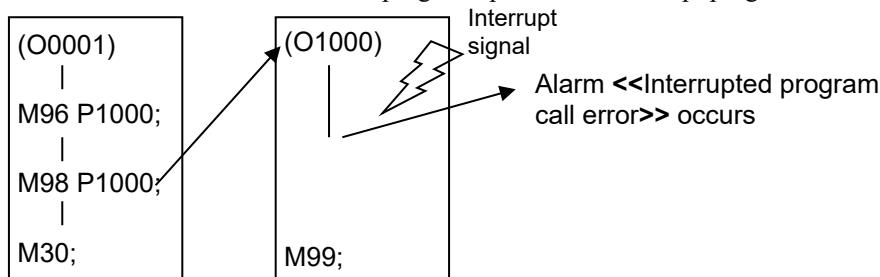
M97 is effectively commanded in an interrupt program or in a program called from an interrupt program.



- (NOTE 1) The alarm <<Simultaneous specified code cannot be used.>> is triggered when another G or M code command is issued on an M96 and M97 command block. In addition, the axis does not travel even if an X-, Y-, Z-, A-, B- or C-axis command is issued.
- (NOTE 2) If there is no command with an interrupt program number or interrupt program name on the M96 command block, the alarm <<Interrupt program number error>> is triggered. If a program number that does not exist is specified, the alarm <<No interrupt program>> is triggered.
- (NOTE 3) When an interrupt signal is detected during execution of a subprogram, the interrupt program is called.

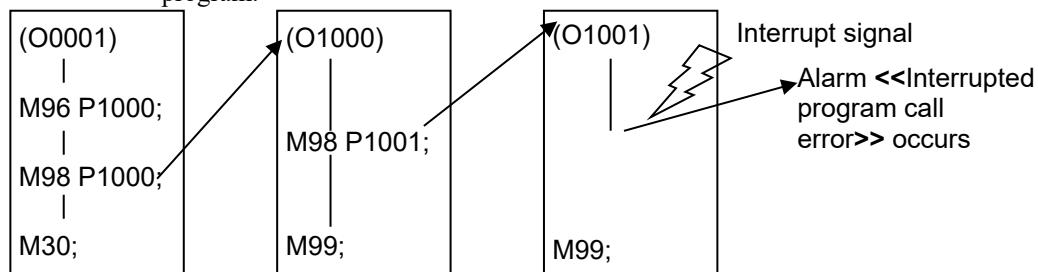


- (NOTE 4) The alarm <<Interrupted program call error>> occurs at the time of calling an interrupt program when an interrupt signal is detected during execution by G65/G66/M98 of subprogram specified as interrupt program.

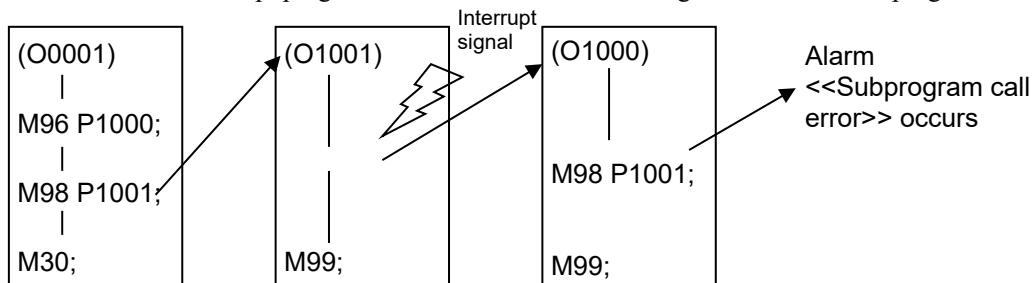


6

- (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.



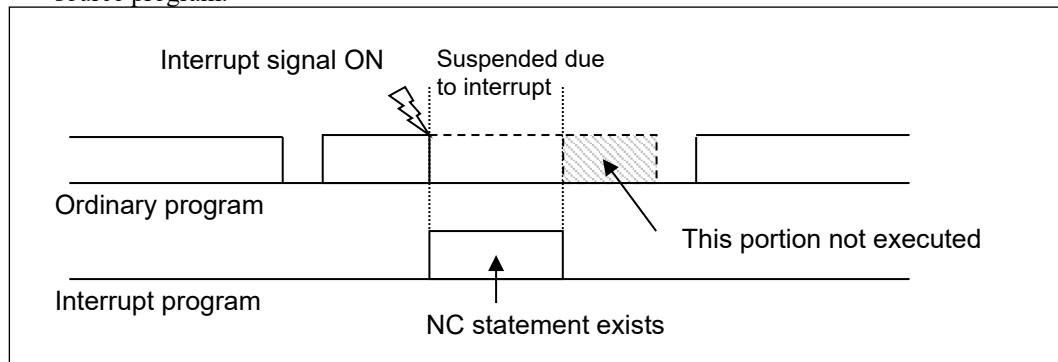
- (NOTE 7) When the total program size that is loaded (including size of interrupt programs) exceeds the set size at 32MB, it operates in the extended memory operation mode. In this mode, the operation may stop for a time period equivalent to the time required for loading if the size of the interrupt 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.
- (NOTE 9) When a command is issued for P\_ and <\*\*\*\*> simultaneously, the data is overwritten after the command.

### 6.7.1 Interrupt Type

When an interrupt signal is detected, the user parameter (switch 1: programming) <Interrupt macro interrupt type> can be used to set how to interrupt the currently executing block.

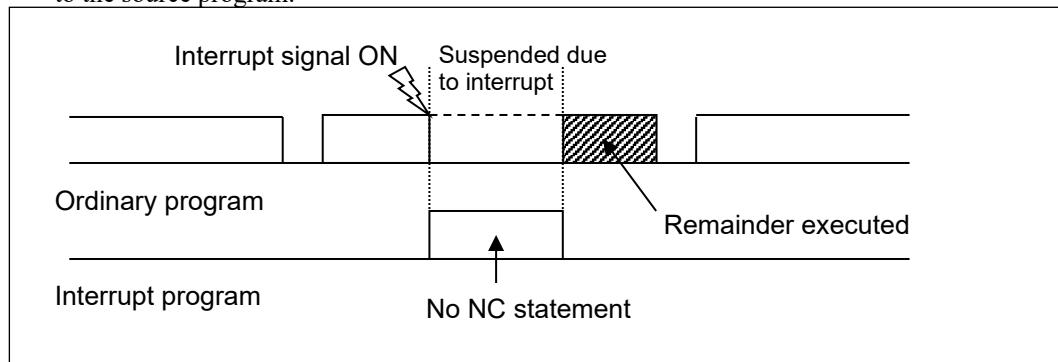
1. Type 1: Interrupt by suspending the execution  
The currently being executed block is suspended on detecting an interrupt signal, and the interrupt program starts immediately.

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.



6

In the absence of NC statements, the commands in the suspended block continue on returning to the source program.



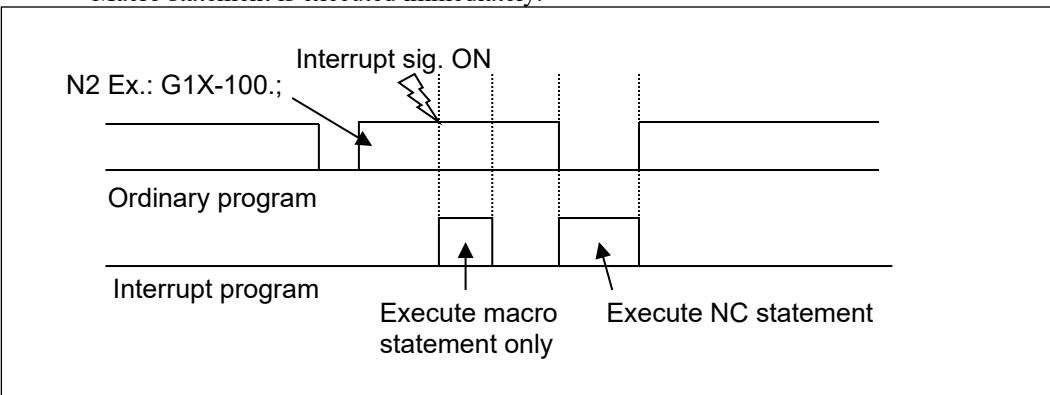
Refer to “6.7.6.Macro Statement and NC Statement” for the detailed description of NC statement.

2. Type 2: Interrupt without suspending the execution  
The interrupt program is executed on detecting an interrupt signal without suspending the currently executed block.

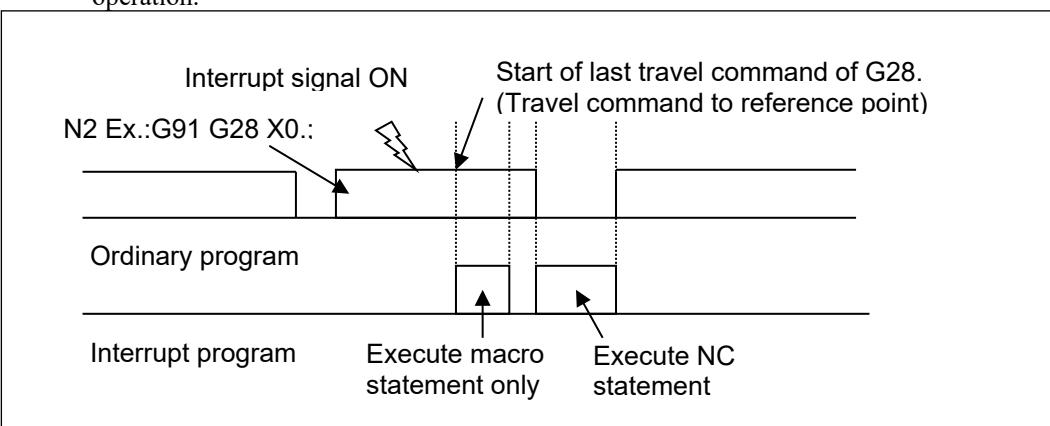
Macro statements up to the first NC statement in the interrupt program or subprogram / macro program called from the interrupt program are processed in parallel with the currently executed block.

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.



6

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)
- Brake load test

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)
- Brake load test

## 6.7.2 Call Type

There are two ways to call an interrupt program in an interrupt type macro. The user parameter (switch 1: programming) <Interrupt macro call type> is used to set the type or method for calling said interrupt program.

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 ways to receive and accept an interrupt signal. The user parameter (switch 1: programming) <Interrupt macro acceptance type> is used to set the type or method for receiving said signal.

1. Type 1: Status trigger system

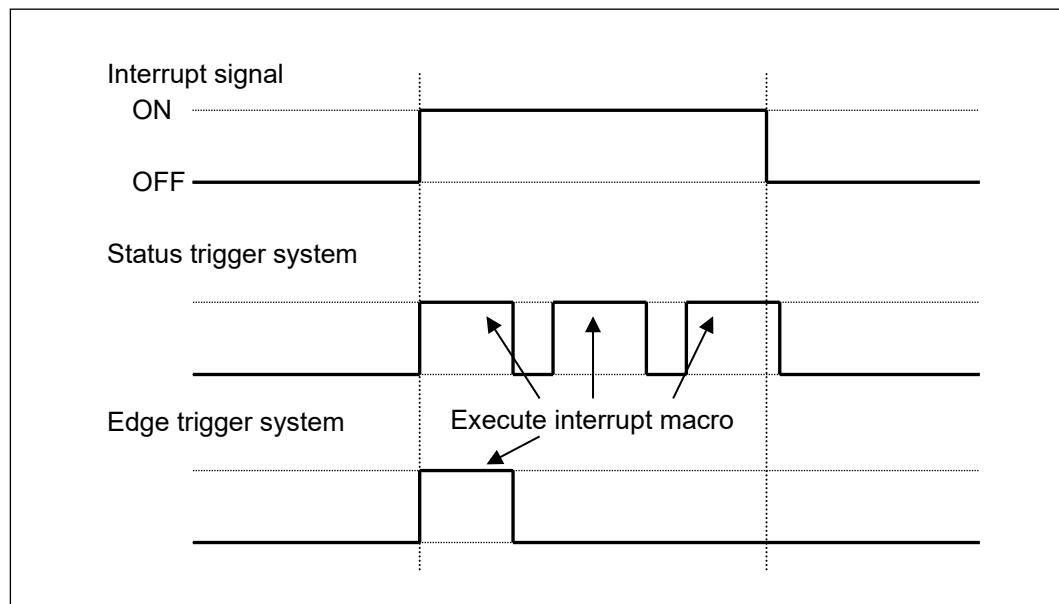
Signals are accepted when interrupt signal is ON.

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.



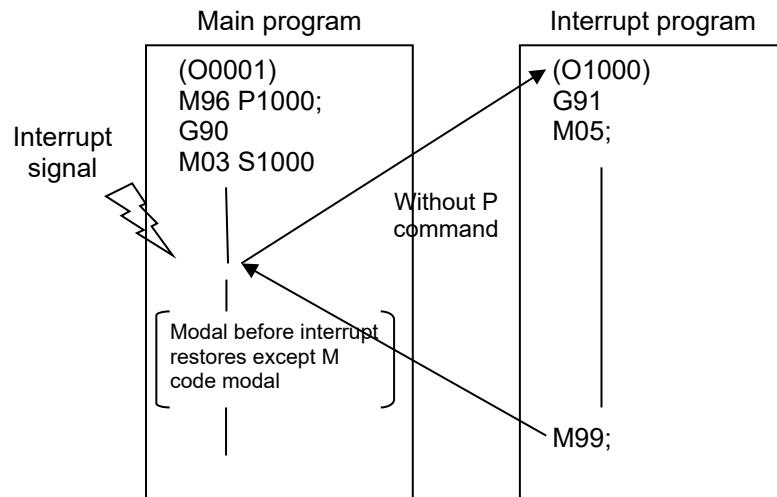
## 6.7.4 Interrupt Macro and Modal Information

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.

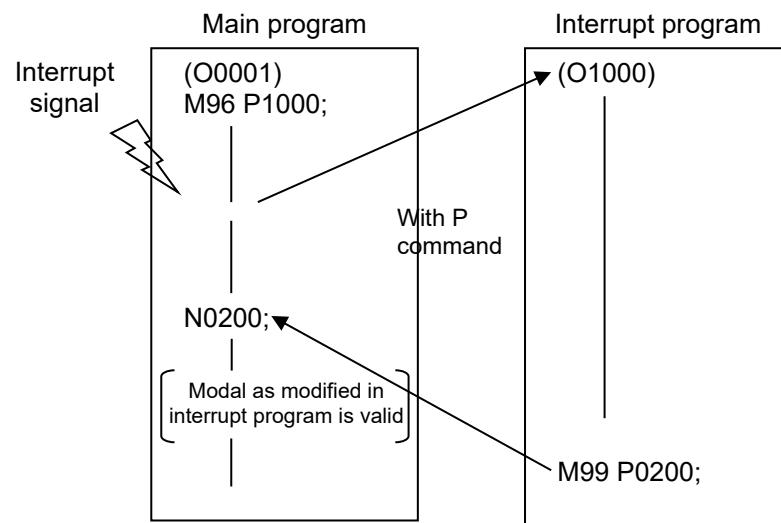


6

2. Returning with M99PXXXX (with a sequence number specified)

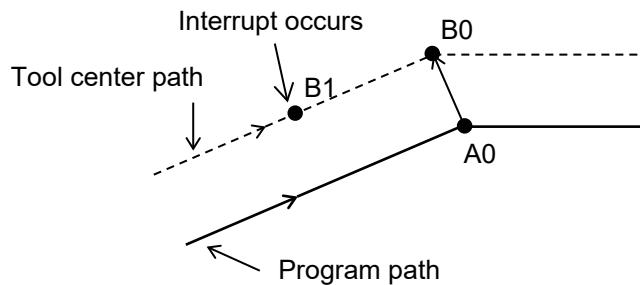
Modal information, modified in the interrupt program, is valid as it is modified.

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)
  - Blocks including sub program call commands (M98) and external sub program call commands (M198)
  - Includes a return command (M99) from macro program / subprogram
- \* Only when the user parameter (switch 1: programming) <Macro command single stop> is set 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 Function” in the Operation Manual I for further details on the 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. The mode cannot be changed during the time from when an interrupt signal is detected until the interrupt program is ended by executing M99. Otherwise, the alarm <>Interrupt type macro execution is being prepared.>> is displayed.
16. The mode cannot be changed during the time from when the interrupt program execution is completed until the processing is returned to the original program. Otherwise, the alarm <>Interrupt type macro post-process being executed.>> is displayed.
17. When restoring from an interrupt program during tap operation (general communications device), the sequence number cannot be specified. If it is specified, the alarm <>Interrupted program return error>> is triggered.
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.
  - Macro program that is called using G code (or M code).
  - Sub program / macro program that is called using the command M98, M198, 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 executes 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.
22. If an interrupt signal (UNIT) is detected during the Z-axis perimeter operation, then the interrupt program is not executed until after the operation is completed, regardless of the user parameter (switch 1: programming) <Interrupt macro interrupt type> setting. The Z-axis perimeter operation begins at the start of the Z-axis up rapid feed and lasts until the X-/Y- axis rapid feed is complete. Refer to "Chapter 12 M function" for further details on the Z-axis perimeter operation.
23. When an M96 command is issued during TCP control, the alarm <>TCP under control<> is triggered. In addition, when the TCP control (G43.4/G43.5) command is issued during M96 modal, the alarm <>TCP control command not possible<> is triggered.

### 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.

(NOTE) 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.)

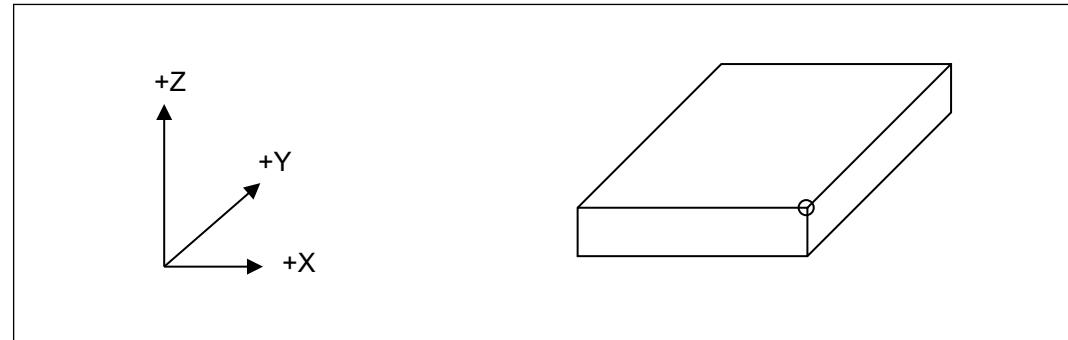
# CHAPTER 7

## AUTOMATIC WORKPIECE MEASUREMENT

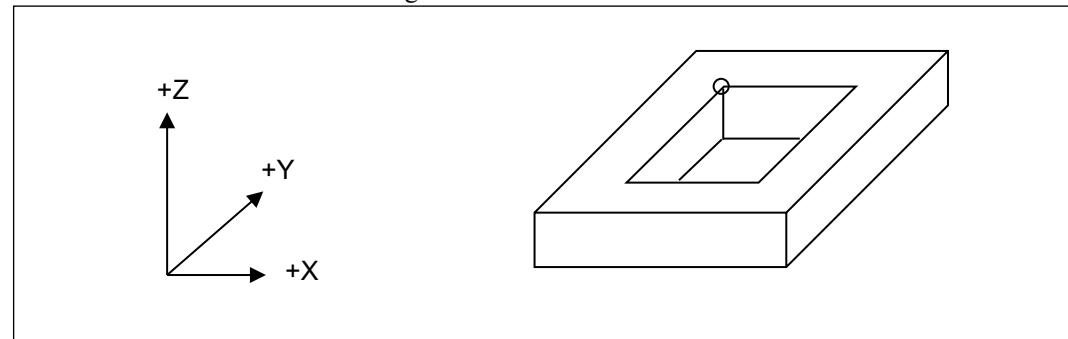
- 7.1 List of Automatic Workpiece Measurement Functions
- 7.2 Before Automatic Workpiece Measurement
- 7.3 Setting Data for Automatic Workpiece Measurement
- 7.4 Command Procedure for Automatic Workpiece Measurement
- 7.5 Measurement Results Processing
- 7.6 Lock Key Operations
- 7.7 Program Restart Operation

## 7.1 List of Automatic Workpiece Measurement Functions

1. G121...X and Y coordinates for corner

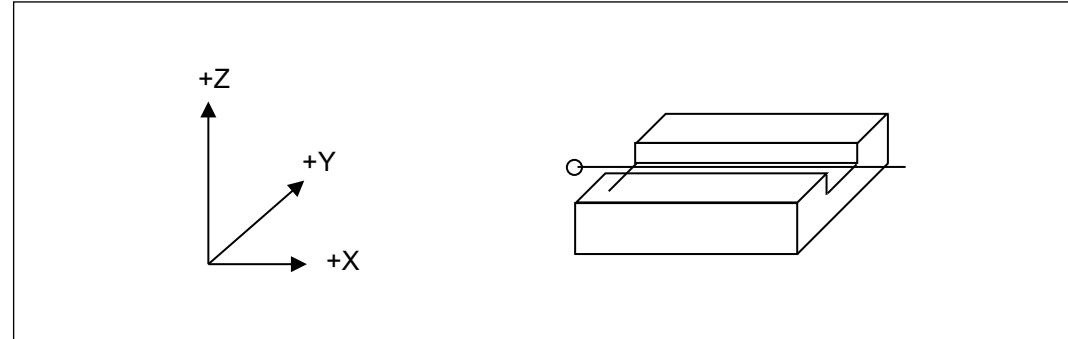


2. G129...X and Y coordinates for groove

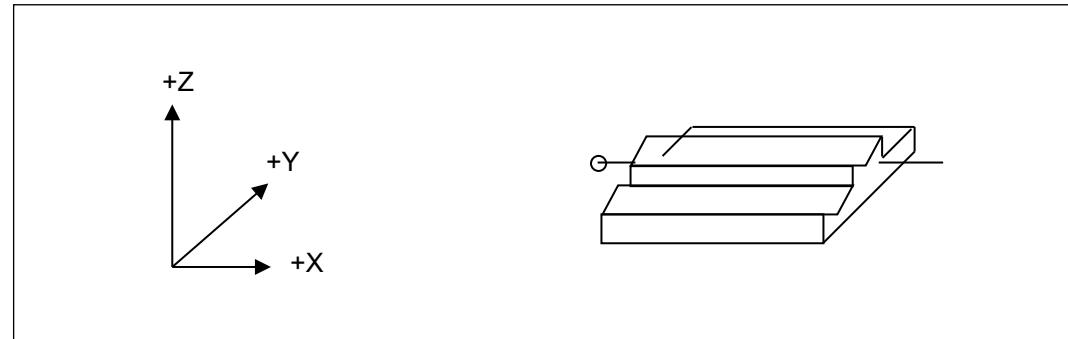


7

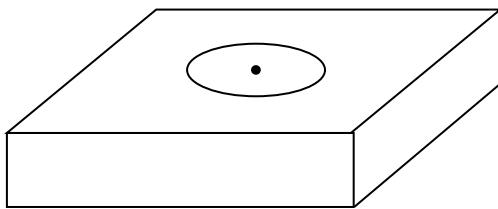
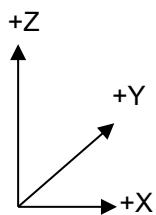
3. G122...X and Y coordinates for center of parallel groove



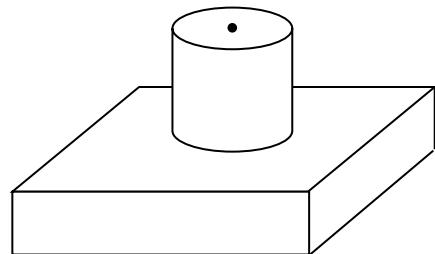
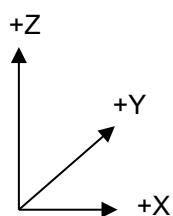
4. G123...X and Y coordinates for center of parallel boss



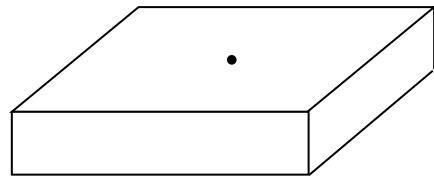
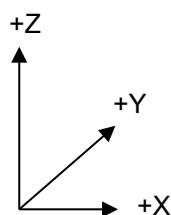
5. G124 and G126...X and Y coordinates for center of hole



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

The user parameter (switch 1: programming) <Measurement setting 1> is used in the automatic workpiece measurement function. Make sure that only the devices or instruments used for the measurement are enabled.

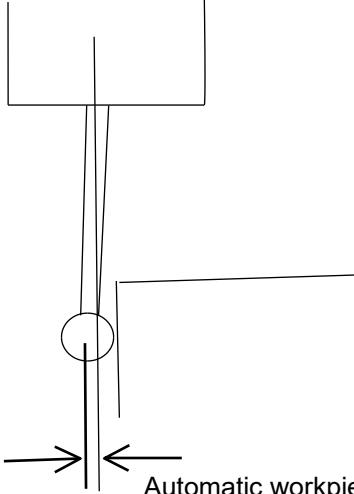
In addition, set the necessary user parameters (automatic workpiece measurement/automatic centering).

If the parameters are not correctly set, the probe can become damaged.

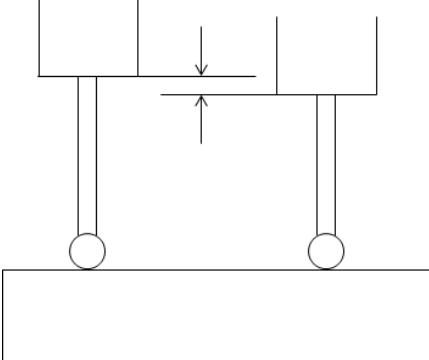
- (NOTE 1) The commands G121 to G129 cannot be issued when using the lathe spindle in the following situations. After switching modals, issue the command.
  - M142 modal in progress
  - G143/G144 modal in progress
- (NOTE 2) The commands G121 to G129 cannot be issued while in feature coordinate manufacturing mode (G68.2 modal in progress).
- (NOTE 3) The G121 to G129 commands are 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.
- (NOTE 4) A command between G121 and G129 cannot be issued while under TCP control (G43.4/G43.5). Otherwise, the alarm <<TCP under control>> is triggered.

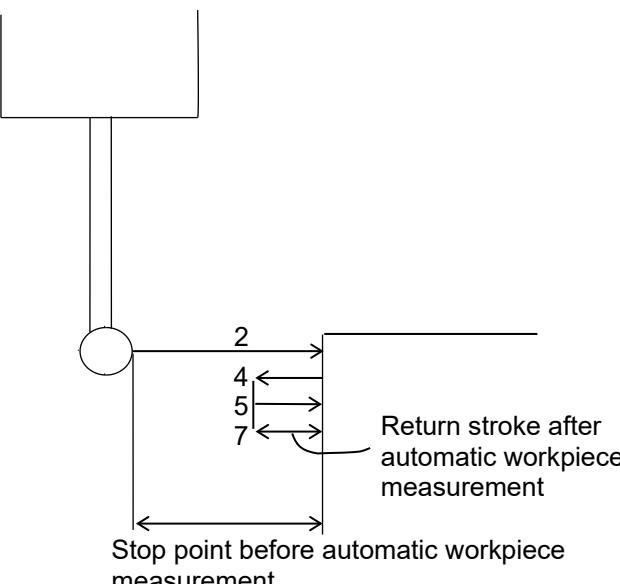
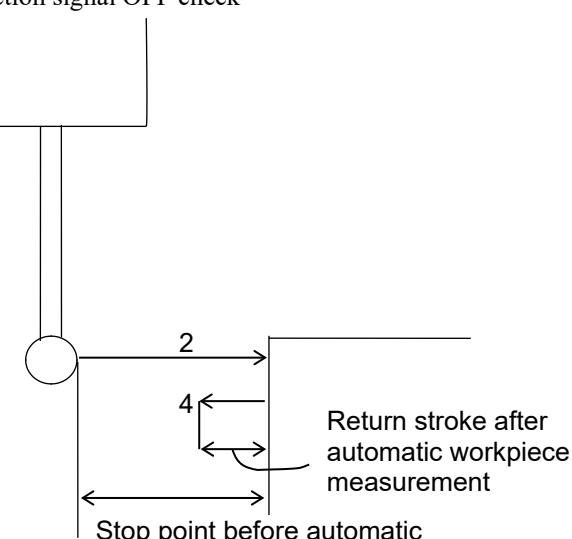
## 7.3 Setting Data for Automatic Workpiece Measurement

Description of the user parameters (automatic workpiece measurement/automatic centering)

| Item                                                                                                                                   | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                               |
|----------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <Automatic workpiece measurement compensation X (corner)><br><Automatic workpiece measurement compensation Y (corner)>                 | <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 the &lt;Automatic workpiece measurement compensation X (corner)&gt; and the Y value of the difference corresponds to the &lt;Automatic workpiece measurement compensation Y (corner)&gt;.</p> <p>We recommend performing the automatic setting on the automatic centering - &lt;Automatic workpiece measurement compensation (corner)&gt; screen in MDI mode.</p>  <p style="text-align: center;">Automatic workpiece measurement compensation</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> |
| <Automatic workpiece measurement compensation X (3-point circle)><br><Automatic workpiece measurement compensation Y (3-point circle)> | <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 the &lt;Automatic workpiece measurement compensation X (3-point circle)&gt; and the Y value of the difference corresponds to the &lt;Automatic workpiece measurement compensation Y (3-point circle)&gt;.</p> <p>We recommend performing the automatic setting on the automatic centering - &lt;Automatic workpiece measurement 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>                                                                                                                                                                            |

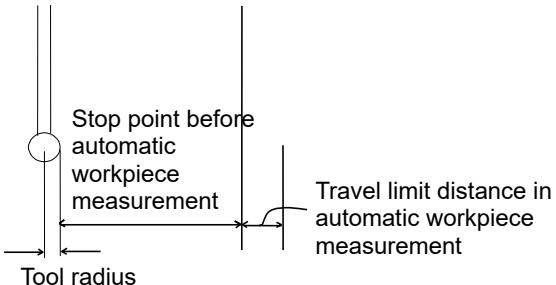
## Chapter 7 Automatic Workpiece Measurement

| Item                                                                     | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                        |
|--------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <Automatic workpiece measurement compensation Z (workpiece top surface)> | <p>Set the difference between the spindle end when the detection signal turns ON and the spindle end when the stylus tip ball touches the workpiece, with the touch probe attached to the spindle. We recommend performing the automatic setting on the automatic centering - &lt;Automatic workpiece measurement compensation (workpiece top surface)&gt; screen in MDI mode.</p> <p>When ball tip of stylus touches workpiece      When detection signal turns ON</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                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                  |
|---------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>&lt;Automatic workpiece measurement motion&gt;</b><br><b>&lt;0: Type 1&gt;</b><br><b>&lt;1: Type 2&gt;</b> | <p>&lt;0: Type 1&gt;</p> <ol style="list-style-type: none"> <li>1. Detection signal OFF check</li> <li>2. Travels at automatic workpiece 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 automatic workpiece 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>Return stroke after automatic workpiece measurement</p> <p>Stop point before automatic workpiece measurement</p> |
| <b>&lt;1: Type 2&gt;</b>                                                                                      | <p>&lt;1: Type 2&gt;</p> <ol style="list-style-type: none"> <li>1. Detection signal OFF check</li> <li>2. Travels at automatic workpiece 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. Detection signal OFF check</li> </ol>  <p>Return stroke after automatic workpiece measurement</p> <p>Stop point before automatic workpiece measurement</p>                                                                                                                                                                                                  |

## Chapter 7 Automatic Workpiece Measurement

| Item                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             |
|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <p>&lt;Automatic workpiece measurement speed 1&gt;</p> <p>Set the first measurement speed for the automatic workpiece measurement motion (Type 1). When this value is increased, the time it takes to measure can be reduced because the axis travel speed increases. However, if the value is increased too much, the probe can become damaged. Therefore, be careful when setting this value.</p> <p>The appropriate value for &lt;Automatic workpiece measurement speed 1&gt; can be calculated by using the following procedure.</p> <ol style="list-style-type: none"> <li>(1) Set the tolerance value L (mm) for the overtravel amount of the probe</li> <li>(2) Check the delay time <math>t_p</math> (msec) for the probe's touch signal detection</li> <li>(3) Calculate the value <math>t</math> (msec) that includes the machine parameter (system 1: X-, Y- and Z-axes) &lt;Skip feed time const. 1/3&gt;</li> <li>(4) Use the equation below to calculate <math>F_1</math>, and set the &lt;Automatic workpiece measurement speed 1&gt; so it is below that value</li> </ol> <div style="border: 1px solid black; padding: 10px; text-align: center;"> <math display="block">F_1 &lt; (120000L \div (8+t+2\times t_p)) \div 1.2</math> <p style="margin-top: -10px;">↑<br/>Safety factor</p> </div> <p>(Reference) Relational expression between each signal<br/> <math>L \geq ((F_1 \times t_d) \div (60 \times 1000))</math></p> <p style="text-align: center;">↑ Overtravel amount due to delay in control system<br/> * Control system delay = <math>t_d</math> (msec) = 4 (msec)</p> <p style="text-align: center;">+ <math>((F_1 \times t/2) \div (60 \times 1000))</math><br/> ↑ Overtravel amount due to acceleration/deceleration</p> <p style="text-align: center;">+ <math>((F_1 \times t_p) \div (60 \times 1000))</math><br/> ↑ Overtravel amount due to probe delay</p> <p>(NOTE 1) The probe overtravel amount or the detection delay time varies according to the device being used. After confirming which probe is being used, by checking the manual or by contacting the probe manufacturer, then decide on the value.</p> <p>(NOTE 2) This setting is also used for the measurement in the automatic centering - &lt;Automatic workpiece measurement compensation (corner)&gt;, &lt;Automatic workpiece measurement compensation (3-point circle)&gt; and &lt;Automatic workpiece measurement compensation (workpiece top surface)&gt; in MDI mode.</p> <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> | <p>&lt;Automatic workpiece measurement speed 2&gt;</p> <p>Set the second measurement speed for the automatic workpiece measurement motion (Type 1) and the measurement speed for the measurement motion (Type 2).</p> <p>When this value is increased, there will be greater variance in the measurement results. Therefore, set a value that corresponds to the targeted accuracy.</p> <p>The variance over repeated measurements is the probe's repeatability plus the variance (approx. 1.0 (um) for F2000) on the control side.</p> <p>(NOTE 1) The probe's repeatability varies according to the device being used. After confirming which probe is being used, by checking the manual or by contacting the probe manufacturer, then decide on the value.</p> <p>(NOTE 2) This setting is also used for the measurement in the automatic centering - &lt;Automatic workpiece measurement compensation (corner)&gt;, &lt;Automatic workpiece measurement compensation (3-point circle)&gt; and &lt;Automatic workpiece measurement compensation (workpiece top surface)&gt; in MDI mode.</p> <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> |

| Item                                                       | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                       |
|------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <Stop point before automatic workpiece 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;Stop point before automatic workpiece measurement&gt; + &lt;Travel limit distance in automatic workpiece measurement&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>                                  |
| <Travel limit distance in automatic workpiece measurement> | <p>Set the overtravel when the measurement skip operation exceeds the predicted value (programmed value).</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>                                                                                                                                                                                                                                                                                                                                                                                              |
| <Tolerance 1 for automatic workpiece measurement>          | <p>The alarm &lt;&lt;Large error in measured value (1)&gt;&gt; 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.</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>                                                                                                                                                                                                                                                      |
| <Tolerance 2 for automatic workpiece measurement>          | <p>The alarm &lt;&lt;Large error in measured value (2)&gt;&gt; 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.</p> <ul style="list-style-type: none"> <li>• When 0 is set</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> <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> |

## Chapter 7 Automatic Workpiece Measurement

| Item                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                        | Description |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------|
| <p>&lt;Return stroke after automatic workpiece measurement&gt;</p> <p>Set the return distance from the position where the measurement probe makes contact with the workpiece.</p> <p>However, if this value is decreased too much, the probe and workpiece may come into contact while the speed is not at a constant value, causing inappropriate measurement results.</p> <p>We recommend following the procedure below to set the calculated value.</p> <ol style="list-style-type: none"> <li>(1) Calculate the value t (msec) that includes the machine parameter (system 1: X-, Y- and Z-axes) &lt;Skip feed time const. 1/3&gt;</li> <li>(2) Confirm the value F2 (mm/min) for the &lt;Automatic workpiece measurement speed 2&gt;</li> <li>(3) Calculate the values for (a) and (b) below, and set the &lt;Return stroke after automatic workpiece measurement&gt; so it is greater than the bigger of the two values           <ul style="list-style-type: none"> <li>(a) 1.0</li> <li>(b) <math>F2 \times t / 60000</math></li> </ul> </li> </ol> <p>(Reference) Significance of (a) and (b)</p> <p>Set a minimum return stroke of 1.0 (mm) because there may be interference between the probe and workpiece (due to any variance in the workpiece configuration) when the Z-axis rises during the measurement motion.</p> <p>It is preferable to have the probe and workpiece come into contact during measurement while at a constant speed (not during acceleration). Therefore, set the return stroke so it is greater than the travel amount needed to reach the constant speed.</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> |             |

7

(NOTICE) 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

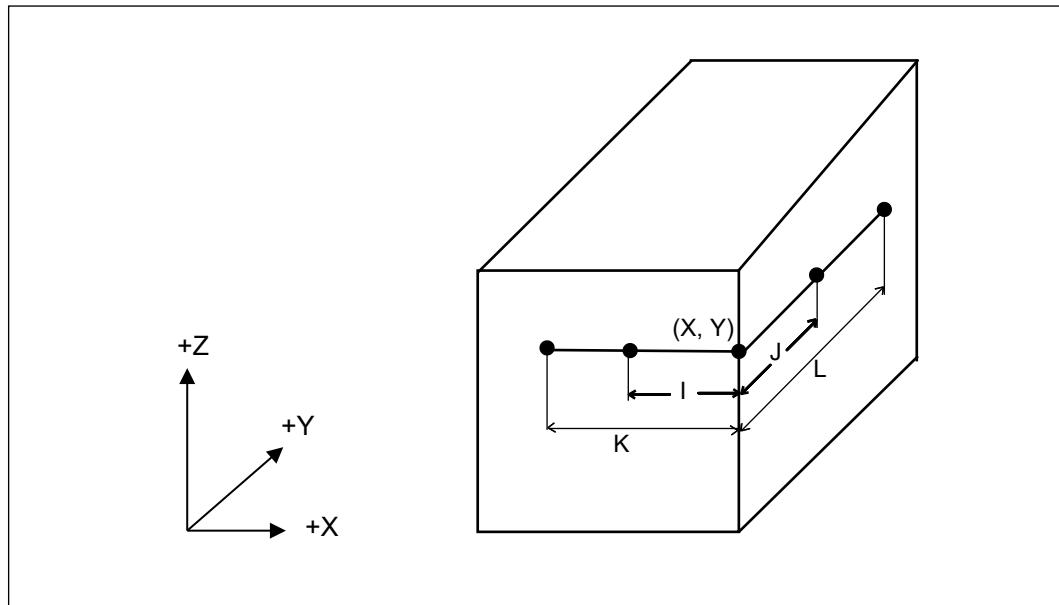
Command format

Boss

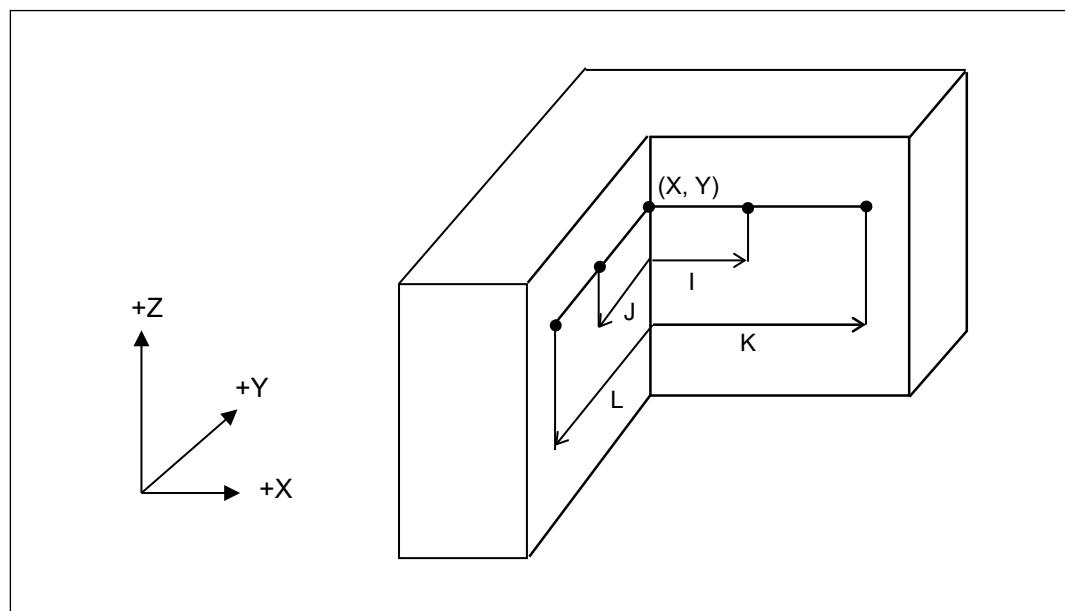
Groove

**G121 X\_Y\_I\_J\_K\_L\_D\_Z\_R\_Q;**

**G129 X\_Y\_I\_J\_K\_L\_D\_Z\_R\_Q;**



7



- 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 : Measurement number (1 to 4 and 1 is used when omitted)

## Chapter 7 Automatic Workpiece Measurement

- (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 carrying out (G121/G129), perform <Automatic workpiece measurement compensation (corner)> for the automatic centering, and set the compensation in the user parameters (automatic workpiece measurement / automatic centering) <Automatic workpiece measurement compensation X (corner)> and <Automatic workpiece measurement compensation Y (corner)>.
- (NOTE 4) The D modal does not change with the D command.

Boss

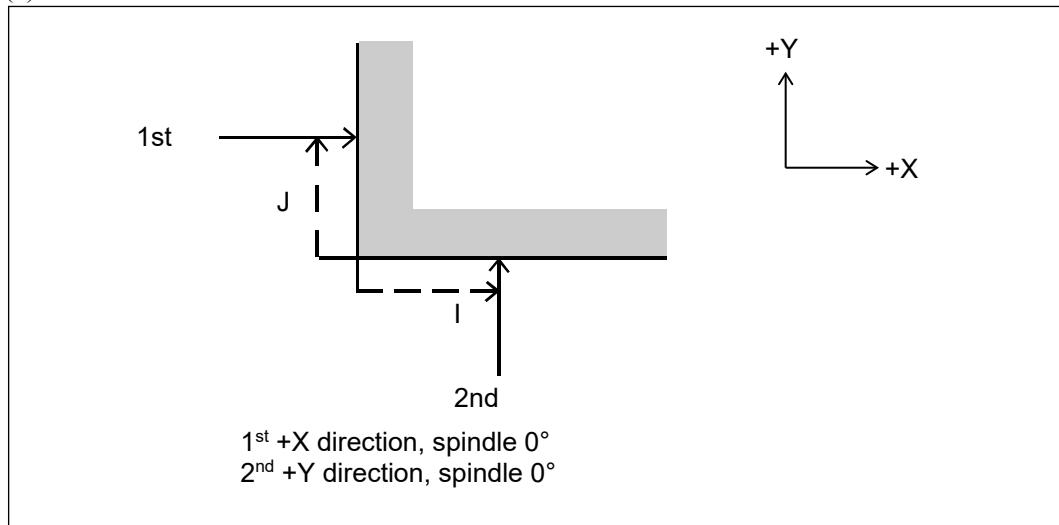
(Motion)

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.

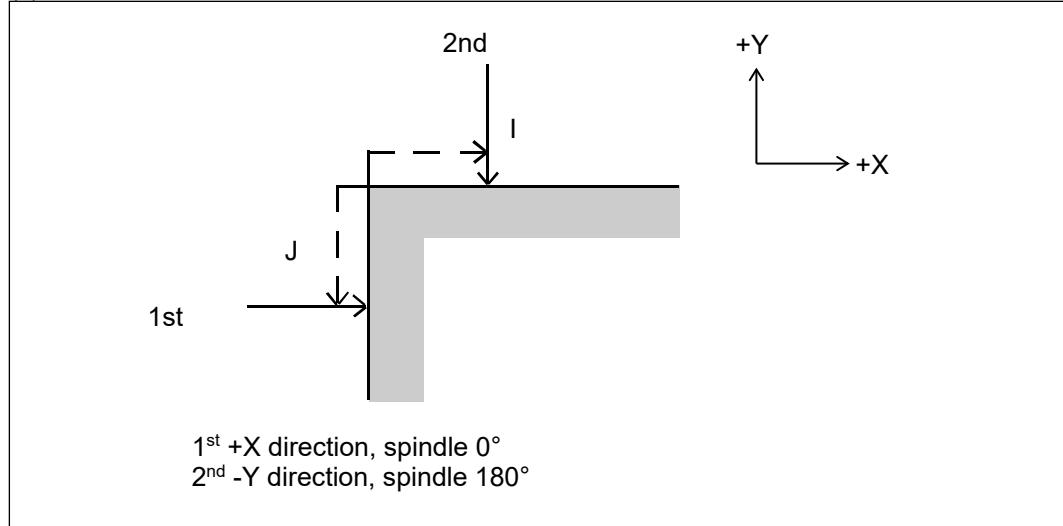
I and J signs, spindle orientation, and measurement skip travel direction

(1) I > 0 and J > 0

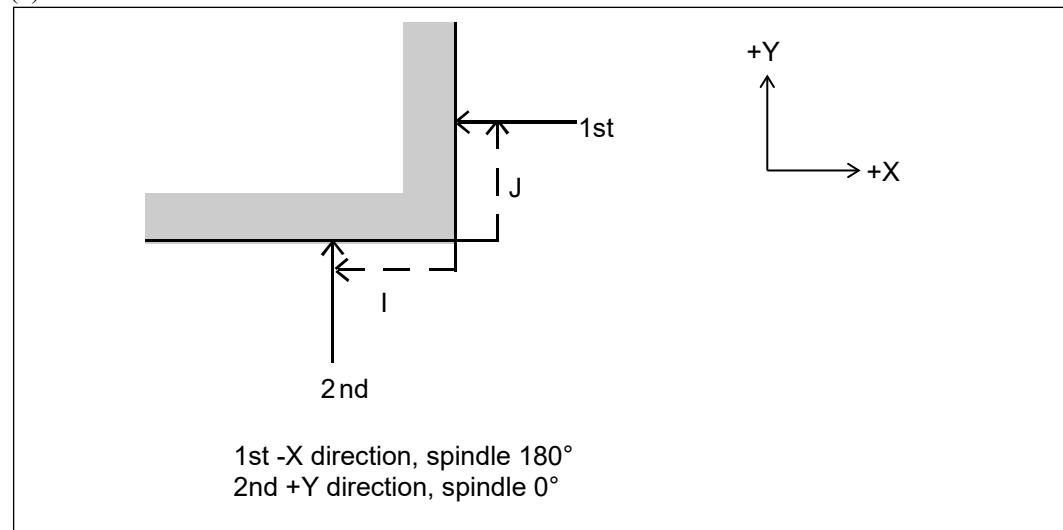
7



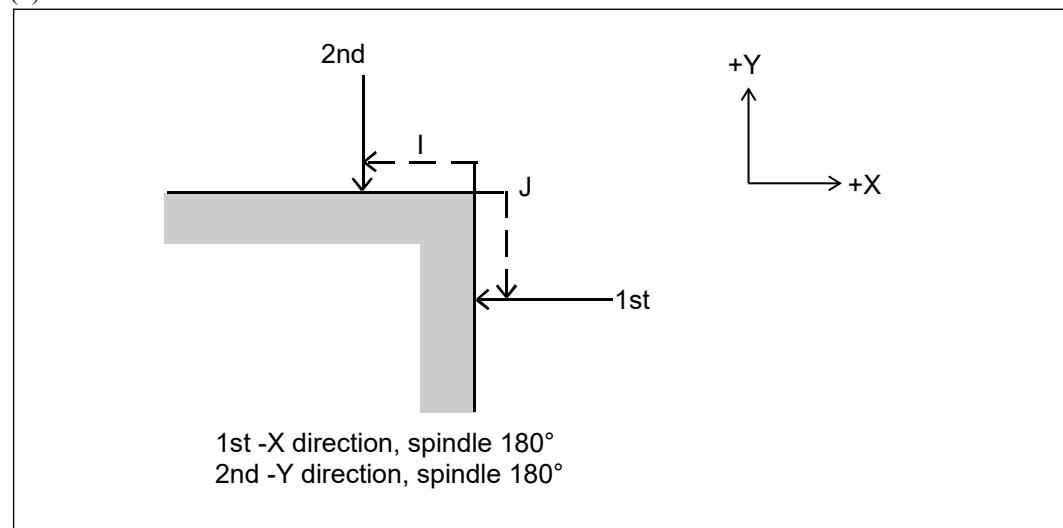
(2)  $I > 0$  and  $J < 0$



(3)  $I < 0$  and  $J > 0$



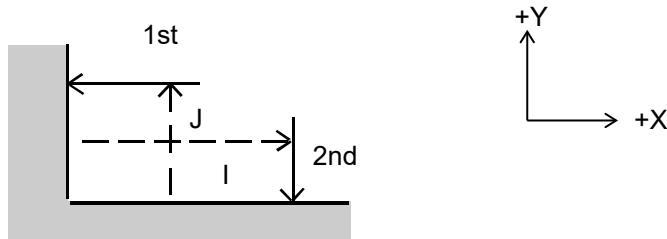
(4)  $I < 0$  and  $J < 0$



### 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.

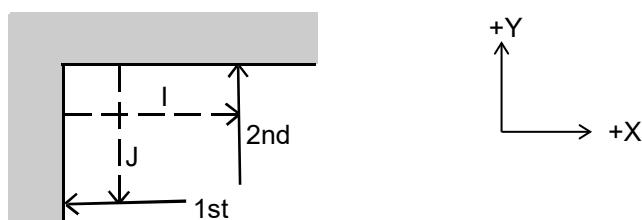
(1)  $I > 0$  and  $J > 0$



1st -X direction, spindle 180°  
2nd -Y direction, spindle 180°

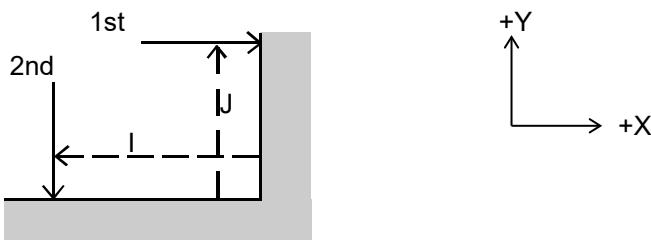
7

(2)  $I > 0$  and  $J < 0$

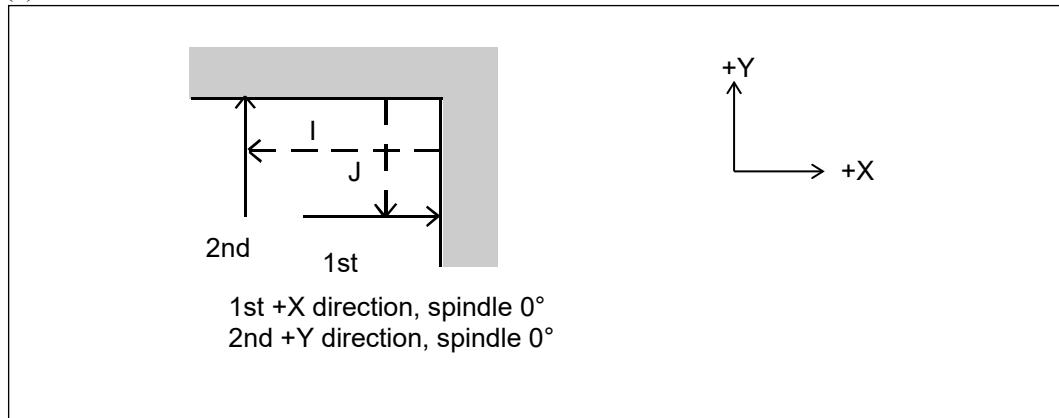


1st -X direction, spindle 180°  
2nd +Y direction, spindle 0°

(3)  $I < 0$  and  $J > 0$



1st +X direction, spindle 0°  
2nd -Y direction, spindle 180°

(4)  $I < 0$  and  $J < 0$ 

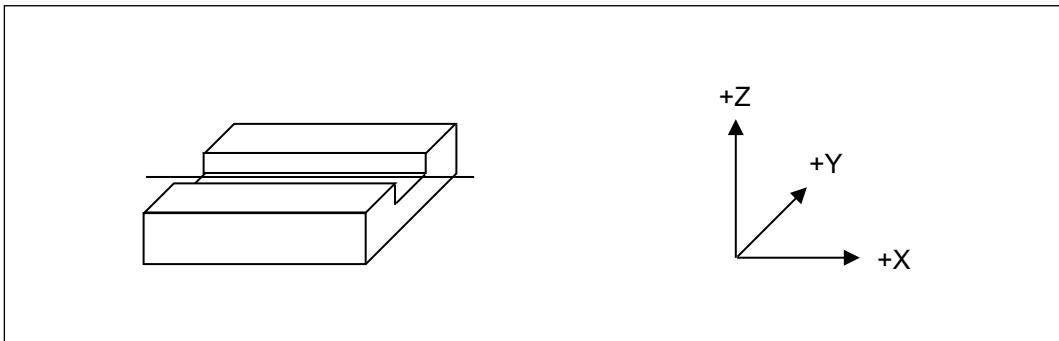
### 7.4.2 Parallel

Command format

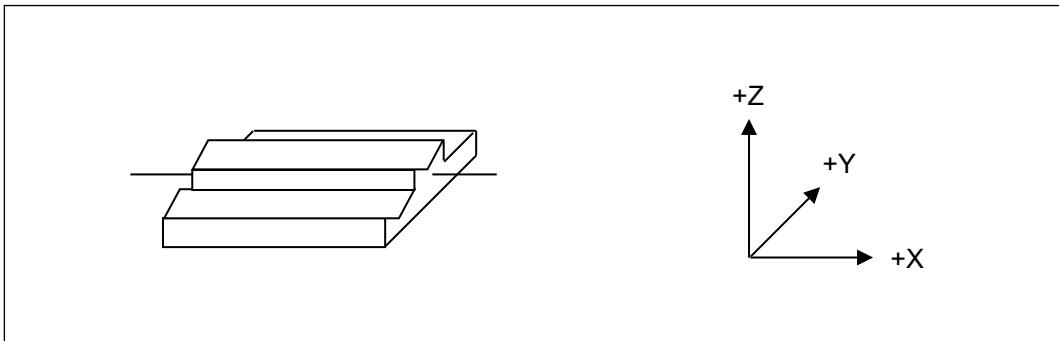
Groove

**G122 X\_ Y\_ I\_(J\_) D\_ Z\_ R\_ Q\_;**

Boss

**G123 X\_ Y\_ I\_(J\_) D\_ Z\_ R\_ Q\_;**

7

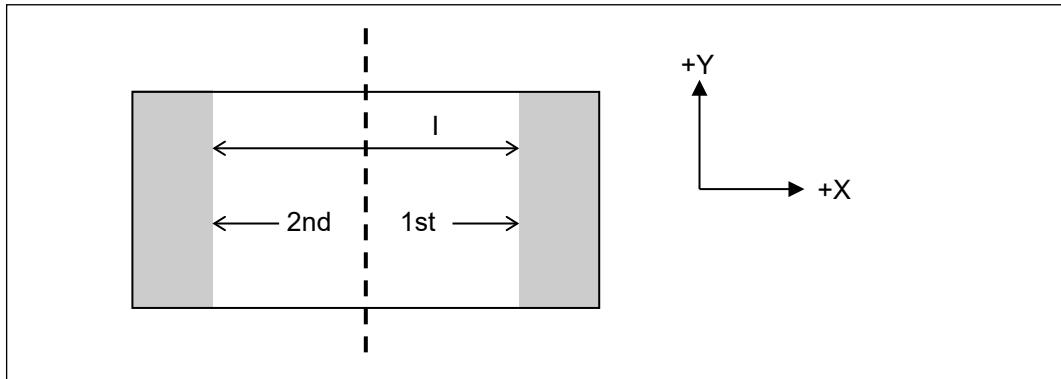


|           |                                                        |
|-----------|--------------------------------------------------------|
| X and Y : | Predicted value for groove (boss) center               |
| I and J : | Groove width                                           |
| I :       | Width 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 :       | Measurement number (1 to 4 and 1 is used when omitted) |

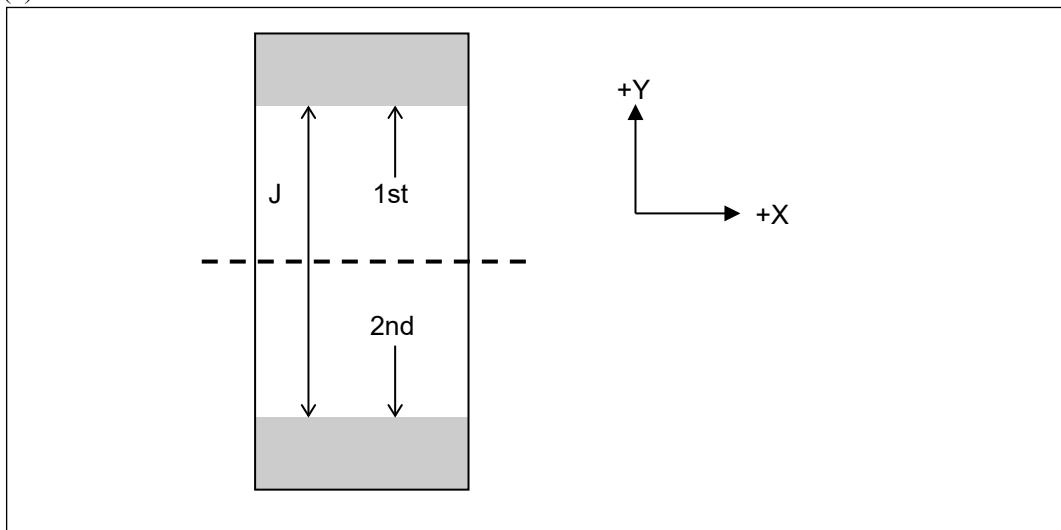
### Groove (Motion)

1. Spindle orientation 0°. 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°. 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



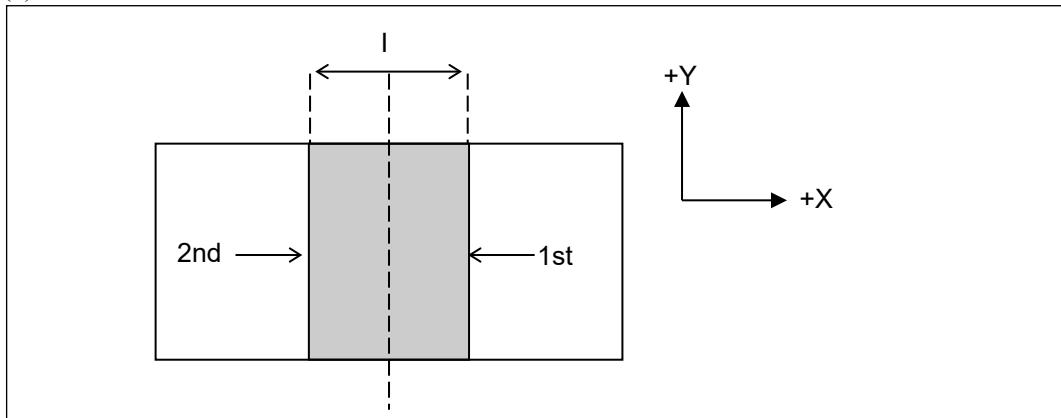
(2) When J command is issued



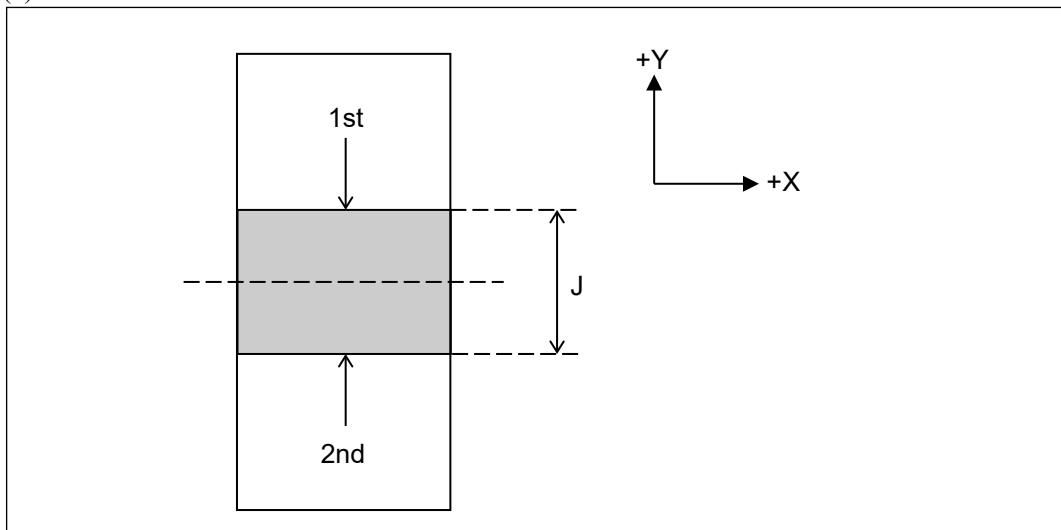
Boss  
(Motion)

1. Spindle orientation 180°. 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°. 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



### 7.4.3 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\_;**

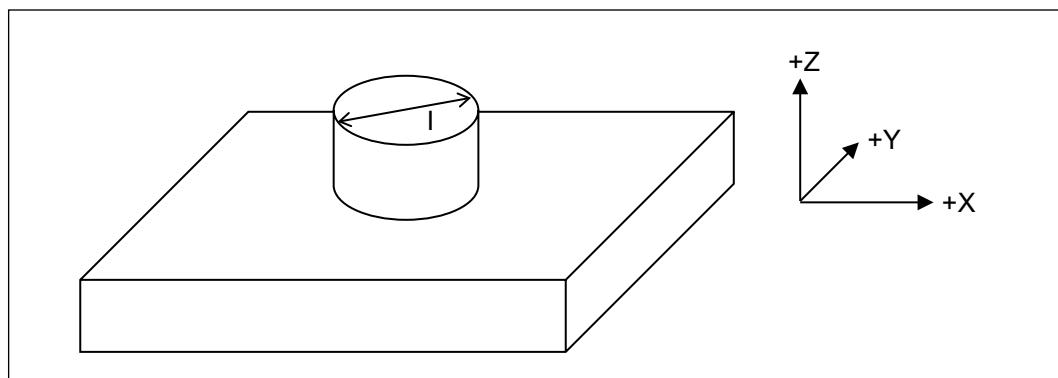
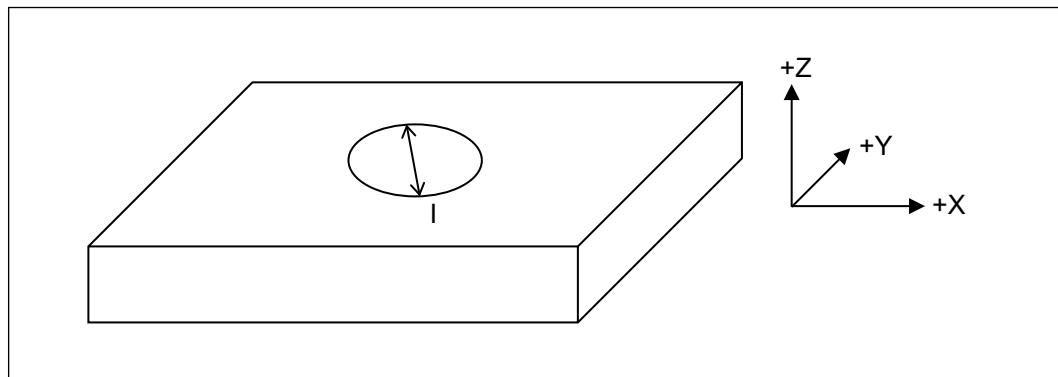
4 points are measured to calculate the center of the circle.

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 :       | Measurement number (1 to 4 and 1 is used when omitted) |

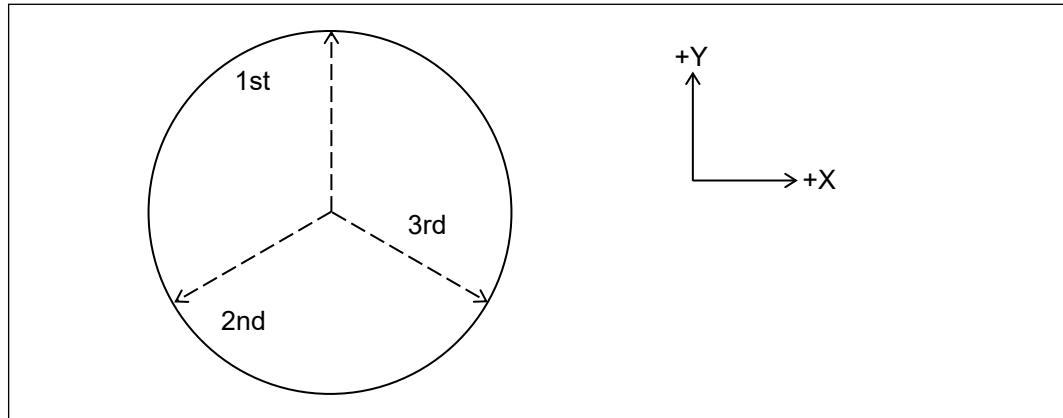
7



(NOTE) Before carrying out (G124/G125), perform <Automatic workpiece measurement compensation (3-point circle)> for the automatic centering, and set the compensation in the user parameters (automatic workpiece measurement / automatic centering) <Automatic workpiece measurement compensation X (3-point circle)> and <Automatic workpiece measurement compensation Y (3-point circle)>.

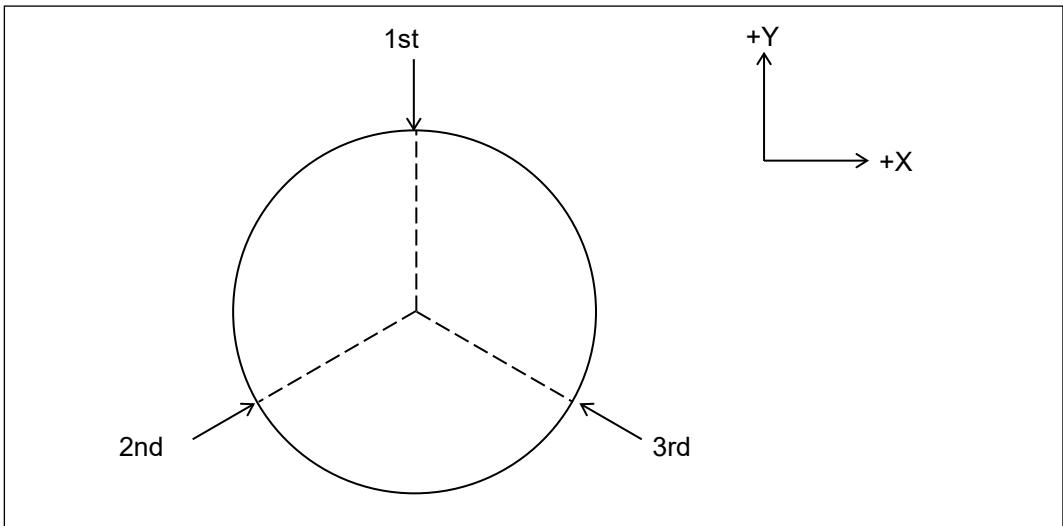
Hole and 3-point measurement  
(Motion)

1. Spindle orientation 0°. 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°. Travels to the second measurement start position for the X- and Y-axes.
5. Second measurement. (Direction that forms the 120° angle with the first)
6. Spindle orientation 0°. Travels to the third measurement start position for the X- and Y-axes.
7. Third measurement. (Direction that forms the 240° angle with the first)
8. Travels to the return height point on the Z-axis.



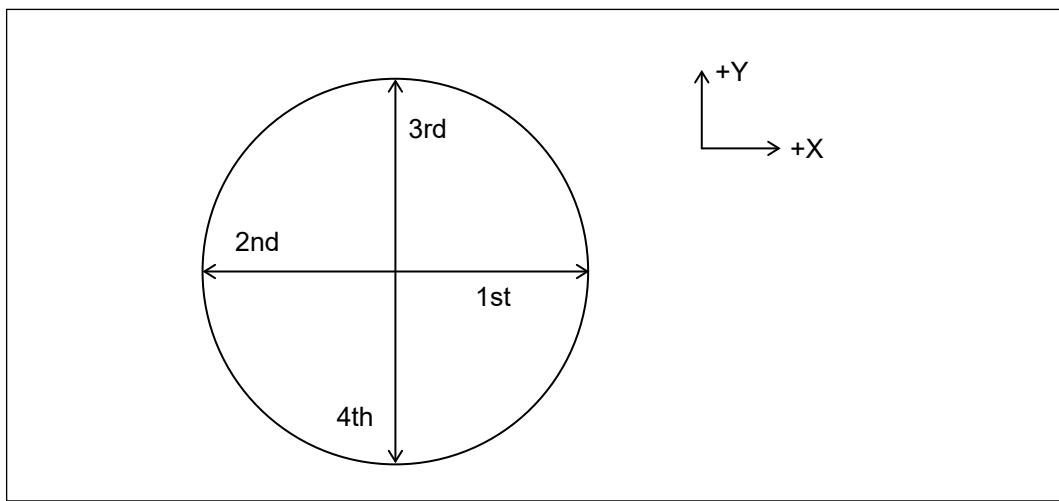
Boss and 3-point measurement  
(Motion)

1. Spindle orientation 180°. 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°. 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° angle with the first)
8. Returns to the return height point on the Z-axis.
9. Spindle orientation 180°. 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° angle with the first)
12. Returns to the return height point on the Z-axis.



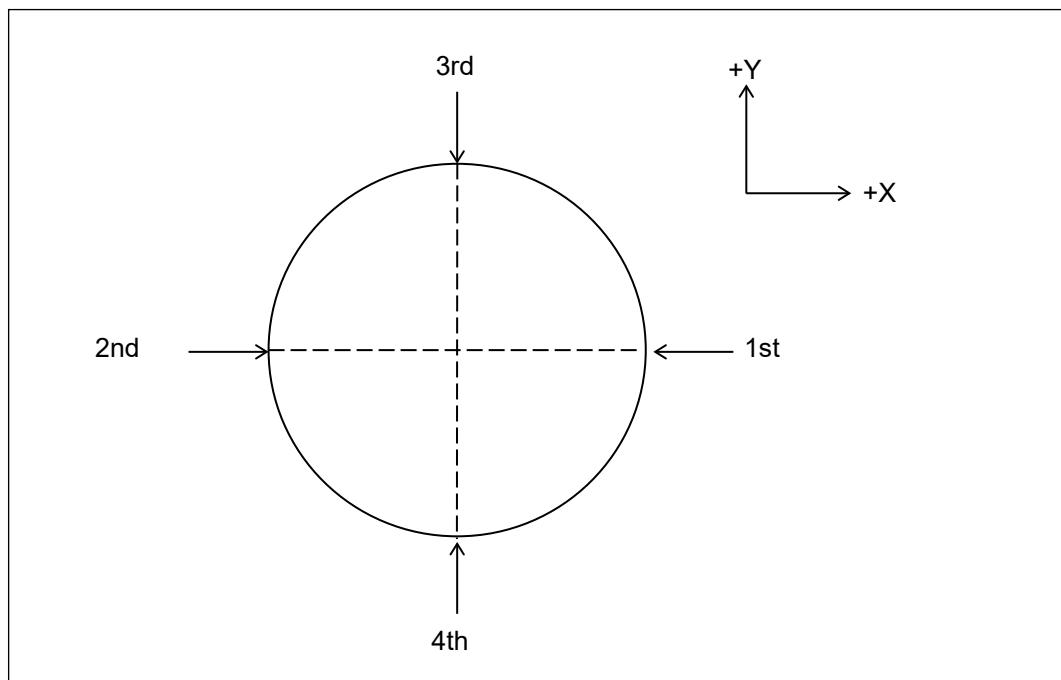
Hole and 4-point measurement  
(Motion)

1. Spindle orientation 0°. 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°. Travels to the second measurement start position for the X- and Y-axes.
5. Second measurement. (X-axis minus direction)
6. Spindle orientation 0°. Travels to the third measurement start position for the X- and Y-axes.
7. Third measurement. (Y-axis plus direction)
8. Spindle orientation 180°. 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°. 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°. 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°. 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°. 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.

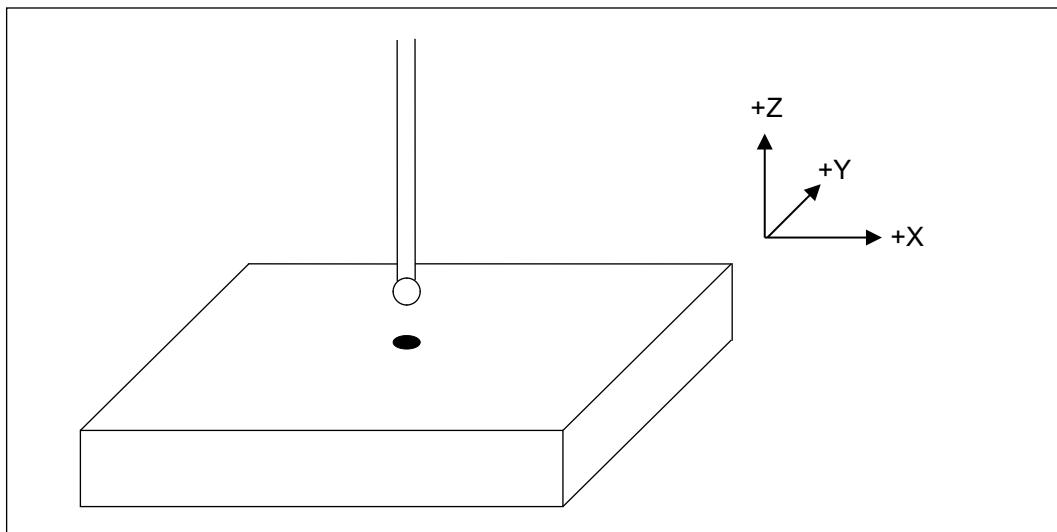


#### 7.4.4 Workpiece Top Surface

Command format

**G128 X\_ Y\_ Z\_ Q\_;**

- X and Y : X and Y coordinates at measurement point  
Z : Estimate for the top surface of the workpiece  
Q : Measurement number (1 to 4 and 1 is used when omitted)



**(NOTE)** Before carrying out (G128), perform <Automatic workpiece measurement compensation (workpiece top surface)> for the automatic centering, and set the compensation in the user parameter (automatic workpiece measurement / automatic centering) <Automatic workpiece measurement compensation Z (workpiece top surface)>.

**7**

(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)

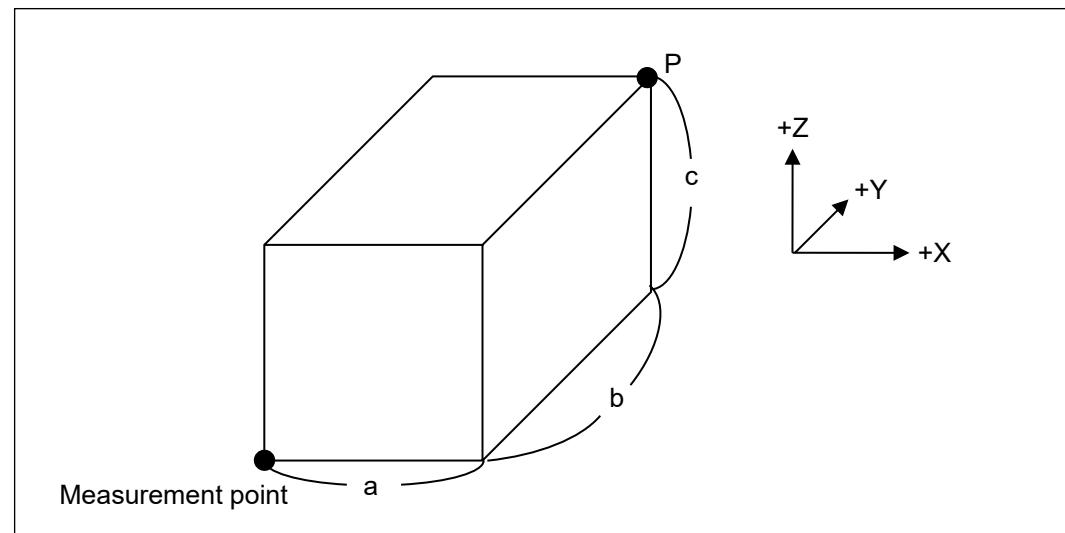
### 7.4.5 Positioning to Measurement Position

Command format

**G120 X\_ Y\_ Z\_ Q\_;**

X, Y and Z : Incremental amount from measurement position

Q : Measurement result number (1 to 4 and 1 is used when omitted)



**G120 Xa Yb Zc**

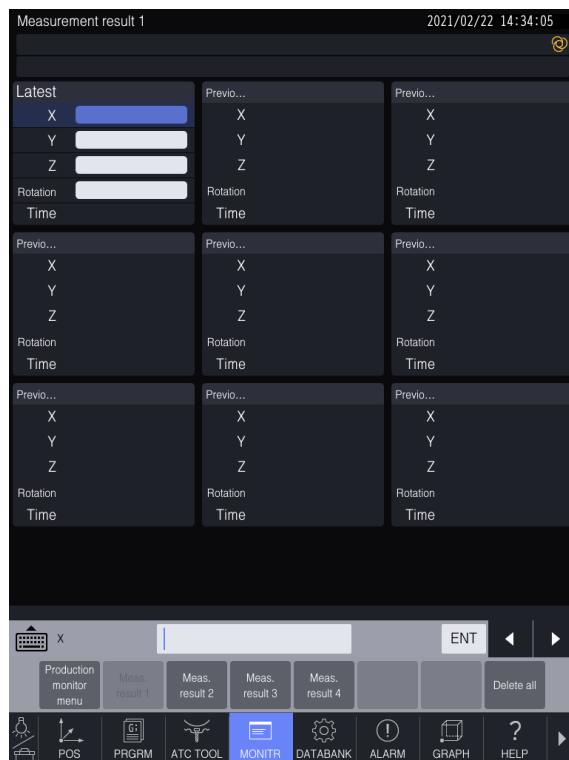
When the above command is issued, it travels to P in the figure.

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.



7

When continuing to perform measurements, the measurement results that were previously carried out are displayed.



## 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

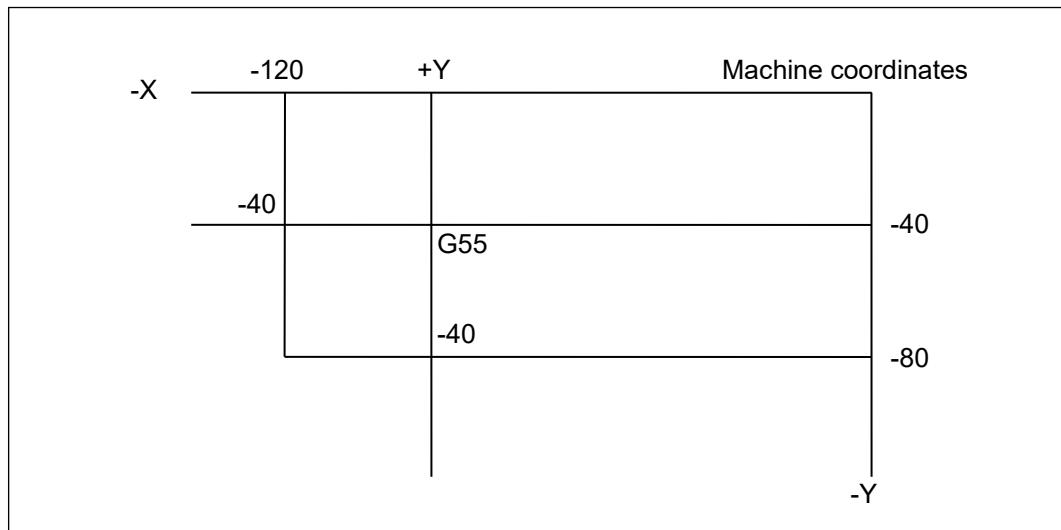
G10 L98 Pn X\_ Y\_ Z\_ Q\_;

Pn : n = 1 G54.1 P1  
               2 G54.1 P2  
               •       •  
               •       •  
               •       •  
               300 G54.1 P300  
 X, Y and Z : Coordinates for measurement position  
 Q            : Measurement result number (1 to 4)

Ex: When the measurement value for Measurement Result Number 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;

7



The workpiece coordinate data G55 is rewritten as:

X -80.000  
 Y -40.000

(NOTICE) 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.

# CHAPTER 8

## SUB PROGRAM FUNCTION

- 8.1      Overview**
- 8.2      Create Sub Program**
- 8.3      Simple Call Function**
- 8.4      External Sub Program Call Function**
- 8.5      Specify Return Number from Sub Program Function**
- 8.6      Call Specifying Sequence No.**

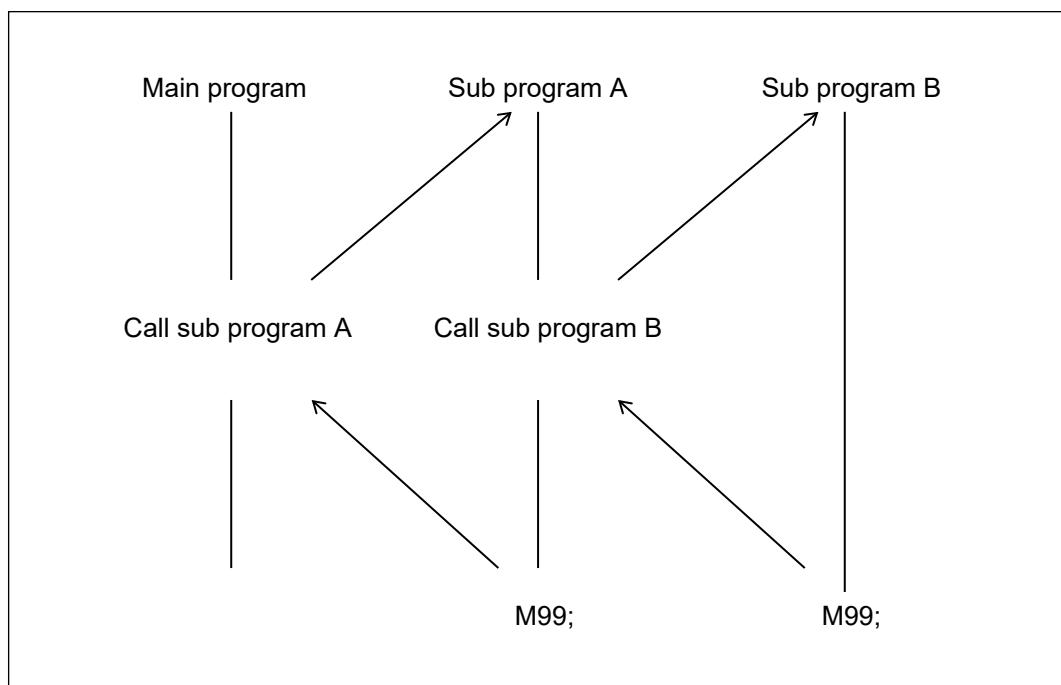
## 8.1 Overview

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. In addition, a sub program registered in an external device (memory card) can be called. However, a sub program registered in an external device (general communications device) cannot be called.

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.

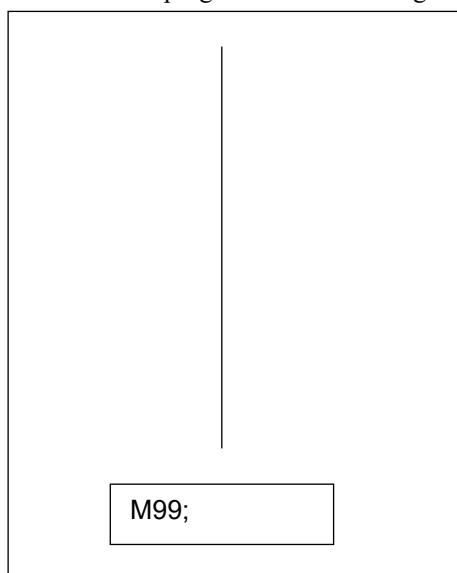
Furthermore, a separate sub program can also be called from a sub program which was called. In this situation, up to 8 sub programs can be called.



A sub program can also be called repeatedly with one call command.

## 8.2 Create Sub Program

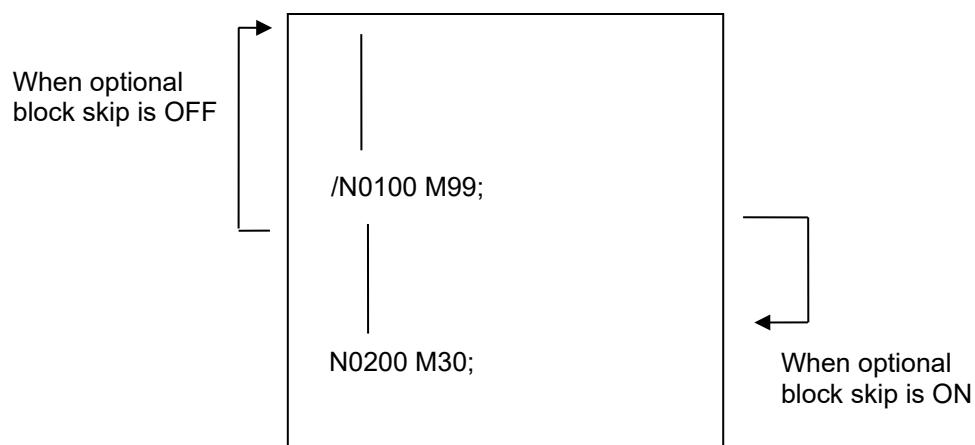
A standard sub program is created using a format like the following.



- (NOTE 1) Always specify M99 at the last block of the sub program.  
 (NOTE 2) Another G or M code command cannot be issued on the M99 command block. The alarm <<Simultaneous specified code cannot be used.>> is triggered when a G code command is issued.  
 In addition, if a M code command is issued and the user parameter (switch 1: programming) <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 <<Multiple M codes cannot be used.>> is triggered.

### Special use of M99

When the M99 command is issued in the main program, excluding tape operation (general communications device), it returns to the beginning of the program and operation starts from the beginning of the program again.



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 MDI operation and tape operation (with a memory card or general communications device).  
There are two methods to call a program: using a program number and using a program name.

Command format

|                                   |                            |
|-----------------------------------|----------------------------|
| <b>M98 P_H_L_;</b> ...            | <b>Program number call</b> |
| <b>M98 &lt;****&gt;H_L -;</b> ... | <b>Program name call</b>   |

P : Call sub program number (current program when omitted), or  
<\*\*\*\*> : Call sub program name (\*\*\*\* refers to the program name)  
H : Sequence No. for sub program called (beginning block when omitted)  
L : Number of repeated calls (9999 times or less) (1 when omitted)

There are 3 folders that can be specified for calling a sub program. The description below shows how to specify a folder. In the default settings, the folder is the same as the main program.

| Sub program call folder                                                                                              | User parameter (switch 1: programming)<br>Selection: <Sub program folder selection *> (*:1 to 3) | User parameter (switch 1: programming)<br>Setting: <Sub program folder designation *> (*:1 to 3) |
|----------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------|
| Same folder as the main program<br>(NOTE) This folder is the root folder when using MDI operation or tape operation. | <1: Current folder>                                                                              | Grayed out and disabled.                                                                         |
| Specified folder                                                                                                     | <0: Specified folder>                                                                            | Folder name                                                                                      |
| Root directory or root folder                                                                                        | <0: Specified folder>                                                                            | Blank field                                                                                      |

When a program is called, a search is conducted starting from <Sub program folder selection 1> - <Sub program folder designation 1> and continues up to <Sub program folder selection 3> - <Sub program folder designation 3>. The first program that is found in the search is called.

8

- (NOTE 1) Another G or M code command cannot be issued on the M98 command block. The alarm <<Simultaneous specified code cannot be used>> is triggered when a G code command is issued. In addition, if an M code command is issued and the user parameter (switch 1: programming) <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 <<Multiple M codes cannot be used>> 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 32MB, after an M30 (M02) command, all programs that can be called with the macro variables, such as “M98P1” (1st 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.

Ex 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;      } After M30, all alternative “M98P\*\*” commands  
 M98P5;      } are entered as shown on the left  
 M98P100;

Ex 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;      } After M30, all alternative G code macros are  
 G500;      } entered as shown on the left  
 G600;

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 32MB, 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 32MB, the machine operates in the extended memory operation mode.
- (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) The axis does not travel even if an X-, Y-, Z-, A-, B- or C- axis command is issued on the M98 command block.
- (NOTE 8) When a sequence number (H address) and the number of repetitions (L address) are specified during tape operation (general communications device), the alarm <<Subprogram call error>> is triggered.
- (NOTE 9) When a command is issued for P\_ and <\*\*\*\*> simultaneously, the data is overwritten after the command.

## 8.4 External Sub Program Call Function

Any sub program (hereafter, “external sub program”) registered in the root folder of an external device (memory card) is called and executed from the main program or the master’s sub program. When using the external sub program call function in memory operation, it runs in extended memory operation mode.

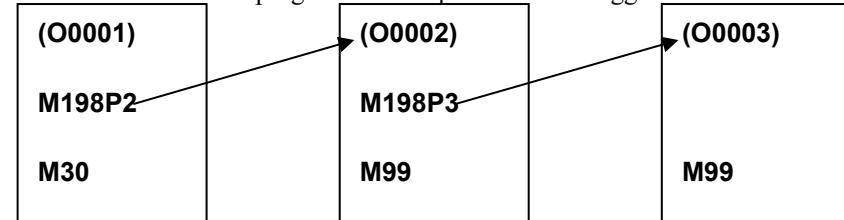
There are two methods to call a program: using a program number and using a program name.

Command format

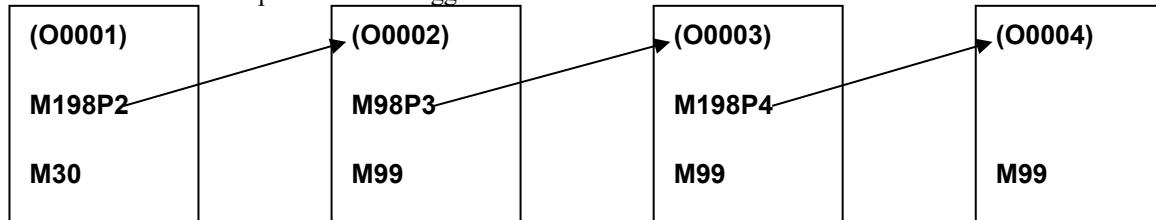
|                             |                         |
|-----------------------------|-------------------------|
| <b>M198 P_ L_;</b>          | ... Program number call |
| <b>M198 &lt;****&gt;L_;</b> | ... Program name call   |

P : Call sub program number, or  
 <\*\*\*\*> : Call external sub program name (\*\*\*\* refers to the program name)  
 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 M198 command block. The alarm <<Simultaneous specified code cannot be used>> is triggered when a G code command is issued. In addition, if an M code command is issued and the user parameter (switch 1: programming) <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 <<Multiple M codes cannot be used>> is triggered.
- (NOTE 2) If the specified external sub program does not exist, the alarm <<No subprogram>> is triggered when the external sub program call is executed.
- (NOTE 3) The axis does not travel even if an X-, Y-, Z-, A-, B- or C-axis command is issued on the M198 command block.
- (NOTE 4) Macro variables can be used in an external sub program number to call.
- (NOTE 5) The external 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 6) If M98 is executed while an external device (memory card) program is being executed with M198, 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) Calling an external sub program from tape operation is not possible. The alarm <<External sub program call not possible>> is triggered.
- (NOTE 8) When the external I/O device connection is not set to memory card, calling an external sub program is not possible. The alarm <<External sub program call not possible>> is triggered.
- (NOTE 9) Calling an external sub program from an external sub program is not possible. The alarm <<External sub program call not possible>> is triggered.



- (NOTE 10) Calling a sub program from an external sub program is possible, but after a sub program is called by an external sub program, any sub program thereafter cannot call an external sub program. Otherwise, the alarm <<External sub program call not possible>> is triggered.



## 8.5 Specify Return Number from Sub Program Function

Command format

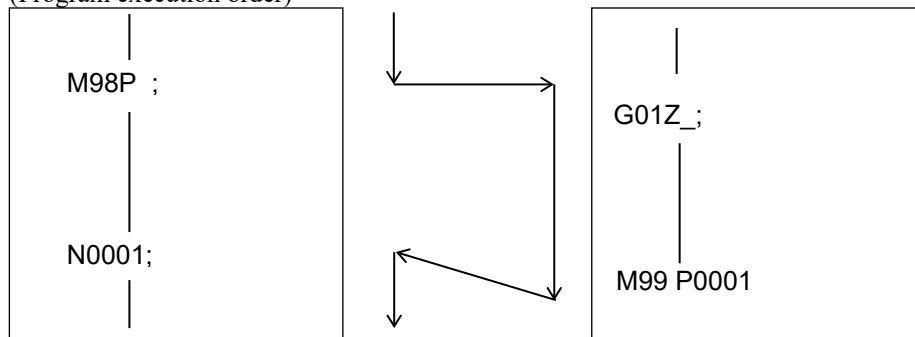
**M99 P\_;**

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)



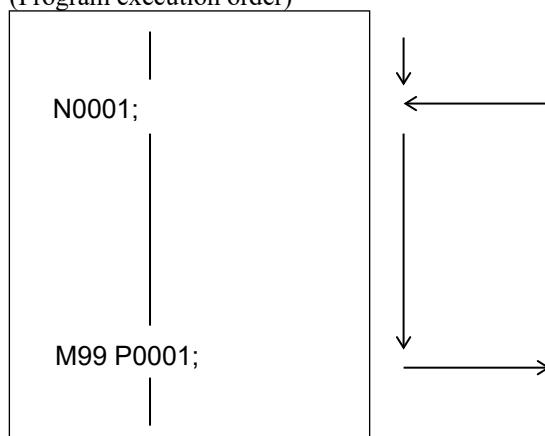
(NOTE 1) The alarm <<Subprogram return error>> is triggered if the number of repetitions in the M98 command is not 1.

(NOTE 2) The return number is ignored when carrying out tape operation (general communications device).

### 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)



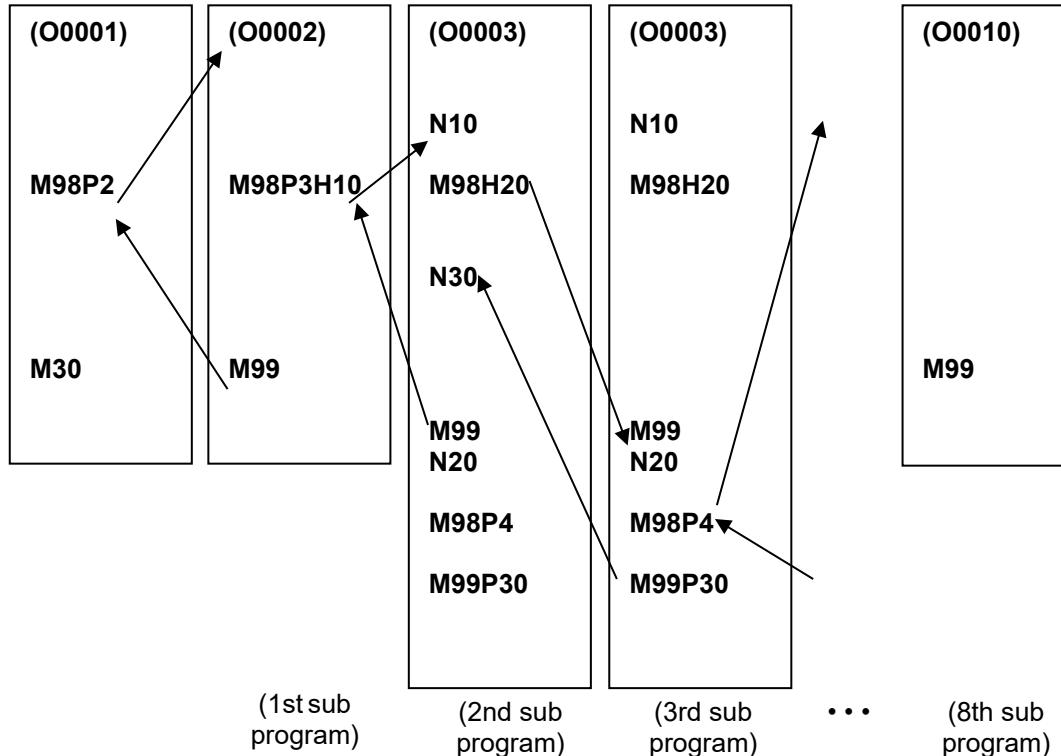
(NOTE) A command cannot be issued in tape operation (general communications device). Otherwise, the alarm <<Subprogram return error>> is triggered.

## 8.6 Call Specifying Sequence No.

When issuing a command with M98P\_H\_ or M98<\*\*\*\*>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



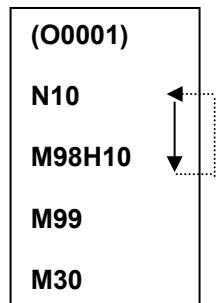
8

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 both the program number or program name command and the H address. When carrying out tape operation (general communications device), program number or program name command cannot be omitted because an H address command is not possible.
- (NOTE 5) The alarm <<Subprogram call error>> is triggered when the current program is called. 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”. However, a macro variable cannot be used when a program name command is issued such as <#100>.
- (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.

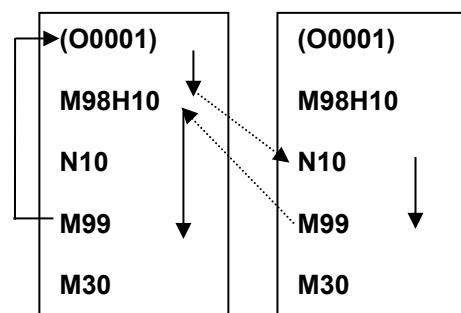
- (NOTE 9) Calls (M98H\_) which specify sequence numbers in the same program cannot be issued during tape operation (general communications device). Otherwise, the alarm <<Subprogram return error>> is triggered.

Ex 1:



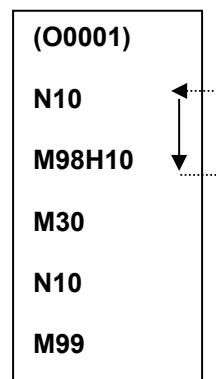
The alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time in a program as shown on the left.

Ex 2:



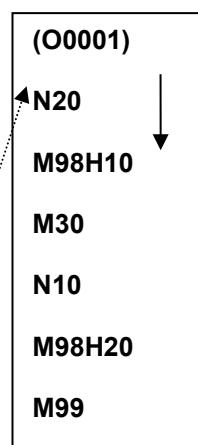
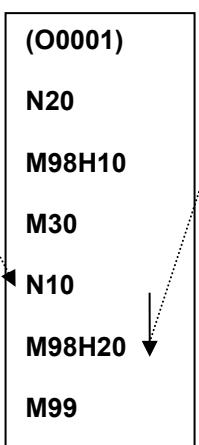
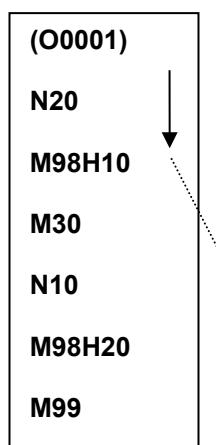
When creating a program as shown on the left, the operation continues because it will not reach M30.  
When creating a sub program in one program, create it after M30 (M02).

Ex 3:



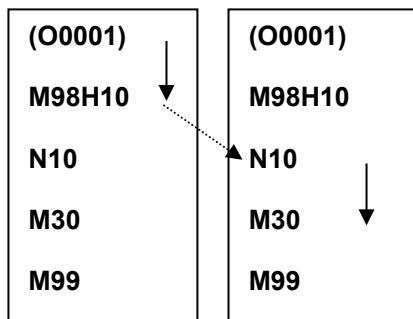
The sequence number search for the H address starts from the beginning of the program. As shown on the left, it jumps close to the block closer to beginning of the program when there are two blocks for “N10”.  
(The alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time.)

Ex 4:



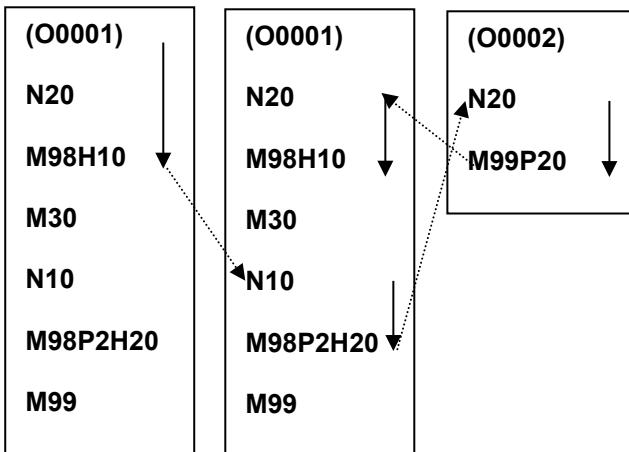
When creating a program as shown on the left, the alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time.

Ex 5:



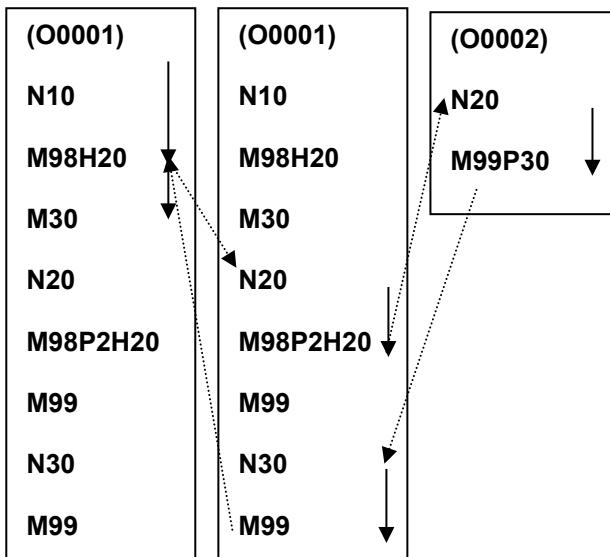
When creating a program as shown on the left, the program ends with M30.

Ex 6:



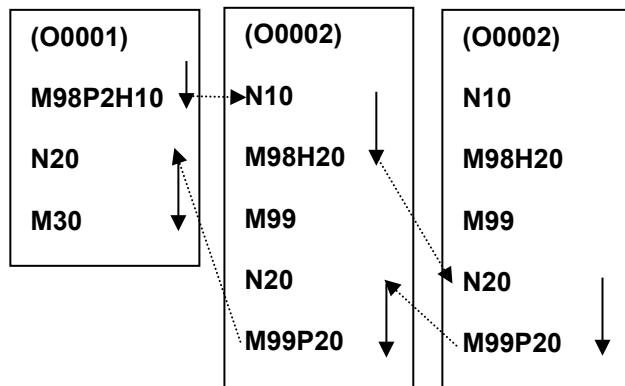
When creating a program as shown on the left, the alarm <<Subprogram call error>> is triggered when the “M98H10” block is executed the second time.

Ex 7:



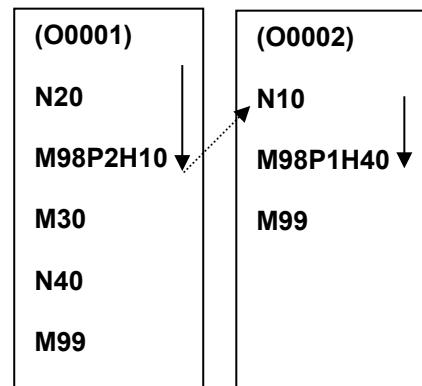
When creating a program as shown on the left and the H20 is executed for the second time in the “M98P2H20” block, an error does not trigger because a different program number is called. This program ends normally with M30.

Ex 8:



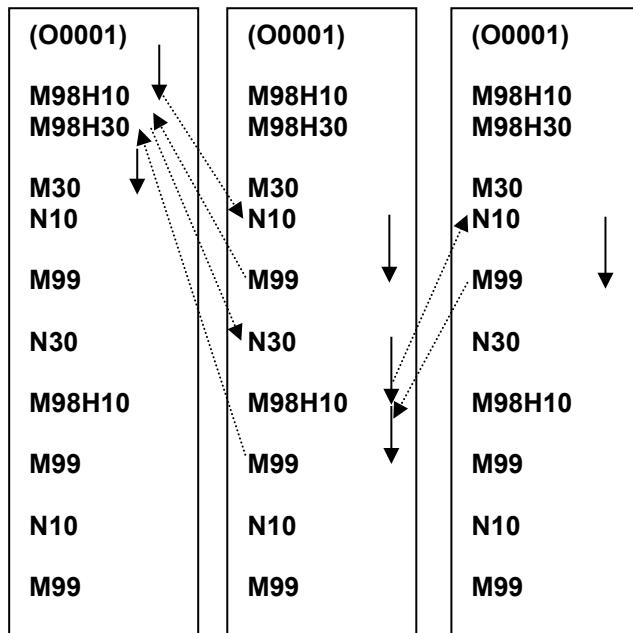
When creating a program as shown on the left, “N20” is called in the “M98H20” block and it is called again in the “M99P20” block, but an error does not trigger. This program ends normally with M30.

Ex 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.

Ex 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.  

$$\text{Execution speed [mm (inch)/min]} = \text{Feedrate [mm (inch)/rev]} \times \text{Spindle speed [min}^{-1}\text{]} \\ \times (\text{Spindle override [%]} / 100) \times (\text{Cutting override [%]} / 100)$$

(NOTE 2) If the spindle rotation is zero when the feedrate per rotation command is executed, the alarm <<Feedrate error>> is triggered.

(NOTE 3) When a command is issued during TCP control (G43.4/G43.5), the alarm <<TCP under control>> is triggered. In addition, when a TCP control command is issued during feedrate per rotation, the alarm <<TCP control command not possible>> 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.

9

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

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

```

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 the NO<sub>2</sub> block is as follows:

In the above example, the feedrate for the NCZ block is as follows.  
 Feedrate (mm/min) = Radius of arc from start point (mm)  $\times$  F(1/min)  
 $= 50 \times 10$   
 $= 500$

Example of inverse time calculation

Ex 1: When ending 1 block, the following command calculation is used.

$$\begin{aligned}\text{Inverse time [1/min]} &= 1 \div 1 \text{ block end time [sec]} \div 60[\text{min}] \\ &= (1 \div 60) \div (1 \div 60) \\ &= 1 \quad (\text{command is issued with F1})\end{aligned}$$

Ex 2: Travel time when issuing a F0.25 command is as follows.

$$\begin{aligned}\text{Travel time [min]} &= 1 \div \text{Inverse time [1/min]} \\ &= 1 \div 0.25 \\ &= 4.0 (\text{travel time is 4 minutes})\end{aligned}$$

|                                                    | 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
  - TCP control (G43.4/G43.5) in progress

#### 9.1.4 Command Range

| Minimum unit setting | Command                     | Command range                                                     |                                                                         |
|----------------------|-----------------------------|-------------------------------------------------------------------|-------------------------------------------------------------------------|
|                      |                             | Meters                                                            | Inches                                                                  |
| Type 1 (Micron)      | Feedrate per minute (G94)   | 0.01 ~ 999999.99 mm/min<br>0.01 ~ 999999.99 deg/min               | 0.001 ~ 99999.999 inch/min<br>0.001 ~ 99999.999 deg/min                 |
|                      | Feedrate per rotation (G95) | 0.0001 ~ 999999.9999 mm/rev<br>0.0001 ~ 999999.9999 deg/rev       | 0.000001 ~ 99999.99999 inch/rev<br>0.000001 ~ 99999.99999 deg/rev       |
|                      | Inverse time feed (G93)     | 0.001 ~ 999999.999 1/min                                          |                                                                         |
| Type 2 (Submicron)   | Feedrate per minute (G94)   | 0.001 ~ 999999.999 mm/min<br>0.001 ~ 999999.999 deg/min           | 0.00001 ~ 99999.99999 inch/min<br>0.00001 ~ 99999.99999 deg/min         |
|                      | Feedrate per rotation (G95) | 0.000001 ~ 999999.99999 mm/rev<br>0.000001 ~ 999999.99999 deg/rev | 0.0000001 ~ 99999.9999999 inch/rev<br>0.0000001 ~ 99999.9999999 deg/rev |
|                      | Inverse time feed (G93)     | 0.001 ~ 999999.999 1/min                                          |                                                                         |

Feedrate per minute (G94) and feedrate per rotation (G95):

The alarm <<Max. speed exceeded>> or <<Feedrate error>> is triggered for a feedrate when an attempt is made to move the axis and the speed exceeds the user parameter (switch 1) setting <Max. actual cutting travel speed (linear axis / rotation axis)>, the machine parameter (system 1: X-, Y- and Z-axes) setting <X-(to Z)-axis max. cutting feedrate> or the machine parameter (system 2: additional axis) setting <5th-(to 8th)-axis max. cutting rotation speed>.

Inverse time feed (G93):

When the calculated feedrate has been exceeded, the operation follows the user parameter setting (switch 1: programming) <Cutting speed exceeded for inverse time feed>.

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<br>0: Meter | 0.01 mm/min<br>0.01 deg /min     | 0.001 mm/min<br>0.001 deg/min    |
| Machine unit setting<br>1: Inch  | 0.001 inch/min<br>0.001 deg /min | 0.0001 mm/min<br>0.0001 deg /min |

When calculating the execution speed of the feedrate per rotation, if the value (Feedrate [mm/inch]/rev × Spindle speed [ $\text{min}^{-1}$ ]) before applying the override is less than the minimum value of the executable range for the feedrate per minute, then the minimum value becomes the feedrate.

Ex: When the minimum unit setting is Type 1 (micron) in meters and when the feedrate is 0.0001 mm/rev and the spindle speed is 99  $\text{min}^{-1}$ :

Value (mm/min) before applying override =  $0.0001 \text{ mm/rev} \times 99 \text{ min}^{-1} = 0.0099 \text{ mm/min}$   
Since the value is less than the minimum 0.01 mm/min, the calculation result is adjusted to 0.01 mm/min.

In addition, when the value after applying the override (actual speed) is less than the minimum of the command range, the actual speed becomes 0 mm/inch/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 (switch 1: canned cycle) <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 “5.7 Canned cycle for tool change (nonstop ATC) (G100)” 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

## 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.

10

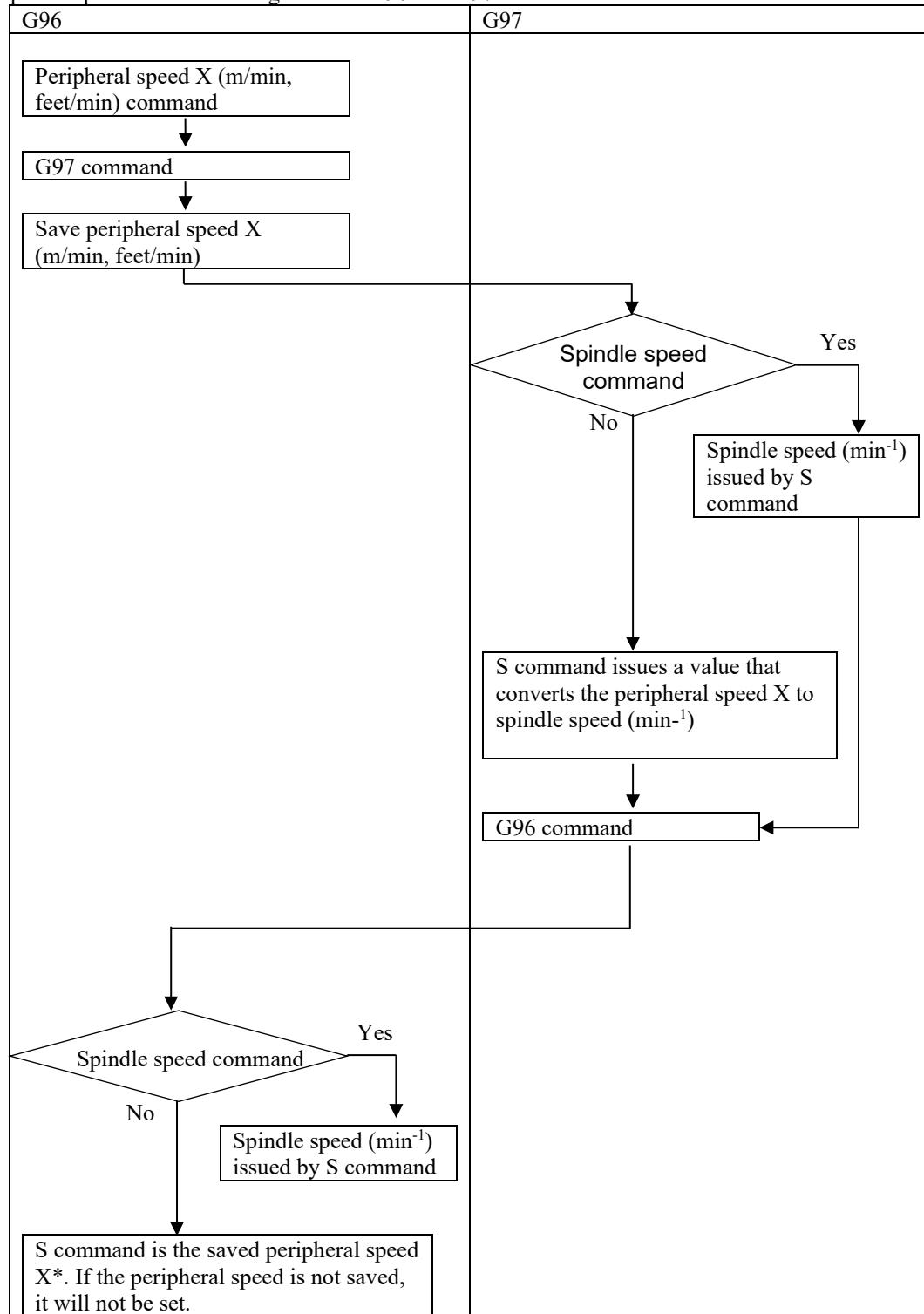
When using the constant peripheral speed control, configure the settings so that the workpiece coordinate zero for the control axis is in the lathe spindle center.

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.
- (NOTE 3) When a command for the constant peripheral speed control (G96) is issued during TCP control, the alarm <<TCP under control>> is triggered. In addition, when the TCP control (G43.4/G43.5) command is issued during G96 modal, the alarm <<TCP control command not possible>> 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

**G92 S \_Q\_;**

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: canned cycle) <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 “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: programming) <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: common) <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, T201 to T299)

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 are 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: program) <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 “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.<br>- Standard tool turn speed when a standard tool is used (NOTE 2).<br>- 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 D00 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.<br>- Tool type being changed is a large tool<br>- 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)

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: programming) <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 mechanisms, the magazine turn speed can be changed from the program.

(NOTE 1) (NOTE 2)

M435: Maximum magazine turn speed

M436: Magazine turn speed 1

M437: Magazine turn speed 2

Use the following operations to change back to the M435 modal.

- 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 (switch 1: ATC/Magazine) <Magazine turn speed ratio 1(M436)> and <Magazine turn speed ratio 2(M437)>.

(NOTE 1) This function is disabled on a turret type ATC mechanism. When a command is issued, it follows the user parameter (switch 1: programming) <Unregistered M-code>.

(NOTE 2) When the M435, M436, M437 commands are issued on the same block as a T command, the alarm <<Invalid command code and invalid command address>> 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) The tool numbers 1 to 99 can be specified. The tool numbers 201 to 299 cannot be specified.

(NOTE 2) This function is disabled on a turret type ATC mechanism. When a command is issued, it follows user parameter (switch 1: programming) <Unregistered M-code>.

(NOTE 3) 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 3 Machining load monitor function” in the Data Bank & Alarm 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 (In-position Check Distance)**
- 12.7    M Function (Time Constant Switch)**
- 12.8    M Function (Brake Load Test)**

## 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 “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                                   |                        |                               |                   |
|       | 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                             | -                      |                               |                   |
|       | M99    | Return from Subprogram                       | -                      |                               |                   |
|       | M198   | External sub program call                    | -                      |                               |                   |
|       | M120   | Tool breakage error check (ON)               |                        |                               |                   |
|       | M121   | Tool breakage error check (OFF)              | After                  | One-shot                      | Possible          |
|       | M141*  | Selection of spindle                         |                        |                               |                   |
|       | M142   | Selection of turning spindle                 | Simultaneous           | Modal                         | Possible          |
|       | 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                      |                        |                               |                   |
|       | M221*  | Workpiece counter 1 cancel                   | Simultaneous           | Modal                         | Possible          |
|       | M212   | Workpiece counter 2 set                      |                        |                               |                   |
|       | M222*  | Workpiece counter 2 cancel                   | Simultaneous           | Modal                         | Possible          |
|       | M213   | Workpiece counter 3 set                      |                        |                               |                   |
|       | M223*  | Workpiece counter 3 cancel                   | Simultaneous           | Modal                         | Possible          |
|       | M214   | Workpiece counter 4 set                      |                        |                               |                   |
|       | M224*  | Workpiece counter 4 cancel                   | Simultaneous           | Modal                         | Possible          |
|       | M230*  | Tool life counter set                        |                        |                               |                   |
|       | M231   | Tool life counter cancel                     | Simultaneous           | Modal                         | Possible          |
|       | M238   | To start waveform display measurement        |                        |                               |                   |
|       | M239   | To finish waveform display measurement       | -                      | Modal<br>(NOTE 4)<br>(NOTE 5) | Impossible        |

| Group | M code       | Contents                                                                          | Timing of axis motions | Modal / One-shot           | Multiple commands |
|-------|--------------|-----------------------------------------------------------------------------------|------------------------|----------------------------|-------------------|
|       | M241 to M249 | Tap time constant 10 ~ 90%                                                        | Simultaneous           | Modal (NOTE 1)             | Possible          |
|       | M250         | Tap time constant selection                                                       |                        |                            |                   |
|       | M251         |                                                                                   |                        |                            |                   |
|       | M252 to M254 | Tap acceleration kept constant selection<br>High speed / Medium speed / Low speed |                        |                            |                   |
|       | M258         | Production monitor - Time measurement start                                       | Simultaneous           | Modal (NOTE 4)<br>(NOTE 5) | Possible          |
|       | M259         | Production monitor - Time measurement end                                         |                        |                            |                   |
|       | 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         | Cancel in-position check distance change                                          | Before                 | Modal (NOTE 2)             | Possible          |
|       | M271 to M279 | Change in-position check distance                                                 |                        |                            |                   |
|       | M290*        | Tool replacement Z axis lower speed 100%                                          | Before                 | Modal                      | Possible          |
|       | M291 to M293 | Tool replacement Z axis lower speed 1~3                                           |                        |                            |                   |
|       | M295         | Diagnosis of tool wash liquid surface sensor failure                              | -                      | One-shot                   | Impossible        |
|       | M298         | Machining mode specification                                                      |                        |                            |                   |
|       | M299         | Cancel machining mode specification                                               | -                      | Modal                      | Impossible        |
|       | M300         | Z-axis perimeter mode on                                                          |                        |                            |                   |
|       | M301         | Z-axis perimeter mode off                                                         | Before                 | Modal (NOTE 3)             | Possible          |
|       | M303         | Forward rotation of turning spindle                                               |                        |                            |                   |
|       | M304         | Backward rotation of turning spindle                                              | Simultaneous           | Modal                      | Possible          |
|       | M305*        | To stop turning spindle                                                           |                        |                            |                   |
|       | M320         | Measurement device sensor ON confirmation                                         | Before                 | One-shot                   | Possible          |
|       | M321         | Signal OFF check 1 for measuring instrument detection                             |                        |                            |                   |
|       | M324         | Signal OFF check 2 for measuring instrument detection                             |                        |                            |                   |
|       | M325         | Signal OFF check 3 for measuring instrument detection                             |                        |                            |                   |
|       | M326         | Signal OFF check 4 for measuring instrument detection                             |                        |                            |                   |
|       | M327         | Signal OFF check 90 for measuring instrument detection                            |                        |                            |                   |
|       | 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)                       | -                      | One-shot                   | Impossible        |
|       | M368         | Cancel thermal distortion compensation (Ball screw)                               |                        |                            |                   |
|       | M369         | Cancel thermal distortion compensation (Spindle)                                  |                        |                            |                   |
|       | M400         | M400ON (chip shower ON)                                                           | Simultaneous           | Modal                      | Possible          |
|       | M401*        | M400OFF (chip shower OFF)                                                         |                        |                            |                   |
|       | M402         | M402ON                                                                            | Simultaneous           | Modal                      | Possible          |
|       | M403*        | M402OFF                                                                           |                        |                            |                   |

## Chapter 12 M Function

| Group | M code | Contents                                                                                        | Timing of axis motions | Modal / One-shot | Multiple commands |  |  |  |
|-------|--------|-------------------------------------------------------------------------------------------------|------------------------|------------------|-------------------|--|--|--|
|       | 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   | ATC arm turn speed 1                                                                            |                        |                  |                   |  |  |  |
|       | M422   | ATC arm turn speed 2                                                                            |                        |                  |                   |  |  |  |
|       | 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                                                                 |                        |                  |                   |  |  |  |
|       | M436   | Magazine rotation speed 1                                                                       | -                      | Modal            | Possible          |  |  |  |
|       | M437   | Magazine rotation speed 2                                                                       |                        |                  |                   |  |  |  |
|       | M438   | Magazine/Pot shutter open                                                                       | -                      | One-shot         | Impossible        |  |  |  |
|       | M439   | Magazine/Pot shutter close                                                                      |                        |                  |                   |  |  |  |
|       | 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 turned off)                 | Simultaneous           | One-shot         | Possible          |  |  |  |
|       | M451   | To output one-shot (Proceeds to the next block without waiting until the signal has turned off) |                        |                  |                   |  |  |  |
|       | M455   | To output one-shot (Proceeds to the next block without waiting until the signal has turned off) |                        |                  |                   |  |  |  |
|       | 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                                                                     |                        |                  |                   |  |  |  |
|       | M470   | Forced execution of brake load test                                                             | -                      | One-shot         |                   |  |  |  |
|       | M471   | Brake load test                                                                                 |                        |                  |                   |  |  |  |
|       | M474*  | Coil conveyor automatic mode: Enable                                                            | Before                 | Modal            | Possible          |  |  |  |
|       | M475   | Coil conveyor automatic mode: Disable                                                           |                        |                  |                   |  |  |  |
|       | M476   | Tool cleaning blowoff valve ON (MDI operation only)                                             | -                      | One-shot         | Impossible        |  |  |  |
|       | M477   | Tool cleaning blowoff valve OFF (MDI operation only)                                            |                        |                  |                   |  |  |  |
|       | M478   | Automatic oiling / greasing cycle ON                                                            | -                      | One-shot         | Impossible        |  |  |  |
|       | M479   | Liquid removal cycle ON                                                                         |                        |                  |                   |  |  |  |
|       | M480   | M480 signal ON                                                                                  | Simultaneous           | One-shot         | Impossible        |  |  |  |
|       | M481*  | M480 signal OFF                                                                                 |                        |                  |                   |  |  |  |
|       | M482   | M482 signal ON                                                                                  | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M483*  | M482 signal OFF                                                                                 |                        |                  |                   |  |  |  |
|       | M484   | M484 signal ON                                                                                  | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M485*  | M484 signal OFF                                                                                 |                        |                  |                   |  |  |  |
|       | M486   | M486 signal ON                                                                                  | Simultaneous           | Modal            | Possible          |  |  |  |
|       | M487*  | M486 signal OFF                                                                                 |                        |                  |                   |  |  |  |
|       | M494   | Coolant through center ON                                                                       | Before                 |                  |                   |  |  |  |
|       | M495*  | Coolant through center OFF                                                                      |                        |                  |                   |  |  |  |
|       | M489   | Back washing cycle ON (Waiting for completion)                                                  | -                      | One-shot         | Impossible        |  |  |  |

| Group | M code             | Contents                                    | Timing of axis motions | Modal / One-shot | Multiple commands |
|-------|--------------------|---------------------------------------------|------------------------|------------------|-------------------|
|       | 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          |

- (NOTE 1) The modal during power startup is specified in the user parameter (switch 1: canned cycle) <Tap accel. setting>.
- (NOTE 2) The modal during power startup is specified in the user parameter (switch 3: in-position check distance) <Initial modal for in-position check distance>.
- (NOTE 3) The modal during power startup is specified in the user parameter (switch 1:programming) <Z-axis perimeter function>.
- (NOTE 4) Even when the user parameter (switch 1: operation) <S.T.M. return> is set to <1: Yes>, it does not restore to the last modal status in the M code group when restarting the program.
- (NOTE 5) Even if the user parameters (switch 3: modal display setting): <M code for modal display 1> to <M code for modal display 16> are not set to 0 (zero), the M code modal does not display on the <Modal info 1> and <MDI operation> screens.

### 12.2.1 Multiple M Code Commands in One Block

If the user parameter (switch 1: programming) <Multiple M codes in one block> is set to <1:Yes>, M codes with the mark “Possible” refer to “multiple commands” in the “12.2 M code list” that can be used to issue up to three M code commands on one block at the same time.

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.

When an M code that cannot be specified at the same time is issued on 1 block, the alarm <<Multiple M codes cannot be used.>> is triggered.

|                     | 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 | 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  |
| M159 | M198 | M203 | M238 |
| M239 | M298 | M299 | M340 |
| M341 | M342 | M343 | M350 |
| M351 | M352 | M353 | M355 |
| M360 | M361 | M362 | M363 |
| M364 | M365 | M366 | M367 |
| M368 | M369 | M489 | M434 |
| M438 | M439 | M448 | M449 |
| M470 | M471 | M476 | M477 |
| M478 | M479 | M498 |      |

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 “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 |
| M258, M259                             |
| M260 to M267, M269                     |
| M280 to M287, M289                     |
| M270, M271 to M279                     |
| M290, M291 to M293                     |
| M300, M301                             |
| M320, M321, M324 to M327               |
| 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                             |
| 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: operation) 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 cancel 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.

### 12.3.7 Time Measurement (M258 and M259)

The time measurement starts when M258 is executed. If an M258 command is specified during the time measurement, the measurement time up until then is recorded and a new measurement starts. The time measurement ends when M259 is executed. In addition, if the power is turned OFF during the measurement, or if the operation is reset by pressing the [RST] key or by executing M02 or M30, the time measurement ends.

The M258 and M259 commands can be issued even during MDI operation.

The measurement is shown in intervals of 100 ms and can be checked on the production monitor screen.

(NOTE) If the time measurement (M258, M259) extends over into successive blocks for the cutting feed, the in-position check is carried out.

As a result, it may not be possible to obtain the expected value in program example 1 below. Therefore, specify the commands on the same block, as shown in program example 2.

Ex.1: G01 X-10.000 Y-10.000;  
M258;

G01 X-11.500 Y-11.000;

Ex.2: G01 X-10.000 Y-10.000;  
G01 X-11.500 Y-11.000 M258;

### 12.3.8 Z-axis Perimeter Mode (M300 and M301)

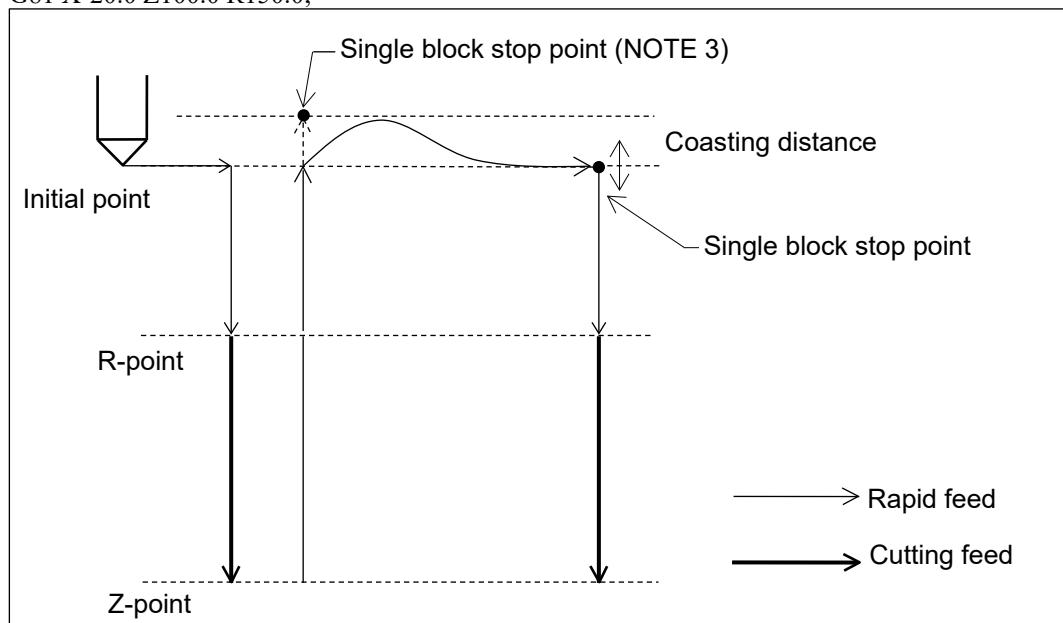
When an M300 command is issued, the Z-axis perimeter mode is turned ON. The user can shorten the rapid feed operation between blocks when the Z-axis rapid feed up operation and X-/Y-axis rapid feed operation are continuous. In this situation, the perimeter motion is carried out at the top only for the distance (Note 1) the Z-axis coasts to the command position. By starting the rapid feed on the X- and Y-axes while the Z-axis is coasting, the user can shorten the waiting time while the Z-axis decelerates and thereby shorten the cycle time.

Make sure that the tool, workpiece and jig do not interfere with each other during the perimeter motion. In addition, the total travel distance on the Z-axis increases compared to when the Z-axis perimeter motion mode is OFF.

After an M301 command is issued, the Z-axis perimeter mode is turned OFF. During the M301 modal, the perimeter motion will not be carried out.

The example below shows the operation when the Z-axis perimeter mode is turned ON.

Program example  
G00 G98 X0.0 Z200.0;  
M300;  
G81 Z100.0 R150.0;  
G81 X-20.0 Z100.0 R150.0;



- (NOTE 1) The Z-axis coating distance varies depending on the Z-axis travel distance and the X-/Y-axis travel distance. The upper limit of the coasting distance is set in the user parameter (switch 1: programming) <Upper limit for Z-axis perimeter coasting distance>. The perimeter motion is not carried out when the set value has been exceeded.
- (NOTE 2) The perimeter motion is carried out at the top within the stroke limit or within the stroke range.
- (NOTE 3) If there is a single block stop, the X- and Y-axes do not travel, and the Z-axis stops at a position that is equal to the command position plus the coasting distance.
- (NOTE 4) If the additional axis moves at the same time as the Z-axis up rapid feed operation or X-/Y-axis rapid feed operation, then the perimeter motion is not carried out.
- (NOTE 5) MDI intervention is not possible during the perimeter motion. The operator message <<Z-axis perimeter operation in progress>> appears when an attempt is made to change the mode.
- (NOTE 6) If an interrupt signal is detected during the M96 modal and the Z-axis perimeter motion, then the interrupt program is executed after the X-/Y-axis rapid feed operation is completed, regardless of the user parameter (switch 1: programming) <Interrupt macro interrupt type> setting.
- (NOTE 7) The machine is initialized in the modal that is set in the user parameter (switch 1: programming) <Z-axis perimeter function> 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 8) When the Z-axis perimeter mode is ON, the following commands will carry out the perimeter motion. Even if the rapid feed operation is not included in other commands (not listed below), the perimeter motion is not carried out regardless of the modal.
- Positioning (G00)
  - Reference position return (G28 to G30)
  - Complex thread cutting cycle (G376)
  - Thread cutting cycle (G392)
  - General canned cycles (Excluding right after the tapping cycle (G74, G77, G78, G84, G177, G178, G277 and G278) recovery operation)
- (NOTE 9) While under TCP control (G43.4/G43.5), the perimeter motion will not be carried out even when Z-axis perimeter mode is ON.

## 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, 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-D00, M96-M97 became internal use codes. If they are used, change the BCDoutput codes.

### 12.4.2 Tool Breakage Error Check (M120 and M121)

Before using the M120/M121 command, go to the user parameter (tool breakage detection) <Check location measurement setting for tool breakage detection> and use a signal setting from one of the user parameters (switch 1: programming): <Measurement setting 1> to <Measurement setting 4>.

When the M120 command is issued, the processing checks whether the measuring instrument detection is enabled. If the signal 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 Checking Measurement Instrument Detection Signal ON (M320)

When the M320 command is issued, if the measuring instrument detection signal is ON while a skip command (G31/131/132 issued right before) is being executed, the operation ends. If the signal is not ON, the alarm <<Detection signal off>> is triggered.

(NOTE) When an M320 command is issued, the check signal is turned OFF by one of the following operations: pressing the [RST] key, locking/unlocking the machine or turning OFF the power. In addition, the signal status is updated not just with a skip command but when a measurement function (automatic workpiece measurement (G121 to 129) and automatic centering) is used as well.

### 12.4.4 Checking Measurement Instrument Detection Signal OFF (M321, M324 to M327)

When the commands M321, M324, M325, M326 and M327 are issued, the processing first checks the user parameters (switch 1: programming) that correspond to the aforementioned M codes to confirm where the measuring instrument detection signal is enabled: <Signal setting 1 for measuring instrument detection>, <Signal setting 2 for measuring instrument detection>, <Signal setting 3 for measuring instrument detection>, <Signal setting 4 for measuring instrument detection> or <Signal setting 90 for measuring instrument detection>. If the signal is OFF in all of those parameters, the operation ends. If it is ON in one of the parameters, the alarm <<Detection signal on>> is triggered.

| M code | Corresponding parameter                                |
|--------|--------------------------------------------------------|
| M321   | <Signal setting 1 for measuring instrument detection>  |
| M324   | <Signal setting 2 for measuring instrument detection>  |
| M325   | <Signal setting 3 for measuring instrument detection>  |
| M326   | <Signal setting 4 for measuring instrument detection>  |
| M327   | <Signal setting 90 for measuring instrument detection> |

### 12.4.5 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 |

- After the user parameter (switch 1: installation) <Chip shower retention time> has elapsed, the M401 command turns OFF the chip shower.

### 12.4.6 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. The signal output time is set in the user parameter (switch 1: programming) <One shot signal M450/451/455/456 output time>.

### 12.4.7 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 wait time is set in the user parameter (switch 1: programming) <External signal reading time limit>.

After this time, the <<Signal output timeout>> alarm appears.

### 12.4.8 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 4 OM” for details.

### 12.4.9 Expansion Signal Output (M900 to M999)

Output the M900 signal and a BCD code in the lowest two digits.

## 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 <>Address where command is not possible>> is triggered.

Ex.1: G00 X-50. M444; ← When it travels along the X-axis, the C-axis unclamp is carried out at the same time.

Ex.2: G00 C90. M444; ← The alarm <>Address where command is not possible>> is triggered.

(NOTE) 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 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 <>Address where command is not possible>> is triggered.

Ex.1: G00 X-50. M445; ← When it travels along the X-axis, the C-axis clamp is carried out at the same time.

Ex.2: G00 C90. M445; ← The alarm <>Address where command is not possible>> is triggered

(NOTICE) 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.

(NOTE) 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 (NOTE) <>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.

(NOTE) When the user parameter (switch 1: programming) <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 <>Simultaneous specified code cannot be used.>> is triggered.

Ex.1: G00 Y-50. M440; ← When it travels along the Y-axis, the B-axis unclamp is carried out at the same time.

Ex.2: G00 B90. M440; ← The alarm <>Simultaneous specified code cannot be used.>> is triggered.

(NOTE) 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.

Ex.1: G00 Y-50. M441; ← When it travels along the Y-axis, the B-axis clamp is carried out at the same time.

Ex.2: G00 B90. M441; ← The alarm <>Address where command is not possible>> is triggered.

(NOTE) 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 (NOTE) <>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.

(NOTE) When the user parameter (switch 1: programming) <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 a A-axis travel command on the same block, the alarm <>Address where command is not possible<> is triggered.

Ex.1: G00 Z-50. M442; ← When it travels along the Z-axis, the A-axis unclamp is carried out at the same time.

Ex.2: G00 A90. M442; ←The alarm <>Address where command is not possible<> is triggered.

(NOTE) 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 a A-axis travel command on the same block, the alarm <>Address where command is not possible<> is triggered.

Ex.1: G00 Z-50. M443; ← When it travels along the Z-axis, the A-axis clamp is carried out at the same time.

Ex.2: G00 A90. M443; ← The alarm <>Address where command is not possible<> is triggered.

(NOTE) 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 (NOTE)<>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.

(NOTE) When the user parameter (switch 1: programming) <Unregistered M-code> is set to <0: Error>.

## 12.6 M Function (In-position Check Distance)

### 12.6.1 Change In-position Check Distance (M270 to M279)

M270: The values from the machine parameters (system 1: X-, Y- and Z-axes) <Positioning end check distance> and (system 2: additional axis) <Positioning end check angle> are used for the in-position width in the positioning operation.

The value from the machine parameter (system 1: X, Y and Z-axes/system 2: additional axis) <In-position width> is used for the in-position width between the positioning operation and cutting operation.

M271: The values from the user parameters (switch 3: in-position check distance) <M271 positioning end check distance> and <M271 positioning end check angle> are used for the in-position width in the positioning operation.

The X-, Y- and Z-axes use the <M271 positioning end check distance> value in the same way. The A-, B- and C-axes use the <M271 positioning end check angle> value in the same way.

The values from the user parameters (switch 3: in-position check distance) <M271 in-position width (X-, Y- and Z-axes)> is used for the in-position width between the positioning operation and cutting operation.

M272 to M279: These work as same as M271.

When one of the following operations is performed, the data is initialized to the modal that is set in the user parameter (switch 3: in-position check distance) <Initial modal for in-position check distance>.

- Program ends
- Operation reset
- Power OFF
- Reset (including switching the machine lock setting)

## 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).

- (NOTICE) 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) The machine is initialized in the modal that is set in the user parameter (switch 1: canned cycle) <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.

## 12.8 M Function (Brake Load Test)

### 12.8.1 Execute Brake Load Test (M470 and M471)

M470 : Forces the execution of the brake load test. This executes the test even on axes that have already been tested.

M471 : Executes the brake load test on axes that have not been tested.

- (NOTE 1) This test does not execute on the external additional axis (only machine models with QT) / PLC-axis.
- (NOTE 2) When the lathe spindle is selected and the test is executed for the lathe spindle, the alarm <<The lathe spindle is being selected.>> is triggered and the test cannot be executed.

(This page was intentionally left blank.)

# 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-dimentional shape workpiece machining to attain high-speed, high-accuracy, and high-dignity machining. The high accuracy mode AIII is equipped with two functions: one for improving the shape accuracy and one for improving the surface quality. The user can change the machining shape accuracy and surface quality by setting the override value in the user parameter.

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:

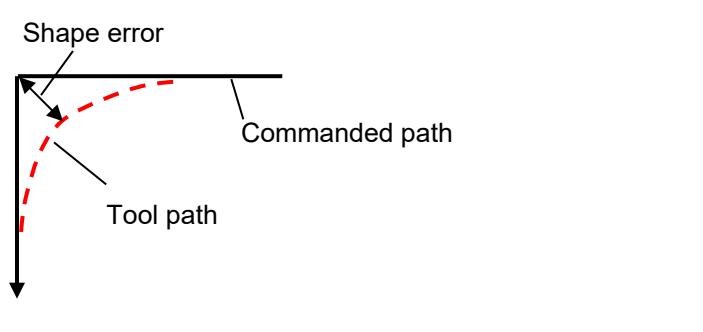
### 1. Improving shape accuracy

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

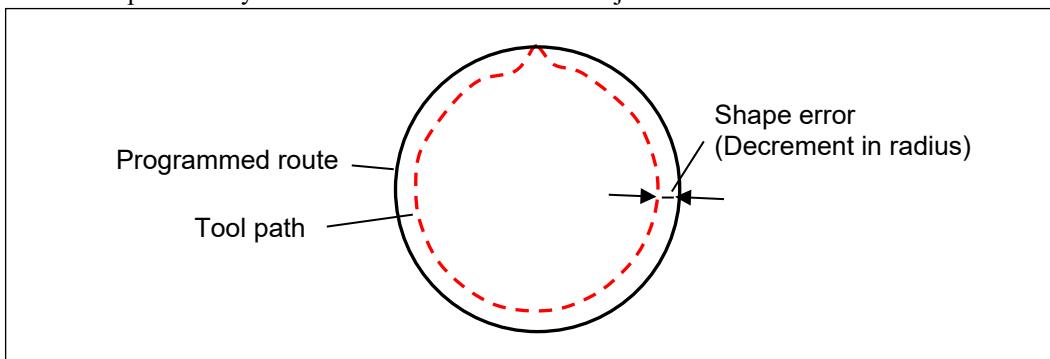
#### (1) Automatic corner decelerating function

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.



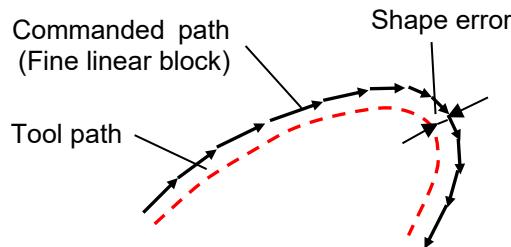
#### (2) Automatic arc decelerating function

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-dimentional shape. Curve approximate deceleration override can adjust the shape error.



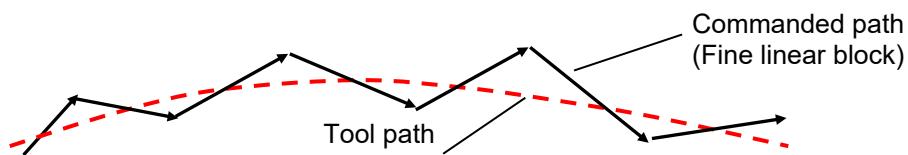
## (4) Fully automatic deceleration function

This function limits all errors in the command paths and tool paths in order to improve the shape accuracy. The user parameter (high accuracy: high accuracy A) <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-dimentional 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

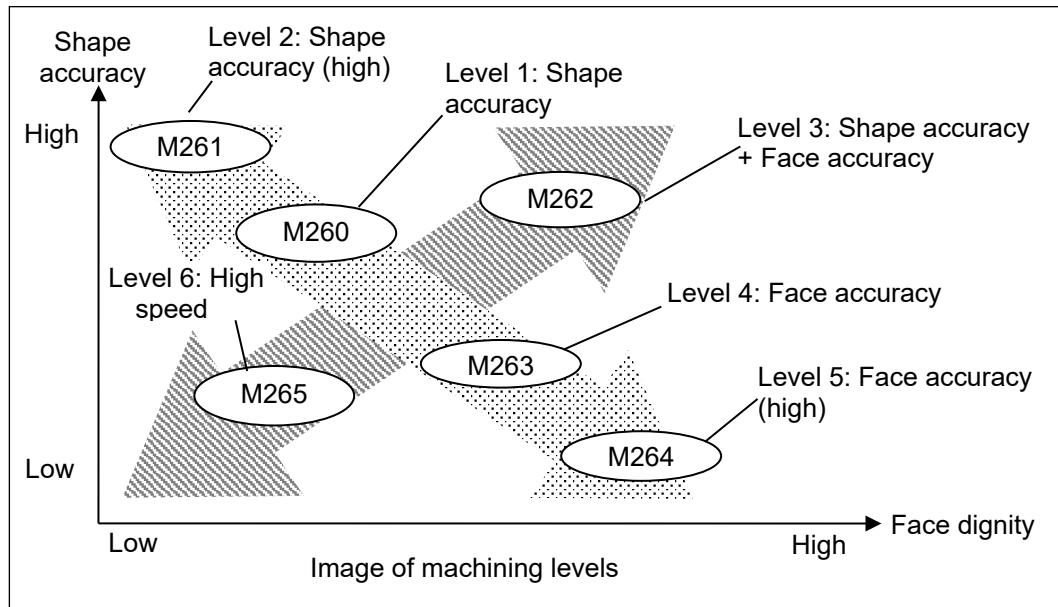
### 2.1 To Select a Machining Level

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)                                                                                                                                                                                                  | M260               |
| 2               | Shape accuracy (High)         | 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.                                                                                                            | 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<br>to<br>M267 |



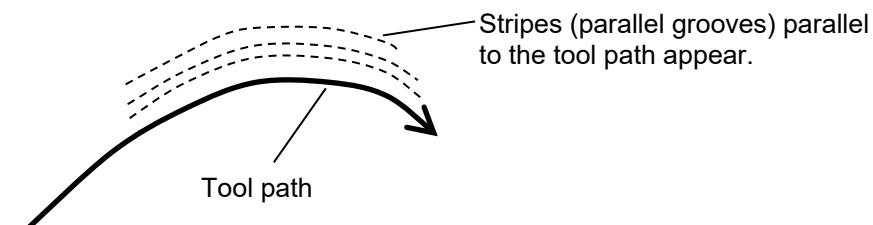
#### Points for selection of machining level

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.



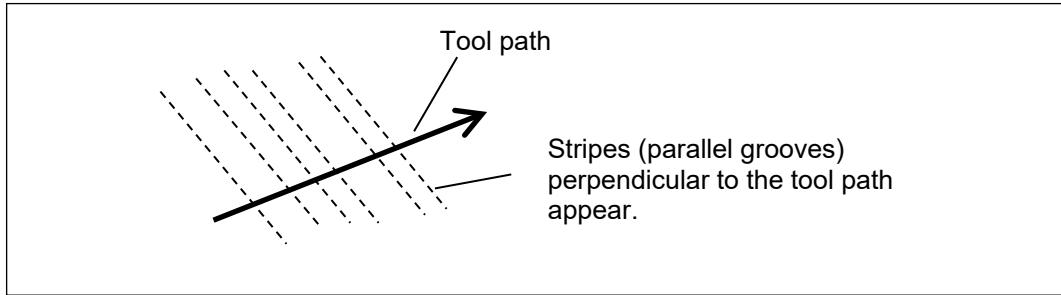
#### Presumed cause

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 gone 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; ← High-accuracy mode A (Level 1) ON
G01 X20. Y30. Z50.; } High-accuracy mode A (Level 1) in operation
X40. Y20. Z30.; ;
M269; ← High-accuracy mode A (Level 1) OFF
;
M261; ← High-accuracy mode A (Level 2) ON
G01 X20. Y30. Z50.; } High-accuracy mode A (Level 2) in operation
X40. Y20. Z30.; ;
M269; ← High-accuracy mode A (Level 2) OFF
;
M262; ← High-accuracy mode A (Level 3) ON
G01 X20. Y30. Z50.; } High-accuracy mode A (Level 3) in operation
X40. Y20. Z30.; ;
M269; ← High-accuracy mode OFF
M30;
```

## 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.

|                                                             |
|-------------------------------------------------------------|
| <b>M97: To cancel interruptive macro program</b>            |
| <b>M299: Cancel machining mode specification</b>            |
| <b>G43/G44/G143/G144/G49: TCP control (G43.4/G43.5) OFF</b> |

## 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. Except for TCP control (G43.4/G43.5), G code commands are possible 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) and TCP control (G43.4/G43.5), 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  
Ex.) M260G1X-10.Y-10.  
G1X-20.Y-20.  
M261G1X-30.Y-30...(1)  
G1X-40Y-40

When using a program similar to the one above, if the setting in the user parameter (high accuracy: high accuracy A) <Cutting feed time constant selection> for M260 and M261 is different, or if the setting for <Smooth override> is different, then the in-position check is performed before the program (1) runs.

## 3.4 Feedrate When Commanding of High-accuracy Mode A

The feedrate for a high accuracy mode A command is restricted by the machine parameter (high accuracy: X-, Y- and Z-axes) <Reference feedrate A>.

However, the user can adjust the restricting speed in the user parameter (high accuracy: high accuracy A) setting for <Smooth override>. (Refer to “4.1 Detailed description of functions” for further details.)

However, when a speed command is issued that is greater than the machine parameter (high accuracy: X-, Y- and Z-axes) <Maximum feedrate A>, the alarm <<Feedrate error>> is triggered. The machine parameter (system: X-, Y- and Z-axes) <X-(to Z-)axis max. cutting feedrate> and the user parameter (switch 1: programming) <Max. actual cutting travel speed (linear axis / rotation axis)> cannot be used in high accuracy mode.

## 3.5 Notes in Motions by Smooth Path Offset Function

- Operates only when the user parameter (high accuracy: common) <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 unchangeable. 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: common) <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: system) <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-dimentional 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:

- When the user parameter (high accuracy: high accuracy A) <Smooth override> is 100% or more, the limit value is equal to the product from the following equation: the machine parameter (high accuracy: X-, Y- and Z-axes) “Reference feedrate A” × “ $\sqrt{(\text{Smooth override} / 100)}$ ”. (Decimal point is rounded off)
- When the <Smooth override> is less than 100%, the limit value is equal to the product from the following equation: the machine parameter (high accuracy: X-, Y- and Z-axes) “Reference feedrate A” × “ $\sqrt{(\text{Smooth override} / 100)}$ ”. (Decimal point is rounded off)

If the smooth override value is set at 100% or more, the cutting feedrate can be raised.

However, if the restricted value exceeds the machine parameter (high accuracy: X-, Y- and Z-axes) <Maximum feedrate A>, it clamps to ensure it does not exceed <Maximum feedrate A>.

Ex.) 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

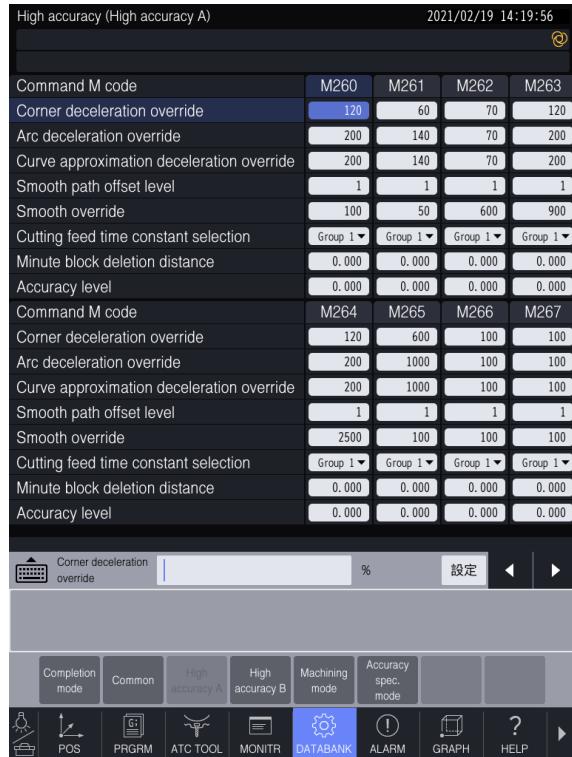
The setting area of the <Smooth path offset level> is “1-5” (The default value is “1”). If this value is made bigger, the path is offset by more command points to have much smoother path. However, if it is made big, the path may error.

(When the user parameter (high accuracy: common) <Smooth path offset function> is set to “Invalid”, this set value is invalid.)

## 4.2 Explanation About Parameters

Each level is controlled by referring to the setting values of the user parameter (high-accuracy).

(High accuracy A)



(High-accuracy common)



## Chapter 13 High-accuracy Mode

User parameter (high accuracy: high accuracy A)

| Parameter name                            | Contents                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                         | Setting range  |
|-------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------|
| Corner deceleration override              | <p>This is used together with the automatic corner deceleration function. 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.</p> <p>When the &lt;Accuracy level&gt; is set to another value besides "0", this parameter is disabled.</p>                                                     | 0, 10~9999 (%) |
| Arc deceleration override                 | <p>This is used together with the automatic arc deceleration function. 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.</p> <p>When the &lt;Accuracy level&gt; is set to another value besides "0", this parameter is disabled.</p>                                                              | 0, 10~9999 (%) |
| Curve approximation deceleration override | <p>This is used together with the automatic curve approximate deceleration function.</p> <p>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.</p> <p>When the &lt;Accuracy level&gt; is set to another value besides 0, this parameter is disabled.</p>           | 0, 10~9999 (%) |
| Smooth path offset level                  | <p>This is used when the &lt;Smooth path offset function&gt; (explained later) is set &lt;Valid&gt;. The larger it is set, the smoother it becomes.</p> <p>Making the set value larger may cause a significant deviation from the program path. Be extra careful when changing the value.</p>                                                                                                                                                                                                                                                                                                                                                    | 1~5            |
| Smooth override                           | <p>When "100" is set, the cutting feed time constant selected by the &lt;Cutting feed time constant selection&gt; is used.</p> <p>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.</p>                                                                                                                                                                                                                                                                                                                                                             | 10~9999 (%)    |
| Cutting feed time constant selection      | <p>The cutting feed time constant of the high-accuracy mode A is selected.</p> <p>&lt;0: Group 1&gt;<br/>Uses &lt;High accuracy A cutting feed time const. 1-3 (Group 1)&gt; in the machine parameter (high accuracy: X-, Y- and Z-axes).</p> <p>&lt;1: Group 2&gt;<br/>Uses &lt;High accuracy A cutting feed time const. 1-3 (Group 2)&gt; in the machine parameter (high accuracy: X-, Y- and Z-axes).</p> <p>&lt;2: Group 3&gt;<br/>Uses &lt;High accuracy A cutting feed time const. 1-3 (Group 3)&gt; in the machine parameter (high accuracy: X-, Y- and Z-axes).</p>                                                                      | 0~2            |
| Minute block deletion distance            | <p>If the travel distance of a block is less than the parameter value, the block is deleted.</p> <p>If the &lt;Smooth path offset function&gt; is &lt;Valid&gt;, it is performed at a command point after path approximation.</p>                                                                                                                                                                                                                                                                                                                                                                                                                | 0~2.000 (mm)   |
| Accuracy level                            | <p>Used with the fully automatic deceleration function.</p> <p>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: &lt;Corner deceleration override&gt;, &lt;Arc deceleration override&gt; and &lt;Curve approximation deceleration override&gt;.</p> <p>When set to 0, the fully automatic deceleration function is disabled and the following are enabled: &lt;Corner deceleration override&gt;, &lt;Arc deceleration override&gt; and &lt;Curve approximation deceleration override&gt;.</p> | 0~5.000 (mm)   |

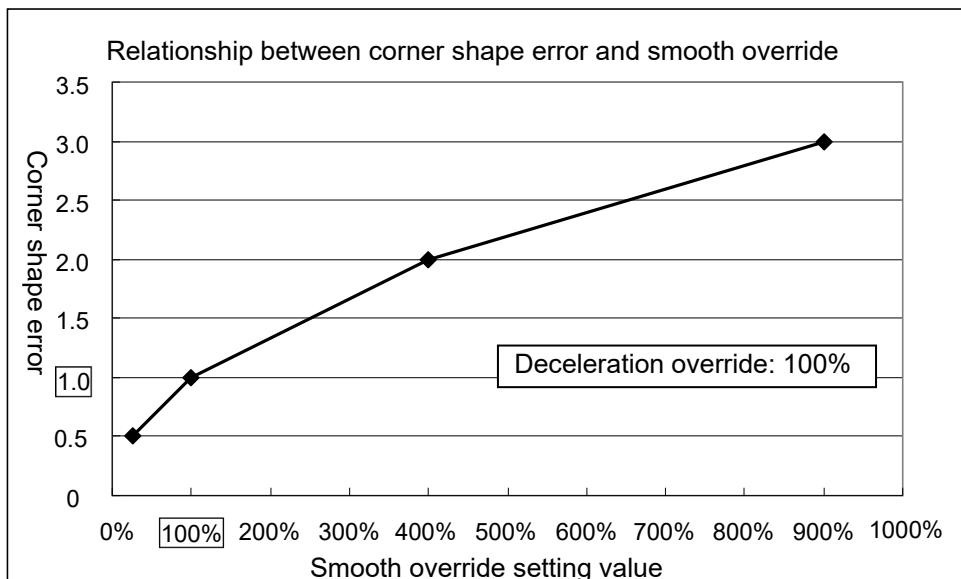
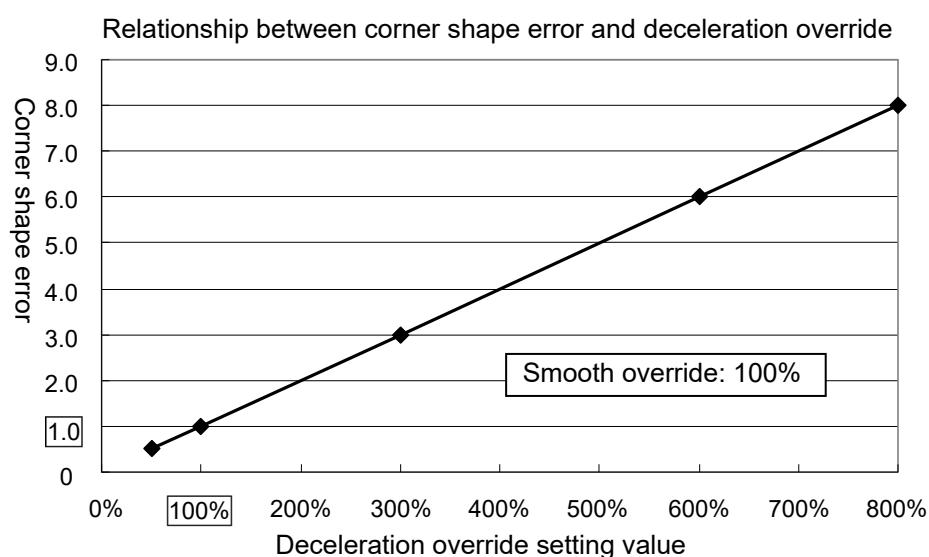
## User parameter (high accuracy: common)

| 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>. 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 (mm) |
| Smooth path offset cancel angle                            | This is used when the <Smooth path offset function> is set <Valid>. 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 (°) |
| Smooth path offset angle determination invalid distance    | This is used when the <Smooth path offset function> is set <Valid>. 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 (mm)  |
| Smooth path offset reference length                        | Used when <Smooth path offset function> is set to <Valid>. If the travel distance of one block is longer than the set value, the path is offset using the set value.                                                                                                                                                                          | 0~9.999 (mm)  |
| Minute block deletion when smooth path offset is cancelled | Used when the <Smooth path offset function> is set to <Valid>. 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. | 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

$$\text{Corner shape error} = E * D * \sqrt{S}$$

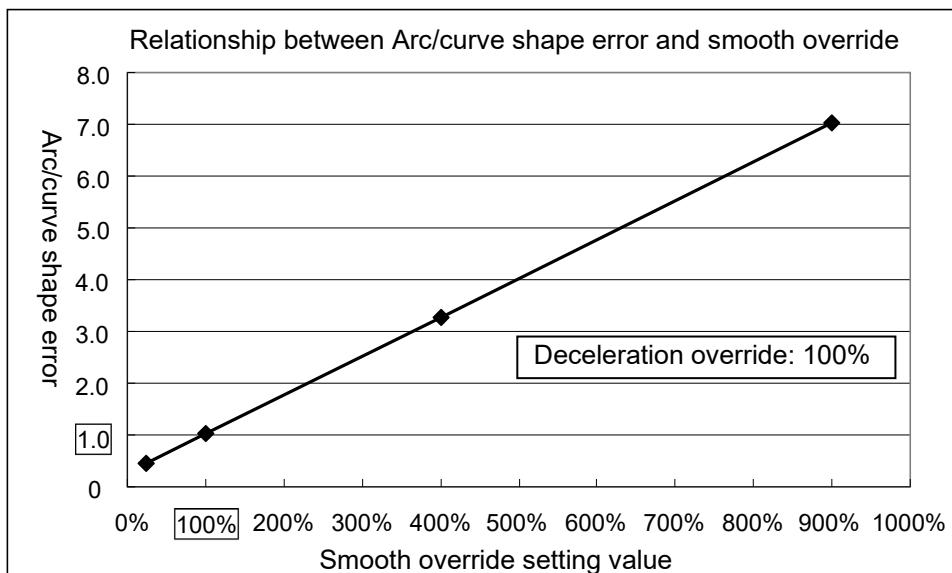
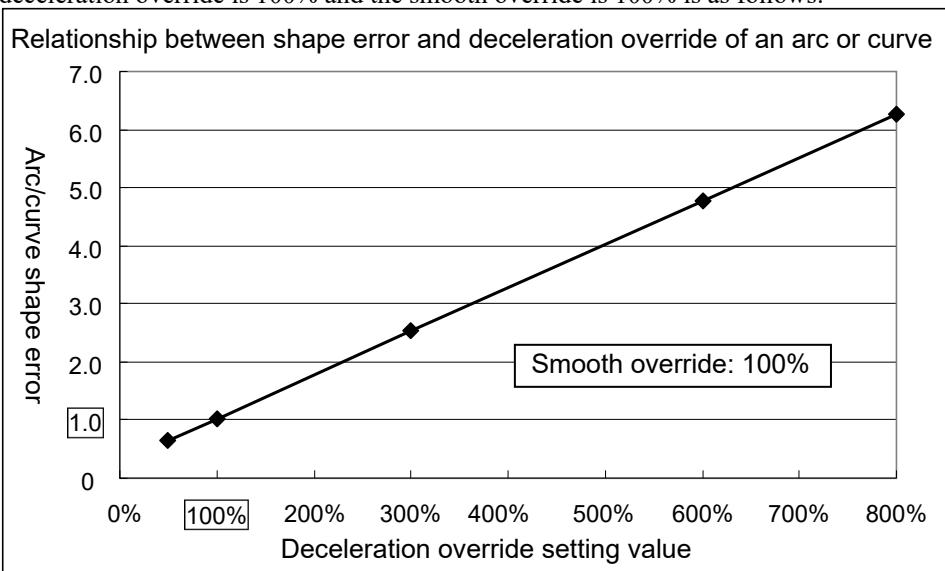
E : Error when the corner deceleration override is 100% and the smooth override is 100% (Guide amount: 100-120 μm)

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.

|               | <b>Parameter for adjusting shape accuracy</b>                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                       | <b>Smooth override</b>                                                                                                                                                                                                                                                                                                                           |
|---------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 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    | 160                                 | Machine parameter (high accuracy: common) <No. of look-ahead blocks B> |
| Smooth path offset function | Not available                       | Available                                                              |

In addition, when the machine parameter (high accuracy: common) <No. of look-ahead blocks B> is less than 160, 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:

|                                                  |
|--------------------------------------------------|
| <b>M97: To cancel interruptive macro program</b> |
| <b>M299: Cancel machining mode specification</b> |

In addition, this condition is applied when power is supplied.

- (NOTE 1) If the high-accuracy mode B is used when the above modal status is not applied, the alarm <<High-accuracy B unable modal>> appears.
- (NOTE 2) Commands are possible while under TCP control (G43.4/G43.5). However, refer to “14.2.6 Machining parameter” for further details on the functions and parameter content.

### 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 “3.2 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

When a speed command is issued that is greater than the machine parameter (high accuracy: X-, Y- and Z-axes) <Maximum feedrate B> and the machine parameter (high accuracy: additional axis) <Maximum feedrate B>, the alarm <<Feedrate error>> is triggered.

The machine parameter (system 1: X-, Y- and Z-axes) <Maximum cutting travel speed> (X- to Z-axes) and the user parameter (switch 1: programming) <Max. actual cutting travel speed (linear axis / rotation axis)> cannot be used in high accuracy mode B.

## 3.7 Notes in Smooth Path Offset Function

- Operates only when the user parameter (high accuracy: common) <Smooth path offset function> is set to “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 user parameter (high accuracy: common) <Smooth path offset cancel angle> is set too large, the alarm <<Curve speed error>> may be triggered.

## 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: common).

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

User parameter (high accuracy: high accuracy B)

| Parameter name                            | Contents                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                     | Setting range  |
|-------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------|
| Corner deceleration override              | <p>Used for automatic corner deceleration function.</p> <p>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.</p> <p>This parameter is disabled when the smooth override type is set to &lt;1: Type 2&gt;.</p>                                                                                                                                | 0, 10-9999 (%) |
| Arc deceleration override                 | <p>Used for automatic arc deceleration.</p> <p>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.</p>                                                                                                                                                                                                                                     | 0, 10-9999 (%) |
| Curve approximation deceleration override | <p>Used for automatic curve approximation deceleration.</p> <p>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.</p>                                                                                                                                                                                     | 0, 10-9999 (%) |
| Smooth path offset level                  | <p>Used when &lt;Smooth path offset function&gt; (described later) is set to &lt;Valid&gt;.</p> <p>The larger the value is, the smoother the path becomes.</p> <p>Making the set value larger may cause a significant deviation from the program path. Be extra careful when changing the value.</p>                                                                                                                                                                                                                                                                                                                                                                                         | 1-5            |
| Smooth override                           | <p>Used machine parameter (high accuracy) &lt;High accuracy B cutting feed time constant&gt; at a value of 100.</p> <p>If the value is larger than 100, the path becomes smoother.</p>                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                       | 100-9999 (%)   |
| Minute block deletion distance            | <p>If the travel distance of one block is less than the parameter value, the block is deleted.</p> <p>If &lt;Smooth path offset function&gt; is set to &lt;Valid&gt;, it is executed at a programmed point after path approximation.</p>                                                                                                                                                                                                                                                                                                                                                                                                                                                     | 0-2.000 (mm)   |
| Smooth override type                      | <p>Change the smooth override type.</p> <p>&lt;0: Type 1&gt;</p> <p>Even if the smooth override setting is changed, the speed difference that is permitted between blocks does not change.</p> <p>&lt;1: Type 2&gt;</p> <p>The speed difference allowed between blocks is equal to the "Speed difference between the machine parameter (high accuracy: X-, Y- and Z-axes) &lt;Speed difference B(min.)&gt; (X-, Y- and Z-axes) and &lt;Speed difference B(max.)&gt;" × "Smooth override (value)" / 100.</p> <p>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.</p> | 0-1            |

## User parameter (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>. 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 (mm) |
| Smooth path offset cancel angle                            | This is used when the <Smooth path offset function> is set <Valid>. 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 (°) |
| Smooth path offset angle determination invalid distance    | This is used when the <Smooth path offset function> is set <Valid>. 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 (mm)  |
| Smooth path offset reference length                        | Use when <Smooth path offset function> is set to <Valid>. If the travel distance of one block is longer than the set value, the path is offset using the set value.                                                                                                                                                                           | 0-9.999 (mm)  |
| Minute block deletion when smooth path offset is cancelled | Used when the <Smooth path offset function> is set to <Valid>. 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. | 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 14

## 5 AXES MACHINING FUNCTION

**14.1 Interpolation Using 5 Axes Simultaneously**

**14.2 TCP Control**

- \* The 5 axes machining function is available on the following machine models.

**S300Xd1-5AX/S500Xd1-5AX/S700Xd1-5AX**

**M200Xd1-5AX**

## 14.1 Interpolation Using 5 Axes Simultaneously

### 14.1.1 Overview

This chapter provides the specific programming specifications for interpolation using 5 axes simultaneously. When the interpolation function using 5 axes simultaneously is enabled, the G codes shown below are used for those specific motions.

- G01: Linear interpolation
- G02: Helical thread cutting interpolation
- G00: Positioning (NOTE)
- G43.4/G43.5: TCP control

(NOTE) The 5-axes simultaneous interpolation may not enable depending on the user parameter setting for (switch 1: programming) <Positioning method>. Refer to "14.1.4 Positioning (G00)" for further details.

### 14.1.2 Linear Interpolation (G01)

Linear travel is carried out at the command feedrate from the current position to the end point.

Command format

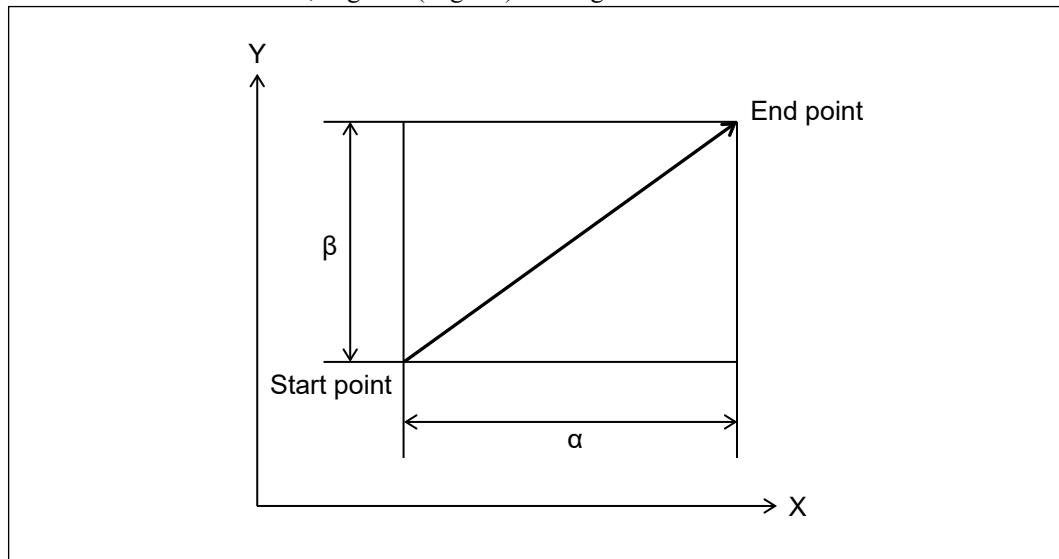
**G01 X\_Y\_Z\_A\_B\_C\_F\_;**

The axis command is valid for up to 3 linear travel axes + 2 additional axes.

If an additional axis command is issued when there is no additional axis option, the alarm <<No \*-axis option.>> is triggered.

The feedrate command is issued in F. The command is valid from the time it is issued until the feedrate is changed to another value.

When the command axis is the X-, Y- or Z-axis, mm/min (mm/rev) is recognized for the feedrate. When it is an additional axis, deg/min (deg/rev) is recognized for the feedrate.



The feedrate in the direction of each axis is as follows.

When using G01 G91 X $\alpha$  Y $\beta$  Z $\gamma$  Ff;

$$\text{Feedrate in X-axis direction: } F_x = \frac{\alpha}{L} \cdot f$$

$$\text{Feedrate in Y-axis direction: } F_y = \frac{\beta}{L} \cdot f$$

$$\text{Feedrate in Z-axis direction: } F_z = \frac{\gamma}{L} \cdot f$$

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

Linear interpolation using a linear axis and rotation axis is as follows.

When the user parameter (switch 1: programming) <Rotation axis speed command type> is set to <Type 2>, the speed of each axis changes depending on the user parameter (switch 1: programming) <Standard circle radius> setting. Refer to “3.3.1.1 Speed command for standard circle on rotation axis” for further details.

When using G01 G91 X $\alpha$  Y $\beta$  Z $\gamma$  B $\delta$  C $\varepsilon$  Ff;

$$\text{Time required for distributing between B-axis and C-axis: } T = \frac{L}{f}$$

$$\text{Feedrate in B-axis direction: } F_b = \frac{\delta}{T}$$

$$\text{Feedrate in C-axis direction: } F_c = \frac{\varepsilon}{T}$$

$$\text{Feedrate in X-axis direction: } F_x = \frac{\alpha}{L} \cdot f$$

$$\text{Feedrate in Y-axis direction: } F_y = \frac{\beta}{L} \cdot f$$

$$\text{Feedrate in Z-axis direction: } F_z = \frac{\gamma}{L} \cdot f$$

The L value is calculated based on the user parameter (switch 1: programming) <Rotation axis speed command type> settings below.

- <Type 1>  $L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \delta^2 + \varepsilon^2}$
- <Type 2>  $L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + (\delta \times \gamma \times \pi \div 180)^2 + (\varepsilon \times \gamma \times \pi \div 180)^2}$   
r: User parameter (switch 1: programming) <Standard circle radius>

(NOTE) When the <Rotation axis speed command type> is set to <Type 2> and the <Standard circle radius> is set to 0, the alarm <<User param. setting error (switch 1)>> is triggered.

### 14.1.3 Helical Thread Cutting Interpolation (G02)

When an axis command is inserted outside of the selected plane in an arc command block, helical thread cutting is carried out.

Command format

**For X-Y plane:**

**G17 G02 X\_Y\_Z\_- [ I\_J\_- ] (A\_B\_C\_) F\_;**  
**R\_-**

**G17 G03 X\_Y\_Z\_- [ I\_J\_- ] (A\_B\_C\_) F\_;**  
**R\_-**

**For Z-X plane:**

**G18 G02 Z\_X\_Y\_- [ K\_I\_- ] (A\_B\_C\_) F\_;**  
**R\_-**

**G18 G03 Z\_X\_Y\_- [ K\_I\_- ] (A\_B\_C\_) F\_;**  
**R\_-**

**For Y-Z plane:**

**G19 G02 Y\_Z\_X\_- [ J\_K\_- ] (A\_B\_C\_) F\_;**  
**R\_-**

**G19 G03 Y\_Z\_X\_- [ J\_K\_- ] (A\_B\_C\_) F\_;**  
**R\_-**

An axis command outside of the selected plane is valid for up to 1 linear travel axis + 1 additional axis. When only using additional axes, it is valid for up to 2 additional axes.

The feedrate command on a circular interpolation axis is issued in F.

When F is greater than the machine parameter (system 1: X, Y or Z-axis) <Maximum cutting travel speed> (X, Y or Z-axes) or <Rapid feedrate> (X, Y or Z-axis), the alarm <<Feedrate error>> is triggered.

The axis feed outside of the selected plane is determined by the values for “Feed”, “End point X”, “End point Y” and “End point Z” for the circular interpolation axis as shown in the following formula.

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

F : Command speed (Selected plane axis)

R : Radius

θ : Angle

Fz : Axis feedrate outside of selected plane

L : Axis travel distance outside of selected plane

Ex: When command speed = 500 (mm/min), radius = 10 (mm), angle = 360 (°), travel distance = 2 (mm),

$$F_z = (180 * 2 * 500) / (\pi * 10 * 360) \\ \approx 15.9 \text{ (mm/min).}$$

If the calculated feedrate for the axis outside of the selected plane is: greater than the machine parameter (system 1: X-, Y- and Z-axes) <Maximum cutting travel speed> or the <Rapid feedrate> (X-, Y- or Z-axes), or greater than the machine parameter (system 2: additional axis) <Maximum cutting rotation speed> (5th- to 8th-axes) or <Rapid feedrate> (5th- to 8th-axes), then the slowest speed is applied and that minimum speed on both the selected plane axis and the axis outside of the selected plane are fixed and used.

When issuing a cutter compensation command, cutter compensation is carried out for 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.

#### **14.1.4 Positioning (G00)**

Command format

**G00 X\_Y\_Z\_A\_B\_C\_;**

The axis command is valid for up to 3 linear travel axes + 2 additional axes.

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 (switch 1: programming) <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>

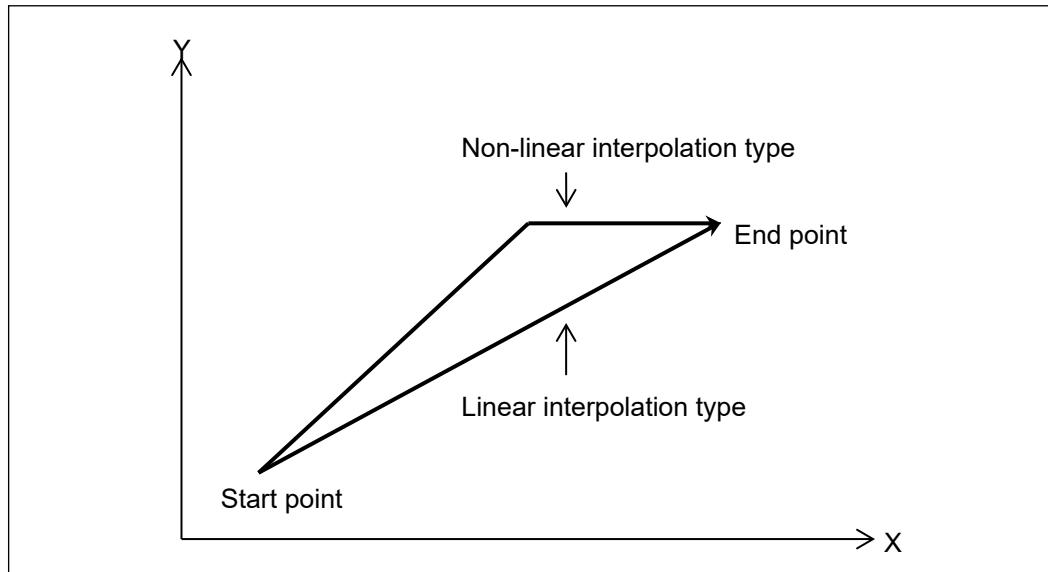
Positioning is carried out so the tool path is a straight line and is completed in the shortest time period at a speed that does not exceed the rapid feedrate on each axis.

<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.

However, if the user parameter (switch 1: programming) <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 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>.
- (NOTE 5) While the Z-axis perimeter mode is on (M300), the Z-axis perimeter operation is carried out when there are consecutive operations for the Z-axis up positioning and the X-axis or Y-axis positioning operations. Refer to “Chapter 12 M function” for details on the Z-axis perimeter operation.

### 14.1.5 TCP Control (G43.4/G43.5)

Refer to “14.2 TCP control” for further details.

### 14.1.6 Restrictions

The restrictions are shown below for the simultaneous interpolation command for 5 axes.

- When a command is issued in conversation language mode for 2 additional axes simultaneously in a job for a specified cutting axis, the alarm <<Interpolation command using 5 axes simultaneously is not possible>> is triggered.
- When the machine parameter (high accuracy: common) <High accuracy B additional axes> is set to <1:Valid> and a command is issued simultaneously for 2 additional axes using G01/G02, the alarm <<High-accuracy B unable command>> is triggered.

## 14.2 TCP Control

### 14.2.1 Overview

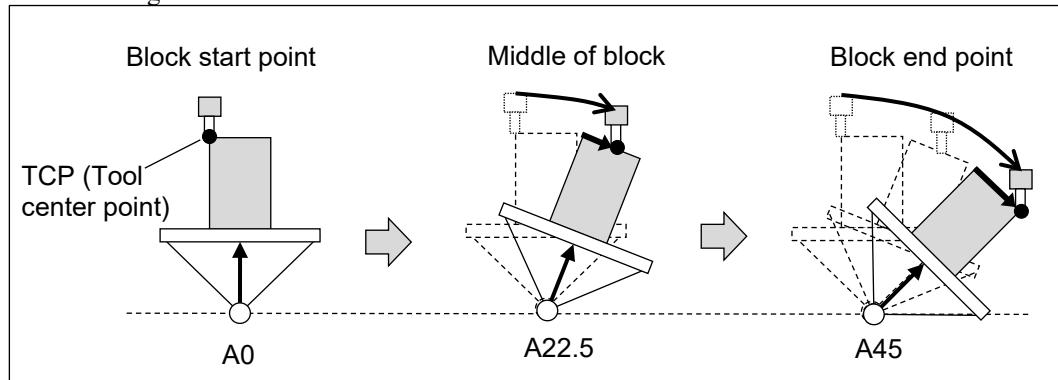
#### 14.2.1.1 Introduction

When using tool center point (TCP) control, the end point of the tool is controlled so that it always follows the program command path.

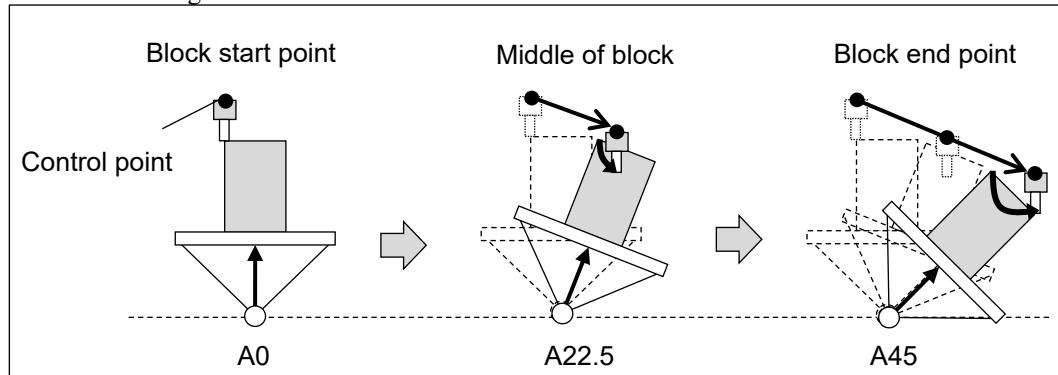
As shown in the images below, when there is an additional axis travel command on a block with linear interpolation, the TCP control keeps the path linear for the tool center point (when looking from the table).

When TCP control is not used, it maintains the linear path of the control point in the machine coordinate system. This prevents the TCP path from being linear, and it may cut in or leave something uncut in the middle of the block.

#### ■ When using TCP control



#### ■ When not using TCP control



In addition, when using the TCP control, the speed of the TCP (from the perspective of the workpiece) is controlled so that it reaches the command speed in the program (F command value). The user can directly issue commands in the program for the cutting speed (from the perspective of the workpiece).

When not using the TCP control, the speed of the control point (resultant speed of X-, Y- and Z-axes and additional axis) in the machine coordinate system is controlled so that it follows the command speed in the program (F command value). The cutting speed (from the perspective of the workpiece) will not reach command speed in the program.

#### 14.2.1.2 Prerequisites

This function can only be used on machine models equipped with the interpolation function using 5 axes simultaneously.

It can only be used in NC language mode. It cannot be used in conversation language mode.

### 14.2.1.3 Machine configuration

This function can be used on the following machine configurations.

- Configuration with two axes (rotation axis and tilt axis) for the table  
(When user parameters (rotation axis/tilt axis settings) <Tilt axis 1> and <Rotation axis 1> are not set to <0: Not specified>)
- Configuration with one axis (rotation axis or tilt axis) for the table  
(When user parameter (rotation axis/tilt axis settings) <Tilt axis 1> or <Rotation axis 1> is not set to <0: Not specified>)

In addition, this function is used only when the rotation direction of the rotation axis/tilt axis is set to one of the following.

#### Tilt axis

- Forward direction around X-axis rotates from the Z-axis forward direction to the Y-axis forward direction  
(When user parameters (rotation axis/tilt axis settings) <Forward direction for the coordinate system on tilt axis 1> is set to <1: Z→Y>)
- Forward direction around Y-axis rotates from the X-axis forward direction to the Z-axis forward direction  
(When user parameters (rotation axis/tilt axis settings) <Forward direction for the coordinate system on tilt axis 1> is set to <3: X→Z>)

#### Rotation axis

- Forward direction around Z-axis rotates from the Y-axis forward direction to the X-axis forward direction  
(When user parameters (rotation axis/tilt axis settings) <Forward direction for the coordinate system on rotation axis 1> is set to <5: Y→X>)

When the TCP control ON command is issued on another machine configuration with the rotation direction set, the alarm <<Rotation axis/Tilt axis parameter setting error>> is triggered.

## 14.2.2 Program Commands

### 14.2.2.1 TCP control ON command

ABC command format (G43.4)

The command is issued at the address for A, B or C for the coordinates of the tilt axis and rotation axis at the end point of each block (or the travel amount).

Command format

```
G43.4 X_ Y_ Z_ α_ β_ Hn ;
X_ Y_ Z_ α_ β_ ;
...
```

Hn : Tool number (n = 0 to 99 and 201 to 299), or group number (n = 901 to 930)

X\_ Y\_ Z\_ : End point coordinate (for G90 modal) or travel amount (for G91 modal)

α\_ β\_ : End point coordinate for tilt axis and rotation axis (for G90 modal)

Travel amount for tilt axis and rotation axis (for G91 modal)

α and β indicate the address “A”, “B”, or “C” for the tilt axis and rotation axis.

(NOTE 1) A command is issued for X, Y and Z in the coordinate system that is set in the user parameter (5 axes machining: common) <Programming coordinate system>. Refer to “14.2.3 Coordinate system” for further details.

(NOTE 2) X, Y, Z, α and β for the same block as G43.4 can be omitted. An omitted axis is processed as a no travel command. Refer to “14.2.4 Startup operation” for further details.

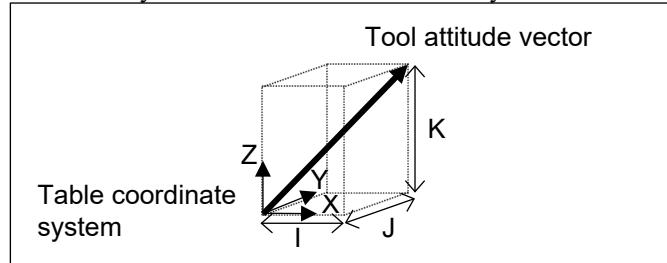
(NOTE 3) If a command is issued at an address that is not specified for the tilt axis/rotation axis in α and β, the alarm <<Address where command is not possible>> is triggered.

(NOTE 4) When “H0” is specified, the tool length is always 0. If “H” is omitted, the current H modal is used. However, if there is no H modal, it operates with a 0 tool length.

(NOTE 5) 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.

### IJK command format (G43.5)

A command for the orientation of the tool in the table coordinate system at the end point of each block is issued in a vector format by the IJK address. (This is called the “Tool posture vector”)  
To use this command format, set the user parameter (5 axes machining: common) <Programming coordinate system> to <0: Table coordinate system>.



Command format

```
G43.5 X_ Y_ Z_ α_ β_ Hn ;
X_ Y_ Z_ I_ J_ K_ ;
...
```

Hn : Tool number (n = 0 to 99 and 201 to 299), or group number (n = 901 to 930)

X\_ Y\_ Z\_ : End point coordinate (for G90 modal) or travel amount (for G91 modal)

I\_ J\_ K\_ : Tool posture vector (tool orientation at the end point of the block in the table coordinate system)

(NOTE 1) X, Y and Z commands are issued in the table coordinate system. Refer to “14.2.3 Coordinate system” for further details on the table coordinate system.

(NOTE 2) X, Y, and Z on the same block as G43.5 can be omitted. An omitted axis is processed as a no travel command. Refer to “14.2.4. Startup operation” for further details.

(NOTE 3) If the user parameter (5 axes machining: common) <Programming coordinate system> is set to <1: Wrkpc. coord. sys.>, then the alarm <<TCP control command error>> is triggered.

(NOTE 4) If the user parameter (rotation axis/tilt axis setting) <Tilt axis 1> or <Rotation axis 1> is set to <Not specified>, then the alarm <<TCP control command error>> is triggered.

(NOTE 5) On the G43.5 block, commands for I, J and K are not possible. If a command is issued, the alarm <<Address where command is not possible>> is triggered.

(NOTE 6) When one of the following is omitted: I, J or K, it is processed as a 0 value. When all are omitted, the tool posture in the previous block is maintained and the additional axis does not travel.

(NOTE 7) When “H0” is specified, the tool length is always 0. If “H” is omitted, the current H modal is used. However, if there is no H modal, it operates with a 0 tool length.

(NOTE 8) 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 9) If a rotation axis address command is issued while using the G43.5 modal, the alarm <<Address where command is not possible>> is triggered.

IJK address range/valid digits

Command range and valid digits for IJK address in IJK command format (G43.5) are as follows.

- Up to 12 digits for the integer and decimal digits can be used for commands (excluding symbols and decimal points)
 

Ex: K-1.12345678901 (1 integer digit + 11 decimal digits = 12 total digits) → All digits valid  
   K-1.123456789012 (1 integer digit + 12 decimal digits = 13 total digits) → <<Command data range error>>
- Up to 6 digits for integer digits can be used for commands
 

Ex: K-123456.123456 (6 integer digits + 6 decimal digits = 12 total digits) → All digits valid  
   K-1234567.12345 (7 integer digits + 5 decimal digits = 12 total digits) → <<Command data range error>>
- Number of valid decimal digits is the difference after subtracting integer digits from total of 12 digits (maximum of 11 digits)
 

Ex: K[-1000/6]→166.666666667 (9 decimal digits are valid because there are 3 integers)  
   K[-1/6]→0.16666666667 (11 decimal digits are valid because there is 1 integer)  
   K-.123456789012 (0 integer digits + 12 decimal digits = 12 total digits) → Up to the 11th decimal digit is valid

How angle is determined for IJK command

There are normally 2 sets of angles in the range between 0° and 360° for the combination of the tilt axis and rotation axis angles, which is calculated from the tool posture vector in IJK command format (G43.5). 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.

- (1) Travel amount of tilt axis is the small angle
- (2) Travel amount of rotation axis is the small angle when the travel amount in the positive and negative directions is the same for (1).
- (3) 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 (1) or (2).
- (4) 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 (1), (2) or (3).
- (5) Angle where the tilt axis travels in a positive direction when the set of angles is not decided by (1), (2), (3) or (4).

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: stroke) <\*-axis stroke control> is set to <1: Yes>, it selects an angle within the stroke range. In addition, it selects an angle where the axis can travel within the stroke range. The alarm <<TCP control 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.

How the selection is made as explained above is the same as the feature coordinate setting function.

Refer to “3.8.8.2 Command format - (2) How the travel angle on the additional axis is established” for specific examples.

### 14.2.2.2 Cancel command (G49)

The G49 command cancels the TCP control.

Command format

**G49 X\_ Y\_ Z\_ α\_ β\_ ;**

X\_ Y\_ Z\_ : End point coordinate (for G90 modal) or travel amount (for G91 modal)

α\_ β\_ : End point coordinate for tilt axis and rotation axis (for G90 modal)

Travel amount for tilt axis and rotation axis (for G91 modal)

α and β indicate the address “A”, “B”, or “C” for the tilt axis and rotation axis.

(NOTE 1) XYZ commands on the same block as G49 are issued in the workpiece coordinate system. Refer to “14.2.5 Cancel operation” for further details.

(NOTE 2) The tool length offset command (G43/G44) and the tool change command (G100) can also be cancelled. Refer to the “14.2.5.3 Cancel operation by other commands” for further details.

(NOTE 3) Before executing the end of program (M02 and M30), always cancel the TCP control. When executing without cancelling TCP control, the alarm <<TCP control did not turn OFF.>> is triggered.

### 14.2.2.3 Command conditions

Issue a TCP control ON command in the following situations.

If a TCP control ON command is carried out when one of the following does not apply, the alarm <<TCP control command not possible>> is triggered.

#### Command conditions

- G00 modal (positioning) or G01 modal (linear interpolation)
- G40 modal (cutter / nose R compensation cancel)
- G43/G44 modal (tool length offset) or G49 modal (cancel)
- G50 modal (cancel scaling)
- G50.1 modal (cancel mirror image)
- Zero point shift amount is not set in G52 one shot command (local coordinate) → (NOTE 1)
- G54.2P0 modal (cancel rotary fixture offset)
- G69 modal (cancel rotational transformation/feature coordinate manufacturing mode)
- G80 modal (canned cycle cancel mode)
- G94 modal (feedrate per minute)
- G97 modal (cancel constant peripheral speed control)
- M97 modal (cancel interrupt type macro)
- M141 modal (spindle selection)
- M269/M289 modal (high accuracy mode A/B OFF) or M280 to M287 modal (high accuracy mode B ON) →(NOTE 2)
- M299 modal (cancel machining mode specification)
- Additional axis position is set →(NOTE 3)
- Travel is complete for tool offset →(NOTE 4)
- LOCK signal is OFF for all axes (X-, Y-, Z-, rotation and tilt axes) that are used in TCP control

(NOTE 1) An alarm is triggered when the zero point shift amount is set in G52 (local coordinate). An alarm does not trigger when the zero point shift amount is set in G92 (workpiece coordinate system).

(NOTE 2) TCP control cannot be turned ON for modals M260 to M267 (high accuracy mode A ON).

(NOTE 3) After changing from the lathe spindle (M142) to the spindle (M141), the alarm <<TCP control command error>> is triggered if the following commands are not issued to the additional axis that is assigned as the lathe spindle.

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

- (NOTE 4) The alarm <<TCP control command error>> is triggered when travel is not complete for the tool offset as described below.
- When a Z-axis travel command is not carried out after the tool length offset command is executed without a Z-axis command (or tool position offset command is executed with an X-, Y- and Z-axes command) and the user parameter (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change> is set to <Type 2>.

### 14.2.2.4 Command functions

The functions available while using TCP control are as follows.

If a function command is issued that is unavailable, the alarm <<TCP under control>> is triggered.

#### Command functions

- Positioning (G00) / linear interpolation (G01) →(NOTE 1)
- Dwell (G04)
- Exact stop check (G09)
- Programmable data input (G10) →(NOTE 2)
- Skip (G31/G131/G132) →(NOTE 3)
- Exact stop mode (G61)
- Cutting mode (G64)
- Macro call (G65) and macro modal call (G66)
- Absolute / Incremental command (G90/G91)
- Programmable data input (high accuracy) (G210)
- S code command
- T code command
- M code command excluding unavailable command functions

(NOTE 1) Angle chamfering/corner R cannot be used. Otherwise, the alarm <<Address where command is not possible>> is triggered.

(NOTE 2) Input command is not possible for workpiece coordinate zero data (G10 L2/L20/L98/L99) and for tool length offset data (G10 L10/L11/L90Z/L91Z). Otherwise, the alarm <<TCP under control>> is triggered.

(NOTE 3) Skip command is only possible for X-, Y- and Z-axes commands. If an additional axis command is issued, the alarm <<Address where command is not possible>> is triggered.

#### Unavailable command functions

- Arc (G02/G03), helical/spiral (G02/G03) and involute interpolation (G02.2/G03.2)
- Circular cutting (G12/G13)
- Programmable stroke limit (G22)
- Reference position return (G28/G29/G30)
- Thread cutting (G33/G392)
- Coordinate calculation (G36 to G39)
- Cutter / nose R compensation (G41/G141)
- TCP control (G43.4/G43.5) →(NOTE 1)
- Scaling (G51) and mirror image (G51.1)
- Local coordinate system (G52)
- Machine coordinate command (G53)
- Feature coordinate index command (G53.1)
- Workpiece coordinate system selection (G54 to G59/G54.1) and rotary fixture offset (G54.2)
- Single direction positioning (G60)
- Rotational transformation (G68) and feature coordinate setting (G68.2)
- General canned cycle (G77, G81, etc.)
- Workpiece coordinate system setting (G92)
- Inverse time feed (G93) and feedrate per rotation (G95)
- M96 (interrupt type macro)
- M142 (lathe spindle selection)
- M203 (tool breakage detection)
- M260 to M267 (high accuracy mode A)
- M298 (high accuracy machining mode specification)
- M300 (Z-axis perimeter mode on) →(NOTE 3)
- M303/M304 (lathe spindle clockwise/counterclockwise)

- H code command

(NOTE 1) An alarm is triggered when a command is issued again while the TCP control is already ON.

(NOTE 2) TCP control can be turned ON with the M300 modal, but the Z-axis perimeter function is normally disabled.

Only typical functions are noted in this section.

Refer to “3.17 G code priority” for further details.

### 14.2.2.5 Feedrate

In the linear interpolation (G01), the TCP speed relative to the workpiece (TCP speed in the table coordinate system) is controlled by the “F” command feedrate.

Refer to “14.2.3 Coordinate system” for further details on the table coordinate system.

Refer to “14.2.2.6 Positioning” for details on the positioning (G00) operation.

- (NOTE 1) Only feedrate per minute (G94) commands are possible. An error is triggered when feedrate per rotation (G95) and inverse time feed (G93) commands are issued.
- (NOTE 2) When the user parameter (5 axes machining: common) <Programming coordinate system> is set to <1: Wrkpc. coord. sys.> as well, then the TCP speed relative to the workpiece is controlled so that it reaches the feedrate.
- (NOTE 3) The user parameter (switch 1: programming) <Rotation axis speed command type> is invalid.
- (NOTE 4) The user parameters (switch 1: programming) <Max. actual cutting travel speed (linear axis)> and <Max. actual cutting travel speed (rotation axis)> are invalid.

#### Command range

When a feedrate command is issued that exceeds the largest value below, then the alarm <<Feedrate error>> is triggered.

- Machine parameter (5 axes machining: X-, Y- and Z-axes) <Maximum speed (Cutting)> (X-axis) (mm/min)
- Machine parameter (5 axes machining: X-, Y- and Z-axes) <Maximum speed (Cutting)> (Y-axis) (mm/min)
- Machine parameter (5 axes machining: X-, Y- and Z-axes) <Maximum speed (Cutting)> (Z-axis) (mm/min)
- Machine parameter (5 axes machining: additional axis) <Maximum speed (Cutting)> ( $\alpha$ -axis)  $\times 360$  (deg/min)  
( $\alpha$  refers to the axis set in the user parameter (rotation axis/tilt axis settings) <Tilt axis 1>.)
- Machine parameter (5 axes machining: additional axis) <Maximum speed (Cutting)> ( $\beta$ -axis)  $\times 360$  (deg/min)  
( $\beta$  refers to the axis set in the user parameter (rotation axis/tilt axis settings) <Tilt axis 1>.)

#### When additional axis only travels

As shown below, when there is a command in which the TCP does not move in the table coordinate system, the user can select how to handle the feedrate for the additional axis. In the user parameter (5 axes machining: common) <Feedrate selection (when only rotation axis moves)>, select from <F command value> or <Maximum speed>.

- <F command value>: Controlled so the resultant speed of the rotation axis and tilt axis becomes the F command value (deg/min).
- <Maximum speed>: Controlled so that each axis speed for the rotation/tilt axis speed becomes the <Maximum speed (Cutting)> (5th- to 8th-axes) set in the machine parameter (5 axes machining: additional axis).

Example 1: Additional axis only command

(G43.4 modal)  
G91 G01 A10. C10. F10000

Example 2: Command with no travel on X-, Y- and Z-axes

(G43.4 modal, G90)  
G90 G01 X10. Y10. A10. C10. F10000  
X10. Y10. A20. C20.

### 14.2.2.6 Positioning

Positioning (G00) command operates with TCP control.

The speed of each axis in the machine coordinate system is controlled so that it becomes within the set value of the following parameters.

#### Speed on each axis

- Machine parameter (5 axes machining: X-, Y- and Z-axes) <Maximum speed (Rapid feed)> (X-axis) (mm/min)
- Machine parameter (5 axes machining: X-, Y- and Z-axes) <Maximum speed (Rapid feed)> (Y-axis) (mm/min)
- Machine parameter (5 axes machining: X-, Y- and Z-axes) <Maximum speed (Rapid feed)> (Z-axis) (mm/min)
- Machine parameter (5 axes machining: additional axis) <Maximum speed (Rapid feed)> ( $\alpha$ -axis)  $\times 360$  (deg/min)  
( $\alpha$  refers to the axis set in the user parameter (rotation axis/tilt axis settings) <Tilt axis 1>.)
- Machine parameter (5 axes machining: additional axis) <Maximum speed (Rapid feed)> ( $\beta$ -axis)  $\times 360$  (deg/min)  
( $\beta$  refers to the axis set in the user parameter (rotation axis/tilt axis settings) <Tilt axis 1>.)

(NOTE 1) The operation functions always with the TCP control ON regardless of the set value in the user parameter (switch 1: programming) <Positioning method>.

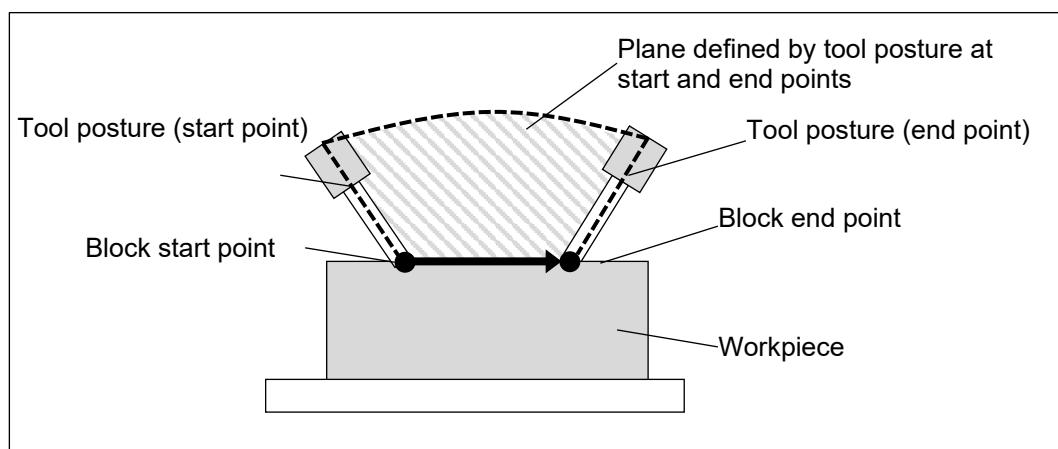
(NOTE 2) The same block as the TCP control ON command operates with the TCP control OFF. Refer to “14.2.4 Startup operation” for further details.

### 14.2.2.7 Interpolation method on additional axis

The respective angle speed for the tilt axis and rotation axis interpolates so that the speed remains constant throughout the block.

Tool posture at the start and end points in the block (orientation of the tool relative to the workpiece) follows the orientation according to the start and end points in the program, but orientation of the tool in the middle of the block is not controlled.

When the tilt axis and rotation axis operate at the same time, the tool posture in the middle of the block does not exist on the same plane as the tool posture at the start and end points on the block, because the two axes operate independently.



### 14.2.3 Coordinate System

#### 14.2.3.1 Programming coordinate system

A coordinate system used in the program (referred to as “programming coordinate system”) can be selected for TCP control. In the user parameter (5 axes machining: common) <Programming coordinate system>, select from <Table coordinate system> or <Wrkpc. coord. sys.>.

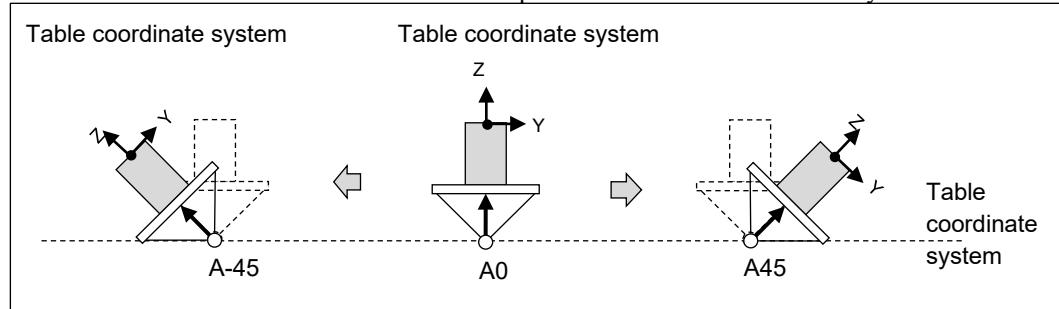
##### Table coordinate system

This coordinate system is fixed to the table.

The zero position in the coordinate system and the XYZ directions change according to the rotation of the table.

The workpiece coordinate system is fixed to the table and defines the table coordinate system.

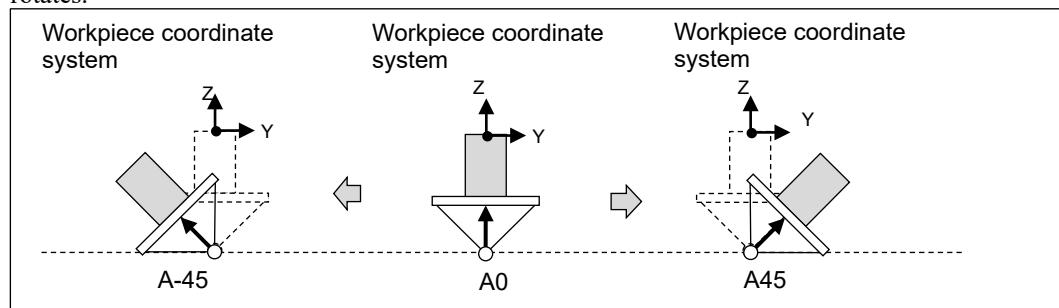
Refer to the next section for details about fixed position in the table coordinate system.



##### Workpiece coordinate system

This coordinate system is not fixed to the table.

The zero position in the coordinate system and the XYZ directions do not change even if the table rotates.



#### 14.2.3.2 Fixed position in table coordinate system

When the workpiece coordinate system is fixed to the table, the coordinates for the rotation axis and tilt axis can be selected.

In the user parameter (5 axes machining: common) <Fixed position in table coordinate system>, select from <Zero position in workpiece coordinates> or <Additional axis position at start>.

##### Zero position in workpiece coordinates

The workpiece coordinate system is fixed to the table for the zero position of the rotation axis/tilt axis that is set in the workpiece coordinate system.

##### Additional axis position at start

The workpiece coordinate system is fixed to the table for the coordinate position of the rotation axis/tilt axis when TCP control is ON.

If a travel command is issued for the rotation axis/tilt axis on the same block as the TCP control ON command, then the workpiece coordinate system is fixed to the table for the end position after rotation travel.

### 14.2.3.3 Operation example (additional axis at ON command is 0 degrees)

The operation example below is when the additional axis position is 0 degrees when TCP control is ON.

#### Assumed conditions

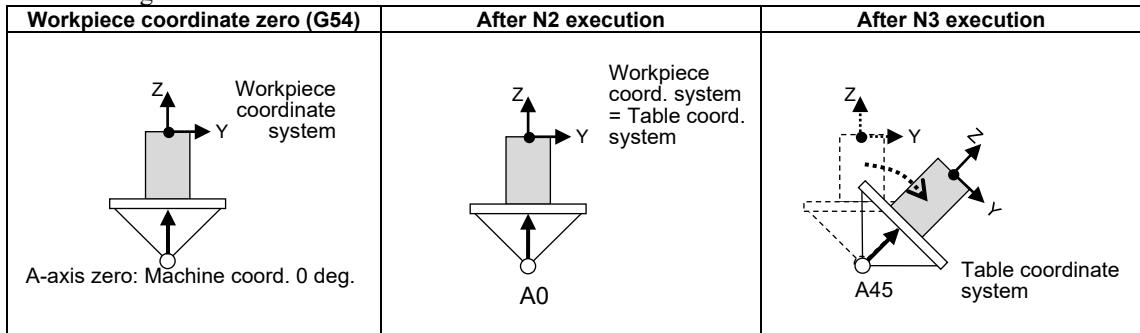
- A-axis: Tilt axis around the X-axis

#### Program

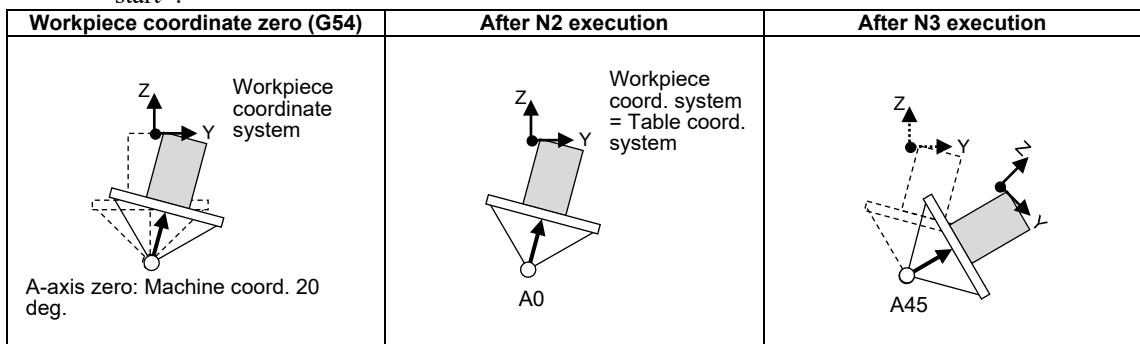
```
N1 G54 G90 X_Y_Z_A0. ;...A-axis default value: 0 degrees
N2 G43.4 Hn ;...TCP control ON
N3 X_Y_Z_A45. ; ...A-axis rotation: 45 degrees
...
```

#### Operation

- When <Zero position in workpiece coordinates> or <Additional axis position at start> applies. The workpiece coordinate system is fixed to the table with the A-axis machine coordinate at 0 degrees.



As shown below, even if the workpiece coordinate system zero point (G54) is set to another value besides 0 degrees, the workpiece coordinate system is fixed to the table at a position where the A-axis is 0 degrees in the workpiece coordinates (20 degrees in machine coordinates) for the <Zero position in workpiece coordinates> and <Additional axis position at start>.



### 14.2.3.4 Operation example (additional axis at ON command is not 0 degrees)

The operation example below is when the additional axis position is not 0 degrees when TCP control is ON.

#### Assumed conditions

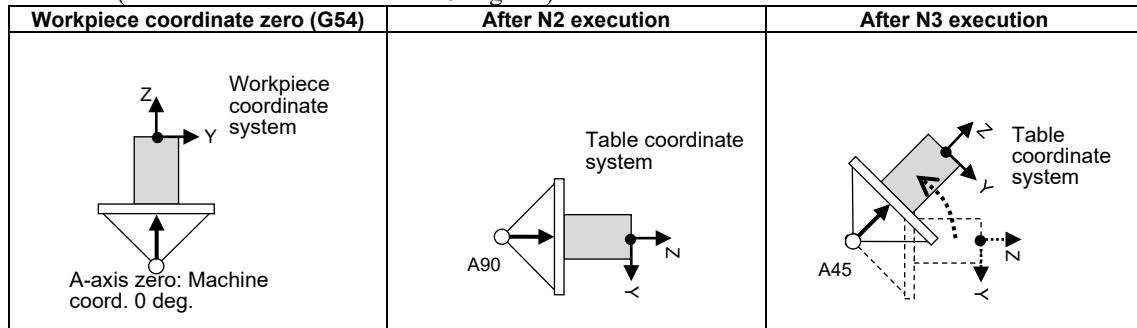
- A-axis: Tilt axis around the X-axis

#### Program

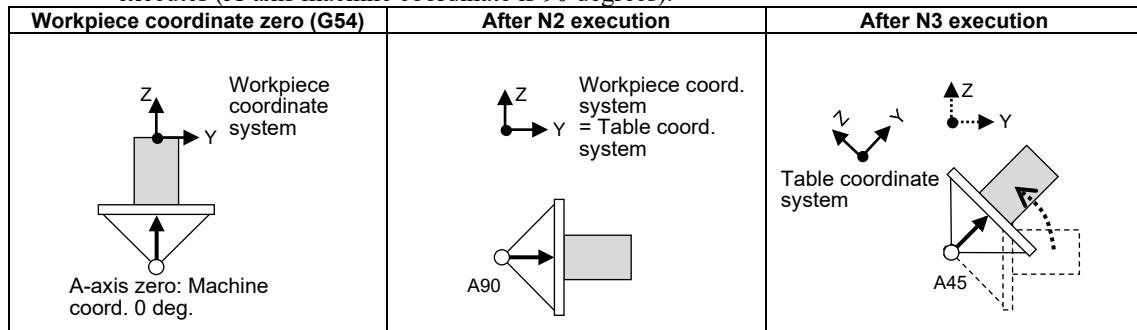
```
N1 G54 G90 X_Y_Z_A90. ;...A-axis default value: 90 degrees
N2 G43.4 Hn ; ...TCP control ON
N3 X_Y_Z_A45. ; ...After A-axis rotation: 45 degrees
...
```

#### Operation

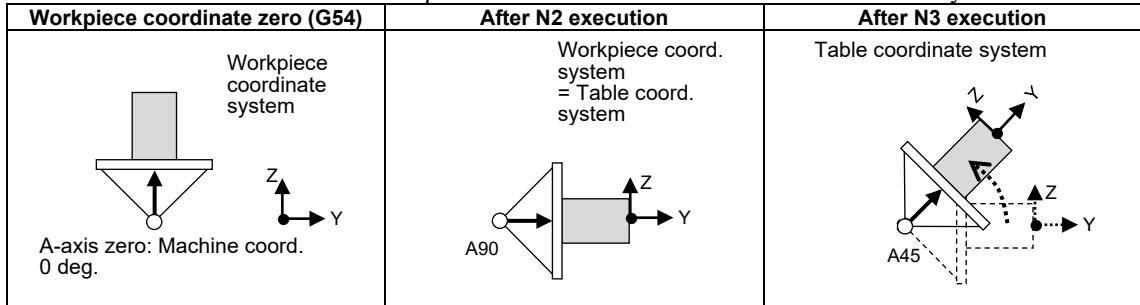
- When set to <Zero position in workpiece coordinates>  
The workpiece coordinate system is fixed to the table at the workpiece coordinate zero point (A-axis machine coordinate is 0 degrees).



- When set to <Additional axis position at start>  
The workpiece coordinate system is fixed to the table at the additional axis position when N2 executes (A-axis machine coordinate is 90 degrees).



As shown below, to fix the coordinate system at the top end of the workpiece, the zero point in the workpiece coordinate system must be set beforehand to the top end position of the workpiece at a position when the machine coordinate is 90 degrees. At this time, the upward direction relative to the workpiece becomes the Y direction in the coordinate system.



#### 14.2.4 Startup Operation

The tool length offset is applied from the TCP control ON command.  
The startup operation refers to the travel in this situation.

##### 14.2.4.1 No travel command

As shown below, if there is no axis travel command on the same block as the TCP control ON command, then the Z-axis just travels the tool length offset at the startup operation.

ABC command format

**G43.4Hn ;**

IJK command format

**G43.5Hn ;**

(NOTE 1) The startup operation includes the travel for TCP control OFF.

(NOTE 2) It travels only for the tool offset regardless of the set value in the user parameter (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change>.

The following is a program example.

##### Parameter setting

- User parameter (5 axes machining: common) <Programming coordinate system>: <Table coordinate system>
- Tool length offset H1: 80 mm

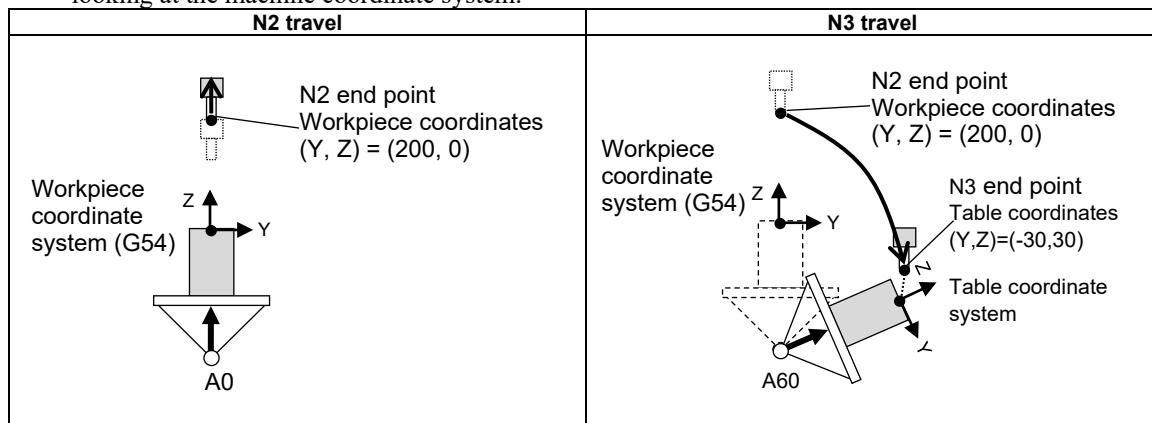
##### Program

```
N1 G90 G54 G00 Y0. Z200. A0. ;
N2 G43.4 H1 ; ... No axis travel command
N3 Y-30. Z30. A60. ;... X-, Y- and Z-axes command is issued in programming
coordinate system
...
```

#### Operation

On the N2 block, the Z-axis travels in the plus direction only for the tool length offset.

Starting from N3 block, it travels for the TCP control ON, and then travels along a curve when looking at the machine coordinate system.



#### 14.2.4.2 Travel command

As shown below, an axis travel command on the same block as the TCP control ON command can be carried out.

The startup operation includes travel to the position that is offset by the tool length relative to the end position.

ABC command format

**G43.4 X\_ Y\_ Z\_ α\_ β\_ Hn ;**

IJK command format

**G43.5 X\_ Y\_ Z\_ Hn ;**

Refer to “14.2.2.1 TCP control ON command” for details on the significance of each address.

- (NOTE 1) The startup operation includes the travel for TCP control OFF.
- (NOTE 2) When startup operation includes the positioning operation (G0 modal) and the user parameter (switch 1: programming) <Positioning method> is set to <Non-linear interpol.pos.>, the path changes depending on the [RAPID TRAVERSE OVERRIDE] switch status. Pay particular attention that there is no tool or jig interference.
- (NOTE 3) The end position of the X-, Y- and Z-axes on the same block as the G43.4/G43.5 follows the coordinate system (table coordinate system or workpiece coordinate system) that is set in the user parameter (5 axes machining: common) <Programming coordinate system>.
- (NOTE 4) The user parameter (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change> is invalid. Even if there is no Z-axis travel command, it always travels the tool offset amount.

The following is a program example.

#### Parameter setting

- User parameter (5 axes machining: common) <Programming coordinate system>: <Table coordinate system>
- Tool length offset H1: 80 mm

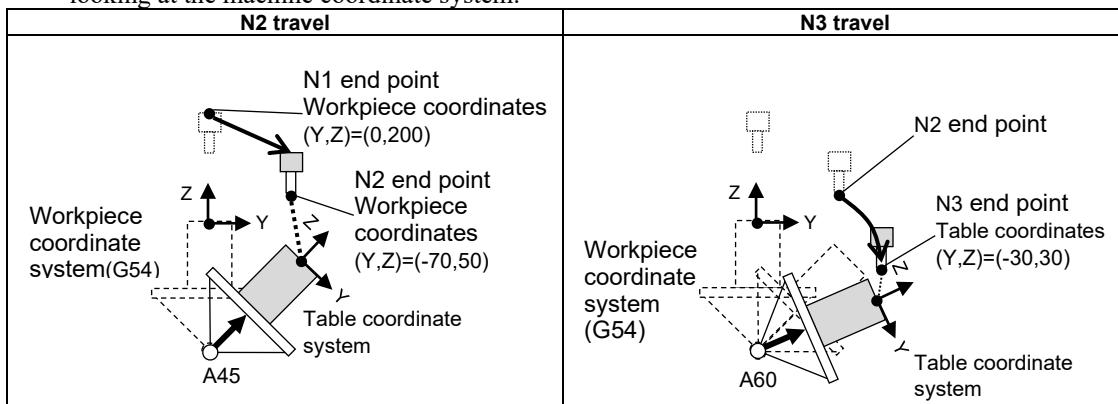
#### Program

```
N1 G90 G54 G00 Y0. Z200. A0. ;
N2 G43.4 Y-70. Z50. A45. H1 ; ...X-, Y- and Z-axes commands are issued in the table
coordinate system.
N3 Y-30. Z30. A60. ; ... X-, Y- and Z-axes commands are issued in the table
coordinate system.
...
```

#### Operation

On the N2 block, it travels for the TCP control OFF, and then travels along a linear path when looking at the machine coordinate system.

Starting from N3 block, it travels for the TCP control ON, and then travels along a curve when looking at the machine coordinate system.



### 14.2.4.3 Simultaneous Commands with Tool Change

As shown below, it is possible to issue commands on the same block as the tool change command and the TCP control ON command.

ABC command format

**G100 T\_ G43.4 X\_ Y\_ Z\_ α\_ β\_ Hn ;**

IJK command format

**G100 T\_ G43.5 X\_ Y\_ Z\_ Hn ;**

T: Tool number (1 to 99, 201 to 299), or pot number (magazine number) (101 to 199), or group number (901 to 930).

Refer to “14.2.2.1 TCP control ON command” for details on the significance of each address apart from those above.

(NOTE 1) The end position of the X-, Y- and Z-axes on the same block as the G100 follows the coordinate system (table coordinate system or workpiece coordinate system) that is set in the user parameter (5 axes machining: common) <Programming coordinate system>.

(NOTE 2) An alarm is triggered if a command is carried out that cannot be executed on the same block as G43.4/G43.5 or G100. Refer to “3.17 G code priority” for further details.

## Chapter 14 5 axes machining function

The operation is the same as the standard tool change canned cycle.  
Refer to the following operation example.

### Parameter setting

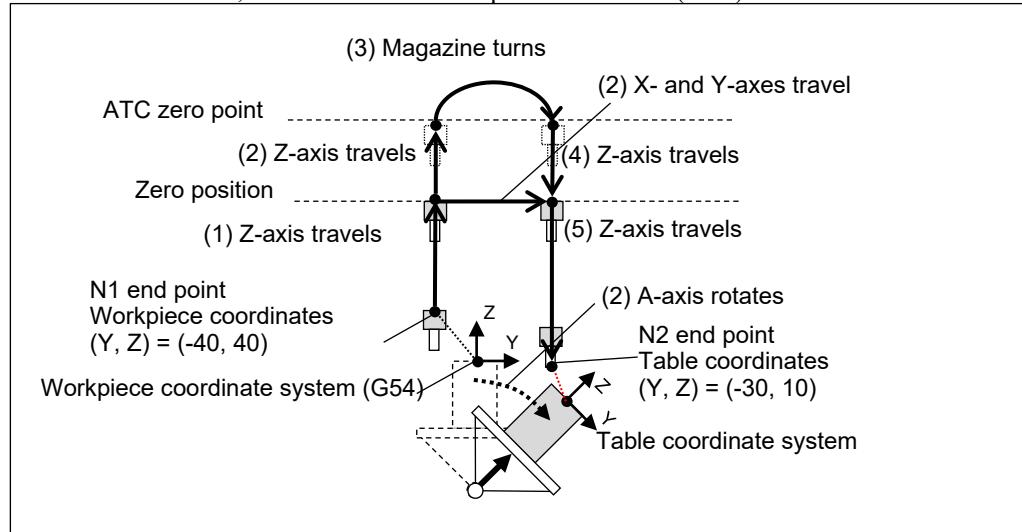
- User parameter (5 axes machining: common) <Programming coordinate system>: <Table coordinate system>  
Assumed conditions
- A-axis: Tilt axis around the X-axis
- Spindle mounted tool: T1

### Program command

```
N1 G54 G90 G0 Y-40. Z40.
N2 G100 G43.4 T2 X0. Y-30. Z10. A45. C0. H2 M03 S10000 ;
N3 X_ Y_ Z_ A_ C_ ;
...
```

### Tool change operation at N2

- (1) The Z-axis travels from the N1 end position (workpiece coordinate system: Z = 40) to the zero position (distance to zero point).  
At the same time, the spindle orientation is carried out.
- (2) The Z-axis travels from the zero position (distance to zero point) to the ATC zero point.  
At the same time, the X- and Y-axes travel to the end position (table coordinate system: X = 0, Y = 30) as specified in N2.  
In addition, the A- and C-axes travel to the end position (A = 45, C = 30) as specified in N2.
- (3) The magazine rotates and indexes the T2 tool.
- (4) The Z-axis travels from the ATC zero point to the zero position.
- (5) The Z-axis travels from the zero position to the end position (table coordinate system: Z = 10) as specified in N2.  
At the same time, rotation starts for the spindle command (M03).



Other specifications are the same as the standard tool change canned cycle.  
Refer to “5.7 Canned cycle for tool change (nonstop ATC) (G100)” for further details.

## 14.2.5 Cancel operation

The tool length offset is cancelled from the TCP control OFF command.  
Travel at this time is referred to as a “cancel operation”.

### 14.2.5.1 No travel command

As shown below, if only the TCP control OFF command is carried out, then the Z-axis just travels the tool length offset amount because of the cancel operation.

Command format

**G49 ;**

- (NOTE 1) The cancel operation includes the travel for TCP control OFF.
- (NOTE 2) When the user parameter (switch 1: compensation function) <Error check when traveling during tool length/tool position offset cancel> is set to <Yes>, an error is triggered.
- (NOTE 3) It travels only for the tool offset regardless of the set value in the user parameter (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change>.
- (NOTE 4) Only the Z-axis operates in the cancel operation, but if the lock signal for the X- and Y-axes is ON, then the alarm <<Lock signal for axis with TCP control is on.>> may trigger. In this situation, set the lock signal so it turns on in the next block for the cancel operation.

### 14.2.5.2 Travel command

As shown below, an axis travel command on the same block as the TCP control OFF command can be carried out.

The cancel operation includes travel to the position where the tool length offset (relative to the end position) is cancelled.

Command format

**G49 X\_ Y\_ Z\_ α\_ β\_ ;**

- (NOTE 1) The cancel operation includes the travel for TCP control OFF.
- (NOTE 2) When cancel operation includes the positioning operation (G0 modal) and the user parameter (switch 1: programming) <Positioning method> is set to <Non-linear interpol.pos.>, the path changes depending on the [RAPID TRAVERSE OVERRIDE] switch status. Pay particular attention that there is no tool or jig interference.
- (NOTE 3) The command position of X-, Y- and Z-axes on the same block as G49 is specified in the workpiece coordinate system.
- (NOTE 4) When using an IJK command format, a tool posture (IJK address) command is not possible on a G49 block. An ABC address command is possible for the tilt axis/rotation axis.
- (NOTE 5) When one of the following commands: M260 to M267 (high accuracy mode A ON), M280 to M287 (high accuracy mode B ON) or M298 (machining mode specification) is issued on the same block on G49, it becomes valid starting on the next block. OFF travel is carried out for high accuracy mode and machining mode on the same block as G49.
- (NOTE 6) When the Z-axis travel command is omitted and when the user parameter (switch 1: compensation function) <Error check when traveling during tool length/tool position offset cancel> is set to <Yes>, an error is triggered.
- (NOTE 7) The user parameter (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change> is invalid. Even if there is no Z-axis travel command, it always travels the tool offset amount.

### 14.2.5.3 Cancelled by other commands

When one of the following commands that is not the cancel command (G49) is carried out during TCP control, then the TCP control is cancelled.

#### Tool length offset

Command format

**G43/G44 X\_ Y\_ Z\_  $\alpha$ \_  $\beta$ \_ ;**

TCP control is cancelled and the tool length offset turns ON.

It travels to the position that is offset by the tool length regardless if there is a Z-axis travel command or not.

- (NOTE) When there is a command that does not include a Z-axis travel command and the user parameter (switch 1: compensation function) <X/Y/Z-axis travel at tool length/tool pos. offset change> is set to <Type 2>, be careful because it behaves differently as shown below.
- When under TCP control, it travels to the position that is offset by the tool length.
  - When not under TCP control, the Z-axis does not travel.

#### Tool change command

Command format

**G100 Tn X\_ Y\_ Z\_  $\alpha$ \_  $\beta$ \_ ;**

TCP control is cancelled and a tool change is carried out.

After the tool change, the TCP control stays cancelled.

## 14.2.6 Machining Parameter

### 14.2.6.1 Overview of acceleration/deceleration control

When under TCP control, it executes the program look ahead operation and carries out acceleration/deceleration before interpolation.

When there is a continuous cutting operation (G01) as per the program look ahead, acceleration/deceleration can be carried out smoothly without stopping at each block.

- (NOTE 1) The set value in the machine parameter (5 axes machining: common) <No. of look-ahead blocks> is valid for the number look ahead blocks.
- (NOTE 2) The acceleration/deceleration is carried out without any stops even if there are heavy travel blocks, empty lines and/or comment lines.
- (NOTE 3) The acceleration/deceleration before interpolation is disabled for the following operations, and the time constant after interpolation is used to carry out acceleration/deceleration.
- Axis travel on the same block as the TCP control ON command
  - Axis travel on the same block as the TCP control cancel command
  - Axis travel for the skip command
  - Axis travel after restoring from a manual intervention operation
  - Axis travel when the dry run key is ON

### 14.2.6.2 Stop command

If one of the following commands is carried out, it stops first at the end point of the previous block before it is executed.

- Rapid feed (G00)
- Skip (G31/G131/G132)
- Dwell (G04)
- Programmable data input (G10)
- Programmable data input (high accuracy) (G210)
- S code command
- T code command
- M code command
- Additional axis travel command under unclamp/clamp automatic control (NOTE 1)

(NOTE 1) Before the additional axis travel command, it first stops and then automatically unclamps. In addition, after the additional axis travel is complete, it always stops first and then automatically clamps. To keep it from stopping at every block, issue a command M442/M440/M444 (A-/B-/C-axis unclamp) beforehand and turn OFF the automatic control as shown below.

```
N1 M442 M444 ;...A-axis/C-axis unclamp
N2 G43.4 Hn X_ Y_ Z_ A_ C_ ;...TCP control ON
N3 G01 X_ Y_ Z_ A_ C_ ;
N4 G01 X_ Y_ Z_ A_ C_ ; A- and C-axes stay unclamped
N5 G01 X_ Y_ Z_ A_ C_ ; (Stops at each block)
...
```

### 14.2.6.3 Override functions

There are three override functions to adjust the machining parameters.

- Corner deceleration override
- Acceleration override
- Smooth override

The set values for each override function can be changed in the user parameters.  
Refer to “14.2.6.6 Parameter adjustment screen” for further details.

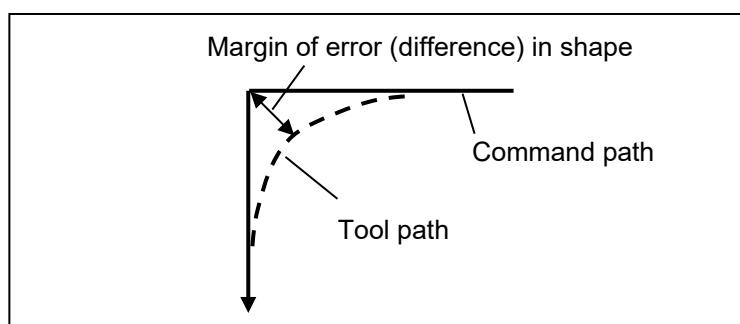
#### Corner deceleration override

This function limits any differences between the corner command path and the tool path in order to improve the shape accuracy on the corner.

The X-, Y-, Z-, tilt and rotation axes can be adjusted with the <Corner deceleration override(\*axis)>. The setting range can be set to “0%” or between “10% and 999%”.

The smaller the set value, the more the deceleration function is applied, making the difference between the paths smaller and improving the shape accuracy.

However, the machining time takes longer as a result.



### Acceleration override

When the acceleration is gradual after decelerating on the corner or curve section, the surface quality improves.

The X-, Y-, Z-, tilt and rotation axes can be adjusted with the <Acceleration override(\*axis)>. The setting range can be set between “10% and 100%”.

The smaller the set value, the more gradual the acceleration becomes, making the surface quality better.

However, the machining time takes longer as a result.

### Smooth override (cutting time constant adjustment)

The cutting time constant can be adjusted.

The common value on all axes can be set using <Smooth override>. The setting range can be set between “10% and 999%”.

When there is a sudden speed change on a corner or curve section that adversely affects the surface quality, the user can increase the smooth override to improve the surface quality.

However, the difference between the command path and tool path becomes larger.

When the machining level (M280 to M287) is not selected, all become 100 (%). Refer to the next chapter for details on selecting the machining level.

These override functions are enabled for the cutting feed. They do not affect the positioning operation.

### **14.2.6.4 Machining level change (M280 to M287)**

The machining level can be changed using the M code.

### **14.2.6.5 Program usage**

M codes are not needed when not changing the machining level.

When the TCP control ON (G43.4 or G43.5) command is issued, it operates using the factory default value for the machining level.

```
(Program example 1)
G00 X_ Y_ Z_;
G01 X_ Y_ Z_;
X_ Y_ Z_; } High accuracy mode OFF
...
G00 X_ Y_ Z_;
G43.4 H1; ← TCP control ON
...
G01 X_ Y_ Z_ A_ C_; } TCP control ON (factory default value) executing
X_ Y_ Z_ A_ C_; ...
G0 X_ Y_ Z_ A_ C_;
G49; ← TCP control OFF
M30;
```

To use a machining level between 1 and 8, insert an M code (M280 to M287) to change the desired machining level.

|                            |                               |                                          |
|----------------------------|-------------------------------|------------------------------------------|
| (Program example 2)        |                               |                                          |
| G00 X_Y_Z_;                |                               |                                          |
| M281;                    ← | Machining level (level 2) ON  |                                          |
| G01 X_Y_Z_;                | }                             | High accuracy mode B (level 2) executing |
| X_Y_Z_;                    |                               |                                          |
| ...                        |                               |                                          |
| G00 X_Y_Z_;                |                               |                                          |
| G43.4 H1;              ←   | TCP control ON                |                                          |
| ...                        |                               |                                          |
| M282;                    ← | Machining level (level 3) ON  |                                          |
| G01 X_Y_Z_A_C_;            | }                             | TCP control ON (level 3) executing       |
| X_Y_Z_A_C_;                |                               |                                          |
| ...                        |                               |                                          |
| G0 X_Y_Z_A_C_;             |                               |                                          |
| M283;                    ← | Machining level (level 4) ON  |                                          |
| G01 X_Y_Z_A_C_;            | }                             | TCP control ON (level 4) executing       |
| X_Y_Z_A_C_;                |                               |                                          |
| ...                        |                               |                                          |
| G0 X_Y_Z_A_C_;             |                               |                                          |
| G49 Z_;                    | ←                             | TCP control OFF                          |
| ...                        |                               |                                          |
| M289;                    ← | Machining level selection OFF |                                          |
| G01 X_Y_Z_;                | }                             | High accuracy mode B OFF executing       |
| X_Y_Z_;                    |                               |                                          |
| ...                        |                               |                                          |
| M30;                       |                               |                                          |

- (NOTE 1) Machining level selection for M280 to M287 is valid regardless if the TCP control is ON or OFF. When the TCP control is OFF, the machining level selection is used for high accuracy mode B. Refer to “Chapter 13 (2) High accuracy mode B” for further details.
- (NOTE 2) When one of the following operations is performed, the machining level selection is turned OFF.
  - 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
- (NOTE 3) Machining level selection for M260 to M267 is valid only when the TCP control is OFF. Refer to “Chapter 13 (1) High accuracy mode AIII” for further details. When the TCP control ON command is executed in a modal between M260 and M267, the alarm <<TCP control command error>> is triggered.
- (NOTE 4) The machining mode, selected on the <Machining parameter adjustment (Machining mode)> screen or on the <Machining parameter adjustment (Machining mode)> screen using the machining mode setting function, is valid only when the TCP control is OFF. Refer to “6.2.5 Machining mode settings” in Operation Manual II for further details.
- (NOTE 5) When there is a TCP control ON command and the mode is selected by a program command using the machining mode setting function, the alarm <<TCP control command error>> is triggered.

#### **14.2.6.6 Parameter adjustment screen**

The set value can be adjusted with an M code.

Select <5 axes machining> on the <User parameter menu> screen.

Set on the <5 axes machining (by M code)> screen below.

| 5th-axis machining (By M code)                                                                                            |                    | 2022/06/10 08:47:36                                                                  |           |        |          |
|---------------------------------------------------------------------------------------------------------------------------|--------------------|--------------------------------------------------------------------------------------|-----------|--------|----------|
|                                                                                                                           |                    | M280                                                                                 | M281      | M282   | M283     |
| Command M code                                                                                                            |                    |                                                                                      |           |        |          |
| Corner deceleration override(XYZ-ax)                                                                                      |                    |                                                                                      |           |        |          |
| Corner deceleration override(Tilt axis)                                                                                   |                    |                                                                                      |           |        |          |
| Corner deceleration override(Rotation axis)                                                                               |                    |                                                                                      |           |        |          |
| Acceleration override (XYZ-ax)                                                                                            |                    |                                                                                      |           |        |          |
| Acceleration override (Tilt axis)                                                                                         |                    |                                                                                      |           |        |          |
| Acceleration override (rotation axis)                                                                                     |                    |                                                                                      |           |        |          |
| Smooth override                                                                                                           |                    |                                                                                      |           |        |          |
|                                                                                                                           |                    |                                                                                      |           |        |          |
| Command M code                                                                                                            |                    | M284                                                                                 | M285      | M286   | M287     |
| Corner deceleration override(XYZ-ax)                                                                                      |                    |                                                                                      |           |        |          |
| Corner deceleration override(Tilt axis)                                                                                   |                    |                                                                                      |           |        |          |
| Corner deceleration override(Rotation axis)                                                                               |                    |                                                                                      |           |        |          |
| Acceleration override (XYZ-ax)                                                                                            |                    |                                                                                      |           |        |          |
| Acceleration override (Tilt axis)                                                                                         |                    |                                                                                      |           |        |          |
| Acceleration override (rotation axis)                                                                                     |                    |                                                                                      |           |        |          |
| Smooth override                                                                                                           |                    |                                                                                      |           |        |          |
|                                                                                                                           |                    |                                                                                      |           |        |          |
|  Corner deceleration<br>override(XYZ-ax) |                    |                                                                                      | %         | ENT    |          |
|                                                                                                                           |                    |                                                                                      |           |        | 1/1      |
|                                          | Completion<br>mode |     | By M code |        |          |
|                                          | Common             |     |           |        |          |
|                                          |                    |     |           |        |          |
|                                          |                    |     |           |        |          |
|                                          |                    |    |           |        |          |
|                                        | POS                |   | ATC TOOL  | MONITR | DATABASE |
|                                        |                    |   |           |        |          |
|                                        |                    |   |           |        |          |
|                                        |                    |   |           |        |          |
|                                        |                    |  |           |        |          |
|                                      |                    |                                                                                      |           |        |          |

#### **14.2.6.7 Programmable data input (G210)**

The parameters can be temporarily changed in the program.

Refer to “3.18.3 Temporary parameter change with TCP control” for further details.

## 14.2.7 Other

### 14.2.7.1 Operation pause

When the [FEED HOLD] switch is used to stop operation while under TCP control, operation may not stop on the block when the switch is pressed, because it decelerates gradually within the allowable acceleration.

(NOTE) The execution pointer (“>>”) on the program operation screen refers to the block where operation was actually stopped.

### 14.2.7.2 Emergency stop

When the [EMERGENCY] switch is used to stop operation while under TCP control, operation may not stop on the block when the switch is pressed, because it decelerates gradually within the allowable acceleration.

(NOTE) The execution pointer (“>>”) on the program operation screen refers to the block that was executing when the switch was pressed. The block may be different from the block where operation was actually stopped.

### 14.2.7.3 Single operation

When starting single operation while under TCP control, it may not stop on the block where the [SINGL] key is pressed for the same reason noted in “14.2.7.1 Operation pause”.

### 14.2.7.4 Cutting override

When using the [FEEDRATE OVERRIDE] switch, the override (0-200%) is applied to the feedrate (TCP speed relative to the workpiece) specified in “F”.

### 14.2.7.5 Rapid traverse override

When the [RAPID TRAVERSE OVERRIDE] switch is used to change between speeds 1 to 4, the speed on each axis is restricted so that the X-, Y-, Z- and additional axes stay within the values set in the user parameters (switch 2: override speed) <Rapid traverse override speed 1 - 4> and <\*-axis override rotation speed 1 - 4> (\* refers to axis between 5th- and 8th-axis).

(NOTE) When the switch is set to 100%, the speed on each axis is restricted so that the X-, Y-, Z- and additional axes stay within the values set in the machine parameter (5 axes machining: X-, Y- and Z-axes) <Maximum speed(Rapid feed)> and machine parameter (5 axes machining: additional axis) <Maximum speed(Rapid feed)>.

### 14.2.7.6 MDI operation

TCP control (G43.4/G43.5) command is not possible while in MDI operation mode. The alarm <<Specified G code cannot be used.>> is triggered when an attempt is made to issue a command. MDI intervention is not possible while under TCP control. The operator message <<TCP under control>> appears when an attempt is made to change the mode.

### 14.2.7.7 Manual operation

When operation is stopped while under TCP control by a command or an operation below while in memory operation mode, one of the following intervention operations can be carried out by pressing the [MANU] key and changing to manual operation.

- When paused pressing the [FEED HOLD] switch.
- When the [SINGL] key is pressed to perform a block stop.
- When an M00/M01 command is used to perform a block stop.

In addition, when the mode is changed to manual operation mode without performing one of the above stop operations, memory operation pauses and an intervention operation below can be carried out.

### Available intervention operations

- Spindle clockwise/stop (key operations below)
  - [SP.CW] and [SP.STOP]
- Jog and step operations (key operations below)
  - [RPD], [JOG] or [STEP]
  - [+/-] or [-/+]
  - [+X], [-X], [+Y], [-Y], [+Z], [-Z]
- Axis travel using manual pulse generator (X-, Y- and Z-axes only)

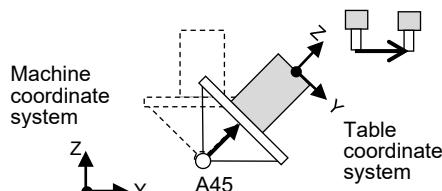
- (NOTE 1) When one of the following intervention operations is performed while under TCP control, an operator message appears.
- [ATC], [Z.RTN] and [L.SP] keys
  - Additional axis is selected on the <Manual conditions> screen and [+4] or [-4] key is pressed
  - Additional axis is selected on the <Manual conditions> screen and [Axis selection] switch is changed to <4> on the manual pulse generator
- (NOTE 2) If the mode is changed to manual operation mode during the G91 modal, an axis travel operation cannot be carried out. The operator message <<TCP under control>> appears when one of the following operation is carried out.
- [+X], [-X], [+Y], [-Y], [+Z] or [-Z] key is pressed
  - When the [Axis selection] switch on the manual pulse generator is changed to a setting besides <OFF>

### Coordinate system for intervention operation

All axis travel in an intervention operation are commands in the machine coordinate system.

Operation direction examples are shown below.

When the [+Y] key is used to carry out a jog or step operation, or when the <Y-axis> is selected on the manual pulse generator and the handle is turned to the plus direction.



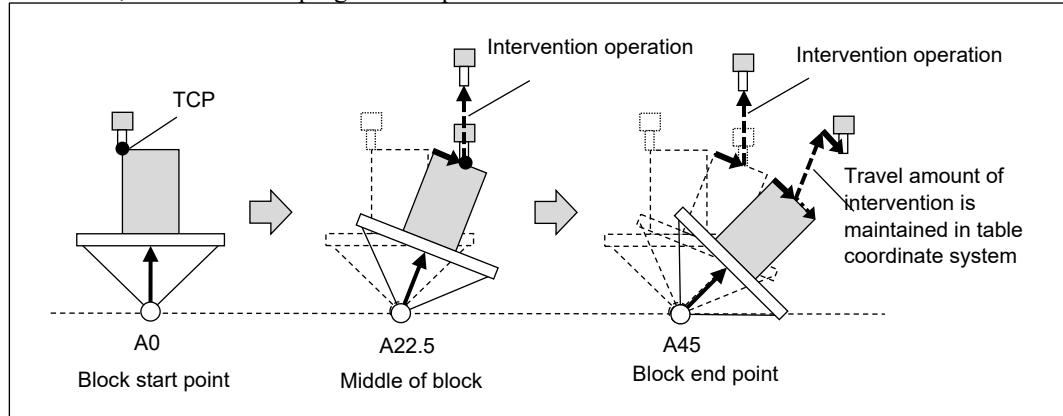
### Axis travel speed of intervention operation

Refer to "Chapter 5 Manual operation" in Operation Manual I for details on the axis travel speed.

Axis travel after restoring from intervention operation

When an intervention operation is carried out during travel on the block, the travel amount of the intervention is maintained in the table coordinate system until the end point on the block prior to the intervention.

Thereafter, it travels to the programmed position on the next block.

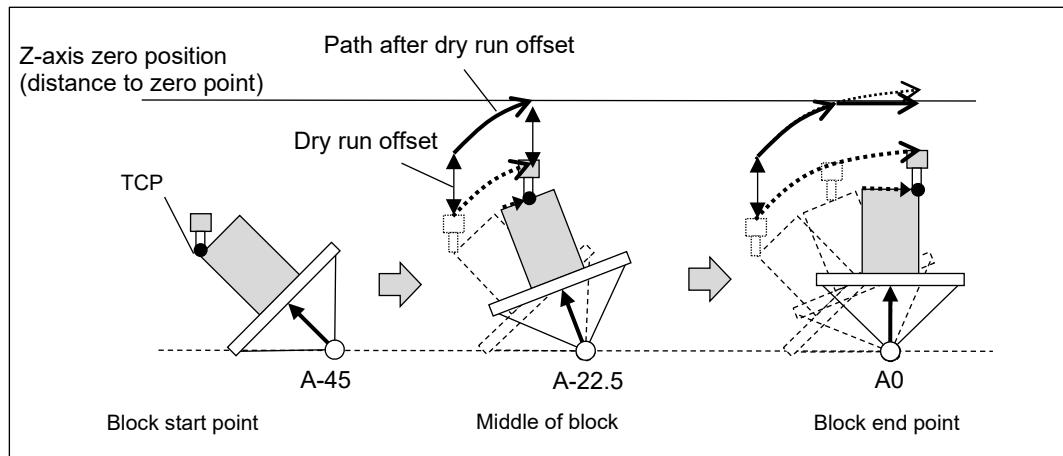


- (NOTE) Acceleration/deceleration before interpolation is disabled for travel up to the end point on the block prior to the intervention. Compared to when a manual intervention has not been carried out, the difference in the paths is larger, because acceleration/deceleration is carried out using only the interpolation time constant.

### 14.2.7.8 Dry run

In the same way as when the TCP control is OFF, after pressing the [DRY] key, the offset is applied in the Z-axis direction in the machine coordinates. Set the offset amount in the user parameter (switch 1: operation) <Dry run offset amount>.

- (NOTE 1) The dry run ON/OFF cannot be changed while TCP control is being used. When the [DRY] key is pressed, the operator message <<TCP under control>> appears.
- (NOTE 2) As shown in the image below, when the path exceeds the Z-axis zero position (distance to Z-axis zero) after the dry run offset is applied, then the Z-axis is clamped at the zero position (distance to zero point) and operates accordingly.
- (NOTE 3) When a manual intervention is carried out while clamped at the Z-axis zero position, then the Z-axis travel cannot be carried out. The operator message <<TCP under control>> appears when one of the following operations is carried out.
- When [+Z] or [-Z] key is pressed for the jog or step operation
  - When the [Axis selection] switch on the manual pulse generator is changed to the <Z-axis> and the handle is turned



## Chapter 14 5 axes machining function

### Dry run speed (positioning)

As shown in the table below, the TCP speed follows the ON/OFF of the PLC signal <Positioning speed restriction>.

| <Positioning speed restriction> | TCP speed (positioning)                                                                                                                                                                                                                                                                                                                                                                                 |
|---------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| OFF                             | <p>The travelling axes operate at a maximum TCP speed that does not exceed the following speeds.</p> <ul style="list-style-type: none"> <li>• X-, Y- and Z-axes<br/>Machine parameter (system 1: X-, Y- and Z-axes) &lt;Rapid feedrate&gt; (X-, Y- and Z-axes)</li> <li>• Tilt axis/rotation axis<br/>Machine parameter (system 2: additional axis) &lt;Rapid feedrate&gt; (5th to 8th-axes)</li> </ul> |
| ON                              | <p>The travelling axes operate at a maximum TCP speed that does not exceed the following speeds.</p> <ul style="list-style-type: none"> <li>• X-, Y- and Z-axes<br/>Positioning clamp speed 1 (BDY160/161)</li> <li>• Tilt axis/rotation axis<br/>Positioning clamp speed 2 (BDY162/163)</li> </ul>                                                                                                     |

### Dry run speed (cutting)

- When the user parameter (switch 1: operation) <Axis travel speed method for dry run> is set to <Method 1>

| <Positioning speed restriction> | TCP speed (Cutting)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                 |
|---------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| OFF                             | <p>Whichever one of the following is greater</p> <ul style="list-style-type: none"> <li>- F command value</li> <li>- Maximum TCP speed where travelling axes do not exceed the following speeds</li> <li>• X-, Y- and Z-axes<br/>Machine parameter (system 1: X-, Y- and Z-axes) &lt;Maximum cutting travel speed&gt; (X-, Y- and Z-axes) × &lt;High travel spd(XYZ/P)&gt; on manual conditions screen (10/25/50/75/100%)</li> <li>• Tilt axis/rotation axis<br/>Machine parameter (system 2: additional axis) &lt;Maximum cutting rotation speed&gt; (5th- to 8th-axes) × &lt;High travel spd(XYZ/P)&gt; on manual conditions screen (10/25/50/75/100%)</li> </ul> |
| ON                              | <p>The travelling axes operate at a maximum TCP speed that does not exceed the following speeds.</p> <ul style="list-style-type: none"> <li>• X-, Y- and Z-axes<br/>Cutting clamp speed 1 (BDY164/165) × &lt;High travel spd(XYZ/P)&gt; on manual conditions screen</li> <li>• Tilt axis/rotation axis<br/>Cutting clamp speed 2 (BDY166/167) × &lt;High rotation spd(4/P)&gt; on manual conditions screen</li> </ul>                                                                                                                                                                                                                                               |

- When the user parameter (switch 1: operation) <Axis travel speed method for dry run> is set to <Method 2>

| <Positioning speed restriction> | TCP speed (positioning)                                                                                                                                                                                                                                                                                                                                                                                   |
|---------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| OFF                             | <p>Maximum TCP speed where travelling axes do not exceed the following speeds</p> <ul style="list-style-type: none"> <li>X-, Y- and Z-axes<br/>Machine parameter (system 1: X-, Y- and Z-axes) &lt;Maximum cutting travel speed&gt; (X- to Z-axes)</li> <li>Tilt axis/rotation axis<br/>Machine parameter (system 2: additional axis) &lt;Maximum cutting rotation speed&gt; (5th to 8th-axes)</li> </ul> |
| ON                              | <p>The travelling axes operate at a maximum TCP speed that does not exceed the following speeds.</p> <ul style="list-style-type: none"> <li>X-, Y- and Z-axes<br/>Cutting clamp speed 1 (BDY164/165)</li> <li>Tilt axis/rotation axis<br/>Cutting clamp speed 2 (BDY166/167)</li> </ul>                                                                                                                   |

(NOTE) Refer to “7.1.11 Dry run” in Operation Manual I for details about the dry run speed when TCP control is OFF.

### 14.2.7.9 Software limit

The following software limit checks are carried out.

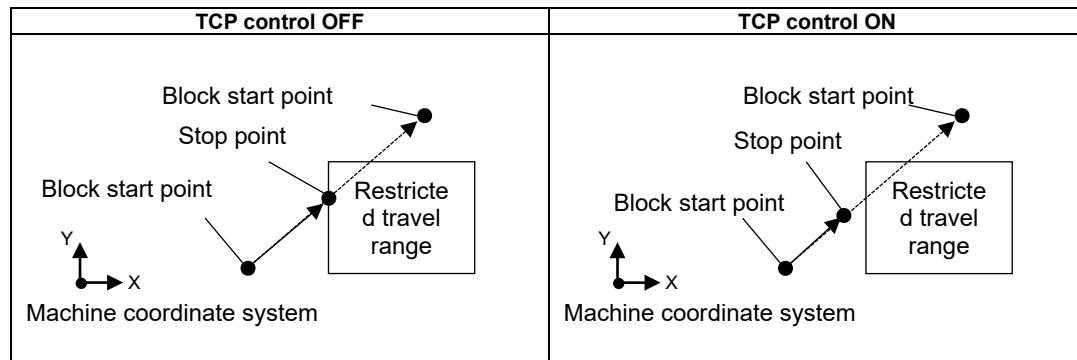
In the same way as when the TCP control is OFF, the machine coordinates for the spindle end are checked.

- Stroke setting in machine parameter
- Stroke limit setting in user parameter

Refer to “3.9 Software limit” for further details.

(NOTE 1) The TCP machine coordinates are not checked.

(NOTE 2) While under TCP control, operation may stop a little before the restricted travel range.



### 14.2.7.10 Skip function

Skip (G31/G131/132) can be used while under TCP control.

However, it is only possible for X-, Y- and Z-axes commands. If an additional axis command is issued, the alarm <>Address where command is not possible<> is triggered.

### 14.2.7.11 Graphic function

The graphic function (tool path simulation and operation graphs) can be used.

To draw a graph that includes the additional axis travel, set the parameter <Drawing method> to <Method 2> or <Method 3>.

In addition, to draw a path of the tool center (end), set the following parameters to <Added>.

- Tool path simulation
  - Parameter <Tool length/Tool position offset (Tool length offset)>
- Operation drawing
  - When drawing before operation, parameter <Tool length/Tool position offset (Tool length offset) (Drawing before operation)>
  - When drawing during operation, parameter <Tool length/Tool position offset (Tool length offset) (Drawing during operation)>

Refer to “2.2.7.1 Drawing parameter” in Operation Manual II for further details.

### 14.2.7.12 Smooth path offset function

The smooth path offset function is invalid while under TCP control.

The path offset is not carried out even if the user parameter (high accuracy: common) <Smooth path offset function> is set to <Valid>.

---

(This page was intentionally left blank.)

---

©2021-2022 BROTHER INDUSTRIES, LTD.  
All Rights Reserved.  
Printed in Japan.  
This is the original instructions.

69A799001  
2208 (6)