

PROGRAMMING MANUAL

MAZATROL MATRIX 2

(for Machining Centers)

EIA/ISO Program

MANUAL No.: H740PB6042E

Serial No.:

Before using this machine and equipment, fully understand the contents of this manual to ensure proper operation. Should any questions arise, please ask the nearest Technical Center or Technology Center.

IMPORTANT NOTICE

1. Be sure to observe the safety precautions described in this manual and the contents of the safety plates on the machine and equipment. Failure may cause serious personal injury or material damage. Please replace any missing safety plates as soon as possible.
2. No modifications are to be performed that will affect operation safety.
3. For the purpose of explaining the operation of the machine and equipment, some illustrations may not include safety features such as covers, doors, etc. Before operation, make sure all such items are in place.
4. This manual was considered complete and accurate at the time of publication, however, due to our desire to constantly improve the quality and specification of all our products, it is subject to change or modification. If you have any questions, please contact the nearest Technical Center or Technology Center.
5. Always keep this manual near the machinery for immediate use.
6. If a new manual is required, please order from the nearest Technical Center or Technology Center with the manual No. or the machine name, serial No. and manual name.

Issued by *Manual Publication Section, Yamazaki Mazak Corporation, Japan*

SAFETY PRECAUTIONS

Preface

Safety precautions relating to the CNC unit (in the remainder of this manual, referred to simply as the NC unit) that is provided in this machine are explained below. Not only the persons who create programs, but also those who operate the machine must thoroughly understand the contents of this manual to ensure safe operation of the machine.

Read all these safety precautions, even if your NC model does not have the corresponding functions or optional units and a part of the precautions do not apply.

Rule

1. This section contains the precautions to be observed as to the working methods and states usually expected. Of course, however, unexpected operations and/or unexpected working states may take place at the user site.
During daily operation of the machine, therefore, the user must pay extra careful attention to its own working safety as well as to observe the precautions described below.
2. Although this manual contains as great an amount of information as it can, since it is not rare for the user to perform the operations that overstep the manufacturer-assumed ones, not all of “what the user cannot perform” or “what the user must not perform” can be fully covered in this manual with all such operations taken into consideration beforehand.
It is to be understood, therefore, that functions not clearly written as “executable” are “inexecutable” functions.
3. The meanings of our safety precautions to DANGER, WARNING, and CAUTION are as follows:



DANGER

: Failure to follow these instructions could result in loss of life.



WARNING

: Failure to observe these instructions could result in serious harm to a human life or body.



CAUTION

: Failure to observe these instructions could result in minor injuries or serious machine damage.

Basics



- After turning power on, keep hands away from the keys, buttons, or switches of the operating panel until an initial display has been made.
- Before proceeding to the next operations, fully check that correct data has been entered and/or set. If the operator performs operations without being aware of data errors, unexpected operation of the machine will result.
- Before machining workpieces, perform operational tests and make sure that the machine operates correctly. No workpieces must be machined without confirmation of normal operation. Closely check the accuracy of programs by executing override, single-block, and other functions or by operating the machine at no load. Also, fully utilize tool path check, Virtual Machining, and other functions, if provided.
- Make sure that the appropriate feed rate and rotational speed are designated for the particular machining requirements. Always understand that since the maximum usable feed rate and rotational speed are determined by the specifications of the tool to be used, those of the workpiece to be machined, and various other factors, actual capabilities differ from the machine specifications listed in this manual. If an inappropriate feed rate or rotational speed is designated, the workpiece or the tool may abruptly move out from the machine.
- Before executing correction functions, fully check that the direction and amount of correction are correct. Unexpected operation of the machine will result if a correction function is executed without its thorough understanding.
- Parameters are set to the optimum standard machining conditions prior to shipping of the machine from the factory. In principle, these settings should not be modified. If it becomes absolutely necessary to modify the settings, perform modifications only after thoroughly understanding the functions of the corresponding parameters. Modifications usually affect any program. Unexpected operation of the machine will result if the settings are modified without a thorough understanding.

Remarks on the Cutting Conditions Recommended by the NC



- Before using the following cutting conditions:
 - Cutting conditions that are the result of the MAZATROL Automatic Cutting Conditions Determination Function
 - Cutting conditions suggested by the Machining Navigation Function
 - Cutting conditions for tools that are suggested to be used by the Machining Navigation Function

Confirm that every necessary precaution in regards to safe machine setup has been taken – especially for workpiece fixturing/clamping and tool setup.
- Confirm that the machine door is securely closed before starting machining.
Failure to confirm safe machine setup may result in serious injury or death.

Programming



- Fully check that the settings of the coordinate systems are correct. Even if the designated program data is correct, errors in the system settings may cause the machine to operate in unexpected places and the workpiece to abruptly move out from the machine in the event of contact with the tool.
- During surface velocity hold control, as the current workpiece coordinates of the surface velocity hold control axes approach zeroes, the spindle speed increases significantly. For the lathe, the workpiece may even come off if the chucking force decreases. Safety speed limits must therefore be observed when designating spindle speeds.
- Even after inch/metric system selection, the units of the programs, tool information, or parameters that have been registered until that time are not converted. Fully check these data units before operating the machine. If the machine is operated without checks being performed, even existing correct programs may cause the machine to operate differently from the way it did before.
- If a program is executed that includes the absolute data commands and relative data commands taken in the reverse of their original meaning, totally unexpected operation of the machine will result. Recheck the command scheme before executing programs.
- If an incorrect plane selection command is issued for a machine action such as arc interpolation or fixed-cycle machining, the tool may collide with the workpiece or part of the machine since the motions of the control axes assumed and those of actual ones will be interchanged. (This precaution applies only to NC units provided with EIA/ISO functions.)
- The mirror image, if made valid, changes subsequent machine actions significantly. Use the mirror image function only after thoroughly understanding the above. (This precaution applies only to NC units provided with EIA/ISO functions.)
- If machine coordinate system commands or reference position returning commands are issued with a correction function remaining made valid, correction may become invalid temporarily. If this is not thoroughly understood, the machine may appear as if it would operate against the expectations of the operator. Execute the above commands only after making the corresponding correction function invalid. (This precaution applies only to NC units provided with EIA/ISO functions.)
- The barrier function performs interference checks based on designated tool data. Enter the tool information that matches the tools to be actually used. Otherwise, the barrier function will not work correctly.
- The system of G-code and M-code commands differs, especially for turning, between the machines of INTEGREX e-Series and the other turning machines.
Issuance of the wrong G-code or M-code command results in totally non-intended machine operation. Thoroughly understand the system of G-code and M-code commands before using this system.

Sample program	Machines of INTEGREX e-Series	Turning machines
S1000M3	The milling spindle rotates at 1000 min ⁻¹ .	The turning spindle rotates at 1000 min ⁻¹ .
S1000M203	The turning spindle rotates at 1000 min ⁻¹ .	The milling spindle rotates at 1000 min ⁻¹ .

- For the machines of INTEGREX e-Series, programmed coordinates can be rotated using an index unit of the MAZATROL program and a G68 command (coordinate rotate command) of the EIA/ISO program. However, for example, when the B-axis is rotated through 180 degrees around the Y-axis to implement machining with the turning spindle No. 2, the plus side of the X-axis in the programmed coordinate system faces downward and if the program is created ignoring this fact, the resulting movement of the tool to unexpected positions may incite collisions.

To create the program with the plus side of the X-axis oriented in an upward direction, use the mirror function of the WPC shift unit or the mirror imaging function of G-code command (G50.1, G51.1).

- After modifying the tool data specified in the program, be sure to perform the tool path check function, the Virtual Machining function, and other functions, and confirm that the program operates properly. The modification of tool data may cause even a field-proven machining program to change in operational status.

If the user operates the machine without being aware of any changes in program status, interference with the workpiece could arise from unexpected operation.

For example, if the cutting edge of the tool during the start of automatic operation is present inside the clearance-including blank (unmachined workpiece) specified in the common unit of the MAZATROL program, care is required since the tool will directly move from that position to the approach point because of no obstructions being judged to be present on this path.

For this reason, before starting automatic operation, make sure that the cutting edge of the tool during the start of automatic operation is present outside the clearance-including workpiece specified in the common unit of the MAZATROL program.

**CAUTION**

- If axis-by-axis independent positioning is selected and simultaneously rapid feed selected for each axis, movements to the ending point will not usually become linear. Before using these functions, therefore, make sure that no obstructions are present on the path.
- Before starting the machining operation, be sure to confirm all contents of the program obtained by conversion. Imperfections in the program could lead to machine damage and operator injury.

Operations



- Single-block, feed hold, and override functions can be made invalid using system variables #3003 and #3004. Execution of this means the important modification that makes the corresponding operations invalid. Before using these variables, therefore, give thorough notification to related persons. Also, the operator must check the settings of the system variables before starting the above operations.
- If manual intervention during automatic operation, machine locking, the mirror image function, or other functions are executed, the workpiece coordinate systems will usually be shifted. When making machine restart after manual intervention, machine locking, the mirror image function, or other functions, consider the resulting amounts of shift and take the appropriate measures. If operation is restarted without any appropriate measures being taken, collision with the tool or workpiece may occur.
- Use the dry run function to check the machine for normal operation at no load. Since the feed rate at this time becomes a dry run rate different from the program-designated feed rate, the axes may move at a feed rate higher than the programmed value.
- After operation has been stopped temporarily and insertion, deletion, updating, or other commands executed for the active program, unexpected operation of the machine may result if that program is restarted. No such commands should, in principle, be issued for the active program.



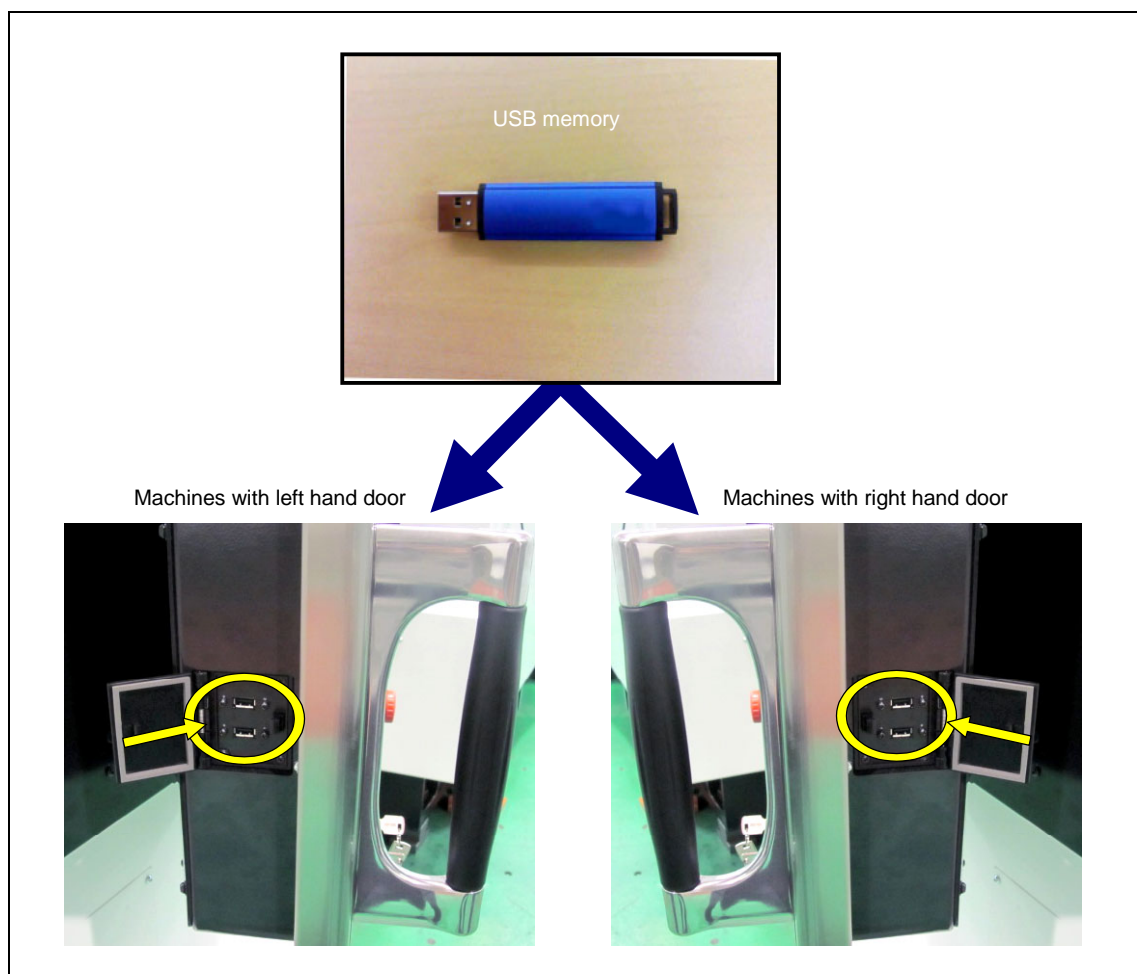
- During manual operation, fully check the directions and speeds of axial movement.
- For a machine that requires manual homing, perform manual homing operations after turning power on. Since the software-controlled stroke limits will remain ineffective until manual homing is completed, the machine will not stop even if it oversteps the limit area. As a result, serious machine damage will result.
- Do not designate an incorrect pulse multiplier when performing manual pulse handle feed operations. If the multiplier is set to 1000 times and the handle operated inadvertently, axial movement will become faster than that expected.

REQUEST TO THE USER

Request for Saving Data of Machining Programs

Machining programs saved on the hard disk of the NC unit may not be read out in the event of accidental hard disk trouble. The user, therefore, is earnestly requested to back up and store every machining program of importance at regular intervals onto an external memory (USB memory, memory card, etc.).

- The procedure for data storage is detailed in the Operating Manual, Part 3 (OPERATING NC UNIT AND PREPARATION FOR AUTOMATIC OPERATION), Chapter 9, (DISPLAY RELATED TO DATA STORAGE).
- Always use an initialized USB memory. The location of the USB connector depends on the machine model, as shown below.



On machines with a random ATC feature, each actual ATC operation changes the tool data (in pocket numbers). Be sure not to run the machine after loading the externally stored data of the **TOOL DATA** display without having confirmed the data's correspondence to the current tooling on the magazine. Otherwise the machine cannot be guaranteed to operate normally.

BEFORE USING THE NC UNIT

Limited Warranty

The warranty of the manufacturer does not cover any trouble arising if the NC unit is used for its non-intended purpose. Take notice of this when operating the unit.

Examples of the trouble arising if the NC unit is used for its non-intended purpose are listed below.

1. Trouble associated with and caused by the use of any commercially available software products (including user-created ones)
2. Trouble associated with and caused by the use of any Windows operating systems
3. Trouble associated with and caused by the use of any commercially available computer equipment

Operating Environment

1. Ambient temperature

During machine operation: 0° to 50°C (32° to 122°F)

2. Relative humidity

During machine operation: 10 to 75% (without bedewing)

Note: As humidity increases, insulation deteriorates causing electrical component parts to deteriorate quickly.

Keeping the Backup Data

Note: Do not attempt to delete or modify the data stored in the following folder.

Recovery Data Storage Folder: D:\MazakBackUp

Although this folder is not used when the NC unit is running normally, it contains important data that enables the prompt recovery of the machine if it fails.

If this data has been deleted or modified, the NC unit may require a long recovery time. Be sure not to modify or delete this data.

- NOTE -

CONTENTS

Page

1	CONTROLLED AXES.....	1-1
1-1	Coordinate Words and Controlled Axes	1-1
2	UNITS OF PROGRAM DATA INPUT	2-1
2-1	Units of Program Data Input	2-1
2-2	Units of Data Setting.....	2-1
2-3	Ten-Fold Program Data	2-1
3	DATA FORMATS.....	3-1
3-1	Tape Codes	3-1
3-2	Program Formats	3-5
3-3	Tape Data Storage Format.....	3-6
3-4	Optional Block Skip	3-6
3-5	Program Number, Sequence Number and Block Number: O, N	3-7
3-6	Parity-H/V	3-8
3-7	List of G-Codes	3-10
4	BUFFER REGISTERS.....	4-1
4-1	Input Buffer	4-1
4-2	Preread Buffer	4-2
5	POSITION PROGRAMMING.....	5-1
5-1	Dimensional Data Input Method	5-1
5-1-1	Absolute/Incremental data input: G90/G91	5-1

5-2	Inch/Metric Selection: G20/G21	5-3
5-3	Decimal Point Input	5-4
6	INTERPOLATION FUNCTIONS	6-1
6-1	Positioning (Rapid Traverse): G00	6-1
6-2	One-Way Positioning: G60	6-4
6-3	Linear Interpolation: G01	6-5
6-4	Circular Interpolation: G02, G03	6-6
6-5	Radius Designated Circular Interpolation: G02, G03	6-9
6-6	Spiral Interpolation: G2.1, G3.1 (Option)	6-11
6-7	Plane Selection: G17, G18, G19	6-19
6-7-1	Outline	6-19
6-7-2	Plane selection methods	6-20
6-8	Polar Coordinate Interpolation ON/OFF: G12.1/G13.1	6-21
6-9	Virtual-Axis Interpolation: G07	6-24
6-10	Spline Interpolation: G06.1 (Option)	6-25
6-11	Modal Spline Interpolation: G61.2 (Option)	6-36
6-12	NURBS Interpolation: G06.2 (Option)	6-37
6-13	Cylindrical Interpolation: G07.1	6-44
6-14	Helical Interpolation: G02, G03	6-53
6-15	Fixed Gradient Control for G0 (Option)	6-55
7	FEED FUNCTIONS	7-1
7-1	Rapid Traverse Rates	7-1

7-2	Cutting Feed Rates.....	7-1
7-3	Asynchronous/Synchronous Feed: G94/G95	7-1
7-4	Selecting a Feed Rate and Effects on Each Control Axis.....	7-3
7-5	Automatic Acceleration/Deceleration.....	7-6
7-6	Limitation of Speed.....	7-7
7-7	Exact-Stop Check: G09.....	7-7
7-8	Exact-Stop Check Mode: G61	7-10
7-9	Automatic Corner Override: G62	7-10
7-10	Tapping Mode: G63.....	7-15
7-11	Cutting Mode: G64	7-15
7-12	Geometry Compensation/Accuracy Coefficient: G61.1/,K.....	7-16
7-12-1	Geometry compensation function: G61.1	7-16
7-12-2	Accuracy coefficient (,K)	7-17
7-13	Inverse Time Feed: G93 (Option).....	7-18
8	DWELL FUNCTIONS	8-1
8-1	Dwell Command in Time: (G94) G04.....	8-1
8-2	Dwell Command in Number of Revolutions: (G95) G04	8-2
9	MISCELLANEOUS FUNCTIONS	9-1
9-1	Miscellaneous Functions (M3-Digit).....	9-1
9-2	No. 2 Miscellaneous Functions (A8/B8/C8-Digit).....	9-2
10	SPINDLE FUNCTIONS	10-1
10-1	Spindle Function (S5-Digit Analog).....	10-1

10-2	Spindle Speed Range Setting: G92.....	10-1
11	TOOL FUNCTIONS	11-1
11-1	Tool Function (4-Digit T-Code)	11-1
11-2	Tool Function (8-Digit T-Code)	11-1
12	TOOL OFFSET FUNCTIONS	12-1
12-1	Tool Offset.....	12-1
12-2	Tool Length Offset/Cancellation: G43, G44, or T-Code/G49	12-5
12-3	Tool Length Offset in Tool-Axis Direction: G43.1 (Option).....	12-7
12-4	Tool Position Offset: G45 to G48.....	12-14
12-5	Tool Radius Compensation: G40, G41, G42	12-20
12-5-1	Overview	12-20
12-5-2	Tool radius compensation.....	12-20
12-5-3	Tool radius compensation using other commands	12-29
12-5-4	Corner movement	12-36
12-5-5	Interruptions during tool radius compensation.....	12-36
12-5-6	General precautions on tool radius compensation	12-38
12-5-7	Offset number updating during the offset mode	12-39
12-5-8	Excessive cutting due to tool radius compensation	12-41
12-5-9	Interference check	12-43
12-6	Three-Dimensional Tool Radius Compensation (Option)	12-50
12-6-1	Function description.....	12-50
12-6-2	Programming methods	12-51
12-6-3	Correlationships to other functions	12-55

12-6-4	Miscellaneous notes on three-dimensional tool radius compensation.....	12-55
12-7	Programmed Data Setting: G10	12-56
12-8	Tool Offsetting Based on MAZATROL Tool Data	12-64
12-8-1	Selection parameters	12-64
12-8-2	Tool length offsetting	12-65
12-8-3	Tool radius compensation.....	12-66
12-8-4	Tool data update (during automatic operation)	12-67
12-9	Shaping Function (Option).....	12-68
12-9-1	Overview	12-68
12-9-2	Programming format	12-69
12-9-3	Detailed description	12-69
12-9-4	Remarks	12-76
12-9-5	Compatibility with the other functions	12-77
12-9-6	Sample program	12-78
13	PROGRAM SUPPORT FUNCTIONS.....	13-1
13-1	Hole Machining Pattern Cycles: G34.1/G35/G36/G37.1.....	13-1
13-1-1	Overview	13-1
13-1-2	Holes on a circle: G34.1	13-2
13-1-3	Holes on a line: G35	13-3
13-1-4	Holes on an arc: G36.....	13-4
13-1-5	Holes on a grid: G37.1	13-5
13-2	Fixed Cycles.....	13-7
13-2-1	Outline	13-7
13-2-2	Fixed-cycle machining data format	13-8

13-2-3	G71.1 (Chamfering cutter CW)	13-11
13-2-4	G72.1 (Chamfering cutter CCW)	13-12
13-2-5	G73 (High-speed deep-hole drilling)	13-13
13-2-6	G74 (Reverse tapping)	13-14
13-2-7	G75 (Boring)	13-15
13-2-8	G76 (Boring)	13-16
13-2-9	G77 (Back spot facing)	13-17
13-2-10	G78 (Boring)	13-18
13-2-11	G79 (Boring)	13-19
13-2-12	G81 (Spot drilling)	13-19
13-2-13	G82 (Drilling)	13-20
13-2-14	G83 (Deep-hole drilling)	13-21
13-2-15	G84 (Tapping)	13-22
13-2-16	G85 (Reaming)	13-23
13-2-17	G86 (Boring)	13-23
13-2-18	G87 (Back boring)	13-24
13-2-19	G88 (Boring)	13-25
13-2-20	G89 (Boring)	13-25
13-2-21	Synchronous tapping (Option)	13-26
13-3	Suppression of Single-Block Stop for Fixed Cycles	13-30
13-3-1	Function description	13-30
13-3-2	Examples of operation	13-30
13-4	Initial Point and R-Point Level Return: G98 and G99	13-31
13-5	Scaling ON/OFF: G51/G50	13-32

13-6	Mirror Image ON/OFF: G51.1/G50.1	13-45
13-7	Subprogram Control: M98, M99	13-46
13-8	End Processing: M02, M30, M998, M999.....	13-53
13-9	Linear Angle Commands	13-54
13-10	Macro Call Function: G65, G66, G66.1, G67.....	13-55
13-10-1	User macros	13-55
13-10-2	Macro call instructions	13-56
13-10-3	Variables	13-65
13-10-4	Types of variables.....	13-67
13-10-5	Arithmetic operation commands	13-88
13-10-6	Control commands.....	13-92
13-10-7	External output commands (Output via RS-232C).....	13-96
13-10-8	External output command (Output onto the hard disk)	13-98
13-10-9	Precautions	13-100
13-10-10	Specific examples of programming using user macros	13-102
13-11	Geometric Commands (Option).....	13-106
13-12	Corner Chamfering and Corner Rounding Commands.....	13-108
13-12-1	Corner chamfering (,C_)	13-108
13-12-2	Corner rounding (,R_).....	13-109
14	COORDINATE SYSTEM SETTING FUNCTIONS.....	14-1
14-1	Fundamental Machine Coordinate System, Workpiece Coordinate Systems, and Local Coordinate Systems	14-1
14-2	Machine Zero Point and Second, Third, and Fourth Reference Points.....	14-2
14-3	Fundamental Machine Coordinate System Selection: G53	14-3

14-4	Coordinate System Setting: G92	14-4
14-5	Automatic Coordinate System Setting	14-5
14-6	Reference Point Return: G28, G29	14-6
14-7	Second, Third, or Fourth Reference Point Return: G30.....	14-8
14-8	Reference Point Check Command: G27	14-10
14-9	Workpiece Coordinate System Setting and Selection: (G92) G54 to G59.....	14-11
14-10	Additional Workpiece Coordinate System Setting and Selection: G54.1	14-16
14-11	Local Coordinate System Setting: G52.....	14-22
14-12	Reading/Writing of MAZATROL Program Basic Coordinates	14-27
14-12-1	Calling a macroprogram (for data writing)	14-27
14-12-2	Data reading	14-27
14-12-3	Rewriting	14-28
14-13	Workpiece Coordinate System Rotation.....	14-29
14-14	Three-Dimensional Coordinate Conversion: G68	14-42
15	MEASUREMENT SUPPORT FUNCTIONS.....	15-1
15-1	Skip Function: G31	15-1
15-1-1	Function description.....	15-1
15-2	Skip Coordinate Reading.....	15-2
15-3	Amount of Coasting in the Execution of a G31 Block	15-3
15-4	Skip Coordinate Reading Error.....	15-4
15-5	Multi-Step Skip: G31.1, G31.2, G31.3, G04	15-5
16	PROTECTIVE FUNCTIONS.....	16-1

16-1	Pre-move Stroke Check ON/OFF: G22/G23	16-1
17	THREADING: G33 (OPTION).....	17-1
17-1	Equal-Lead Threading	17-1
17-2	Continuous Threading	17-4
17-3	Inch Threading	17-4
18	DYNAMIC OFFSETTING: M173, M174 (OPTION)	18-1
19	HIGH-SPEED SMOOTHING CONTROL FUNCTION (OPTION)	19-1
19-1	Programming Format.....	19-2
19-2	Commands Available in the High-Speed Smoothing Control Mode	19-2
19-3	Additional Functions in the High-Speed Smoothing Control Mode	19-4
19-4	Related Parameters.....	19-4
19-5	Restrictions.....	19-5
19-6	Related Alarms	19-5
20	FUNCTION FOR SELECTING THE CUTTING CONDITIONS.....	20-1
21	TORNADO TAPPING (G130).....	21-1
22	HIGH-SPEED MACHINING MODE FEATURE (OPTION)	22-1
22-1	Programming Format.....	22-2
22-2	Commands Available in the High-Speed Machining Mode	22-3
22-3	Additional Functions in the High-Speed Machining Mode	22-4
22-4	Restrictions.....	22-6
23	AUTOMATIC TOOL LENGTH MEASUREMENT: G37 (OPTION)	23-1

24	DYNAMIC OFFSETTING II: G54.2P0, G54.2P1 - G54.2P8 (OPTION).....	24-1
25	COMPENSATION FOR DEVIATION OF THE AXIS OF ROTATION OF THE TILTING TABLE	25-1
26	FIVE-AXIS MACHINING FUNCTION	26-1
26-1	Tool Tip Point Control for Five-Axis Machining (Option).....	26-1
26-1-1	Function outline	26-1
26-1-2	Detailed description	26-2
26-1-3	Relationship to other functions	26-18
26-1-4	Restrictions	26-24
26-1-5	Related parameters	26-26
26-2	Inclined-Plane Machining: G68.2, G68.3, G68.4, G53.1 (Option).....	26-30
26-2-1	Function description.....	26-30
26-2-2	Relationship to other functions	26-46
26-2-3	Restrictions	26-50
26-3	Tool Radius Compensation for Five-Axis Machining (Option)	26-53
26-3-1	Function outline	26-53
26-3-2	Function description.....	26-53
26-3-3	Operation of tool radius compensation for five-axis machining	26-55
26-3-4	Method of computing the offset vector.....	26-55
26-3-5	Relationship to other functions	26-57
26-3-6	Restrictions	26-59
26-4	Workpiece Setup Error Correction: G54.4P0, G54.4P1 to G54.4P7 (Option)	26-61
26-4-1	Function outline	26-61

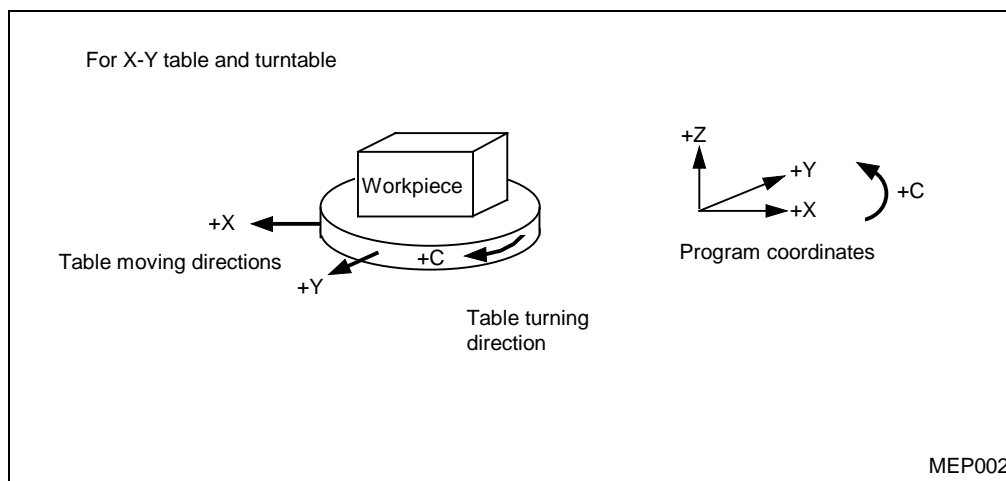
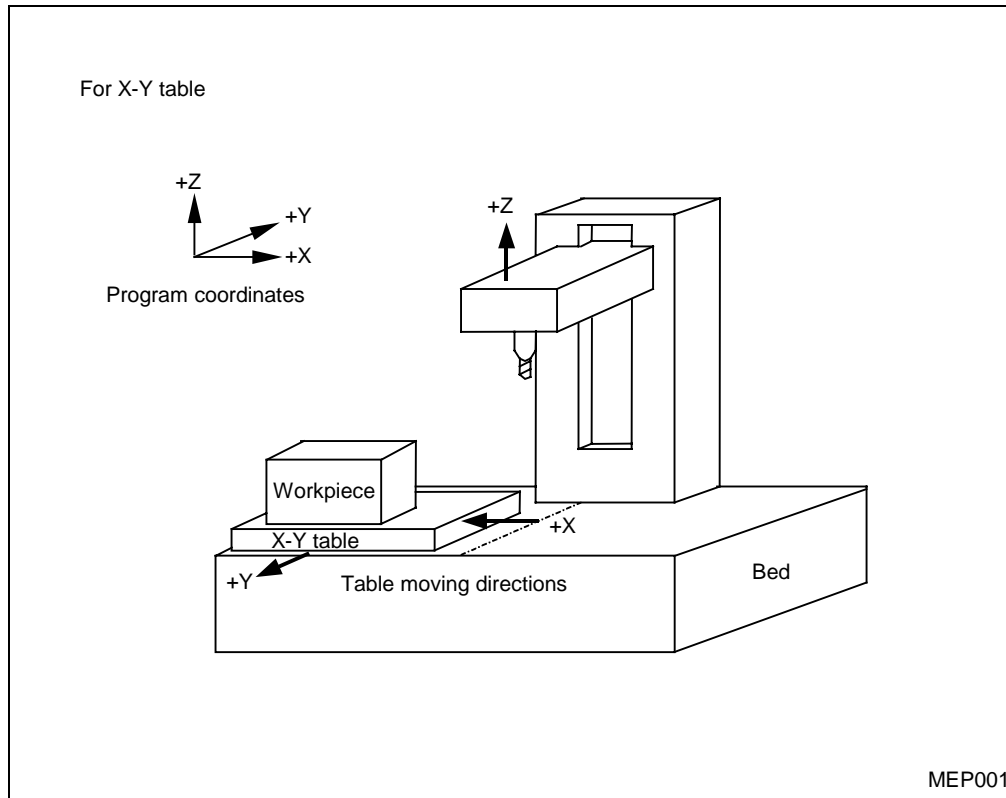
26-4-2	Function description.....	26-61
26-4-3	Relationship to other functions	26-68
26-4-4	Restrictions	26-69
27	CHOPPING FUNCTION (OPTION).....	27-1
28	EIA/ISO PROGRAM DISPLAY	28-1
28-1	Procedures for Constructing an EIA/ISO Program	28-1
28-2	Editing Function of EIA/ISO PROGRAM Display.....	28-2
28-2-1	Outline	28-2
28-2-2	Operation procedure.....	28-2
28-3	Macro-Instruction Input.....	28-8
28-4	Division of Display (Split Screen).....	28-9
28-5	Editing Programs Stored in External Memory Areas	28-12

- NOTE -

1 CONTROLLED AXES

1-1 Coordinate Words and Controlled Axes

Under standard specifications, there are three-dimensional controlled axes. With an added feature and a special option, the machine can control up to a maximum of six axes, including the three fundamental axes. The direction of machining can be designated using a predetermined coordinate word consisting of an alphabetic character.



- NOTE -

2 UNITS OF PROGRAM DATA INPUT

2-1 Units of Program Data Input

The movements on coordinate axes are to be commanded in the MDI mode or machining program. The movement data are expressed in millimeters, inches or degrees.

2-2 Units of Data Setting

Various data commonly used for control axes, such as offsetting data, must be set for the machine to perform an operation as desired.

The units of data setting and those of program data input are listed below.

	Linear axis		Rotational axis
	Metric system	Inch system	
Units of program data input	0.0001 mm	0.00001 in	0.0001 deg
Units of data setting	0.0001 mm	0.00001 in	0.0001 deg

Note 1: Inch/metric selection can be freely made using either bit 4 of parameter **F91** ("0" for metric, "1" for inches; validated through power-off and -on) or G-code commands (G20, G21).

Selection using the G-code commands is valid only for program data input.

Variables and offsetting data (such as tool offsetting data) should therefore be set beforehand using the appropriate unit (inch or metric) for the particular machining requirements.

Note 2: Metric data and inch data cannot be used at the same time.

2-3 Ten-Fold Program Data

Using a predetermined parameter, machining program data can be processed as set in units of one μm . There may be cases that a machining program which has been set in units of one μm is to be used with a numerical control unit based on 0.1 μm increments. In such cases, use of this parameter allows the machine to perform the required machining operations without rewriting the program.

Use bit 0 of user parameter **F91** for this purpose.

All types of coordinate data (axis movement data) not provided with the decimal point will be multiplied by a factor of 10. This does not apply, indeed, to preset tool-offsetting data designated with addresses H and D.

Controlled axis	Program command	Moving distance when program commands are executed			Program applicability (A) → (B)
		NC (A) for which the program was prepared	MAZATROL (B)		
			Bit 0 of F91 = 0	Bit 0 of F91 = 1	
Linear axis	X1 (Y1 / Z1)	1 μm	0.1 μm	1 μm	Applicable
Rotational axis	B1	0.001°	0.0001°	0.001°	Applicable

- NOTE -

3 DATA FORMATS

3-1 Tape Codes

This numerical control unit (in the remainder of this manual, referred to as the NC unit) uses command information that consists of letters of the alphabet (A, B, C Z), numerics (0, 1, 2 9), and signs (+, -, /, and so on). These alphanumerics and signs are referred to collectively as characters. On paper tape, these characters are represented as a combination of a maximum of eight punched holes.

Such a representation is referred to as a code.

The NC unit uses either the EIA codes (RS-244-A) or the ISO codes (R-840).

Note 1: Codes not included in the tape codes shown in Fig. 3-1 will result in an error when they are read.

Note 2: Of all codes specified as the ISO codes but not specified as the EIA codes, only the following codes can be designated using the data I/O (Tape) parameters **TAP9** to **TAP14**:

- [Bracket Open
-] Bracket Close
- # Sharp
- * Asterisk
- = Equal sign
- : Colon

However, you cannot designate codes that overlap existing ones or that result in parity error.

Note 3: EIA/ISO code identification is made automatically according to the first EOB/LF code appearing after the NC unit has been reset. (EOB: End Of Block, LF: Line Feed)

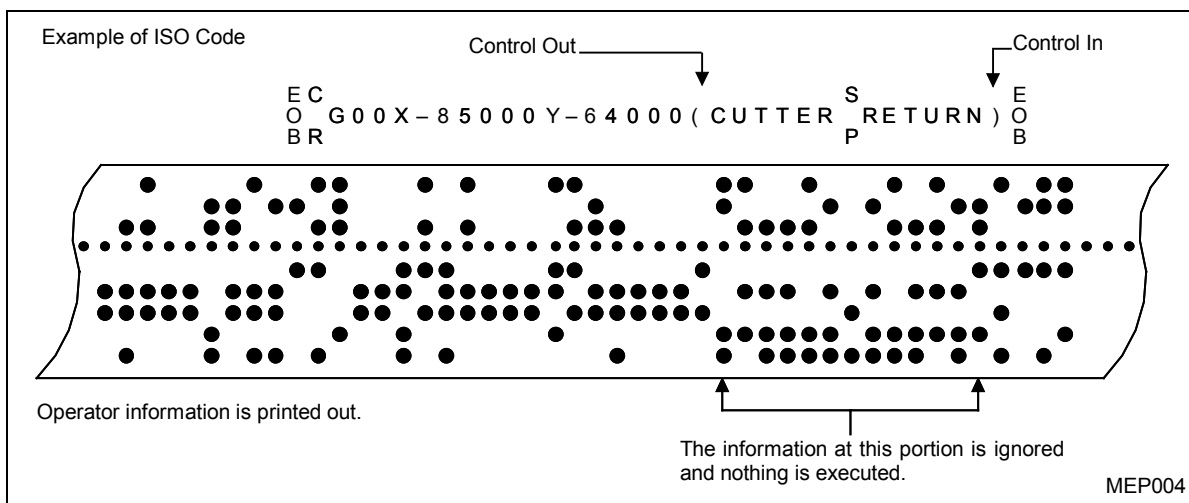
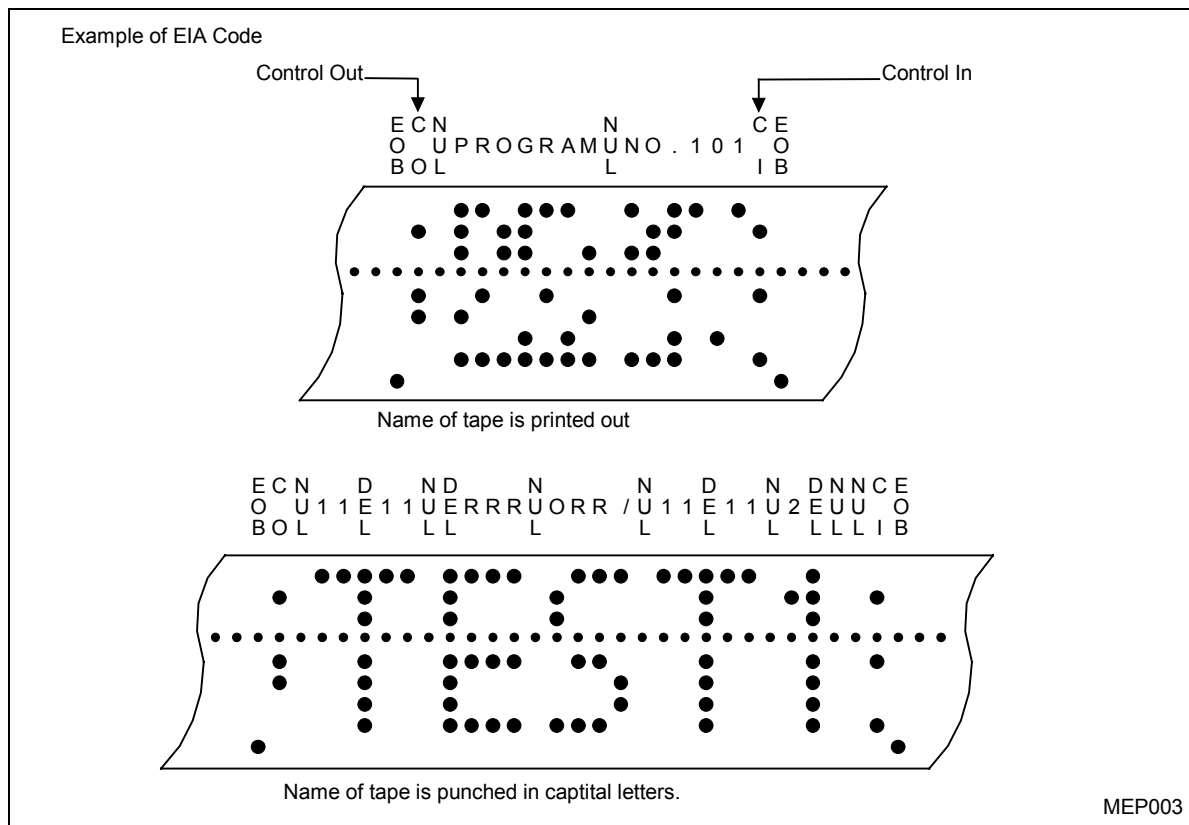
1. Significant information area (LABEL SKIP function)

During tape-based automatic operation, data storage into the memory, or data searching, the NC unit will ignore the entire information up to the first EOB code (;) in the tape when the unit is turned on or reset. That is, significant information in a tape refers to the information contained in the interval from the time a character or numeric code appears, following the first EOB code (;) after the NC unit has been reset, until a reset command is given.

2. Control Out, Control In

The entire information in the area from Control Out "(" to Control In ")" will be ignored in regard to machine control, while they will surely be displayed on the data display unit. Thus, this area can be used to contain information, such as the name and number of the command tape, that is not directly related to control.

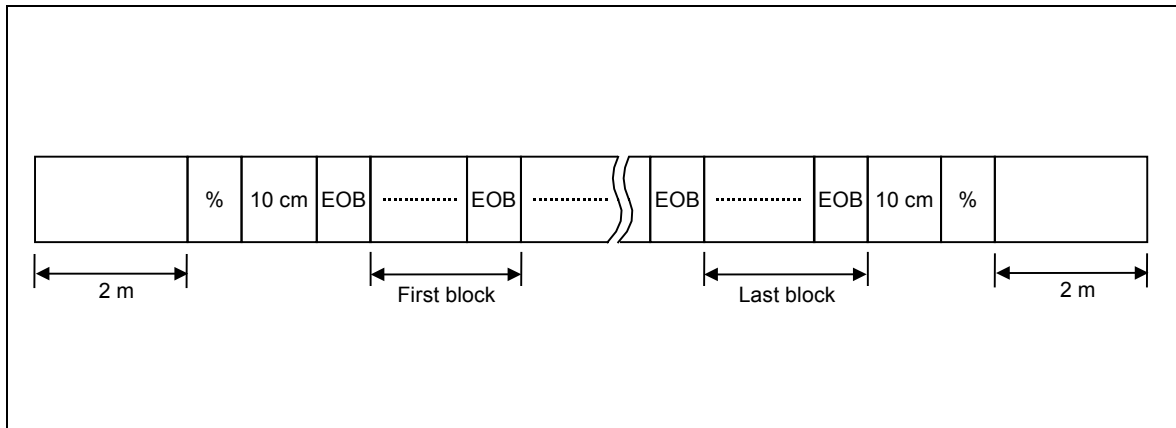
During tape storage, however, the information in this area will also be stored. The NC unit will enter the Control In status when power is turned on.



3. EOR code (%)

In general, the EOR (End Of Record) code is punched at both ends of a tape and has the following functions:

- To stop rewinding (only when a rewinding device is provided)
- To start rewinding during tape data search (only when a rewinding device is provided)
- To terminate the storage of tape data.

4. Tape creation method for tape operation (Only when a rewinding device is used)

The two meters of dummy at both ends and the EOR (%) at the head are not required when a rewinding device is not used.

EIA/ISO identification is made automatically by detecting whether EOB or LF initially appears after the NC unit has been reset.

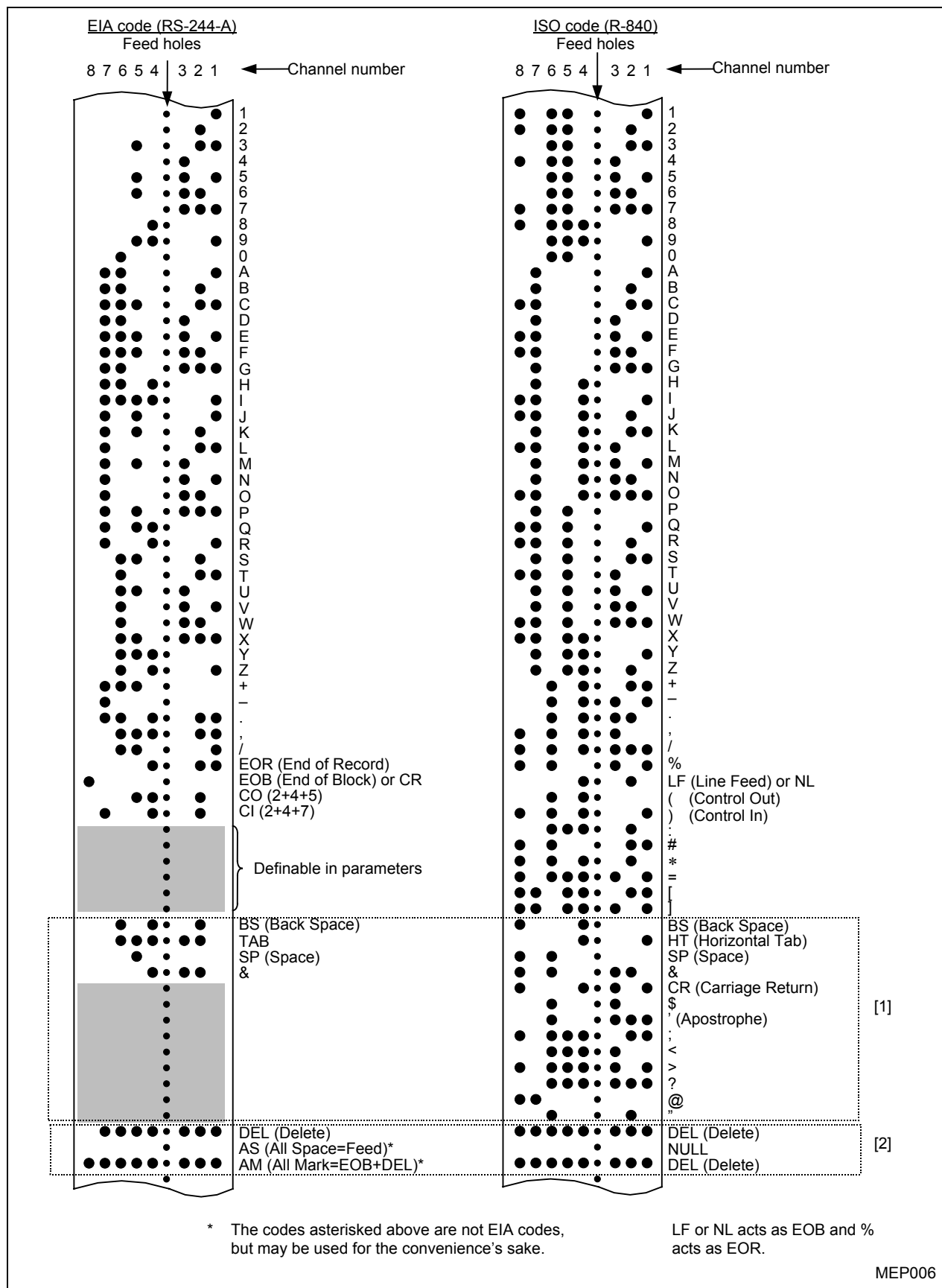


Fig. 3-1 Tape codes

Codes in section [1] will only be stored as tape data when they are present in a comment section, and ignored elsewhere in the significant information area.

Codes in section [2] are non-operative and will always be ignored (but undergo the parity-V check).

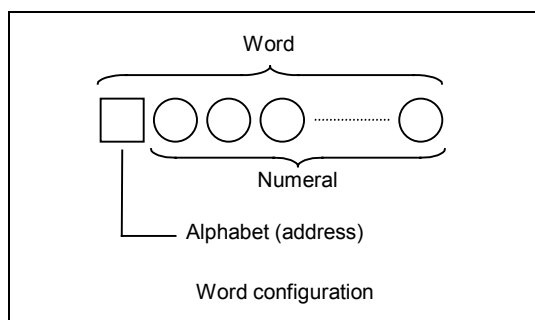
A dotted area indicates that the EIA Standard provides no corresponding codes.

3-2 Program Formats

A format predetermined for assigning control information to the NC unit is referred to as a program format. The program format used for our NC unit is word address format.

1. Words and addresses

A word is a set of characters arranged as shown below, and information is processed in words.



The alphabetic character at the beginning of a word is referred to as an address, which defines the meaning of its succeeding numeric information.

Table 3-1 Type and format of words

Item			Metric command	Inch command
Program No.			O8	
Sequence No.			N5	
Preparatory function			G3 or G21	
Moving axis	Input unit	0.0001 mm (deg.), 0.00001 in	X+54 Y+54 Z+54 α+54	X+45 Y+45 Z+45 α+45
Auxiliary axis		0.0001 mm (deg.), 0.00001 in	I+54 J+54 K+54	I+45 J+45 K+45
Dwell		0.001 mm (rev), 0.0001 in	X54 P8 U54	
Feed		0.0001 mm (deg.), 0.00001 in	F54 (per minute) F33 (per revolution)	F45 (per minute) F24 (per revolution)
Fixed cycle		0.0001 mm (deg.), 0.00001 in	R+54 Q54 P8 L4	R+45 Q45 P8 L4
Tool offset			H3 or D3	
Miscellaneous function			M3 × 4	
Spindle function			S5	
Tool function			T4 or T8	
No. 2 miscellaneous function			B8, A8 or C8	
Subprogram			P8 H5 L4	
Variables number			#5	

1. Code O8 here indicates that program number can be set as an unsigned integer of eight digits following O, and for X+54, "+" indicates that the value can be signed (negative) and the two-digit number (54) indicates that the decimal point can be used and that five digits before and four after the decimal point are effective (5 + 4 = 9 digits are effective for a designation without decimal point).
2. The alpha sign (α) denotes additional axis address. +44 will be used when α is specified for rotational axis.
3. The number of digits in the words is checked by the maximum number of digits in the addresses.
4. When data with decimal point is used for address for which decimal input is not available, decimal figures will be ignored.
5. If the number of integral digits exceeds the specified format, an alarm will result.
6. If the number of decimal digits exceed the specified format, the excess will be rounded.

2. Blocks

A block, unit of instruction, contains a number of words which constitute information necessary for the NC machine to perform an operation. The end of each block must be indicated by an EOB (End Of Block) code.

3. Programs

A number of blocks form one program.

4. Program end

M02, M30, M99, M998, M999 or % is used as program end code.

3-3 Tape Data Storage Format

As with tape operation, tape data to be stored into the memory can be either of ISO or EIA code. The first EOB code read in after resetting is used by the NC unit for automatic identification of the code system ISO or EIA.

The area of tape data to be stored into the memory is, if the NC unit has been reset, from the character immediately succeeding the first EOB code the EOR code, and in all other cases, from the current tape position to the EOR code. Usually, therefore, start tape data storage operation after resetting the NC unit.

3-4 Optional Block Skip

1. Function and purpose

Optional block skip is a function that selectively ignores that specific block within a machining program which begins with the slash code "/".

Any block beginning with "/" will be ignored if the **[BLOCK SKIP]** menu function is set to ON, or will be executed if the menu function is set to OFF.

For example, if all blocks are to be executed for a type of parts but specific blocks are not to be executed for another type, then different parts can be machined using one and the same program that contains the "/" code at the beginning of the specific blocks.

2. Operating notes

1. Blocks that have already been read into the pre-read buffer cannot be skipped.
2. This function is valid even during sequence number search.
3. During tape data storage (input) or output, all blocks, including those having a “/” code, are in- or outputted, irrespective of the status of the **[BLOCK SKIP]** menu function.

3-5 Program Number, Sequence Number and Block Number: O, N

Program numbers, sequence numbers, and block numbers are used to monitor the execution status of a machining program or to call a machining program or a specific process within a machining program.

Program numbers are assigned to command blocks as required. A program number must be set using the letter O (address) and a numeric of a maximum of eight digits that follow O.

Sequence numbers identify command blocks forming a machining program. A sequence number must be set using the letter N (address) and a numeric of a maximum of five digits that follow N. Block numbers are counted automatically within the NC unit, and reset to 0 each time a program number or a sequence number is read. These numbers will be counted up by one if the block to be read does not have an assigned program number or sequence number.

All blocks of a machining program, therefore, can be uniquely defined by combining program number, sequence number, and block number as shown in the table below.

NC input machining program	NC monitor display		
	Program No.	Sequence No.	Block No.
O1234 (DEMO. PROG)	1234	0	0
G92X0Y0	1234	0	1
G90G51X-150. P0.75	1234	0	2
N100G00X-50. Y-25.	1234	100	0
N110G01X250. F300	1234	110	0
Y-225.	1234	110	1
X-50.	1234	110	2
Y-25.	1234	110	3
N120G51Y-125. P0.5	1234	120	0
N130G00X-100. Y-75.	1234	130	0
N140G01X-200.	1234	140	0
Y-175.	1234	140	1
X-100.	1234	140	2
Y-75.	1234	140	3
N150G00G50X0Y0	1234	150	0
N160M02	1234	160	0
%			

3-6 Parity-H/V

One method of checking if the tape is correctly created is by parity checks. Parity checks are performed to check a tape for errors in punched codes, that is, for punching errors. There are two types of parity checks: parity-H and parity-V.

1. Parity-H check

Parity-H checks are intended to check the quantity of punched holes which form one character, and performed during tape operation, tape loading, and sequence-number searching.

A parity-H error occurs in the following cases:

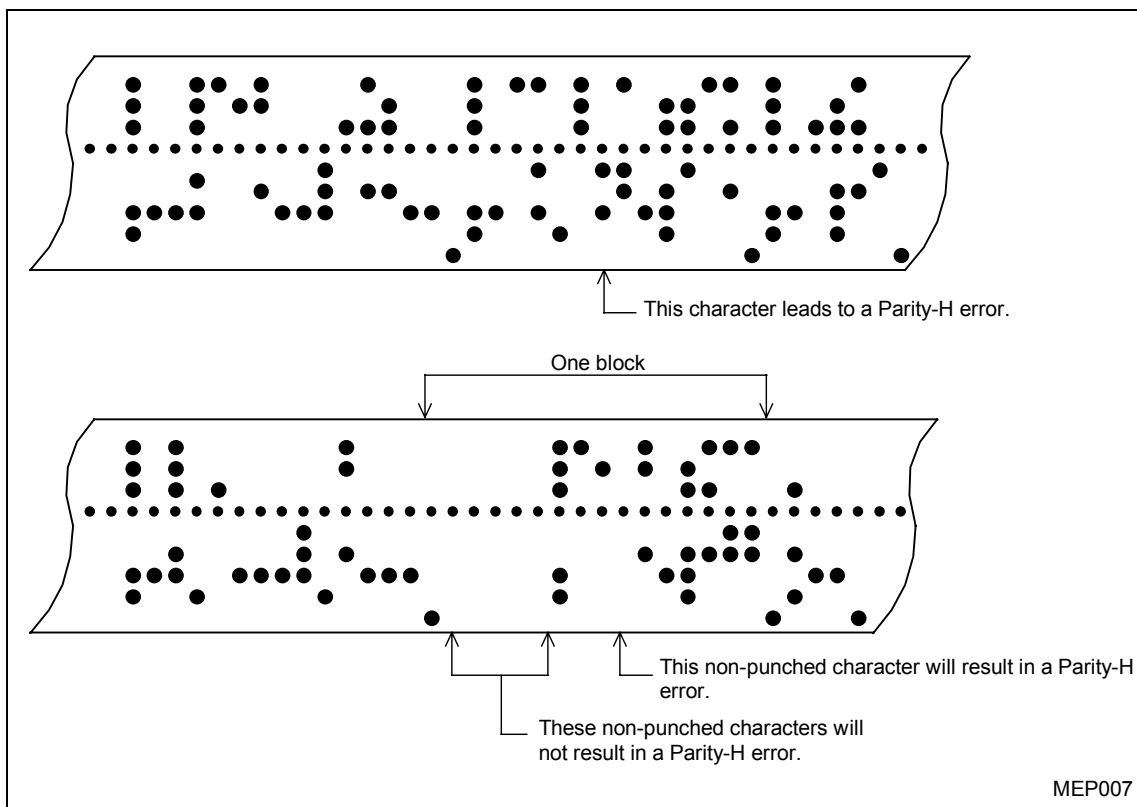
- ISO Codes

If a code with an odd number of punched holes is present in the significant information area.

- EIA Codes

If a code with an even number of punched holes is present in the significant information area or if non-punched holes (sprockets only) are present after a significant code in one block.

Example 1: Parity-H error (for EIA codes)



If a parity-H error occurs, the tape will stop at the position next to the error code.

2. Parity-V check

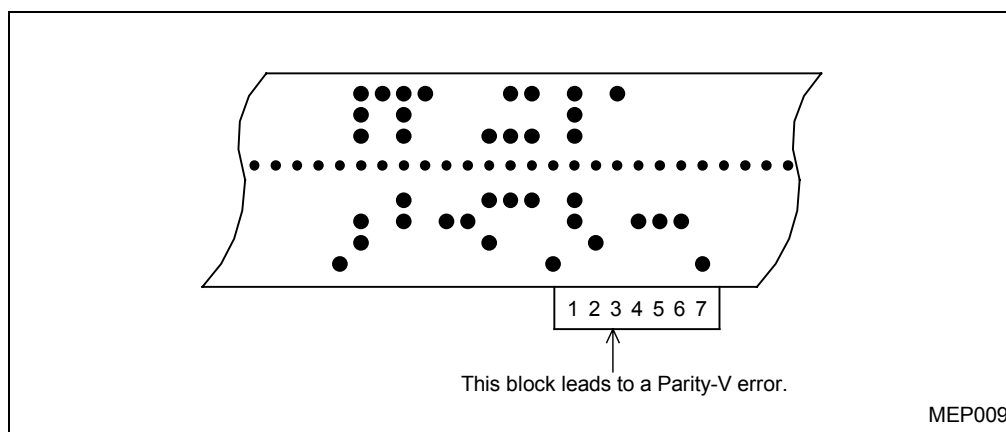
Parity-V checks will be performed during tape operation, tape loading, or sequence-number searching, if parity-V check item on the **PARAMETER** display is set to ON. Parity-V during memory operation, however, will not be checked.

A parity-V error occurs in the following case:

If an odd number of codes are present in the significant information area from the first significant code in the vertical direction to the EOB code (;), that is, if an odd number of characters are present in one block.

In the event of a parity-V error, the tape stops at a code next to the EOB (;).

Example 2: An example of parity-V error



Note 1: During a parity-V check, some types of code are not counted as characters. See Fig. 3-1, "Tape codes" for further details.

Note 2: Space codes in the area from the first EOB code to the first address code or slash code "/" are not subjected to counting for parity-V check.

3-7 List of G-Codes

G functions are described in the list below.

Function	G-code	Group
Positioning	■ G00	01
Linear interpolation	■ G01	01
Circular interpolation (CW)	G02	01
Circular interpolation (CCW)	G03	01
Spiral interpolation (CW)	G02.1	01
Spiral interpolation (CCW)	G03.1	01
Dwell	G04	00
High-speed machining mode	G05	00
Fine spline interpolation	G06.1	01
NURBS interpolation	G06.2	01
Virtual-axis interpolation	G07	00
Cylindrical interpolation	G07.1	00
Exact-stop check	G09	00
Data setting mode ON	G10	00
Command address OFF	G10.1	00
Data setting mode OFF	G11	00
X-Y plane selection	■ G17	02
Z-X plane selection	■ G18	02
Y-Z plane selection	■ G19	02
Inch command	■ G20	06
Metric command	■ G21	06
Pre-move stroke check ON	G22	04
Pre-move stroke check OFF	▲ G23	04
Reference point check	G27	00
Reference point return	G28	00
Return from reference point	G29	00
Return to 2nd, 3rd and 4th reference points	G30	00
Skip function	G31	00
Multi-step skip 1	G31.1	00
Multi-step skip 2	G31.2	00
Multi-step skip 3	G31.3	00
Thread cutting (straight, taper)	G33	01
Variable lead thread cutting	G34	01
Hole machining pattern cycle (on a circle)	G34.1	00
Hole machining pattern cycle (on a line)	G35	00
Hole machining pattern cycle (on an arc)	G36	00
Hole machining pattern cycle (on a grid)	G37.1	00
Automatic tool length measurement	G37	00
Vector selection for tool radius compensation	G38	00
Corner arc for tool radius compensation	G39	00
Tool radius compensation OFF	▲ G40	07
Tool radius compensation (left)	G41	07
Tool radius compensation for five-axis machining (left)	G41.2/G41.4/G41.5	07
Tool radius compensation (right)	G42	07
Tool radius compensation for five-axis machining (right)	G42.2/G42.4/G42.5	07
Tool length offset (+)	G43	08

Function	G-code	Group
Tool tip point control (Type 1) ON	G43.4	08
Tool tip point control (Type 2) ON	G43.5	08
Tool length offset (–)	G44	08
Tool position offset, extension	G45	00
Tool position offset, reduction	G46	00
Tool position offset, double extension	G47	00
Tool position offset, double reduction	G48	00
Tool position offset OFF	▲ G49	08
Scaling OFF	▲ G50	11
Scaling ON	G51	11
Mirror image OFF	▲ G50.1	19
Mirror image ON	G51.1	19
Local coordinate system setting	G52	00
Machine coordinate system selection	G53	00
Tool-axis direction control	G53.1	00
Selection of workpiece coordinate system 1	▲ G54	12
Selection of workpiece coordinate system 2	G55	12
Selection of workpiece coordinate system 3	G56	12
Selection of workpiece coordinate system 4	G57	12
Selection of workpiece coordinate system 5	G58	12
Selection of workpiece coordinate system 6	G59	12
Additional workpiece coordinate systems	G54.1	12
Selection of fixture offset	G54.2	23
Workpiece setup error correction	G54.4	27
One-way positioning	G60	00
Exact stop mode	G61	13
High-accuracy mode (Geometry compensation)	G61.1	13
Modal spline interpolation	G61.2	13
Automatic corner override	G62	13
Tapping mode	G63	13
Cutting mode	▲ G64	13
User macro single call	G65	00
User macro modal call A	G66	14
User macro modal call B	G66.1	14
User macro modal call OFF	▲ G67	14
Programmed coordinate rotation ON	G68	16
Programmed coordinate rotation OFF	G69	16
3-D coordinate conversion ON	G68	16
Inclined-plane machining ON	G68.2	16
Inclined-plane machining (by specifying tool-axis direction) ON	G68.3	16
Incremental coordinate system establishment for inclined-plane machining	G68.4	16
3-D coordinate conversion OFF	▲ G69	16
Fixed cycle (Chamfering cutter 1, CW)	G71.1	09
Fixed cycle (Chamfering cutter 2, CCW)	G72.1	09
Fixed cycle (High-speed deep-hole drilling)	G73	09
Fixed cycle (Reverse tapping)	G74	09
Fixed cycle (Boring 1)	G75	09
Fixed cycle (Boring 2)	G76	09
Fixed cycle (Back spot facing)	G77	09

Function	G-code	Group
Fixed cycle (Boring 3)	G78	09
Fixed cycle (Boring 4)	G79	09
Fixed cycle OFF	▲ G80	09
Fixed cycle (Spot drilling)	G81	09
Chopping	G81.1	00
Fixed cycle (Drilling)	G82	09
Fixed cycle (Deep-hole drilling)	G83	09
Fixed cycle (Tapping)	G84	09
Fixed cycle (Synchronous tapping)	G84.2	09
Fixed cycle (Synchronous reverse tapping)	G84.3	09
Fixed cycle (Reaming)	G85	09
Fixed cycle (Boring 5)	G86	09
Fixed cycle (Back boring)	G87	09
Fixed cycle (Boring 6)	G88	09
Fixed cycle (Boring 7)	G89	09
Absolute data input	■ G90	03
Incremental data input	■ G91	03
Coordinate system setting/Spindle speed range setting	G92	00
Workpiece coordinate system rotation	G92.5	00
Inverse time feed	G93	05
Feed per minute (asynchronous)	■ G94	05
Feed per revolution (synchronous)	■ G95	05
Initial point level return in fixed cycles	▲ G98	10
R-point level return in fixed cycles	G99	10
Measurement macro, workpiece/coordinate measurement	G136	
Compensation macro	G137	

Notes:

1. The codes marked with ▲ are automatically selected in each group upon switching-on or resetting with initializing the modal conditions.
2. The codes marked with ■ are able to be selected by a parameter as an initial modal condition which is to become valid upon switching-on or resetting with initializing the modal conditions. Changeover of inch/metric system, however, can be made valid only by switching-on.
3. G-codes of group 00 are those which are not modal, and they are valid only for blocks in which they are entered.
4. If a G-code not given in the G-code list is used, an alarm is displayed. And if a G-code without corresponding option is used, an alarm is displayed (**808 MIS-SET G CODE**).
5. If G-codes belong to different groups each other, any G-code can be used in the same block. The G-codes are then processed in order of increasing group number. If two or more G-codes belonging to the same group are given in the same block, the G-code entered last is valid.

4 BUFFER REGISTERS

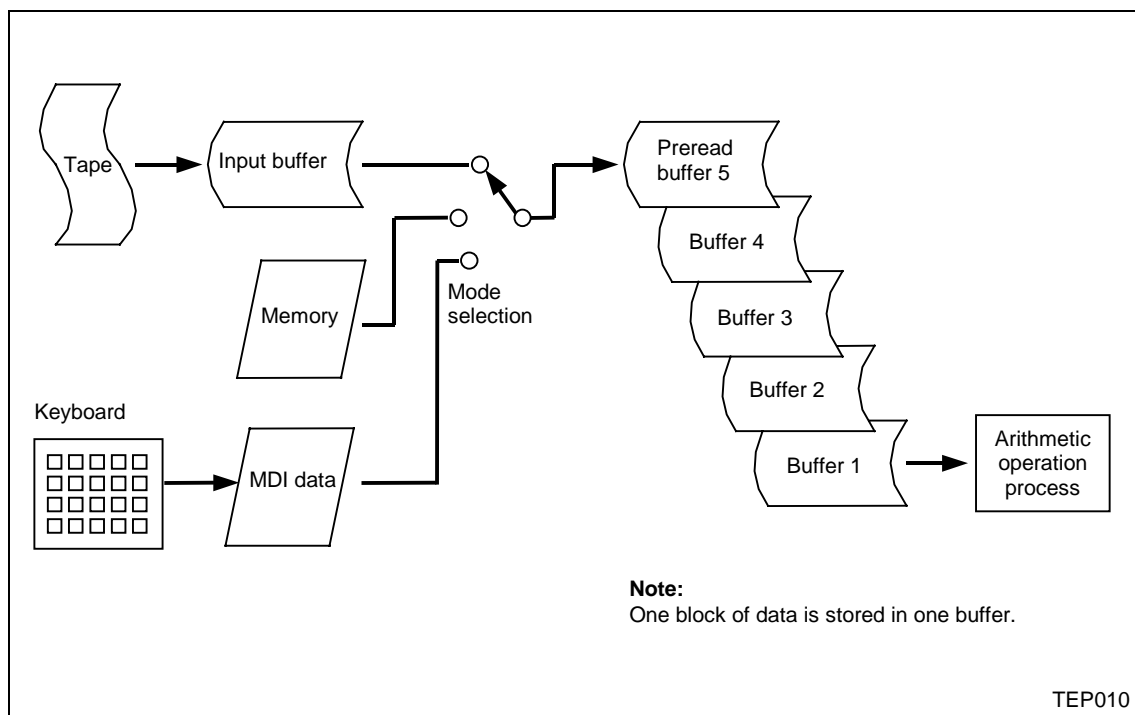
4-1 Input Buffer

1. Overview

During tape operation or RS-232C operation, when the preread buffer becomes empty, the contents of the input buffer will be immediately shifted into the pre-read buffer and, following this, if the memory capacity of the input buffer diminishes to 248×4 characters or less, next data (up to 248 characters) will be preread from the tape and then stored into the input buffer.

The input buffer makes block-to-block connections smooth by eliminating any operational delays due to the tape-reading time of the tape reader.

These favorable results of prereading, however, will be obtained only if the execution time of the block is longer than the tape-reading time of the next block.



2. Detailed description

- The memory capacity of the input buffer is 248×5 characters (including the EOB code).
- The contents of the input buffer register are updated in 248-character units.
- Only the significant codes in the significant information area are read into the buffer.
- Codes, including "(" and ")", that exist between Control Out and Control In, are read into the input buffer. Even if optional block skip is valid, codes from / to EOB will also be read into the input buffer.
- The contents of the buffer are cleared by a reset command.

4-2 Preread Buffer

1. Overview

During automatic operation, one block of data is usually preread to ensure smooth analysis of the program. During tool radius compensation, however, maximal 23 blocks of data are preread to calculate crossing point or to check the interference.

In the high-speed machining mode (G05P2), moreover, up to 8 blocks of data are preread, and in the mode of high-speed smoothing control up to 24 blocks of data are stored with the currently executed block in the middle (i. e. 12 blocks being preread).

2. Detailed description

- One block of data is stored into the prepared buffer.
- Only the significant codes in the significant information area are read into the pre-read buffer.
- Codes existing between Control Out and Control In are not read into the pre-read buffer. If optional block skip is valid, codes from / to EOB will not also be read into the pre-read buffer.
- The contents of the buffer are cleared by a reset command.
- If the single block operation mode is selected during continuous operation, processing will stop after pre-reading the next block data.

5 POSITION PROGRAMMING

5-1 Dimensional Data Input Method

5-1-1 Absolute/Incremental data input: G90/G91

1. Function and purpose

Setting of G90 or G91 allows succeeding dimensional data to be processed as absolute data or incremental data.

Setting of arc radius (with address R) or arc center position (with addresses I, J, K) for circular interpolation, however, must always refer to incremental data input, irrespective of preceding G90 command.

2. Programming format

G90 (or G91) $Xx_1 Yy_1 Zz_1 \alpha\alpha_1$ (α : Additional axis)

where G90: Absolute data input

G91: Incremental data input

3. Detailed description

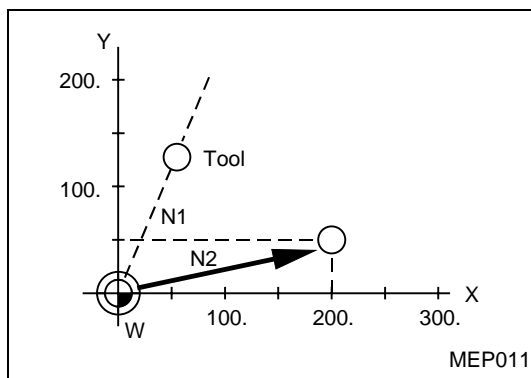
1. In the absolute data mode, axis movement will be performed to the program-designated position within the workpiece coordinate system, irrespective of the current position.

```
N1 G90G00X0 Y0
```

In the incremental data mode, axis movement will be performed through the program-designated distance as relative data with respect to the current position.

```
N2 G91G01X200. Y50. F100
```

```
N2 G90G01X200. Y50. F100
```



Commands for a movement from the origin of the workpiece coordinate system are given with the same values, irrespective of whether the absolute data mode or the incremental data mode is used.

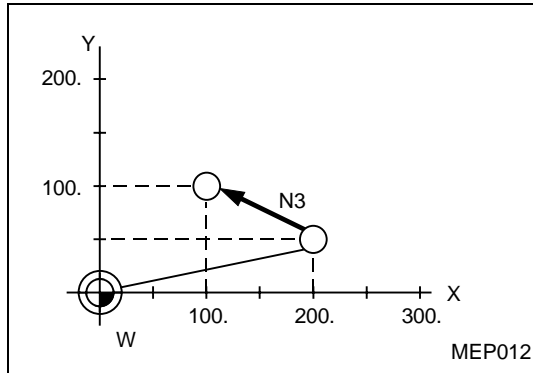
2. The last G90 or G91 command works as a modal one for the following blocks.

(G90) N3 X100. Y100.

This block will perform a movement to the position of $X = 100$ and $Y = 100$ in the workpiece coordinate system.

(G91) N3 X-100. Y50.

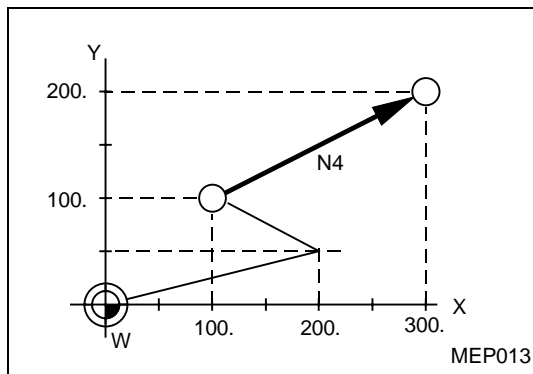
This block will perform a movement of -100 on the X-axis and $+50$ on the Y-axis, and thus result in a movement to the position of $X = 100$ and $Y = 100$.



3. Multiple G90 or G91 commands can be set in one block, and thus only a specific address can be set as absolute data or incremental data.

N4 G90X300. G91Y100.

In this example, dimensional data X300 preceded by G90 will be processed as an absolute data input, and Y100 preceded by G91 as an incremental data input. Therefore, this block will result in a movement to the position of $X = 300$ and $Y = 200$ ($100 + 100$) in the workpiece coordinate system.



Moreover, G91 (incremental data input mode) will work for the succeeding blocks.

4. Either the absolute data mode or the incremental data mode can be freely selected as initial mode by setting the bit 2 of user parameter **F93**.
5. Even in the MDI (Manual Data Input) mode, G90 and G91 will also be handled as modal commands.

5-2 Inch/Metric Selection: G20/G21

1. Function and purpose

Inch command/metric command selection is possible with G-code commands.

2. Programming format

G20: Inch command selection

G21: Metric command selection

3. Detailed description

1. Changeover between G20 and G21 is effective only for linear axes; it is meaningless for rotational axes.

Example: Preset unit of data input and G20/G21 (for decimal-point input I)

Axis	Example	Initial Inch (parameter) OFF		Initial Inch (parameter) ON	
		G21	G20	G21	G20
X	X100	0.0100 mm	0.0254 mm	0.00039 inches	0.00100 inches
Y	Y100	0.0100 mm	0.0254 mm	0.00039 inches	0.00100 inches
Z	Z100	0.0100 mm	0.0254 mm	0.00039 inches	0.00100 inches
B	B100	0.0100 deg	0.0100 deg	0.0100 deg	0.0100 deg

2. To perform G20/G21 changeover in a program, you must first convert variables, parameters, and offsetting data (such as tool length/tool position/tool radius compensation data) according to the unit of data input for the desired system (inch or metric) and then set all these types of data either on each data setting display or using the programmed parameter input function.

Example: If Initial inch selection is OFF and offsetting data is 0.05 mm, the offsetting data must be converted to 0.002 ($0.05 \div 25.4 \approx 0.002$) before changing the G21 mode over to the G20 mode.

3. In principle, G20/G21 selection should be done before machining. If you want this changeover to be performed in the middle of the program, temporarily stop the program by an M00 command after G20 or G21 and convert the offsetting data as required.

Example: G21 G92 XX₁ Y₁ ZZ₁

⋮

G20 G92 XX₂ Y₂ ZZ₂

M00 → Convert offsetting data here.

⋮

F10 → Set an F (Feed rate) command anew.

Note: Do not fail to give an F command appropriate to the new unit system after changeover between G20 and G21. Otherwise, axis movements would be performed using the last F value before the changeover, without any conversion, on the basis of the new unit system.

4. Whether G20 or G21 is to be selected upon switching-on can be specified by the bit 4 of user parameter **F91** (Initial Inch parameter).

5-3 Decimal Point Input

1. Function and purpose

The decimal point can be used to determine the units digit (mm or inch) of dimensional data or feed rate.

2. Programming format

○○○○○.○○○○ Metric system

○○○○.○○○○○ Inch system

3. Detailed description

- Decimal-point commands are valid only for the distance, angle, time, speed, and scaling factor (only after G51) that have been set in the machining program.
- As listed in the table below, the meaning of command data without the decimal point differs between decimal-point input types I and II according to the type of command unit system.

Command	Command unit × 10	Type I	Type II
x1	OFF	0.0001 (mm, inches, deg)	1.0000 (mm, inches, deg)
	ON	0.0010 (mm, inches, deg)	1.0000 (mm, inches, deg)

- Decimal-point commands are only valid for addresses X, Y, Z, U, V, W, A, B, C, I, J, K, E, F, P, Q and R, where address P only refers to a scaling factor.
- The number of effective digits for each type of decimal-point command is as follows:

	Move command (Linear)		Move command (Rotational)		Feed rate		Dwell	
	Integral part	Decimal part	Integral part	Decimal part	Integral part	Decimal part	Integral part	Decimal part
mm	0. - 99999.	.0000 - .9999	0. - 99999.	.0000 - .9999	0. - 200000.	.0000 - .9999	0. - 99999.	.000 - .999
inch	0. - 9999.	.00000 - .99999	0. - 99999. (359.)	.0000 - .9999	0. - 20000.	.00000 - .99999	0. - 99999.	.000 - .999

- Decimal-point commands are also valid for definition of variables data used in subprograms.
- For data which can be, but is not specified with the decimal point, either the minimum program data input unit or mm (or in) unit can be selected using bit 5 of parameter **F91**.
- A decimal-point command issued for an address which does not accept the decimal point will be processed as data that consists of an integral part only. That is, all decimal digits will be ignored. Addresses that do not accept the decimal point are D, H, L, M, N, O, S and T. All types of variables command data are handled as the data having the decimal point.

4. Sample programs

A. Sample programs for addresses accepting the decimal point

Command category Program example	For 1 = 1 μm	For 1 = 0.1 μm	1 = 1 mm
G0X123.45 (With the decimal point always given as the millimeter point)	X123.450 mm	X123.450 mm	X123.450 mm
G0X12345	X12.345 mm*	X1.2345 mm**	X12345.000 mm***
#111=123 #112=5.55 X#111 Y#112	X123.000 mm Y5.550 mm		
#113=#111+#112 (ADD)	#113 = 128.550		
#114=#111-#112 (SUBTRACT)	#114 = 117.450		
#115=#111.#112 (MULTIPLY)	#115 = 682.650		
#116=#111/#112 #117=#112/#111 (DIVIDE)	#116 = 22.162 #117 = 0.045		

* The least significant digit is given in 1 μm .

** The least significant digit is given in 0.1 μm .

*** The least significant digit is given in 1 mm.

B. Validity of decimal point for each address

Address	Decimal point command	Application	Remarks	Address	Decimal point command	Application	Remarks
A	Valid	Coordinate position data		O	Invalid	Program number	
	Invalid	Rotary table Miscellaneous function code		P	Invalid	Dwell time	
	Valid	Linear angle data			Valid	Subprogram call number	
B	Valid	Coordinate position data			Invalid	Number of helical pitches	
	Invalid	Rotary table Miscellaneous function code			Invalid	Offset amount (in G10)	
C	Valid	Coordinate position data			Valid	Scaling factor	
	Invalid	Rotary table Miscellaneous function code			Valid	Rank for NURBS curve	
	Valid	Corner chamfering amount		Q	Valid	Cutting depth for deep-hole drilling cycle	
D	Invalid	Offset number (tool position, tool length and tool radius)			Valid	Shift amount for back boring	
E	Valid				Valid	Shift amount for fine boring	
F	Valid	Rate of feed		R	Valid	R point in fixed cycle	
G	Valid	Preparatory function code			Valid	Radius of an arc for circular interpolation	
H	Invalid	Offset number (tool position, tool length and tool radius)			Valid	Radius of an arc for corner rounding	
	Invalid	Intra-subprogram sequence number			Valid	Offset amount (in G10)	
I	Valid	Coordinate of arc center			Valid	Weight for NURBS curve	
	Valid	Vector component for tool radius compensation		S	Invalid	Spindle function code	
J	Valid	Coordinate of arc center		T	Invalid	Tool function code	
	Valid	Vector component for tool radius compensation		U	Valid	Coordinate position data	
K	Valid	Coordinate of arc center		V	Valid	Coordinate position data	
	Valid	Vector component for tool radius compensation		W	Valid	Coordinate position data	
	Valid	Knot for NURBS curve		X	Valid	Coordinate position data	
L	Invalid	Fixed cycle/Subprogram repetition			Valid	Dwell time	
M	Invalid	Miscellaneous function code		Y	Valid	Coordinate position data	
N	Invalid	Sequence number		Z	Valid	Coordinate position data	

Note: The decimal point is valid in all the arguments for a user macroprogram.

6 INTERPOLATION FUNCTIONS

6-1 Positioning (Rapid Traverse): G00

1. Function and purpose

Positioning command G00 involves use of a coordinate word. This command positions a tool by moving it linearly to the ending point specified by a coordinate word.

2. Programming format

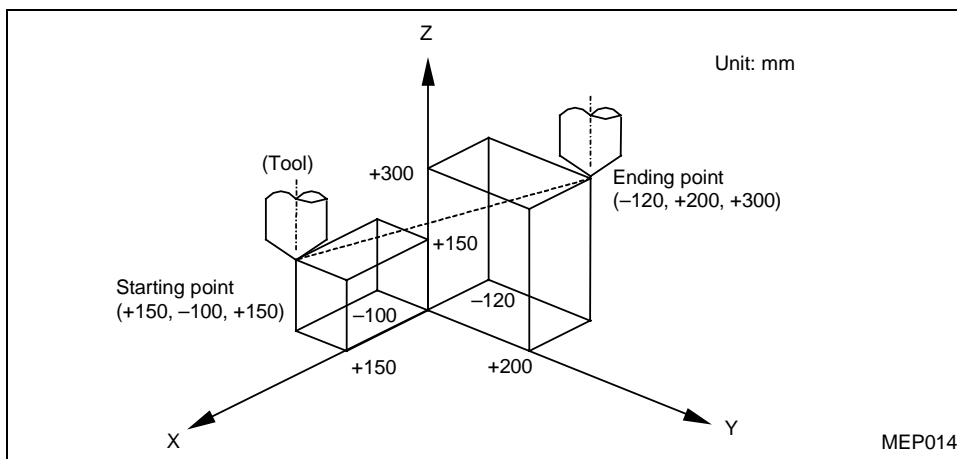
G00 Xx Yy Zz $\alpha\alpha$; (α : Additional axis)

The command addresses are valid for all additional axis. The absolute or the incremental data input is used according to the status of G90/G91 existing at the particular time.

3. Detailed description

1. Once this command has been given, the G00 mode will be retained until any other G-code command that overrides this mode, that is, either G01, G02, G03, or G32 of command group 01 is given. Thus, a coordinate word will only need be given if the next command is also G00. This function is referred to as the modal function of the command.
2. In the G00 mode, acceleration/deceleration always takes place at the starting/ending point of a block and the program proceeds to the next block after confirming that the pulse command in the present block is 0 and the tracking error of the acceleration/deceleration cycle is 0. The width of in-position can be changed using a parameter (**S13**).
3. The G-code functions (G71.1 to G89) of command group 09 are canceled by the G00 command (G80).
4. The tool path can be made either linear or nonlinear using a parameter (**F91** bit 6) but the positioning time remains unchanged.
 - Linear path
As with linear interpolation (G01), the tool speed is limited according to the rate of rapid traverse of each axis.
 - Nonlinear path
The tool is positioned according to the separate rate of rapid traverse of each axis.
5. When no number follows G address, this is treated as G00.

4. Sample programs



The diagram above is for:

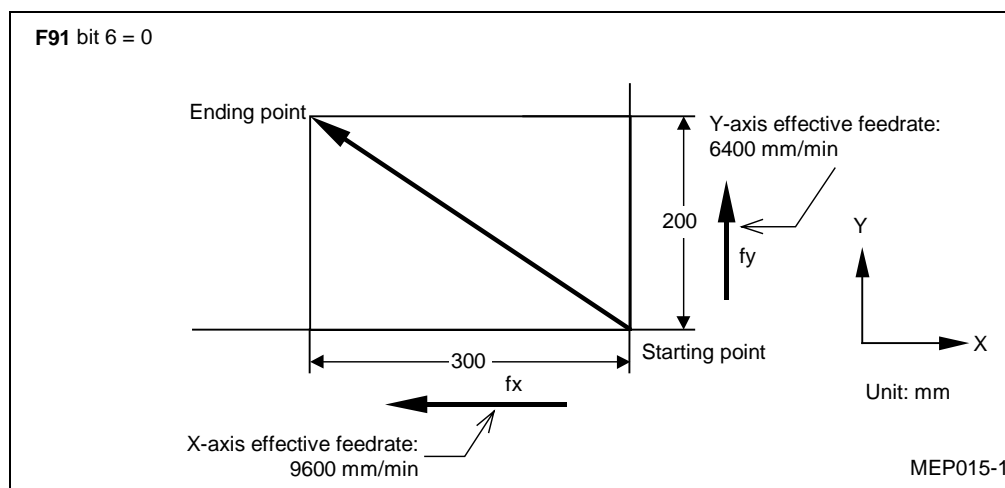
```
G90 G00 X-120.000 Y200.000 Z300.000; Absolute data command
G91 G00 X-270.000 Y300.000 Z150.000; Incremental data command
```

5. Remarks

- If bit 6 of user parameter **F91** is 0, the tool will take the shortest path connecting the starting and ending points. The positioning speed will be calculated automatically to give the shortest allocation time within the limits of the rapid feed rate of each axis. For example, if a rapid feed rate of 9600 mm/min is preset for both X- and Y-axes, then the command

```
G91 G00 X-300.000 Y200.000
```

will move the tool as shown in the figure below.

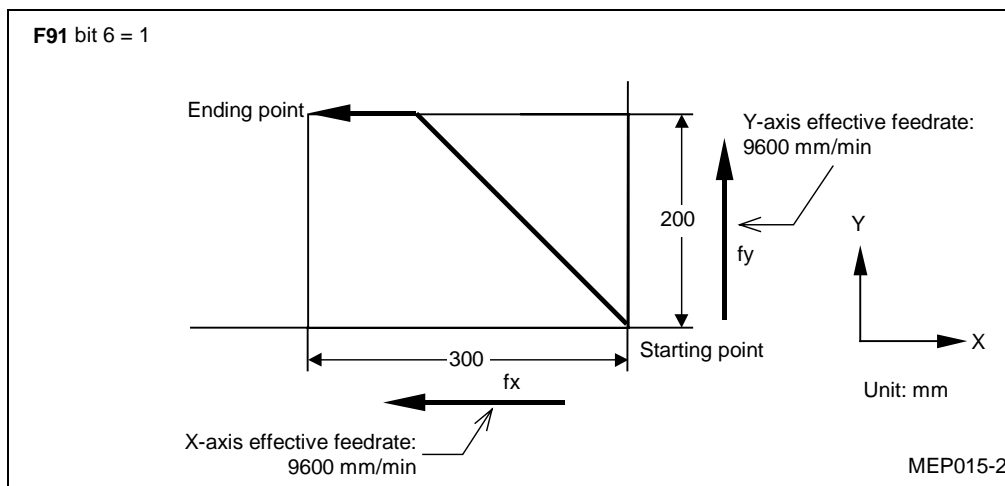


2. If bit 6 of user parameter **F91** is 1, the tool will move from the starting point to the ending point according to the rapid feed rate of each axis.

For example, if a rapid feed rate of 9600 mm/min is preset for both X- and Y-axes, then the command

```
G91 G00 X-300.000 Y200.000
```

will move the tool as shown in the figure below.



3. The rapid feed rate that you can set for each axis using the G00 command varies from machine to machine. Refer to the relevant machine specification for further details.

4. Rapid feed (G00) deceleration check

When processing of rapid feed (G00) is completed, the next block will be executed after the deceleration check time (T_d) has passed.

The deceleration check time (T_d) is calculated by following expressions depending on the acceleration/deceleration type.

Linear acceleration/linear deceleration $T_d = T_s + \alpha$

Exponential acceleration/linear deceleration $T_d = 2 \times T_s + \alpha$

Exponential acceleration/exponential deceleration $T_d = 2 \times T_s + \alpha$

(Where T_s is the acceleration time constant, $\alpha = 0$ to 14 ms)

The time required for the deceleration check during rapid feed is the longest among the rapid feed deceleration check times of each axis determined by the rapid feed acceleration/deceleration time constants and by the rapid feed acceleration/deceleration mode of the axes commanded simultaneously.

6-2 One-Way Positioning: G60

1. Function and purpose

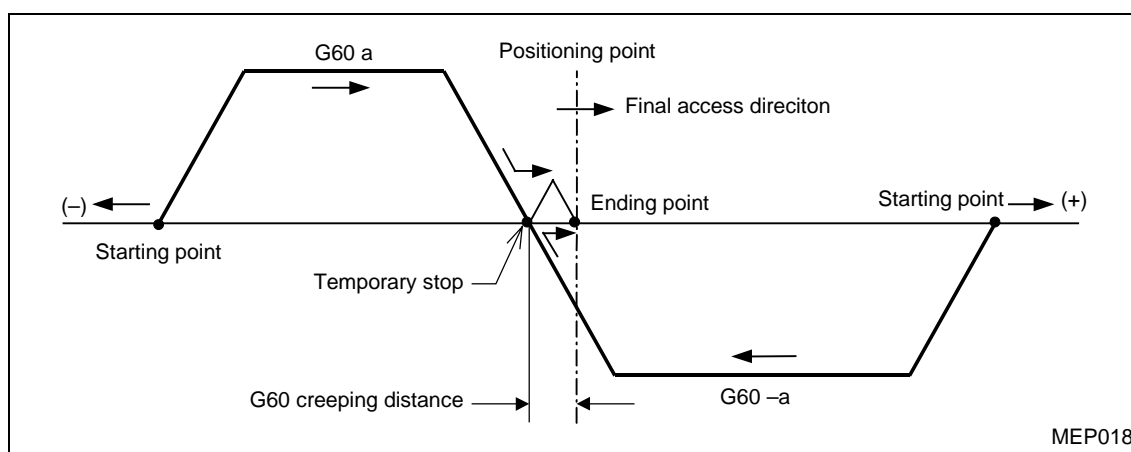
Highly accurate positioning free from any backlash error can be performed when the axis movement is controlled by the G60 command so that the final access always takes place in one determined direction.

2. Programming format

G60 Xx Yy Zz $\alpha\alpha$; (α : Additional axis)

3. Detailed description

1. The direction of final access and its creeping distance must be set in parameter I1.
2. After rapid approach to a position away from the ending point by the creeping distance, the final access is performed in the predetermined direction at a speed corresponding with the rapid feed.



3. The positioning pattern described above also applies during machine locking or for a Z-axis command with the Z-axis cancellation activated.
4. In the dry run mode (G00 mode), the whole positioning is carried out at the dry-running speed.
5. The creeping to the ending point can be halted with Reset, Emergency stop, Interlock, or Feed hold, or by setting the rapid feed override to 0 (zero).
The creeping is performed according to the setting of the rapid feed, and the rapid feed override function is also effective for the creeping.
6. One-way positioning is automatically invalidated for the hole-drilling axis in hole-drilling fixed-cycle operations.
7. One-way positioning is automatically invalidated for shifting in fine-boring or back-boring fixed-cycle operations.
8. Usual positioning is performed for an axis not having a parameter-set creeping distance.
9. One-way positioning is always of non-interpolation type.
10. An axis movement command for the same position as the ending point of the preceding block (movement distance = 0) will cause reciprocation through the creeping distance so that the final access can be performed in the predetermined direction for an accurate positioning to the desired point.

6-3 Linear Interpolation: G01

1. Function and purpose

This command moves (interpolates) a tool from the current position to the ending point specified by a coordinate word, at the feed rate specified with address F. The specified feed rate acts here as the linear velocity relative to the direction of movement of the tool center.

2. Programming format

G01 Xx Yy Zz $\alpha\alpha$ Ff; (α : Additional axis)

where x, y, z, and α each denote a coordinate. The absolute or the incremental data input is used according to the status of G90/G91 existing at the particular time.

3. Detailed description

Once this command has been given, the G01 mode will be retained until any other G-code command that overrides this mode, that is, either G00, G02, G03 or G33 of command group 01 is given. Thus, it is merely required to input coordinate words for linear interpolations in the succeeding blocks unless the feed rate must be changed.

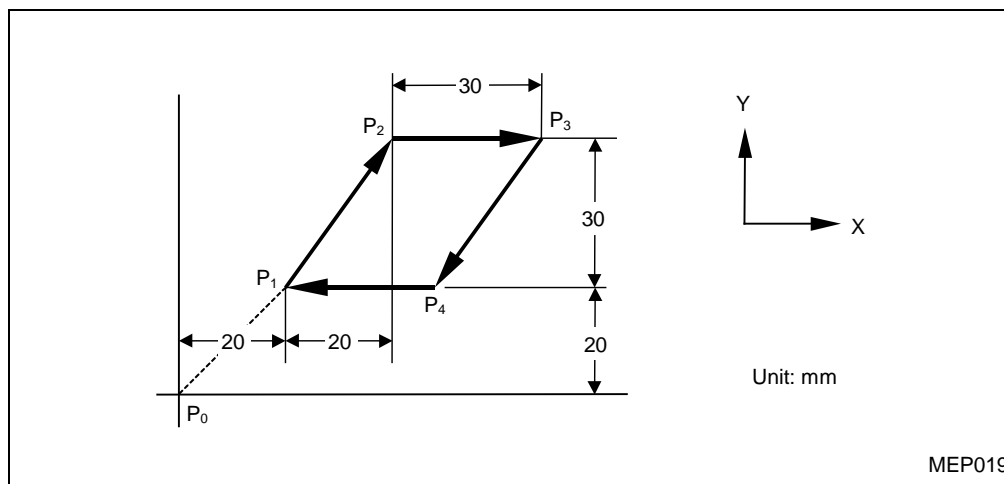
An alarm (**FEEDRATE ZERO**) will result if no F-code command has been given to the first G01 command.

The feed rates for rotational axes must be set in deg/min. (Example: F300 = 300 deg/min)

The G-code functions (G71.1 to G89) of command group 09 are cancelled by G01 (set to G80).

4. Sample program

The following shows a program for moving the tool at a cutting feed rate of 300 mm/min on the route of $P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow P_4 \rightarrow P_1$ (where the section from $P_0 \rightarrow P_1$ forms a positioning route for the tool):



G91	G00	X20.	Y20.	$P_0 \rightarrow P_1$
	G01	X20.	Y30. F300	$P_1 \rightarrow P_2$
		X30.		$P_2 \rightarrow P_3$
		X-20.	Y-30.	$P_3 \rightarrow P_4$
		X-30.		$P_4 \rightarrow P_1$

6-4 Circular Interpolation: G02, G03

1. Function and purpose

Commands G02 and G03 feed the tool along an arc.

2. Programming format

G02 (**G03**) Xx Yy (Zz) Ii Jj (Kk) Ff ;
Coordinates of the ending point Coordinates of the arc center Feedrate
Counter-clockwise (CCW)
Clockwise (CW)

X : Arc ending point coordinate, X-axis
Y : Arc ending point coordinate, Y-axis
Z : Arc ending point coordinate, Z-axis
I : Arc center, X-axis
J : Arc center, Y-axis
K : Arc center, Z-axis
F : Feed rate

Use addresses X, Y and Z (or their parallel axes) to specify the coordinates of the ending point of arc, and addresses I, J and K for the coordinates of arc center.

Combined use of absolute and incremental data input is available for setting the coordinates of the ending point of arc. For the coordinates of the arc center, however, incremental data relative to the starting point must always be set.

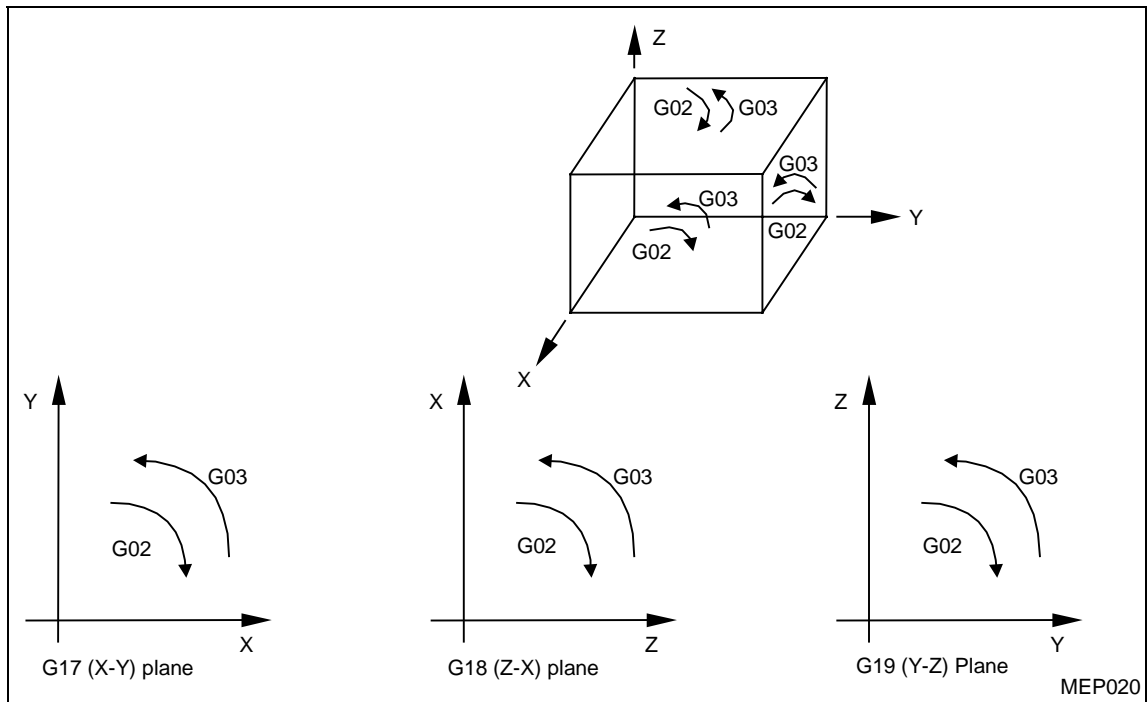
3. Detailed description

- Once the G02 (or G03) command has been given, this command mode will be retained until any other G-code command used to override the G02 (or G03) command mode, that is, G00 or G01 of command group 01 is given.

2. The direction of circular movement is determined by G02/G03.

G02: CW (Clockwise)

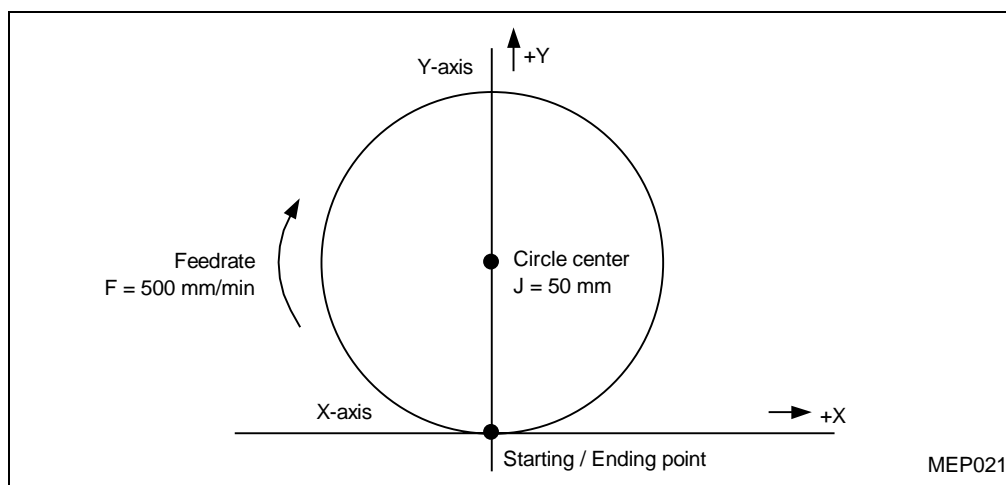
G03: CCW (Counterclockwise)



3. Interpolation of an arc that spans multiple quadrants can be defined with one block.
4. To perform circular interpolation, the following information is required:
- Rotational direction CW (G02) or CCW (G03)
 - Arc ending point coordinates Given with address X, Y, Z.
 - Arc center coordinates Given with address I, J, K. (Incremental dimension)
 - Feed rate Given with address F.
5. If none of the addresses I, J, K and R is specified, an alarm will occur.
6. Addresses I, J and K are used to specify the coordinates of the arc center in the X, Y and Z directions respectively as seen from the starting point, therefore, care must be taken for signs.

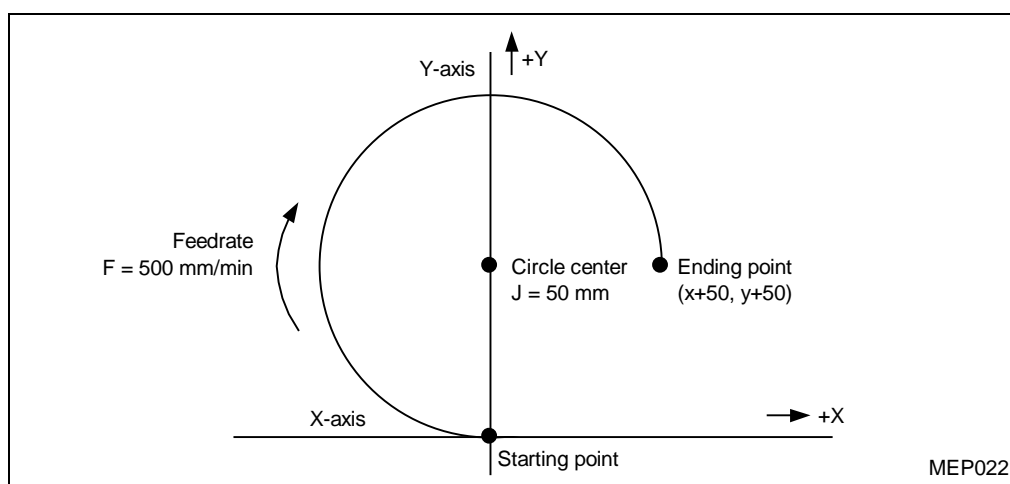
4. Sample programs

Example 1: Complete-circle command



G02 J50.000 F500

Example 2: Three-quarter circle command

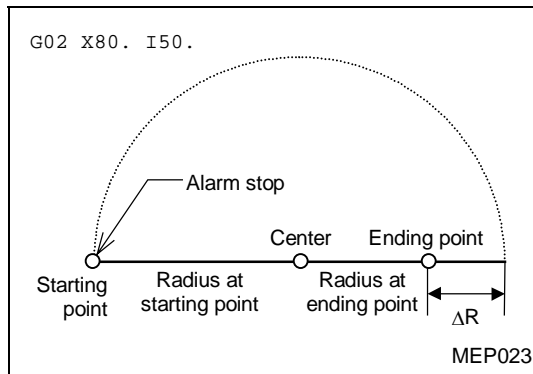


G91 G02 X50.000 Y50.000 J50.000 F500

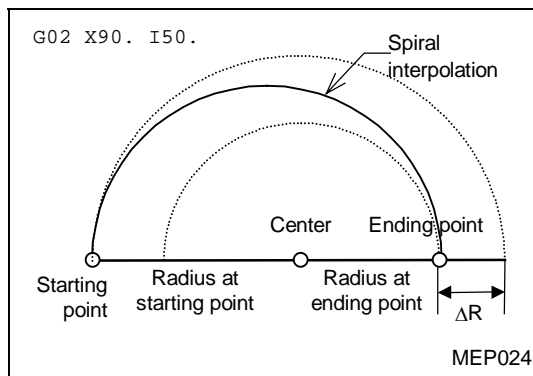
5. Notes on circular interpolation

1. Clockwise (G02) or Counterclockwise (G03) during circular interpolation refers to the rotational direction in the right-handed coordinate system when seen from the plus side toward the minus side of the coordinate axis perpendicular to the plane to be interpolated.
2. If the coordinates of the ending point are not set or if the starting and ending points are set at the same position, designating the center using address I, K or J will result in an arc of 360 degrees (true circle).

3. The following will result if the starting-point radius and the ending-point radius are not the same.
- If error ΔR is larger than the parameter **F19** (tolerance for radial value difference at ending point), an alarm (**817 INCORRECT ARC DATA**) will occur at the starting point of the arc.



- If error ΔR is equal to or smaller than the parameter data, interpolation will take a spiral form heading for the programmed ending point of the arc.



The examples shown above assume that excessively large parameter data is given to facilitate your understanding.

6-5 Radius Designated Circular Interpolation: G02, G03

1. Function and purpose

Circular interpolation can be performed by designating directly the arc radius R as well as specifying conventional arc center coordinates (I, J, K).

2. Programming format

G02 (G03) Xx Yy Rr Ff;

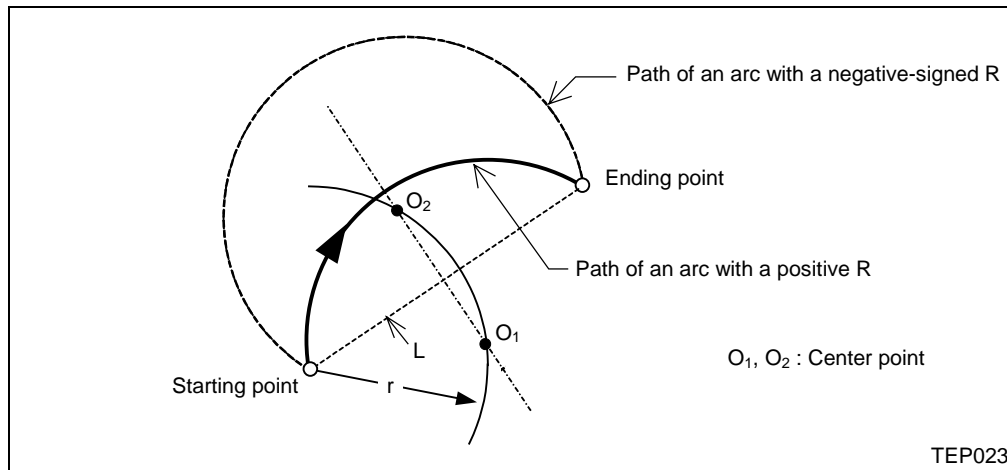
where x : X-axis coordinate of the ending point
 y : Y-axis coordinate of the ending point
 r : Radius of the arc
 f : Feed rate

3. Detailed description

The arc center is present on the mid-perpendicular to the segment which connects the starting point and the ending point. The crossing point of the mid-perpendicular and that circle of the designated radius r that has the center set at the starting point gives the center coordinates of the designated arc.

A semi-circle or smaller will be generated if R is a positive value.

An arc larger than the semi-circle will be generated if R is a negative value.



To use the radius-designated arc interpolation commands, the following requirement must be met:

$$\frac{L}{2 \cdot r} \leq 1$$

where L denotes the length of the line from the starting point to the ending point.

If radius data and arc center data (I, J, K) are both set in the same block, the circular interpolation by radius designation will have priority in general.

For complete-circle interpolation (the ending point = the starting point), however, use center-designation method with addresses I, J and K , since the radius-specification command in this case will immediately be completed without any machine operation.

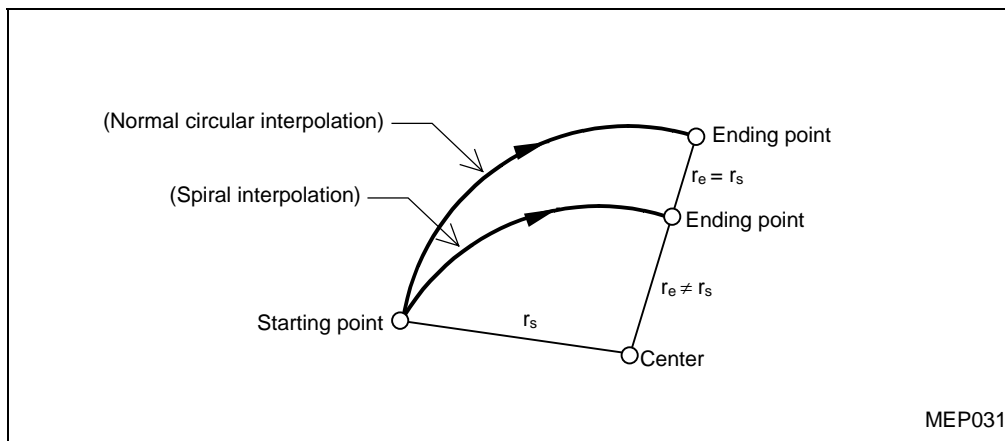
4. Sample programs

- G02 XX₁ YY₁ Rr₁ Ff₁ XY-plane, radius-designated arc
- G03 ZZ₁ XX₁ Rr₁ Ff₁ ZX-plane, radius-designated arc
- G02 XX₁ YY₁ Jj₁ Rr₁ Ff₁ XY-plane, radius-designated arc
(If radius data and center data (I, J, K) are set in the same block, circular interpolation by radius designation will have priority.)
- G17 G02 Ii₁ Ji₁ Rr₁ Ff₁ XY-plane, center-designated arc
(Radius-specification is invalid for complete circle)

6-6 Spiral Interpolation: G2.1, G3.1 (Option)

1. Function and purpose

Commands G2.1 and G3.1 provide such an interpolation that the starting and ending points are connected smoothly for an arc command where the radii of the both points differ from each other.



2. Programming format

G17 G2.1 (or G3.1) $\underbrace{X_p \ Y_p}_{\text{Arc ending point coordinates}} \underbrace{I \ J}_{\text{Arc center coordinates}} (\alpha) F \ P$

G18 G2.1 (or G3.1) $Z_p \ X_p \ K \ I (\alpha) F \ P$

G19 G2.1 (or G3.1) $Y_p \ Z_p \ J \ K (\alpha) F \ P$

P : Number of pitches (revolutions) (P can be omitted if equal to 0.)

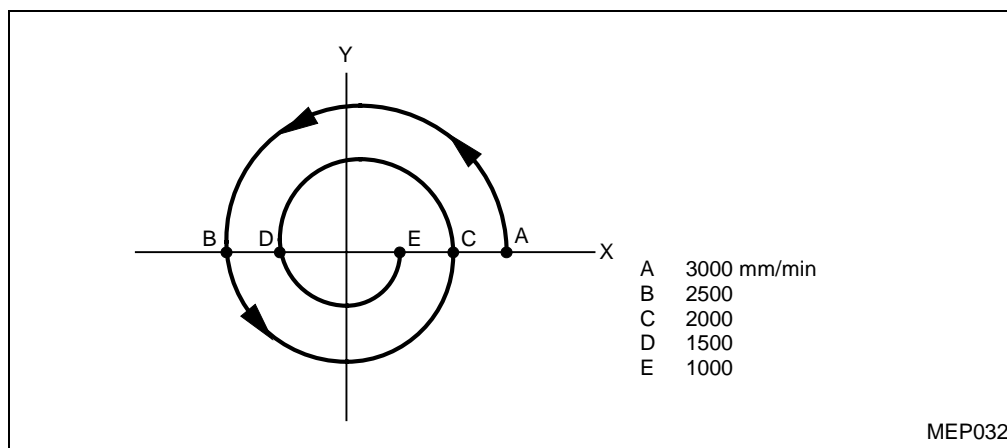
α : Any axis other than circular interpolation axes (For helical cutting only)

F : Rate of feed along the tool path

3. Detailed description

1. Circular movement directions of G2.1 and G3.1 correspond with those of G02 and G03, respectively.
2. Radius designation is not available for spiral interpolation. (The starting and ending points must lie on the same arc for a radius designation.)
Note: When a radius is designated, this command will be regarded as a radius-designated circular interpolation.
3. Conical cutting or tapered threading can be done by changing the radii of the arc at its starting and ending points and designating a linear-interpolation axis at the same time.
4. Even for normal circular command G2 or G3, spiral interpolation will be performed if the difference between the radii of the starting point and the ending point is smaller than the setting of parameter **F19**.

Example:



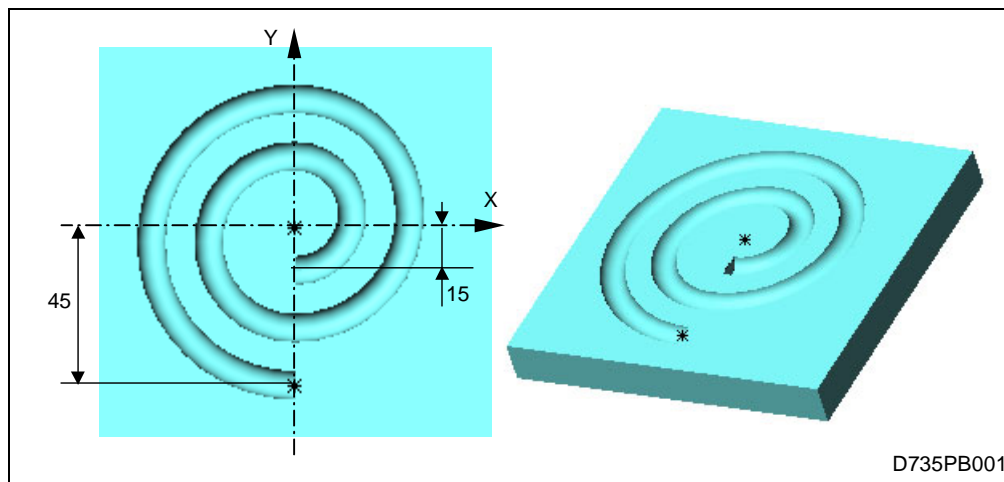
```
G28 X0 Y0
G00 Y-200.
G17 G3.1 X-100. Y0 I-150. J0 F3000 P2
M30
```

When the above program data are executed, the feed rates for each of the points will be as shown in the diagram above.

4. Sample programs

Example 1: Spiral cutting

Shown below is an example of programming for spiral contouring with incremental data input of the arc center (X = 0, Y = 45.0) and absolute data input of the arc ending point (X = 0, Y = -15.0).



G91 G28 Z0	Zero point return on the Z-axis
G80 G40	Fixed-cycle cancellation
T15 T00 M06	Tool change
G54.1 P40	Coordinate system setting
G94 G00 X0 Y-45.0	Approach in the XY-plane to the starting point (0, -45.0)
G43 Z30.0 H01	Positioning on the Z-axis to the initial point
Z3.0	
S1500 M03	Normal rotation of the spindle
M50	Air blast ON
G01 Z-1.0 F150	Infeed on the Z-axis
G2.1 X0 Y-15.0 I0 J45.0 F450 P2	Command for spiral interpolation with arc ending point = (0, -15.0), arc center = (0, 0)*, and pitch = 2. * I- and J-values refer to increments to the starting point.
G00 Z3.0	Return on the Z-axis
M05 M09	Spindle stop and Air blast OFF
Z30.0	
M30	End of machining

The rate of feed at the starting point is 450 mm/min, as specified in the block of G2.1, and the rate of feed at the ending point can be calculated as follows:

(Ending point's radius/Starting point's radius) × Command value of the rate of feed.

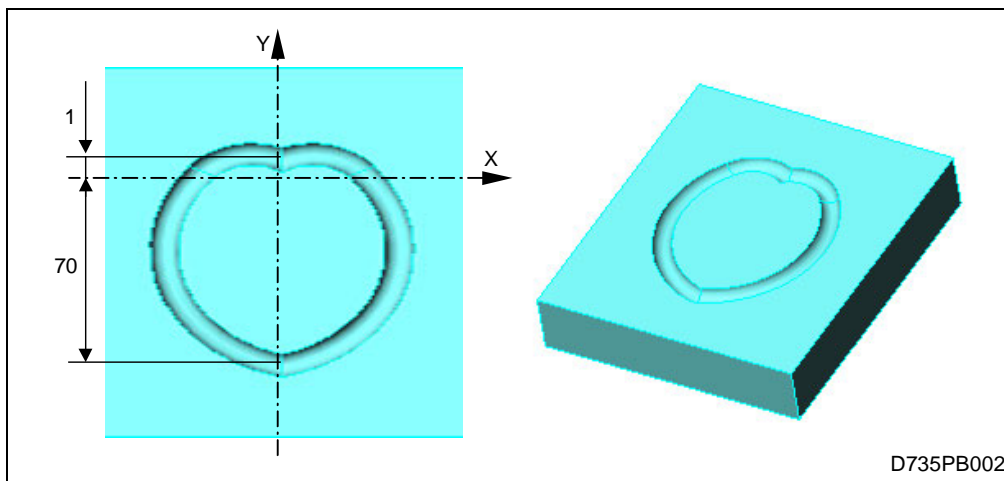
As the radius of the starting point = 45.0, that of the ending point = 15.0, and the command rate of feed (F) = 450, the rate of feed results in

$$(15.0/45.0) \times 450 = 150 \text{ mm/min}$$

at the ending point.

Note 1: Take care not to use radius designation (argument R) for spiral interpolation; otherwise a normal circular interpolation (by G02 or G03) will be executed.

Note 2: It is not possible to give the command for a spiral interpolation the starting and ending points of which should have different centers specified.

Example 2: Heart-shaped cam (by absolute data input)

```

G91 G28 Z0
G80 G40
T15 T00 M06
G54.1 P40
G94 G00 X0 Y-70.0
G43 Z30.0 H01
S1500 M03
Z3.0
M50
G01 Z-1.0 F150
G2.1 X0 Y1.0 I0 J70.0 F450
X0 Y-70.0 I 0 J-1.0
G00 Z3.0
M05 M09
Z30.0
M30

```

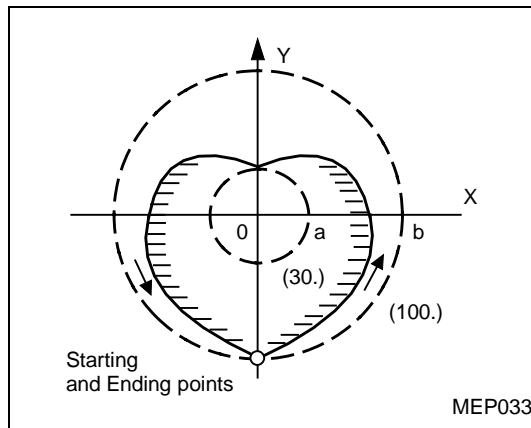
```

Zero point return on the Z-axis
Fixed-cycle cancellation
Tool change
Coordinate system setting
Approach in the XY-plane to the starting point (0, -70.0)
Positioning on the Z-axis to the initial point
Normal rotation of the spindle

Air blast ON
Infeed on the Z-axis
Command for the left-hand half curve
Command for the right-hand half curve
Return on the Z-axis
Spindle stop and Air blast OFF

End of machining

```

Example 3: Heart-shaped cam (by incremental data input)

The difference $(b-a)$ between the radii of the starting point and ending point denotes a displacement for heart shape.

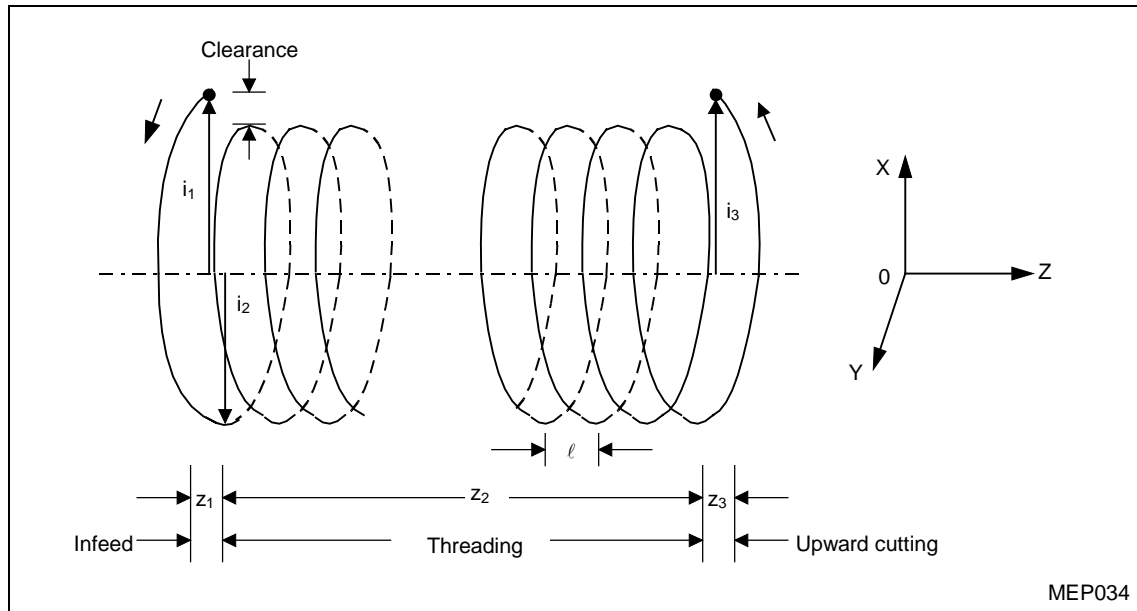
Use two blocks for programming separately the right-half and the left-half shape.

A sample program in incremental data input:

```
G3.1 Y130. J100. F1000..... (Right half)
      a+b    b
G3.1 Y-130. J-30 ..... (Left half)
      -a-b    -a
      a = 30. (Minimum arc radius)
      b = 100. (Maximum arc radius)
      a + b = 130. (Ending-point coordinate of the right half-circle)
      -a - b = -130. (Ending-point coordinate of the left half-circle)
```

Example 4: Large-size threading

To perform large-size threading, use three helical-interpolation blocks for programming separately infeed section, threading section and upward-cutting section. Spiral interpolation is required to designate the amounts of diameter clearance for both the infeed block and the upward-cutting block. (The starting and ending points are shifted through the designated clearance amounts from the circumference of threading section.)



G3.1 X- i_1 - i_2 Y0 ZZ₁ I- i_1 J0 Ff₁ (Infeed block, half-circle)

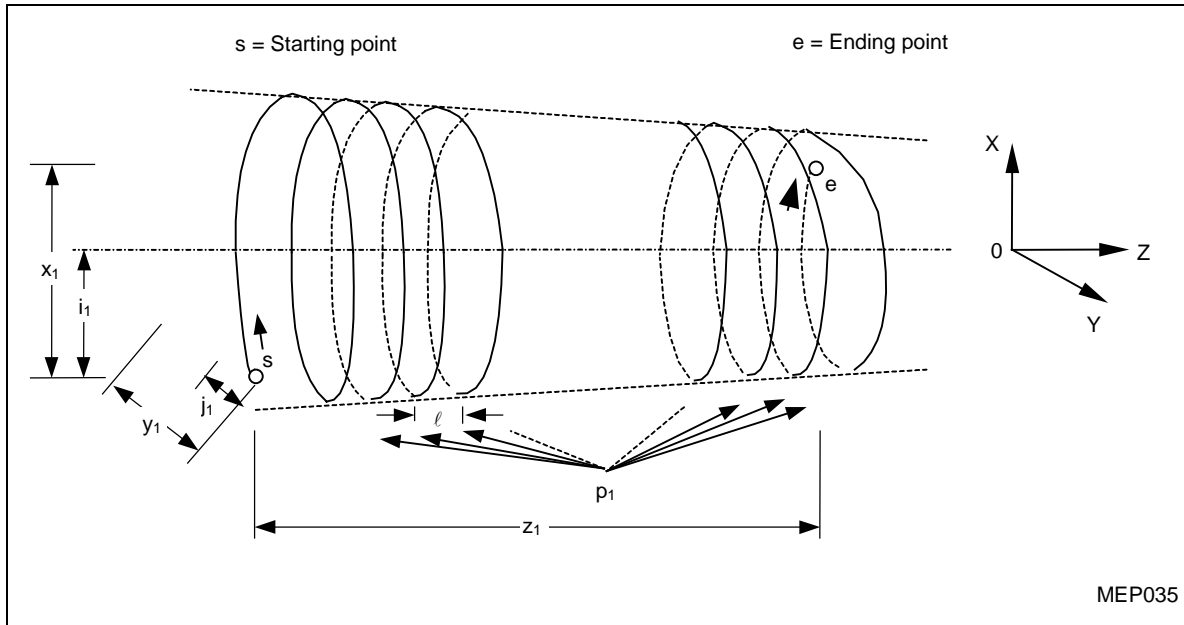
G03 X0 Y0 ZZ₂ I i_2 J0 Pp₂ (Threading block, complete circle)

G3.1 X i_2 + i_3 Y0 ZZ₃ I i_2 J0 (Upward-cutting block, half-circle)

- * The number of pitches, p_2 , in the threading block is given by dividing the stroke z_2 by the pitch ℓ . Note that the value p_2 must be an integer.

Example 5: Tapered threading

As shown in the figure below, tapered helical cutting that begins at any angle can be performed.



Data with addresses X, Y and Z must be the increments x_1 , y_1 and z_1 respectively, from the starting point s to the ending point e ; data of I and J must be the increments i_1 and j_1 respectively, from the starting point s to the circular center, and data of P must be equal to the number of pitches p_1 .

G3.1 $XX_1 YY_1 ZZ_1 II_1 JJ_1 PP_1 FF_1$

The amount of taper t and the pitch ℓ are calculated as follows:

$$t = \frac{2(re - rs)}{x_1}$$

where $rs = \sqrt{i_1^2 + j_1^2}$, $re = \sqrt{(x_1 - i_1)^2 + (y_1 - j_1)^2}$;

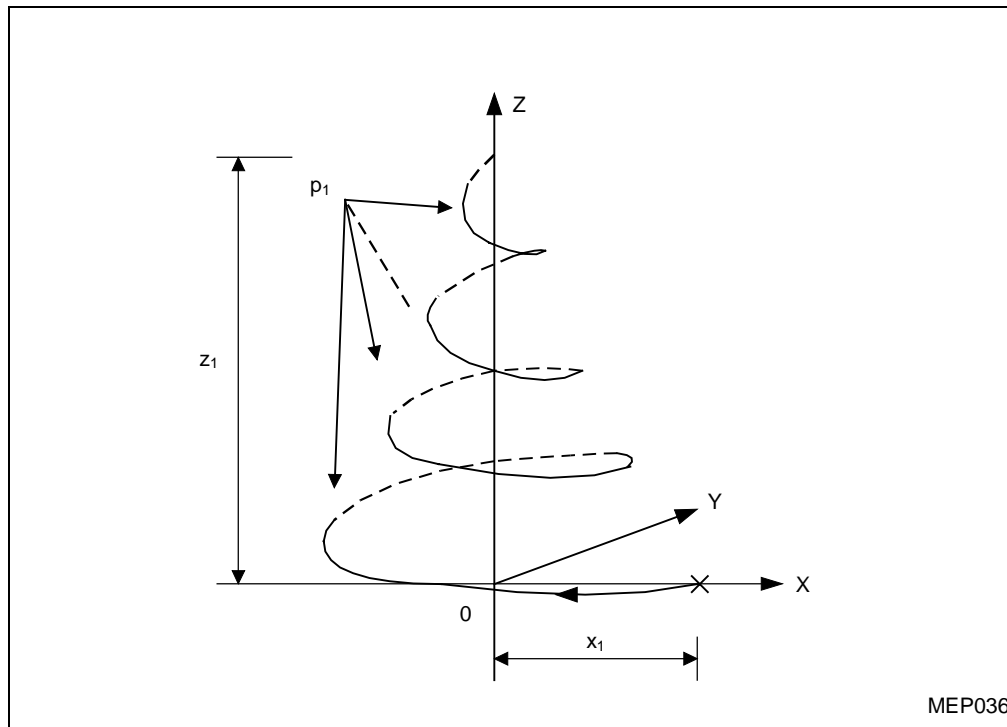
$$\ell = \frac{z_1}{(2\pi \cdot p_1 + \theta) / 2\pi}$$

$$\text{where } \theta = \theta_e - \theta_s = \tan^{-1} \frac{j_1 - y_1}{i_1 - x_1} - \tan^{-1} \frac{-j_1}{-i_1}$$

where rs and re denote the radii at the starting point and the ending point respectively, and qs and qe denote the angles at the starting point and the ending point respectively.

Example 6: Conical cutting

Conical cutting is an application of tapered threading, and have its starting or ending point on the center line. Tapering results from gradually increasing or decreasing the arc diameter. The pitch is determined by z_1/p_1 .



MEP036

G2.1 X-x₁ Y0 ZZ₁ I-x₁ Pp₁ Ff₁

x₁ : Radius of the base

z₁ : Height

p₁ : Number of pitches

f₁ : Feed rate

Note: Use the **TRACE** display to check the tool path during spiral interpolation.

6-7 Plane Selection: G17, G18, G19

6-7-1 Outline

1. Function and purpose

Commands G17, G18 and G19 are used to select a plane for motion control.

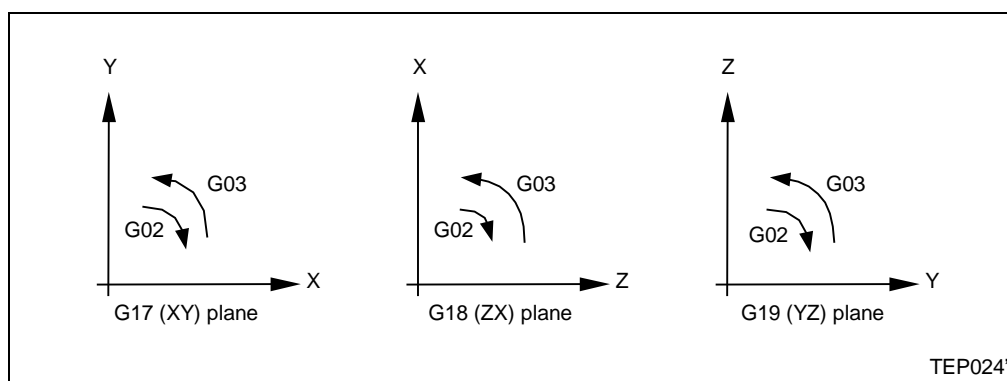
Registering the three fundamental axes as parameters allows you to select a plane generated by any two non-parallel axes.

Use these G-codes to select the plane for the following:

- Circular interpolation
- Polar coordinate interpolation
- Chamfering
- Positioning for fixed cycle operation
- Corner rounding/chamfering

2. Programming format

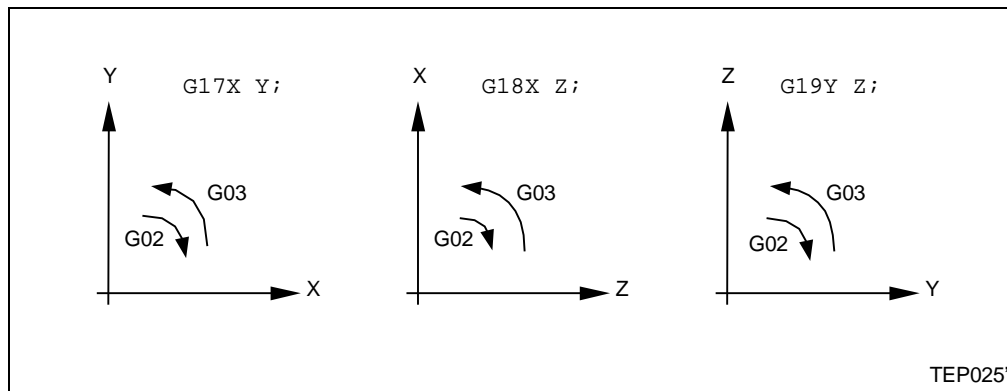
G17; (XY-plane selection)	} X, Y, and Z denote respective coordinate axes or their corresponding parallel axes.
G18; (ZX-plane selection)	
G19; (YZ-plane selection)	



6-7-2 Plane selection methods

Plane selection by parameter setting is explained in this section.

- Which of the fundamental axes or their parallel axes are to form the plane you want to select is determined by the type of plane selection command (G17, G18 or G19) and the axis address specified in the same block.



- Automatic plane selection does not occur for blocks that do not have an issued plane-selection command (G17, G18 or G19)

```
G18 X_ Z_; ZX-plane
Y_ Z_; ZX-plane (No plane change)
```

- If axis addresses are not set for blocks having an issued plane-selection command (G17, G18 or G19), the fundamental three axes will be regarded as set.

```
G18_; (ZX-plane = G18 XZ ;)
```

Note 1: Use bits 0 and 1 of parameter **F92** to set the initial plane which is to be selected upon power-on or resetting.

Note 2: The G-codes for plane selection (G17, G18 or G19) should be commanded in a block independently. If such a G-code is commanded in a block containing the axis move command, a movement independent from the selected plane can be caused.

6-8 Polar Coordinate Interpolation ON/OFF: G12.1/G13.1

1. Function and purpose

Polar coordinate interpolation is a function to convert a command programmed by the rectangular coordinate system into the linear axis movement (tool movement) and the rotational axis movement (workpiece rotation) to give contouring control.

2. Programming format

The polar coordinate interpolation is commanded by the following G-codes (group 19).

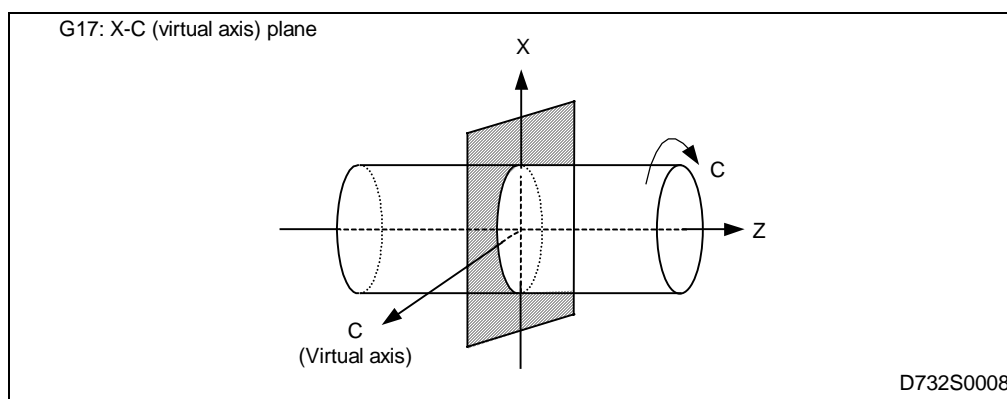
G12.1: Polar coordinate interpolation mode (Mode by which the polar coordinate is interpolated)

G13.1: Polar coordinate interpolation cancel mode (Mode by which the polar coordinate is not interpolated)

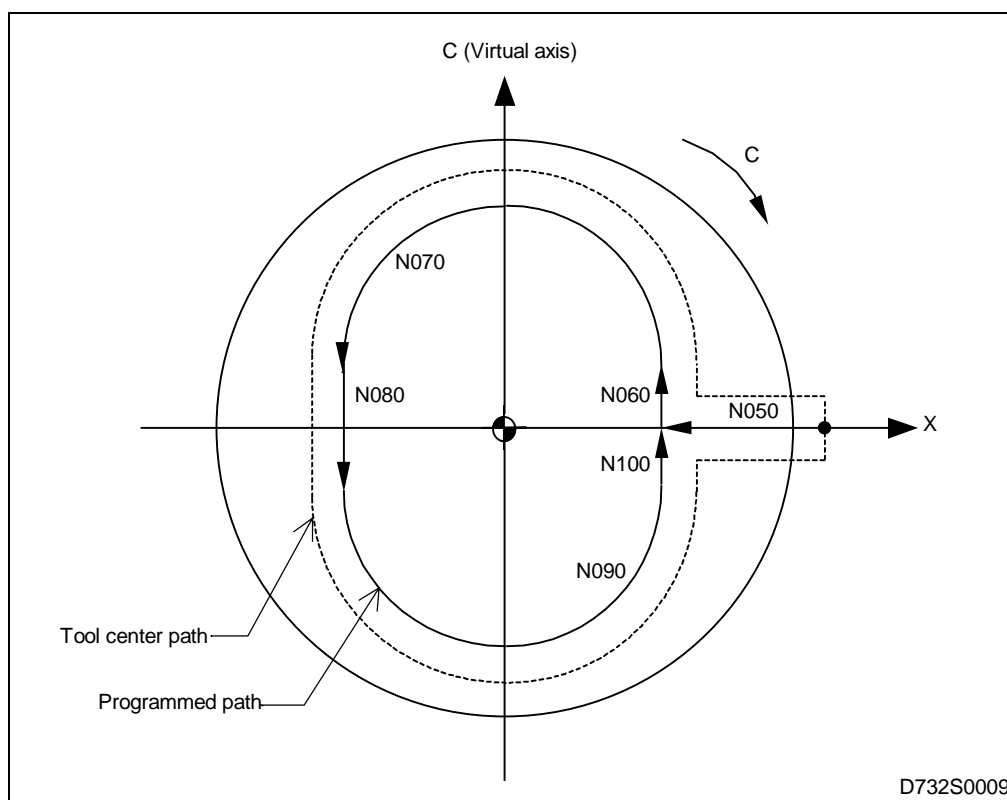
These G-codes should be commanded in an independent block.

3. Detailed description

1. When turning on the power and resetting, the polar coordinate interpolation cancel mode (G13.1) is provided. Commanding G12.1 provides a plane selected by G17.
2. The polar coordinate interpolation uses the zero point of workpiece coordinate system as that of the coordinate system. A plane (hereinafter referred to as "polar coordinate interpolation plane") is selected using the linear axis as the 1st axis of the plane and the virtual axis perpendicular to the linear axis as the 2nd axis of the plane. The polar coordinate interpolation is given on that plane.
3. The program during polar coordinate interpolation mode is commanded by the rectangular coordinate value on the polar coordinate interpolation plane. The axis address of the rotational axis (C) is used for that of the command of the 2nd axis of the plane (virtual axis).
A command is given in mm or inches as with the 1st axis of the plane (command by the axis address of the linear axis), and not in degrees.
4. Absolute command and incremental command for the linear interpolation (G01) and the circular interpolation (G02, G03) can be commanded during the polar coordinate interpolation mode.
The tool radius compensation can also be made for the program command, and the polar coordinate interpolation is given to the path after the tool radius compensation. However, the polar coordinate interpolation mode (G12.1, G13.1) cannot be changed during the tool radius compensation mode (G41, G42). G12.1 and G13.1 must be commanded in G40 mode (Tool radius compensation cancel mode).
5. The feed rate is commanded using tangential speed (relative speed of the workpiece and a tool) on the polar coordinate interpolation plane (rectangular coordinate system) as F (mm/min or in/min is used for a unit of F).
6. The coordinate value of the virtual axis when G12.1 is commanded provides "0". That is, the polar coordinate interpolation is started taking the position where G12.1 is commanded as the angle = 0.



4. Sample programs



```

N001 G00 G97 G98;
N004 G28 U0 W0;
N008 M200;
N010 T01T00M06;
N020 G00 X100.0 Z10.0 C0.0;
N030 G12.1;
N040 G42;
N050 G01 X50.0 F500;
N060 C10.0;
N070 G03 X-50.0 C10.0 I-25.0;
N080 G01 C-10.0;
N090 G03 X50.0 C-10.0 R25.0;
N100 G01 C0.0;
N110 G00 X100.0;
N120 G40;
N130 G13.1;
N140 M202;

```

Positioning to the start point
Polar coordinate interpolation ON

Contour program
(Program with rectangular coordinate values
on the XC-plane)

Polar coordinate interpolation OFF

5. Notes

1. Before G12.1 is commanded, a workpiece coordinate system must be set using the center of rotational axis as the zero point of the coordinate system. The coordinate system must not be changed during G12.1 mode.
2. The plane before G12.1 is commanded (plane selected by G17, G18 or G19) is temporarily cancelled, and it is restored when G13.1 (polar coordinate interpolation cancel) is commanded. The polar coordinate interpolation mode is cancelled in resetting, and the G17 plane is provided.
3. The method of commanding the circular radius (which address of I, J and K is used) when the circular interpolation (G02, G03) is given on the polar coordinate interpolation plane depends on which axis of the basic coordinate system the 1st axis of the plane (linear axis) corresponds to.
 - Command is given by I and J taking the linear axis as the X-axis of X_p - Y_p plane.
 - Command is given by J and K taking the linear axis as the Y-axis of Y_p - Z_p plane.
 - Command is given by K and I taking the linear axis as the Z-axis of Z_p - X_p plane.The circular radius can also be designated by R command.
4. G-codes capable of command during G12.1 mode are G04, G65, G66, G67, G00, G01, G02, G03, G98, G99, G40, G41 and G42.
5. Move command of an axis other than those on the selected plane during G12.1 mode is executed independently of the polar coordinate interpolation.
6. Tool offset must be commanded in the polar coordinate interpolation cancel mode before G12.1 is commanded. It cannot be commanded during the polar coordinate interpolation mode. Offset amount must not be changed during the polar coordinate interpolation mode.
7. Current position display during G12.1 mode
Every current position during the polar coordinate interpolation mode is displayed with an actual coordinate value. However, only "residue moving distance" (**REMAIN**) is displayed with the residue moving distance on the polar coordinate command plane.
8. Program restart cannot be made for a block during G12.1 mode.

6-9 Virtual-Axis Interpolation: G07

1. Function and purpose

Specify with G07 code one of the two circular-interpolation axes for helical or spiral interpolation with synchronous linear interpolation as a virtual axis (a pulse-distributed axis without actual movement), and an interpolation on the plane defined by the remaining circular axis and the linear axis can be obtained along the sine curve which corresponds with the side view of the circular interpolation with synchronous linear interpolation.

2. Programming format

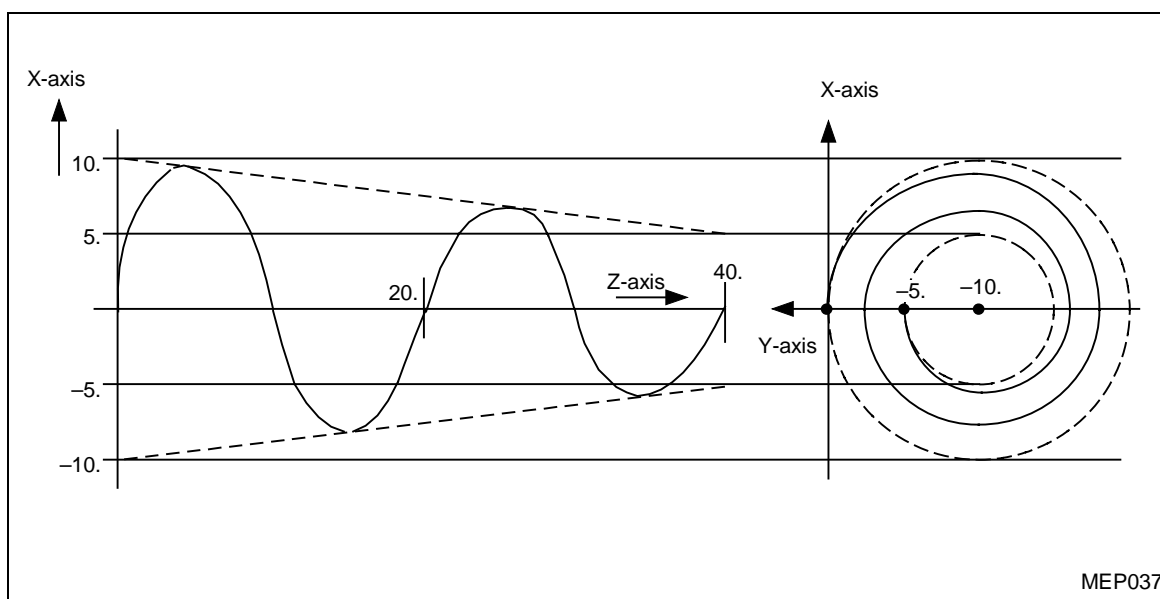
G07 α0	To set a virtual axis
⋮	} To interpolate with the virtual axis
G07 α1	

3. Detailed description

- Only helical or spiral interpolation can be used for the virtual-axis interpolation.
- In the program section from G07α0 to G07α1, the “alpha” axis is processed as a virtual axis. If, therefore, the alpha axis is included independently in this section, the machine will remain in dwell status until pulse distribution to the virtual axis is completed.
- The virtual axis is valid only for automatic operation; it is invalid for manual operation.
- Protective functions, such as interlock, stored stroke limit, etc., are valid even for the virtual axis.
- Handle interruption is also valid for the virtual axis. That is, the virtual axis can be shifted through the amount of handle interruption.

4. Sample program

G07 Y0	Sets the Y-axis as a virtual axis.
G17G2.1X0Y-5.I0J-10.Z40.P2F50	Sine interpolation on the XZ-plane
G07 Y1	Resets the Y-axis to an actual axis.



6-10 Spline Interpolation: G06.1 (Option)

1. Function and purpose

The spline interpolation automatically creates a curve that smoothly traces specified points, and thus enables a high-speed and high-accuracy machining for free shapes along smoothly curved tool path.

2. Programming format

G06.1 Xx₁ Yy₁

3. Detailed description

A. Setting and cancellation of spline interpolation mode

The spline interpolation mode is set by the preparatory function G06.1, and cancelled by another Group 01 command (G00, G01, G02 or G03).

Example 1:

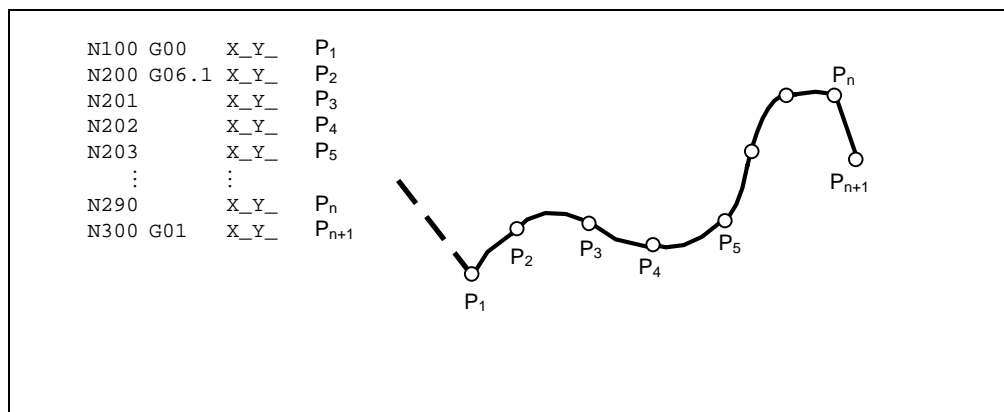


Fig. 6-1 Interpolated line by spline interpolation

In the above example, the spline interpolation is activated at N200 (block for movement from P₁ to P₂) and it is cancelled at N300. Therefore, a spline curve is created for a group of ending points from P₁ to P_n, and interpolation is applied along the created curve.

For creating a spline interpolation curve, it is generally required to specify two or more blocks (at least three points to be traced) in the mode. If the spline interpolation mode is set just for one block, the path to the ending point of the block is interpolated in a straight line.

Example 2:

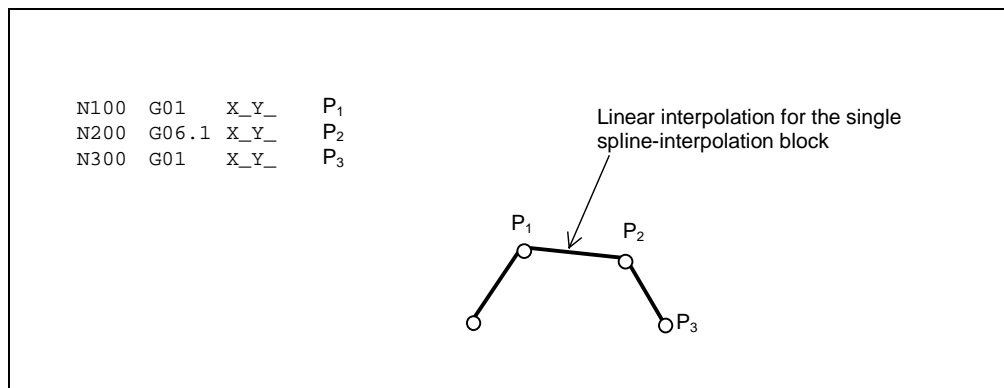


Fig. 6-2 Spline interpolation applied to a single block

B. Division of spline curve in spline-interpolation mode

The spline interpolation mode generally creates a continuous curve that smoothly connects all specified points from the beginning of the mode to the end of it. However, the spline curve is divided into two discontinuous curves as often as one of the following conditions is satisfied:

- When the angle between linear movement lines of two neighboring blocks is beyond the spline-cancel angle,
- When the movement distance of a block exceeds the spline-cancel distance, or
- When there is a block without any movement command in the spline-interpolation mode.

1. When the relative angle of two neighboring blocks is beyond the spline-cancel angle

Spline-cancel angle Parameter **F101**

As to the sequence of points $P_1, P_2, P_3, \dots, P_n$ in a spline interpolation mode, when the angle θ_i made by two continuous vectors $\overrightarrow{P_{i-1}P_i}$ and $\overrightarrow{P_iP_{i+1}}$ is larger than **F101**, the point P_i is regarded as a corner. In that event, the point group is divided into two sections of P_1 to P_i and P_i to P_n at P_i , and spline curve is individually created for each section.

When the spline-cancel angle is not set (**F101** = 0), this dividing function is not available.

Example 1: **F101** = 80 deg

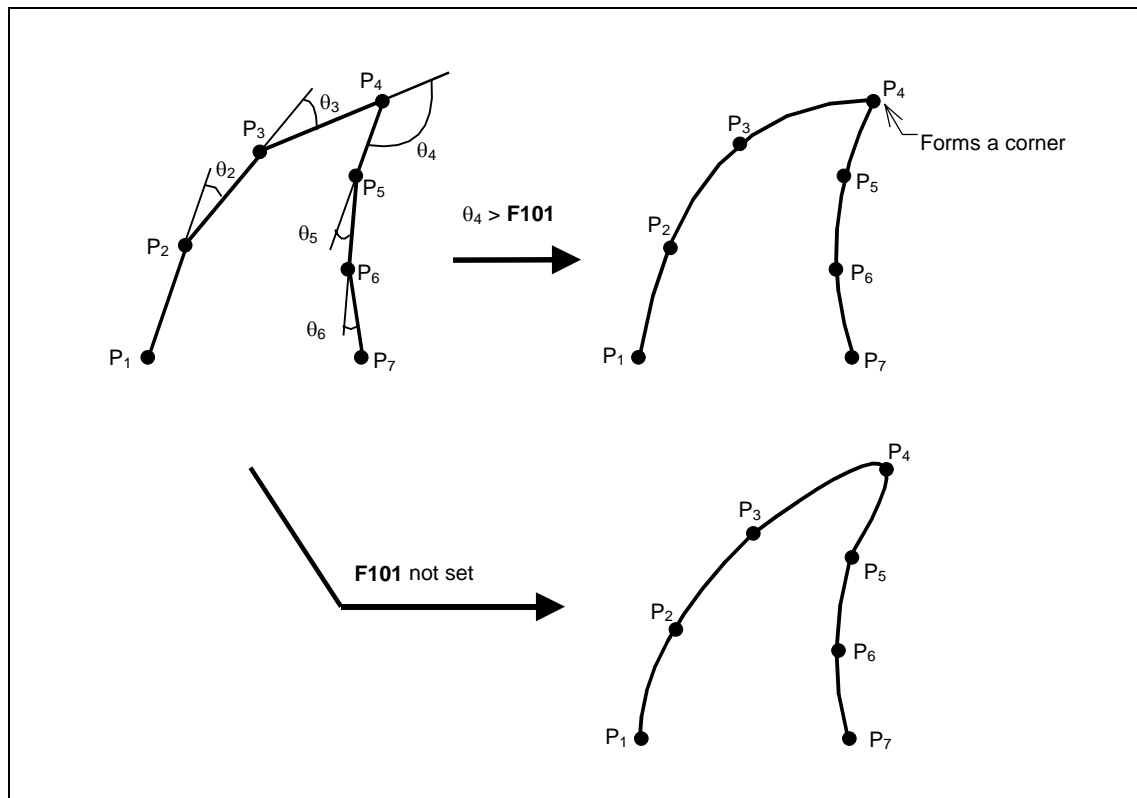


Fig. 6-3 Spline cancel depending on angle

When there are more than one point where $\theta_i > \mathbf{F101}$, such points are treated as corners to divide the point group and multiple spline curves are created for respective sections.

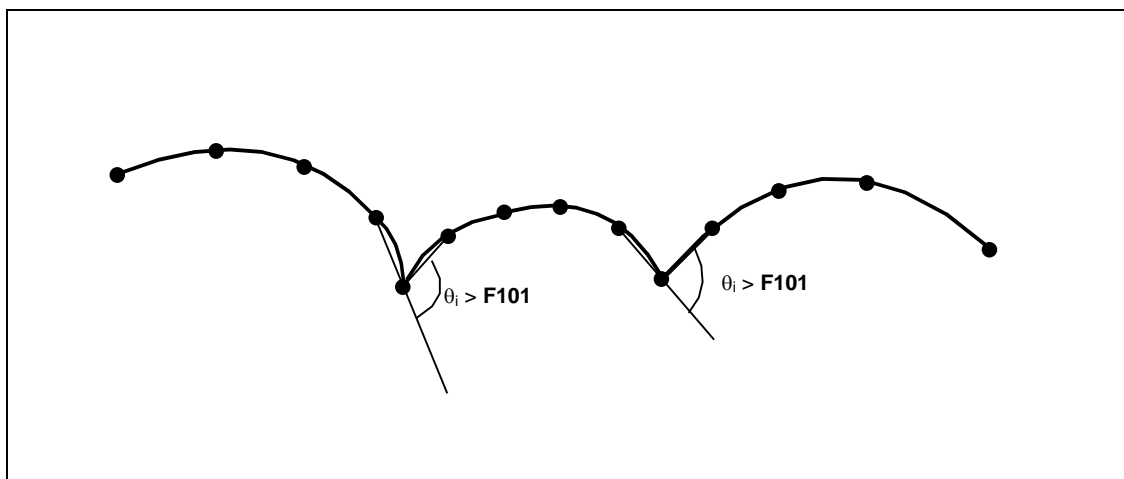


Fig. 6-4 Multiple-cornered spline curve depending on angle

When any two corner points (where $\theta_i > \mathbf{F101}$) successively exist, the block for the second point is automatically set under control of linear interpolation. Therefore, it can be omitted to specify G01 code in each intermediate block of pick feed, for example, during 2.5-dimensional machining, which considerably simplifies the programming.

Example 2: $\mathbf{F101} < 90$ (deg)

In the following program (shown in Fig. 6-5), the angle of the Y-directional pick feed to the X-Z plane (of spline interpolation) is always 90° . If $\mathbf{F101}$ is set slightly smaller than 90° , spline interpolation is automatically cancelled in the pick-feed blocks (N310, N410, ...), which are then linearly interpolated each time. If no value is set for $\mathbf{F101}$, it is required to specify G-codes parenthesized in the program below to change the mode of interpolation.

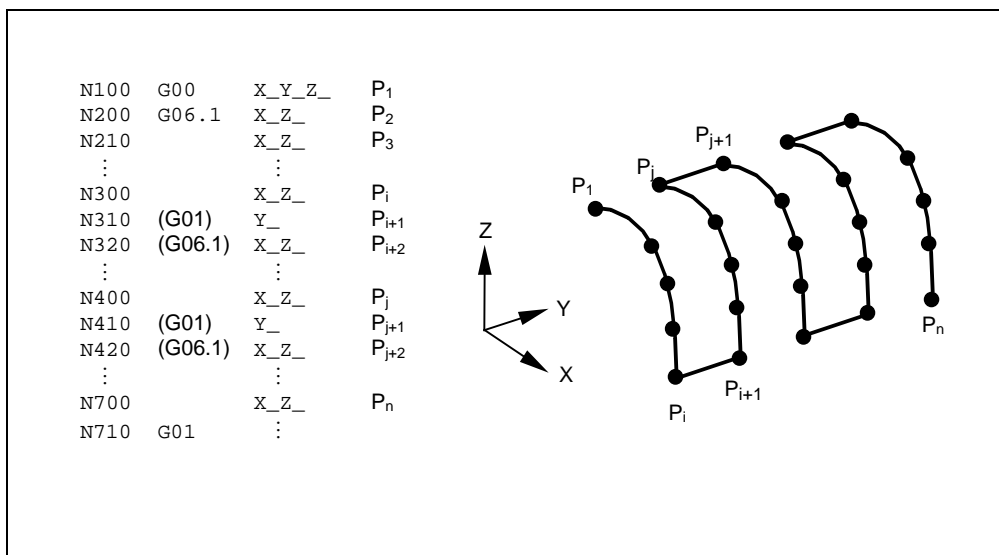


Fig. 6-5 Linear interpolation for pick feed in spline-interpolation mode

2. When the movement distance of a block exceeds the spline-cancel distance

Spline-cancel distance Parameter **F100**

As to the sequence of points $P_1, P_2, P_3, \dots, P_n$ in a spline interpolation mode, when the length $\overline{P_i P_{i+1}}$ of the vector $\overrightarrow{P_i P_{i+1}}$ is longer than **F100**, the block for point P_{i+1} is automatically set under control of linear interpolation, while the preceding and succeeding sections P_1 to P_i and P_{i+1} to P_n are individually interpolated in spline curves.

In this case, the inclination of the tangent vector at P_i (at the end of spline P_1 to P_i) and the inclination of the tangent vector at P_{i+1} (at the beginning of spline P_{i+1} to P_n) do not correspond to that of the line segment $\overline{P_i P_{i+1}}$ in general.

When the spline-cancel distance is not set (**F100** = 0), this dividing function is not available.

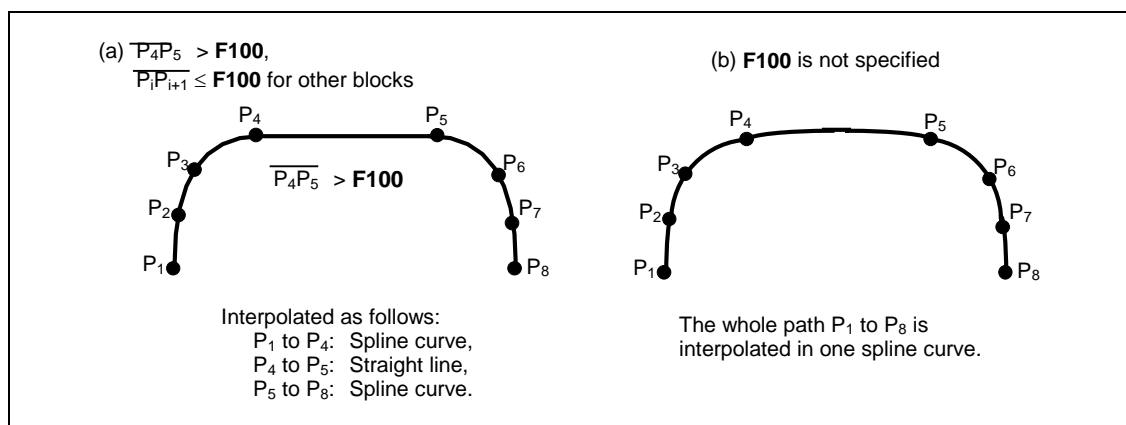


Fig. 6-6 Spline cancel depending on movement distance of a block

When there are more than one block where $\overline{P_i P_{i+1}} > \mathbf{F100}$, all those blocks will individually undergo the linear interpolation.

3. When there is a block without any movement command in the spline-interpolation mode
- Any block without movement command temporarily cancels the spline interpolation, and the sections before and after such a block will independently be spline-interpolated.

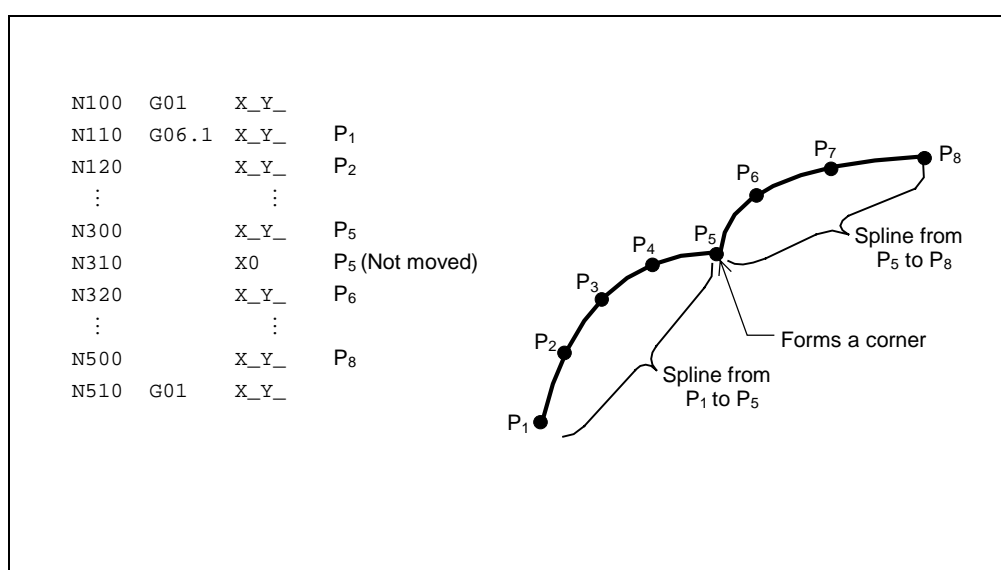


Fig. 6-7 Spline cancel by a block without movement command

C. Fine spline function (curved shape correction)

The fine spline function works with spline interpolation and automatically corrects the shape of a spline curve, as required, to make the path of the curve smoother.

More specifically, the fine spline function works in the following two cases:

- The case that the curve errors in blocks are significant
- The case that an unusually short block exists (automatic correction in this case is referred to as fairing.)

Automatic correction in the above cases is explained below.

1. Automatic correction for significant curve errors in blocks

When the curve data in CAD undergoes micro-segmentation with CAM, approximation using a polygonal line is usually executed with a curve tolerance (chord error) of about 10 microns. At this time, if any inflection points are included in the curve, the micro-segment block including the inflection points may increase in length (see $\overline{P_3 P_4}$ in the figure below)

Also, if the length of this block becomes unbalanced against those of the immediately preceding and succeeding blocks, the spline curve in this zone may have a significant error with respect to the original curve.

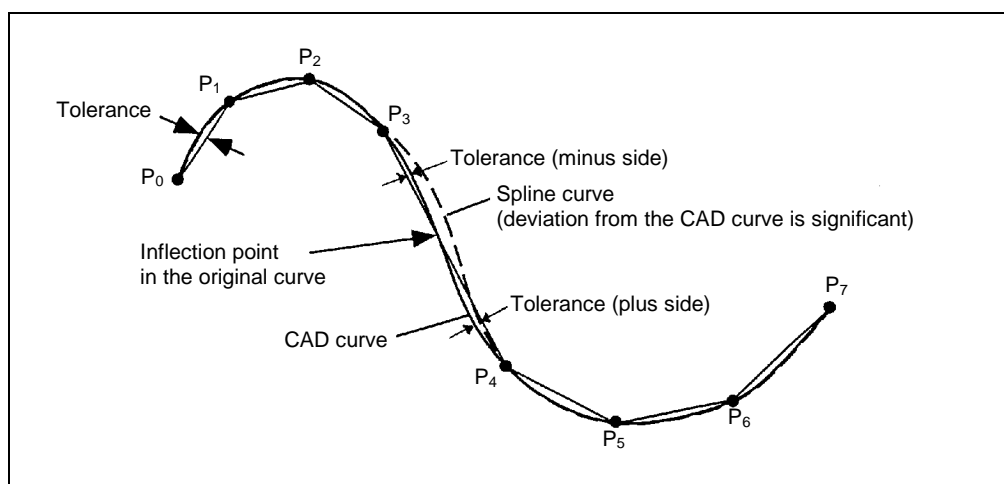


Fig. 6-8 Spline curve having a significant chord error (inflection points present)

This function detects the sections whose chord errors in the curve due to the presence of inflection points become significant, and corrects the shape of the spline curve in that zone automatically so that the chord errors in the curve fall within the data range of the specified parameter.

Curve error 1 Parameter **F102**

If a block in the spline interpolation mode is judged to have inflection points in the spline curve and the maximum chord error of the spline curve from the segment is greater than the value of **F102**, the shape of that spline curve will be corrected for a maximum chord error not exceeding the value of **F102**.

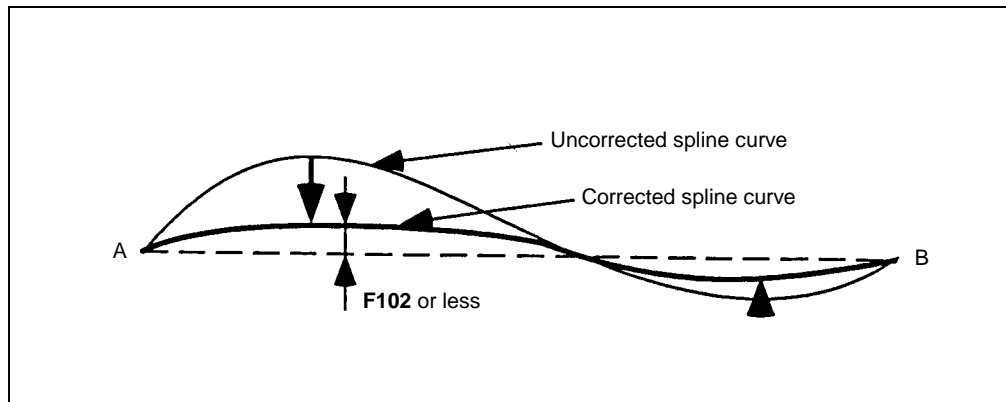


Fig. 6-9 Shape correction 1 for spline curve

The shape of a curve can also be corrected if the chord error in the spline curve increases due to an imbalance in the lengths of adjoining blocks occurs for any reasons other than the presence of inflection points or for other reasons.

Curve error 2 Parameter **F104**

If a blocks in the spline interpolation mode is judged to have no inflection points in the spline curve and the maximum chord error in the spline curve and block is greater than the value of **F104**, the shape of that spline curve will be corrected for a maximum chord error not exceeding the value of **F104**.

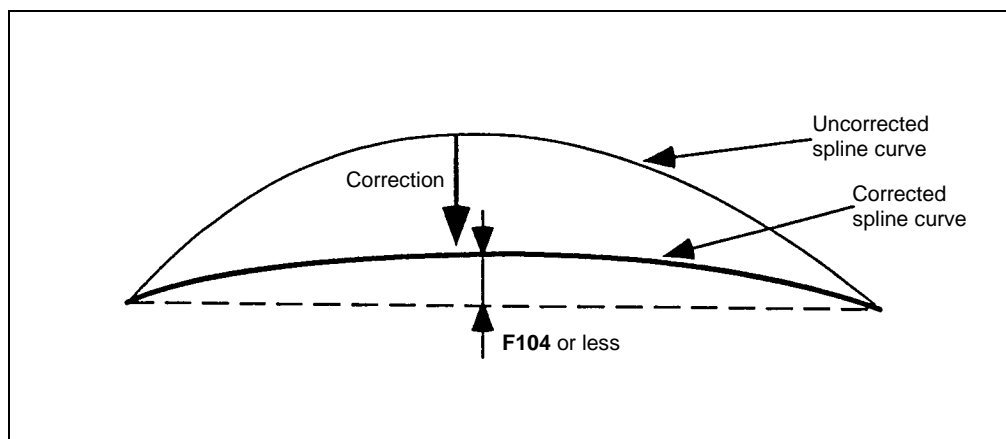


Fig. 6-10 Spline curve having a significant chord error (no inflection points)

- Remark 1:** In all types of spline curve correction, the curve correction function works only for the corresponding block. Therefore, the tangential vectors at the boundaries with the immediately preceding and succeeding blocks become discontinuous.
- Remark 2:** If parameter **F102** is set to 0, all blocks regarded as including inflection points will become linear. If parameter **F104** is set to 0, all blocks regarded as including no inflection points will become linear.
- Remark 3:** Curved-shape correction based on parameter **F102** or **F104** usually becomes necessary when adjoining blocks are unbalanced in length. If the ratio of the adjoining block lengths is very large, however, spline interpolation may be temporarily cancelled between the blocks prior to evaluation of the chord error.

2. Automatic correction of the spline curve in an unusually short block (Fairing)

When CAD data is developed into micro-segments by CAM, a very small block may be created in the middle of the program because of internal calculation errors. Such a block is often created during creation of a tool radius compensation program which requires convergence calculation, in particular. Since this unusually small block usually occurs at almost right angles to the direction of the spline curve, this curve tends not to become smooth.

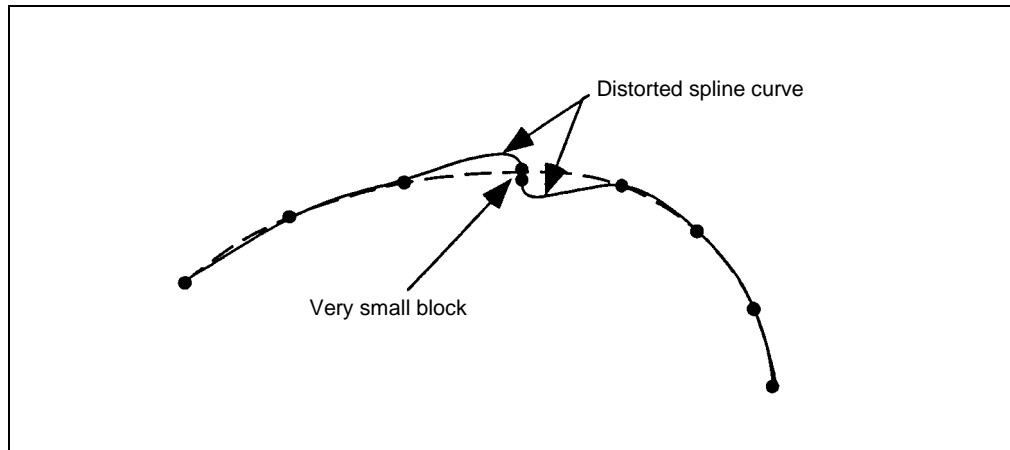


Fig. 6-11 Distortion of a spline curve due to the effects of a very small block

If it detects such an extremely small block during spline interpolation, the shape correction function will remove that block and then connect the preceding and succeeding blocks directly (this is referred to as fairing) to create a smooth spline curve free from distortion.

Block fairing length Parameter **F103**

Assume that the length of the i -th block in spline interpolation mode is taken as l_i and that the following expressions hold:

$$l_{i-1} > \mathbf{F103} \times 2$$

$$l_i \leq \mathbf{F103}$$

$$l_{i+1} > \mathbf{F103} \times 2$$

In the above case, the ending point of the $(i-1)$ -th block and the starting point of the $i+1$ block are moved to the mid-point of the i th block and as a result, the i th block is deleted. Spline interpolation is executed for the sequence of points that has thus been corrected.

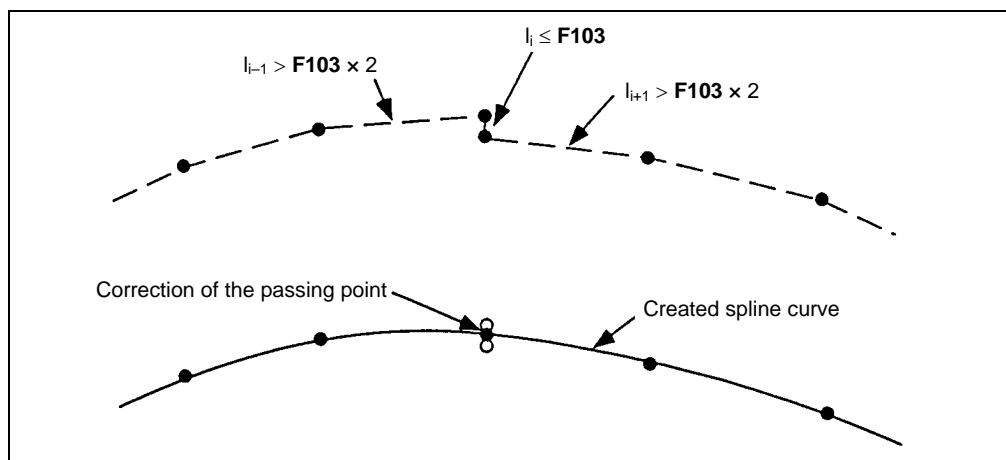


Fig. 6-12 Correction of spline curve passing points by fairing

Assume that the first block in spline interpolation mode is very small and that the following expressions hold:

$$l_1 \leq F103$$

$$l_2 > F103 \times 2$$

In the above case, the starting point of the second block is changed to that of the first block and as a result, the first block is deleted.

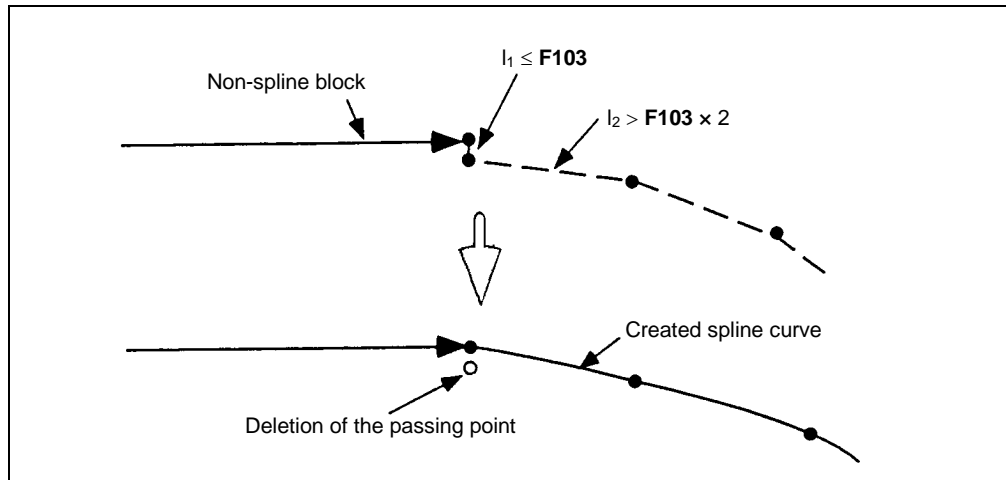


Fig. 6-13 Fairing at the starting point of a spline curve

Assume that the last block in spline interpolation mode is very small and that the following expressions hold:

$$l_{n-1} > F103 \times 2$$

$$l_n \leq F103$$

In the above case, the ending point of the (n-1)-th block is changed to that of the nth block and as a result, the nth block is deleted.

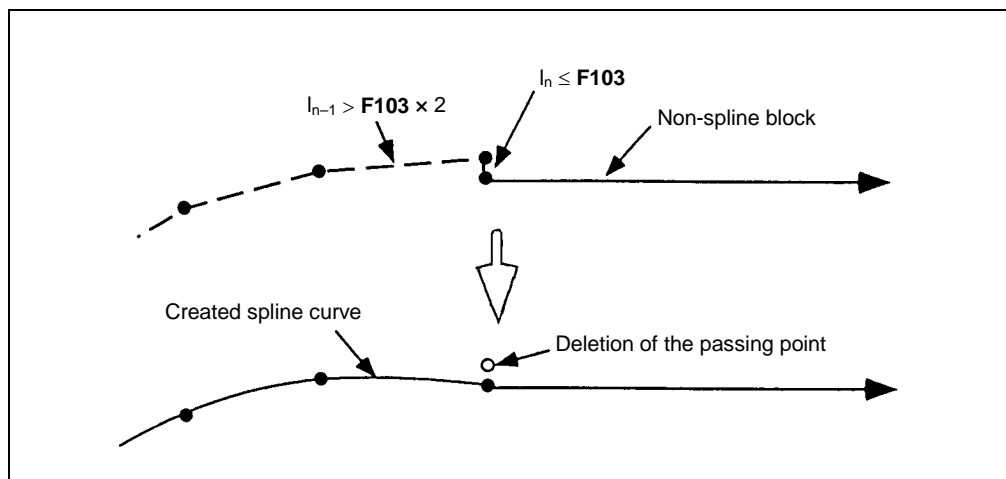


Fig. 6-14 Fairing at the ending point of a spline curve

This function is executed preferentially over the curve slitting function based on the angle of spline cancellation.

D. Feed-rate limitation in spline-interpolation mode

The modal cutting feed rate F remains valid in general for the spline interpolation; however, if the feed rate should be kept constant, it would yield excessively high acceleration at portions where the curvature is big (the curvature radius is small) as shown in Fig. 6-15.

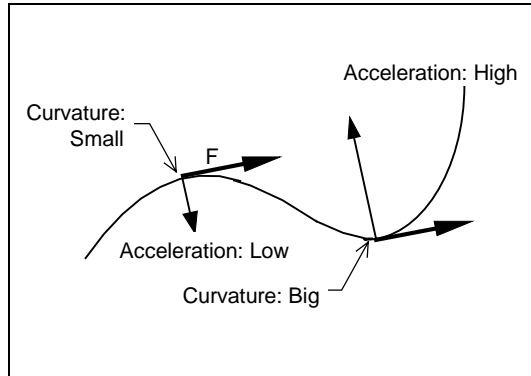


Fig. 6-15 Change of acceleration depending on curvature

In the spline-interpolation mode of our NC, the feed rate can be controlled so that it does not exceed the allowable limit, calculated from the related parameters, for pre-interpolation acceleration.

To obtain an appropriate feed rate for each block of spline interpolation, the limit feed rate F' is calculated by the equation [1] shown below where the smaller between two radii R_s (curvature radius at the starting point of the block) and R_e (curvature radius at its ending point) will be regarded as the reference radius R for the block. The modal feed rate F will then be temporarily overridden by F' for the respective block if $F > F'$, so that the whole spline curve can be interpolated block-by-block at the appropriate feed rate according to the curvature radius.

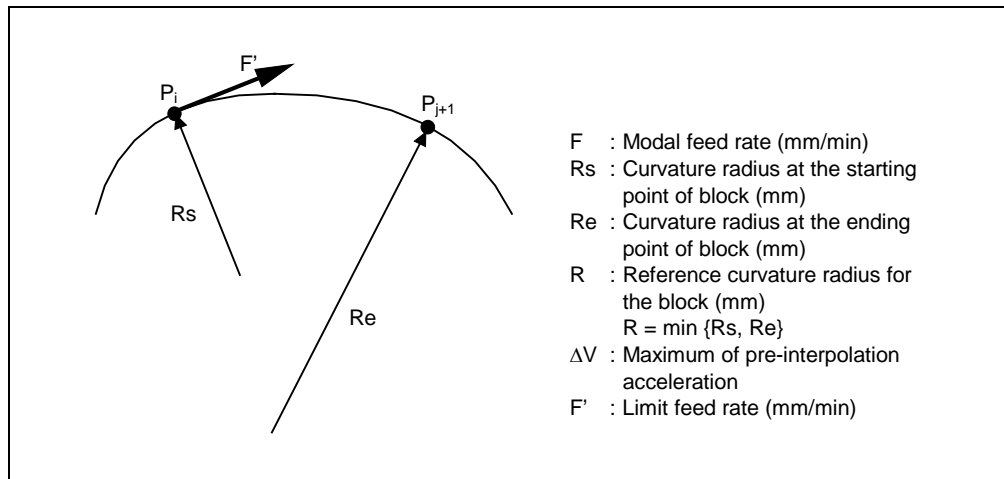


Fig. 6-16 Feed-rate limitation for spline interpolation

$$F' = \sqrt{R \times \Delta V \times 60 \times 1000} \quad \dots\dots [1]$$

$$\Delta V = \frac{G1bF \text{ (mm/min)}}{G1btL \text{ (ms)}}$$

E. Spline interpolation during tool radius compensation

The spline interpolation can be performed during tool radius compensation as follows.

1. Tool radius compensation (2-dimensional)

Shown in Fig. 6-17 is an example that the command route is straight in the section P_0P_1 , polygonal line in the section $P_1P_2 \dots P_n$ that is the object of spline interpolation, and straight in the section P_nP_{n+1} . The interpolation route with tool radius compensation is created by the following procedure.

- 1) In the first step is created a polygonal line $P_0'P_1'P_2' \dots P_n'P_{n+1}'$ that is offset by the radius compensation value r compared with the original polygonal line $P_0P_1P_2 \dots P_nP_{n+1}$.
- 2) Next, a point P_i'' where $\overline{P_iP_i''} = r$ on the vector $\overrightarrow{P_iP_i'}$ is determined for all the pass points P_i ($i = 2, 3, \dots, n-1$) other than the starting point P_1 and the ending point P_n of the spline curve.
- 3) Spline interpolation is now conducted for the polygonal line $P_1'P_2''P_3'' \dots P_{n-1}''P_n'$ and the curve thus created will act an offset path of tool center for the commanded spline curve.

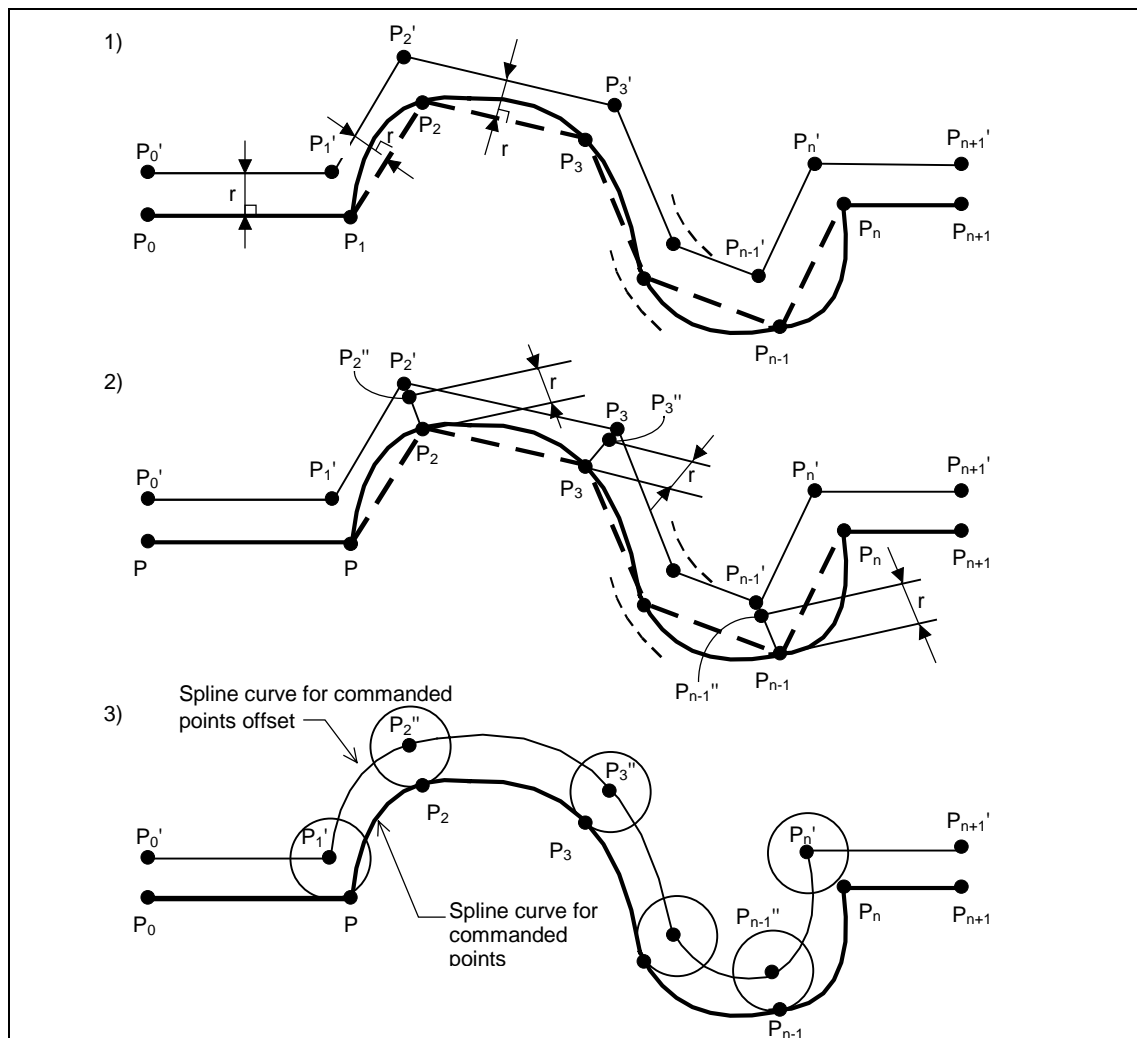


Fig. 6-17 Spline interpolation during tool radius compensation

The spline curve created in the above-mentioned procedure is not the strict offset, indeed, of the commanded spline curve, but an approximation of it.

2. 3-dimensional tool radius compensation

In the 3-dimensional tool radius compensation, each point defined with programmed coordinates is first offset through the tool radius “r” in the direction of the specified normal vector (i, j, k) and then, the serial points thus offset in the spline-interpolation section are connected in a smooth curve, which will act as the path of tool-radius center for the 3-dimensional spline interpolation.

F. Others

1. The spline interpolation targets the basic coordinate axes of X, Y and Z; however, it is not always required to specify objective axes on commanding the spline interpolation. Moreover, the spline-interpolation command code (G06.1) can be given in a block without any movement command.

Example:

N100	G06.1	X_Y_Z0	→	N100	G06.1	X_Y_
N200		X_Y_Z_		N200		X_Y_Z_
N300		X_Y_Z_		N300		X_Y_Z_
⋮		⋮		⋮		⋮
N100	G06.1	F_ (← No movement commands)				
N200		X_Y_Z_				
N300		X_Y_Z_				
⋮		⋮				

2. The spline-interpolation command (G06.1) falls under the G-code group 01.
3. In the single-block operation mode, the spline interpolation is cancelled and all the respective blocks will individually undergo the linear interpolation.
4. In tool-path check, the blocks of spline interpolation are not actually displayed in a spline curve but in a polygonal line that connects linearly the respective points, which, in case of tool radius compensation, will have been offset in the same manner as described in the foregoing article E.
5. During spline interpolation, when feed hold is executed, the block for which the feed hold function has been executed will be interpolated, at the beginning of the restart operation along the spline curve existing before the feed hold function was executed, and then the spline curve in the next block onward will be re-created and interpolation executed.
6. Although spline interpolation can also be executed in the high-speed machining mode (G05P2 mode), curve shape correction by fairing becomes invalid in the G05P2 mode.

6-11 Modal Spline Interpolation: G61.2 (Option)

1. Function and purpose

The modal preparatory function G61.2 is used to select a geometry compensation mode which is in general equivalent to G61.1 with a particular difference in that all blocks of G1 (linear interpolation) are processed as those of fine spline interpolation. This function is especially useful in applying fine spline interpolation to a program with microsegment blocks created by a CAM.

2. Programming format

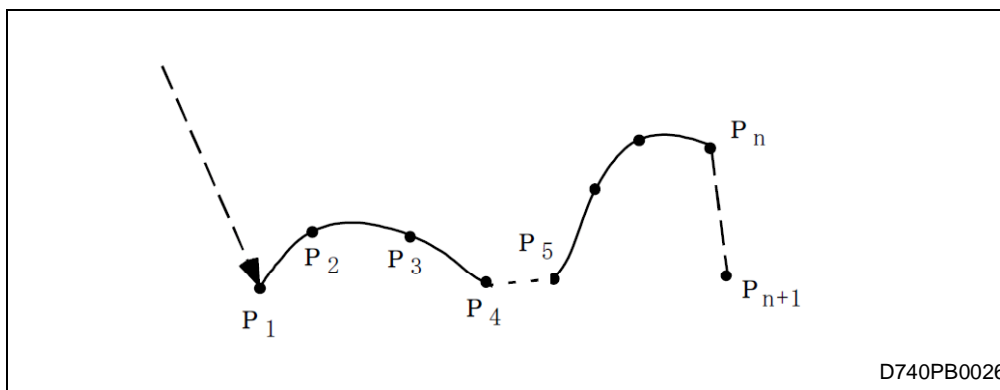
G61.2 Modal spline interpolation ON
:
G64 Modal spline interpolation OFF

3. Detailed description

The mode of fine spline interpolation for G1 blocks under G61.2 is temporarily cancelled by any other G-code of group 01, and the mode of G61.2 is cancelled by any other G-code of group 13. The table below is given to show how various interpolation types are processed under different modal states of group 13. For example, G1 under G61.2 is equivalent to G06.1 under G61.1.

	G61.2	G61.1	G64
G0	G61.1 + G0	G61.1 + G0	G0
G1	G61.1 + G06.1	G61.1 + G1	G1
G2/G3	G61.1 + G2/G3	G61.1 + G2/G3	G2/G3
G06.1	G61.1 + G06.1	G61.1 + G06.1	G06.1

4. Sample program



N099 G61.2		N205 X_Y_Z_	P ₇
N100 G00 X_Y_Z_	P ₁	N206 X_Y_Z_	P ₈
N200 G01 X_Y_Z_	P ₂	:	
N201 X_Y_Z_	P ₃	N290 G01 X_Y_Z_	P _n
N202 X_Y_Z_	P ₄	N300 G00 X_Y_Z_	P _{n+1}
N203 G00 X_Y_Z_	P ₅		
N204 G01 X_Y_Z_	P ₆		

A simple insertion of G61.2 allows fine spline interpolation to be applied to the linear interpolation blocks without having to replace the G01 codes in the blocks N200, N204 etc. with G06.1.

6-12 NURBS Interpolation: G06.2 (Option)

1. Function

The NURBS interpolation function provides interpolation by performing NURBS-defined CNC-internal computations on the command issued from the CAD/CAM system in the NURBS format. With this optional function, a very smooth interpolation path can be obtained since the interpolation process is performed directly without dividing a NURBS-formatted free-form curve into minute line segments.

2. Definition of the NURBS curve

NURBS, short for Non-Uniform Rational B-Spline, provides rationalization of the B-spline function.

The NURBS curve is defined as follows:

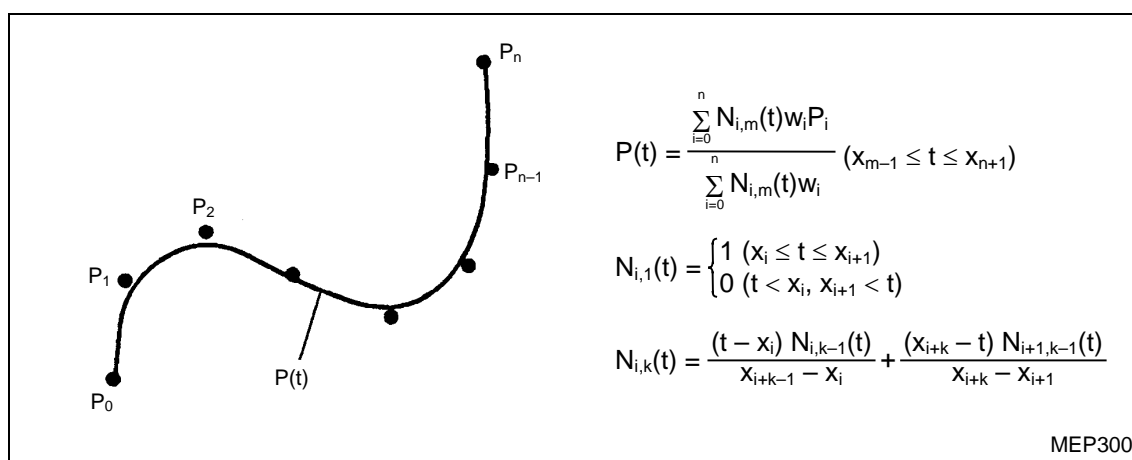


Fig. 6-18 NURBS curve

- “ P_i ” and “ w_i ” denote respectively a control point and the weight on the control point.
- “ m ” denotes the rank, and the NURBS curve of rank “ m ” is a curve of the $(m-1)$ -th order.
- “ x_i ” denotes a knot ($x_i \leq x_{i+1}$), and an array of knots $[x_0 \ x_1 \ x_2 \ \dots \ x_{n+m}]$ is referred to as the knot vector.
- A variation in parameter “ t ” from x_{m-1} to x_{n+1} produces NURBS curve $P(t)$.
- $N_{i,k}(t)$ is the B-spline basis function expressed by the above recurrence equation.

Thus the NURBS curve is uniquely defined from the weighted control points and the knot vector.

3. Programming format

G6.2[P] K_X_Y_Z_[R_] [F_] ← NURBS interpolation ON

K_X_Y_Z_[R_]

K_X_Y_Z_[R_]

K_X_Y_Z_[R_]

⋮

K_X_Y_Z_[R_]

K_

K_

K_

K_

← NURBS interpolation OFF

P : Rank (omissible)

X, Y, Z : Coordinates of the control point

R : Weight on the control point (omissible)

K : Knot

F : Speed of interpolation (omissible)

4. Detailed description

Set the G6.2 code to select the NURBS interpolation mode. Subsequently, designate the rank, the coordinates and weights of the control points, and the knots to determine the shape of the NURBS curve.

The modal code G6.2, which belongs to group 1 of G-codes, is of temporary validity and the modal function relieved by a G6.2 code will automatically be retrieved upon cancellation (termination) of the NURBS interpolation. The G6.2 code can only be omitted for an immediately subsequent setting of the next NURBS curve.

Address P is used to set the rank, and the NURBS curve of rank “m” is of the (m–1)-th order, that is, set as the rank

- P2 for a straight line (curve of the first order),
- P3 for a quadratic curve (of the second order) or
- P4 for a cubic curve (of the third order).

Setting another value than 2, 3 and 4 will cause an alarm, and P4 will be used in default of argument P. The rank, moreover, should be specified in the first block (containing the G6.2 code).

Designate the control points in as many sequential blocks as required by specifying their respective coordinates and weights at addresses X, Y, Z and R. Argument R denotes the weight proper to each control point (R1.0 will be used in default), and the more the weight is applied, the closer will be drawn the NURBS curve to the control point.

Address K is assigned to knots, and the NURBS curve of rank “m” for an “n” number of control points requires an (n+m) number of knots. The required array of knots, referred to as knot vector, is to be designated in sequential blocks, namely: the first knot in the same block as the first control point, the second knot in the same block as the second control point, and so forth. Following the “n” blocks entered thus, designate the remaining “m” knots in single-command blocks. The leading single-command block of argument K also notifies the NC of the completion of entering the control points, and the NURBS interpolation function itself will be terminated with the last block for the “m” knots.

5. Remarks

1. Only the fundamental axes X, Y and Z can undergo the NURBS interpolation.
2. Do not fail to explicitly designate all the required axes X, Y and/or Z in the first block (containing G6.2). Designating a new axis in the second block onward will cause an alarm.
3. Since the first control point serves as the starting point of the NURBS curve, set in the first block (with G6.2) the same coordinates as the final point of the previous block. Otherwise, an alarm will be caused.
4. The setting range for the weight (R) is from 0.0001 to 99.9999. For a setting without decimal point, the least significant digit will be treated as units digit (for example, 1 = 1.0).
5. The knot (K) must be designated for each block. Omission results in an alarm.
6. Knots, as with the weight, can be set down to four decimal digits, and the least significant digit of a setting without decimal point will be regarded as units digit.
7. Knots must be monotonic increasing. Setting a knot smaller than that of the previous block will result in an alarm.
8. The order of addresses in a block can be arbitrary.

9. The shape of the NURBS curve can theoretically be modified very flexibly by changing the rank, the positions and weights of the control points, and the knot vector (the relative intervals of knots).

In practice, however, manual editing is almost impossible, and a special CAD/CAM system should be used to edit the NURBS curve and create the program for the interpolation. Generally speaking, do not edit manually the program created by a CAD/CAM system for the NURBS interpolation.

6. Variation of curve according to knot vector

The NURBS curve, which in general passes by the control points, can be made to pass through a specific control point by setting a certain number of knots in succession with the same value. In particular, setting as many leading and trailing knots as the rank (value of P) with the respective identical values will cause the NURBS curve to start from the first control point (P_0) and to end in the last one (P_5).

The examples given below exhibit a variation of the NURBS curve according to the knot vector with the control points remaining identical.

Example 1: Rank : 4
 Number of control points : 6
 Knot vector : [0.0 1.0 2.0 3.0 4.0 5.0 6.0 7.0 8.0 9.0]

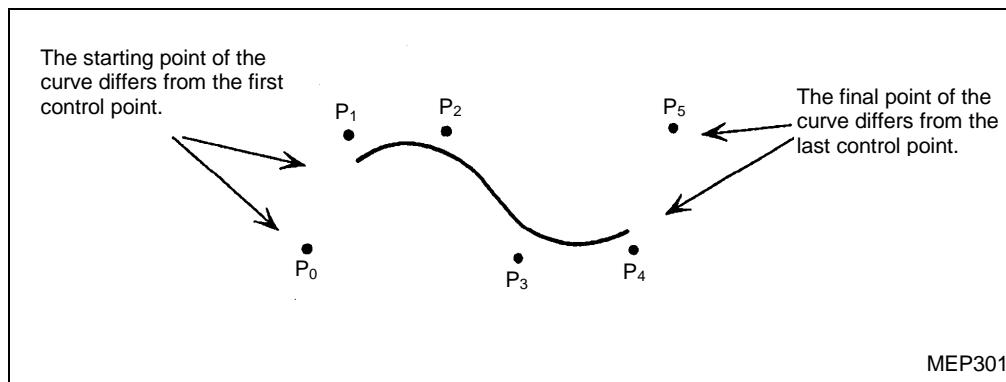


Fig. 6-19 NURBS curve for continuously increasing knots

Example 2: Rank : 4
 Number of control points : 6
 Knot vector : [0.0 0.0 0.0 0.0 0.0 1.0 2.0 3.0 3.0 3.0 3.0]

[1] [2]

Point [1]: The first four (=rank) knots have the same value assigned.

Point [2]: The last four (=rank) knots have the same value assigned.

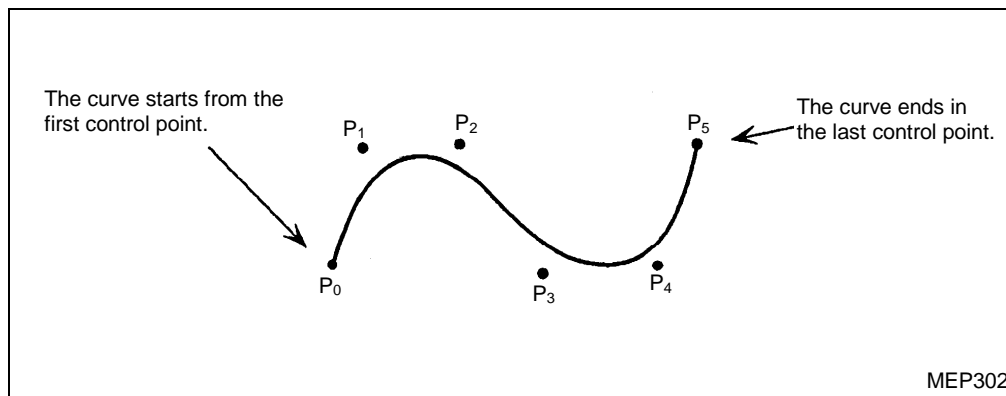


Fig. 6-20 NURBS curve for some identical knots

Note 1: The NURBS interpolation can be performed only for the NURBS curve that starts and ends from the first and in the last control point. Do not fail, therefore, to set as many leading and trailing knots as the rank with the respective identical values.

Note 2: The NURBS interpolation is executed at the designated feed rate (F-code). During the shape correction mode, however, the interpolation speed is controlled in order that the maximum available acceleration may not be exceeded in the section of a considerable curvature.

7. Compatibility with the other functions

The tables in this section specify the compatibility of the NURBS interpolation with the other functions. Pay attention to the incompatible functions, especially G-codes.

A. Preparatory, feed and auxiliary functions

The table below enumerates the G-codes, F-, M-, S-, T- and B-codes with regard to their availability before, with and after G6.2.

○: available x: not available

Function	Code	before G6.2	with G6.2	after G6.2
G-codes of group 00	all	○	x	x
G-codes of group 01	all	○	○ (*)	x
G-codes of group 02	G17	○	○	x
	G18			
	G19			
G-codes of group 04	G22	x	x	x
	G23	○	x	x
G-codes of group 05	G93	○	○	x
	G98			
	G99			
G-codes of group 06	G20	○	○	x
	G21			
G-codes of group 07	G40	○	x	x
	G41	x	x	x
	G42	x	x	x
G-codes of group 09	G80	○	x	x
	the others	x	x	x
G-codes of group 12	G54 - G59	○	○	x
G-codes of group 13	G61.1	○	x	x
	G61.2	○	x	x
	G61	x	x	x
	G62	x	x	x
	G63	x	x	x
	G64	○	x	x
G-codes of group 14	G66	x	x	x
	G66.1	x	x	x
	G66.2	x	x	x
	G67	○	x	x
G-codes of group 16	G68.5	x	x	x
	G69.5	○	x	x
G-codes of group 23	G54.2P0	○	x	x
	G54.2P1 - P8	x	x	x
High-speed machining mode	G5P0	○	x	x
	G5P2	x	x	x
Feed function	F	○	○	x
Auxiliary function	MSTB	○	x	x

(*) The G-code given last in the block takes priority in group 01.

B. Skip instructions

The table below enumerates the skip instructions with regard to their availability before, with and after G6.2.

○: available ×: not available

Instruction	before G6.2	with G6.2	after G6.2
Optional block skip	○	○	×
Control Out/In	○	○	×

Note: Designating another address than X, Y, Z, R and K in the mode of (i. e. after) G6.2 will cause an alarm.

C. Interruption and restart

The table below enumerates the functions for interrupting and restarting the program flow with regard to their availability before, with and after G6.2.

○: available ×: not available

Function	before G6.2	with G6.2	after G6.2
Single-block operation	○	×	○ (Note)
Feed hold	○	×	○
Reset	○	○	○
Programmed stop	○	×	×
Optional stop	○	×	×
Manual interruption (Pulse feed and MDI)	○	×	×
Restart	○	×	×
Comparison stop	○	×	×

Note: The single-block stop only occurs between blocks with different knots.

D. Tool path check

The tool path in a section of the NURBS interpolation can only be displayed as if the control points were linearly interpolated (in the mode of G01).

8. Sample program

The program section below refers to a NURBS interpolation of rank 4 (cubic curve) for seven control points.

```

Control points: P0 P1 P2 P3 P4 P5 P6
Knot vector:   [ 0.0  0.0  0.0  0.0  1.0  2.0  3.0  4.0  4.0  4.0  4.0 ]
                ⋮
                ⋮
G90 G01 X0 Y120.F3000
Y100. .... P0
G6.2 P4 X0 Y100.R1.K0... P0
X10.Y100.R1.K0..... P1
X10.Y60.R1.K0..... P2
X60.Y50.R1.K0..... P3
X80.Y60.R1.K1..... P4
X100.Y40.R1.K2..... P5
X100.Y0 R1.K3..... P6
K4.
K4.
K4.
K4.
G01 X120..... P7
                ⋮
                ⋮

```

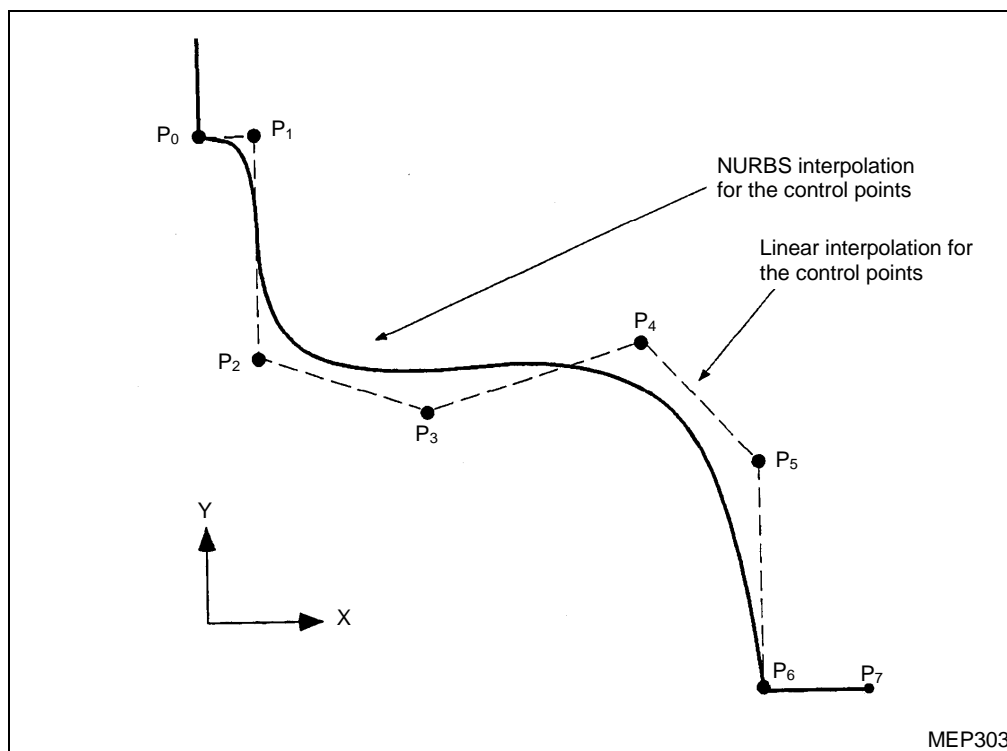


Fig. 6-21 NURBS interpolation and linear interpolation

9. Related alarms

The table below enumerates the alarms related to the NURBS interpolation.

Alarm list

Alarm No.	Alarm message	Cause	Remedy
806	ILLEGAL ADDRESS	Another address than those for the nominated axes (X, Y and/or Z), the weight (R) and the knot (K) is set in the G6.2 mode.	Clear the inadequate address.
807	ILLEGAL FORMAT	1. The modal condition is not appropriate to set G6.2.	1. Satisfy the modal condition with reference to item 7-A.
		2. A block in the G6.2 mode is set without knot (K).	2. Do not fail to set a knot in each block in the G6.2 mode.
		3. The number of blocks with the same knot in succession does not reach the rank.	3. Set an appropriate knot vector with reference to example 2 given in item 6.
809	ILLEGAL NUMBER INPUT	1. The number of digits exceeds the specification of axis commands (X, Y or Z).	1. Specify the axis command within eight digits.
		2. The rank (P) is not admissible.	2. Set 2, 3 or 4 at address P.
		3. The value of a knot is not admissible.	3. Set a value in a range of 0.0001 to 99.9999.
		4. The knot vector is not monotonic increasing.	4. Check the blocks for a decreasing knot.
816	FEEDRATE ZERO	The feed rate (F-code) has not yet been designated.	Set an F-code before or in the same block as the G6.2 code.
936	OPTION NOT FOUND	The system is not equipped with the optional function of the NURBS interpolation.	Purchase and install the optional function.
955	START AND END POINT NOT AGREE	The axis coordinates designated in the block of G6.2 do not correspond to the final point of the previous block.	Designate in the first block of the NURBS interpolation the same position as the final point of the previous block.
956	RESTART OPERATION NOT ALLOWED	The designated restart block falls within the mode of G6.2.	Restart operation is not allowed from the midst of the NURBS interpolation.
957	MANUAL INTERRUPT NOT ALLOWED	An interruption by pulse handle or MDI operation is commanded in the midst of the G6.2 mode.	Manual interruption is not allowed in the midst of the NURBS interpolation.

6-13 Cylindrical Interpolation: G07.1

1. Function and purpose

Cylindrical interpolation refers to a function by which the cylindrical surface of a workpiece can be machined according to a program prepared on its development plane. This function is among others very efficient in the creation of a cam grooving program.

2. Programming format

A. Selection and cancellation of the cylindrical interpolation mode

- When the A-axis functions as the rotational axis:

G07.1 Ar; Cylindrical interpolation mode ON (r = radius of cam groove bottom)

G07.1 A0; Cylindrical interpolation mode OFF

- When the B-axis functions as the rotational axis:

G07.1 Br; Cylindrical interpolation mode ON (r = radius of cam groove bottom)

G07.1 B0; Cylindrical interpolation mode OFF

Note 1: The above preparatory function (G-code) must be given in a single-command block.

Note 2: Enter a precise value for the radius of the cam groove bottom (r), which is used for the internal calculation of the dimensions and the rate of feed in the developed plane.

Note 3: Enter a positive value for the radius of the cam groove bottom (r).

Note 4: In the mode of cylindrical interpolation the radius of the cam groove bottom (r) cannot be otherwise modified than to zero. That is, the modification must be done after canceling the mode temporarily.

Note 5: Cylindrical interpolation is not available for machines with a linear type rotational axis (F85 bit 2 = 1; HV-machines or machines with a tilting table).

Note 6: The figure below refers to a vertical machining center. To perform machining on the surface of the cylinder, position the tool on the Y-axis to the axis of the cylinder first, perform an infeed on the Z-axis even to the groove bottom and then select the mode of a cylindrical interpolation with simultaneous control of the X- and A-axis.

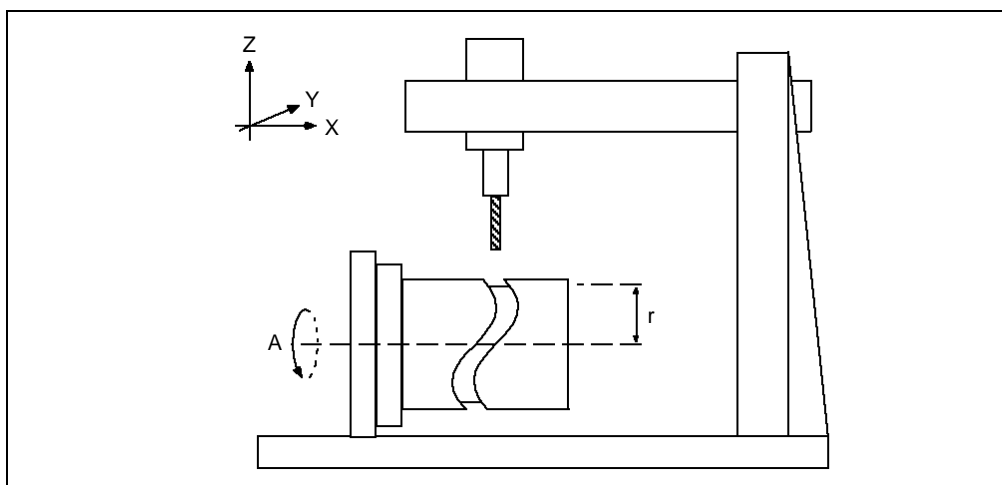


Fig. 6-22 Diagrammatic view of cylindrical interpolation (1/2)

Note 7: The next figure shows a machining center with the fixture mounted perpendicular to the Y-axis. To use the cylindrical interpolation on such a machine, position the tool on the X-axis to the axis of the cylinder first, perform an infeed on the Z-axis even to the groove bottom and then select the mode of a cylindrical interpolation with simultaneous control of the Y- and B-axis.

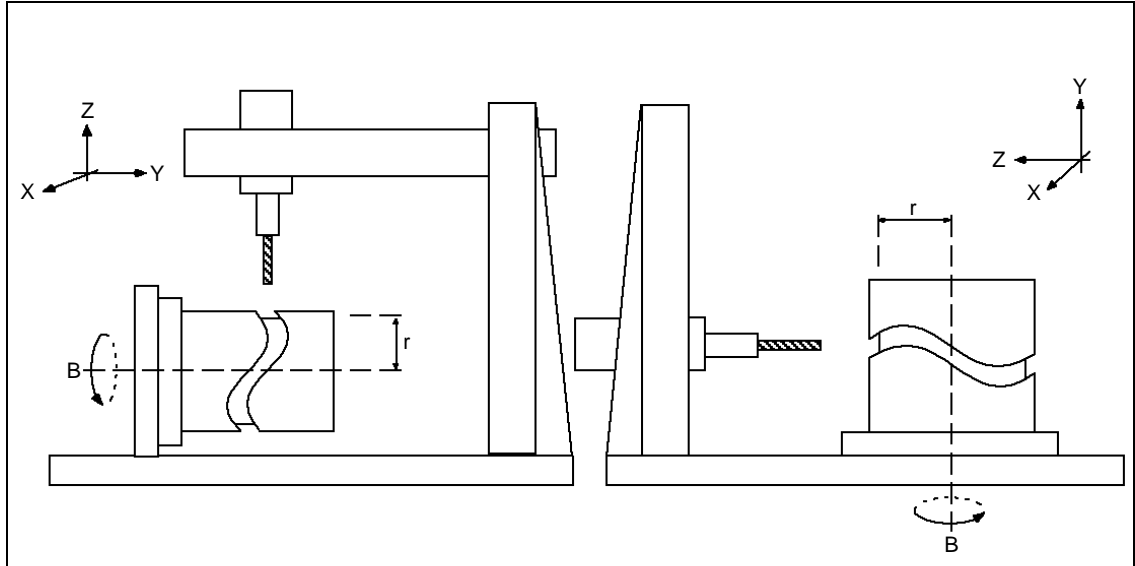


Fig. 6-23 Diagrammatic view of cylindrical interpolation (2/2)

B. Commands in the cylindrical interpolation mode

GgXxAaRrIiJjDdPpFf ;

GgYyBbRrIiJjDdPpFf ;

Gg: Refer to Table 6-2 (in paragraph 1 under 3-B) for the available G-codes.

Xx/Aa: } Cylindrical interpolation occurs in the X-A plane or Y-B plane.

Yy/Bb: } The values of X (or Y) and A (or B) are of linear and angular dimensions, respectively.

Rr: Radius for a circular interpolation

Ii: Abscissa (incremental) of the center for a circular interpolation

Jj: Ordinate (incremental) of the center for a circular interpolation

Dd: Offset number for tool radius compensation

Pp/Xx: Dwell time

Ff: Rate of feed (Refer to the detailed description in paragraph 2 under 3-B.)

3. Detailed description

A. Conditions necessary for the selection of the cylindrical interpolation mode

The G-code modal states required for the cylindrical interpolation mode selection are as follows:

Table 6-1 Conditions necessary for the mode selection

G-code group	Condition
G-code group 1	G0/G1 (Positioning or Linear interpolation only)
G-code group 2	G17 (X-Y plane only)
G-code group 3	G90/G91 (Absolute/Incremental programming) [unconditional]
G-code group 4	G22/G23 (Stroke check ON/OFF) [unconditional]
G-code group 5	G94/G95 (Asynchronous/Synchronous feed) [unconditional]
G-code group 6	G20/G21 (Inch/Metric data input) [unconditional]
G-code group 7	G40 (Tool radius compensation OFF) (Note 1)
G-code group 8	G43/G49 (Tool length offset ON/OFF) [unconditional]
G-code group 9	G80 (Fixed cycle OFF)
G-code group 10	Invalid (Return level selection [between initial and R-point] is only valid for fixed cycles.)
G-code group 11	G50 (Scaling OFF)
G-code group 12	G54 to G59, G54.1 (Standard/Additional workpiece coordinate system) [unconditional]
G-code group 13	G64 (Cutting mode)
G-code group 14	G67 (User macro modal call OFF)
G-code group 15	G40.1 (Shaping OFF)
G-code group 16	G69 (Programmed coordinates rotation OFF) (Note 2)
G-code group 19	G50.1/G51.1 (Mirror image ON/OFF) [unconditional] (Note 3)
G-code group 19	No selection of a plane for five-surface machining
G-code group 23	G54.2P0 (Dynamic offsetting II OFF)
Others	G5P0 (High-speed machining OFF)
Others	G7.1B0 (Cylindrical interpolation OFF)

Otherwise the selection will only lead to an alarm.

Note 1: Select and cancel the tool radius compensation as required in the mode of cylindrical interpolation. An alarm will be caused if the cylindrical interpolation is selected in the mode of tool radius compensation.

Note 2: The cylindrical interpolation cannot be selected in the mode of G68 (3-dimensional coordinate conversion). On an HV machining center, therefore, the cylindrical interpolation function is not available to an inclined or top surface.

Note 3: To use the cylindrical interpolation with the mirror image function being selected, take the following precautions in order to prevent errors from occurring in the development of the cylindrical surface:

- [1] Set the mirroring center to 0° for the rotational axis of the cylindrical interpolation.
- [2] Select the cylindrical interpolation with the rotational axis being positioned at its workpiece origin (0°).
- [3] Also cancel the cylindrical interpolation with the rotational axis being positioned at its workpiece origin (0°).

An example of programming is given later in the sample program under 5-B.

B. Commands in the cylindrical interpolation mode

- The table below enumerates the G-codes available in the cylindrical interpolation mode. Any other G-code will cause an alarm.

Table 6-2 Available G-codes

G-code	Function
G0	Rapid positioning
G1, G2, G3	Linear and circular interpolation
G4	Dwell
G9	Exact-stop check
G17	Plane selection (Note 1)
G40, G41, G42	Tool radius compensation (Note 2)

Note 1: After the mode selection by a block of G7.1, do not fail to give a plane selection command of the following format in order to specify the cylindrical interpolation plane determined by the corresponding linear and rotational axes:

G17X__A__ when the A-axis functions as the rotational axis.

G17Y__B__ when the B-axis functions as the rotational axis.

Note 2: Select and cancel the tool radius compensation as required in the mode of cylindrical interpolation. An alarm will be caused if the cylindrical interpolation is selected in the mode of tool radius compensation.

- In the mode of the cylindrical interpolation the rate of feed refers to a resultant speed (of Fx and Fa, or Fy and Fb) in the plane onto which the surface of the cylinder is developed.

The speed on each component axis is calculated for a block of G1XxAaFf; as follows:

$$F_x = \frac{x}{\sqrt{x^2 + \left(\frac{a}{360} 2\pi r\right)^2}} \cdot f$$

$$F_a = \frac{\frac{a}{360} 2\pi r}{\sqrt{x^2 + \left(\frac{a}{360} 2\pi r\right)^2}} \cdot f$$

x: metric (0.001 mm)
a: degree (0.001°)
r: radius of cam groove bottom
f: command value of speed

The speed on each component axis is calculated for a block of G1YyBbFf; as follows:

$$F_y = \frac{y}{\sqrt{y^2 + \left(\frac{b}{360} 2\pi r\right)^2}} \cdot f$$

$$F_b = \frac{\frac{b}{360} 2\pi r}{\sqrt{y^2 + \left(\frac{b}{360} 2\pi r\right)^2}} \cdot f$$

y: metric (0.001 mm)
b: degree (0.001°)
r: radius of cam groove bottom
f: command value of speed

The speed of rapid traverse and the upper limit of cutting feed, both specified in a parameter, are expressed in an angular velocity (°/min) for a rotational axis. The actual linear speed on the rotational axis of the cylindrical interpolation is therefore allowed to increase in the developed plane just in proportion to the radius of the groove bottom.

4. Remarks

A. Positioning accuracy (on the rotational axis)

Each angular dimension entered is internally converted into linear one on the circumference, which is to be used in the calculation of the interpolation with the other linear axis. The actual angular motion is then determined by the results of that calculation. As a result, depending on the cylinder radius, positioning errors on the rotational axis may occur in the level of the least significant digit, but they are not cumulative.

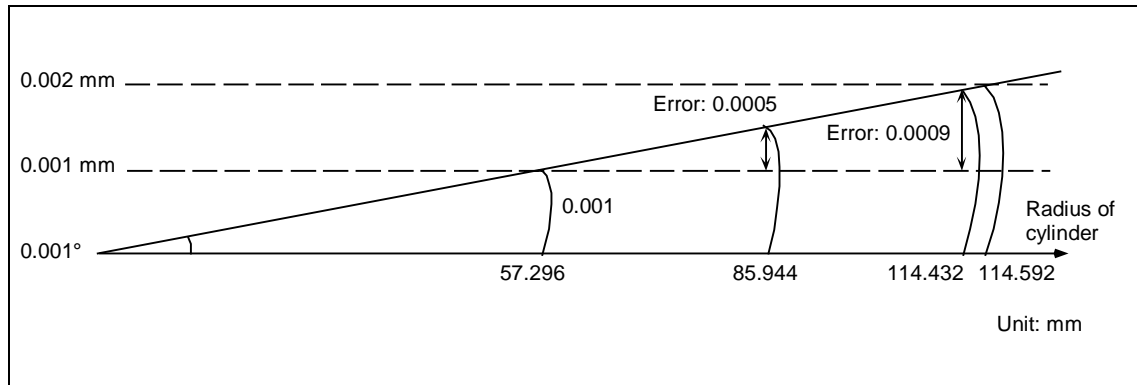


Fig. 6-24 Positioning error according to the angle and radius

B. Changing the radius of the groove bottom

In the mode of cylindrical interpolation, as mentioned before, the radius of the cam groove bottom cannot be otherwise modified than to zero. That is, the modification must be done after canceling the mode temporarily.

C. Rotational axis of the cylindrical interpolation

Only one rotational axis can be used for the cylindrical interpolation. It is not possible to designate multiple rotational axes in a command of G7.1.

D. Manual interruption

1. With "manual absolute" ON

The "manual absolute" function is suspended during cylindrical interpolation, and the first motion block after the cancellation of cylindrical interpolation is executed for the very target position as programmed by canceling the amount of manual interruption.

Refer to the example given later under 5-D.

2. With "manual absolute" OFF

The amount of manual interruption remains intact, irrespective of the selection and cancellation of cylindrical interpolation. See the example given under 5-D.

E. Restart operation

For restarting in the middle of the cylindrical interpolation, follow the normal restart procedure to ensure normal operation by retrieving the necessary modal information (on the radius of groove bottom, etc.). Never use the [RESTART 2] menu function that skips the preceding blocks.

F. Resetting

Resetting (by the RESET key on the operating panel) cancels the cylindrical interpolation mode.

5. Sample programs

Note: The examples in this section are all given for a cylindrical interpolation in the Y-B plane (with respect to a machining center as shown in Fig. 6-23).

Replace the axis names X, Y and B with Y, X and A, respectively, for a cylindrical interpolation in the X-A plane (on a vertical machining center as shown in Fig. 6-22).

A. Cam grooving program

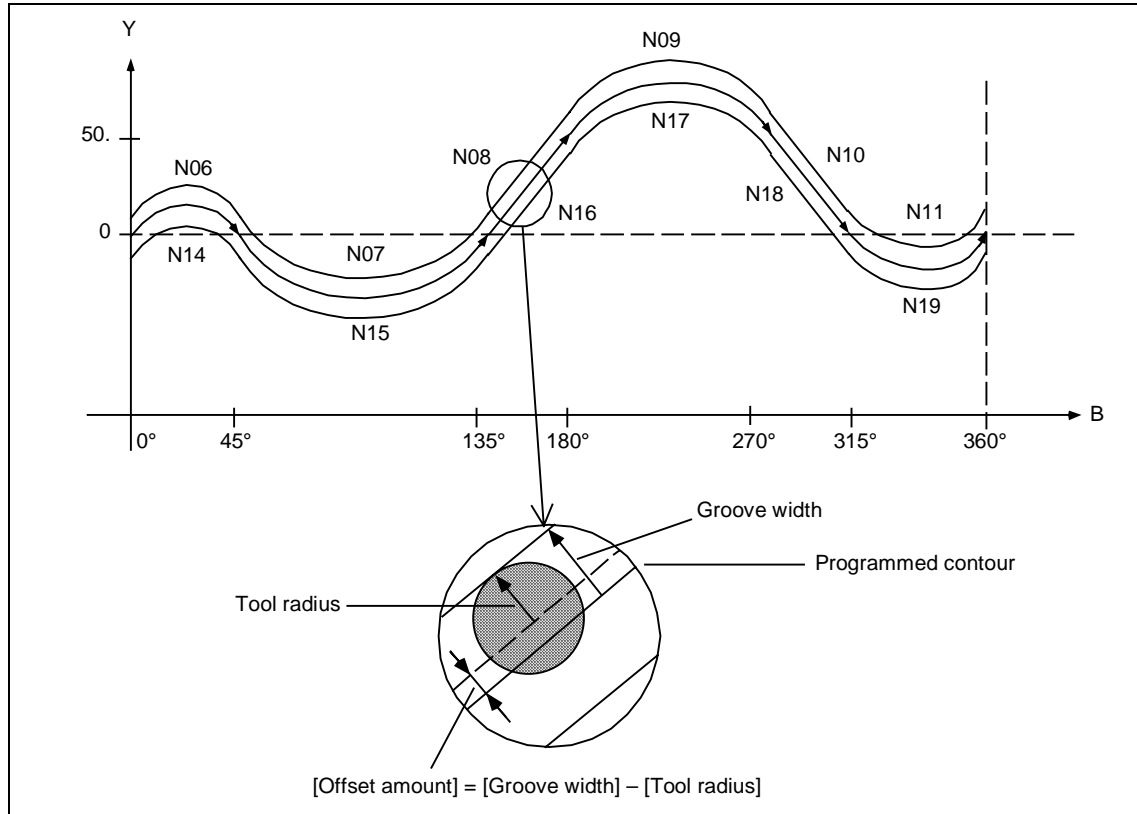


Fig. 6-25 Cam grooving program

```

N01 G54G0G90X0Y0B0; .. Positioning to the cylinder axis
N02 Z0S800M3;
N03 G1Z-5.F2000; ..... Infeed to the groove bottom
N04 G7.1B63.662; ..... Cylindrical interpolation ON
N05 G17G1G41Y0B0D1;
N06 G2B45.R30.;
N07 G3B135.R60.;
N08 G1Y50.B180.;
N09 G2B270.R60.;
N10 G1Y0B315.;
N11 G3B360.R30.;
N12 G1G40;
N13 G1G42Y0B360.;
N14 G2B45.R30.;
N15 G3B135.R60.;
N16 G1Y50.B180.;
N17 G2B270.R60.;
N18 G1Y0B315.;
N19 G3B360.R30.;
N20 G1G40;
N21 G7.1B0; ..... Cylindrical interpolation OFF

```

B. Use of mirror image function

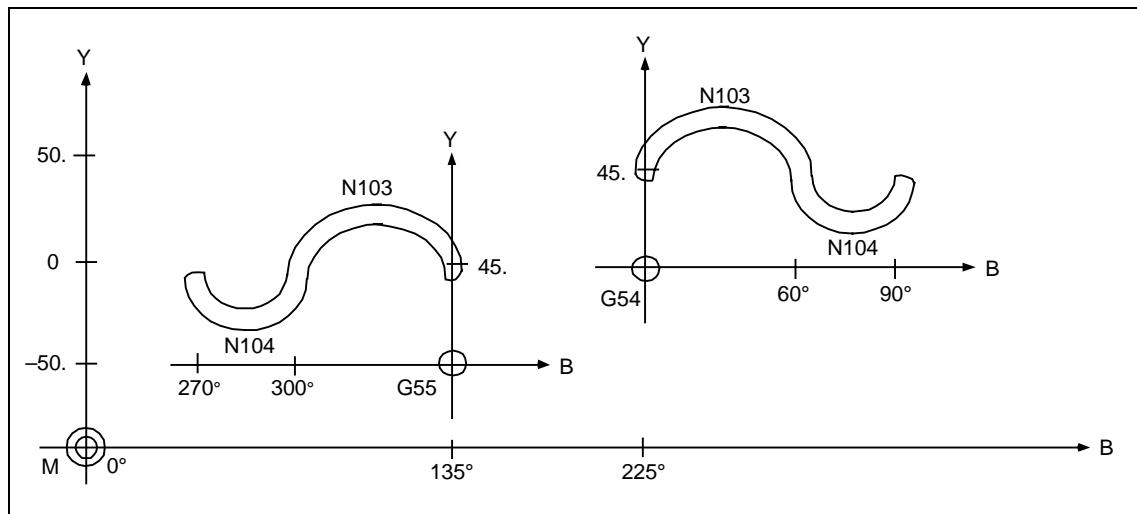


Fig. 6-26 Use of mirror image function

Main program

```

N01 G54G0G17G90X0Y0B0;
N02 Z0S800M3;
N03 M98P2000;
N04 G55G0G17G90X0Y0B0;
N05 Z0S800M3;
N06 G51.1B0; .....Mirror image ON
N07 M98P2000;
N08 G50.1B0; .....Mirror image OFF

```

Subprogram

```

N100 G7.1 B47.746 ;
N101 G17 G0 Y45.B0 ;
N102 G1 Z-5.F1000 ;
N103 G2 B60.R25. ;
N104 G3 B90.R12.5 ;
N105 G0 Z0 ;
N106 G0 B0 ;
N107 G7.1 B0 ;
N108 M99 ;

```

Table 6-3 Workpiece origin data

	G54	G55
X	0.	0.
Y	0.	-50.
Z	0.	0.
B	225.	135.

C. Use of circular interpolation commands

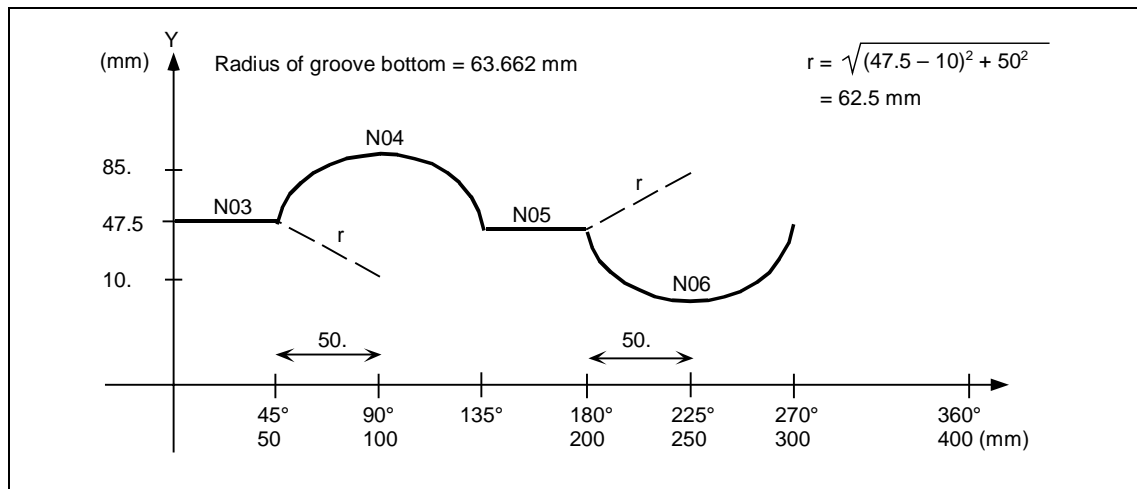


Fig. 6-27 Use of circular interpolation commands

```

N01 G90 G1 Y47.5 F1000 ;
N02 G7.1 B63.662 ;
N03 G17 G1 Y47.5 B45. ;
N04 G2 B90.R62.5 ;
N05 G1 B180. ;
N06 G3 B270.I50.J37.5 ;

```

D. Operation examples with manual interruption

1. With "manual absolute" ON

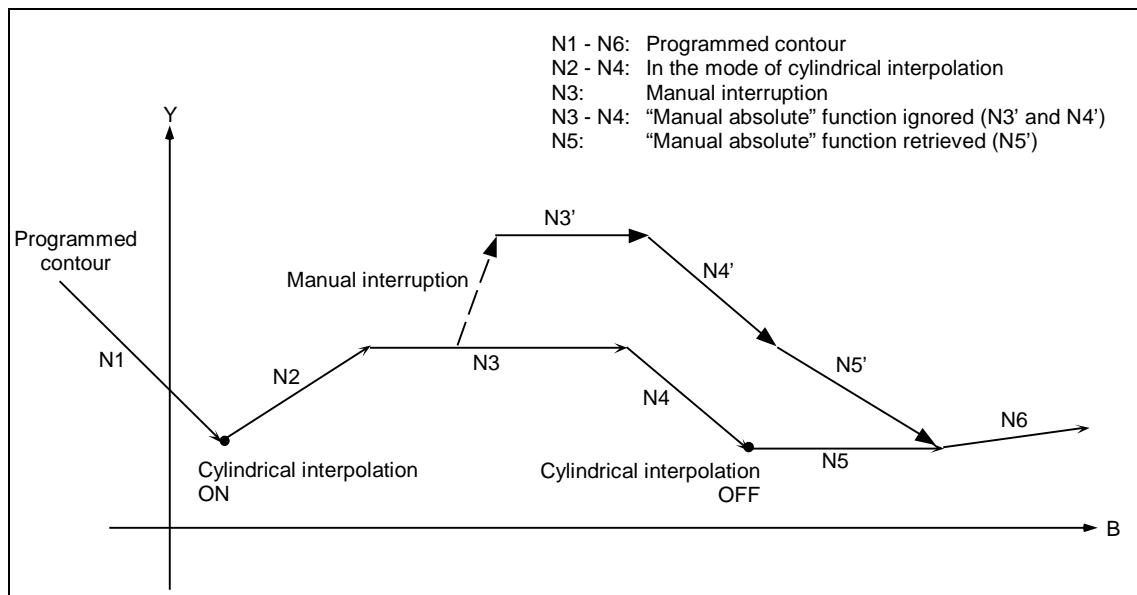


Fig. 6-28 With "manual absolute" ON

2. With “manual absolute” OFF

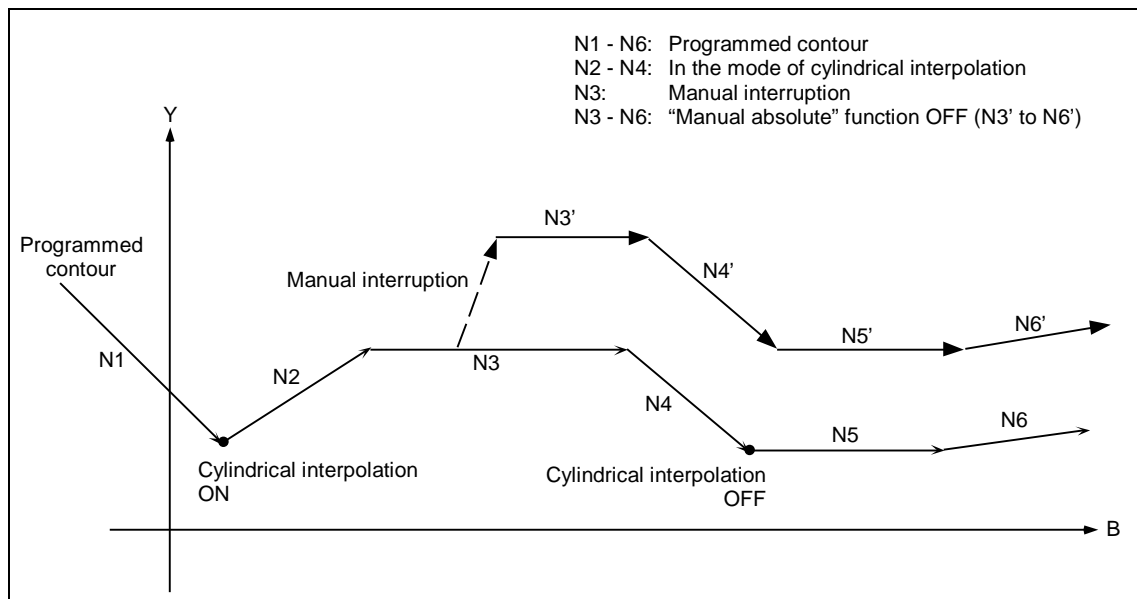


Fig. 6-29 With “manual absolute” OFF

6. Related parameters

- Set the type of the rotational axis (A or B) as “rotational” (with short-cut approach) in order that the initial angular movement should not occur on the roundabout route for a command of tool radius compensation given in the vicinity of the 0° position.

N21 bit 0 = 0: Setting the type of the rotational axis as rotational
= 1: Short-cut approach valid

- Set the B- and A-axis as the primary parallel ones to the axis of abscissa and that of the ordinate, respectively.

SU2 = 66: Setting the B-axis as parallel axis 1 for the axis of abscissa

SU5 = 65: Setting the A-axis as parallel axis 1 for the axis of the ordinate

- The following parameter setting is required for a cylindrical interpolation in the X-A plane on vertical machining centers:

F85 bit 7 = 1

- The following parameter setting is required for inch data input in order that the speed of the angular movement may be calculated appropriately:

F85 bit 4 = 1

7. Related alarms

Table 6-4 Related alarms

Alarm No.	Alarm message	Description	Remedy
806	ILLEGAL ADDRESS	The address of argument in the block of selecting the cylindrical interpolation does not refer to any rotational axis.	Use the correct address.
2110	ILLEGAL FORMAT	The necessary conditions for the selection of cylindrical interpolation are not yet all satisfied.	Refer to Table 6-1.
808	MIS-SET G CODE	An unavailable G-code is given in the mode of cylindrical interpolation.	Refer to Table 6-2.
936	OPTION NOT FOUND (7, 0, 0)	The optional function for cylindrical interpolation is not provided.	Equip the machine with the option as required.

6-14 Helical Interpolation: G02, G03

1. Function and purpose

Command G02 or G03 with a designation for the third axis allows synchronous circular interpolation on the plane specified by plane-selection command G17, G18 or G19 with the linear interpolation on the third axis.

2. Programming format

G17 G02 $\underline{Xx_1} \underline{Yy_1} \underline{Zz_1} \underline{Ii_1} \underline{Jj_1} \underline{Pp_1} \underline{Ff_1};$
 (G03)

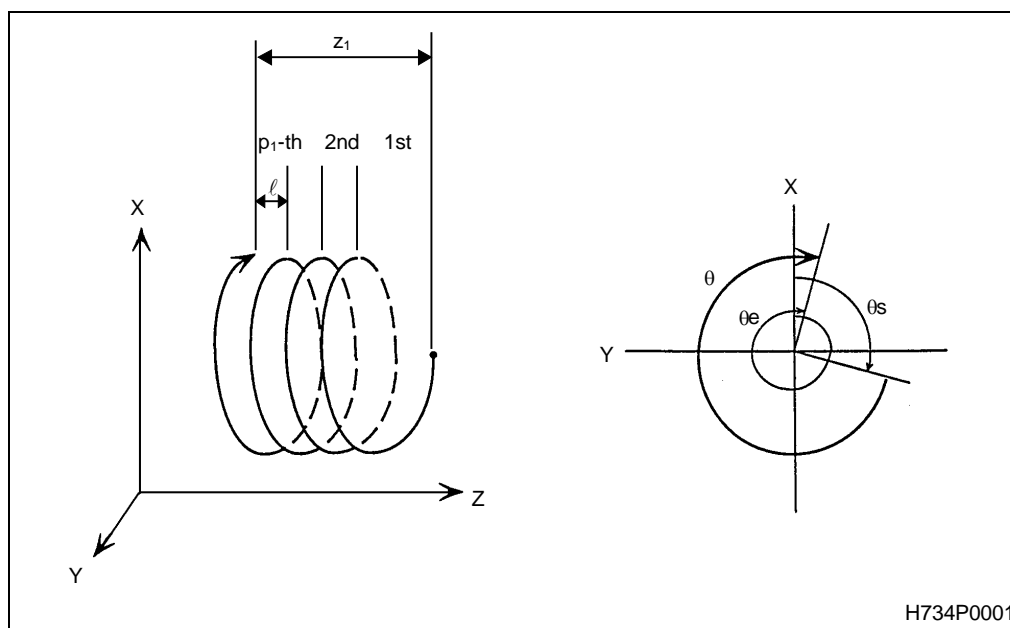
Feed rate
 Number of pitches
 Arc center coordinates
 Linear axis ending point coordinate
 Arc ending point coordinates

or

G17 G02 $\underline{Xx_2} \underline{Yy_2} \underline{Zz_2} \underline{Rr_2} \underline{Pp_2} \underline{Ff_2};$
 (G03)

Feed rate
 Number of pitches
 Arc radius
 Linear axis ending point
 Arc ending point coordinates

3. Detailed description



1. For helical interpolation, movement designation is additionally required for one to two linear axes not forming the plane for circular interpolation.
2. The velocity in the tangential direction must be designated as the feed rate F.
3. The pitch ℓ is calculated as follows:

$$\ell = \frac{Z_1}{(2\pi \cdot p_1 + \theta)/2\pi}$$

$$\theta = \theta_e - \theta_s = \tan^{-1} \frac{y_e}{x_e} - \tan^{-1} \frac{y_s}{x_s} \quad (0 \leq \theta < 2\pi)$$

where (x_s, y_s) : relative coordinates of starting point with respect to the arc center

(x_e, y_e) : relative coordinates of ending point with respect to the arc center

4. Address P can be omitted if the number of pitches is 1.

5. Plane selection

As with circular interpolation, the circular-interpolation plane for helical interpolation is determined by the plane-selection code and axis addresses. The basic programming procedure for helical interpolation is: selecting a circular-interpolation plane using a plane-selection command (G17, G18 or G19), and then designating the two axis addresses for circular interpolation and the address of one axis (perpendicular to the circular-interpolation plane) for linear interpolation.

- XY-plane circular, Z-axis linear

After setting G02 (or G03) and G17 (plane-selection command), set the axis addresses X, Y and Z.

- ZX-plane circular, Y-axis linear

After setting G02 (or G03) and G18 (plane-selection command), set the axis addresses Z, X and Y.

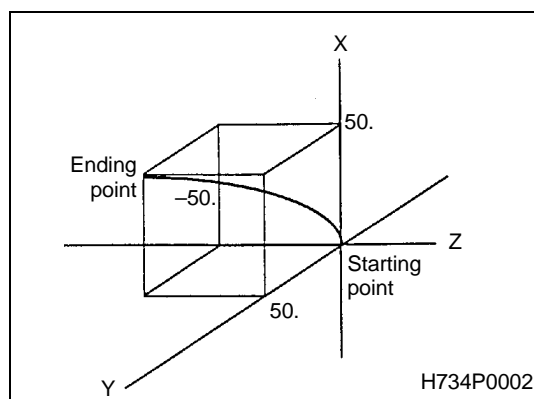
- YZ-plane circular, X-axis linear

After setting G02 (or G03) and G19 (plane-selection command), set the axis addresses Y, Z and X.

4. Sample programs

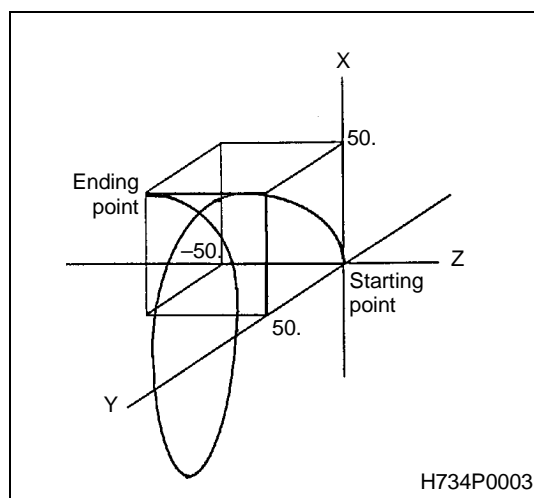
Example 1:

```
G91 G28 X0 Y0 Z0;
G92 X0 Z0 Y0;
G17 G03 X50. Y50. Z-50. R50. F1000;
```



Example 2:

```
G91 G28 X0 Y0 Z0;
G92 X0 Z0 Y0;
G17 G03 X50. Y50. Z-50. R50. P2 F1000;
```



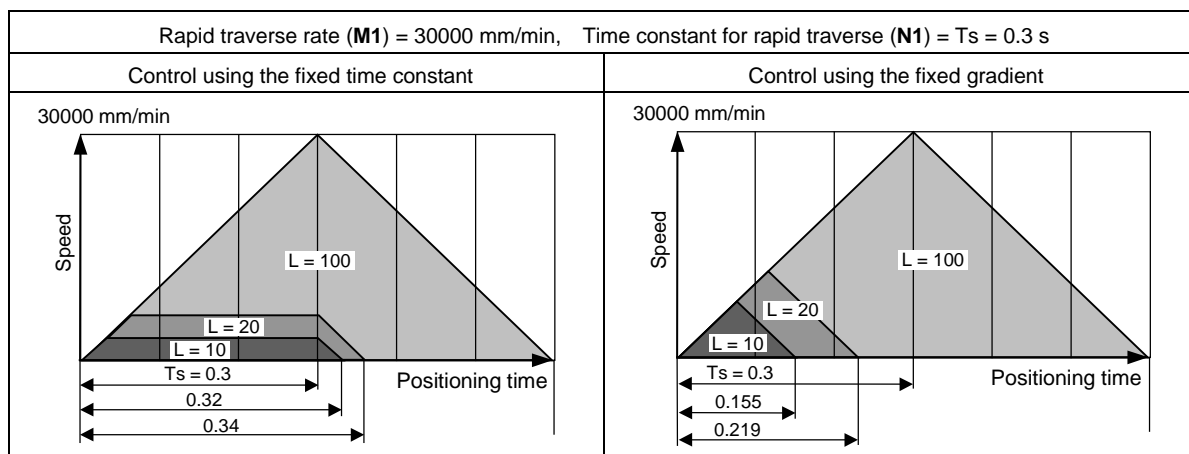
6-15 Fixed Gradient Control for G0 (Option)

1. Function and purpose

Fixed gradient control for G0 refers to a method of executing rapid traverse commands (under G00) using linear acceleration and deceleration of a fixed gradient. The gradient in question denotes an acceleration and corresponds to the quotient of the rapid traverse rate (parameter **M1**) and the time constant concerned (parameter **N1**). As compared with the control using the fixed time constant, positioning time can be reduced for a distance traveled which is shorter than is required for acceleration up to the rapid traverse rate and immediately succeeding deceleration to zero.

2. Example for comparison with the control using the fixed time constant

Given below is an example to illustrate the difference in positioning time by rapid traverse over distance L between the two control methods.



3. Remarks

1. The optional function in question works on the correspondingly executed machines when rapid traverse of interpolation type by G0 and the fixed gradient control for G0 are made valid by the corresponding parameter settings (**F91** bit 6 = 0, and **K96** bit 7 = 1).
2. As for the operation during five-axis machining (in the mode of tool tip point control, inclined-plane machining, and workpiece setup error correction), see the description given under the restrictions for the relevant control modes.

- NOTE -

7 FEED FUNCTIONS

7-1 Rapid Traverse Rates

A separate rapid traverse rate can be set for each axis. The maximum rate of rapid traverse, however, is limited according to the particular machine specifications.

Refer to the Operating manual for the machine for rapid traverse rates.

Two types of tool paths are available for positioning: an interpolation type, which uses a line to perform interpolation from the starting point through the ending point, and a non-interpolation type, which moves the tool at the maximum speed of each axis.

Use a parameter to select the interpolation type or the non-interpolation type. The positioning time is the same for both types.

7-2 Cutting Feed Rates

A cutting feed rate must be designated using address F and an eight-digit number (F8-digit direct designation).

The F8 digits must consist of five integral digits and three decimal digits, with the decimal point. Cutting feed rates become valid for commands G01, G02, G03, G33 and G34.

Example:

	Feed rate
G01 X100. Y100. F200*	200.0 mm/min
G01 X100. Y100. F123.4	123.4 mm/min
G01 X100. Y100. F56.789	56.789 mm/min

* It means the same if F200. or F200.000 is set in stead of F200.

Note: An alarm (No. 713) will result if a feed rate command is not set for the first cutting command (G01, G02, G03, G33 or G34) that is read firstly after power-on.

7-3 Asynchronous/Synchronous Feed: G94/G95

1. Function and purpose

Command G95 allows a feed rate per revolution to be set using an F-code.

Command G94 retrieves the mode of asynchronous feed (per minute).

2. Programming format

G94: Feed per minute (/min) [Asynchronous feed]

G95: Feed per revolution (/rev) [Synchronous feed]

Since command G95 is modal, it will remain valid until command G94 is issued.

3. Detailed description

1. Feed rates that can be set using F-codes are listed in the table below.
The table below also lists synchronous feed rates, which are to be set in millimeters (or inches) per spindle revolution using F-codes.

	G94F_ (Feed per minute)	G95F_ (Feed per revolution)
Input in mm	1 to 240000 mm/min (F1 to F240000)	0.0001 to 500.0000 mm/rev (F1 to F5000000)
Input in inches	0.01 to 9600.00 in/min (F1 to F960000)	0.000001 to 9.999999 in/rev (F1 to F9999999)

2. The effective feed rate per revolution, that is, the actual moving speed of the machine, can be calculated as follows:

$$FC = F \times N \times OVR \text{ (Expression 1)}$$

where FC: Effective feed rate (mm/min or in/min)
F: Designated feed rate (mm/rev or in/rev)
N: Spindle speed (min^{-1})
OVR: Cutting feed override

If multiple axes are selected at the same time, effective feed rate FC given by expression 1 above will become valid for the corresponding vectorial direction.

4. Remarks

1. On the **POSITION** display, FEED denotes an effective feed rate that is expressed in a feed rate per minute (mm/min or in/min), based on the selected feed rate, the spindle speed, and the cutting feed override.
2. If the effective feed rate should become larger than the cutting feed limit speed, that limit speed will govern.
3. In the dry run mode, feed will become asynchronous and the machine will operate at an externally preset feed rate (mm/min or in/min).
4. According to the setting of bit 1 of parameter **F93**, synchronous or asynchronous feed mode (G95 or G94) is automatically made valid upon power-on or by execution of M02 or M30.

7-4 Selecting a Feed Rate and Effects on Each Control Axis

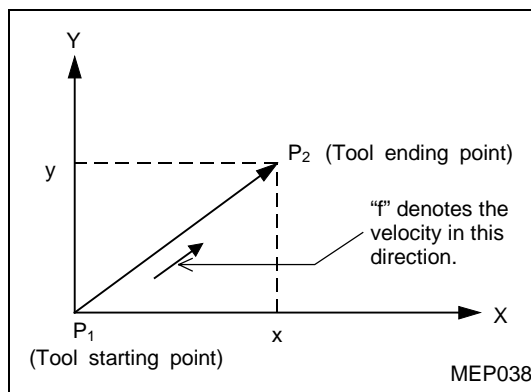
As mentioned earlier, the machine has various control axes. These control axes can be broadly divided into linear axes, which control linear motions, and rotational axes, which control rotational motions. Feed rates for control axes have different effects on the tool speed, which is of great importance for machining quality, according to the particular type of axis controlled.

The amount of displacement must be designated for each axis, whereas the feed rate is to be designated as a single value for the intended tool movement. Before letting the machine control two or more axes at the same time, therefore, you must understand how the feed rate designated will act on each axis. In terms of this, selection of a feed rate is described below.

1. Controlling linear axes

The feed rate that has been selected using an F-code acts as a linear velocity in the moving direction of the tool, irrespective of whether only one axis is to be controlled or multiple axes simultaneously.

Example: If linear axes (X- and Y-axes) are to be controlled using a feed rate of f :



When only linear axes are to be controlled, setting of a cutting feed rate itself is only required. The feed rate for each axis refers to that component of the specified feed rate which corresponds with the ratio of movement stroke on the respective axis to the actual movement distance.

In the example shown above:

$$\text{X-axis feed rate} = f \times \frac{x}{\sqrt{x^2 + y^2}}$$

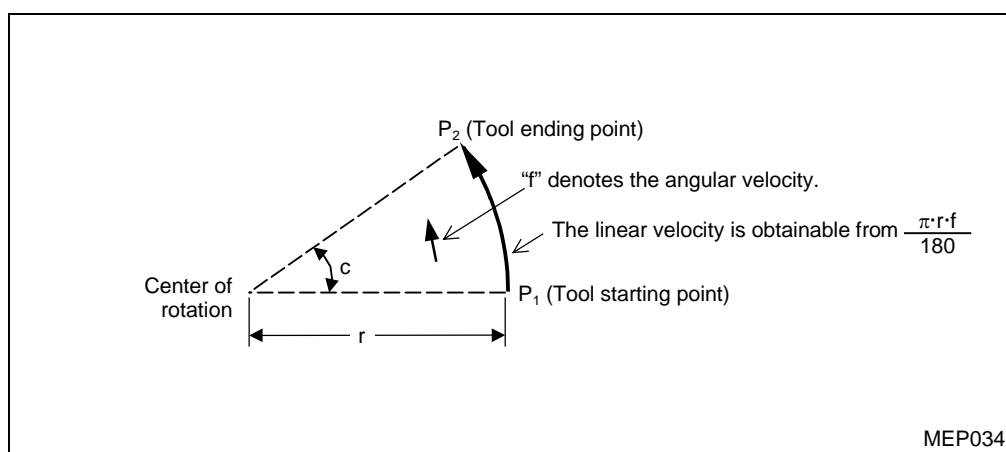
$$\text{Y-axis feed rate} = f \times \frac{y}{\sqrt{x^2 + y^2}}$$

2. Controlling a rotational axis

When a rotational axis is to be controlled, the selected feed rate acts as the rotating speed of the rotational axis, that is, as an angular velocity.

Thus, the cutting speed in the moving direction of the tool, that is, a linear velocity varies according to the distance from the rotational center to the tool. This distance must be considered when setting a feed rate in the program.

Example 1: If a rotational axis (C-axis) is to be controlled using a feed rate of f (deg/min):



In this case, the cutting speed in the moving direction of the tool (linear velocity) " f_c " is calculated by:

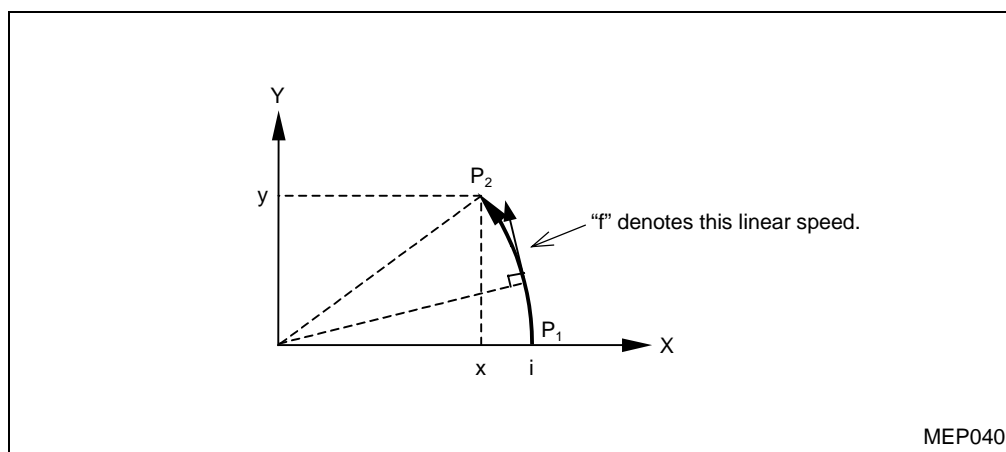
$$f_c = f \times \frac{\pi \cdot r}{180}$$

Hence, the feed rate to be programmed for the required value f_c is:

$$f = f_c \times \frac{180}{\pi \cdot r}$$

Note: If the tool is to be moved by controlling linear axes along the circumference using the circular interpolation function, the feed rate programmed is the velocity acting in the moving direction of the tool, that is, in the tangential direction.

Example 2: If linear axes (X- and Y-axes) are to be controlled at a feed rate of f using the circular interpolation function:



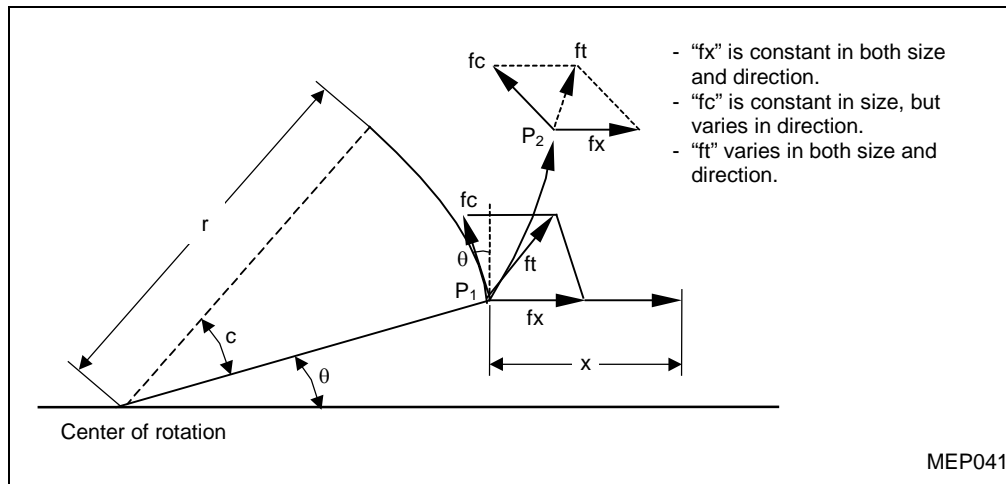
In this case, the X- and Y-axis feed rates will change with the movement of the tool. The resultant velocity, however, will be kept at the constant value, f .

3. Controlling a linear axis and a rotational axis at the same time

The NC unit controls linear axes and rotational axes in exactly the same manner.

For control of rotational axes, data given as a coordinate word (with A, B or C) is handled as an angle, and data given as a feed rate (F) is handled as a linear velocity. In other words, an angle of one degree for a rotational axis is handled as equivalent to a moving distance of 1 mm for a linear axis. Thus, for simultaneous control of a linear axis and a rotational axis, the magnitudes of the individual axis components of the data that has been given by F are the same as those existing during linear axis control described previously in Subparagraph 1. above. In this case, however, the velocity components during linear axis control remain constant in both magnitude and direction, whereas those of rotational axis control change in direction according to the movement of the tool. Therefore, the resulting feed rate in the moving direction of the tool changes as the tool moves.

Example: If a linear axis (X-axis) and a rotational axis (C-axis) are to be controlled at the same time at a feed rate of f:



X-axis incremental command data is expressed here as x, and that of C-axis as c.

The X-axis feed rate (linear velocity), f_x , and the C-axis feed rate (angular velocity), ω , can be calculated as follows:

$$f_x = f \times \frac{x}{\sqrt{x^2 + c^2}} \quad \dots\dots [1] \quad \omega = f \times \frac{c}{\sqrt{x^2 + c^2}} \quad \dots\dots [2]$$

The linear velocity "fc" that relates to C-axis control is expressed as:

$$f_c = \omega \cdot \frac{\pi \cdot r}{180} \quad \dots\dots [3]$$

If the velocity in the moving direction of the tool at starting point P_1 is taken as "ft", and its X- and Y-axis components as "ftx" and "fty" respectively, then one can express "ftx" and "fty" as follows:

$$f_{tx} = -r \sin \left(\frac{\pi}{180} \theta \right) \times \frac{\pi}{180} \omega + f_x \quad \dots\dots [4]$$

$$f_{ty} = -r \cos \left(\frac{\pi}{180} \theta \right) \times \frac{\pi}{180} \omega \quad \dots\dots [5]$$

where r denotes the distance (in millimeters) from the rotational center to the tool, and q denotes the angle (in degrees) of starting point P_1 to the X-axis at the rotational center.

From expressions [1] through [5] above, the resultant velocity “ft” is:

$$\begin{aligned}
 ft &= \sqrt{ftx^2 + fty^2} \\
 &= f \times \frac{\sqrt{x^2 - x \cdot c \cdot r \sin\left(\frac{\pi}{180} \theta\right) \frac{\pi}{90} + \left(\frac{\pi \cdot r \cdot c}{180}\right)^2}}{\sqrt{x^2 + c^2}} \dots\dots [6]
 \end{aligned}$$

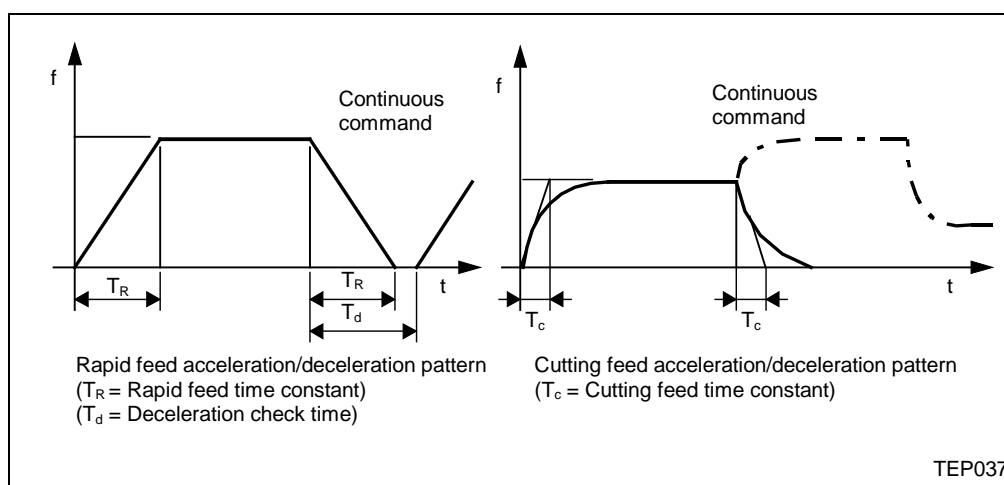
The feed rate f that is to be set in the program must be therefore:

$$f = ft \times \frac{\sqrt{x^2 + c^2}}{\sqrt{x^2 - x \cdot c \cdot r \sin\left(\frac{\pi}{180} \theta\right) \frac{\pi}{90} + \left(\frac{\pi \cdot r \cdot c}{180}\right)^2}} \dots\dots [7]$$

In expression [6], “ft” is the velocity at starting point P₁ and thus the value of ft changes with that of θ which changes according to the rotational angle of the C-axis. To keep cutting speed “ft” as constant as possible, the rotational angle of the C-axis in one block must be minimized to ensure a minimum rate of change of θ.

7-5 Automatic Acceleration/Deceleration

The rapid traverse and manual feed acceleration/deceleration pattern is linear acceleration and linear deceleration. Time constant T_R can be set independently for each axis using parameters in 1 ms steps within a range from 1 to 500 ms. The cutting feed (not manual feed) acceleration/deceleration pattern is exponential acceleration/deceleration. Time constant T_C can be set independently for each axis using parameters in 1 ms steps within a range from 1 to 500 ms. (Normally, the same time constant is set for each axis.)



During rapid traverse and manual feed, the following block is executed after the command pulse of the current block has become “0” and the tracking error of the acceleration/deceleration circuit has become “0”. During cutting feed, the following block is executed as soon as the command pulse of the current block becomes “0” and also the following block can be executed when an external signal (error detection) can detect that the tracking error of the acceleration/deceleration circuit has reached “0”. When the in-position check has been made valid (selected by machine parameter) during the deceleration check, it is first confirmed that the tracking error of the acceleration/deceleration circuit has reached “0”, then it is checked that the position deviation is less than the parameter setting, and finally the following block is executed.

7-6 Limitation of Speed

This function exercises control over the actual cutting feed rate in which override has been applied to the cutting feed rate command so that the speed clamp value preset independently for each axis is not exceeded.

Note: Speed limitation is not applied to synchronous feed.

7-7 Exact-Stop Check: G09

1. Function and purpose

Only after the in-position status has been checked following machine deceleration and stop or after deceleration checking time has been passed, may you want to start the next block command in order to reduce possible machine shocks due to abrupt changes in tool feed rate and to minimize any rounding of workpieces during corner cutting. An exact-stop check function is provided for these purposes.

2. Programming format

G09 G01 (G02, G03) ;

Exact-stop check command G09 is valid only for the cutting command code (G01, G02, or G03) that has been set in that block.

3. Sample program

N001 G09 G01 X100.000 F150; The next block is executed after an in-position status check following machine deceleration and stop.

N002 Y100.000 ;

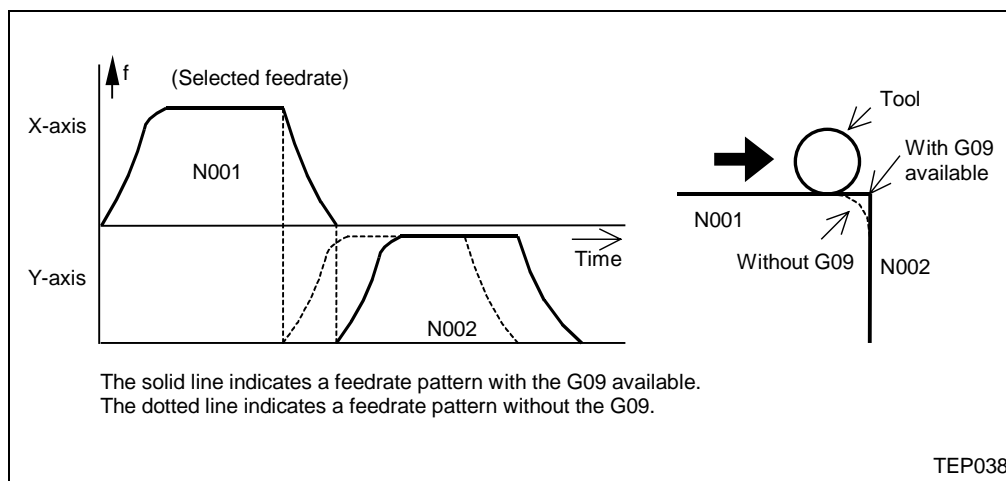


Fig. 7-1 Validity of exact-stop check

4. Detailed description

A. Continuous cutting feed commands

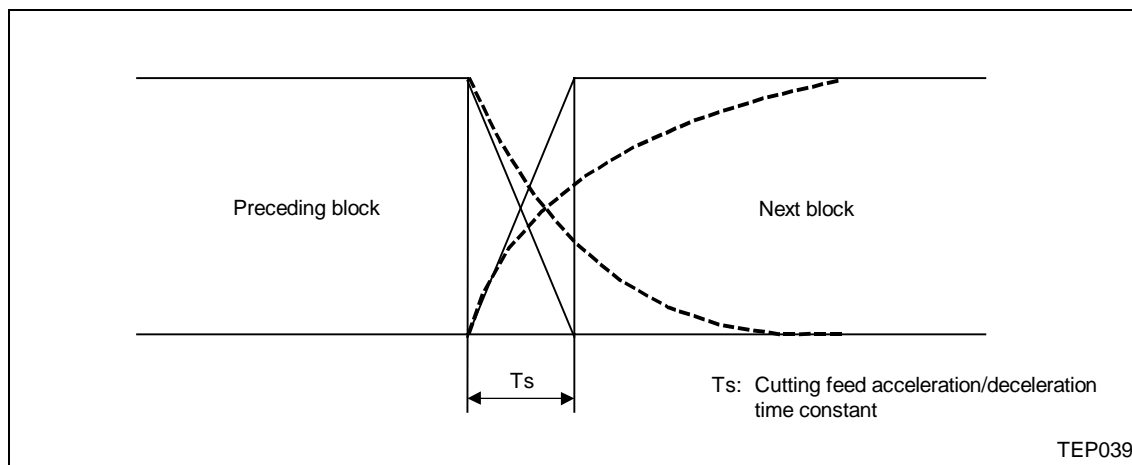


Fig. 7-2 Continuous cutting feed commands

B. Cutting feed commands with in-position status check

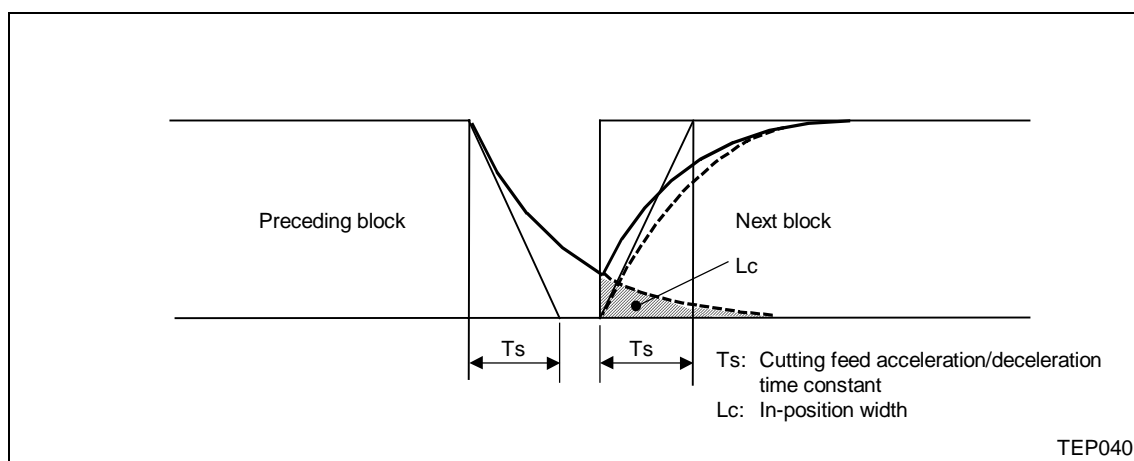
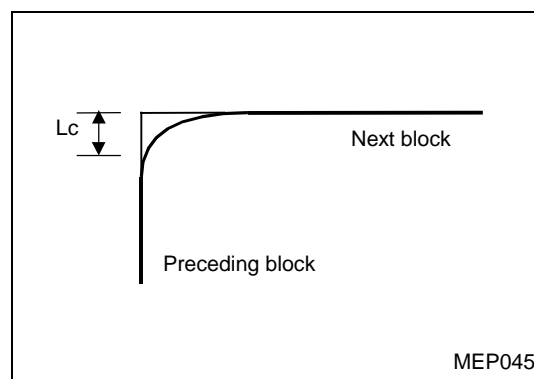


Fig. 7-3 Block-to-block connection in cutting feed in-position status check mode

As shown in Fig. 7-3, in-position width L_c represents the remaining distance within the block immediately preceding the next block to be executed.

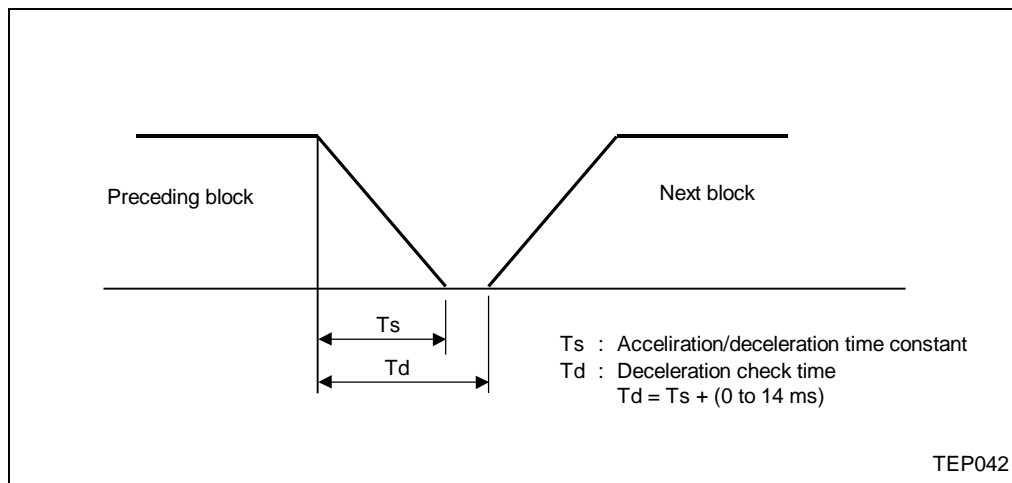
The in-position width helps keep any rounding of workpieces during corner cutting within a fixed level.

If rounding of workpieces at corners is to be completely suppressed, include dwell command G04 between cutting blocks.

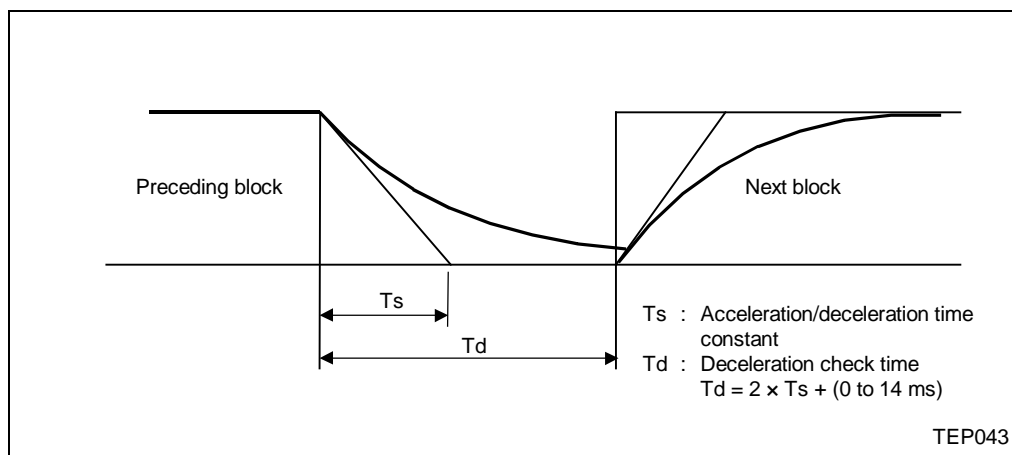


C. With deceleration check

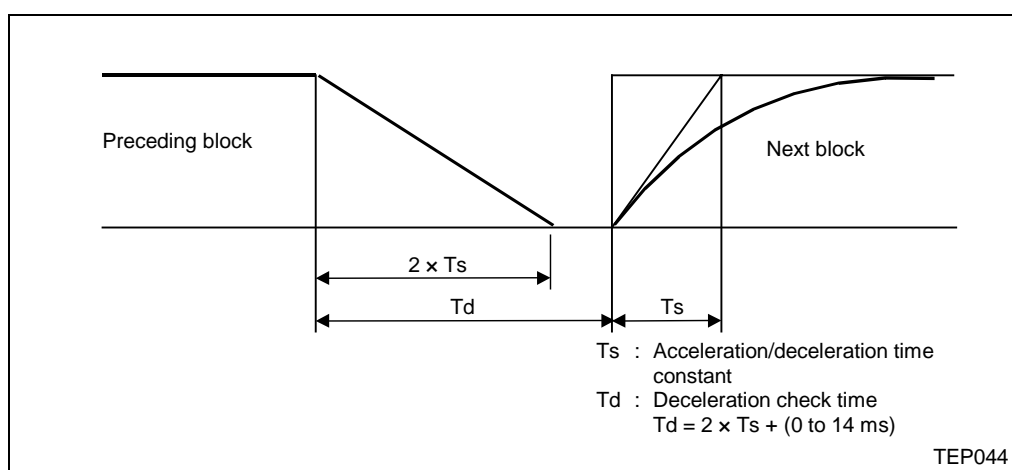
- With linear acceleration/deceleration



- With exponential acceleration/deceleration



- With exponential acceleration/linear deceleration



The time required for the deceleration check during cutting feed is the longest among the cutting feed deceleration check times of each axis determined by the cutting feed acceleration/deceleration time constants and by the cutting feed acceleration/ deceleration mode of the axes commanded simultaneously.

7-8 Exact-Stop Check Mode: G61

1. Function and purpose

Unlike exact-stop check command G09 which performs an in-position status check on that block only, command G61 functions as a modal command. That is, this command acts on all its succeeding cutting commands (G01, G02, and G03) so that deceleration occurs at the end of each block, followed by an in-position status check. Command G61 is cleared by geometry compensation command G61.1, automatic corner override command G62, tapping mode command G63, or cutting mode command G64.

2. Programming format

G61;

7-9 Automatic Corner Override: G62

1. Function and purpose

Command G62 automatically overrides in the tool radius compensation mode the selected feed rate to reduce the tool load during inner-corner cutting or automatic inner-corner rounding.

Once command G62 has been issued, the automatic corner override function will remain valid until it is cancelled by tool radius compensation cancellation command G40, exact-stop check mode command G61, geometry compensation command G61.1, tapping mode command G63, or cutting mode command G64.

2. Programming format

G62;

3. Detailed description

A. Inner-corner cutting

When inner corner of a workpiece is cut as shown in the figure below, the load on the tool increases because of large amount of cutting. Using G62 in such a case allows the cutting feed rate to be automatically overridden within the preset zone, and thus the tool load to be reduced to accomplish appropriate cutting.

This function, however, is valid only for programming the as-finished shape of a workpiece.

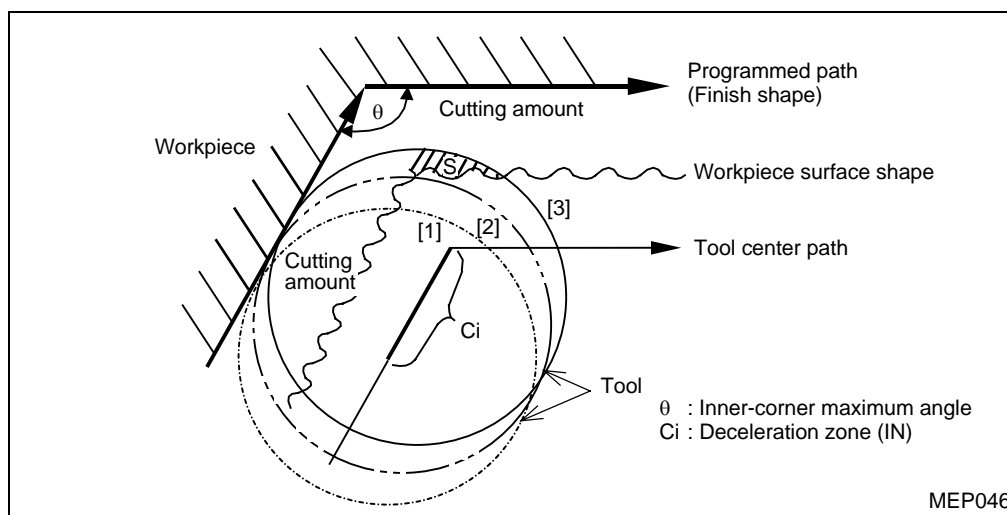


Fig. 7-4 Inner-corner cutting

<Machine operation>

- When the automatic corner override function is not used:

In the figure above, as the tool is moving in order of positions [1]→[2]→[3], the load on the tool increases because the cutting amount at position [3] is larger than that of position [2] by the area of hatched section S.

- When the automatic corner override function is used:

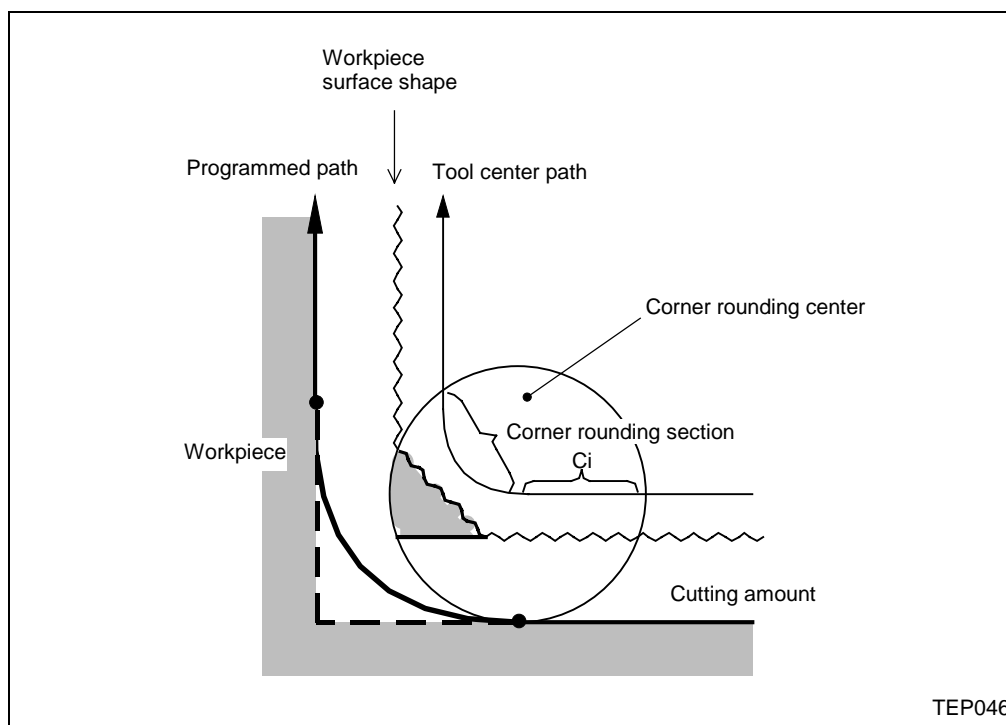
In the figure above, if maximum angle q of the inner corners is smaller than that preset in the appropriate parameter, the feed rate is automatically overridden with the preset value for movement through deceleration zone C_i .

<Setting parameters>

Set the following parameters as user parameters:

- **F29:** Override 0 to 100 (%)
- **F21:** Inner-corner maximum angle θ 0 to 180 (deg)
- **F22:** Deceleration zone C_i data 0 to 99999.999 (mm) or to 3937.000 (in)

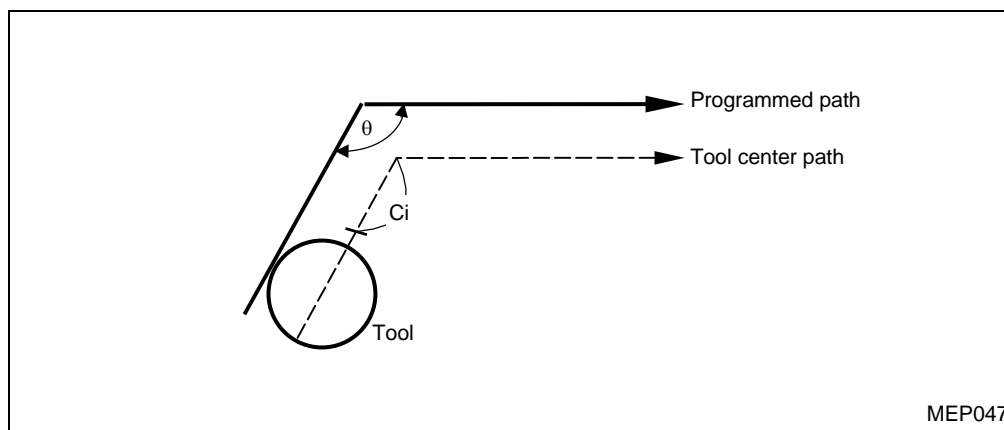
For further details of parameter setting, refer to the description in the Parameter list.

B. Automatic corner rounding

For inner corner cutting with automatic corner rounding, override will be effected as set in parameter through the deceleration zone C_i and corner rounding section (No check made about angle).

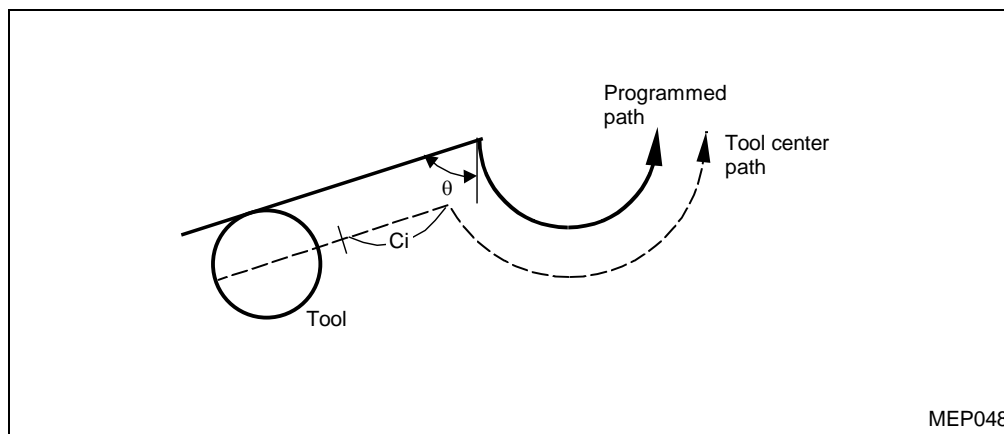
4. Operation examples

- Line-to-line corner



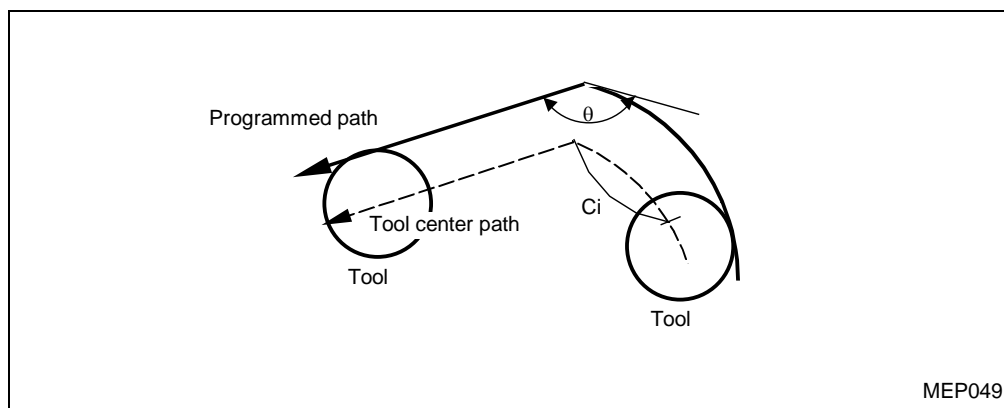
The feed rate is automatically overridden with the preset value by the parameter **F29** through deceleration zone C_i .

- Line-to-circular (outside offsetting) corner



The feed rate is automatically overridden with the preset value by the parameter **F29** through deceleration zone C_i .

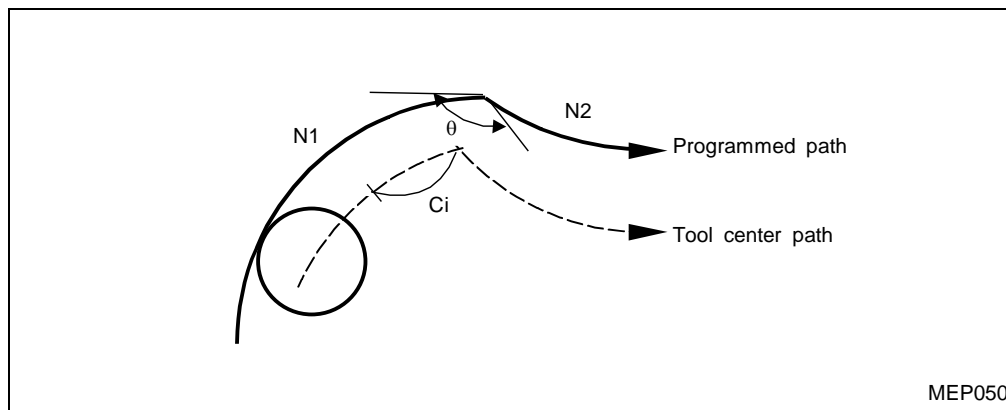
- Arc(internal compensation)-to-line corner



The feed rate is automatically overridden with the preset value by the parameter **F29** through deceleration zone C_i .

Note: Data of deceleration zone C_i at which automatic overriding occurs represents the length of the arc for a circular interpolation command.

- Arc(internal compensation)-to-arc (external compensation) corner



The feed rate is automatically overridden with the preset value by the parameter **F29** through deceleration zone C_i .

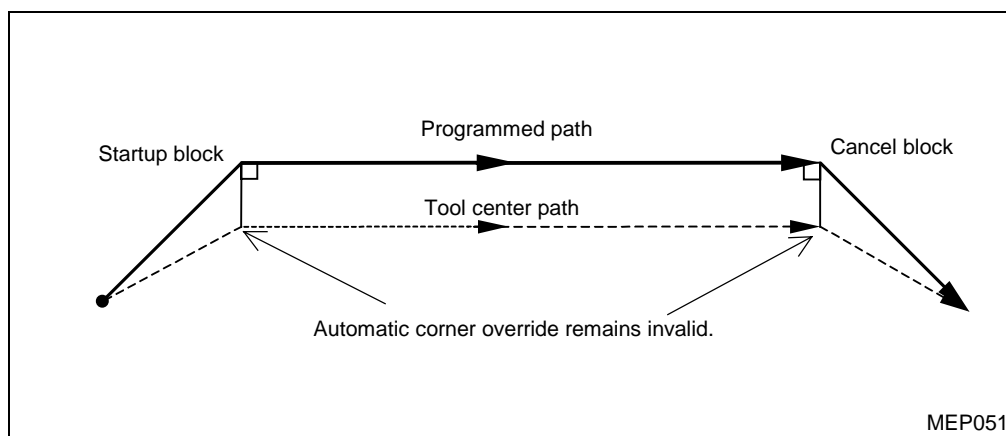
5. Correlationships to other command functions

Function	Override at corners
Cutting feedrate override	Automatic corner override is applied after cutting feed override.
Override cancel	Automatic corner override is not cancelled by override cancel.
Feed rate clamp	Valid (for the feed rate after automatic corner override)
Dry run	Automatic corner override is invalid.
Synchronous feed	A synchronous feed rate is automatically corner-overridden.
Skip (G31)	During tool radius compensation, G31 will result in an alarm.
Machine lock	Valid
G00	Invalid
G01	Valid
G02, G03	Valid

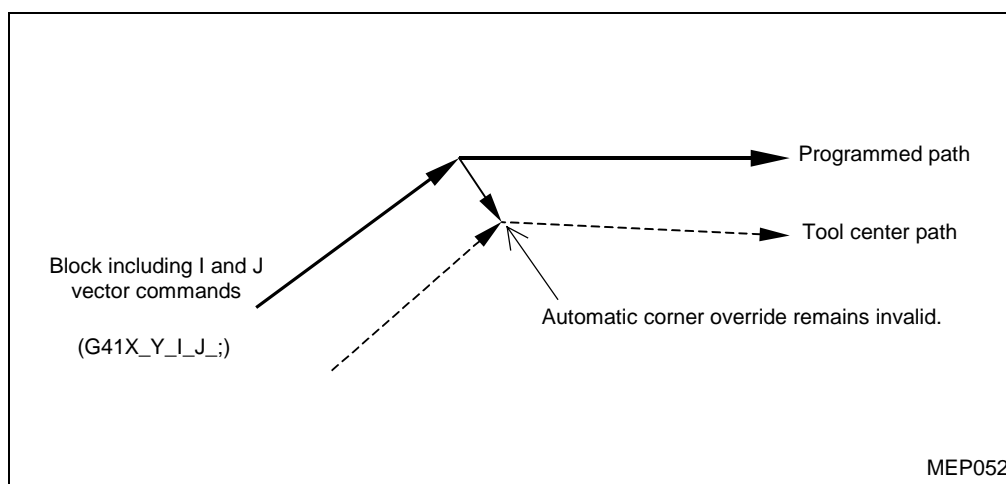
6. Precautions

- Automatic corner override is valid only during the G01, G02 or G03 modes; it is invalid during the G00 mode. Also, when the command mode is changed over from G00 to G01, G02, or G03 (or vice versa) at a corner, automatic corner override is not performed on the G00-containing block at that corner.
- Even in the automatic corner override mode, automatic corner override is not performed until the tool radius compensation mode has been set.

3. Automatic corner override does not occur at corners where tool radius compensation is to start or to be cancelled.



4. Automatic corner override does not occur at corners where I, J and K vector commands for tool radius compensation are to be executed.



5. Automatic corner override occurs only when crossing points can be calculated. Crossing points can not be calculated in the following case:
 - Four or more blocks that do not include move command appear in succession.
6. For circular interpolation, the deceleration zone is represented as the length of the arc.
7. The parameter-set angle of an inner corner is applied to the angle existing on the programmed path.
8. Setting the maximum angle to 0 or 180 degrees in the angle parameter results in an automatic corner override failure.
9. Setting the override to 0 or 100 in the override parameter results in an automatic corner override failure.

7-10 Tapping Mode: G63

1. Function and purpose

Command G63 enters the NC unit into a control mode suitable for tapping. This mode has the following features:

- The cutting feed override is fixed at 100%.
- Commands for deceleration at block-to-block connections are invalidated.
- The feed hold function is invalidated.
- The single-block function is invalidated.
- Tapping-mode signal is output.

The G63 command mode will remain valid until it is cancelled by exact-stop check mode command G61, geometry compensation command G61.1, automatic corner override command G62 or cutting mode command G64.

2. Programming format

G63 ;

7-11 Cutting Mode: G64

1. Function and purpose

Command G64 enters the NC unit into a control mode proper to obtain smoothly cut surfaces. Unlike the exact-stop check mode (G61 command mode), the cutting mode allows the next block to be executed without decelerating/stopping the machine between cutting feed blocks.

The G64 command mode is cleared by exact-stop check mode command G61, geometry compensation command G61.1, automatic corner override command G62 or tapping mode command G63.

In the initial state of the NC unit, the cutting mode is selected.

2. Programming format

G64 ;

7-12 Geometry Compensation/Accuracy Coefficient: G61.1/K

7-12-1 Geometry compensation function: G61.1

1. Function and purpose

The geometry compensation function (G61.1) is provided to reduce conventional geometry errors caused by delayed follow-up of smoothing circuits and servo systems.

The geometry compensation function is canceled, or replaced, by the functions of exact stop check mode (G61), automatic corner override (G62), tapping mode (G63) and cutting mode (G64).

The geometry compensation function is composed of the following four functions:

1. Pre-interpolation acceleration/deceleration
2. Feed forward control
3. Optimum corner deceleration
4. Precise vector compensation

Refer to Section 11-2 "Geometry Compensation Function" in PART 3 of the Operating Manual for the description of the above functions.

2. Programming format

G61.1;

3. Sample program

N001 G61.1	Selection of the geometry compensation function
G01X100.F1000	
X100.Y-100.	
X-100.Y-100.	
X-100.	
X-100.Y100.	
X100.Y100.	
G64	Cancellation of the geometry compensation function

4. Remarks

1. The geometry compensation function cannot be selected or canceled for EIA/ISO programs by the setting of the parameter **F72** (which is only effective for MAZATROL programs).
2. The geometry compensation is an optional function. On machines without corresponding option the code G61.1 can only lead to an alarm (**808 MIS-SET G CODE**).
3. The geometry compensation function is suspended during execution of the following operations:
Rapid traverse of non-interpolation type (according to bit 6 of parameter **F91**), Synchronous tapping, Measurement (skipping), Threading.
4. The pre-interpolation acceleration/deceleration is effective from the block of G61.1 onward.

7-12-2 Accuracy coefficient (,K)

1. Function and purpose

In the mode of geometry compensation (G61.1) the feed of the tool is automatically decelerated at relevant corners and for circular motions by the optimal corner deceleration and the circular feed limitation, respectively, in order to enhance the machining accuracy. Specifying an accuracy coefficient in the machining program can further improve the accuracy by additionally decelerating the feed for the sections concerned.

2. Programming format

,K_; Specify the rate of reduction of the corner deceleration speed and the circular feed rate limitation in percentage terms.

The accuracy coefficient is canceled in the following cases:

- Resetting is performed,
- The geometry compensation function is canceled (by G64),
- A command of “,K0” is given.

3. Sample program

<Example 1>

```
N001 G61.1
N200 G1X_Y_,K30
N300 X_Y_
N400 ...
```

← The rate of feed for a corner deceleration or circular motion in the section from this block onward will be reduced to 70% of the value applied in default of the accuracy coefficient command.

<Example 2>

```
N001 G61.1
N200 G2I-10.,K30
N300 G1X10.,K0
N400 ...
```

← Deceleration to 70% occurs for this block only.

← The accuracy coefficient is canceled from this block onward.

4. Remarks

1. The accuracy coefficient cannot be specified in a MAZATROL program.
2. Specifying an accuracy coefficient 1 to 99 at address “,K” increases the machining time according to the additional deceleration at relevant corners and for circular motions.

7-13 Inverse Time Feed: G93 (Option)

1. Function and purpose

When tool radius compensation is performed for a smooth linear or circular small-line-segment command, differences will occur between the contour defined in the program and that existing after tool radius compensation. The feed commands with G94 and G95 only apply for the tool path existing after compensation, and the tool speed at the point of cutting (that is, along the programmed contour), therefore, will not be kept constant so that the resulting speed fluctuations will cause seams on the surface machined.

Setting of an Inverse Time Feed command code makes constant the processing time for the corresponding block of the machining program, and thus provides control to ensure a constant machining feed rate at the point of cutting (along the programmed contour).

Setting of command code G93 specifies the inverse time assignment mode.

In G93 mode, the reciprocal of the machining time for the block of cutting command code G01, G02 or G03 is to be assigned using an F-code. Data that can be assigned with address F is from 0.001 to 99999.999.

The feed rate for the corresponding block is calculated (by NC) from the commanded length of the program block and the value of the F-code.

- For linear interpolation (G01)

$$\text{F-code value} = \frac{[\text{Speed}]}{[\text{Distance}]}$$

[Speed] : mm/min (for metric system) or
in/min (for inch system)

[Distance] : mm (for metric system) or
in (for inch system)

- For circular interpolation (G02 or G03)

$$\text{F-code value} = \frac{[\text{Speed}]}{[\text{Arc radius}]}$$

[Speed] : mm/min (for metric system) or
in/min (for inch system)

[Arc radius] : mm (for metric system) or
in (for inch system)

2. Programming formats

- Linear interpolation: G93 G01 Xx₁ Yy₁ Ff₁

- Circular interpolation: G93 G02 Xx₁ Yy₁ Rr₁ Ff₁

(Code G03 can be used, instead of G02, and code I, J and/or K instead of R.)

3. Precautions

- Code G93, which belongs to the same G-code group as that of G94 (feed per minute) and G95 (feed per revolution), is a modal G-code.

- In G93 mode, since F codes are not handled as modal codes, they must be set for each block. The absence of F-code results in alarm **816 FEEDRATE ZERO**.

- Setting of F0 during G93 mode results in alarm **816 FEEDRATE ZERO**.

- For a corner insertion block during tool radius compensation, the F-code value in the previous block is regarded as the inverse time command value.

- A modal F-code must be set if the G93 mode is changed over to G94 or G95.

4. Description of alarms

No.	Message	Description
940	NO INVERSE TIME OPTION	The Inverse Time Feed option is not present.
941	G93 MODE	An illegal G-code* has been set during G93 mode.

* Illegal G-codes are:

G31	Skip
G32, G33	Threading
G7□, G8□, G2□□	Fixed cycle

5. Sample program

```

N01 G90 G00 X-80. Y80.
N02 G01 G41 X0 Y0 D11 F500
N03      X200.
N04 G93 G02      Y-200.R100. F5
N05      G03      Y-400.R100. F5
N06      G02      Y-600.R100. F5
N07 G94 G01 X0      F500
N08      Y0
N09 G40      X-80. Y80.
N10 M02

```

MEP053

In this example, set data as follows if the machining speed in the circular-interpolation blocks is to be made equal to 500 mm/min that is specified for the linear-interpolation block of G01:

$$\text{F-code value} = \frac{[\text{Speed}]}{[\text{Arc radius}]} = \frac{500}{100}$$

- NOTE -

8 DWELL FUNCTIONS

The start of execution of the next block can be delayed using a G04 command.

8-1 Dwell Command in Time: (G94) G04

1. Function and purpose

Setting command G04 in the feed-per-second mode (command G94) delays the start of execution of the next block for the specified time.

2. Programming format

G94 G04 X_;

or

G94 G04 P_;

Data must be set in 0.001 seconds.

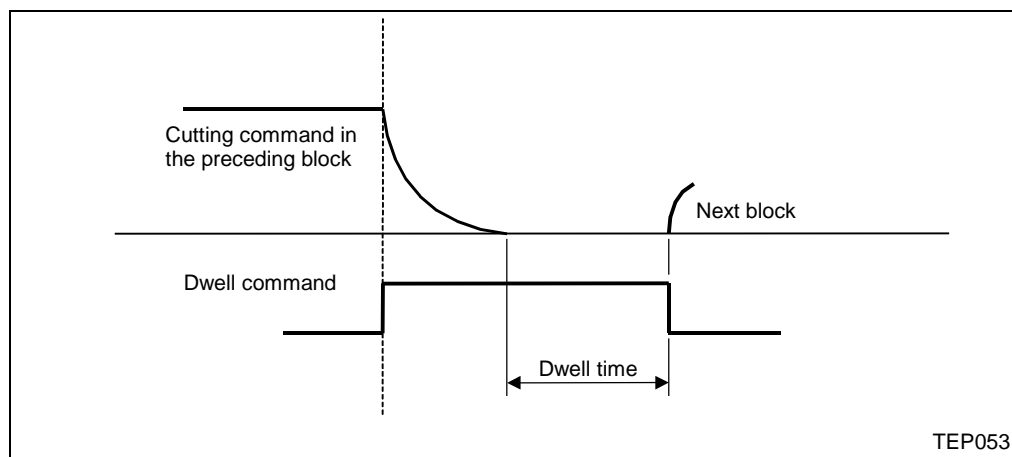
For address P, the decimal point is not available. Setting a decimal point will cause an alarm.

3. Detailed description

- The setting range for dwell time is as follows:

Unit of data setting	Range for address X	Range for address P
0.001 mm	0.001 to 99999.999 (s)	1 to 99999999 (× 0.001 s)
0.0001 inches	0.001 to 99999.999 (s)	1 to 99999999 (× 0.001 s)

- The count for the dwell command which is preceded by a block with cutting-feed command is not started until the movement of the preceding block has been brought to a complete stop.



If the dwell command is given in one block together with an M-, S-, T- or B-code, the dwell count and the execution of the respective code will be started at the same time.

- If the bit 2 of parameter **F92** is set to 1, dwell command value is always processed in time specification irrespective of G94 and G95 modes.

4. Sample programs

- When data is to be set in 0.01 mm, 0.001 mm or 0.0001 inches:
 - G04 X 500 ; Dwell time = 0.5 s
 - G04 X 5000 ; Dwell time = 5.0 s
 - G04 X 5. ; Dwell time = 5.0 s
 - G04 P 5000 ; Dwell time = 5.0 s
 - G04 P 12.345 ; *Alarm*
- When data is to be set in 0.0001 inches and dwell time is included before G04:
 - X5. G04 ; Dwell time = 50 s (Equivalent to X50000G04.)

8-2 Dwell Command in Number of Revolutions: (G95) G04

1. Function and purpose

Setting command G04 in the feed-per-revolution mode (command G95) suspends the start of execution of the next block until the spindle has rotated the specified number of revolutions.

2. Programming format

G95 G04 X_ ;

or

G95 G04 P_ ;

Data must be set in 0.001 revolutions.

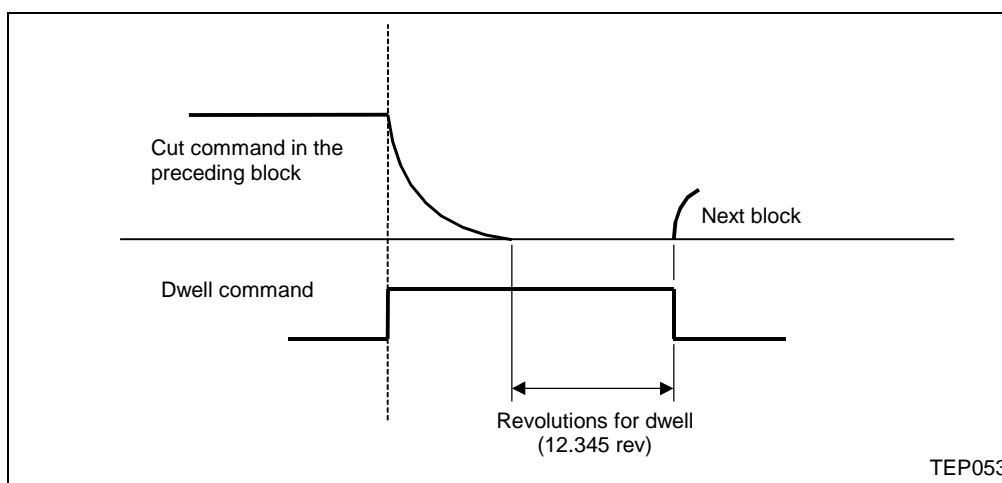
For address P, the decimal point is not available. Setting a decimal point will cause an alarm.

3. Detailed description

1. The setting range for number of dwell revolutions is as follows:

Unit of data setting	Range for address X	Range for address P
0.001 mm	0.001 to 99999.999 (rev)	1 to 99999999 (× 0.001 rev)
0.0001 inches	0.001 to 99999.999 (rev)	1 to 99999999 (× 0.001 rev)

2. The count for the dwell command which is preceded by a block with cutting-feed command is not started until the movement of the preceding block has been brought to a complete stop.



If the dwell command is given in one block together with an M-, S-, T- or B-code, the dwell count and the execution of the respective code will be started at the same time.

3. The dwell function is also valid during the machine lock mode.
4. During rest of the spindle, dwell count is also halted. When the spindle restarts rotating, dwell count will also restart.
5. If the bit 2 of parameter **F92** is set to 1, dwell command value is always processed in time specification.

- NOTE -

9 MISCELLANEOUS FUNCTIONS

9-1 Miscellaneous Functions (M3-Digit)

Miscellaneous functions, which are also referred to as M-code functions, give spindle forward/backward rotation and stop commands, coolant on/off commands, and other auxiliary commands to the NC machine.

For the NC unit, these functions must be selected using M3-digit data (three-digit data preceded by address M). Up to four sets of M3-digit data can be included in one block.

Example: G00 Xx₁ Mm₁ Mm₂ Mm₃ Mm₄;

If five or more sets of M3-digit data are set, only the last four sets will become valid.

Refer to the machine specification for more specific relationships between available data and functions.

For M-codes M00, M01, M02, M30, M98, M99, M998 and M999, the next block of data is not read into the input buffer since pre-reading is disabled automatically.

The M-codes can be included in any block that contains other command codes. If, however, the M-codes are included in a block that contains move commands, then the execution priority will be either

- the M-code functions are executed after completion of movement, or
- the M-code functions are executed together with movement.

It depends on the machine specifications which type of processing is applied.

Processing and completion sequences are required in each case for all M commands except M98 and M99.

The following lists six types of special M-code functions:

1. Programmed Stop: M00

When this M-code is read, the tape reader will stop reading subsequent block. Whether the machine function such as spindle rotation and coolant will also stop depends on the machine specifications. The machine operation is restarted by pressing the cycle start button on the operation panel. Whether resetting can be initiated by M00 or not also depends on the machine specifications.

2. Optional Stop: M01

When the M01 code is read with the **[OPTIONAL STOP]** menu function set to ON, the tape reader will stop operating to perform the same function as M00.

The M01 command will be ignored if the **[OPTIONAL STOP]** menu function is set to OFF.

Example: :
 N10 G00 X1000;
 N11 M01;
 N12 G01 X2000 Z3000 F600;
 :

<**[OPTIONAL STOP]** menu function status and operation>

If the menu function is on, operation stops at N11.

If the menu function is off, operation does not stop at N11 and N12 is executed.

3. Program End: M02 or M30

Usually, the program end command is given in the final block of machining program. Use this command mainly for reading data back to the head of the program during memory operation, or rewinding the tape in the tape operation mode (use an M30 command to rewind the tape). The NC unit is automatically reset after tape rewinding and execution of other command codes included in that block.

Automatic resetting by this command cancels both modal commands and offsetting data, but the designated-position display counter is not cleared to zero.

The NC unit will stop operating when tape rewinding is completed (the automatic run mode lamp goes out). To restart the NC unit, the cycle start button must be pressed.

Beware that if, during the restart of the NC unit following completion of M02 or M30 execution, the first movement command has been set in a coordinate word only, the valid mode will be the interpolation mode existing when the program ended. It is recommended, therefore, that the first movement command be given with an appropriate G-code.

4. Subprogram Call/End: M98, M99

Use M98 or M99 to branch the control into a subprogram or to recall it back to the calling program.

As M98 and M99 are internally processed by the NC M-code signals and strobe signals are not output.

<Internal processing by the NC unit when M00, M01, M02 or M30 is used>

After M00, M01, M02 or M30 has been read, data pre-reading is automatically aborted. Other tape rewinding operations and the initialization of modals by resetting differ according to the machine specification.

Note 1: M00, M01, M02 and M30 output independent signals, which will be cancelled by pressing the RESET key.

Note 2: Tape rewinding is performed only when the tape reader has a rewinding function.

9-2 No. 2 Miscellaneous Functions (A8/B8/C8-Digit)

The No. 2 miscellaneous functions are used for positioning an index table. For the NC unit, these functions must be designated using an eight-digit value (from 0 to 99999999) preceded by address A, B or C.

The output signals are BCD signals of command data and start signals.

A, B or C codes can be included in any block that contains other command codes. If, however, the A, B or C codes can be included in a block that contains move commands, then the execution priority will be either

- the A, B or C code functions are performed after completion of movement, or
- the A, B or C code functions are performed together with movement.

It depends on the machine specifications which type of processing is applied.

Processing and completion sequences are required in each case for all No. 2 miscellaneous functions.

Address combinations are shown below. The same address for both additional axis and the No. 2 miscellaneous functions cannot be used.

Additional axis No. 2 miscellaneous functions	A	B	C
A	x	○	○
B	○	x	○
C	○	○	x

Note: When A has been designated as the No. 2 miscellaneous function address, linear angle commands cannot be used.

10 SPINDLE FUNCTIONS

10-1 Spindle Function (S5-Digit Analog)

When the S5-digit function is added, this function must be set using the numerical command of five digits preceding an S code (0 to 99999) and for other case, two digits preceding by an S code is used.

S command binary outputs must be selected at this time.

By designating a 5-digit number following the S code, this function enables the appropriate gear signals, voltages corresponding to the commanded spindle speed (rpm) and start signals to be output.

Processing and completion sequences are required for all S commands.

The analog signal specifications are given below.

- Output voltage 0 to 10V or -8 to +8V
- Resolution..... 1/4096 (2 to the power of -12)
- Load conditions 10 kilohms
- Output impedance 220 ohms

If the parameters for up to 4 gear range steps are set in advance, the gear range corresponding to the S command will be selected by the NC unit and the gear signal will be output. The analog voltage is calculated in accordance with the input gear signal.

- Parameters corresponding to individual gears... Limit speed, maximum speed, gear shift speed and maximum speed during tapping.
- Parameters corresponding to all gears Orient speed, minimum speed

10-2 Spindle Speed Range Setting: G92

1. Function and purpose

The code G92 can be used to set the maximum and minimum spindle speeds at addresses S and Q, respectively.

2. Programming format

G92 Ss Qq Rr;

- s: Maximum spindle speed
- q: Minimum spindle speed
- r: Spindle for speed limitation (to be specified with 3 for the milling spindle)

3. Detailed description

For gear change between the spindle and spindle motor, four steps of gear range can be set by the related parameters in steps of 1 min^{-1} . In range defined by two ways, parameter setting and G92 SsQq setting, the smaller data will be used for the upper limit and the larger data for the lower limit.

Do not fail to designate the spindle with argument "R3" in the G92 command block.

- NOTE -

11 TOOL FUNCTIONS

11-1 Tool Function (4-Digit T-Code)

Tool function, also referred to as T-code function, is used to designate the tool number and offset number. Of a four-digit integer at address T, upper and lower two digits are respectively used to specify the tool number and offset number.

Only one T-code can be included in a block, and the available range of T-codes depends on the machine specifications. For further details, especially on how to number the actual tools to be used, refer to the operating manual of the relevant machine.

The T-code can be given with any other commands in one block, and the T-code given together with an axis motion command is executed, depending upon the machine specifications, in one of the following two timings:

- The T-code is not executed till completion of the motion command, or
- The T-code is executed simultaneously with the motion command.

11-2 Tool Function (8-Digit T-Code)

This function allows you to select a tool number (from 0 to 99999999) using eight-digit command data preceded by address T. Only one T-code can be included in a block.

Set bit 4 of parameter **F94** to 0 to select the group-number designation for T-code function, or set this bit to 1 to select the tool-number designation.

- NOTE -

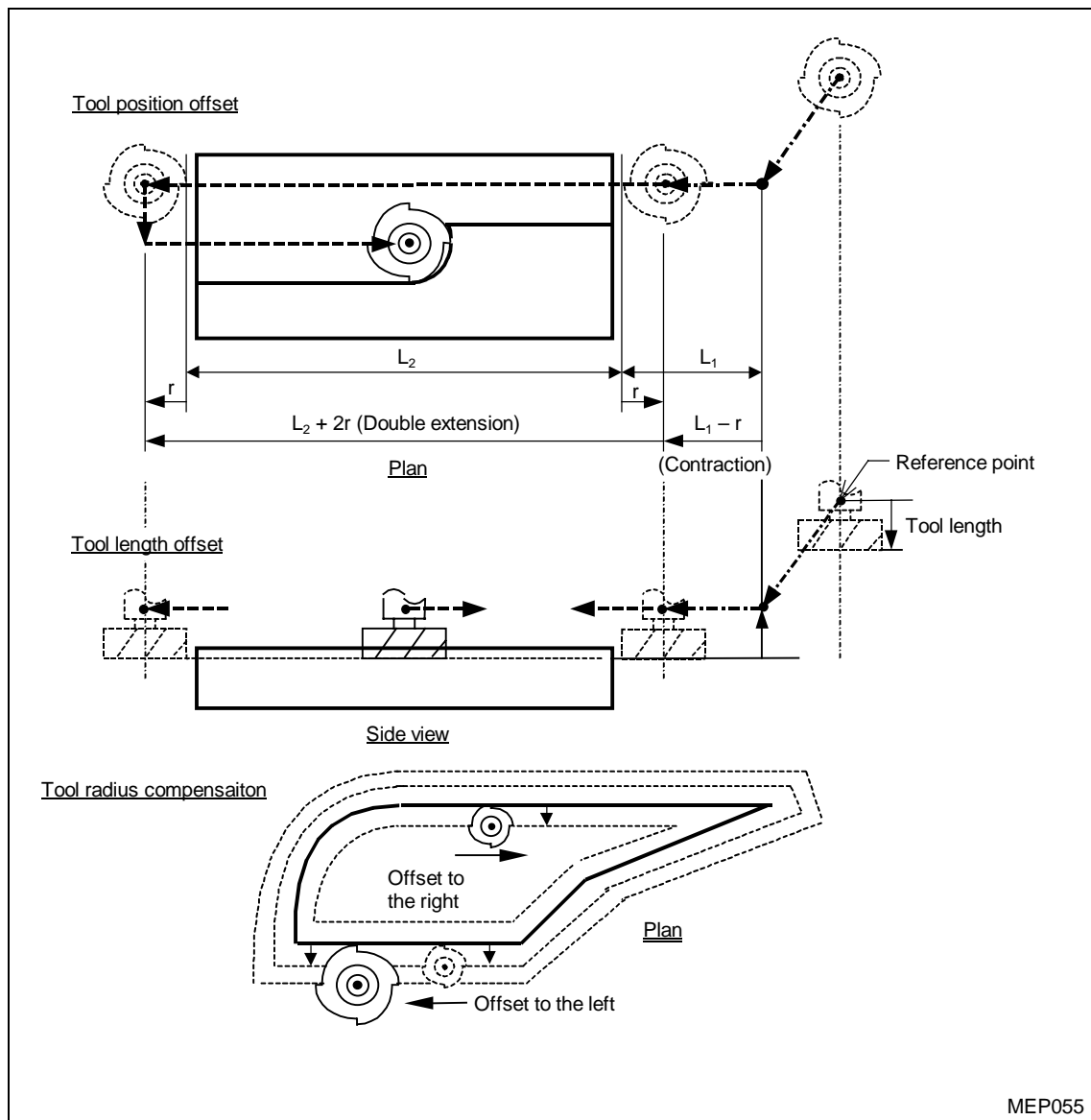
12 TOOL OFFSET FUNCTIONS

12-1 Tool Offset

1. Overview

As shown in the diagram below, three types of basic tool offset functions are available: tool position offset, tool length offset, and tool radius compensation.

These three types of offset functions use offset numbers for designation of offset amount. Set the amount of offset directly using the operation panel or by applying the function of programmed parameter input. MAZATROL tool data can also be used for tool length offset or tool radius compensation operations according to the parameter setting.



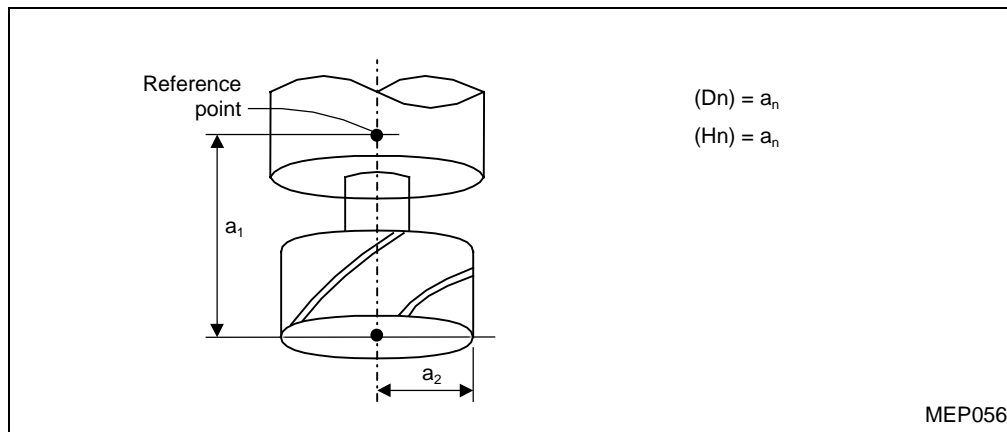
2. Selecting the amounts of tool offset

The amounts of tool offset corresponding to the offset numbers must be prestored on the **TOOL OFFSET** display by manual data input method or programmed data setting function (G10).

The mounts of tool offset can be selected using one of the following two types:

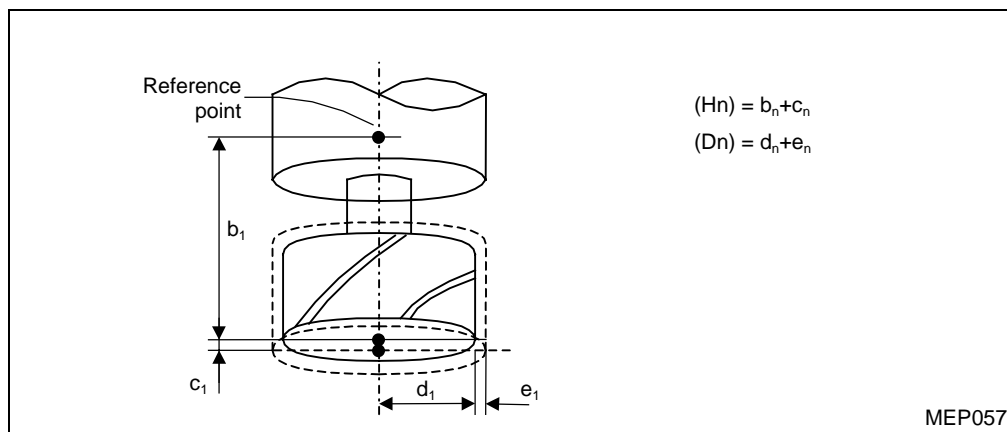
A. Type A

The same amount of offset will be set if identical offset numbers are selected using commands D and H.



B. Type B

Set an H-code and D-code, respectively, to use the total sum of the geometric offset amount and the wear compensation amount for tool length offset and tool radius compensation.



3. TOOL OFFSET display types

As a data storage area for tool offsetting functions, two types of the **TOOL OFFSET** display are provided: Type A and Type B.

Type	Length/Radius distinguished?	Geometric/Wear distinguished?
A	No	No
B	Yes	Yes

A. Type A

As listed in the table below, one offset data is given for one offset number. No distinction is drawn between length, radius, geometric and wear compensation amounts. That is, one set of offset data comprises all these four factors.

$$\begin{aligned} (D1) &= a_1, & (H1) &= a_1 \\ (D2) &= a_2, & (H2) &= a_2 \\ &\vdots & &\vdots \\ (Dn) &= a_n, & (Hn) &= a_n \end{aligned}$$

Offset No.	Offset amount
1	a_1
2	a_2
3	a_3
\vdots	\vdots
\vdots	\vdots
n	a_n

B. Type B

As listed in the table below, two types of offset data can be set for one offset number. That is, different amounts of geometric offset and wear compensation can be set for each of the selected tool length and the selected tool radius.

Use command H to select offset data concerning the tool length, and use command D to select offset data concerning the tool radius.

$$\begin{aligned} (H1) &= b_1 + c_1, & (D1) &= d_1 + e_1 \\ (H2) &= b_2 + c_2, & (D2) &= d_2 + e_2 \\ &\vdots & &\vdots \\ (Hn) &= b_n + c_n, & (Dn) &= d_n + e_n \end{aligned}$$

Offset No.	Tool length (H)		Tool radius (D) / (Position offset)	
	Geometric offset	Wear compensation	Geometric offset	Wear compensation
1	b_1	c_1	d_1	e_1
2	b_2	c_2	d_2	e_2
3	b_3	c_3	d_3	e_3
\vdots	\vdots	\vdots	\vdots	\vdots
\vdots	\vdots	\vdots	\vdots	\vdots
n	b_n	c_n	d_n	e_n

4. Tool offset numbers (H/D)

Tool offset numbers can be selected using address H or D.

- Use address H to offset the selected tool length. Use address D to offset the selected tool position or the selected tool radius.
- Once a tool offset number has been selected, it will remain unchanged until a new H or D is used.
- Offset numbers can be set only once for one block. If offset numbers are set more than once for one block, only the last offset number will be used.
- The offset data range is as listed in the table below.

Offset data for each offset number must be set beforehand on the **TOOL OFFSET** display.

	Metric	Inch
TOOL OFFSET Type A	±1999.9999 mm	±84.50000 in
TOOL OFFSET Type B Length Geom.	±1999.9999 mm	±84.50000 in
TOOL OFFSET Type B Length Wear	±99.9999 mm	±9.99999 in
TOOL OFFSET Type B Dia. Geom.	±999.9999 mm	±84.50000 in
TOOL OFFSET Type B Dia. Wear	±9.9999 mm	±0.99999 in

Note: The tool offset number (H- or D-code) cannot be made effective if it is not designated in the corresponding offset mode.

12-2 Tool Length Offset/Cancellation: G43, G44, or T-Code/G49

1. Function and purpose

Commands G43 and G44 allow the ending point of move commands to be shifted through the previously set offset amount for each axis. Any deviations between programmed tool size and actual length or radius can be set as offset data using these commands to make the program more flexible.

2. Programming format

G43 Zz Hh Tool length offset +
 G44 Zz Hh Tool length offset –
 G49 Zz Cancellation of tool length offset

3. Detailed description

The following represents the relationship between the programming format and the stroke of movement after offsetting.

1. Z-axis motion distance

G43Z±zHh ₁	±z + {±ℓ _{h₁} – (±ℓ _{h₀})}	Positive-direction offset by length offset amount
G44Z±zHh ₁	±z + {±ℓ _{h₁} – (±ℓ _{h₀})}	Negative-direction offset by length offset amount
G49Z±z	±z – (±ℓ _{h₁})	Cancellation of the offset amount

ℓ_{h₁}: **BA62** + Value of offset No. h₁

ℓ_{h₀}: Offset amount existing before the G43 or G44 block

Irrespective of whether absolute or incremental programming method is used, the actual ending point coordinates are calculated by offsetting the programmed end point coordinate through the offset amount.

The initial state (upon turning-on or after M02) is of G49 (tool length offset cancellation).

2. Sample programs

For absolute data input

(H01: Z = 95.)

```

N001 G90 G94 G00 G40 G80
N002 G91 G28 Z0 X0
N003 T01 T00 M06
N004 G90 G54 X0 Y0
N005 G43 Z5. H01
N006 G01 Z-50. F100
  
```

3. Supplement

- 1) Use bit 3 of parameter **F92** to select the axes to which tool length offset is to be applied. Offsetting (by G43H or G44H) is applied to the Z-axis only and to all the three orthogonal axes (X, Y, and Z), respectively, when **F92** bit 3 is set to 1 and 0.
- 2) Offsetting through the specified amount is always performed for a block of G43H or G44H, irrespective of whether or not an axis motion command is given in the same block.

Programming example	When F92 bit 3 = 0	When F92 bit 3 = 1
G43 Hh ₁ ⋮ G49	Motion by the offset amount on the X-, Y-, and Z-axis.	Motion by the offset amount on the Z-axis only.
G43 Xx ₂ Hh ₂ ⋮ G49	X-axis movement to the commanded position shifted by the offset amount, along with the motion on the Y- and Z-axis by the same offset amount.	X-axis movement to the commanded position, along with the motion on the Z-axis by the offset amount.
G43 Yy ₃ Hh ₃ ⋮ G49	Y-axis movement to the commanded position shifted by the offset amount, along with the motion on the X- and Z-axis by the same offset amount.	Y-axis movement to the commanded position, along with the motion on the Z-axis by the offset amount.
G43 Xx ₄ Yy ₄ Zz ₄ Hh ₄ ⋮ G49	Simultaneous movement on the three axes (X, Y, and Z) to the commanded position uniformly shifted by the offset amount.	Simultaneous movement on the three axes (X, Y, and Z) to the commanded position that is shifted only on the Z-axis by the offset amount.

- 3) If reference point (zero point) return is performed in the offsetting mode, the mode is cancelled after completion of the returning operation.

Programming example	
G43 Hh ₁ ⋮ G28 Zz ₂	Upon completion of return to the reference point (zero point), the offset amount is cleared.
G43 Hh ₁ G49 G28 Zz ₂	Reference point return after a Z-axis motion at the current position for clearing the offset amount.

- 4) If command G49 or H00 is executed, length offsetting will be immediately cancelled (the corresponding axis will move to clear the offset amount to zero).
When using MAZATROL tool data, do not use G49 as a cancellation command code; otherwise interference with the workpiece may result since automatic cancellation moves the tool on the Z-axis in minus direction through the distance equivalent to the tool length.
Use an H00 command, rather than a G49 command, if G43/G44 mode is to be cancelled temporarily.
- 5) The alarm **839 ILLEGAL OFFSET No.** will occur if an offset number exceeding the machine specifications is set.
- 6) When tool offset data and MAZATROL tool data are both validated, offsetting is executed by the sum of the two data items concerned.

12-3 Tool Length Offset in Tool-Axis Direction: G43.1 (Option)

1. Function and purpose

The preparatory function G43.1 refers to offsetting the tool, in whatever direction it may be inclined three-dimensionally, in the direction of its axis through the length offset amount.

In the G43.1 mode the tool length offsetting is always performed in the tool-axis direction, in accordance with the motion command for a rotational axis concerned, through the relevant offset amount.

2. Programming format

A. Tool length offset in tool-axis direction ON

G43.1 (XxYyZz) Hh

h: Number of length offset amount

- A block of G43.1 can contain movement commands for the orthogonal axes (X, Y, and Z), but those for rotational axes, any other G-code and an M-, S-, T- or B-code must not be entered in the same block; otherwise an alarm is caused.
- The argument H (for specifying the number of length offset amount) can be omitted when parameter **F93** bit 3 = 1 (use of **LENGTH** data prepared on the **TOOL DATA** display for executing EIA/ISO programs).

B. Tool length offset in tool-axis direction OFF

G49

- The execution of a G49 command does not cause any axis motion.
- The cancellation command G49 must be given independently (without any other instruction codes); otherwise an alarm is caused.

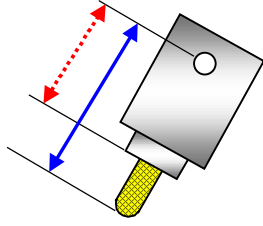


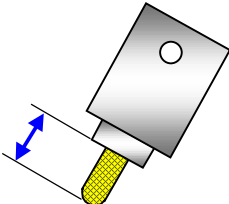

C. Offset amount selection

The table below indicates those usage patterns [1] to [4] of the externally stored tool offset data items which are applied to the tool length offset in tool-axis direction according to the settings of the relevant parameters (**F93** bit 3 and **F94** bit 7).

Pattern	Data items used (Display and Data item names)		Parameter		Programming method
			F94 bit 7	F93 bit 3	
[1]	TOOL OFFSET	Offset data items	0	0	G43.1 with H-code
[2]	TOOL DATA	LENGTH	1	1	T-code + G43.1
		LENGTH + LENG. No. LENGTH + LENG. CO.			T-code + G43.1 with H-code
[3]	TOOL DATA	LENG. No. LENG. CO.	1	0	G43.1 with H-code
[4]	TOOL OFFSET + TOOL DATA	Offset data items + LENGTH	0	1	T-code + G43.1 with H-code

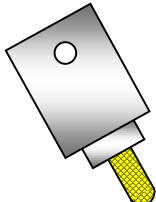
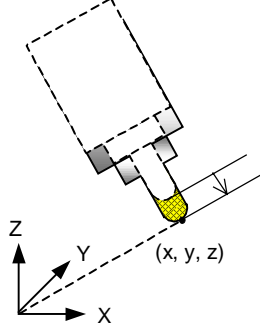
3. On the processing of offset amount for the axis of rotation

According to the definition of the tool length there are two types of tool length offset in the direction of the tool axis as described below.

Type	Inclusion of offset amount for the axis of rotation	Exclusion of offset amount for the axis of rotation
Parameter	F168 bit 2 = 0	F168 bit 2 = 1
Description	<p>Tool length is defined as a distance between the tool tip and the axis of rotation of the tool axis. This is the initial setting.</p>  <p>  : Tool length offset amount  : Offset amount for the axis of rotation </p>	<p>Tool length is defined as a distance between the tool tip and the tool mounting edge. (The distance of the tool mounting edge [spindle nose] from the axis of rotation of the tool axis is excluded.)</p>  <p>  : Tool length offset amount </p>

4. Operation of startup

The operation at the selection of the tool length offset in tool-axis direction depends upon whether or not the motion command for an orthogonal axis is given in the G43.1 block as follows:

	Motion commands for orthogonal axes	
	not given	given
Operation	<p>G43.1Hh;</p>  <p>No motions occur at all. (A motion by the offset amount does not occur.)</p>	<p>G90; G43.1XxYyZzHh;</p>  <p>A motion occurs for a point of the specified coordinates, inclusive of the amount for length offset in tool-axis direction.</p>

- "Given" is the orthogonal-axis motion command when the G43.1 block contains even only one command for an orthogonal axis.

5. Operation of cancellation

The mode of tool length offset in tool-axis direction is cancelled by a G49 command.

- The execution of a G49 command does not cause any axis motion.
- The cancellation command G49 must be given independently (without any other instruction codes); otherwise an alarm is caused.

6. Vector components for tool length offset in tool-axis direction

The orthogonal axis components of the vector for tool length offset in tool-axis direction are internally calculated as follows:

1. When the axes related to rotating the tool axis are A- and C-axis:

$$V_x = L \times \sin(A) \times \sin(C)$$

$$V_y = -L \times \sin(A) \times \cos(C)$$

$$V_z = L \times \cos(A)$$

2. When the axes related to rotating the tool axis are B- and C-axis:

$$V_x = L \times \sin(B) \times \cos(C)$$

$$V_y = -L \times \sin(B) \times \sin(C)$$

$$V_z = L \times \cos(B)$$

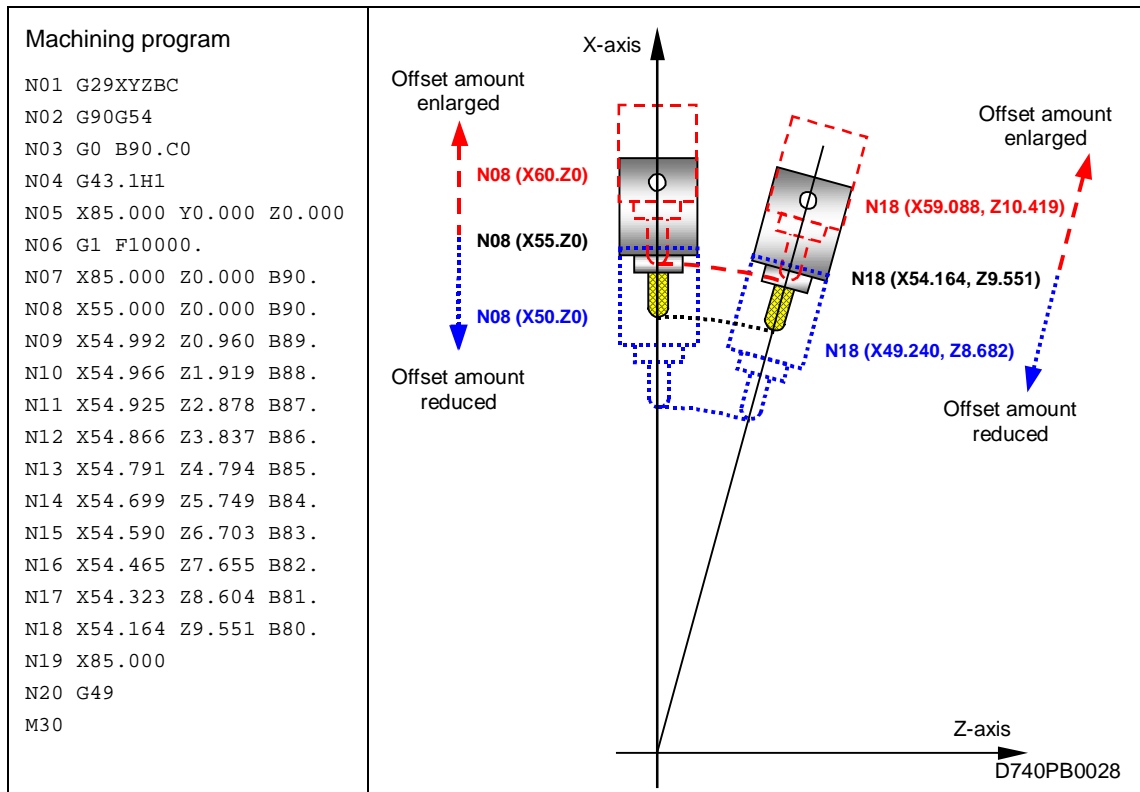
V_x, V_y, V_z : X-, Y-, and Z-components of the vector for tool length offset in tool-axis direction

L : Amount of tool length offset

$(A), (B), (C)$: Amount of angular motion on the rotational axis

7. Operation in the mode of tool length offset in tool-axis direction

The following example shows how the path of the tool tip can be shifted as required simply by changing the offset amount for the tool length offset in tool-axis direction.



The blocks N08 to N18 sequentially describe the points in the ZX-plane, with the Z- and X-components of the position vector, in a distance of 55 mm from the origin $[(X, Z) = (0, 0)]$ according to the angles of rotating the tool axis on the B-axis. Now, an external change of the offset amount (for the offset number H1) allows the tool to be shifted in the direction of the tool axis through the difference in offset amount without having to rewrite the program. Enlarge and reduce the offset amount, respectively, to shift the tool away from, and in the direction of, the origin of the ZX-plane.

	With the initial offset amount	Initial offset amount + 5	Initial offset amount – 5
N08	X55.000, Z0.000	X60.000, Z0.000	X50.000, Z0.000
N09	X54.992, Z0.960	X59.992, Z1.047	X49.992, Z0.873
N10	X54.966, Z1.919	X59.963, Z2.094	X49.970, Z1.745
N11	X54.925, Z2.878	X59.918, Z3.140	X49.931, Z2.617
N12	X54.866, Z3.837	X59.854, Z4.185	X49.878, Z3.488
N13	X54.791, Z4.794	X59.772, Z5.229	X49.810, Z4.358
N14	X54.699, Z5.749	X59.671, Z6.272	X49.726, Z5.226
N15	X54.590, Z6.703	X59.553, Z7.312	X49.627, Z6.093
N16	X54.465, Z7.655	X59.416, Z8.350	X49.513, Z6.959
N17	X54.323, Z8.604	X59.261, Z9.386	X49.384, Z7.822
N18	X54.164, Z9.551	X59.088, Z10.419	X49.240, Z8.682

8. Resetting the tool length offset in tool-axis direction

The tool length offset in tool-axis direction is cancelled in the following cases:

- The reset key is pressed,
- M02, M30, M998 or M999 is executed,
- G49 is executed, or
- A command for offset number zero (G43.1H0) is executed.

9. Compatibility with the other functions

A. Commands available in the mode of tool length offset in tool-axis direction

Code	Function		Code	Function
G00	Positioning		G61	Exact stop check mode
G01	Linear interpolation		G61.1	Geometry compensation (Note 3)
G02	Circular interpolation (CW) (Note 1)		G64	Cutting mode
G03	Circular interpolation (CCW) (Note 1)		G65	User macro single call
G04	Dwell		G90	Absolute data input
G05	High-speed machining mode		G91	Incremental data input
G09	Exact-stop check		G93	Inverse time feed
G17	XY-plane selection		G94	Feed per minute (asynchronous)
G18	ZX-plane selection		G112	M, S, T, B output to opposite system (Note 2)
G19	YZ-plane selection		M98	Subprogram call
G40	Nose R/Tool radius compensation OFF		M99	Subprogram end
G41.5	Tool radius compensation	to the left	F	Feed function
G42.5	for five-axis machining	to the right	MSTB	M-, S-, T-, and B-code (Note 2)
G49	Tool length offset OFF		Macro instruction	Local variable, Common variable, Operation command, Control command
G50	Scaling OFF			

Note 1: A command for helical or spiral interpolation leads to an alarm (**1812 ILLEGAL CMD IN G43.1 MODE**).

The same alarm will be caused when a block of circular interpolation contains a motion command for a rotational axis.

Note 2: Giving a tool change command (T-code) in the mode of tool length offset in tool-axis direction leads to an alarm (**1812 ILLEGAL CMD IN G43.1 MODE**).

Note 3: The use of G61.1 leads to an alarm (**1812 ILLEGAL CMD IN G43.1 MODE**) when the geometry compensation is made invalid for rotational axes.

B. Modes in which the tool length offset in tool-axis direction is selectable

Code	Function		Code	Function
G00	Positioning		G54.4Pp	Workpiece setup error correction
G10.9	Radius data input for X		G61	Exact stop check mode
G13.1	Polar coordinate interpolation OFF		G61.1	Geometry compensation
G15	Polar coordinate input OFF		G64	Cutting mode
G17	XY-plane selection		G65	User macro single call
G18	ZX-plane selection		G67	User macro modal call OFF
G19	YZ-plane selection		G69	3-D coordinate conversion OFF
G20	Inch data input		G80	Fixed cycle OFF
G21	Metric data input		G90	Absolute data input
G23	Pre-move stroke check OFF		G91	Incremental data input
G40.1	Control in perpendicular direction OFF		G93	Inverse time feed
G41.5	Tool radius compensation for five-axis machining	to the left	G94	Feed per minute (asynchronous)
G42.5		to the right	G97	Constant surface speed control OFF
G43	Tool length offset (+)		G98	Initial point level return in fixed cycles
G44	Tool length offset (-)		G99	R-point level return in fixed cycles
G49	Tool length offset OFF		G109	Two-system control by one program
G50	Scaling OFF		G110	Cross machining control ON
G50.1	Mirror image OFF		G111	Cross machining control OFF
G50.2	Polygonal machining mode OFF		G113	Hob milling mode OFF
G54	Selection of workpiece coordinate system			

10. Restrictions and precautions**1. Tracing**

It is not the movement of the actual tool tip, but of the point shifted from the spindle nose through the length offset amount, that is traced in the mode of tool length offset in tool-axis direction. It depends on the parameter setting concerned whether or not the offset amount used in tracing includes the “axis-of-rotation offset” amount.

2. Tool path check

As with tracing, the motion path of the point shifted from the spindle nose through the length offset amount (not of the tool tip) is drawn in the mode of tool length offset in tool-axis direction. It also depends on the parameter setting concerned whether or not the offset amount used in tool path check includes the “axis-of-rotation offset” amount.

3. Measurement

The use of a measuring cycle or the skip function in the mode of tool length offset in tool-axis direction leads to an alarm (**1812 ILLEGAL CMD IN G43.1 MODE**).

4. Tool change command (T-code)

Giving a tool change command (T-code) in the mode of tool length offset in tool-axis direction leads to an alarm (**1812 ILLEGAL CMD IN G43.1 MODE**).

5. Interruption

Do not attempt to interrupt the operation in the mode of tool length offset in tool-axis direction (be it by using manual operation mode, MDI operation mode, or using the pulse handle); otherwise an alarm is caused (**1812 ILLEGAL CMD IN G43.1 MODE**).

6. Circular interpolation

Do not enter any motion command for a rotational axis in a block of circular interpolation in the mode of tool length offset in tool-axis direction; otherwise an alarm is caused (**1812 ILLEGAL CMD IN G43.1 MODE**).

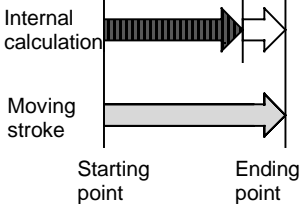
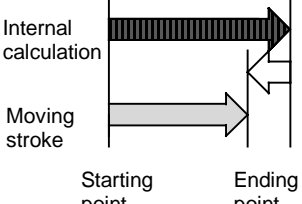
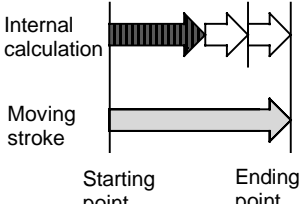
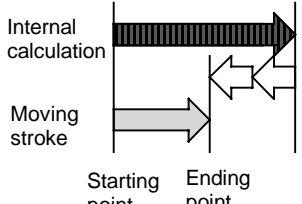
7. Others

Do not give a command for corner rounding/chamfering, a linear angle command, or a geometric command; otherwise an alarm is caused (**1812 ILLEGAL CMD IN G43.1 MODE**).

12-4 Tool Position Offset: G45 to G48

1. Function and purpose

Command G45 or G46 allows the axis movement distance set previously in that block to be increased or decreased, respectively, according to the offset data. Likewise, command G47 or G48 extends or contracts the previously set distance by twice the offset stroke, respectively.

G45 command	G46 command
Extended through offset stroke only	Contracted through offset stroke only
 <p>Internal calculation</p> <p>Moving stroke</p> <p>Starting point</p> <p>Ending point</p>	 <p>Internal calculation</p> <p>Moving stroke</p> <p>Starting point</p> <p>Ending point</p>
G47 command	G48 command
Extended through twice the offset stroke	Contracted through twice the offset stroke
 <p>Internal calculation</p> <p>Moving stroke</p> <p>Starting point</p> <p>Ending point</p>	 <p>Internal calculation</p> <p>Moving stroke</p> <p>Starting point</p> <p>Ending point</p>

$$\begin{array}{c} \text{Hatched Arrow} \end{array} \pm \begin{array}{c} \text{White Arrow} \end{array} = \begin{array}{c} \text{Grey Arrow}$$

(Program command value) (Offset stroke) (Moving stroke after offset)

2. Programming format

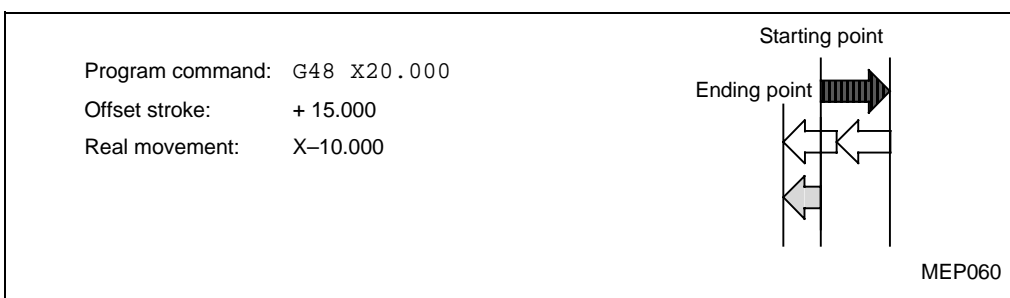
Programming format	Function
G45 Xx Dd	To extend a moving stroke by the offset stroke which has been set in the offset memory.
G46 Xx Dd	To contract a moving stroke by the offset stroke which has been set in the offset memory.
G47 Xx Dd	To extend a moving stroke by twice the offset stroke which has been set in the offset memory.
G48 Xx Dd	To contract a moving stroke by twice the offset stroke which has been set in the offset memory.

3. Detailed description

- Programming based on incremental data is shown below.

Tape command	Stroke of movement by equivalent tape command (selected offset stroke = ℓ)	Example (with $x = 1000$)
G45 Xx Dd	$X \{x + \ell\}$	$\ell = 10 \quad X = 1010$ $\ell = -10 \quad X = 990$
G45 X-x Dd	$X - \{x + \ell\}$	$\ell = 10 \quad X = -1010$ $\ell = -10 \quad X = -990$
G46 Xx Dd	$X \{x - \ell\}$	$\ell = 10 \quad X = 990$ $\ell = -10 \quad X = 1010$
G46 X-x Dd	$X - \{x - \ell\}$	$\ell = 10 \quad X = -990$ $\ell = -10 \quad X = -1010$
G47 Xx Dd	$X \{x + 2 \cdot \ell\}$	$\ell = 10 \quad X = 1020$ $\ell = -10 \quad X = 980$
G47 X-x Dd	$X - \{x + 2 \cdot \ell\}$	$\ell = 10 \quad X = -1020$ $\ell = -10 \quad X = -980$
G48 Xx Dd	$X \{x - 2 \cdot \ell\}$	$\ell = 10 \quad X = 980$ $\ell = -10 \quad X = 1020$
G48 X-x Dd	$X - \{x - 2 \cdot \ell\}$	$\ell = 10 \quad X = -980$ $\ell = -10 \quad X = -1020$

- Even if no offset numbers are set in the same block as that which contains commands G45 to G48, offsetting will be performed, based on previously stored tool position offset numbers.
- An alarm **839 ILLEGAL OFFSET No.** will occur if the designated offset number is an unavailable one.
- These G-code commands are not modal ones, and thus they are valid only for the designated block.
- These commands must be used in modes other than the fixed-cycle mode. They will be ignored if used in the fixed-cycle mode.
- The axis will move in reverse if internal calculation for changing the movement distance results in inversion of the direction of movement.



- The following lists how the machine operates if a movement distance of 0 using the incremental data command mode (G91) is programmed:

NC command	G45 X0 D01	G45 X-0 D01	G46 X0 D01	G46 X-0 D01
Equivalent command	X1234	X-1234	X-1234	X1234

D01: Offset number
1234: Offset amount for D01

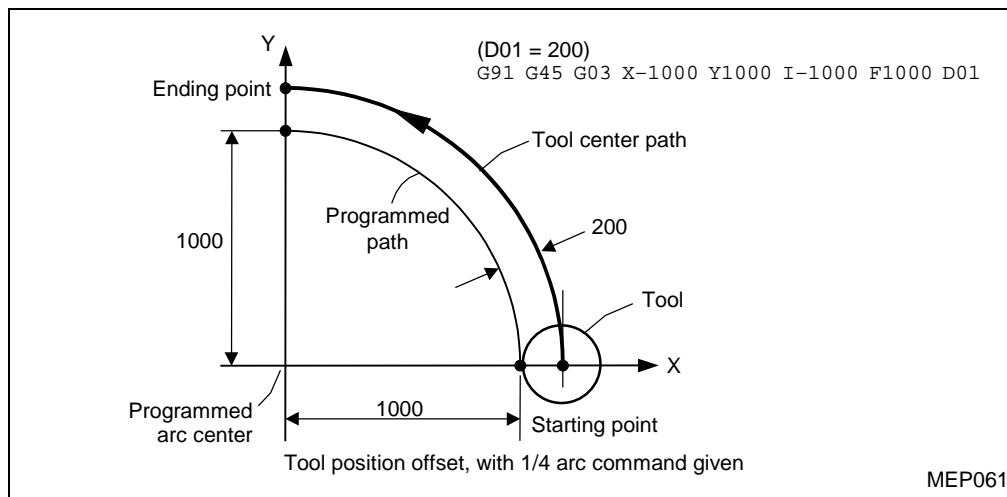
For absolute data commands, if the movement distance is set equal to 0, the block will be immediately completed and no movement through the offset distance will occur.

- When absolute data commands are used, each axis will also move from the ending point preset in the preceding block to the position set in the block that contains commands G45 through G48.

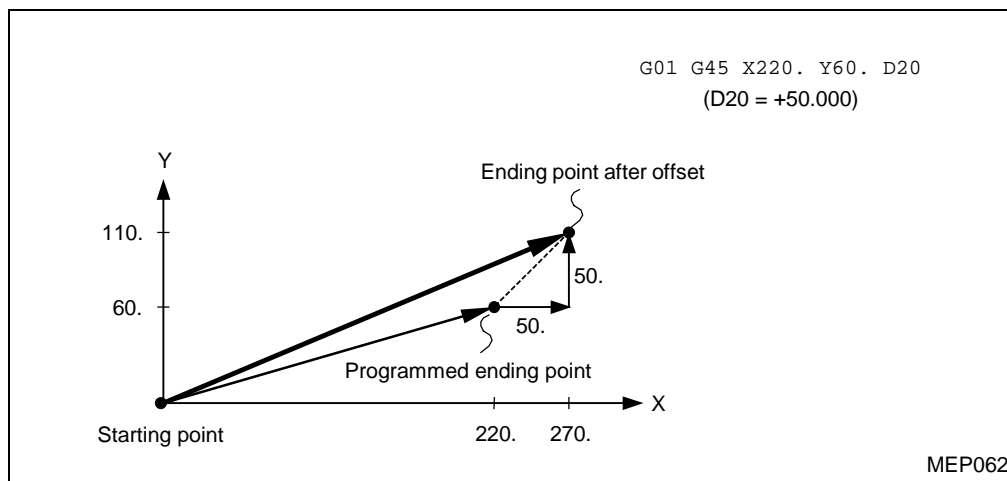
That is, when absolute data commands are used, offsetting will be performed according to the movement distance (increments in distance) set in that block.

4. Sample programs

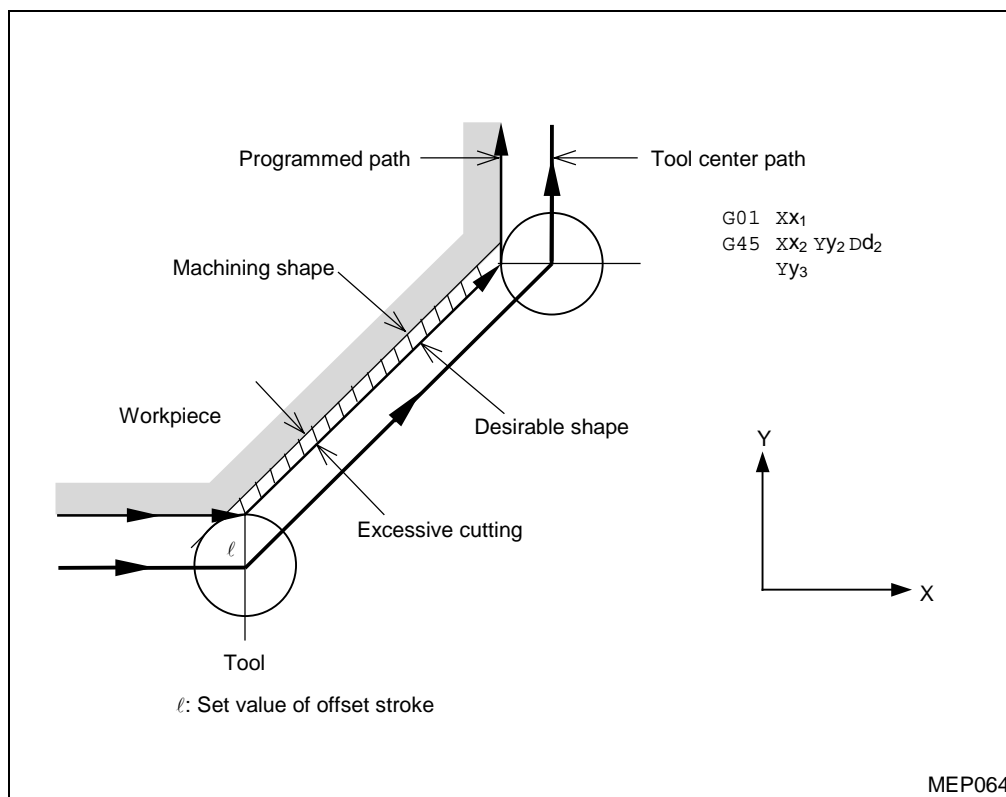
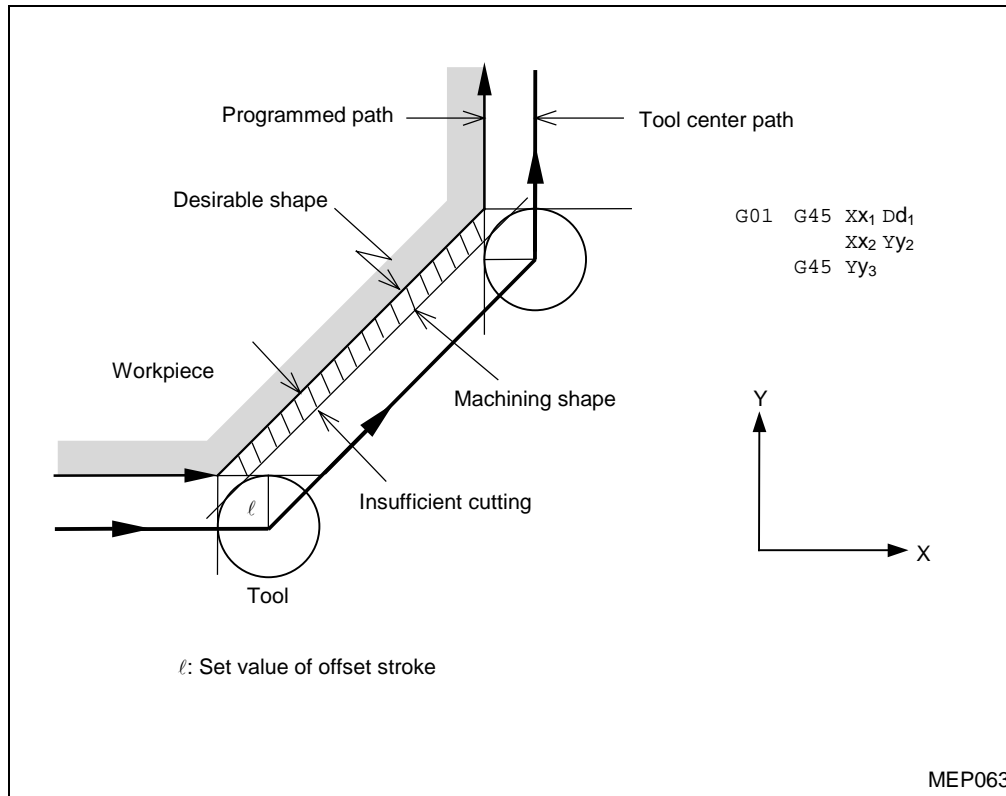
1. During circular interpolation, tool radius compensation using commands G45 to G48 can be done only for a 1/4, 1/2, or 3/4 circle whose starting and ending points are present on a coordinate axis passing through the arc center.



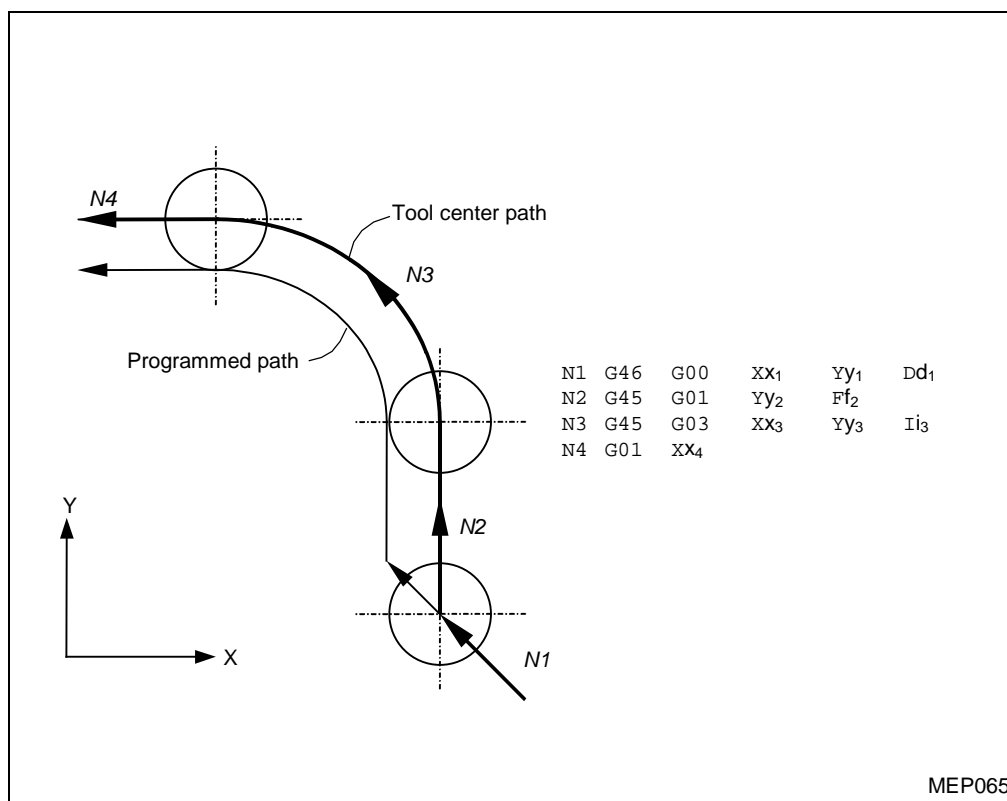
2. If an "n" number of axes are designated at the same time, the same amount of offsetting will be performed on all designated axes. This also applies to additional axes, but within the limits of the simultaneously controllable axis quantity.



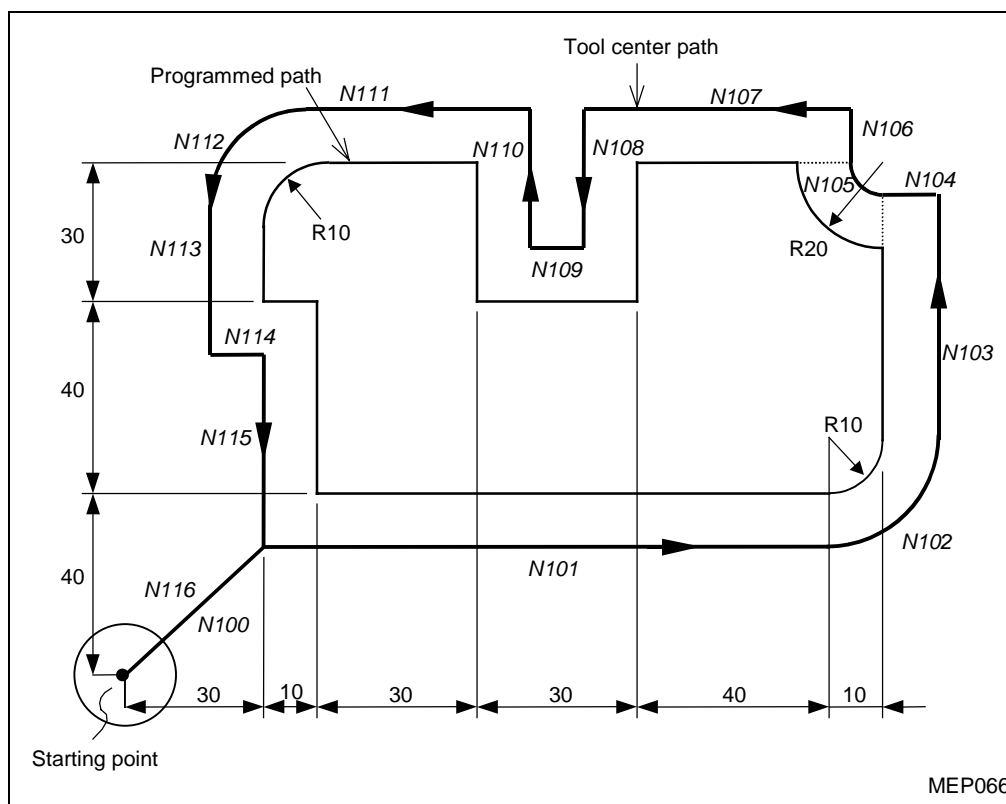
Note: Use tool radius compensation commands G40, G41, or G42 if simultaneous offsetting of two axes is likely to result in excessive or insufficient cutting as shown below.



3. Cornering in a 1/4 circle



4. When commands G45 to G48 are set, each of the corresponding amounts of offsetting will become those designated by the offset numbers; unlike the tool length offset command (G43), these commands will not move the axes through the difference from the previous offset amount.



MEP066

Offset stroke: D01 = 10.000 mm (Tool radius compensation stroke)

```

N100 G91 G46 G00 X40. Y40. D01
N101 G45 G01 X100. F200
N102 G45 G03 X10. Y10. J10.
N103 G45 G01 Y40.
N104 G46 X0
N105 G46 G02 X-20. Y20. J20.
N106 G45 G01 Y0
N107 G47 X-30.
N108 Y-30.
N109 G48 X-30.
N110 Y30.
N111 G45 X-30.
N112 G45 G03 X-10. Y-10. J-10.
N113 G45 G01 Y-20.
N114 X10.
N115 Y-40.
N116 G46 X-40. Y-40.
N117 M02
%
```

12-5 Tool Radius Compensation: G40, G41, G42

12-5-1 Overview

1. Function and purpose

Offsetting in any vectorial direction can be done according to the tool radius preselected using G-codes (G38 to G42) and D-codes. This function is referred to as tool radius compensation.

For turning tools, nose-R compensation can be performed according to the designated direction (only when TOOL OFFSET type C is selected).

2. Programming format

Programming format	Function	Remarks
G40X_Y_	To cancel a tool radius compensation	These commands can be given in the mode of tool radius compensation.
G41X_Y_	To compensate a tool radius (to the left)	
G42X_Y_	To compensate a tool radius (to the right)	
G38 I_J_	To change and hold an offset vector	
G39	To interpolate a corner arc	

3. Detailed description

For tool radius compensation, all H-code commands are ignored and only D-code commands become valid.

Also, tool radius compensation is performed for the plane that is specified by either the plane selection G-code command or two-axis address code command appropriate for tool radius compensation. No such compensation is performed for axes other than those corresponding or parallel to the selected plane. See Section 6-7 to select a plane using a G-code command.

12-5-2 Tool radius compensation

1. Cancellation of tool radius compensation

Tool radius compensation is automatically cancelled in the following cases:

- After power has been turned on
- After the reset key on the NC operation panel has been pressed
- After M02 or M30 has been executed (these two codes have a reset function)
- After G40 (offsetting cancellation command) has been executed

In the offsetting cancellation mode, the offset vector becomes zero and the tool center path agrees with the programmed path.

Programs containing the tool radius compensation function must be terminated during the compensation cancellation mode. Give the G40 command in a single-command block (without any other G-code). Otherwise it may be ignored.

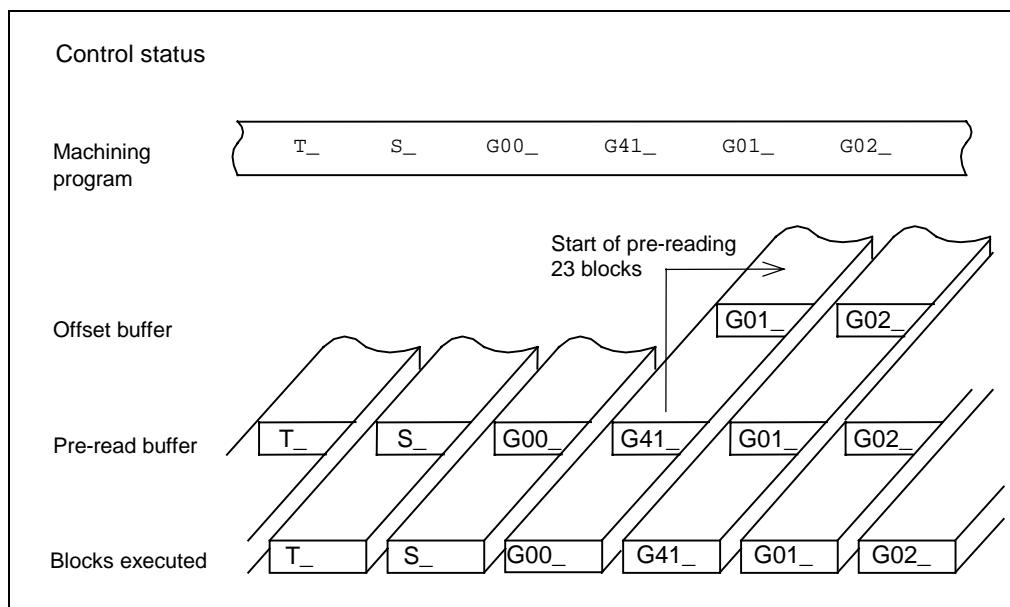
2. Startup of tool radius compensation

Tool radius compensation will begin during the compensation mode when all the following three conditions are met:

- Command G41 or G42 has been executed.
- The offset number for tool radius compensation is larger than zero, but equal to or smaller than the maximum available offset number.
- The command used with the compensation command is a move command other than those used for circular interpolation.

Compensation will be performed only when reading of three motion blocks, if possible within a range of 23 blocks (inclusive of the startup block), is completed, irrespective of whether the single-block operation mode is used.

During compensation, 23 blocks are pre-read and then calculation for compensation is performed.



There are two types of compensation startup operation: Type A and Type B.

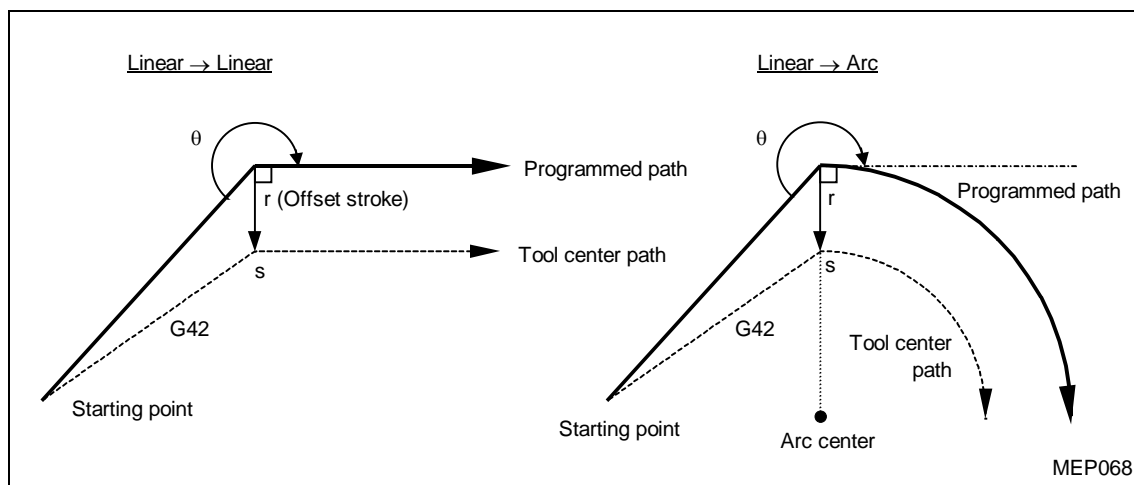
It depends on the setting of bit 4 of parameter **F92** whether Type A or Type B is automatically selected.

These two types of startup operation are similar to those of compensation cancellation.

In the descriptive diagrams below, "s" signifies the ending point of single-block operation.

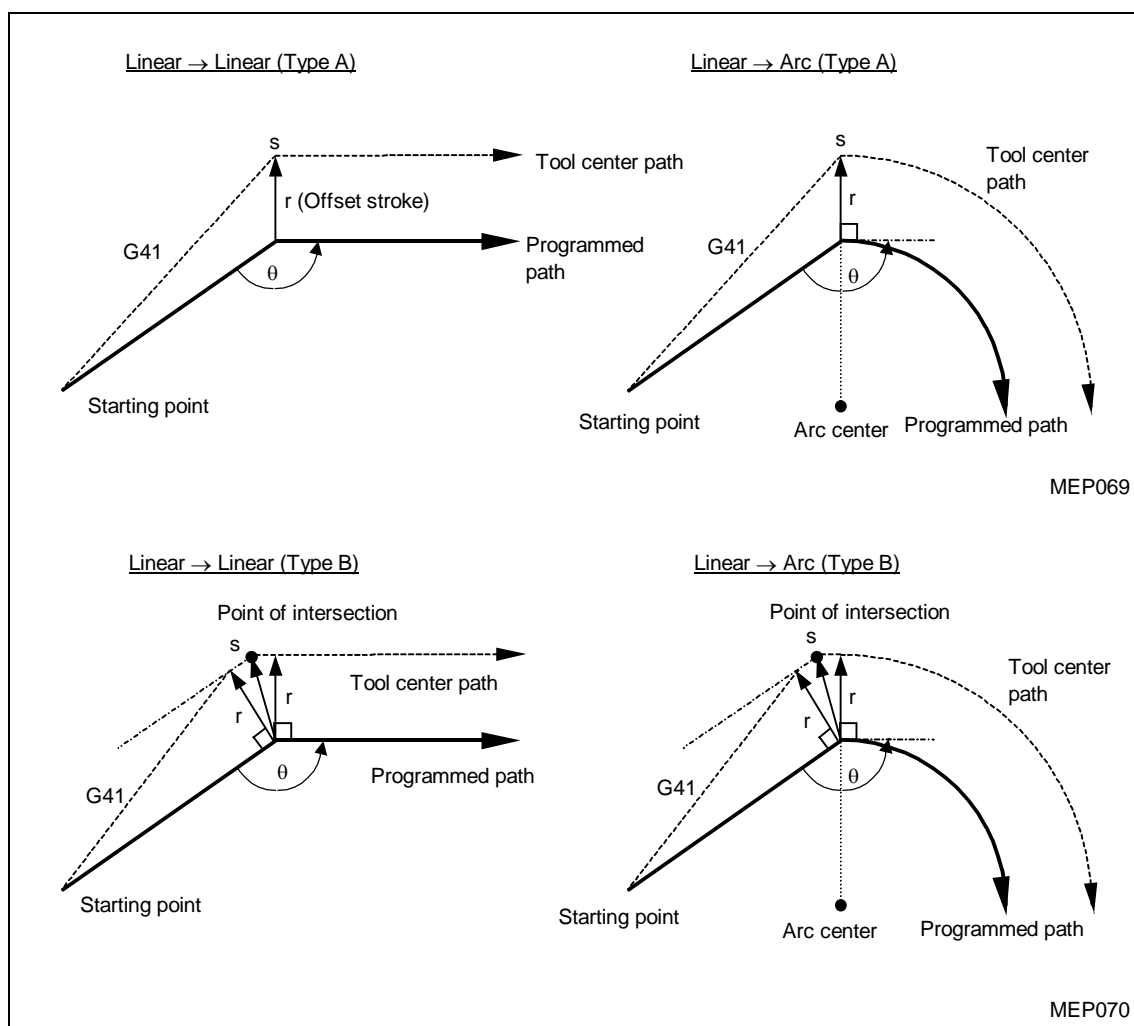
3. Startup operation of tool radius compensation

A. For the corner interior



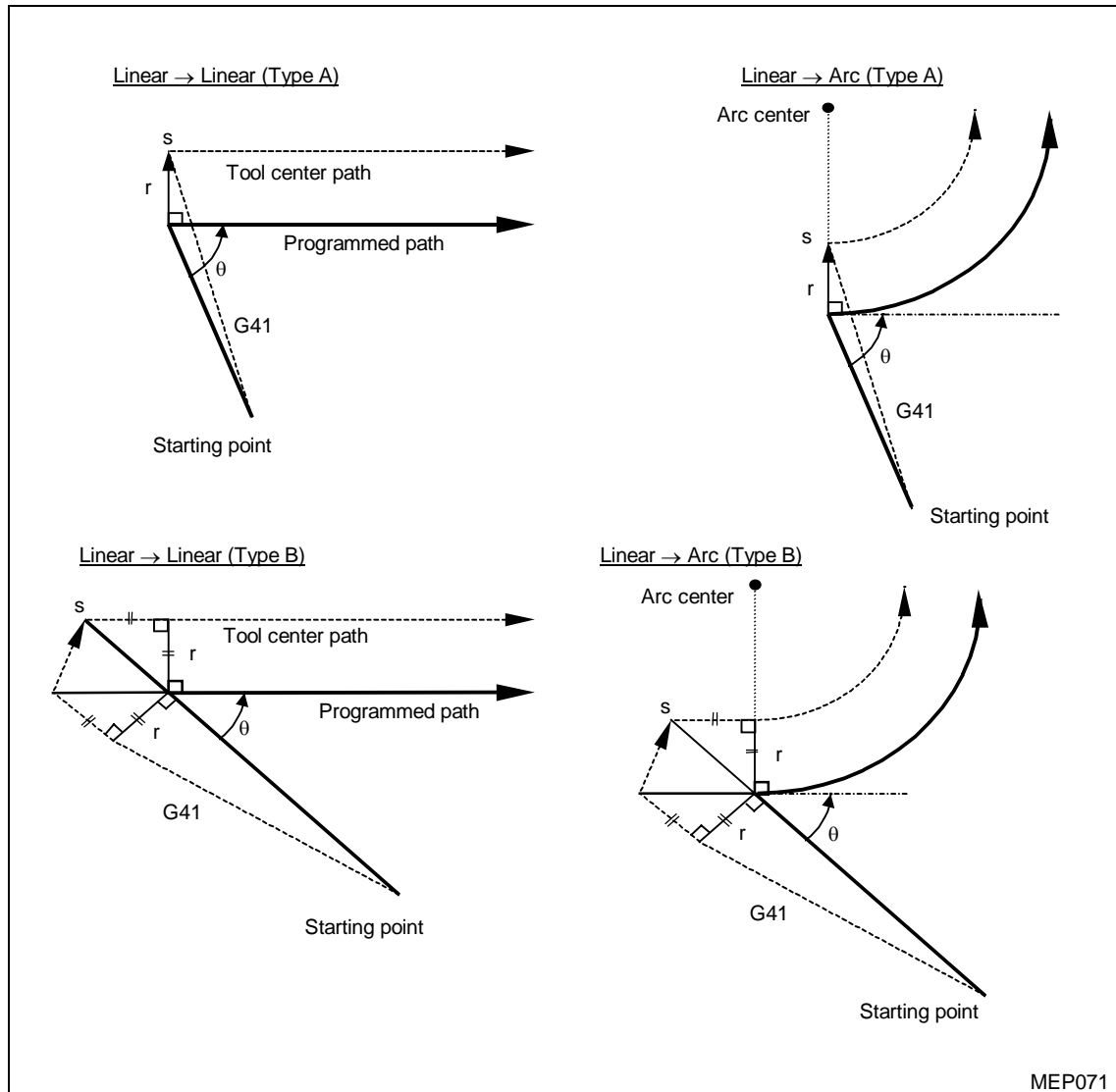
B. For the corner exterior (obtuse angle) [$90^\circ \leq \theta < 180^\circ$]

(Type A/B selection is possible with a predetermined parameter.)



C. For the corner exterior (sharp angle) [$\theta < 90^\circ$]

(Type A/B selection is possible with a predetermined parameter.)



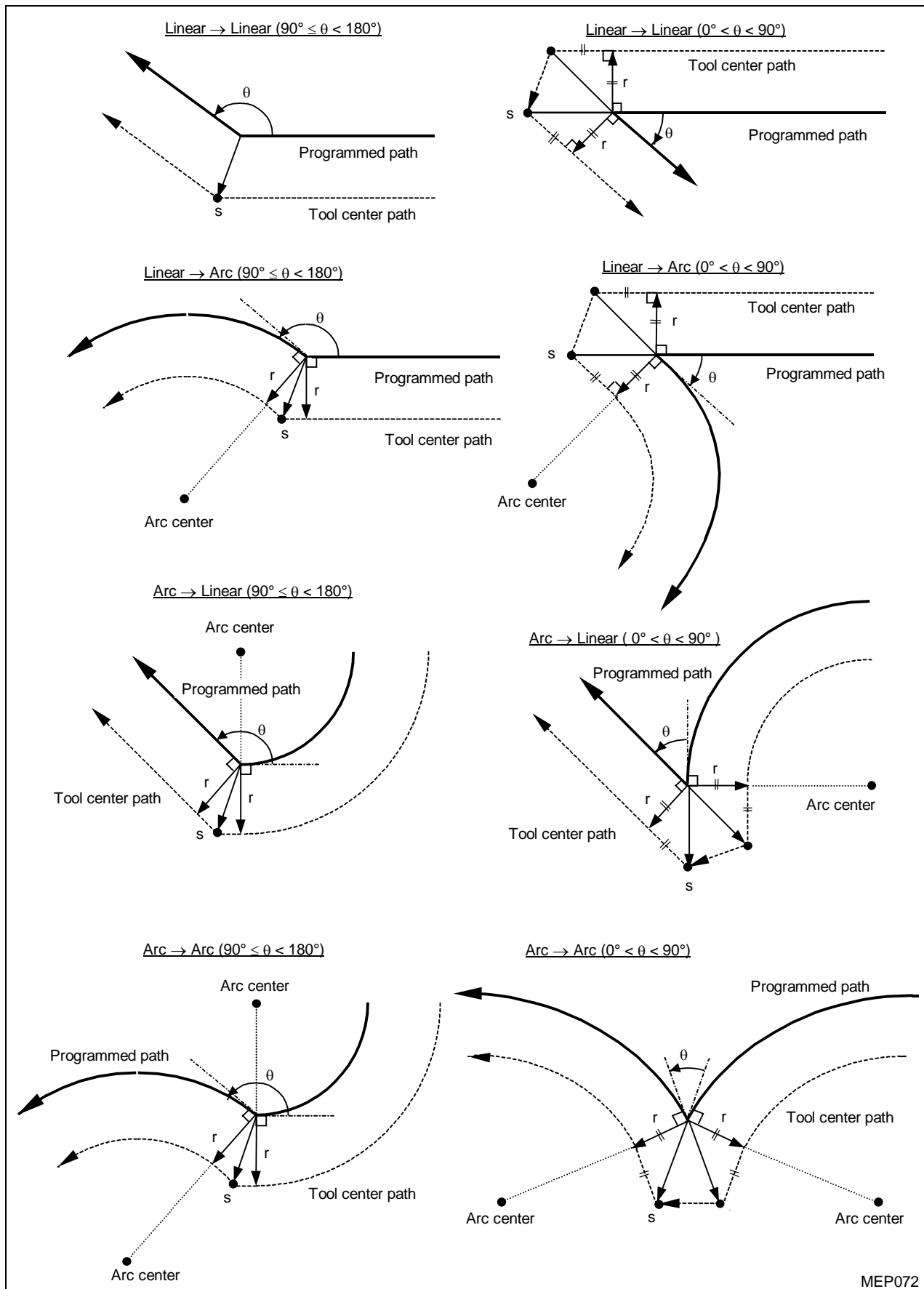
4. Operation during the compensation mode

Compensation is performed for linear or arc interpolation commands and positioning commands. Identical compensation commands G41 or G42 will be ignored if they are used in the compensation mode.

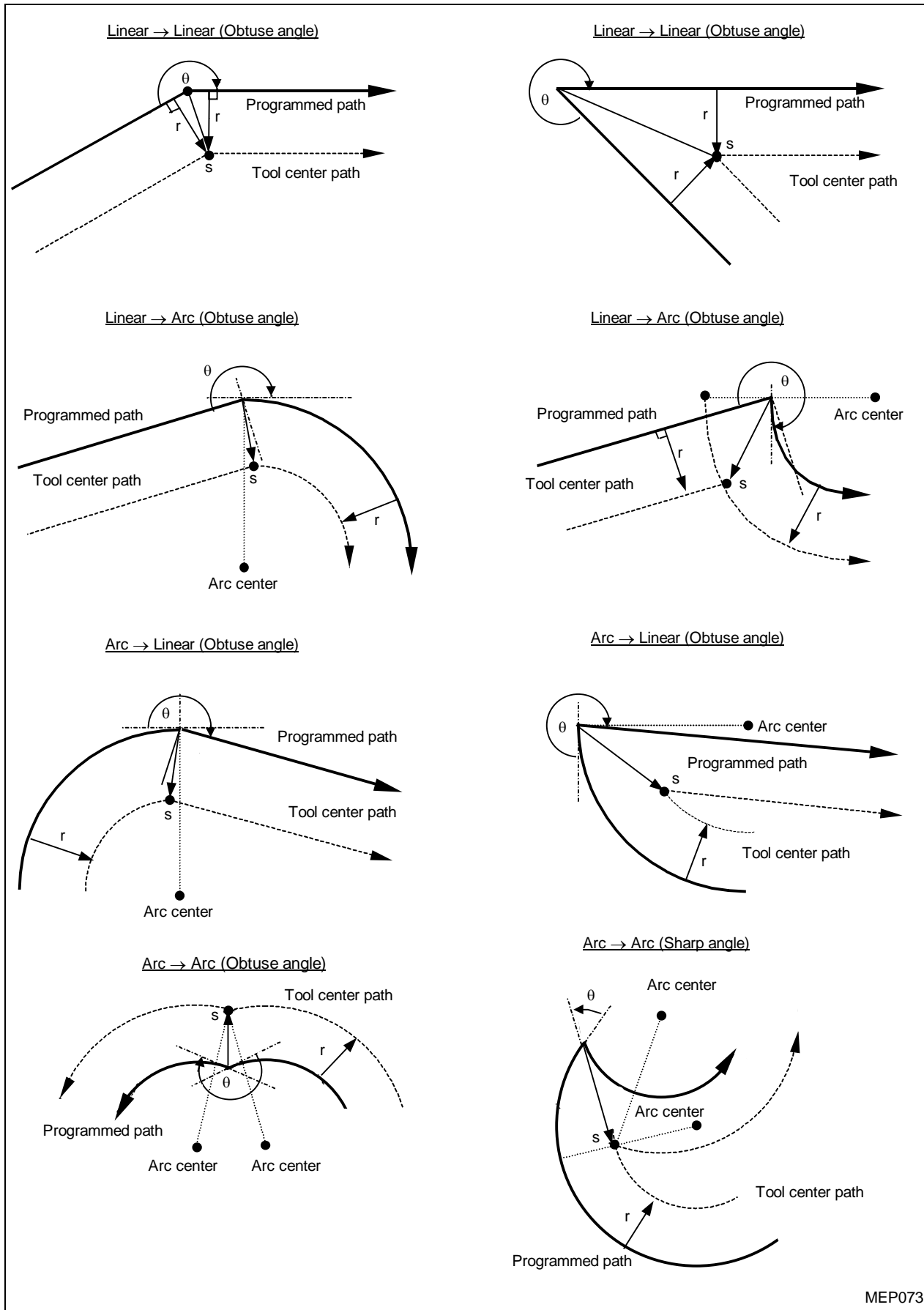
Successive setting of 22 or more move-free blocks (*) during the compensation mode may result in excessive or insufficient cutting.

(*) See Subsection 12-5-3 for a description of 'move-free' blocks.

A. For the corner exterior



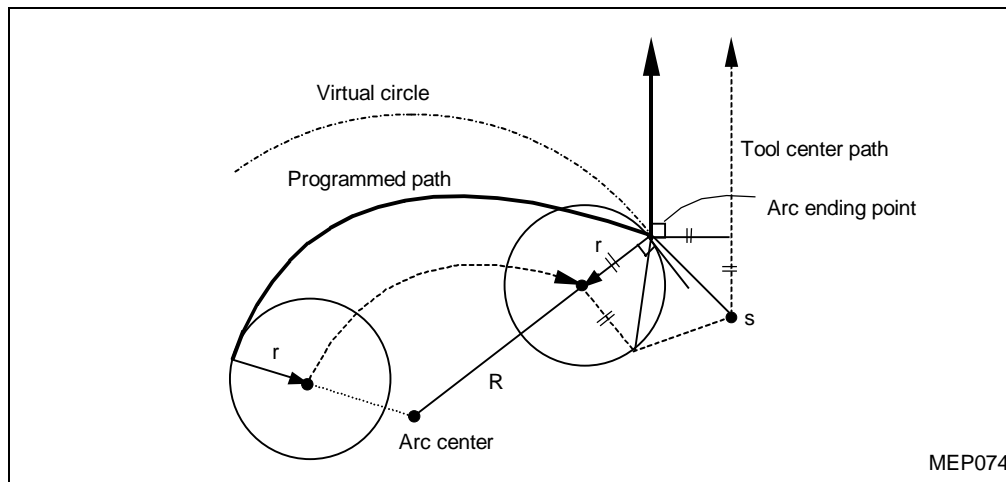
MEP072

B. For the corner interior

MEP073

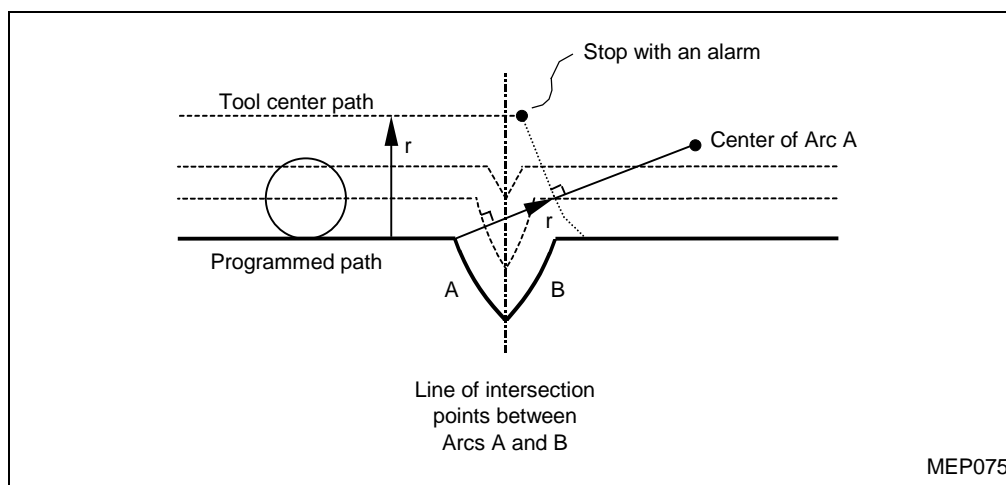
C. For an arc that does not have the ending point on it

The area from the starting point of the arc to the ending point is interpolated as a spiral arc.



D. For arcs that do not have their inner crossing point

In cases such as those shown in the diagram below, there may or may not be a crossing point of arcs A and B, depending on the particular offset data. In the latter case, the program terminates at the ending point of the preceding block after an alarm **836 NO INTERSECTION** has been displayed.



5. Cancellation of tool radius compensation

During the tool radius compensation mode, tool radius compensation will be cancelled in any of the two cases listed below.

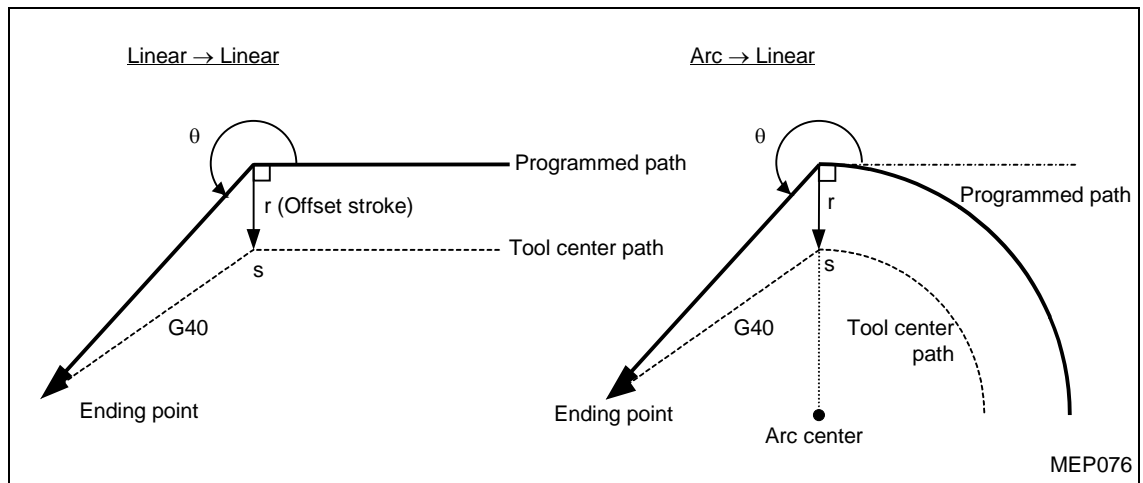
- Command G40 has been executed.
- Offset number code D00 has been executed.

At this time, however, the move command should not be given under the mode of circular interpolation. An alarm **835 G41, G42 FORMAT ERROR** will occur if an attempt is made to cancel the compensation using a circular interpolation command.

After the compensation cancellation command has been read into the offset buffer, the cancellation mode is set automatically and subsequent blocks of data are read into the pre-read buffer, not the offset buffer.

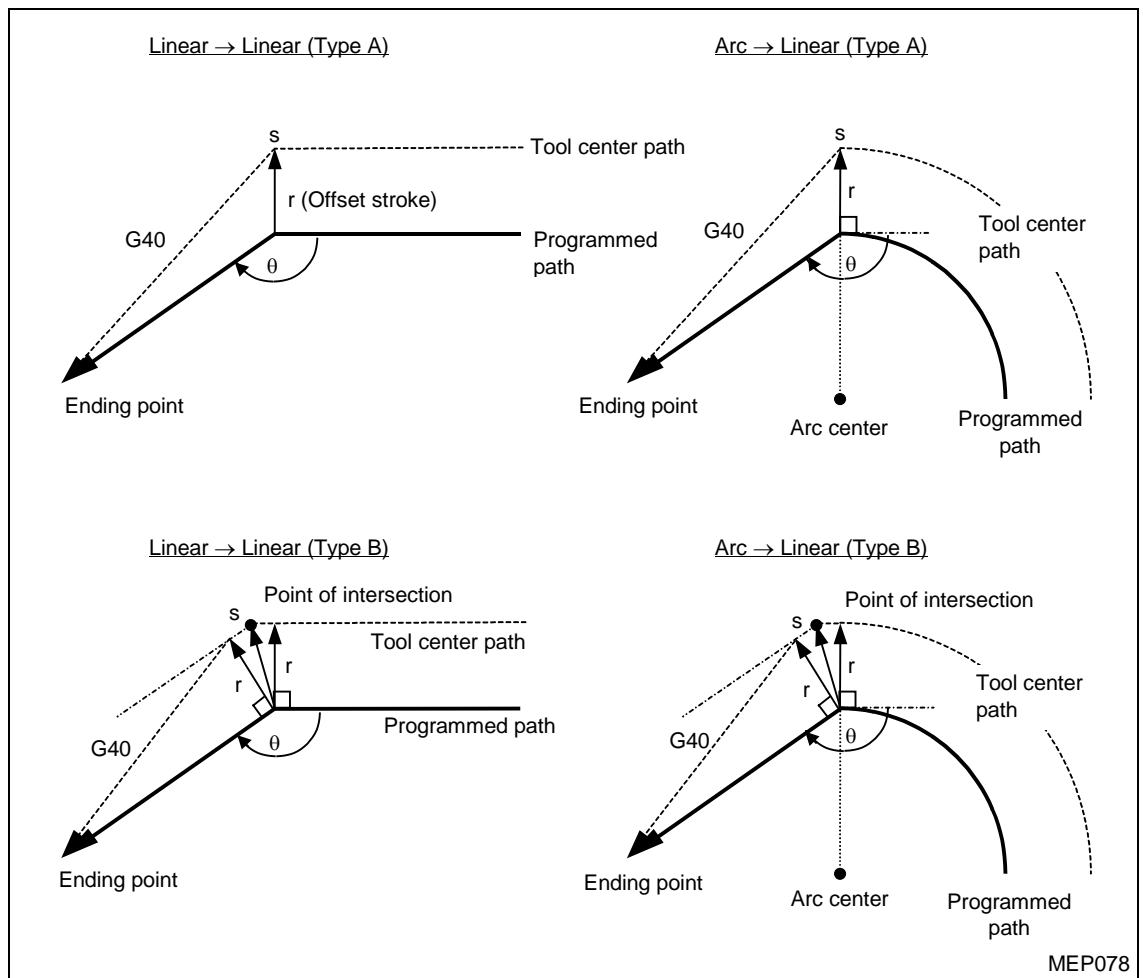
6. Cancellation operation of tool radius compensation

A. For the corner interior



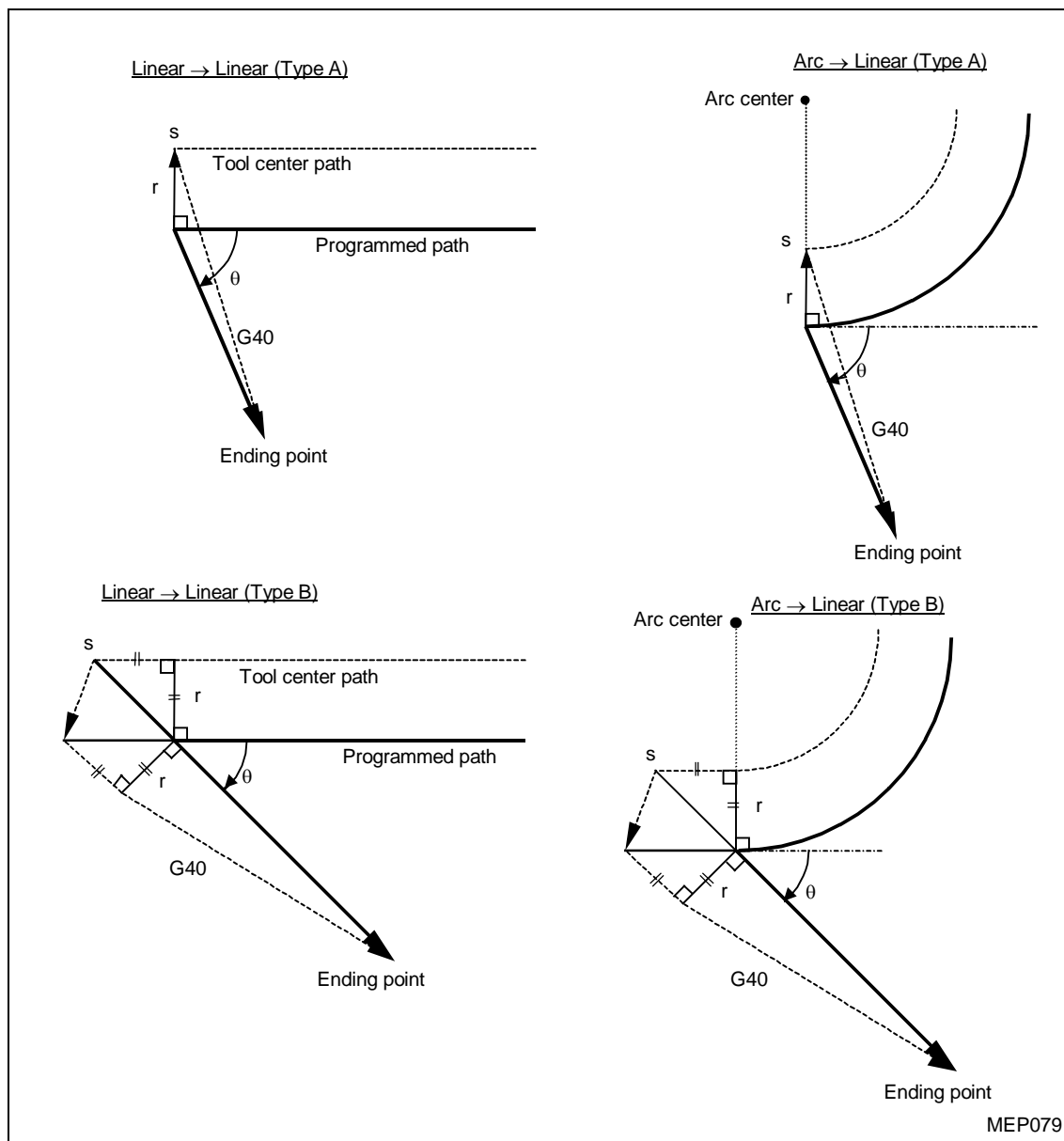
B. For the corner exterior (obtuse angle)

(Type A/B selection is possible with a predetermined parameter)



C. For the corner exterior (sharp angle)

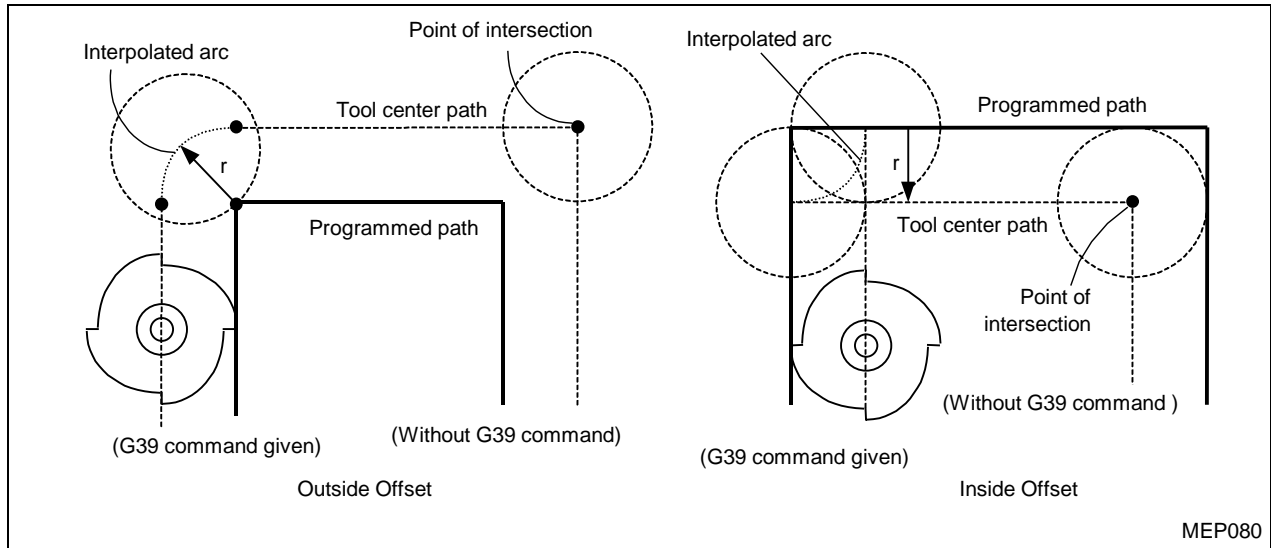
(Type A/B selection is possible with a predetermined parameter)



12-5-3 Tool radius compensation using other commands

1. Interpolation of the corner arc

When command G39 (corner-arc interpolation) is used, the coordinates of the crossing points at workpiece corners will not be calculated and an arc with offset data as its radius will be interpolated.



2. Changing/retaining offset vectors

Using command G38, you can change or retain offset vectors during tool radius compensation.

- Retaining vectors

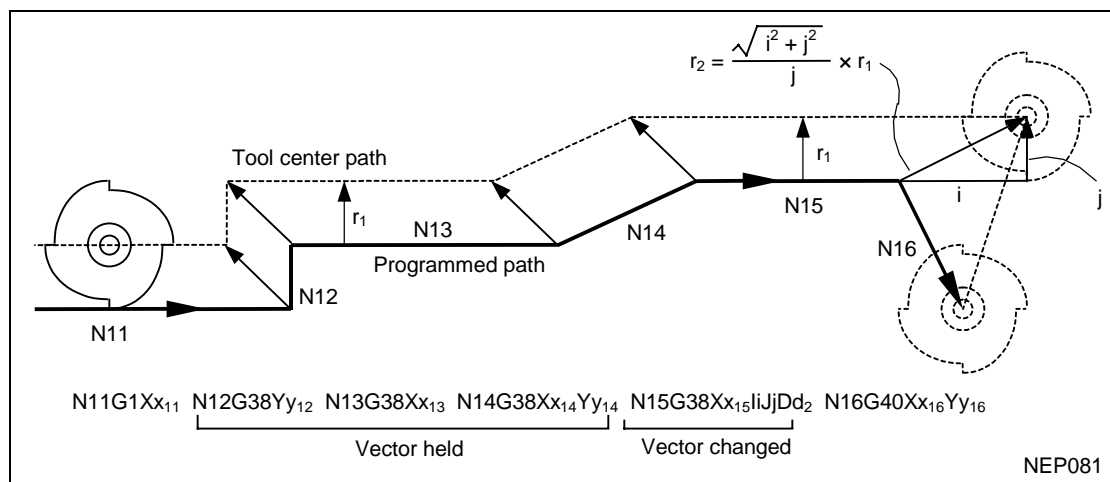
Setting G38 in a block that contains move commands allows crossing-point calculation at the ending point of that block to be cancelled and the vectors in the preceding block to be retained. This can be used for pick and feed operations.

G38 Xx Yy

- Changing vectors

The directions of new offset vectors can be designated using I, J, and K (I, J, and K depend on the selected type of plane), and offset data can be designated using D. (These commands can be included in the same block as that which contains move commands.)

G38 Ii Jj Dd

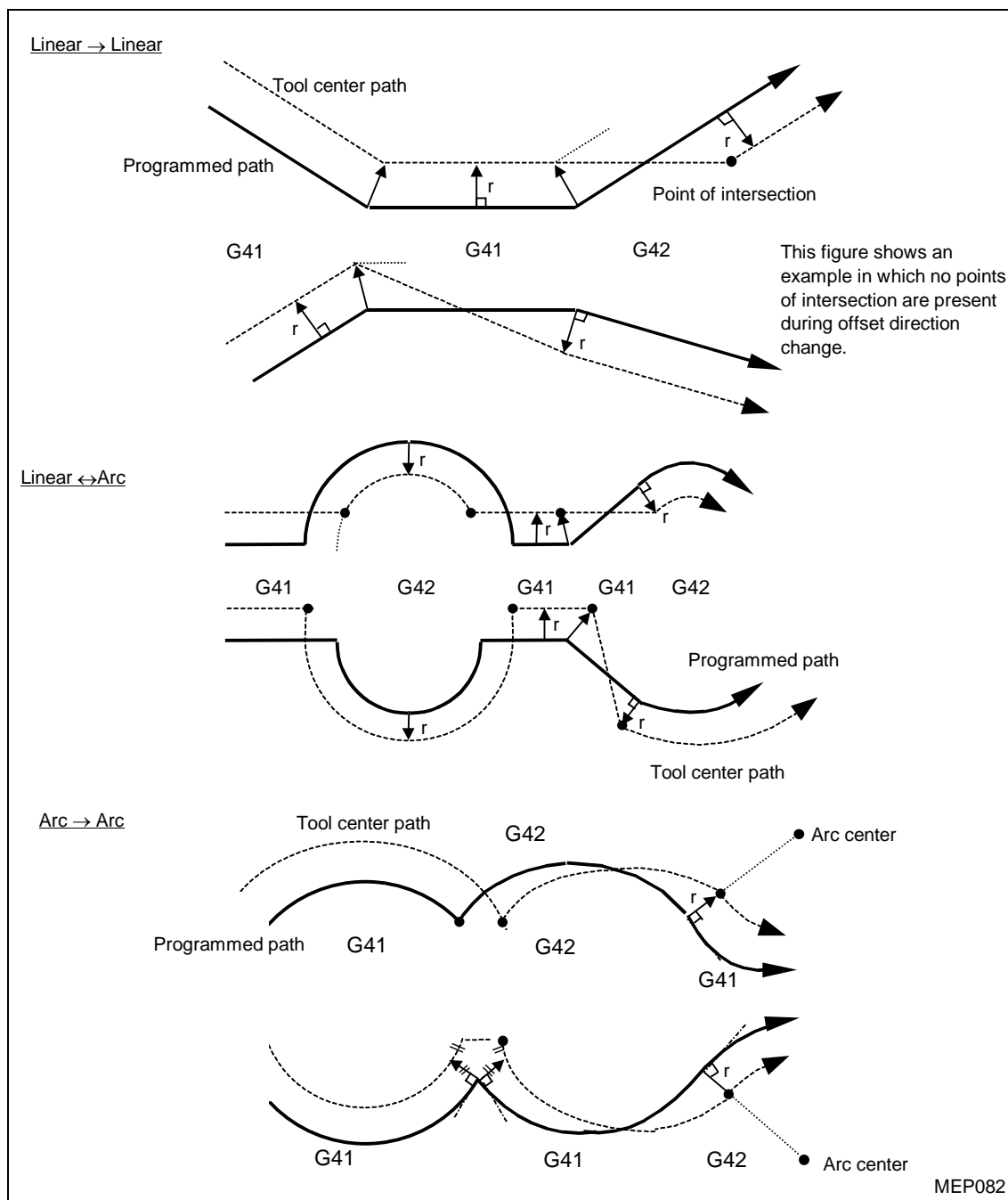


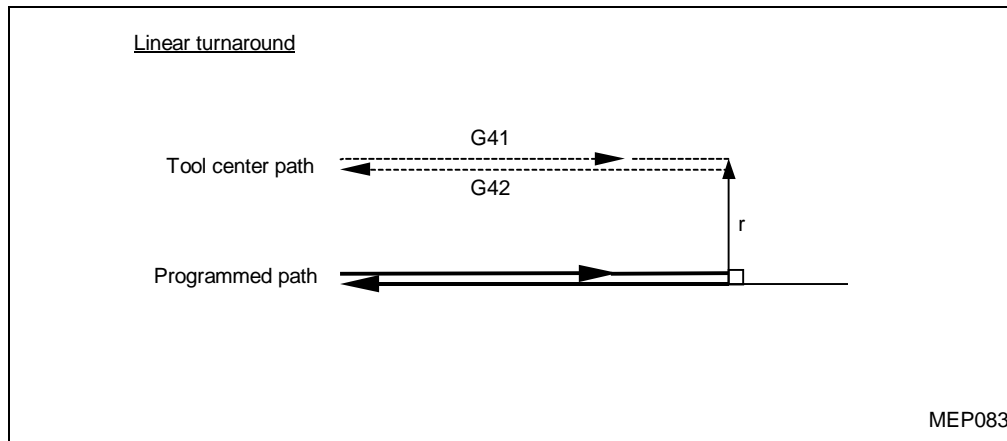
3. Changing the offset direction during tool radius compensation

The offset direction is determined by the type of tool radius compensation command (G41 or G42) and the sign (plus or minus) of the offset data.

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

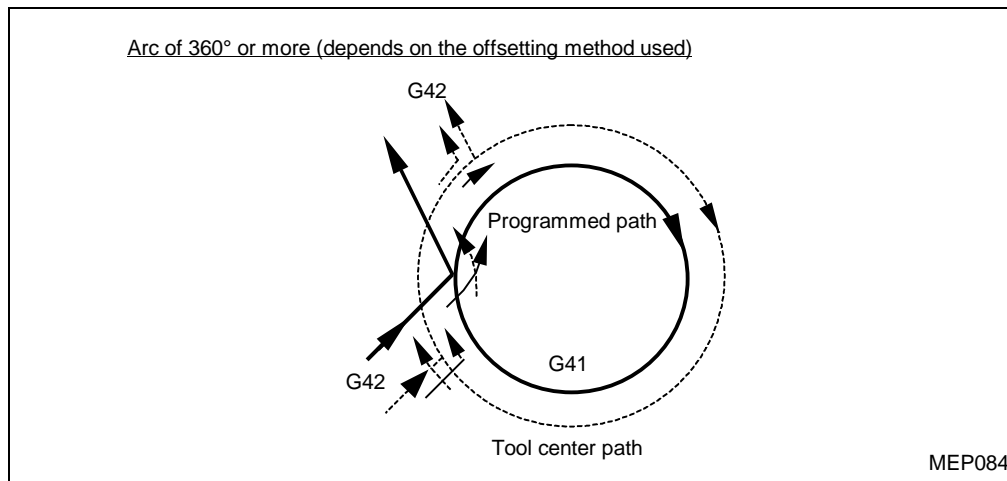
The offset direction can be changed by updating the offset command without having to cancel the compensation mode. This can, however, be done only for blocks other than the compensation startup block and the next block. See Subsection 12-5-6, General precautions on tool radius compensation, for NC operation that will occur if the sign is changed.





The arc of more than 360 degrees may result in the following cases:

- The offset direction has been changed by G41/G42 selection.
- Commands I, J, and K have been set for G40.

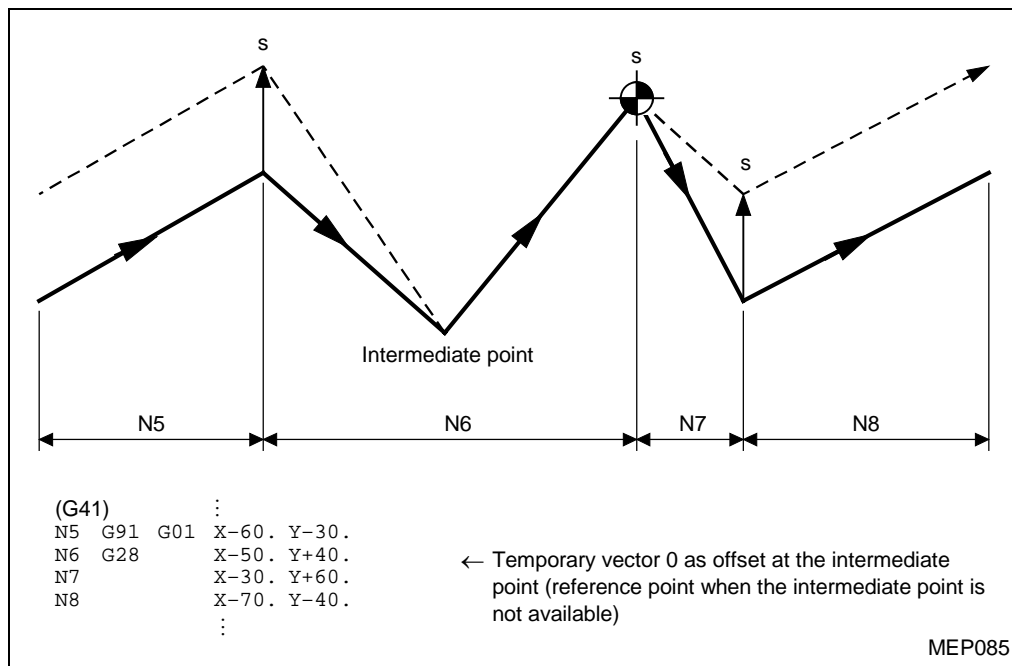


4. Cases where the offset vector is temporarily cancelled

If the command listed below is used during compensation mode, the current offset vector will be cancelled temporarily and then the NC unit will re-enter the compensation mode.

In that case, a movement will occur from the crossing point to a vector-less point, that is, to the programmed point, without movements for cancelling the compensation mode. The next movement will also occur directly to the next crossing point when the compensation mode is restored.

A. Reference-point return command



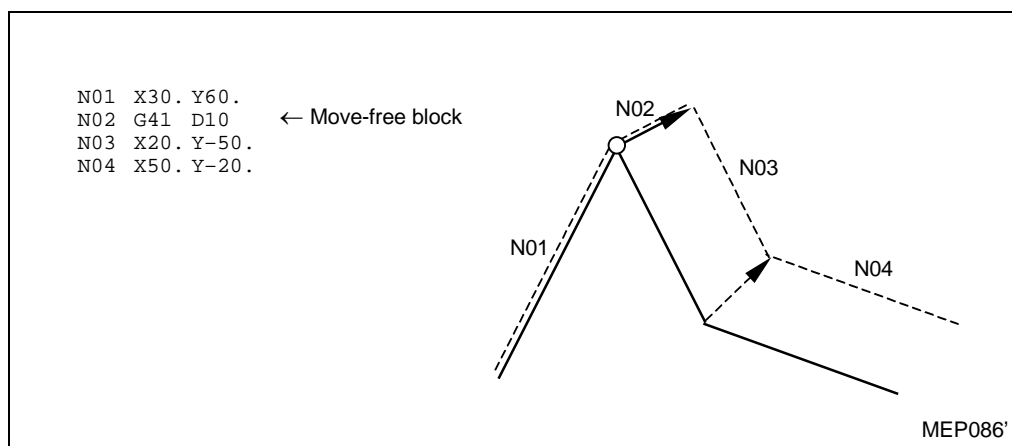
5. Move-free Blocks (that do not include movement)

The blocks listed below are referred to as move-free ones:

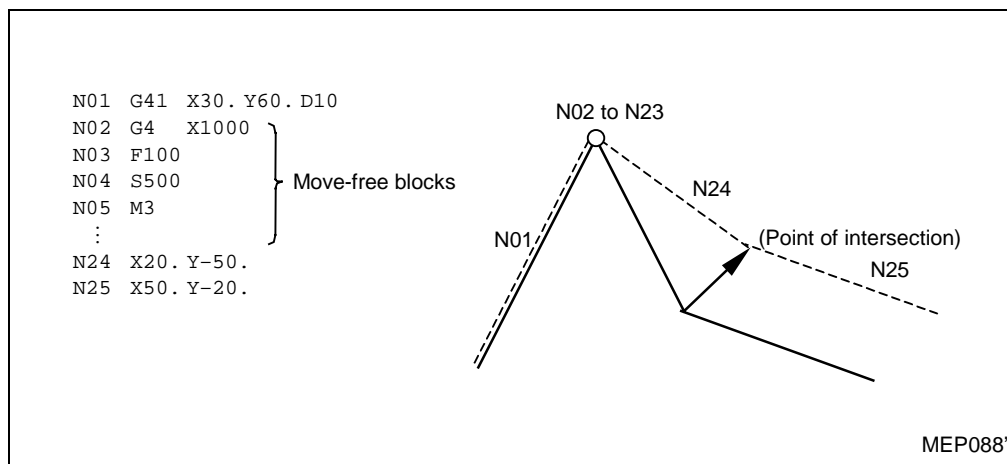
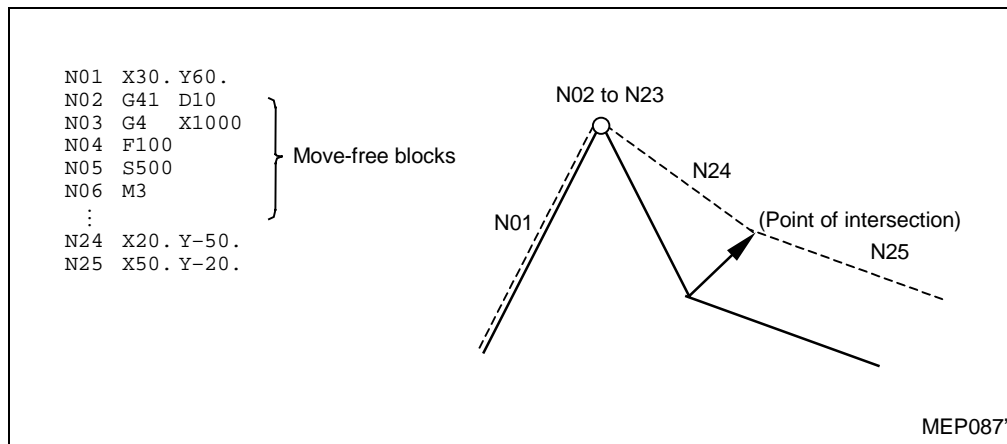
M03.....	M command	}	Move-free
S12.....	S command		
T45.....	T command		
G04 X500.....	Dwell		
G22 X200. Y150. Z100.....	To set a machining-prohibited area		
G10 P01 R50	To set an offset stroke		
G92 X600. Y400. Z500.....	To set a coordinate system		
(G17) Z40.	To move outside the offsetting plane		
G90.....	G-code only		
G91 X0	Moving stroke 0	Moving stroke is 0.

A. When a mve-free block is set at the start of compensation

Vertical offsetting will be performed on the next move block.

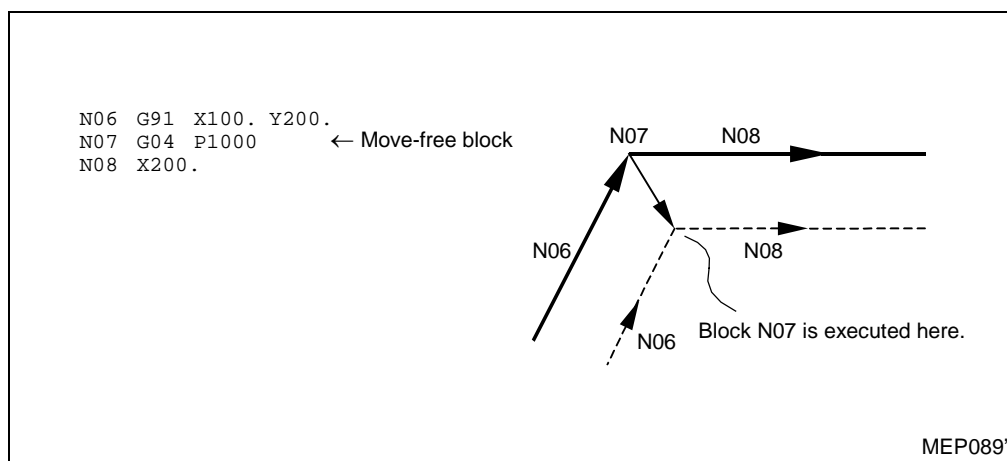


Offset vectors, however, will not be generated if 22 or more blocks that do not include move commands appear in succession.

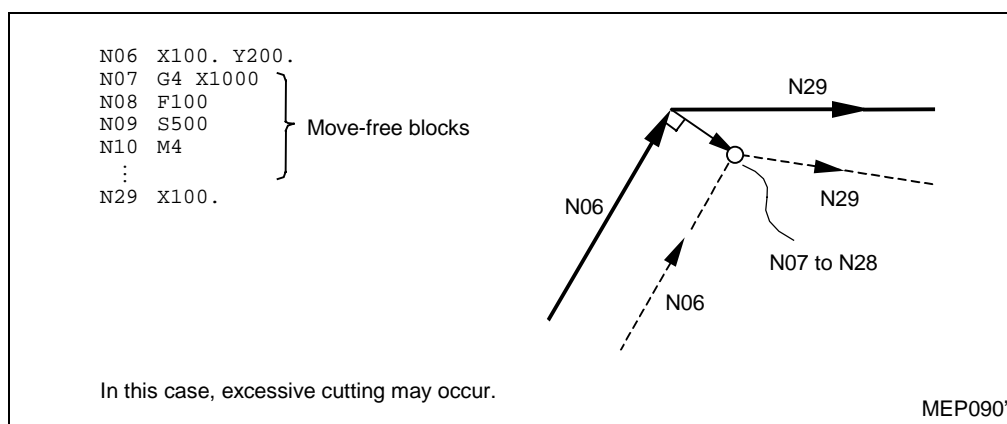


B. When a move-free block is set in the compensation mode

Usual crossing-point vectors will be generated unless 22 or more blocks that do not include movement appear in succession.

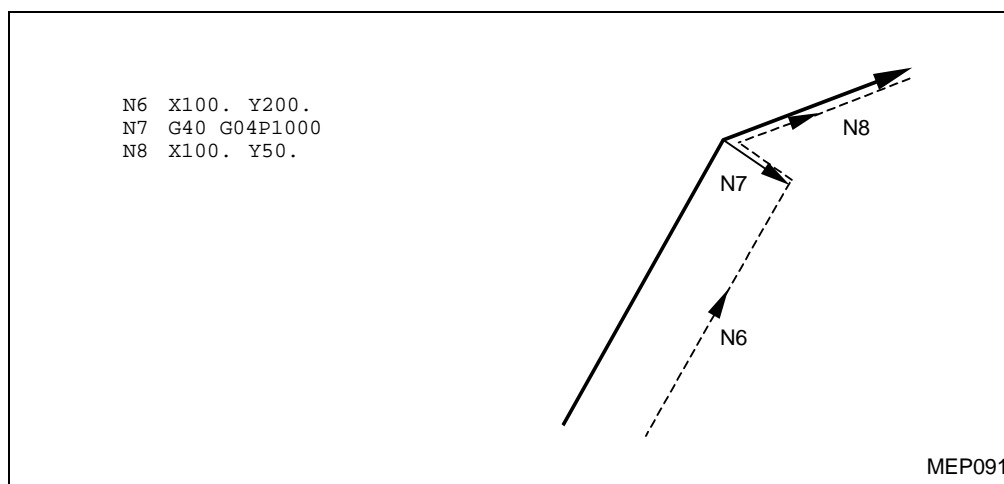


Vertical offset vectors will be generated at the end point of preceding block if 22 or more blocks that do not include movement appear in succession.



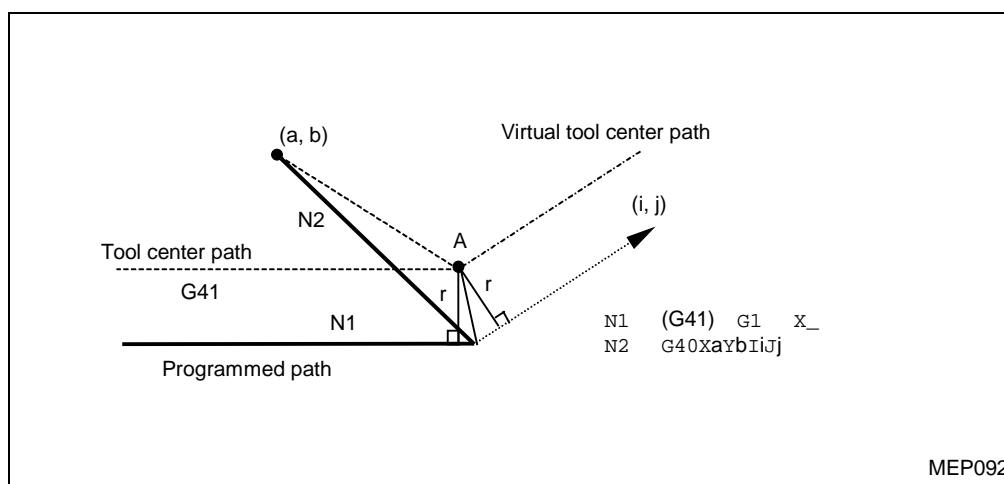
C. When a move-free block contains a cancellation command

Only offset vectors will be cancelled if the block that does not include movement contains G40.

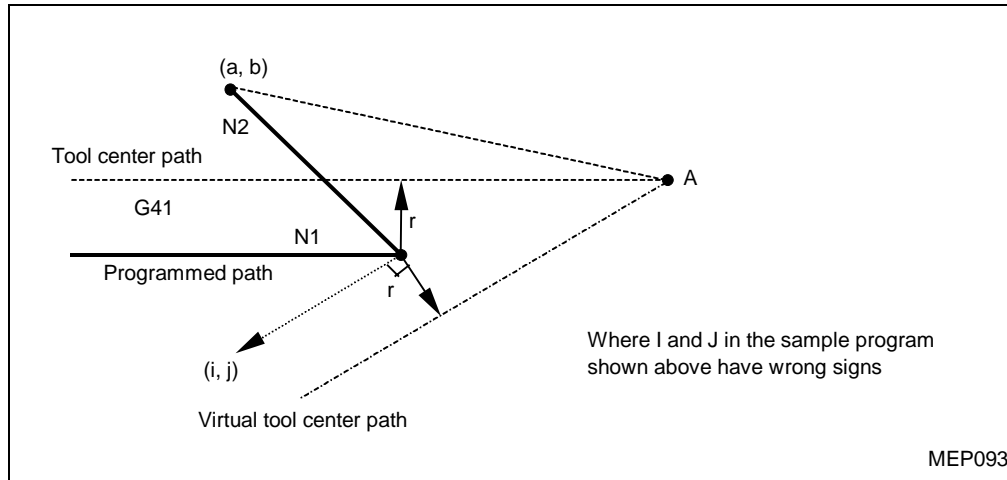


6. If I, J, and K are set with G40

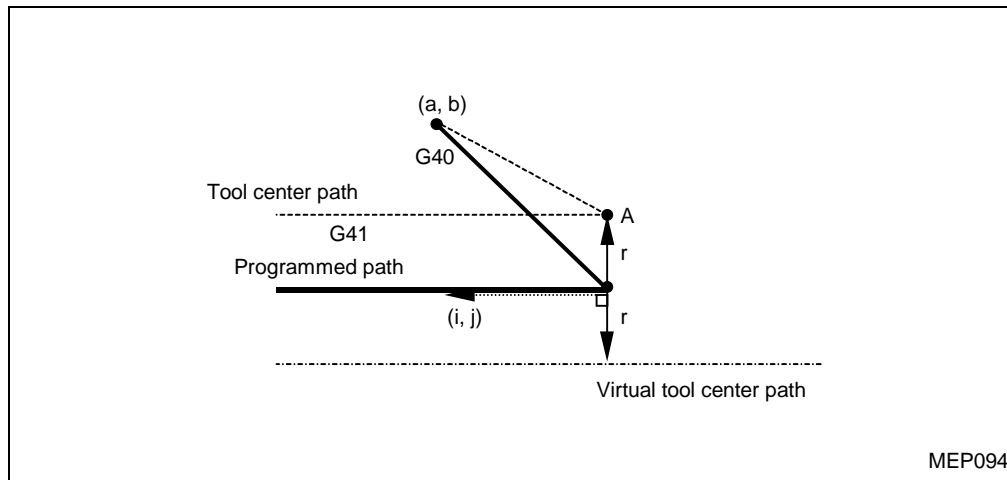
When the last of the four move command blocks which immediately precede the G40 command block contains G41 or G42, movement will be handled as if it had been programmed to occur in the vectorial direction of I, J, and K from the ending point of that last move command. That is, the area up to the crossing point with the virtual tool center path will be interpolated and then compensation will be cancelled. The offset direction will remain unchanged.



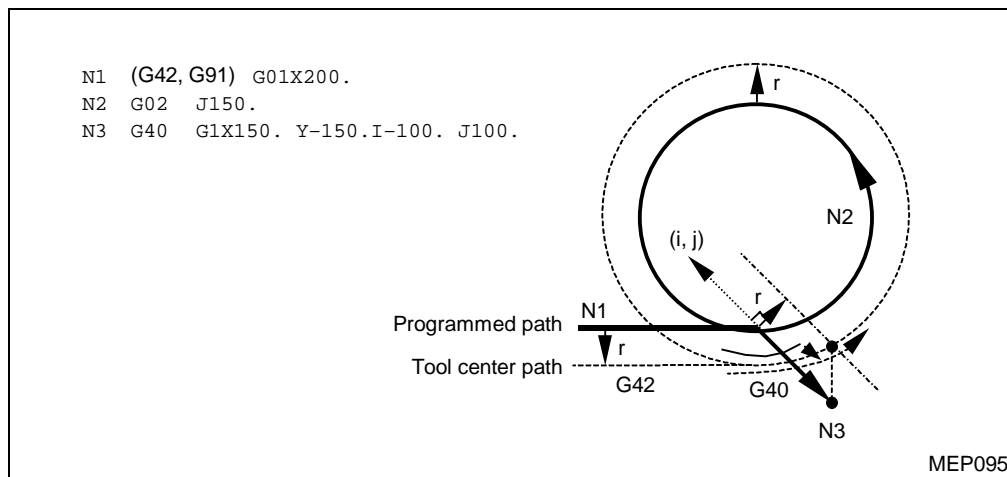
In this case, beware that irrespective of the offset direction, the coordinates of the crossing point will be calculated even if wrong vectors are set as shown in the diagram below.



Also, beware that a vertical vector will be generated on the block before that of G40 if crossing-point calculation results in the offset vector becoming too large.



Note: Part of the workpiece will be cut twice if the I/J/K command in a G40 block preceded by a circular interpolation command generates an arc of more than 360 degrees.

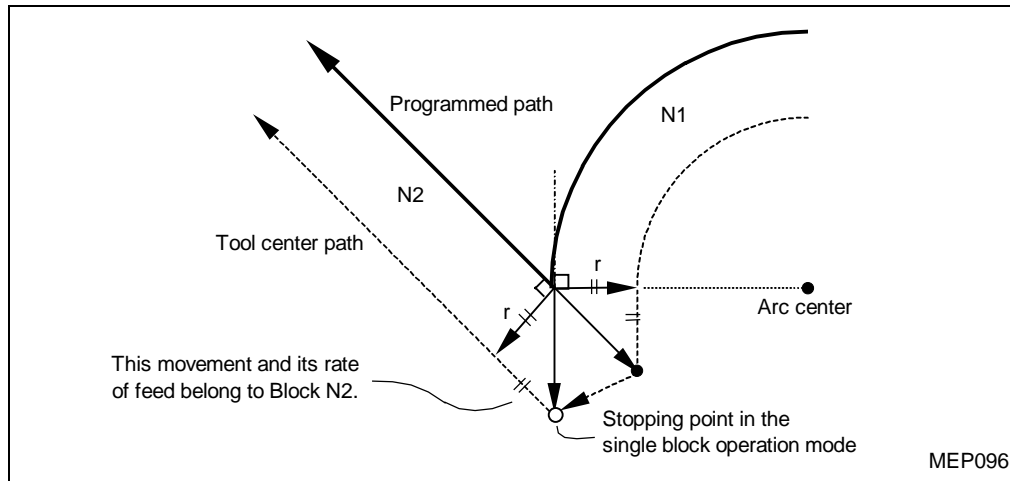


```
N1 (G42, G91) G01X200.
N2 G02 J150.
N3 G40 G1X150. Y-150.I-100. J100.
```

12-5-4 Corner movement

If multiple offset vectors are generated at a connection between move command blocks, a linear (polygonal) movement will occur through the end points of the vectors. This action is referred to as corner movement.

When two different vectors are required for the connection, a linear interpolation will occur around the corner as an addition to the first connecting movement. In the single-block operation, therefore, the first section of "Preceding block + Corner movement" is executed as one block and the second section of "Connecting movement + Next block" as the next one.



12-5-5 Interruptions during tool radius compensation

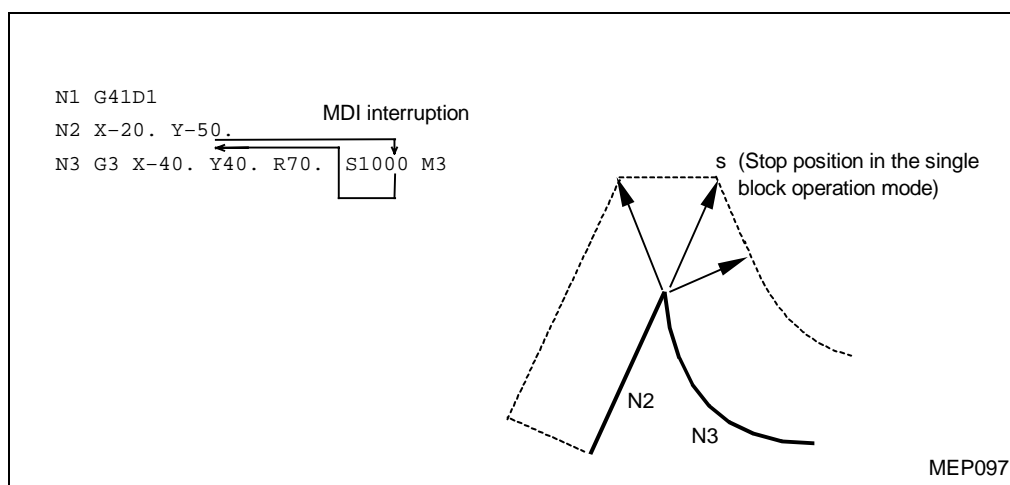
1. Interruption by MDI

Tool radius compensation is valid during automatic operation, whether it is based on the tape, memory, or MDI operation mode.

The following diagrams show what will occur if tape or memory operation is interrupted using the MDI function following termination of the program at a block:

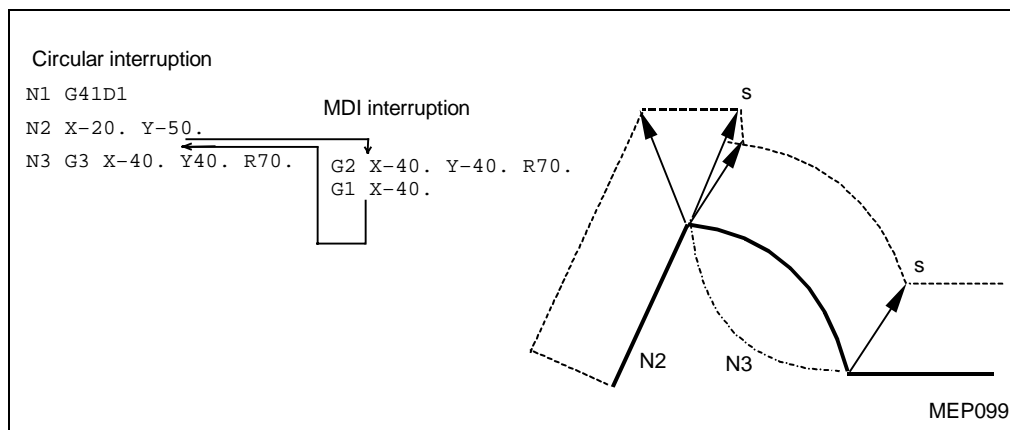
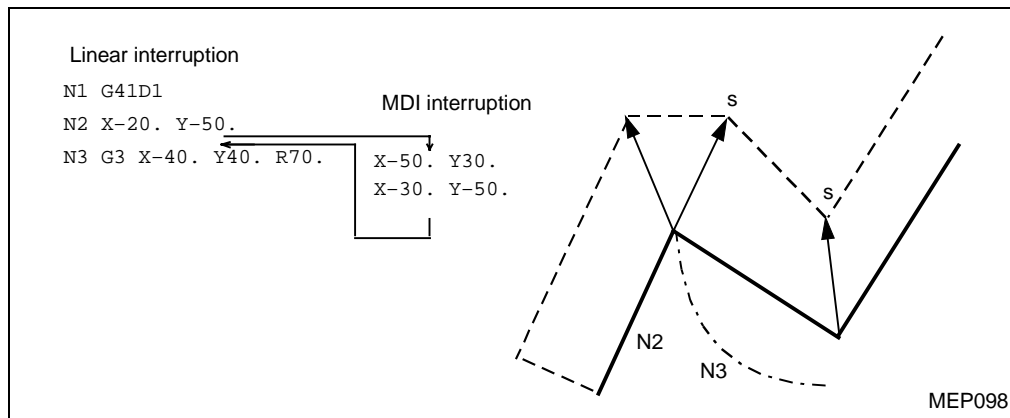
A. Interruption without movement

No change in tool path



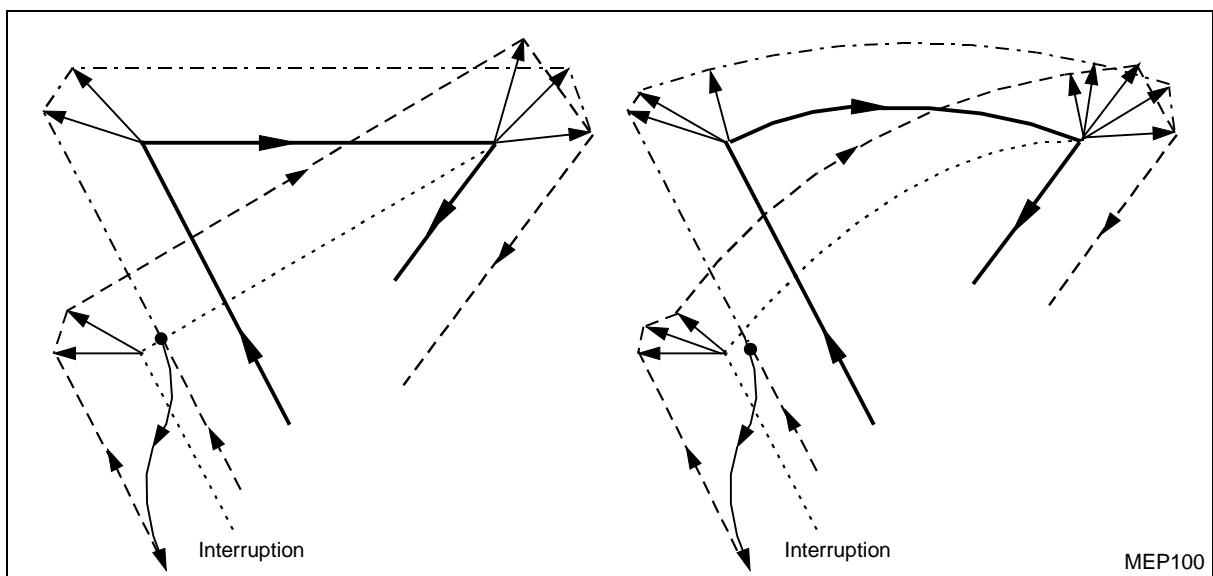
B. Interruption with movement

The offset vectors are recalculated automatically at the first effective move block after interruption.



2. Manual interruption

- For the incremental data command mode, the tool path shifts through the interruption amount.
- For the absolute data command mode, the intended tool path is restored at the ending point of the block immediately following that at which interruption has been performed. This state is shown in the diagram below.



12-5-6 General precautions on tool radius compensation

1. Selecting the amounts of offset

The amounts of offset are selected by specifying an offset number using a D-code. Once a D-code has been used, it will remain valid until a second D-code is used. No H-codes can be used to make these selections.

D-codes are also used to select tool position offset data.

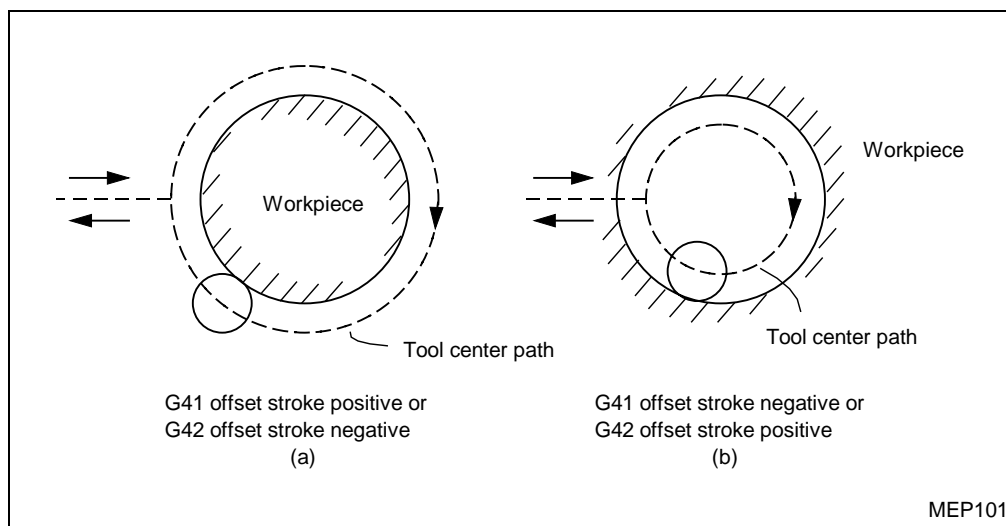
2. Updating the selected amounts of offset

Updating of the selected amounts of offset is usually to be done after a different tool has been selected during the radius compensation cancellation mode. If such updating is done during the compensation mode, vectors at the ending point of a block will be calculated using the offset data selected for that block.

3. The sign of offset data and the tool center path

Minus-signed (–) offset data generates the same figure as that obtained when G41 and G42 are exchanged each other. Therefore, the tool center will move around the inside of the workpiece if it has been moving around the outside. Conversely, the tool center will move around the outside of the workpiece if it has been moving around the inside.

Sample programs are shown below. Usually, offset data is to be programmed as plus (+) data. If the tool center has been programmed to move as shown in diagram (a) below, the movement can be changed as shown in diagram (b) below by changing the sign of the offset data to minus (–). Conversely, if the tool center has been programmed to move as shown in diagram (b) below, the movement can be changed as shown in diagram (a) below by changing the sign of the offset data to plus (+). One tape for machining of both inside and outside shapes can be created in this way. Also, a dimensional tolerance between both shapes can be freely set by selecting appropriate offset data (however, Type A is to be used during the start of offsetting or during its cancellation).



12-5-7 Offset number updating during the offset mode

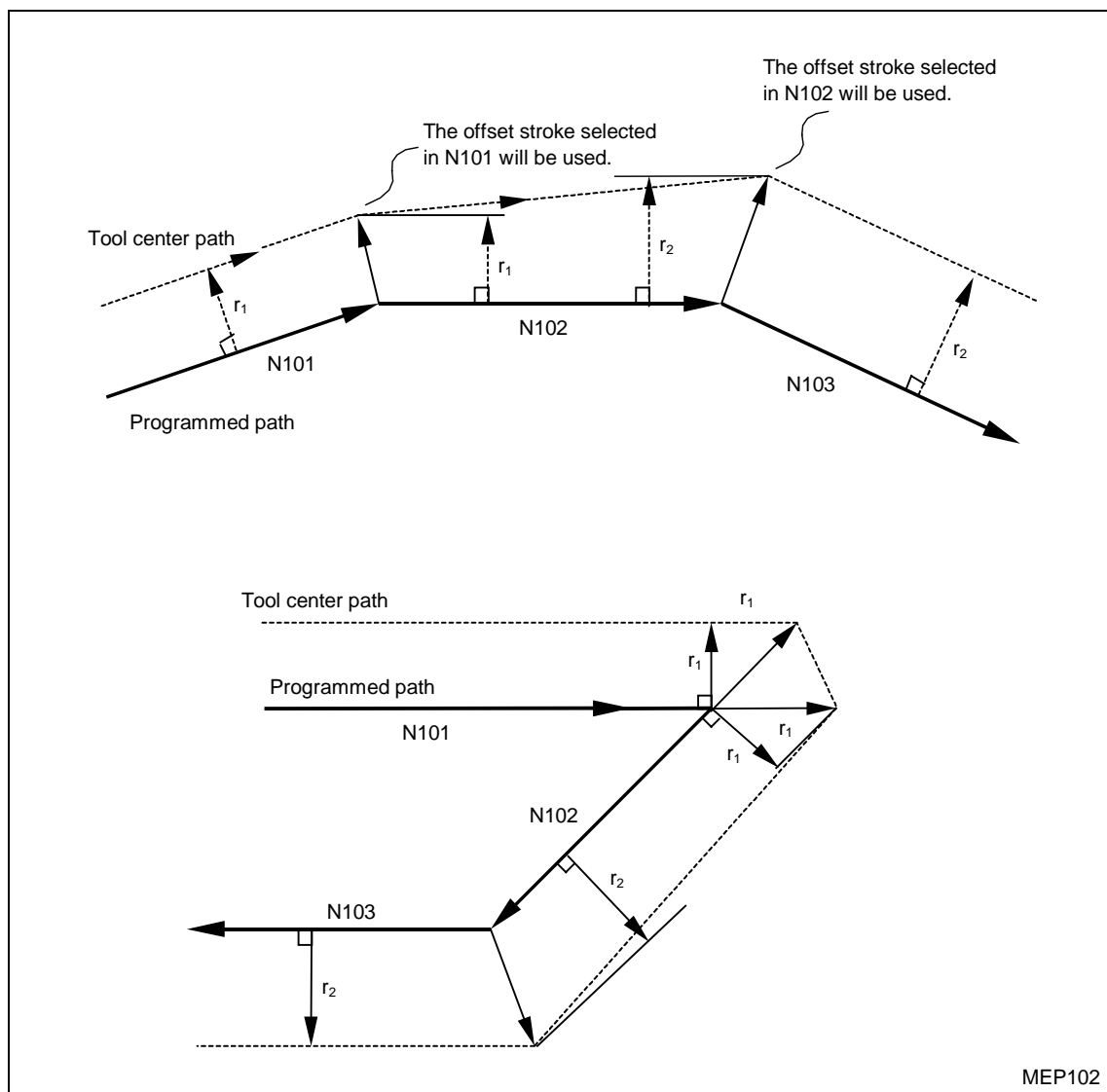
In principle, offset numbers should not be updated during the offset mode. If updating is done, the tool center will move as shown below.

If an offset number (offset data) is updated

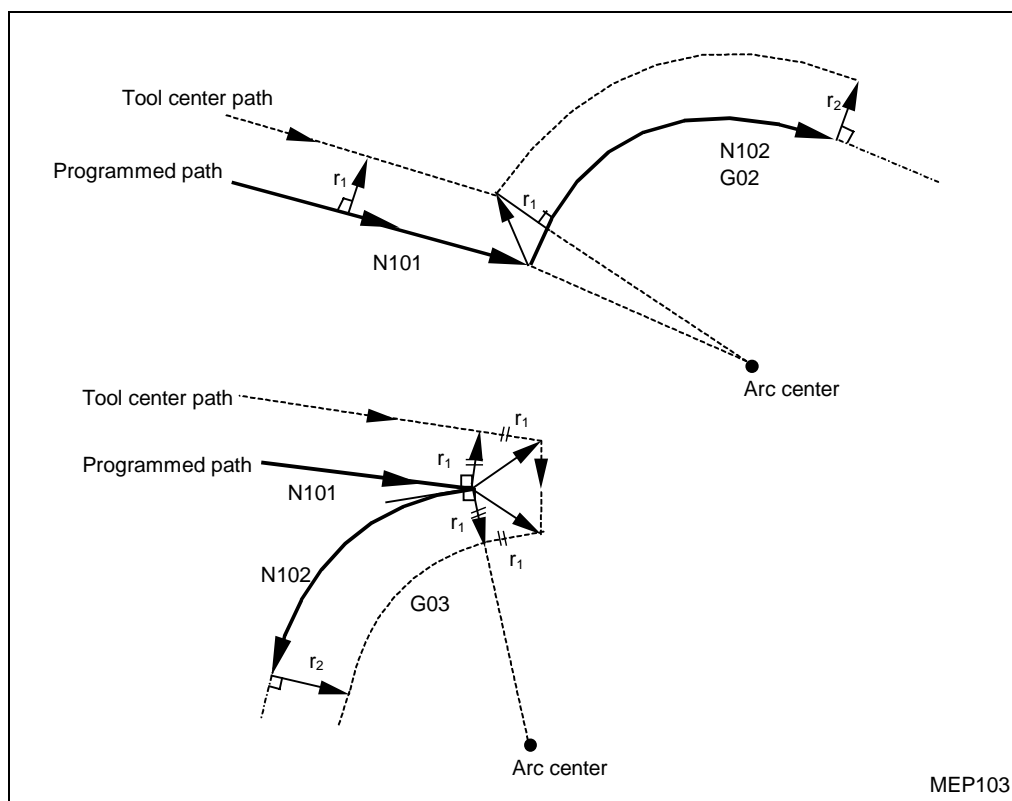
```

G41 G01      Dr1
:
:
:
α = 0, 1, 2, 3
N101 G0α Xx1 Yy1
N102 G0α Xx2 Yy2 Dr2 To change an offset number
N103      Xx3 Yy3
  
```

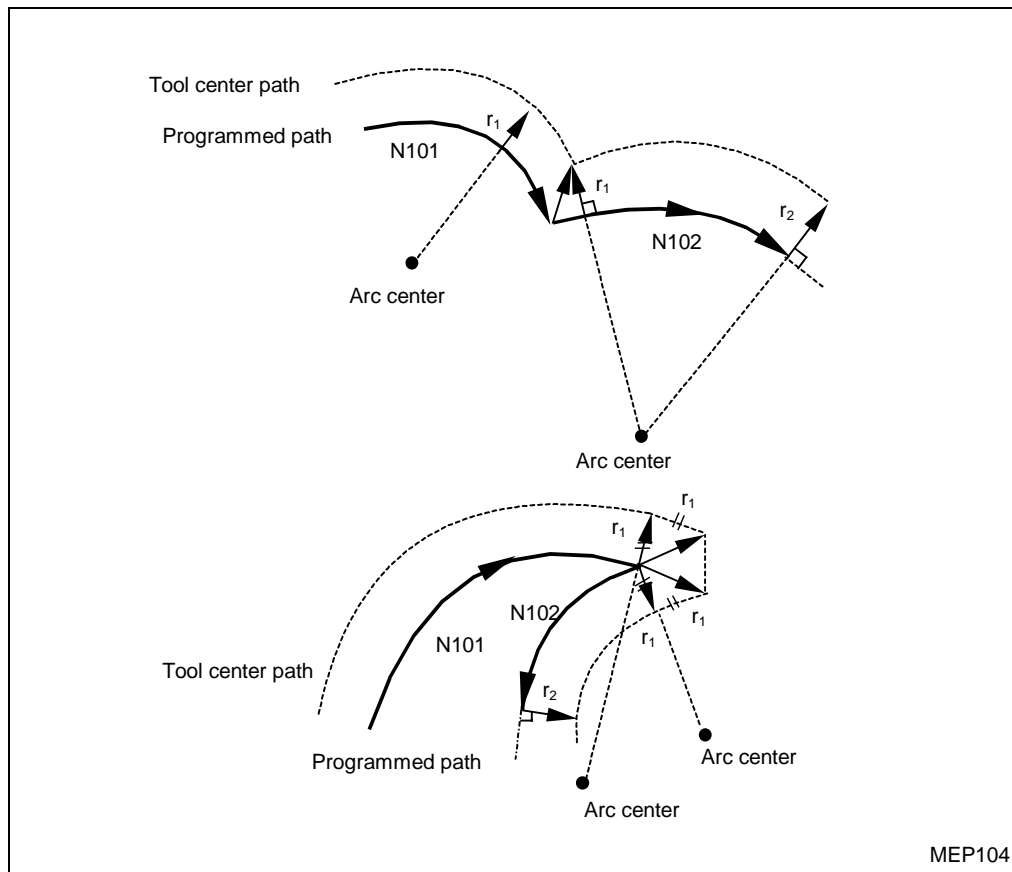
1. Line-to-line movement



2. Line-to-arc movement



3. Arc-to-arc movement

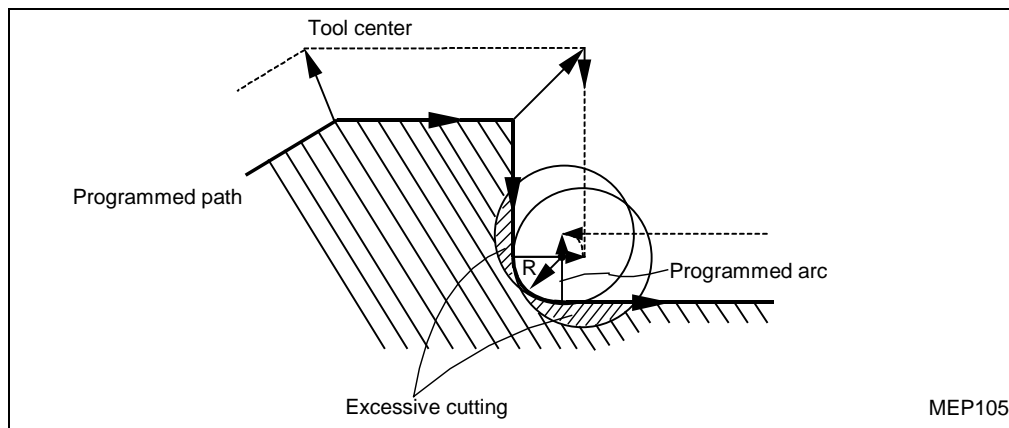


12-5-8 Excessive cutting due to tool radius compensation

If an interference check function is not provided, excessive cutting may result in the following three cases:

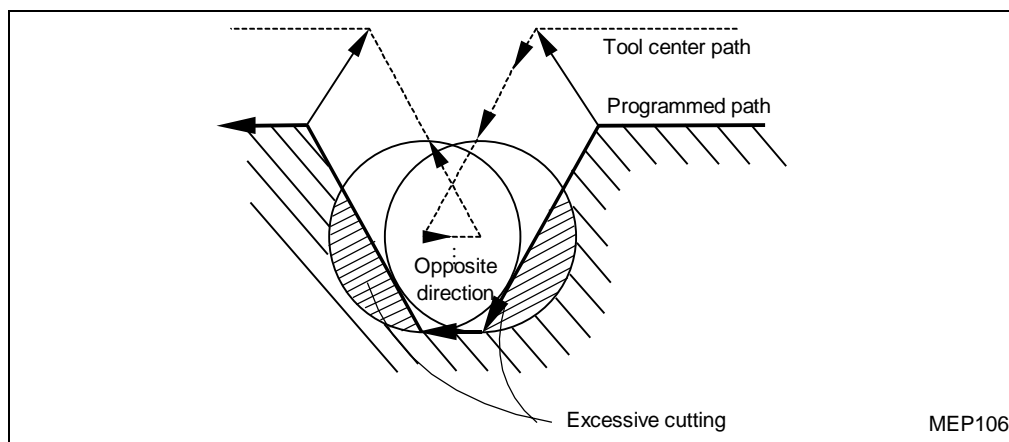
1. Machining of the inside of an arc smaller than the tool radius

If the radius of the programmed arc is smaller than that of the tool, excessive cutting may result from offsetting of the inside of the arc.

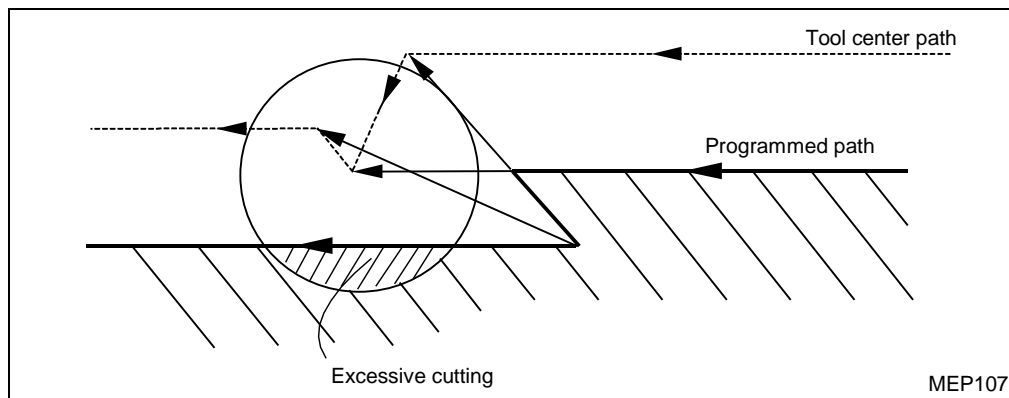


2. Machining of a groove smaller than the tool radius

Excessive cutting may result if tool radius compensation makes the moving direction of the tool center opposite to that of the program.



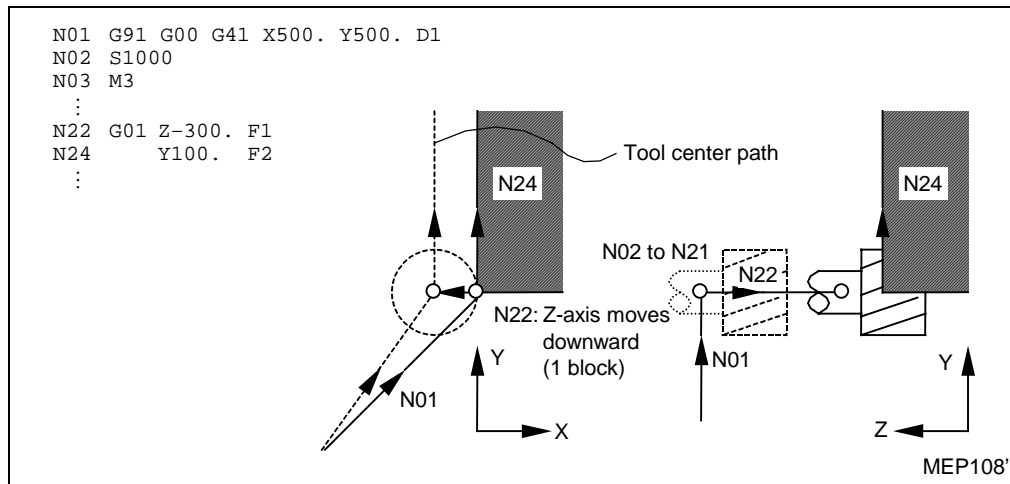
3. Machining of a stepped section smaller than the tool radius



4. Relationship between the start of tool radius compensation and the cutting operation in the Z-axis direction

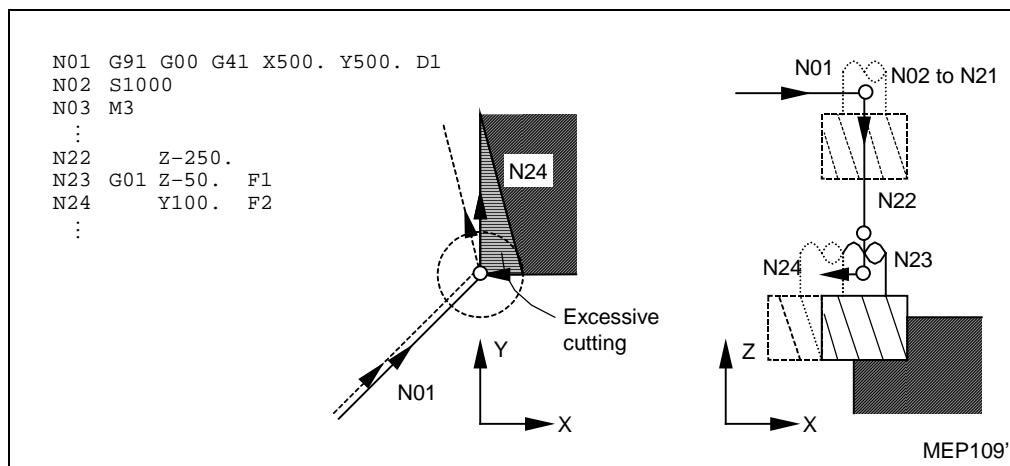
It is generally done that radius compensation (usually, on the XY-plane) is done at a suitable distance from the workpiece during the start of cutting and then the workpiece is cut along the Z-axis. At this time, incorporate the following programming considerations if you want to split the Z-axis action into rapid feed and cutting feed which is to follow only after the Z-axis has moved close to the workpiece:

If you make a program such as that shown below:



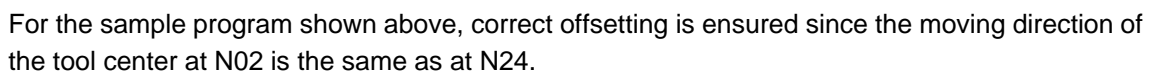
With this program, all blocks up to N24 can be read during the start of offsetting based on N01. Thus, the NC unit will judge the relationship between N01 and N24 and correctly perform the offset operation as shown in the diagram above.

A sample program in which the N22 block in the program shown above has been split into two parts is shown below.

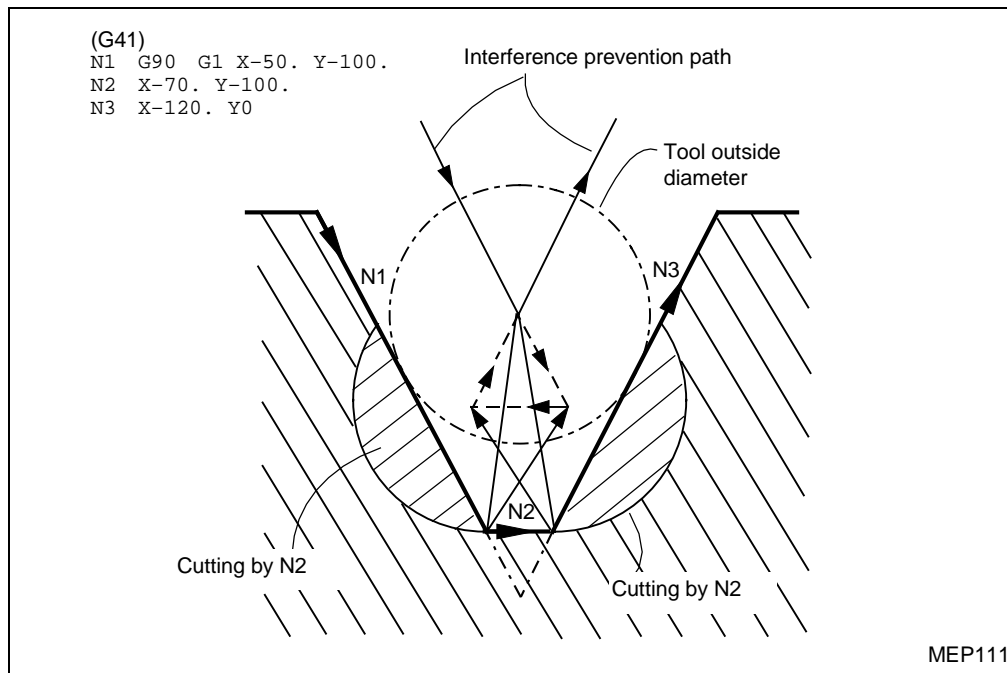


In this case, successive 22 blocks (N02 to N23) do not have any motion command corresponding to the XY-plane and the relevant block N24 cannot be read during the start of offsetting based on N01. As a result, offsetting will be based only on the information contained in the N01 block and thus the NC unit will not be able to create offset vectors during the start of offsetting. This will cause excessive cutting as shown in the diagram above.

Even in such a case, however, excessive cutting can be prevented if a command code that moves the tool in exactly the same direction as that existing after the Z-axis has moved downward is included immediately before the Z-direction cutting block.



Function	Parameter	Operation
Interference check and alarm	Interference check and prevention off	The system will stop, with an alarm resulting before executing the cutting block.
Interference check and prevention	Interference check and prevention on	The path is changed to prevent cutting from taking place.

Example:

- For the alarm function

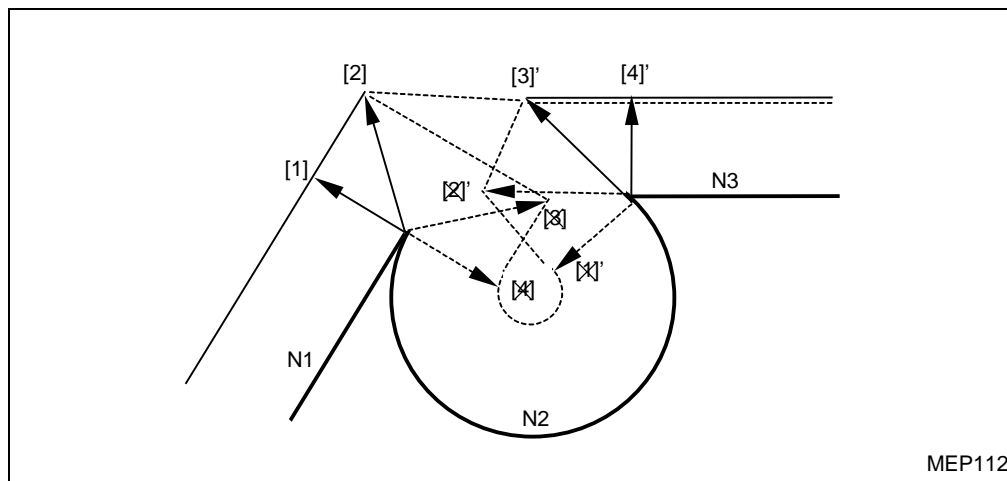
An alarm occurs before N1 is executed. Machining can therefore be proceeded with by updating the program into, for example,

N1 G90 G1 X-20. Y-40.

using the buffer correction function.

- For the prevention function

Interference prevention vectors are generated by N1 and N3 crossing-point calculation.



Vector [1] [4]' check → No interference



Vector [2] [3]' check → No interference



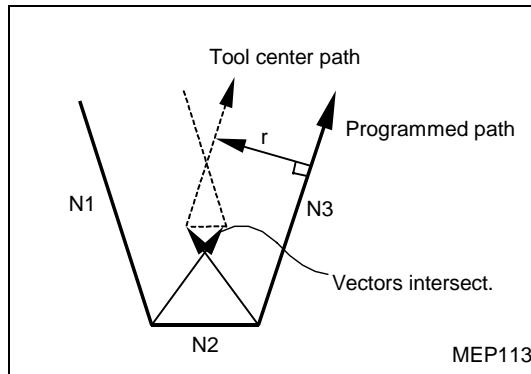
Vector [3] [2]' check → Interference → Vector [3] [2]' deletion → Vector [4] [1]' deletion

The above process is performed to leave vectors [1] [2] [3]' and [4]' as effective ones. Resultantly, the route that connects vectors [1] [2] [3]' and [4]' is taken as a bypass for the prevention of interference.

2. Detailed description

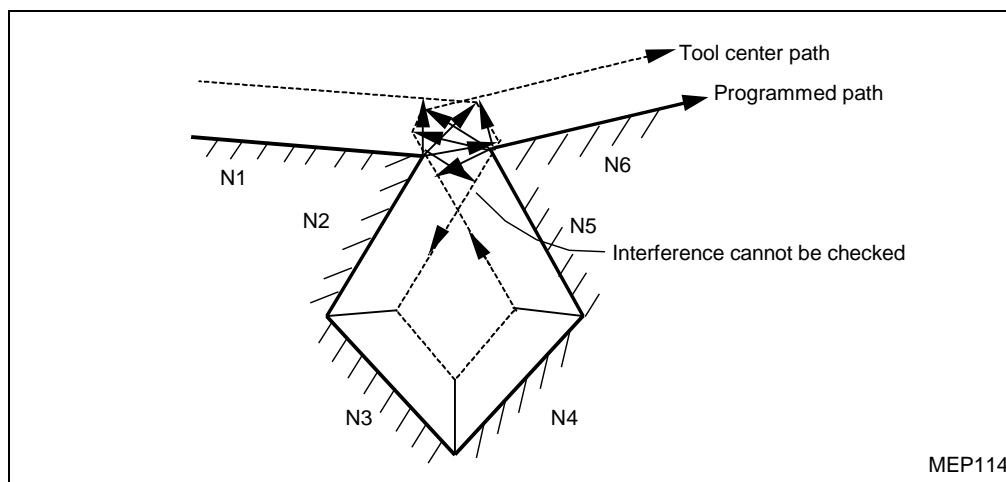
A. The case where interference is regarded as occurring

When move commands are present in three of the 23 command blocks to be pre-read, interference will be regarded as occurring, if the offset calculation vectors at the block connections of the individual move commands intersect.



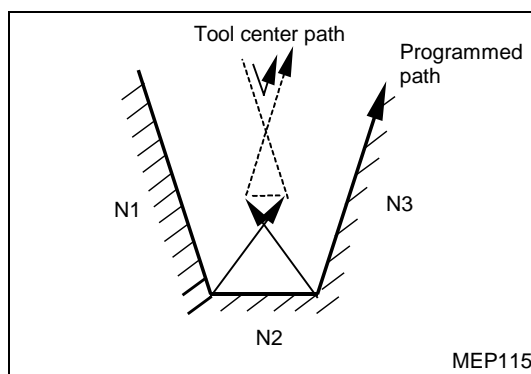
B. Cases where interference check cannot be performed

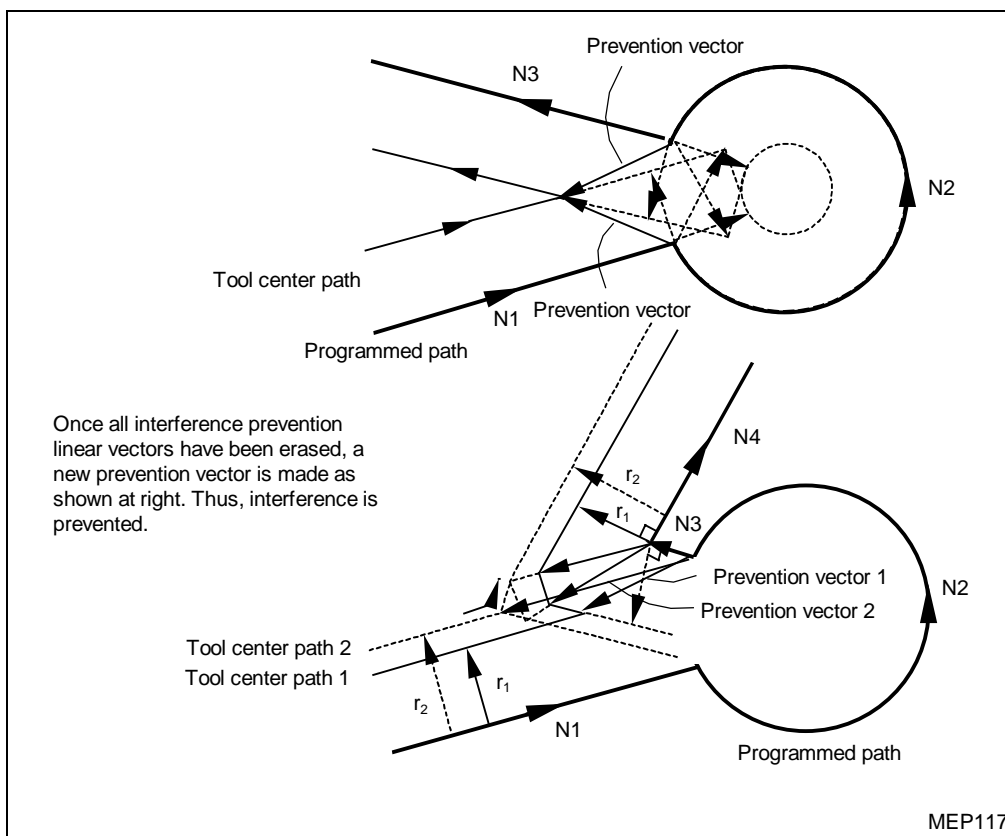
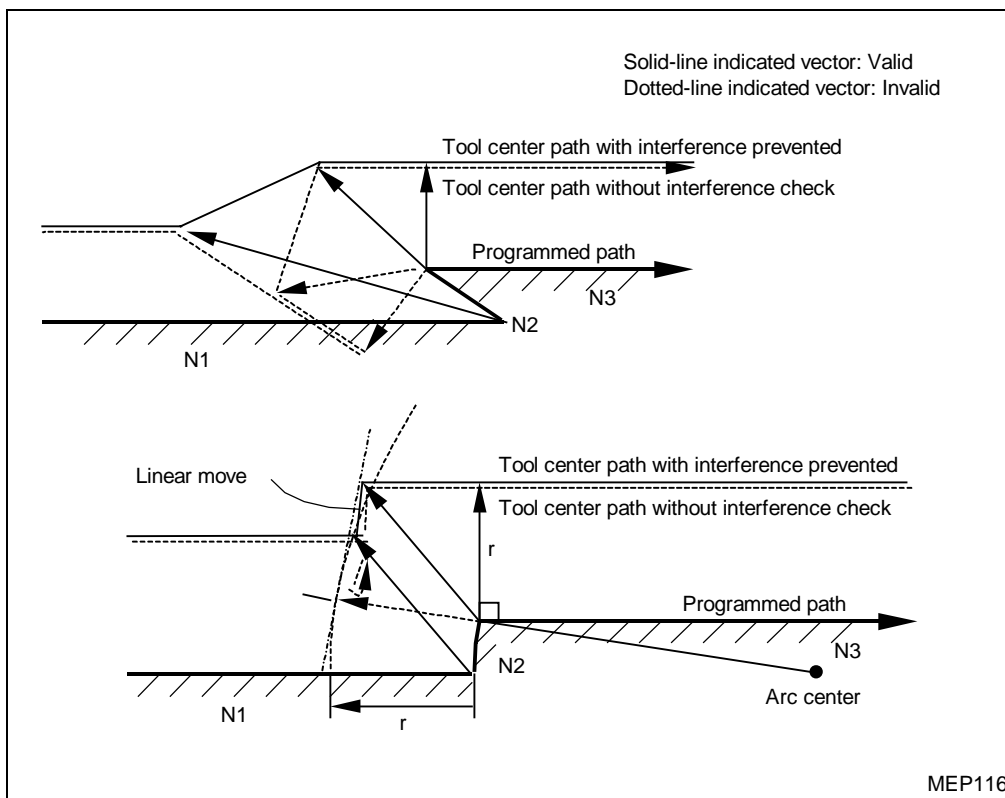
- When pre-reading of three move command blocks within 23 blocks to be pre-read is not possible (since 21 or more blocks do not contain move commands).
- When the fourth and subsequent move command blocks themselves interfere.



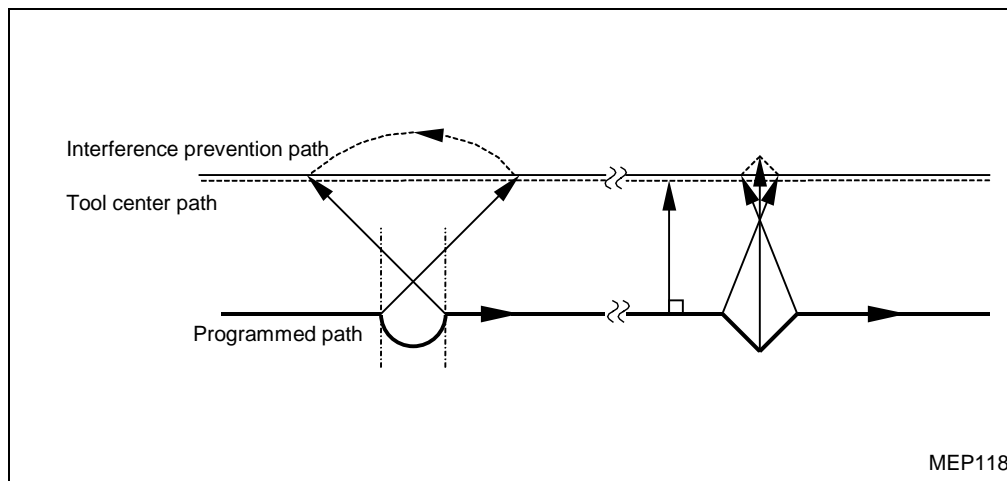
C. Movements during the prevention of interference

The following shows the movements occurring when interference prevention is provided:





In the diagram shown below, part of the groove is left uncut:

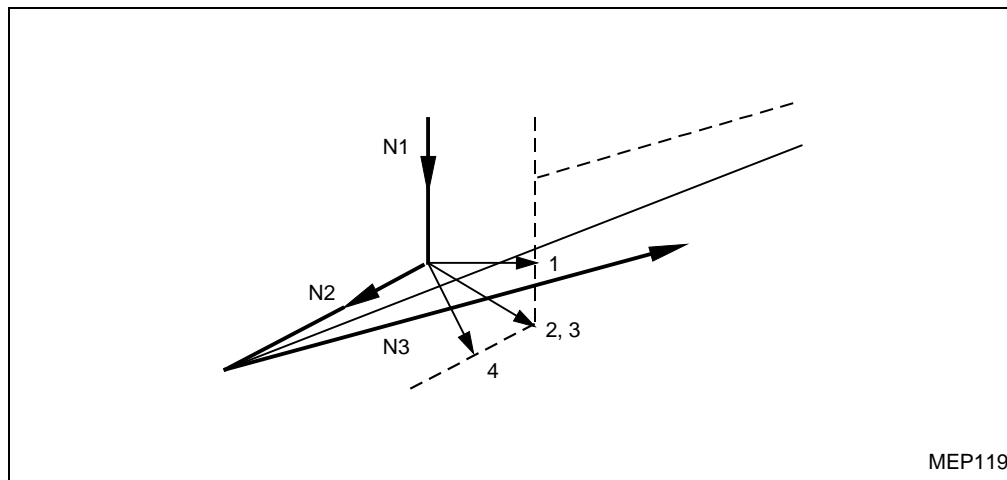


3. Interference alarm

Cases that an interference alarm **837 TOOL OFFSET INTERFERENCE ERROR** occurs are listed below.

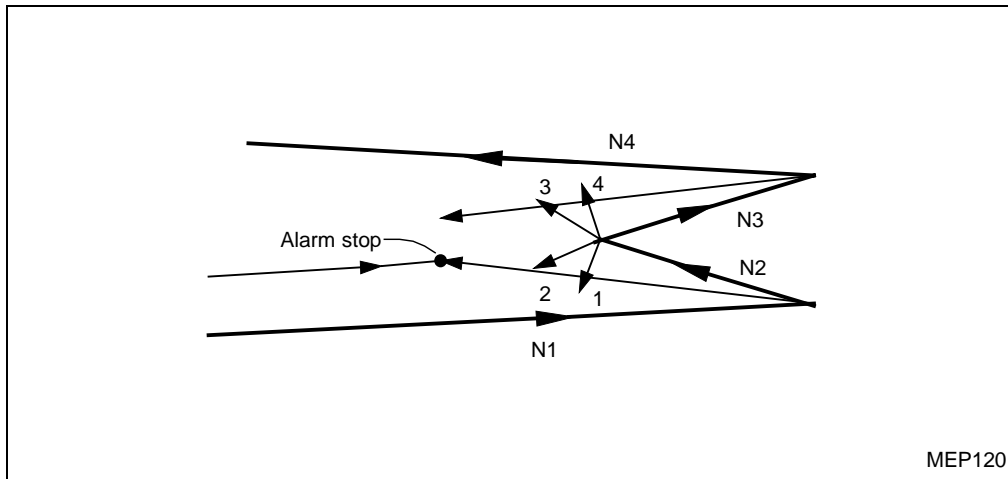
When interference check and alarm is selected

- 1) If all vectors at the ending point of the current block are erased:
Prior to execution of N1, an alarm will result if vectors 1 through 4 at the ending point of the N1 block are all erased as shown in the diagram below.

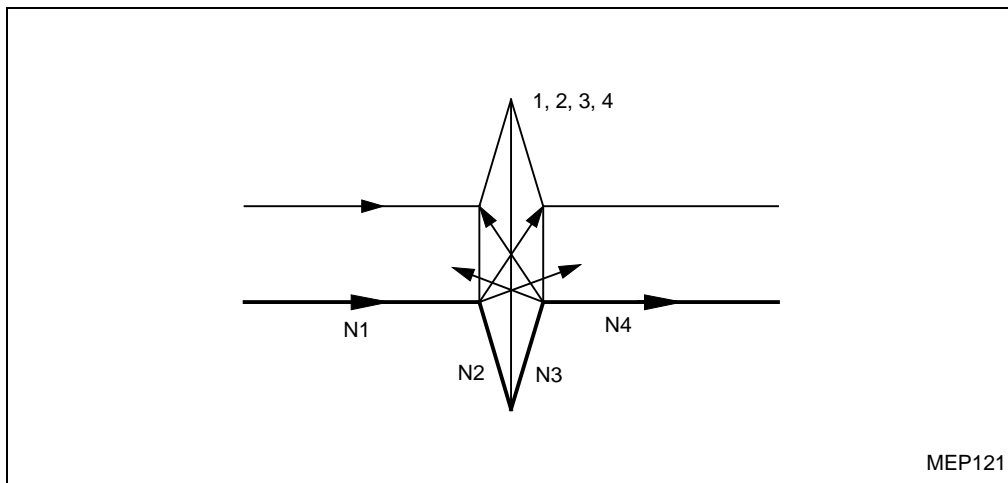


When interference check and prevention is selected

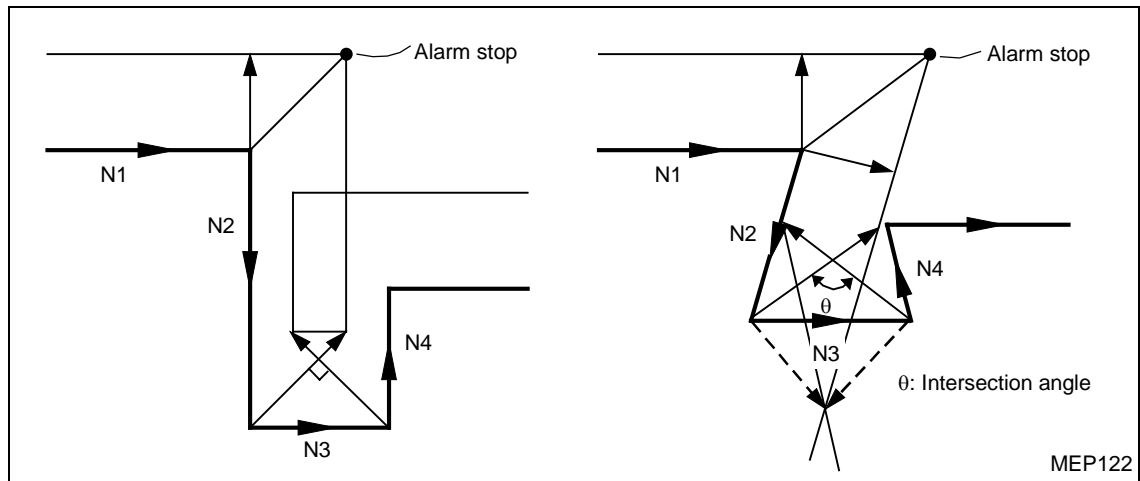
- 2) If all vectors at the ending point of the current block are erased but an effective vector(s) remains at the ending point of the next block:
- For the diagram shown below, interference checking at N2 will erase all vectors existing at the ending point of N2, but leave the vectors at the ending point of N3 effective. At this time, an alarm will occur at the ending point of N1.



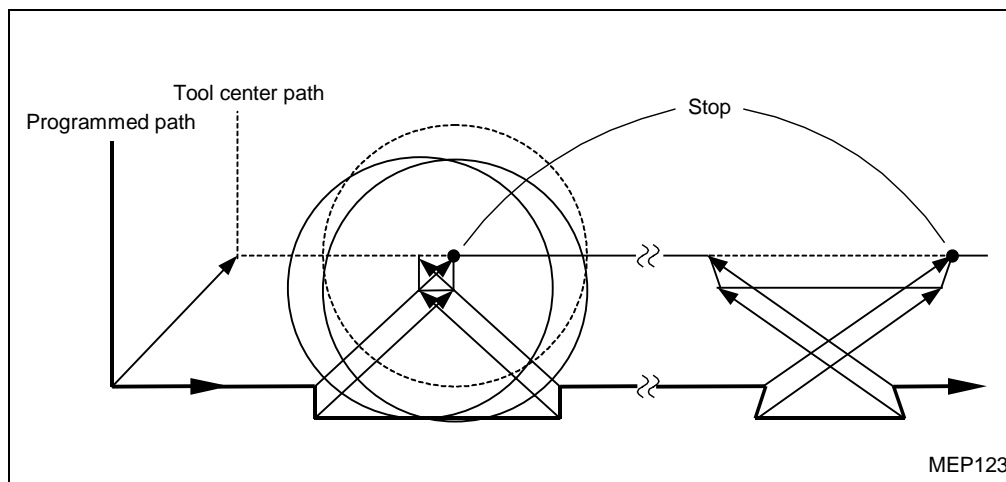
- For the diagram shown below, the direction of movement becomes opposite at N2. At this time, an alarm will occur before execution of N1.



- 3) When prevention vectors cannot be generated:
Prevention vectors may not be generated even when the conditions for generating them are satisfied. Or even after generation, the prevention vectors may interfere with N3.
An alarm will therefore occur at the ending point of N1 if those vectors cross at angles of 90 degrees or more.



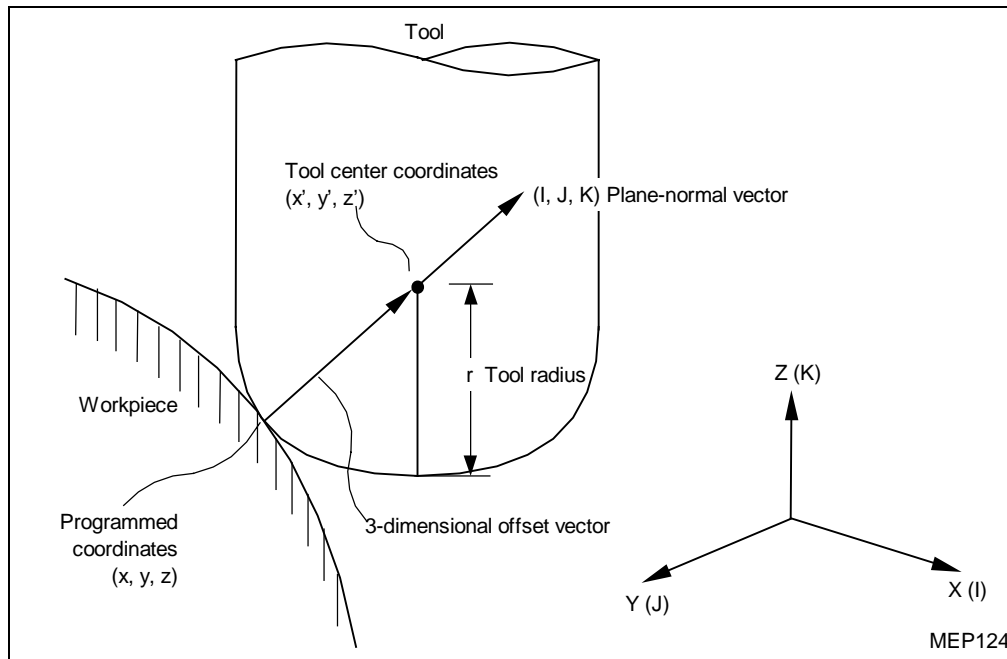
- 4) When the after-offsetting moving direction of the tool is opposite to that of the program:
For a program for the machining of parallel or downwardly extending grooves narrower than the tool diameter, interference may be regarded as occurring even if it is not actually occurring.



12-6 Three-Dimensional Tool Radius Compensation (Option)

Three-dimensional tool radius compensation is performed to offset a tool in three-dimensional space according to the previously designated three-dimensional vectors.

12-6-1 Function description



As shown in the diagram above, the tool is moved through the tool radius r in the plane-normal vectorial direction of (I, J, K) from the program coordinates (x, y, z) to the offset tool center coordinates (x', y', z') . Also, unlike two-dimensional tool radius compensation, which generates vectors perpendicular to the direction of (I, J, K) , three-dimensional tool radius compensation generates vectors in the direction of (I, J, K) . (The vectors are generated at the ending point of that block.) The axis components of three-dimensional offset vectors become:

$$H_x = \frac{I}{\sqrt{I^2 + J^2 + K^2}} \cdot r$$

$$H_y = \frac{J}{\sqrt{I^2 + J^2 + K^2}} \cdot r$$

$$H_z = \frac{K}{\sqrt{I^2 + J^2 + K^2}} \cdot r$$

Hence, the tool center coordinates (x', y', z') are expressed as

$$x' = x + H_x$$

$$y' = y + H_y$$

$$z' = z + H_z$$

where (x, y, z) denote the programmed coordinates.

Note 1: The three-dimensional vectors (H_x, H_y, H_z) refer to plane-normal vectors that are identical to the plane-normal vectors (I, J, K) in direction and have a magnitude of r (tool radius).

Note 2: If parameter **F11** is set to a value other than 0, the value of **F11** will be used as $\sqrt{I^2 + J^2 + K^2}$.

12-6-2 Programming methods

1. G-codes and their functions

G-code	Parameter and feature		
	Offset stroke positive	Offset stroke negative	Offset No. D00
G40	To cancel the 3-dimensional tool radius compensation	To cancel	To cancel
G41	To compensate in (I, J, K) direction	To compensate in the direction opposite to (I, J, K)	To cancel
G42	To compensate in the direction opposite to (I, J, K)	To compensate in (I, J, K) direction	To cancel

2. Offset data

For the tool radius r that is to be offset, the offset number under which that offset amount has been registered must be selected using D.

3. Space in which offsetting is to be performed

The space in which offsetting is to be performed is determined by the axis address commands (X, Y, Z, U, V, W) that are contained in the starting block of three-dimensional tool radius compensation. When the U-, V-, and W-axes are taken as additions to the X-, Y-, and Z-axes, respectively, priority will be given to the X-, Y-, or Z axis if the X-axis and the U-axis (or Y and V, or Z and W) are selected at the same time. Coordinate axes that have not been addressed will be interpreted as the X-axis, the Y-axis, and the Z-axis, respectively.

Example:

G41	X _{x1} Y _{y1} Z _{z1} I _{i1} J _{j1} K _{k1}	XYZ space
G41	Y _{y2} I _{i2} J _{j2} K _{k2}	XYZ space
G41	X _{x3} V _{v3} Z _{z3} I _{i3} K _{k3}	XVZ space
G41	W _{w4} I _{i4} J _{j4} K _{k4}	XYW space

4. Starting a three-dimensional tool radius compensation operation

Offset number D and the plane-normal vectors (I, J, K) must be set in the same block as that which contains three-dimensional tool radius compensation command code G41 (or G42). In that case, (I, J, K) must be set for each of the X-, Y-, and Z-axes. If this vector setting is not complete (setting of zero for I, J or K is effective), the usual tool radius compensation mode will be set. If, however, the machine does not have the three-dimensional tool radius compensation function, an alarm **838 3-D OFFSET OPTION NOT FOUND** will result.

G41 (G42) X_{x1} Y_{y1} Z_{z1} I_{i1} J_{j1} K_{k1} D_{d1}

G41 (G42) : 3-dimensional tool radius compensation command

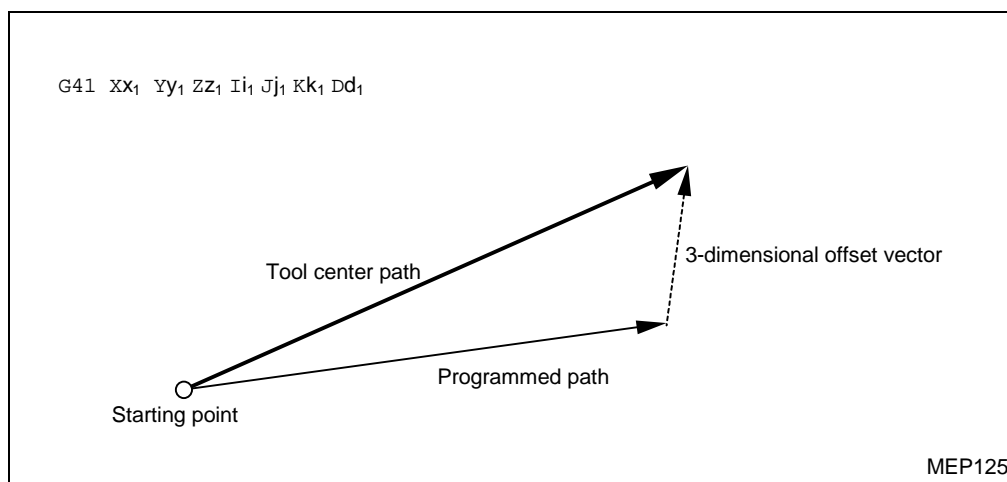
X, Y, Z : Command to move each axis and to determine an offsetting space

I, J, K : To indicate the offsetting direction in plane-normal vectors

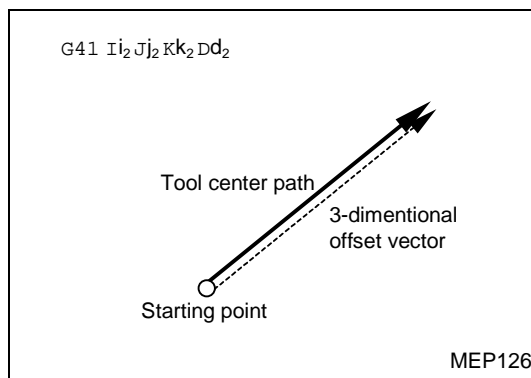
D : Offset number

Use the G00 or G01 mode to start the three-dimensional tool radius compensation operation. Use of the G02 or G03 mode results in an alarm **835 G41, G42 FORMAT ERROR**.

Example 1: If move commands are present:



Example 2: If move commands are not present:

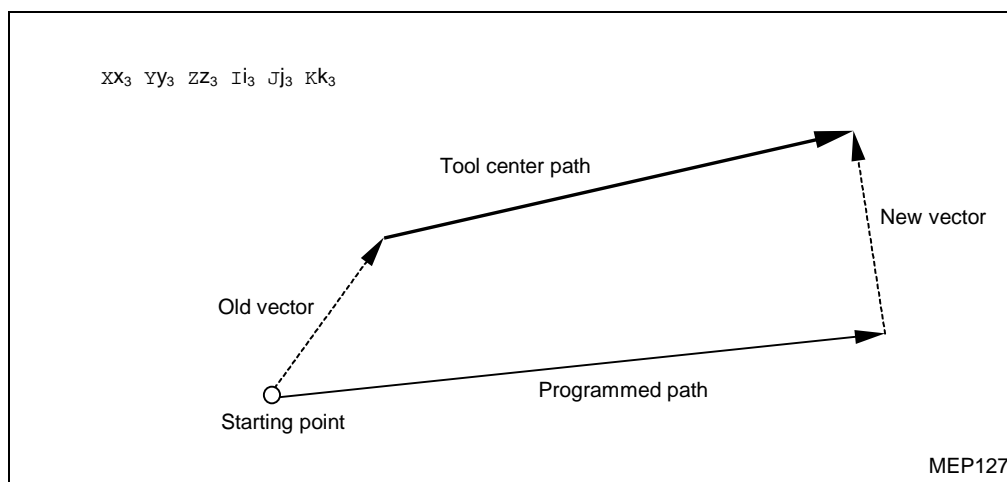


5. During three-dimensional tool radius compensation

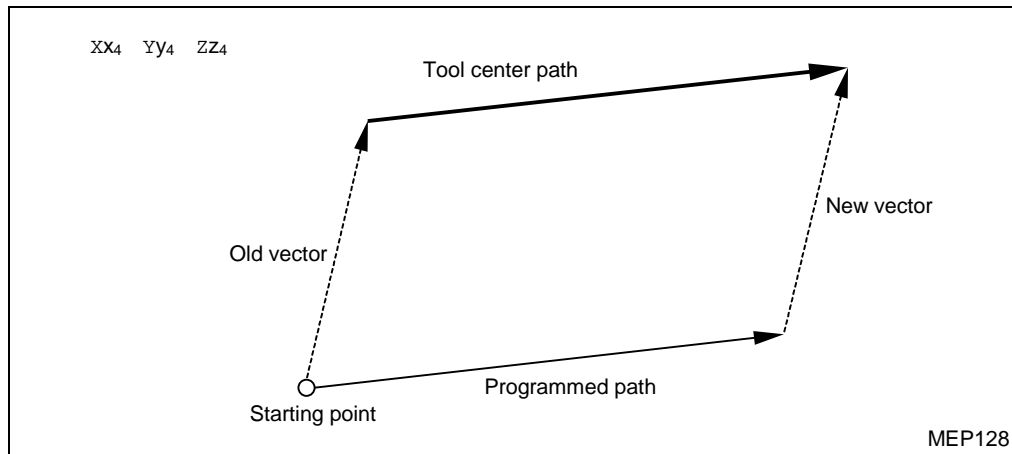
Set move commands and new plane-normal vector commands as follows:

XX_3 YY_3 ZZ_3 Ii_3 Jj_3 Kk_3

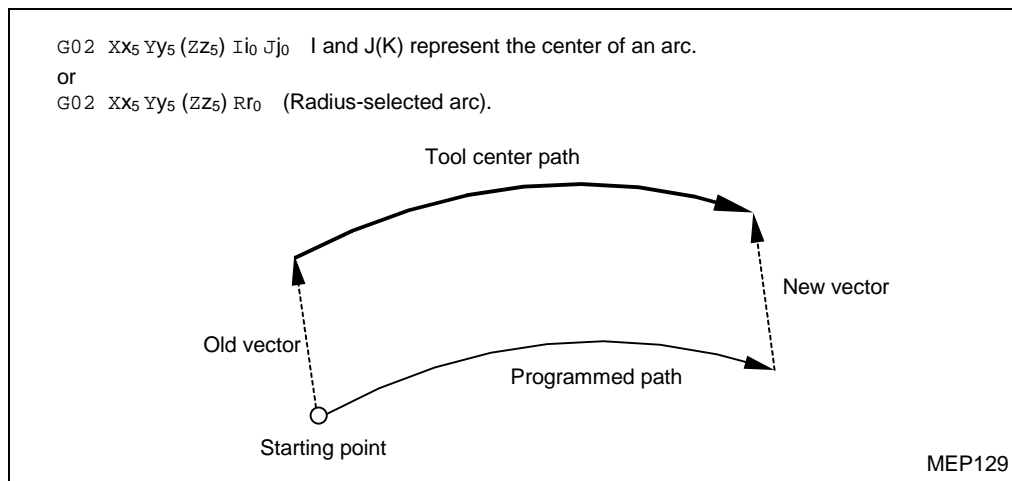
Example 1: If move commands and plane-normal vector commands are present:



Example 2: If plane-normal vector commands are not present:
The new vector is the same as the old one.

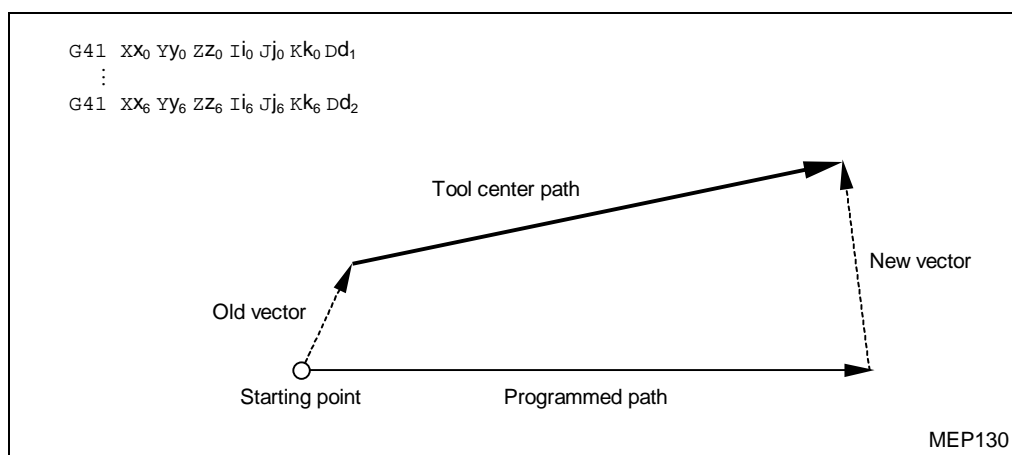


Example 3: For arc or helical cutting:
The new vector is the same as the old one.

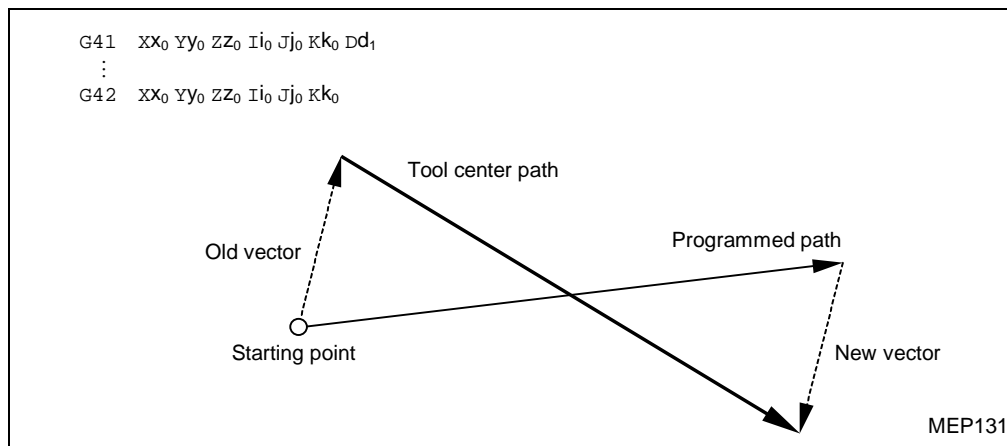


The arc shifts through the amount of vector.

Example 4: For changing the offset data:
Set offset number D in the same block as that of three-dimensional tool radius compensation command G41 or G42. Use the G00 or G01 mode to change the offset data. Use of the arc mode results in an alarm (**835 G41, G42 FORMAT ERROR**).



Example 5: For changing the offset direction:



Use the G00 or G01 mode to change the offset direction. Use of the arc mode results in an alarm (**835 G41, G42 FORMAT ERROR**).

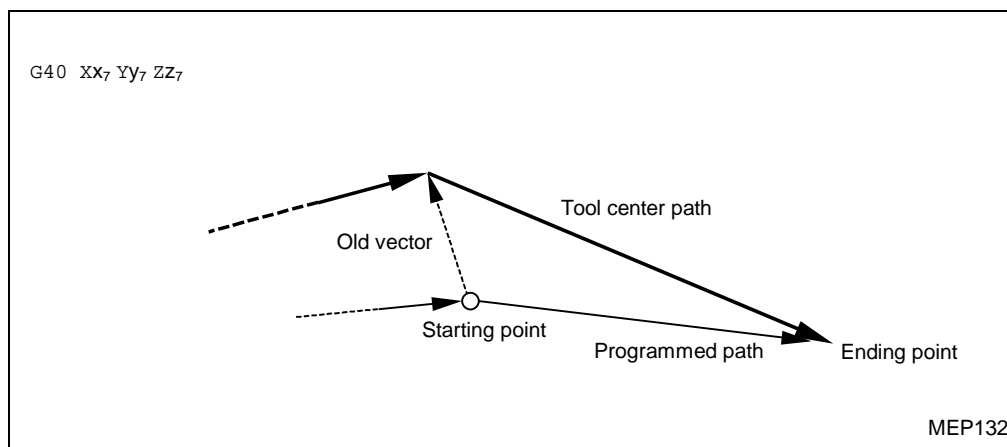
6. Cancelling the three-dimensional tool radius compensation operation

Make the program as follows:

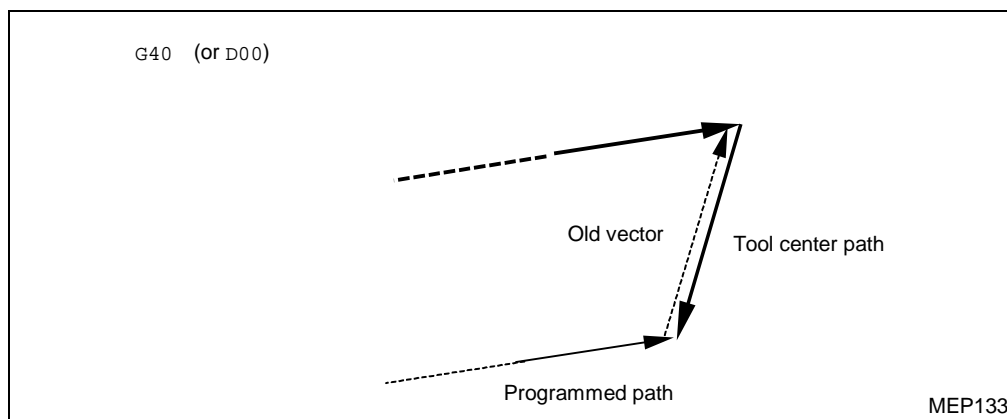
```
G40  Xx7  Yy7  Zz7
```

Use the G00 or G01 mode to cancel three-dimensional tool radius compensation. Use of the G02 or G03 mode results in an alarm (**835 G41, G42 FORMAT ERROR**).

Example 1: If move commands are present:



Example 2: If move commands are not present:



12-6-3 Correlationships to other functions

1. Tool radius compensation
The usual tool radius compensation mode will be selected if setting of plane-normal vectors (I, J, K) in the starting block of three-dimensional tool radius compensation is not done for each of the X-, Y-, and Z-axes.
2. Tool length offset
Tool length offsetting is performed according to the coordinates existing after execution of three-dimensional tool radius compensation.
3. Tool position offset
Tool position offsetting is performed according to the coordinates existing after execution of three-dimensional tool radius compensation.
4. Selection of fixed-cycle operation results in an alarm (**901 INCORRECT FIXED CYCLE COMMAND**).
5. Scaling
Three-dimensional tool radius compensation is performed according to the coordinates existing after execution of scaling.
6. Home position check (G27)
The current offset data is not cancelled.

12-6-4 Miscellaneous notes on three-dimensional tool radius compensation

1. Although they can be used to select offset numbers, D-code commands are valid only after command G41 or G42 has been set. If a D-code command is not present, the previous D-code command becomes valid.
2. Use the G00 or G01 mode to change the offset mode, the offset direction or the offset data. An alarm (**835 G41, G42 FORMAT ERROR**) will occur if an attempt is made to perform these changes in an arc mode.
3. During the three-dimensional tool radius compensation mode using a space, three-dimensional tool radius compensation cannot be done using any other space. The cancel command code (G40 or D00) must be executed to select some other offset space.

Example:

G41 X_ Y_ Z_ I_ J_ K_ To start offsetting in XYZ space
:

G41 U_ Y_ Z_ I_ J_ K_ To offset in XYZ space while a U-axis movement occurs as specified

4. Selection of an offset number falling outside the range from 1 to the maximum available number results in an alarm (**839 ILLEGAL OFFSET No.**).
5. Only the G40 or D00 command code can be used to cancel three-dimensional tool radius compensation. Cancellation is not possible with the NC reset key or external reset functions.
6. An alarm will result if the vectorial magnitude specified by (I, J, K), that is $\sqrt{I^2 + J^2 + K^2}$, overflows.

12-7 Programmed Data Setting: G10

1. Function and purpose

The G10 command allows tool offset data, workpiece offset data and parameter data to be set or modified in the flow of program.

2. Programming formats

A. Programming workpiece offsets

- Programming format for the workpiece origin data

G10 L2 P_ Xp_ Yp_ Zp_α_ (α: Additional axis)

P0: Coordinate shift (Added feature)

P1: G54

P4: G57

P2: G55

P5: G58

P3: G56

P6: G59

Data of P-commands other than those listed above are handled as P = 1.

If P-command setting is omitted, the workpiece offsets will be handled as currently effective ones.

- Programming format for the additional workpiece origin data

G10 L20 P_ Xp_ Yp_ Zp_α_ (α: Additional axis)

P1: G54.1 P1

P2: G54.1 P2

⋮

P299: G54.1 P299

P300: G54.1 P300

The setting ranges of the data at axial addresses are as follows:

	Metric	Inch
Linear axis	±99999.9999 mm	±9999.99999 in
Rotational axis	±99999.9999°	±99999.9999°

B. Programming tool offsets

- Programming format for the tool offset data of Type A

G10 L10 P_R_

P: Offset number

R: Offset amount

- Programming format for the tool offset data of Type B

G10 L10 P_R_ Geometric offset concerning the length

G10 L11 P_R_ Wear compensation concerning the length

G10 L12 P_R_ Geometric offset concerning the diameter

G10 L13 P_R_ Wear compensation concerning the diameter

The setting ranges for programming tool offset amount (R) are as follows:

	Metric	Inch
TOOL OFFSET Type A	±1999.9999 mm	±84.50000 in
TOOL OFFSET Type B Length Geom.	±1999.9999 mm	±84.50000 in
TOOL OFFSET Type B Length Wear	±99.9999 mm	±9.99999 in
TOOL OFFSET Type B Dia. Geom.	±999.9999 mm	±84.50000 in
TOOL OFFSET Type B Dia. Wear	±9.9999 mm	±0.99999 in

C. Programming parameter data

G10 L50..... Parameter input mode ON

N_P_R_

N_R_

G11 Parameter input mode OFF

N: Parameter number

P: Axis number (for axis type parameter)

R: Data of parameter

Specify the parameters with address N as indicated below:

	Parameter	N: Number	P: Axis No.
A	1 to 200	1001 to 1200	—
B	1 to 200	2001 to 2200	—
C	1 to 200	3001 to 3200	—
D	1 to 144	4001 to 4144	—
E	1 to 144	5001 to 5144	—
F	1 to 168 (47 to 66 excluded)	6001 to 6168	—
I	1 to 24	9001 to 9024	1 to 16
J	1 to 144	10001 to 10144	—
K	1 to 144	11001 to 11144	—
L	1 to 144	12001 to 12144	—
M	1 to 48	13001 to 13048	1 to 16
N	1 to 48	14001 to 14048	1 to 16
P	1 to 5	150001 to 150005	1 to 16
S	1 to 48	16001 to 16048	1 to 16
SV	1 to 384	17001 to 17384	1 to 16
SP	1 to 256	18001 to 18256	1 to 8
SA	1 to 144	19001 to 19144	1 to 8
BA	1 to 132	20001 to 20132	—
TC	1 to 154	21001 to 21154	—
SU	1 to 168	22001 to 22168	—
SD	1 to 168	23001 to 23168	—

Note: As for the setting ranges of parameter data, refer to the separate Parameter List/Alarm List/M-code List.

3. Detailed description

A. Workpiece origin data input

1. The G10 command is not associated with movement. However, do not use this command in the same block with a G-code command other than: G21, G22, G54 to G59, G90 and G91.
2. Do not use the G10 command in the same block with a fixed cycle command or a sub-program call command. This will cause a malfunctioning or an alarm.
3. Irrespective of workpiece offset type (G54 - G59 and G54.1), the data to the axial addresses have to refer to the origin of the fundamental machine coordinate system.
4. Depending upon the data input mode — absolute (G90) or incremental (G91) — the designated data will overwrite, or will be added to, the existing data.
5. L-code and P-code commands can be omitted, indeed, but take notice of the following when omitting them:
 - 1) Omit both L-code and P-code commands only when
The axial data should refer to the coordinate system that was last selected.
 - 2) The L-code command only may be omitted when the intended axial data refer to a coordinate system of the same type (in terms of L-code: L2 or L20) as the last selected one; give a P-command in such a case as follows:
 - Set an integer from 0 to 6 with address P to specify the coordinate shift data or one of the coordinate systems from G54 to G59.
 - Set an integer from 1 to 300 with address P to specify one of the additional workpiece coordinate systems of G54.1.
 - 3) If the P-code command only is omitted:
An alarm will result if the value of L mismatches the coordinate system last selected.
6. Axial data without a decimal point can be entered in the range from -99999999 to +99999999. The data settings at that time depend upon the data input unit.

Example: G10 L2 P1 X-100. Y-1000 Z-100 B-1000

The above command sets the following data:

Metric system	X -100.	Y -1.	Z -0.1	B -1.
Metric system (up to 4 dec. places)	X -100.	Y -0.1	Z -0.01	B -0.1
Inch system	X -100.	Y -0.1	Z -0.01	B -1.
Inch system (up to 5 dec. places)	X -100.	Y -0.01	Z -0.001	B -0.1

7. The origin data updated by a G10 command are not indicated as they are on the **WORK OFFSET** display until that display has been selected anew.
8. Setting an illegal L-code value causes an alarm.
9. Setting an illegal P-code value causes an alarm.
10. Setting an illegal axial value causes an alarm.
11. The G10 command is invalid (or skipped) during tool path check.

B. Tool offset data input

1. The G10 command is not associated with movement. However, do not use this command in the same block with a G-code command other than: G21, G22, G54 to G59, G90 and G91.
2. Do not use the G10 command in the same block with a fixed cycle command or a sub-program call command. This will cause a malfunctioning or an alarm.

- Example:** G10 L10 P1 R1000

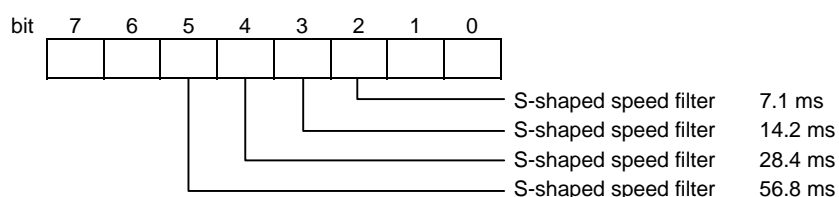
Metric system	1.
Metric system (up to 4 dec. places)	0.1
Inch system	0.1
Inch system (up to 5 dec. places)	0.01

- ### C. Parameter data input

- Example:** For changing a bit type data of 00110110 to 00110111:

6. All decimal places, even if inputted, are ignored.
7. Some specific bit-type parameters require selection of one of multiple bits. For the parameter shown as an example below, set data that turns on only one of bits 2 to 5.

Example: Parameter K107



12-59

8. The parameter data updated by a G10 L50 command are not made valid till the execution of a G11 command.
9. The parameter data updated by a G10 L50 command are not indicated as they are on the **PARAMETER** display until that display has been selected anew.
10. Setting an illegal L-code value causes an alarm.
11. Setting an illegal N-code value (parameter No.) causes an alarm.
12. Omission of P-code for an axis type parameter causes an alarm.
13. Setting an illegal parameter value with address R causes an alarm.
14. The G10 command is invalid (or skipped) during tool path check.
15. As for parameters (**BA** and **SU**) with separate values for each system, a G10 command is only effective for the values of the system to which the current program section belongs.

4. Sample programs

A. Entering tool offset data from tape

```
... G10L10P10R-12345 G10L10P05R98765 G10L10P40R2468 ...
```

H10 = -12345 H05 = 98765 H40 = 2468

B. Updating tool offset data

Example 1: Assumes that H10 has already been set equal to -1000.

```
N1 G01 G90 G43 Z-100000 H10 (Z = -101000)
N2 G28 Z0
N3 G91 G10 L10 P10 R-500 (-500 is added in the G91 mode.)
N4 G01 G90 G43 Z-100000 H10 (Z = -101500)
```

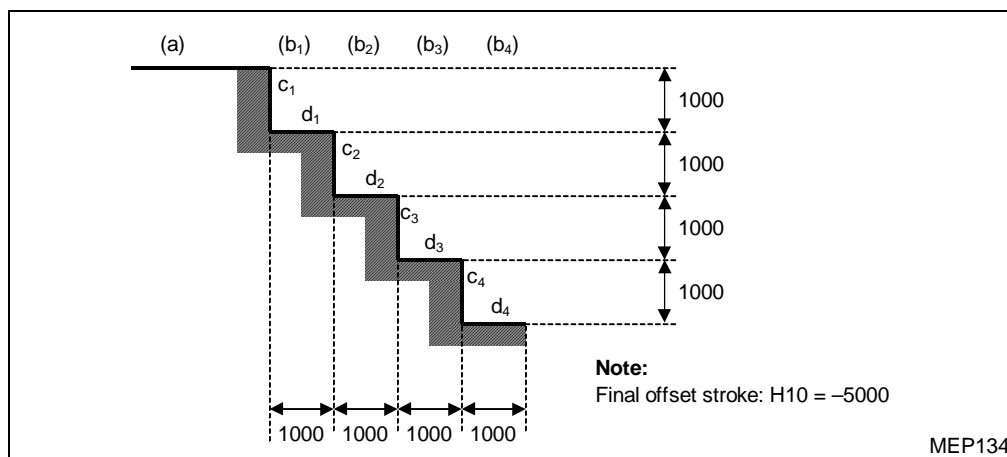
Example 2: Assumes that H10 has already been set equal to -1000.

Main program

```
N1 G00 X100000 ..... a
N2 #1=-1000
N3 M98 P1111L4 ..... b1, b2, b3, b4
```

Subprogram O1111

```
N1 G01 G91 G43 Z0 H10 F100 ..... c1, c2, c3, c4
N2 G01 X1000 ..... d1, d2, d3, d4
N3 #1=#1-1000
N4 G90 G10 L10 P10 R#1
N5 M99
```



Example 3: The programs in Example 2 above can be rewritten as follows:

Main program

```
N1 G00 X100000
N2 M98 P1111 L4
```

Subprogram O1111

```
N1 G01 G91 G43 Z0 H10 F100
N2 G01 X1000
N3 G10 L10 P10 R-1000
N4 M99
```

Note: Even when the command code is displayed on <Next Command>, the current offset number and variables will remain unupdated until that command is executed.

```
N1 G10 L10 P10 R-100
N2 G43 Z-10000 H10
N3 G0 X-10000 Y-10000
N4 G10 L10 P10 R-200
```

Executing block N4 will cause an offset stroke in H10 to be updated.

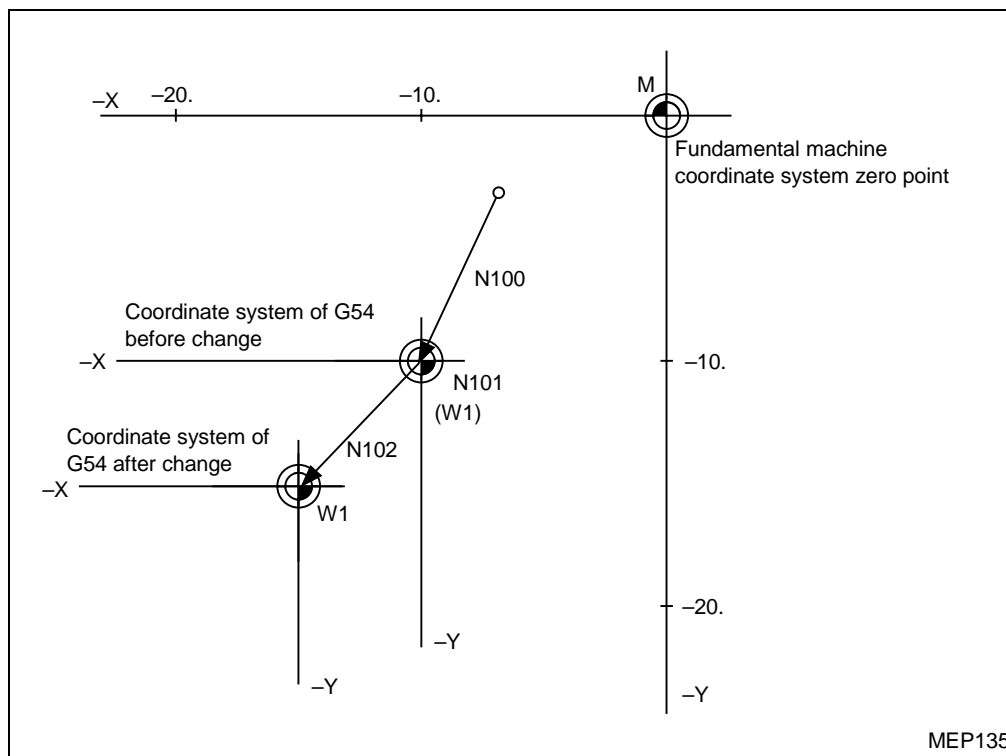
C. Updating the workpiece coordinate system offset data

Assume that the previous workpiece coordinate system offset data is as follows:

$X = -10.000$ $Y = -10.000$

```

:
N100 G00 G90 G54 X0 Y0
N101 G10 L2 P1 X-15.000 Y-15.000
N102 X0 Y0
:
M02
```



Note 1: Changes in the display of the workpiece position at N101

At N101, the display of tool position in the G54 coordinate system changes before and after workpiece coordinate system updating with G10.

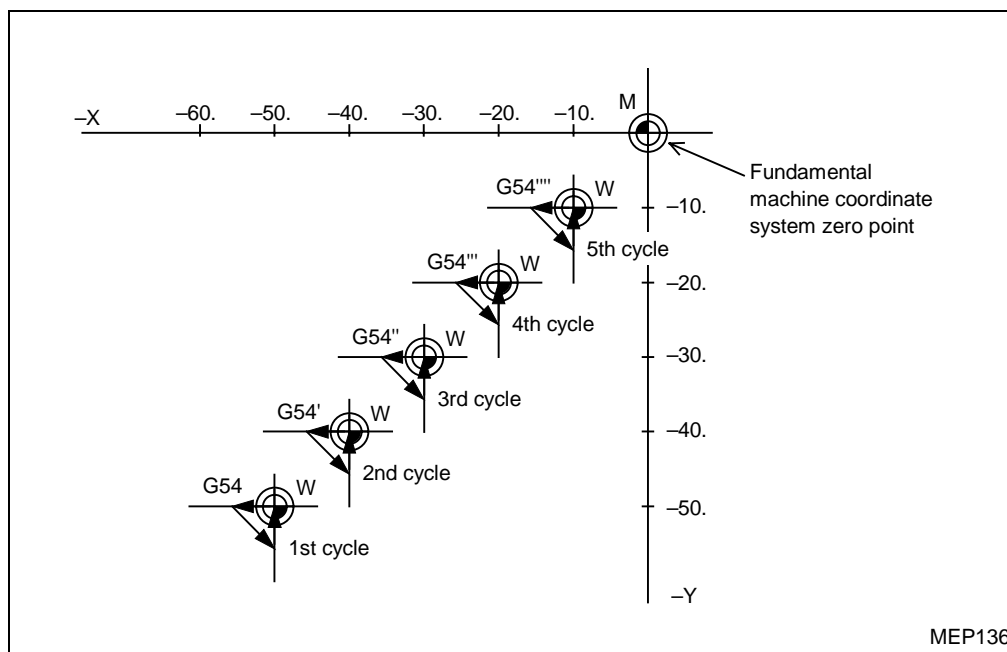
$$\begin{array}{lcl} X = 0 & \Rightarrow & X = +5.000 \\ Y = 0 & & Y = +5.000 \end{array}$$

Note 2: Prepare the following program to set workpiece coordinate system offset data in G54 to G59:

```
G10L2P1X-10.000 Y-10.000
G10L2P2X-20.000 Y-20.000
G10L2P3X-30.000 Y-30.000
G10L2P4X-40.000 Y-40.000
G10L2P5X-50.000 Y-50.000
G10L2P6X-60.000 Y-60.000
```

D. Programming for using one workpiece coordinate system as multiple workpiece coordinate systems

Main program	{	⋮
		#1=-50. #2=10.
		M98 P200 L5
		⋮
		M02
		%
Subprogram (O200)	{	N1 G90 G54 G10 L2 P1 X#1 Y#1
		N2 G00 X0 Y0
		N3 X-5. F100
		N4 X0 Y-5.
		N5 Y0
		N6 #1=#1+#2
		N7 M99
		%



MEP136

E. Programming for parameter data input

G10L50	Parameter input mode ON
N4017R10	D17 is set to "10".
N6088R96	F88 is set to "01100000". [$(01100000)_2 = (96)_{10}$]
N12067R-1000	L67 is set to "-1000".
N12072R67	L72 is set to "0x43". [$(43)_{16} = (67)_{10}$]
N150004P1R50	P4 data for the 1st axis (X-axis) is set to "50".
G11	Parameter input mode OFF

5. Related alarms

Alarm No.	Alarm message	Cause	Remedy
807	ILLEGAL FORMAT	Work offset input: P-command is omitted in a block of G10 L20 (or L2) although the last selected coordinate system is one of the systems from G54 to G59 (or of the G54.1 systems). Parameter input: An illegal parameter number is set.	Review the program data.
809	ILLEGAL NUMBER INPUT	Work offset input: The setting range of the coordinate system number or the offset data is overstepped. Tool offset input: The setting range of the offset data is overstepped. Parameter input: The axis number is not specified for an axis type parameter. The setting range of the axis number or the parameter data is overstepped.	Review the program data.
839	ILLEGAL OFFSET No.	Tool offset input: The specified offset number is greater than the number of available data sets.	Correct the offset number according to the number of available data sets.
903	ILLEGAL G10 L NUMBER	Work offset input: A command of G10 L20 is set although the corresponding function for the G54.1 coordinate systems is not provided.	Give an available L-code command.

12-8 Tool Offsetting Based on MAZATROL Tool Data

Parameter selection allows both tool length offset and tool radius compensation to be performed using MAZATROL tool data (tool diameter and tool length data).

12-8-1 Selection parameters

Using the following parameters, select whether or not MAZATROL tool data is to be used:
User parameters

F92 bit 7 = 1: Tool radius compensation uses the MAZATROL tool data **ACT-φ** (tool diameter data).

F93 bit 3 = 1: Tool length offsetting uses the MAZATROL tool data **LENGTH** (tool length data).

F94 bit 2 = 1: Tool length offsetting using the MAZATROL tool data is prevented from being cancelled by a reference-point return command.

F94 bit 7 = 1: Tool offsetting uses the MAZATROL tool data **ACT-φ CO.** (or **No.**) and **LENG CO.** (or **No.**).
(Set **F94** bit 7 to 0 to use the data stored on the **TOOL OFFSET** display.)

1. Tool length offsetting

Data items used		Parameter		Programming format	Remarks
		F93 bit 3	F94 bit 7		
TOOL OFFSET	Tool offset No.	0	0	G43/G44 H_	
TOOL DATA (MAZATROL)	LENGTH			T_	
	LENGTH + OFFSET No. or LENGTH + LENG CO.	1	1	T_ + H_	- Length offset cancellation not required for tool change. - G43 not required.
	OFFSET No. or LENG CO.	0	1	G43/G44 H_	Length offset cancellation required for tool change. ^[*]
TOOL OFFSET + TOOL DATA	Tool offset No. + LENGTH	1	0	(G43/G44 H_) + (T_)	Length offset cancellation required for tool change. ^[*]

[*] Canceling method - Set G49 before tool change command.
- Set G28/G30 before tool change command (when **F94** bit 2 = 0).

2. Tool radius compensation

Data items used		Parameter		Programming format
		F92 bit 7	F94 bit 7	
TOOL OFFSET	Tool offset No.	0	0	G41/G42 D_
TOOL DATA (MAZATROL)	ACT-φ + ACT-φ CO. or ACT-φ + OFFSET No.	1	1	G41/G42 T_
	ACT-φ CO. or OFFSET No.	0	1	G41/G42 T_
TOOL OFFSET + TOOL DATA	Tool offset No. + ACT-φ	1	0	G41/G42 D_ + T_

12-8-2 Tool length offsetting

1. Function and purpose

Even when offset data is not programmed, tool length offsetting will be performed according to the MAZATROL tool data **LENGTH** that corresponds to the designated tool number.

2. Parameter setting

Set both bit 3 of parameter **F93** and bit 2 of parameter **F94** to 1.

3. Detailed description

1. Tool length offsetting is performed automatically, but its timing and method differ as follows:
 - After a tool change command has been issued, offsetting is performed according to the **LENGTH** data of the tool mounted in the spindle. (A tool change command code must be set in the program before tool length offsetting can be done.)
 - After command G43 has been set, offsetting is performed according to the **LENGTH** data of the tool mounted in the spindle.
2. Tool length offsetting is cancelled in the following cases:
 - When a command for tool change with some other tool is executed
 - When M02 or M30 is executed
 - When the reset key is pressed
 - When command G49 is issued
 - When a reference-point return command is executed with bit 2 of parameter **F94** set to 0
3. Tool length offsetting becomes valid for the block onward that first involves Z-axis movement after tool change. This does not apply, however, if the first motion block in question is of G28, G30, or G53.
4. If this offset function is used with a G43 H-command, offsetting will use as its offset data the sum total of the MAZATROL tool data **LENGTH** and the offset amount specified by the G43 H (or G44 H) command.

Note 1: Set G43 H0 if tool length offsetting is to be done using a G43 H-command and only the offset amount specified by H is to be cancelled.

Note 2: With a G44 command, tool length offsetting based on MAZATROL tool data is not performed.

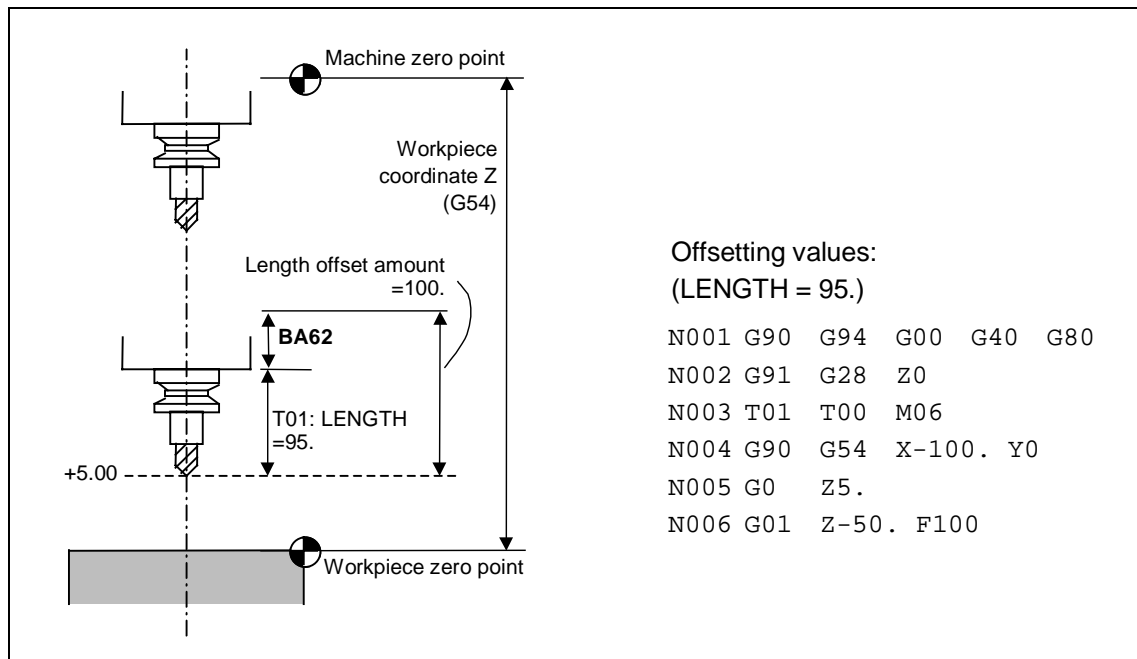
Note 3: The restart operation must begin from a position before a G43 command code or a tool change command code. Even when the spindle has a mounted tool, G43 or the tool change command must be executed before offsetting based on MAZATROL tool data can take place.

Note 4: Offsetting will fail if registered MAZATROL tool data **LENGTH** is not present.

Note 5: For an EIA/ISO program, to carry out tool length offset operations using the tool length data included in MAZATROL tool data, it becomes necessary to set data in the validation parameter for the tool length data of the MAZATROL tool data and to insert a tool change T- and M-code command block. It is to be noted that the tool change command block may not be missed particularly in the following cases:

- During automatic operation, if the first tool to be used has already been mounted in the spindle.
- During call of an EIA/ISO program as a subprogram from the MAZATROL main program, if the tool to be used immediately prior to call of the subprogram is the same as that which is to be designated in that subprogram as the first tool to be used.

4. Sample programs



12-8-3 Tool radius compensation

1. Function and purpose

Tool radius compensation by a G41 or G42 command uses MAZATROL tool data **ACT-φ** for the calculation of offset amount.

2. Parameter setting

Set bit 7 of parameter **F92** to 1.

3. Detailed description

- Tool radius compensation uses as its offset amount the half of the diameter data of the tool which is mounted in the spindle at the issuance of G41/G42.
- Tool radius compensation is cancelled by G40.
- If the tool radius compensation function is used with a D-command, the sum total of the data indicated by the corresponding offset number (D) and the radius of the tool will be used as the offset data.

Note 1: The tool used must be mounted in the spindle before restarting the program.

Note 2: Offsetting based on tool diameter data will not occur if registered MAZATROL tool diameter data is not present or if a tool for which tool diameter data cannot be entered is to be used.

Note 3: To carry out for an EIA/ISO program the tool radius compensation operations using the tool diameter data included in MAZATROL tool data, it is necessary to insert tool change command blocks, as it is the case for tool length offsetting (refer to **Note 5** in Subsection 12-7-2).

12-8-4 Tool data update (during automatic operation)

1. Function and purpose

Tool Data Update allows MAZATROL tool data to be updated during automatic operation based on an EIA/ISO program.

2. Parameter setting

Set parameter **L57** to 1.

3. Detailed description

This function allows the entire tool data, except for spindle tools, to be updated during automatic operation based on an EIA/ISO program.

Parameter	TOOL	NOM- ϕ	ACT- ϕ	LENGTH	LENG COMP.	THRUST F/ HORSE PW	LIFE TIME	CUT TIME	MAT.	MAX. ROT.
L57 = 0	No	No	No	No	No	No	Yes	Yes	No	Yes
L57 = 1	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes

Note 1: In the table given above, "Yes" indicates that you can update the data, and "No" indicates that you cannot update the data.

Identification between MAZATROL programs and EIA/ISO programs is automatically made by whether the program currently being executed, is MAZATROL or EIA/ISO, irrespective of whether it is a main program or subprogram.

If, however, the main program is MAZATROL and its subprograms are EIA/ISO, then the currently active set of programs is regarded as a MAZATROL program.

Note 2: An alarm **428 MEMORY PROTECT (AUTO OPERATION)** will occur if the spindle tool data is modified during automatic operation based on an EIA/ISO program.

12-9 Shaping Function (Option)

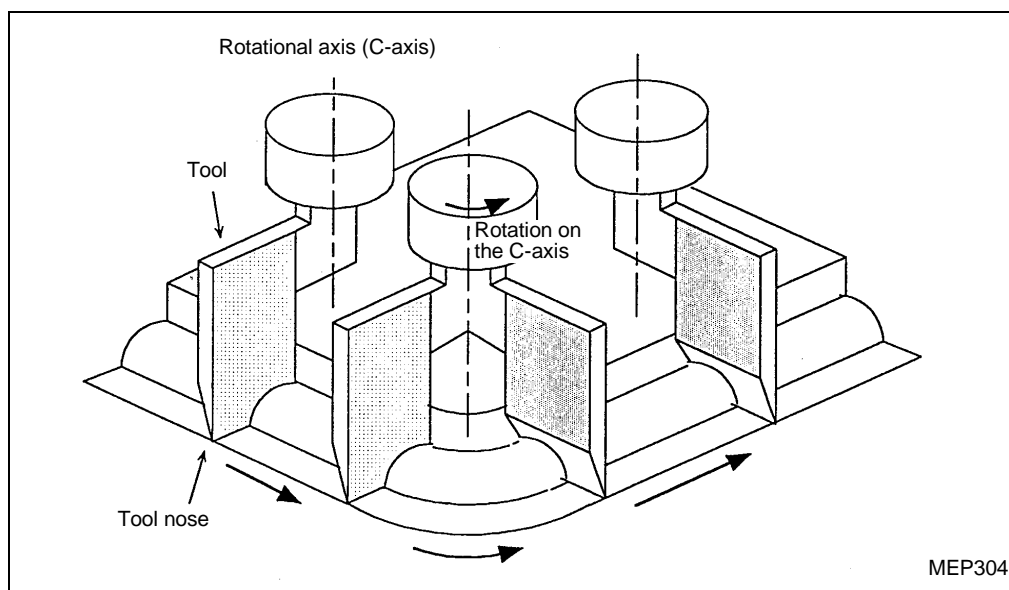
12-9-1 Overview

The shaping function is provided to control the rotational axis concerned in order to keep the tool set normal (perpendicular) to the direction of the movement in the XY-plane.

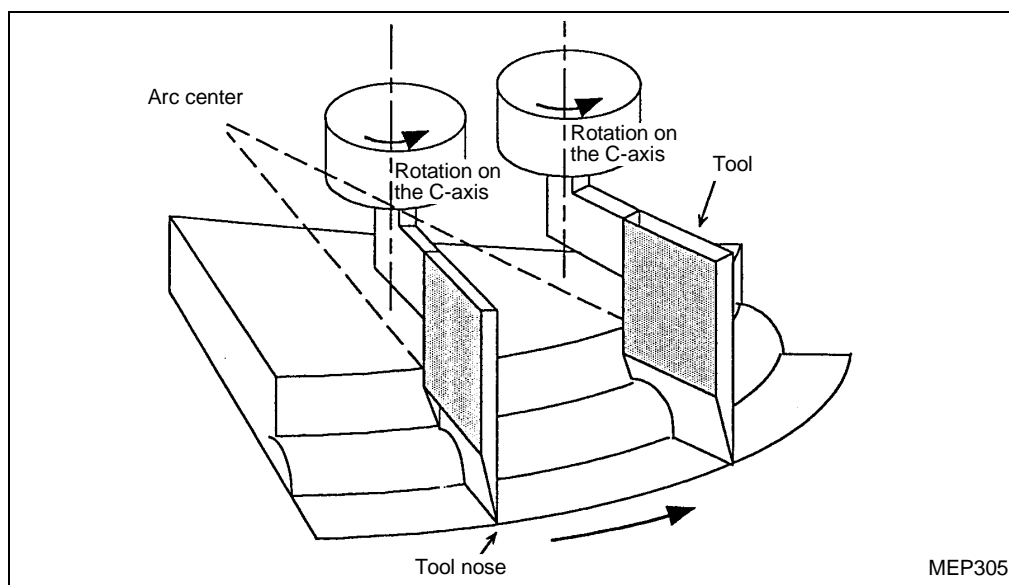
This optional function permits free-form shapes such as the rubber oil-seal surface to be cut out for a better surface finish than with an end-milling tool.

Note: The name of the rotational axis to be controlled for the shaping function depends on the construction of the machine in question. The description in this section assumes that the rotational axis concerned is named C-axis.

- The C-axis control is automatically performed at block connections to keep the tool normally oriented.



- During circular interpolation, the C-axis is continuously controlled in synchronization with the tool movement.



12-9-2 Programming format

```
G40.1 }
G41.1 } Xx Yy Ff
G42.1 }
```

G40.1: Cancellation of shaping

G41.1: Selection of shaping to the left (normal orientation on the left side)

G42.1: Selection of shaping to the right (normal orientation on the right side)

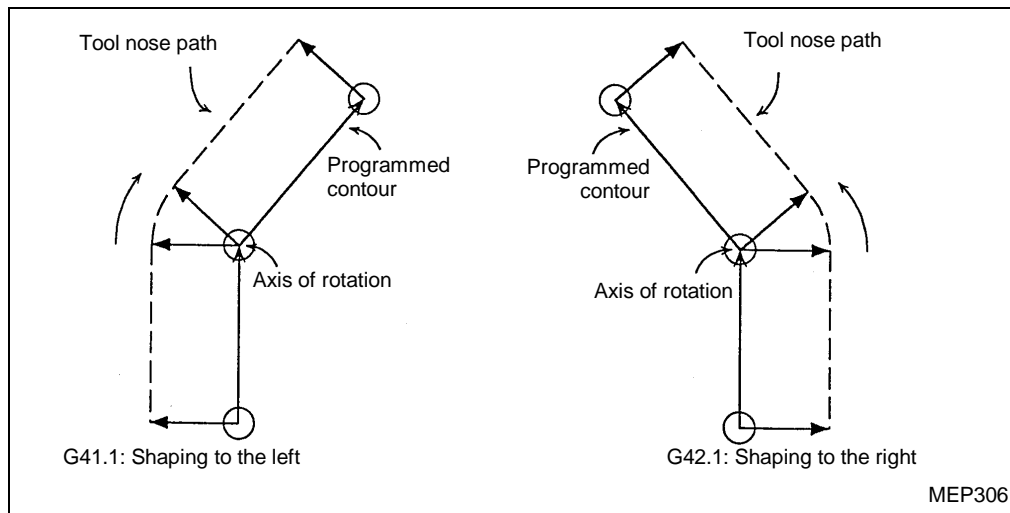
x: X-axial position of ending point

y: Y-axial position of ending point

f: Feed rate

Note 1: The codes G40.1, G41.1 and G42.1 belong to group 15 of G-codes.

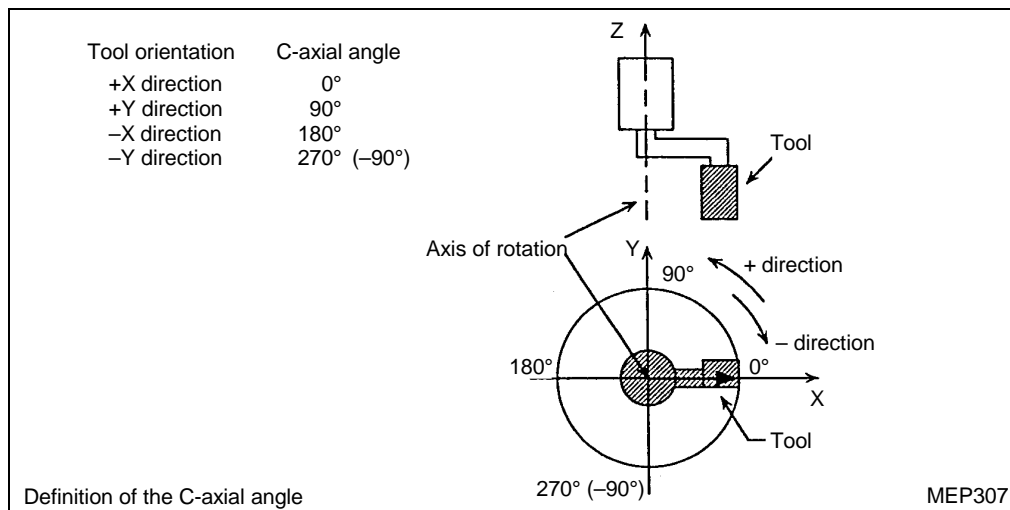
Note 2: The shaping control (orientation of the tool) can only be performed in the XY-plane, regardless of the plane currently selected.



12-9-3 Detailed description

1. Definition of the C-axial angle

The C-axial angle with the tool oriented in the +X direction is defined as 0°, and counterclockwise rotation is defined as positive (+).

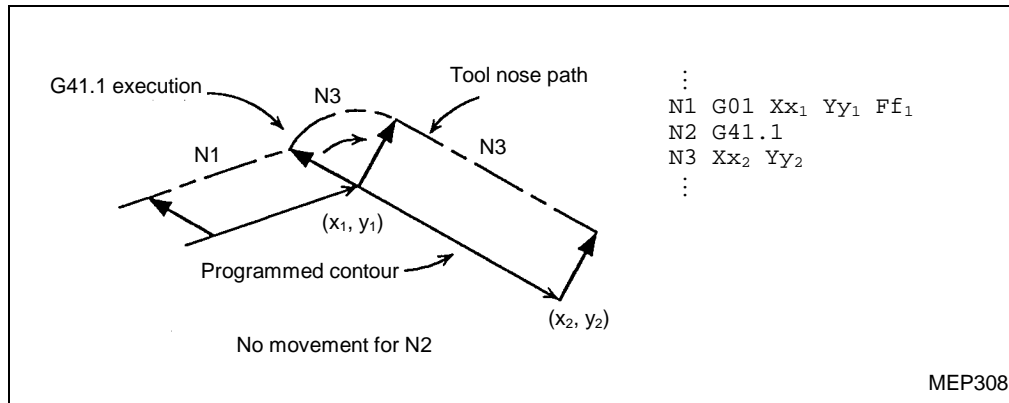


2. Movement

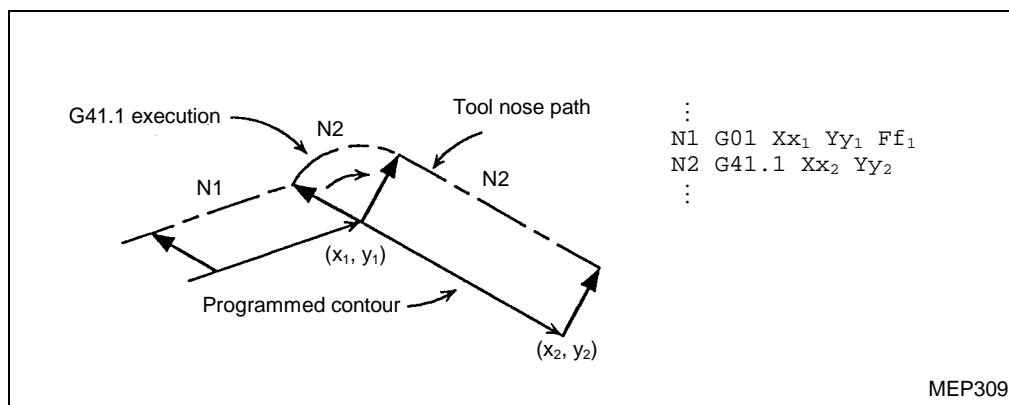
A. Start up

The C-axis rotation is performed at the starting point of the first shaping block and then the X- and Y-axial movement is carried out with the tool normally oriented. The direction of the preparatory rotation is automatically selected for the smallest angle ($\leq 180^\circ$).

- Selection in a single-command block



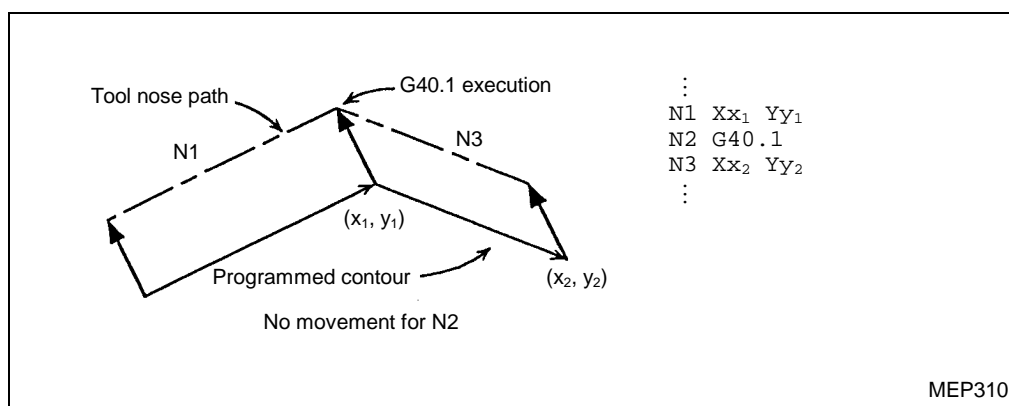
- Selection in a block containing motion command



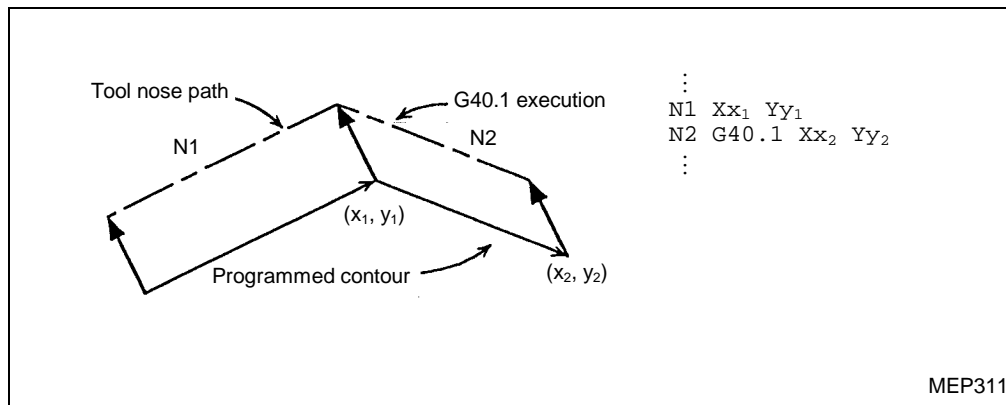
B. Cancellation

After cancellation of shaping, the X- and Y-axial movement is carried out without C-axis rotation.

- Cancellation in a single-command block



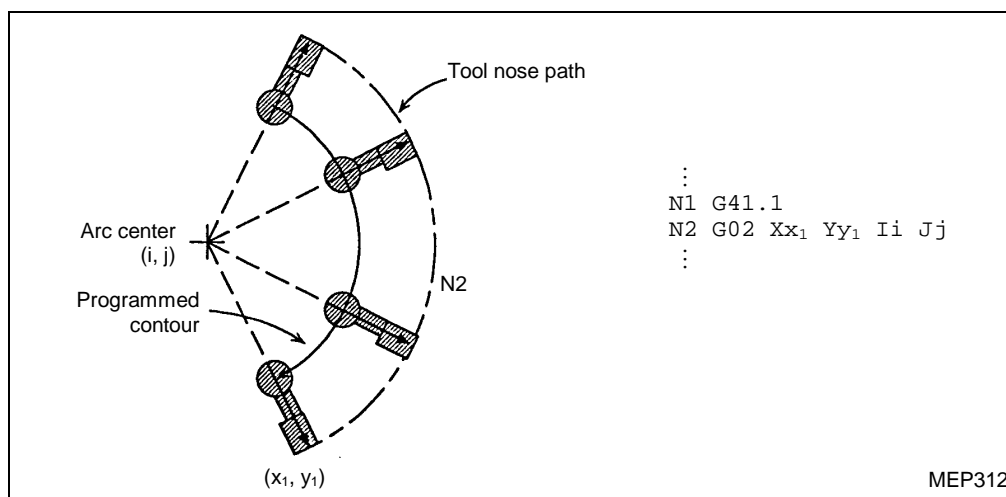
- Cancellation in a block containing motion command



C. Movement in the shaping mode

Execution of a block

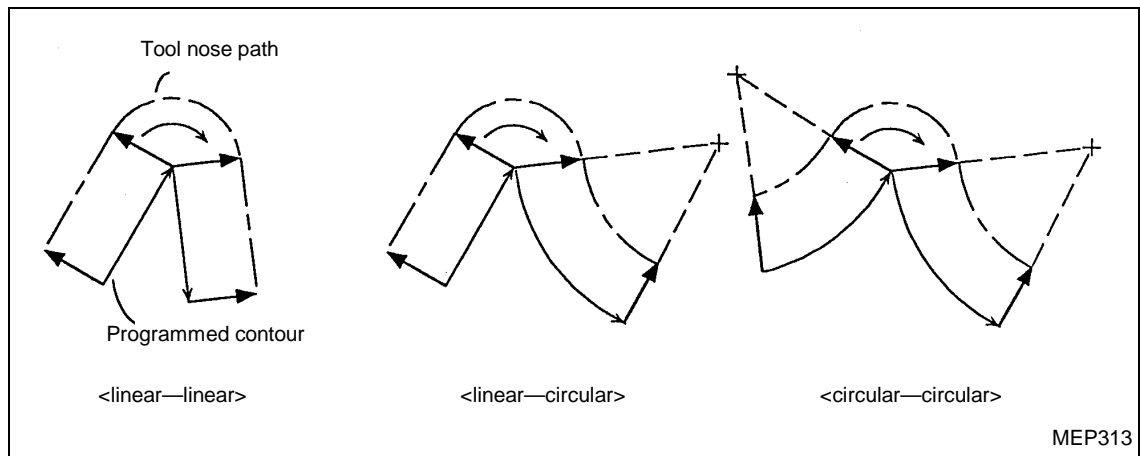
- Block of linear interpolation
The tool moves linearly without C-axis rotation.
- Block of circular interpolation
The angular position on the C-axis is continuously controlled in synchronization with the circular movement of the tool.



Connection between blocks

- Without tool radius compensation

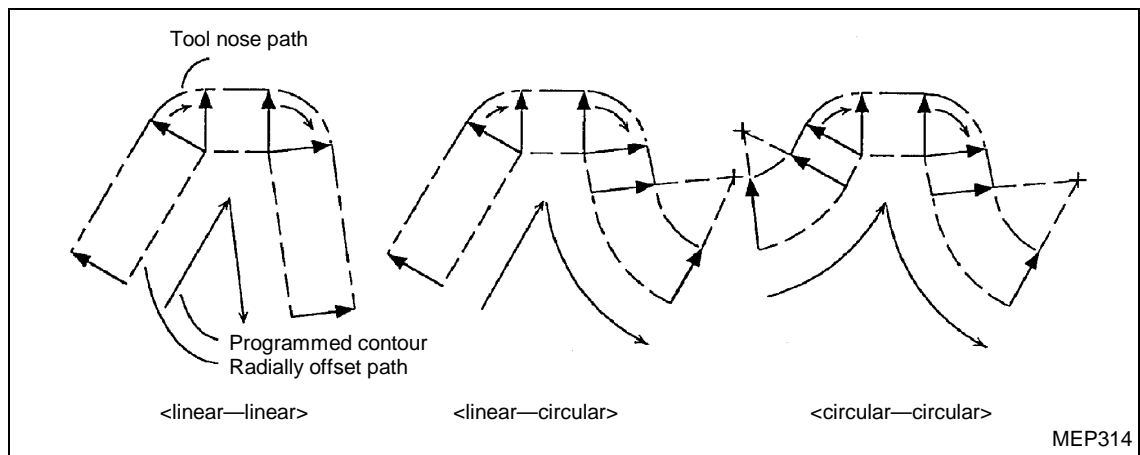
An independent C-axis rotation is performed at the end of the preceding block to orient the tool in the normal direction with respect to the starting motion of the next block.



- With tool radius compensation

The tool radius compensation automatically inserts linear segments for connection between blocks whose paths cross each other at a sharp angle.

The shaping function controls the C-axis so as to orient the tool in accordance with the offset tool path.



Direction of C-axis rotation at block connections

The rotation on the C-axis occurs in the negative direction (clockwise) in the mode of G41.1, or in the positive direction (counterclockwise) in the mode of G42.1, at block connections.

Parameter **K2** (ε : minimum allowable angle of C-axis rotation) is provided to suppress the rotation as described below.

- Direction of C-axis rotation at block connections

For G41.1: negative (CW)

For G42.1: positive (CCW)

- Suppression or prohibition of C-axis rotation at block connections

θ : Rotational angle required

ε : Parameter **K2** (minimum allowable angle of C-axis rotation)

$$|\theta| < \varepsilon$$

The C-axis rotation is suppressed.

In the mode of G41.1:

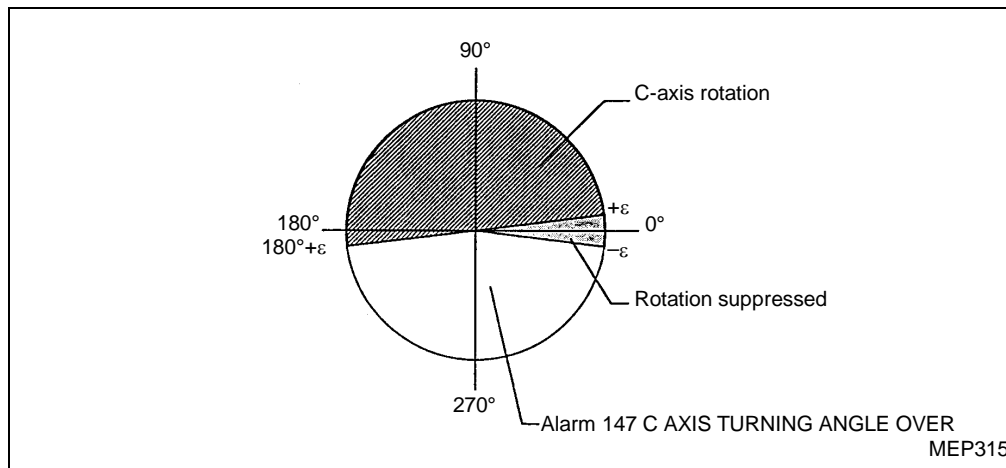
$$\varepsilon \leq \theta < 180^\circ - \varepsilon$$

Alarm No. **147 C AXIS TURNING ANGLE OVER** will be caused.

In the mode of G42.1:

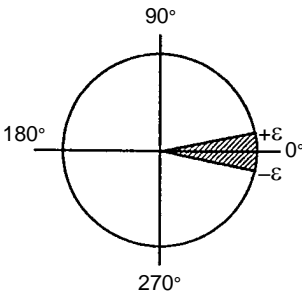
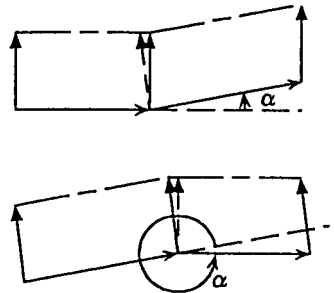
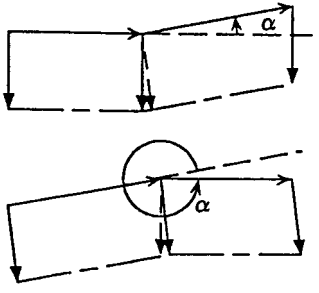
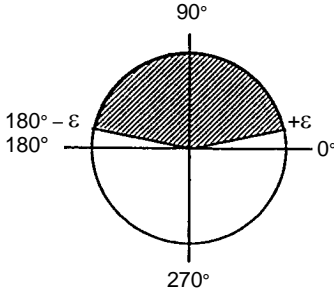
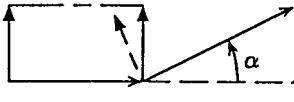
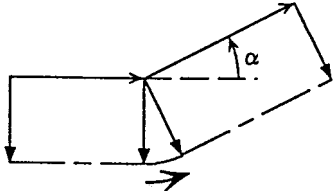
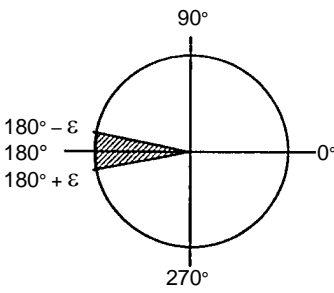
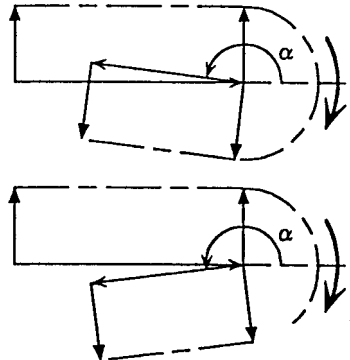
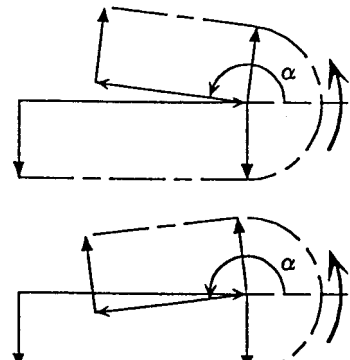
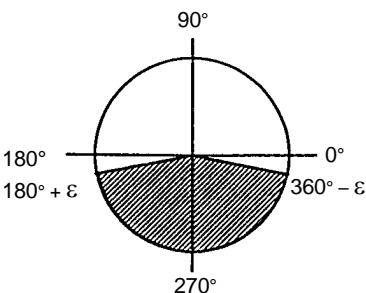
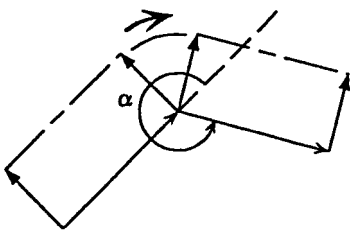
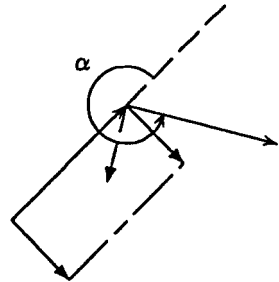
$$180^\circ + \varepsilon \leq \theta < 360^\circ - \varepsilon$$

Alarm No. **147 C AXIS TURNING ANGLE OVER** will be caused.



Note: The C-axis rotation is suppressed if the angle of rotation required is smaller than parameter **K2** ($|\theta| < \varepsilon$).

The rotational angle thus ignored will surely be added to the angle of the next rotation required, which will then be actually performed or further suppressed according to the result of the accumulation.

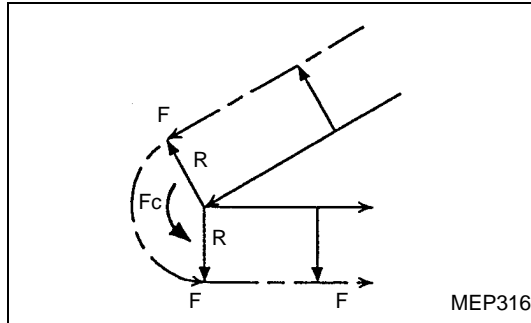
Angle at a block connection: α	G41.1	G42.1
1. $-\varepsilon < \alpha < +\varepsilon$ 	 C-axis rotation suppressed	 C-axis rotation suppressed
2. $+\varepsilon < \alpha < (180^\circ - \varepsilon)$ 	 Alarm 147 C AXIS TURNING ANGLE OVER	
3. $(180^\circ - \varepsilon) \leq \alpha \leq (180^\circ + \varepsilon)$ 		
4. $(180^\circ + \varepsilon) \leq \alpha \leq (360^\circ - \varepsilon)$ 		 Alarm 147 C AXIS TURNING ANGLE OVER

3. Speed of C-axis rotation for shaping

- At block connection

The C-axis rotation is performed at such a speed that the tool nose will move at the speed specified by the F-code.

The C-axis rotational speed F_c is calculated as follows:



If parameter **K1** (radius of C-axis rotation) $\neq 0$

$$F_c = \frac{F}{R} \times \frac{180}{\pi} \text{ (deg/min)}$$

If parameter **K1** (radius of C-axis rotation) $= 0$

$$F_c = F \text{ (deg/min)}$$

F: Feed rate (mm/min)

R: Parameter **K1** (mm) [radius of C-axis rotation (distance between C-axis and tool nose)]

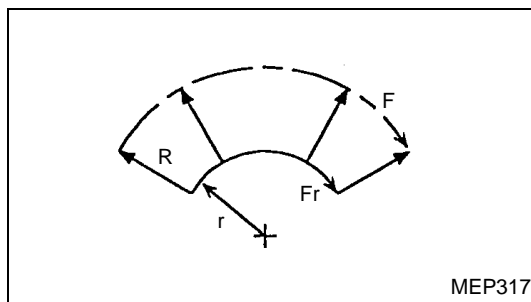
The C-axis rotation, however, is controlled in order that the preset maximum allowable cutting speed of the C-axis should not be exceeded, regardless of the result F_c of the above calculation.

Rapid traverse occurs according to parameter **M1** (rapid traverse speed [for the C-axis]).

- During circular interpolation

The circular interpolation is performed at such a speed that the tool nose will move at the speed specified by the F-code.

The cutting feed rate of the circular interpolation (F_r) is calculated as follows:



$$F_r = F \times \frac{r}{R + r} \text{ (mm/min)}$$

F: Feed rate (mm/min)

r : Radius of circular interpolation (mm)

R: Parameter **K1** (mm) [radius of C-axis rotation (distance between C-axis and tool nose)]

The speed of the circular interpolation (F), however, is automatically controlled in order that the preset maximum allowable cutting speed of the C-axis should not be exceeded.

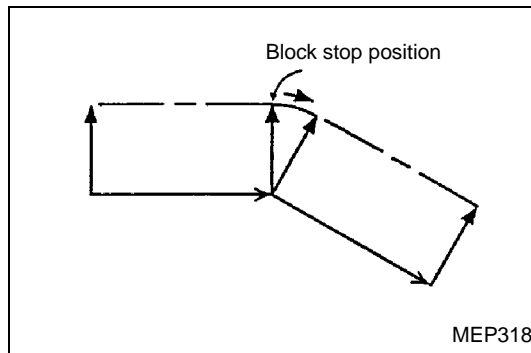
12-9-4 Remarks

1. If the axis of the work spindle is to be used for shaping control, the spindle axis must be changed over to a servoaxis (C-axis). The following M-codes are provided to select the control mode of the work spindle.

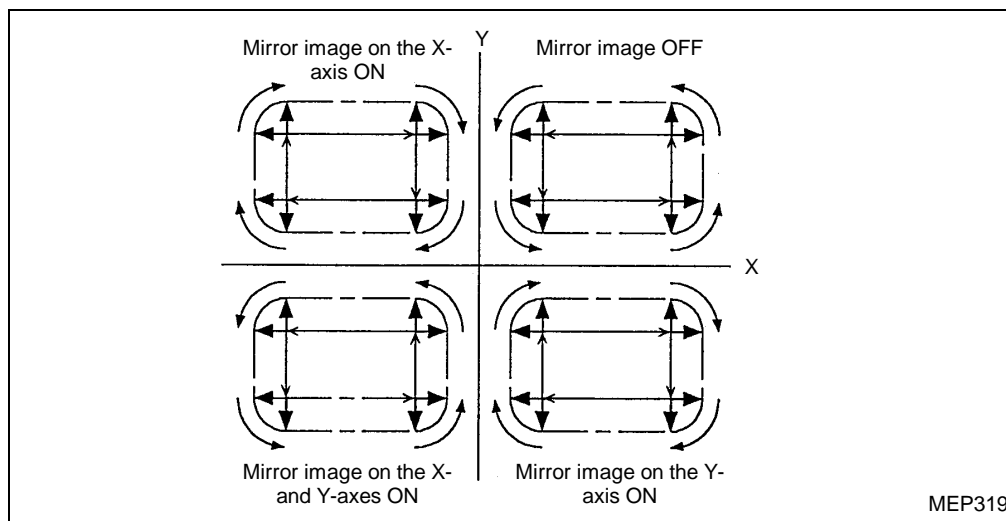
M193: Selection of spindle as the C-axis (Servo On)

M194: Selection of spindle as the milling spindle (Servo Off)

2. In the mode of single-block operation, interlocking at the start of cutting block or each block, the operation will be stopped before the preparatory rotation on the C-axis.



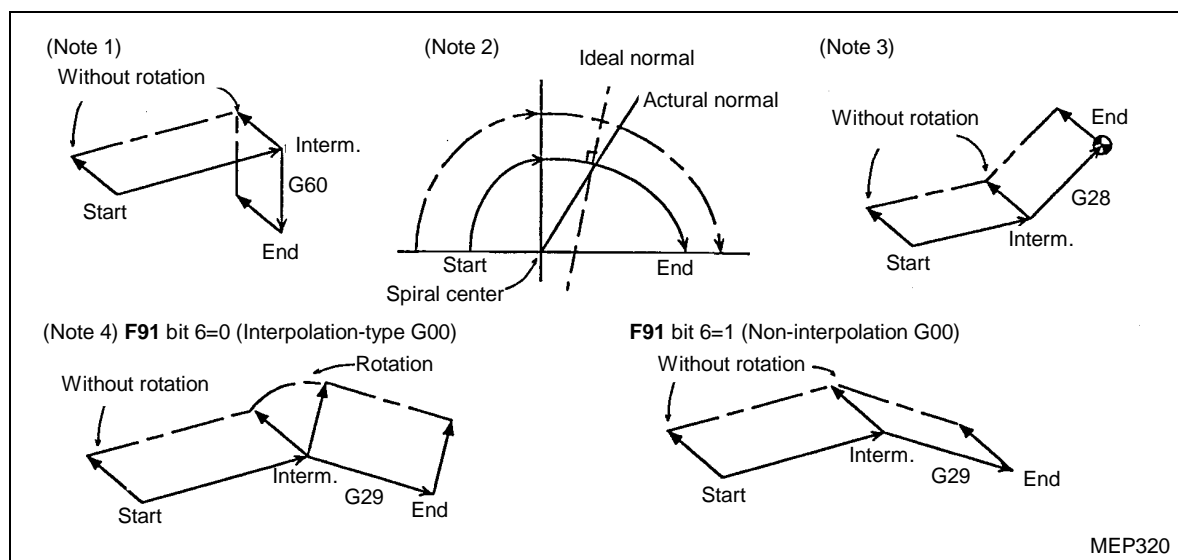
3. The C-axis motion command is ignored in the mode of shaping.
4. The workpiece origin offsetting for the C-axis (G92 Cc) cannot be set in the mode of shaping (G41.1 or G42.1). Setting such a command will only result in alarm **807 ILLEGAL FORMAT**.
5. With the mirror image selected for the X- or Y-axis, the direction of the C-axis rotation is reversed.



6. The indication for the C-axis under **BUFFER** on the **POSITION** display refers to an absolute value.
7. For the connection between blocks, the **BUFFER** area on the **POSITION** display indicates the angle of the C-axis rotation in addition to the distance of the X- and Y-axial movement.

12-9-5 Compatibility with the other functions

Function	Description
One-way positioning	Shaping control is suppressed. (Note 1)
Helical interpolation	Shaping is realized adequately.
Spiral interpolation	Shaping cannot be realized correctly since the starting and ending point do not lie on one and the same circumference. (Note 2)
Synchronous feed	The designated feed rate cannot be obtained since the work spindle is controlled as the C-axis.
Shape correction	Shaping cannot be realized correctly since the control for constant acceleration and deceleration is not applicable to the rotation on the C-axis.
High-speed machining	Alarm 807 ILLEGAL FORMAT will be caused.
Exact-stop check	Deceleration and stop do not occur for the rotation on the C-axis.
Error detection	Deceleration and stop do not occur for the rotation on the C-axis.
Overriding	Overriding is applied adequately to the rotation on the C-axis.
Figure rotation	Shaping control is performed for the rotated figure.
Coordinates system rotation	Shaping control is performed for the rotated figure.
Scaling	Shaping control is performed for the scaled figure.
Mirror image	Shaping control is performed for the mirrored figure.
Linear angle command	Shaping control is performed for the calculated connection between linear segments.
Return to reference point	Shaping control is suppressed. (Note 3)
Return to starting point	Shaping control is suppressed for the movement to the intermediate point, indeed, but it is performed for the movement from the intermediate point to the programmed position if the interpolation-type rapid traverse (G00) is selected [F91 bit 6 = 0]. (Note 4)
Workpiece coordinate system setting	The rotation on the C-axis is performed with reference to the coordinate system established in the shaping mode.
Local coordinate system setting	The rotation on the C-axis is performed with reference to the coordinate system established in the shaping mode.
Dry run	The speed of C-axis rotation is also modified by the external signal.
Modal restart	Restart from a block in the shaping mode can be performed with adequate control of the C-axis.
Non-modal restart	Restart from the midst of the shaping mode is only performed without C-axis control since the modal information before the restart block is ignored.
Tool path check (plane)	The rotation on the C-axis cannot be displayed.
Virtual Machining/ Safety Shield	The rotation on the C-axis cannot be displayed.
Three-dimensional coordinate conversion	Shaping function is available in the mode of 3D coordinate conversion, but on the contrary 3D coordinate conversion cannot be used in the shaping mode.



12-9-6 Sample program

Main program

WNo. 1000

```

O1000
G91G28 X0 Y0 Z0
M193
G28 C0
G90 G92 G53 X0 Y0 Z0
G00 G54 G43 X35.Y0.Z100.H1
G00 Z3.
G01 Z0.1 F3000
G42.1
M98 P1001 L510
M98 P1002 L2
G91 G01 Y10.Z0.05
G40.1
G90 G00 Z100.
G28 X0 Y0 Z0
G00 C0
M194
M30
%
```

Subprogram

WNo. 1001

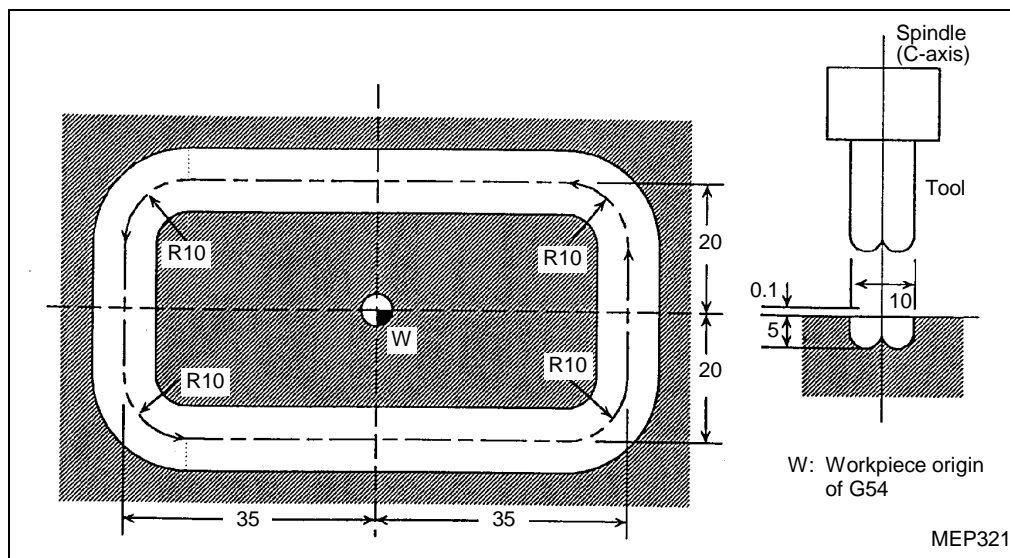
```

O1001
G17 G91 G01 Y20.,R10.Z-0.01
X-70.,R10.
Y-40.,R10.
X70.,R10.
Y20.
M99
%
```

WNo. 1002

```

O1002
G17 G91 G01 Y20.,R10.
X-70.,R10.
Y-40.,R10.
X70.,R10.
Y20.
M99
%
```



13 PROGRAM SUPPORT FUNCTIONS

13-1 Hole Machining Pattern Cycles: G34.1/G35/G36/G37.1

13-1-1 Overview

1. Function and purpose

Hole machining patterns are used to arrange on a predetermined pattern hole positions at which to execute a hole-machining cycle.

- Give beforehand a command of the desired hole-machining cycle without any axis positioning data (which only causes storage of the hole-machining data to be executed at the arranged hole positions).
- The execution of this command begins with the positioning to the first one of the arranged holes. The type of hole machining depends on the corresponding cycle designated last.
- The current mode of hole-machining cycle will remain active over the execution of this command till it is cancelled explicitly.
- This command will only activate positioning when it is given in any other mode than those of hole-machining cycle.
- These commands only cause positioning at the speed of the current modal condition (of G-code group 01) in default of any preceding hole-machining cycle.

2. List of hole machining pattern cycles

G-code	Description	Argument addresses	Remarks
G34.1	Holes on a circle	X, Y, I, J, K	
G35	Holes on a line	X, Y, I, J, K	
G36	Holes on an arc	X, Y, I, J, P, K	
G37.1	Holes on a grid	X, Y, I, P, J, K	

Note: In order that it may function correctly, do not give a command of hole machining pattern cycle in one block together with any one for the following functions (described in this chapter):

- Fixed cycle
- Subprogram control
- Macro call

13-1-2 Holes on a circle: G34.1

As shown in the format below, a command of G34.1 determines a circle of radius “r” around the center designated by X and Y. The circumference is then divided, beginning from the point of the central angle “ θ ”, regularly by “n”, and the hole machining designated beforehand by a fixed cycle (G81 etc.) will be done around all the vertices of the regular n-gon.

The movement in the XY-plane from hole to hole occurs rapidly (under G00). The argument data of the G34.1 command will be cleared upon completion of its execution.

1. Programming format

G34.1 Xx Yy Ir J θ Kn;

X, Y : Coordinates of the center of the circle.

I : Radius (r) of the circle. Always given in a positive value.

J : Central angle (θ) of the first hole. Positive central angles refer to counterclockwise measurement.

K : Number (n) of holes to be machined (from 1 to 9999). The algebraic sign of argument K refers to the rotational direction of the sequential machining of “n” holes. Set a positive and a negative number respectively for counterclockwise and clockwise rotation.

2. Sample programmes

Given below is an example of G81 hole machining with a figure representing the hole positions.

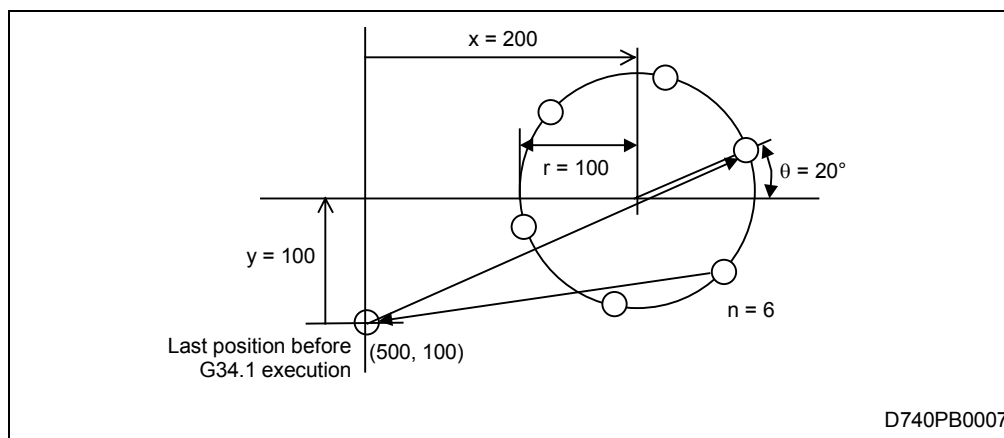
N001 G91;

N002 G81 Z-10. R5. L0. F200;

N003 G90 G34.1 X200. Y100. I100. J20. K6;

N004 G80;

N005 G90 G0 X500. Y100.;



3. Notes

- Give a G90 or G91 command as required to designate the axis position in absolute or incremental values.
- As shown in the above example, the last position of the G34.1 command is on the last one of the arranged holes. Use the method of absolute data input, therefore, to specify the movement to the position for the next operation desired. (An incremental command would require a more or less complicated calculation with respect to that last hole.)

13-1-3 Holes on a line: G35

As shown in the format below, a command of G35 determines a straight line through the starting point designated by X and Y at the angle “ θ ” with the X-axis. On this line “n” holes will be machined at intervals of “d”, according to the current mode of hole machining.

The movement in the XY-plane from hole to hole occurs rapidly (under G00). The argument data of the G35 command will be cleared upon completion of its execution.

1. Programming format

G35 Xx Yy Id J θ Kn;

X, Y : Coordinates of the starting point.

I : Interval (d) between holes. Change of sign for argument I causes a centrically symmetric hole arrangement with the starting point as the center.

J : Angle (θ) of the line. Positive angles refer to counterclockwise measurement.

K : Number (n) of holes to be machined (from 1 to 9999), inclusive of the starting point.

2. Sample programmes

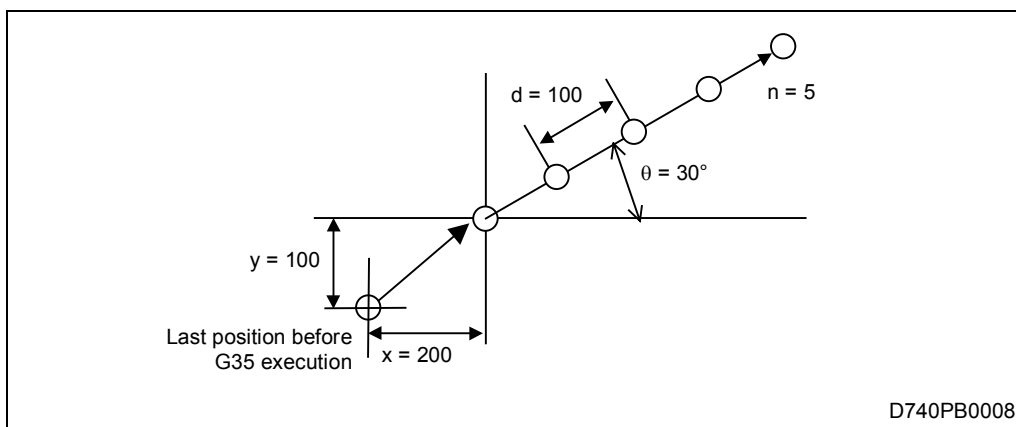
Given below is an example of G81 hole machining with a figure representing the hole positions.

N001 G91;

N002 G81 Z-10. R5. L0. F100;

N003 G35 X200. Y100. I100. J30. K5;

N004 G80;



3. Notes

- Give a G90 or G91 command as required to designate the axis position in absolute or incremental values.
- Omission of argument K or setting “K0” will result in an alarm. A setting of K with five or more digits will lead to the lowest four digits being used.
- In a block with G35 any words with addresses other than G, L, N, X, Y, I, J, K, F, M, S, T and B will simply be ignored.
- Giving a G-code of group 00 in the same block with G35 will cause an exclusive execution of either code which is given later.
- In a block with G35 a G22 or G23 command will simply be ignored without affecting the execution of the G35 command.

13-1-4 Holes on an arc: G36

As shown in the format below, a command of G36 determines a circle of radius “r” around the center designated by X and Y. On the circumference “n” holes will be machined, starting from the point of the central angle “ θ ”, at angular intervals of “ $\Delta\theta$ ”, according to the current mode of hole machining.

The movement in the XY-plane from hole to hole occurs rapidly (under G00). The argument data of the G36 command will be cleared upon completion of its execution.

1. Programming format

G36 Xx Yy Ir J θ P $\Delta\theta$ Kn;

X, Y : Coordinates of the center of the arc.

I : Radius (r) of the arc. Always given in a positive value.

J : Central angle (θ) of the first hole. Positive central angles refer to counterclockwise measurement.

P : Angular interval ($\Delta\theta$) between holes. The algebraic sign of argument P refers to the rotational direction of the sequential machining of “n” holes. Set a positive and a negative value respectively for counterclockwise and clockwise rotation.

K : Number (n) of holes to be machined (from 1 to 9999).

2. Sample programmes

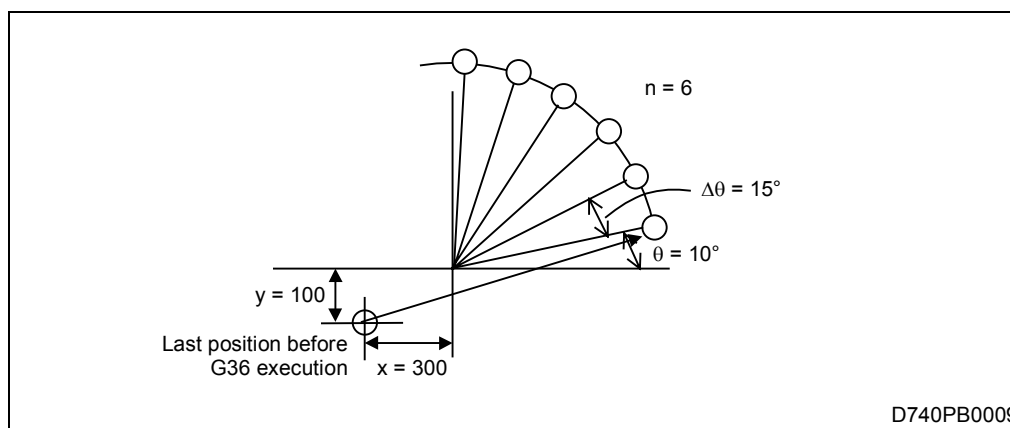
Given below is an example of G81 hole machining with a figure representing the hole positions.

N001 G91;

N002 G81 Z-10. R5. F100;

N003 G36 X300. Y100. I300. J10. P15. K6;

N004 G80;



3. Notes

- Give a G90 or G91 command as required to designate the axis position in absolute or incremental values.

13-1-5 Holes on a grid: G37.1

As shown in the format below, a command of G37.1 determines a grid pattern of $[\Delta x] * [nx]$ by $[\Delta y] * [ny]$ with the point designated by X and Y as starting point. On the grid points the hole machining designated beforehand by a fixed cycle will be done “nx” in number along the X-axis at intervals of “ Δx ”, and “ny” in number along the Y-axis at intervals of “ Δy ”. The main progression of machining occurs in the X-axis direction.

The movement in the XY-plane from hole to hole occurs rapidly (under G00). The argument data of the G37.1 command will be cleared upon completion of its execution.

1. Programming format

G37.1 Xx Yy I Δx Pnx J Δy Kny;

X, Y : Coordinates of the starting point.

I : Hole interval (Δx) on the X-axis. Set a positive and a negative value to arrange holes in respective directions from the starting point on the X-axis.

P : Number (nx) of holes to be arranged on the X-axis (from 1 to 9999).

J : Hole interval (Δy) on the Y-axis. Set a positive and a negative value to arrange holes in respective directions from the starting point on the Y-axis.

K : Number (ny) of holes to be arranged on the Y-axis (from 1 to 9999).

2. Sample programs

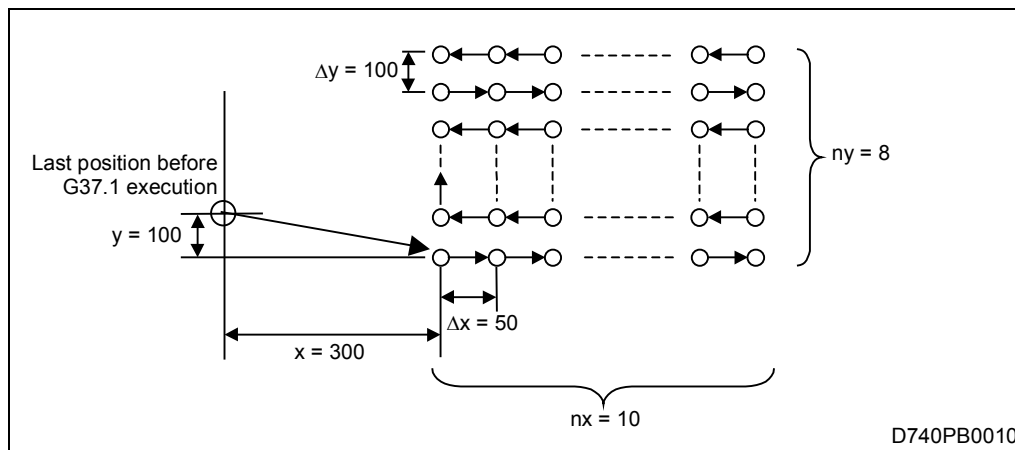
Given below is an example of G81 hole machining with a figure representing the hole positions.

N001 G91;

N002 G81 Z-10. R5. F20;

N003 G37.1 X300. Y-100. I50. P10 J100. K8;

N004 G80;



3. Notes

- Give a G90 or G91 command as required to designate the axis position in absolute or incremental values.
- Omission of argument P or K, or setting “P0” or “K0” will result in an alarm. A setting of K or P with five or more digits will lead to the lowest four digits being used.
- In a block with G37.1 any words with addresses other than G, L, N, X, Y, I, J, K, F, M, S, T and B will simply be ignored.

- Giving a G-code of group 00 in the same block with G37.1 will cause an exclusive execution of either code which is given later.
- In a block with G37.1 a G22 or G23 command will simply be ignored without affecting the execution of the G37.1 command.

13-2 Fixed Cycles

13-2-1 Outline

1. Function and purpose

The fixed-cycle functions allow positioning, hole-drilling, boring, tapping, or other machining programs to be executed according to the predetermined job sequence by the commands of a single block. The available job sequences for machining are listed below.

The fixed-cycle function mode is cancelled on reception of G80 or a G-command (G00, G01, G02, G03, G2.1, or G3.1) of group G01. All related types of data are also cleared to zero at the same time.

2. List of fixed cycles

G-code	Description	Arguments	Notes
G71.1	Chamfering cutter (CW)	[X, Y] Z, Q, R, F [P, D]	
G72.1	Chamfering cutter (CCW)	[X, Y] Z, Q, R, F [P, D]	
G73	High-speed deep-hole drilling	[X, Y] Z, Q, R, F [P, D, K, I, J(B)]	
G74	Reverse tapping	[X, Y] Z, R, F [P, D, J(B), H]	Dwell in seconds
G75	Boring	[X, Y] Z, R, F [Q, P, D, K, I, J(B)]	
G76	Boring	[X, Y] Z, R, F [Q, P, D, J(B)]	
G77	Back spot facing	[X, Y] Z, R, F [Q, P, E, J(B)]	Return to initial point only.
G78	Boring	[X, Y] Z, R, F [Q, P, D, K]	
G79	Boring	[X, Y] Z, R, F [Q, P, D, K, E]	
G81	Spot drilling	[X, Y] Z, R, F	
G82	Drilling	[X, Y] Z, R, F [P, D, I, J(B)]	
G83	Deep-hole drilling	[X, Y] Z, Q, R, F [P, D, K, I, J(B)]	
G84	Tapping	[X, Y] Z, R, F [P, D, J(B), H]	Dwell in seconds
G85	Reaming	[X, Y] Z, R, F [P, D, E]	
G86	Boring	[X, Y] Z, R, F [P]	
G87	Back boring	[X, Y] Z, R, F [Q, P, D, J(B)]	Return to initial point only.
G88	Boring	[X, Y] Z, R, F [P]	
G89	Boring	[X, Y] Z, R, F [P]	

Note 1: The arguments enclosed in brackets ([]) can be omitted.

Note 2: Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1: Argument J-command
= 0: Argument B-command

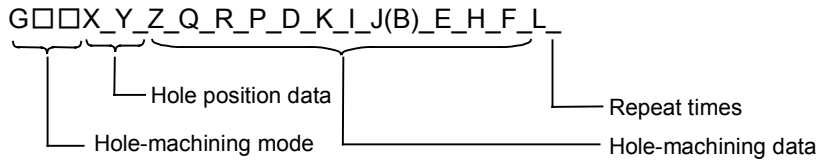
Note 3: In order that it may function correctly, do not give a command of fixed cycle in one block together with any one for the following functions (described in this chapter):

- Hole machining pattern cycle
- Subprogram control
- Macro call

13-2-2 Fixed-cycle machining data format

1. Setting fixed-cycle machining data

Set fixed-cycle machining data as follows:



- Hole-machining mode (G-code)

See the list of the fixed cycles.

- Hole position data (X, Y)

Set hole positions using incremental or absolute data.

- Hole-machining data

Z..... Set the distance from R-point to the hole bottom using incremental data, or set the position of the hole bottom using absolute data.

Q..... Set this address code using incremental data. (This address code has different uses according to the type of hole-machining mode selected.)

R..... Set the distance from the initial point of machining to R-point using incremental data, or set the position of R-point using absolute data.

P..... Set the desired time or the number of spindle revolutions, for dwell at the hole bottom. (Set the overlapping length for the chamfering cutter cycles G71.1 and G72.1.)

D..... Set this address code using incremental data. (This address code has different uses according to the type of hole-machining mode selected.)

K..... Set this address code using incremental data. (This address code has different uses according to the type of hole-machining mode selected.)

I..... Set the feed override distance for the tool to be decelerated during the last cutting operation of drilling with a G73, G82, or G83 command code.

J(B)..... For G74 or G84, set the timing of dwell data output; for G75, G76, or G87, set the timing of M3 and M4 output, or; for G73, G82, or G83, set the feed override ratio for deceleration during the last cutting operation.

E..... Set a cutting feed rate (for G77, G79 and G85).

H..... Select synchronous/asynchronous tapping cycle and set the return speed override during a synchronous tapping cycle.

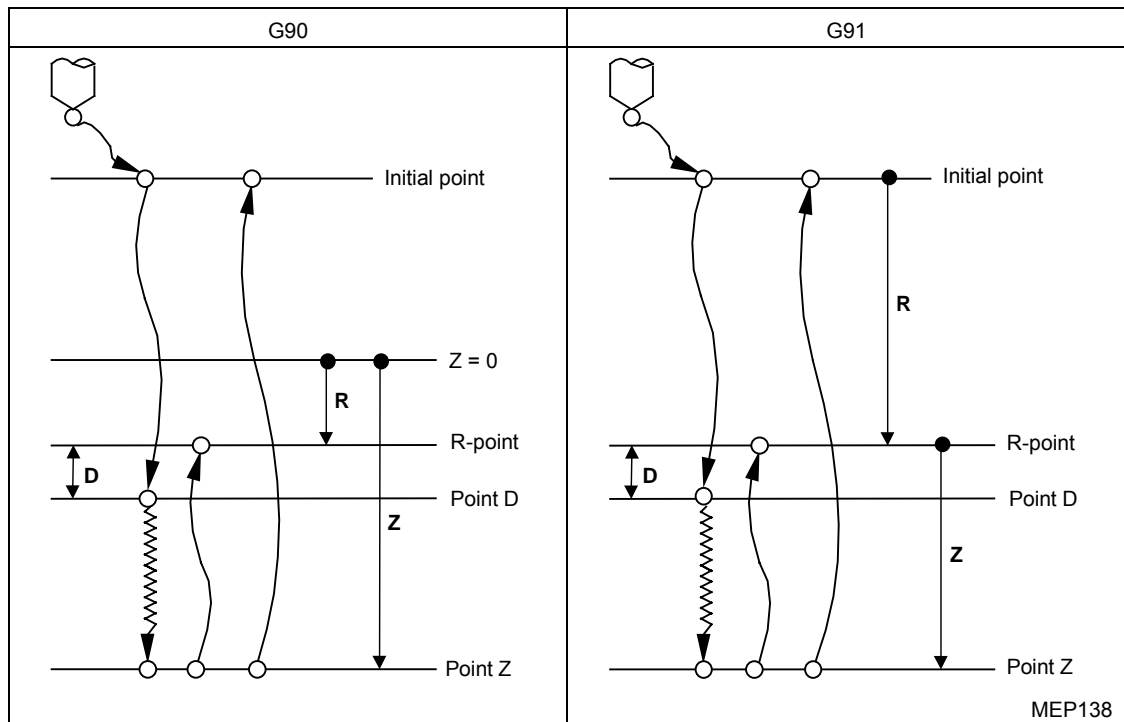
F..... Set a cutting feed rate.

- Repeat times (L)

If no data is set for L, it will be regarded as equal to 1.

If L is set equal to 0, hole-machining will not occur; hole-machining data will only be stored into the memory.

- The differences between the G90 mode data setting method and the G91 mode data setting method are shown in the diagram below.



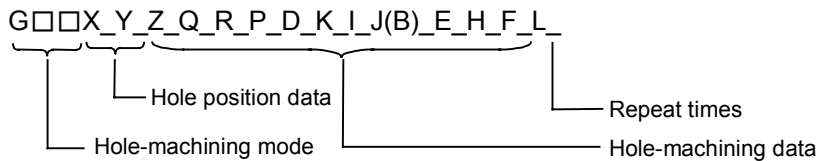
- → : Signifies signed distance data that begins at ●.
- ↔ : Signifies unsigned distance data.

Note 1: The initial point refers to the Z-axis position existing at the moment of the fixed-cycle mode selection.

Note 2: Point D is that at which positioning from R-point can be done further at a rapid feed rate.

2. Programming format

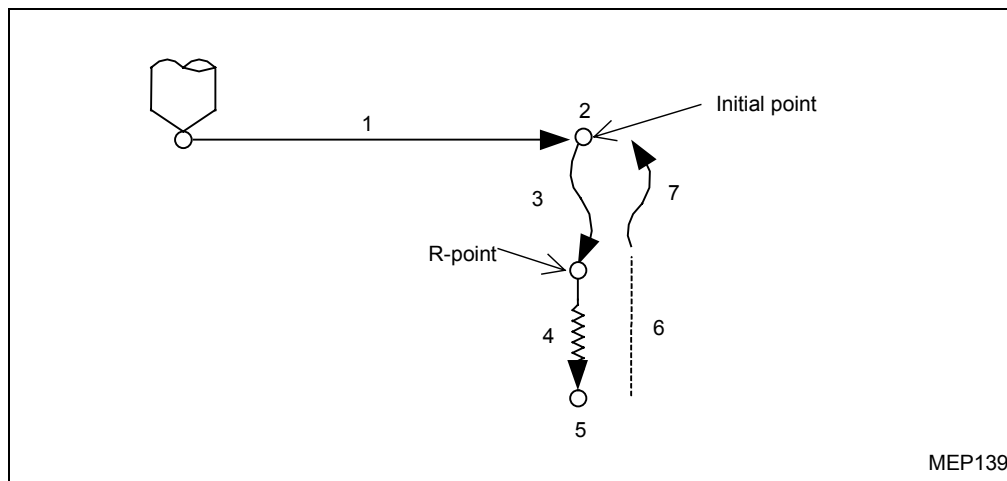
As shown below, the fixed-cycle command consists of a hole-machining mode section, a hole position data section, a hole-machining data section, and a repeat instruction section.



3. Detailed description

1. The hole-machining mode refers to a fixed-cycle mode used for drilling, counterboring, tapping, boring, or other machining operations. Hole position data denotes X- and Y-axis positioning data. Hole-machining data denotes actual machining data. Hole position data and repeat times are non-modal, whereas hole-machining data are modal.
2. If M00 or M01 is set either in the same block as a fixed-cycle command or during the fixed-cycle mode, then the fixed-cycle command will be ignored and then after positioning, M00 or M01 will be outputted. The fixed-cycle command will be executed if either X, Y, Z, or R is set.

3. During fixed-cycle operation, the machine acts in one of the following seven types of manner:
- Action 1 For positioning on the X-, and Y-axes, the machine acts according to the current G-code of group 01 (G02 and G03 will be regarded as G01).
 - Action 2 M19 is sent from the NC unit to the machine at the positioning complete point (initial point) in the G87 mode. After execution of this M-command, the next action will begin. In the single-block operation mode, positioning is followed by block stop.



- Action 3 Positioning to R-point by rapid motion.
- Action 4 Hole-machining by cutting feed.
- Action 5 Depending on the selected fixed-cycle type, spindle stop (M05), spindle reverse rotation (M04), spindle normal rotation (M03), dwell, or tool shift is performed at the hole bottom.
- Action 6 Tool relief to R-point is performed by cutting feed or rapid motion (according to the selected fixed-cycle type).
- Action 7 Return to the initial point is performed by rapid motion.

Whether fixed-cycle mode operation is to be terminated at action 6 or action 7 can be selected with the following G-codes:

G98: Return to the initial point level

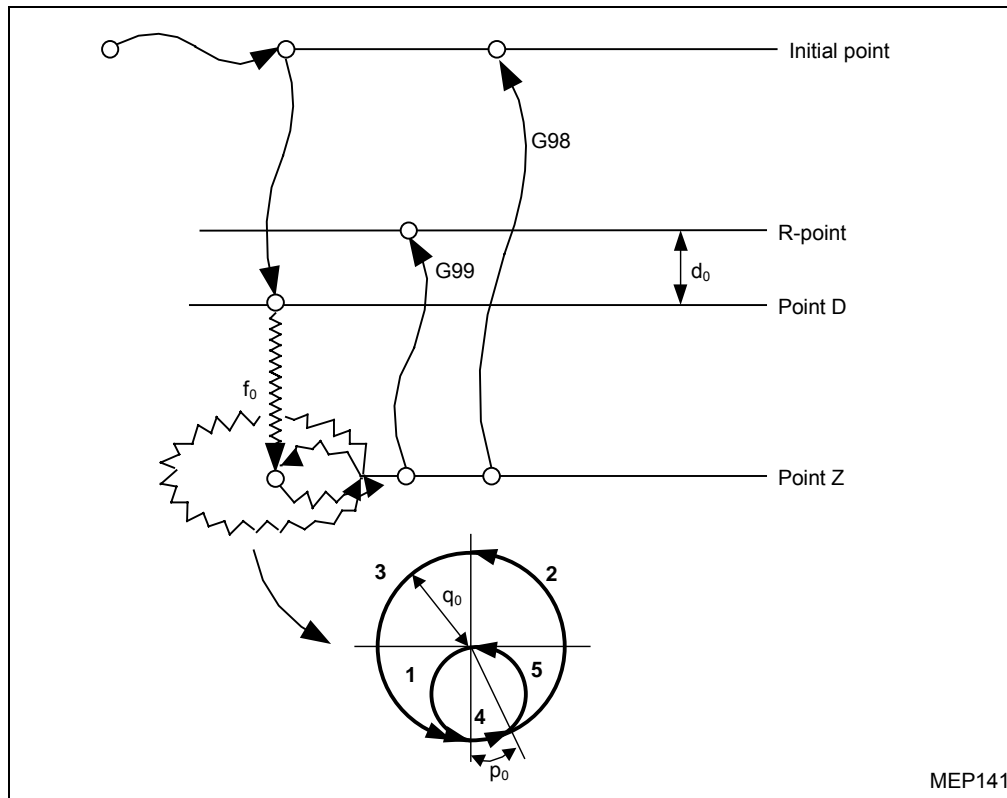
G99: Return to the R-point level

Both commands are modal. Once G98 has been given, for example, the G98 mode remains valid until G99 is given. The G98 mode is the initial state of the NC.

For a block without positioning data, the hole-machining data are only stored into the memory and fixed-cycle operation is not performed.

13-2-4 G72.1 (Chamfering cutter CCW)

G72.1 [Xx Yy] Rr Zz Qq₀ [Pp₀ Dd₀] Ff₀



q_0 : Radius

p_0 : Overlapping length (in arc)

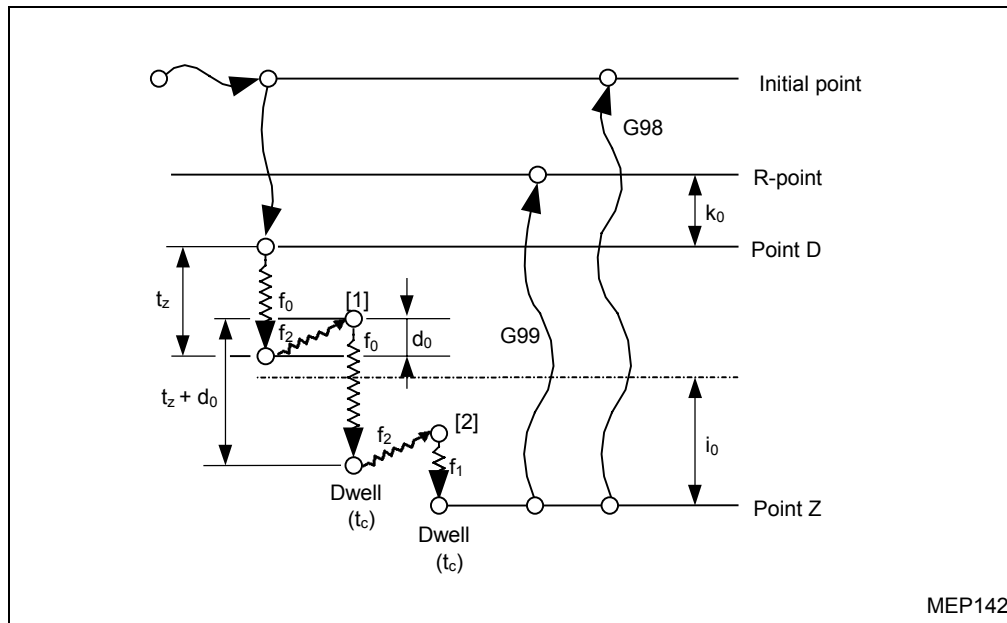
d_0 : Distance from R-point

f_0 : Feed rate

- X, Y, P, and/or D can be omitted.
- Omission of Q or setting "Q0" results in an alarm.

13-2-5 G73 (High-speed deep-hole drilling)

G73 [Xx Yy] Rr Zz Qt_z [Pt_c] Ff₀ [Dd₀ Kk₀ Ii₀ Jj₀(Bb₀)]



t_z : Depth of cut per pass	j_0 : Feed override ratio (%)
t_c : Dwell (in time or No. of revolutions)	(b_0)
d_0 : Return distance	f_0 : Feed rate
k_0 : Distance from R-point to the starting point of cutting feed	f_1 : Feed overridden $f_1 = f_0 \times j_0(b_0)/100$
i_0 : Feed override distance	f_2 : Return speed (fixed)
	Max. speed: 9999 mm/min (for metric spec.)
	999.9 in/min (for inch spec.)

- The feed rate will remain unchanged if either I or J(B) is omitted.
- X, Y, P, D, K, I, and/or J(B) can be omitted.
If D is omitted or set to 0, the machine operates according to the value of parameter **F12**.
- The alarm **809 ILLEGAL NUMBER INPUT** will occur if Q is set to 0.
- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1: Argument J-command
= 0: Argument B-command

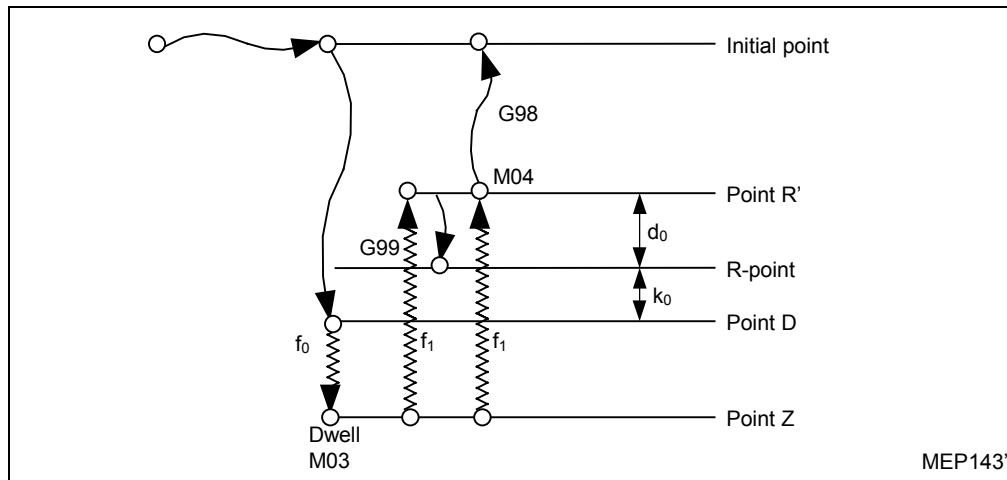
Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

- The feed rate is f_1 only if the starting point of a cutting pass is within the range of i_0 .

Example: In the diagram shown above, during the second cutting operation, since pecking return point [1] falls outside the range of feed override distance i_0 , feeding does not decelerate and cutting is performed at feed rate f_0 ; during the third cutting operation, since pecking return point [2] falls within the range of i_0 , feeding decelerates and cutting is performed at feed rate f_1 .

13-2-6 G74 (Reverse tapping)

G74 [Xx Yy] Rr Zz [Pt_c] Ff₀ [Jj₀(Bb₀) Dd₀ Hh₀ Kk₀]



t_c : Dwell (always in time)

f_0 : Feed rate

j_0 : 1...M03 after dwell at hole bottom

(b_0) 2...M03 before dwell at hole bottom

4...M04 after dwell at R-point

d_0 : Distance from R-point
(Tap lifting distance)

h_0 : Flag for synchronous/asynchronous tapping and
the return speed override (%) for synchronous
tapping

$h_0 = 0$ Asynchronous tapping

$h_0 > 0$ Synchronous tapping

k_0 : Distance from R-point

- X, Y, P, J(B), D, H, and/or K can be omitted.

If, however, J(B) is omitted or set to 0, the setting of J(B) will be regarded as 2.

If H is omitted, synchronous tapping method is automatically selected.

- For synchronous tapping, see Subsection 13-2-21.

- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1: Argument J-command

= 0: Argument B-command

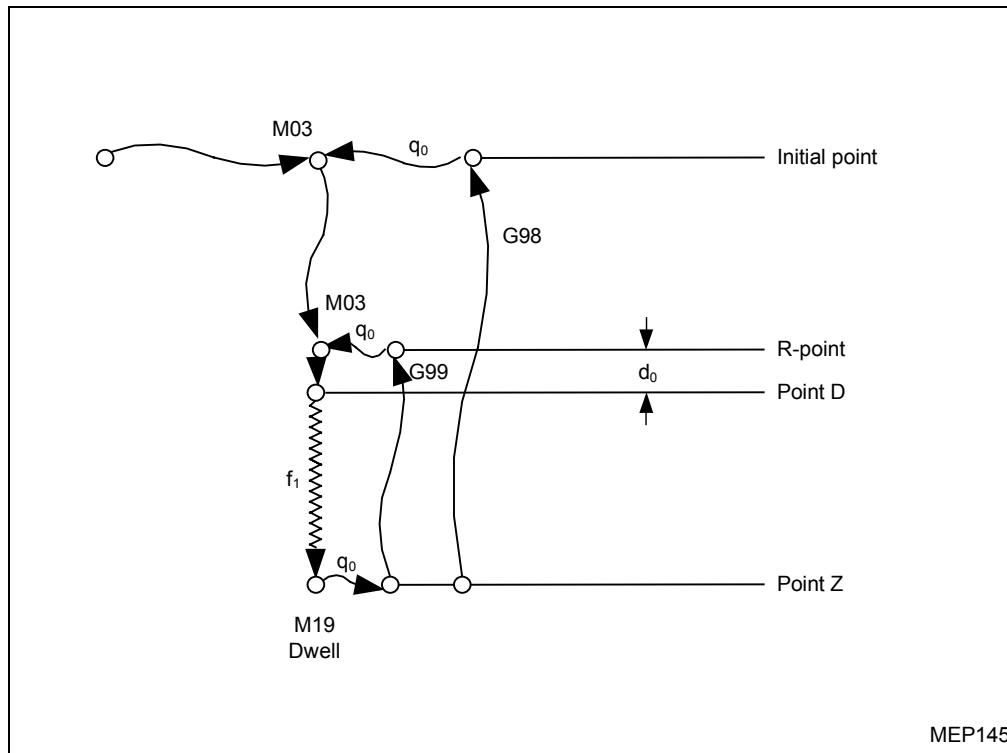
Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

G75 [Xx Yy] Rr Zz [Pt_c Qq₀] Ff₀ [Dd₀ Jj₀(Bb₀) Kk₀ li₀]

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

13-2-8 G76 (Boring)

G76 [Xx Yy] Rr Zz [Pt_c Qq₀] Ff₁ [Dd₀ Jj₀(Bb₀)]



t_c : Dwell (in time or No. of revolutions)

q₀ : Amount of relief on the XY-plane
(Direction determined by bits 3 & 4 of I14)

f₁ : Feed rate

j₀ : 0 or omitted M03 after machining
(b₀) Value except 0 M04 after machining

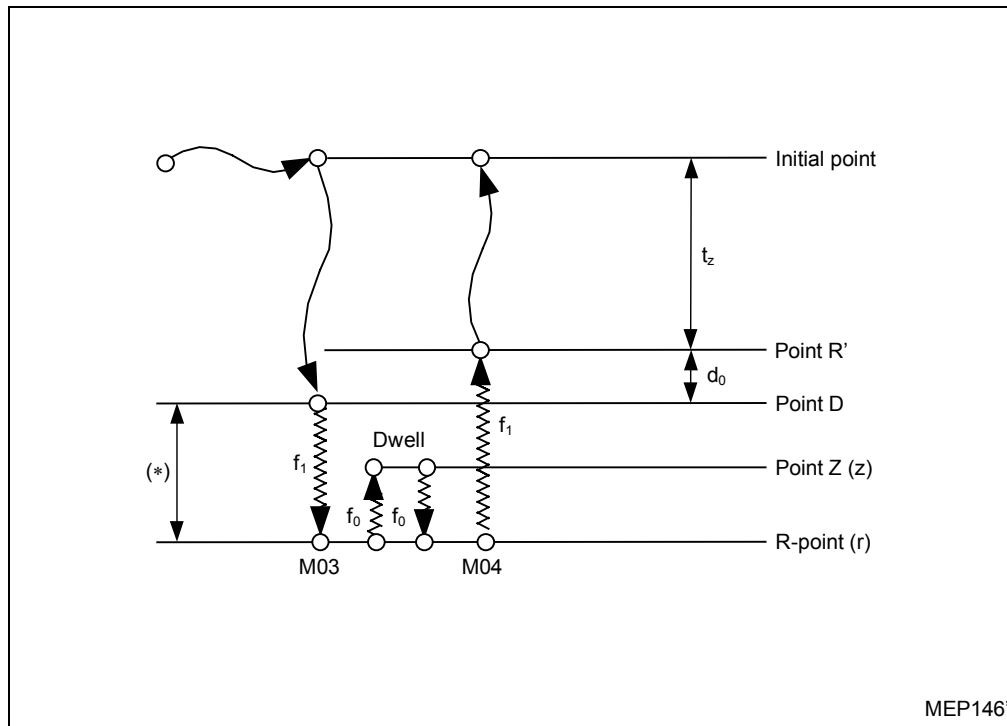
- X, Y, P, Q, D, and/or J(B) can be omitted.
- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1: Argument J-command
= 0: Argument B-command

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

13-2-9 G77 (Back spot facing)

G77 [Xx Yy] Rr Zz [Pt_c Qt_z] Ff₀ [Ef₁ Jj₀(Bb₀) Dd₀]



t_c : Dwell (in time or No. of revolutions) $j_0(b_0)$: Output order of M03 and M04 at hole bottom.
 t_z : Distance from the initial point 0: M03, then M04 (for normal spindle rotation)
 f_0 : Feed rate 0 1: M04, then M03 (for reverse spindle rotation)
 f_1 : Feed rate 1 d_0 : Distance from point R'

- Normally, asynchronous feed (G94) is used for the pass marked with (*). If $f_1 = 0$, or if f_1 is omitted, however, synchronous feed (G95) is used (feed rate = 0.5 mm/rev).
- X, Y, P, Q, E, J (B), and/or D can be omitted.
- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

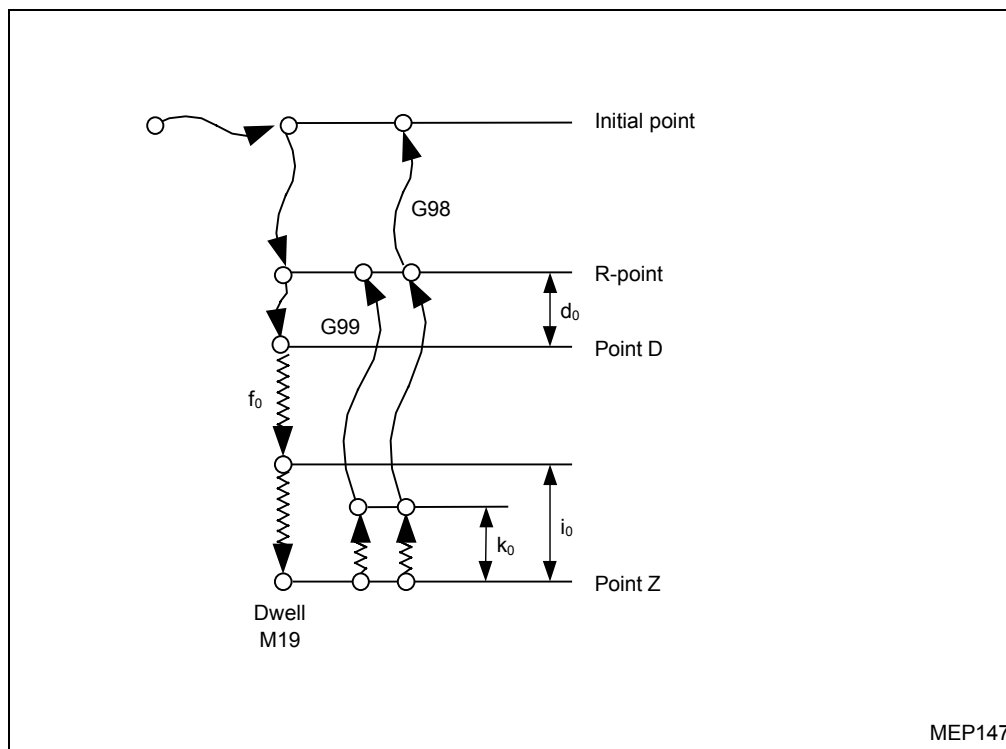
Parameter **F84** bit 1 = 1: Argument J-command
 = 0: Argument B-command

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

- In G91 (incremental data input) mode, the direction of hole machining is automatically determined according to the sign of Z data (the sign of data at address R will be ignored).

13-2-10 G78 (Boring)

G78 [Xx Yy] Rr Zz [Pt_c] Ff₀ [Dd₀ Kk₀ Qi₀]



t_c : Dwell (in time or No. of revolutions)

d_0 : Distance from R-point

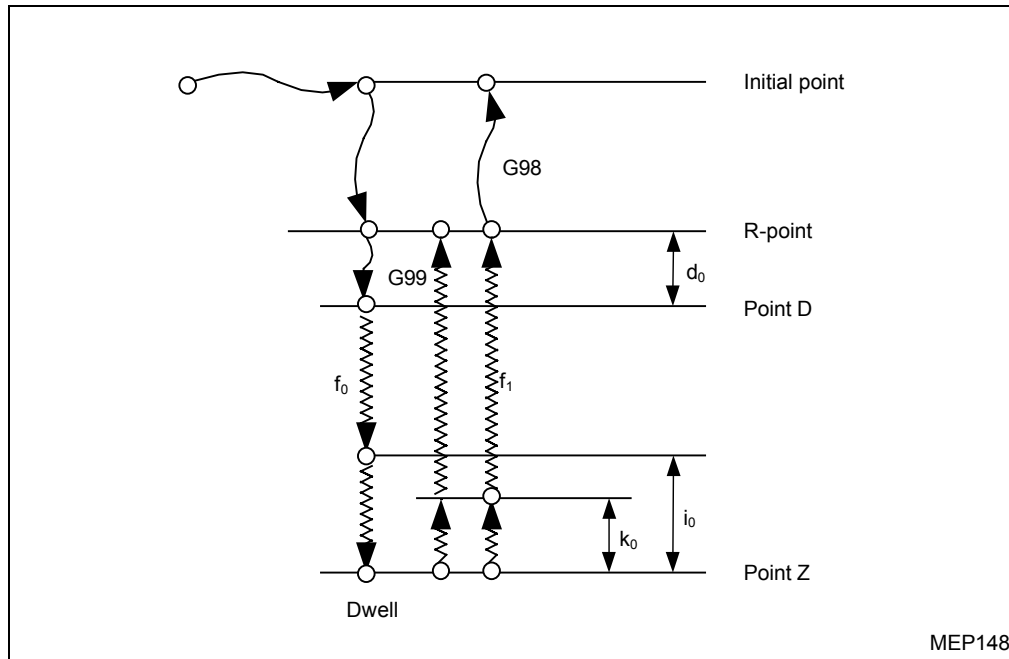
k_0 : Distance from point Z

i_0 : Distance from point Z

- X, Y, P, D, K, and/or Q can be omitted.

13-2-11 G79 (Boring)

G79 [Xx Yy] Rr Zz [Pt_c] Ff₀ [Dd₀ Kk₀ Qi₀ Ef₁]

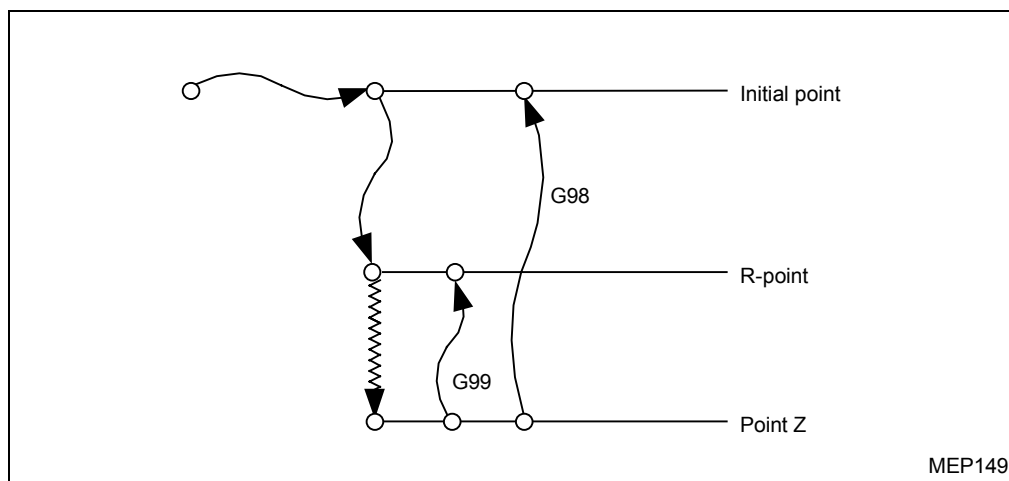


t_c : Dwell (in time or No. of revolutions) k_0 : Distance from point Z
 f_0 : Feed rate 0 i_0 : Distance from point Z
 d_0 : Distance from R-point f_1 : Feed rate 1

- Asynchronous feed is used for f_1 .
 If, however, f_1 is set equal to 0 or is not set, then the tool is fed at the setting of f_0 .
- X, Y, P, D, K, Q, and/or E can be omitted.

13-2-12 G81 (Spot drilling)

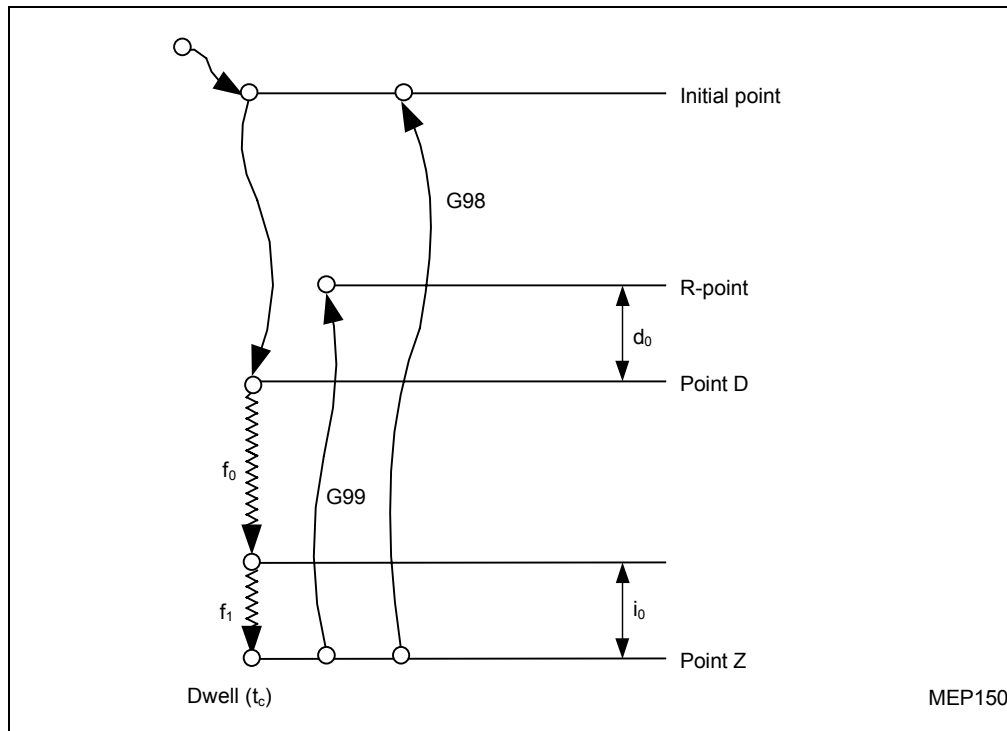
G81 [Xx Yy] Rr Zz



- X and/or Y can be omitted.

13-2-13 G82 (Drilling)

G82 [Xx Yy] Rr Zz [Pt_c] Ff₀ [Dd₀ Ii₀ Jj₀(Bb₀)]



t_c : Dwell (in time or No. of revolutions)	j_0 : Feed override ratio (%)
d_0 : Distance from R-point to the starting point of cutting feed	(b_0)
i_0 : Feed override distance	f_0 : Feed rate
	f_1 : Feed overridden $f_1 = f_0 \times j_0(b_0)/100$

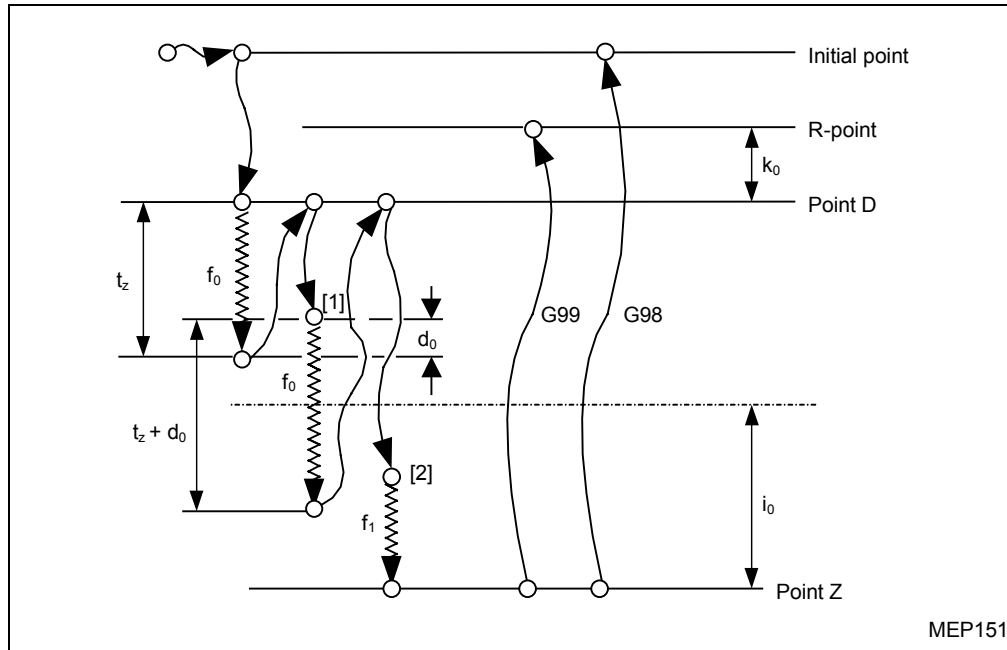
- The feed rate will remain unchanged if either I or J(B) is omitted.
- X, Y, P, D, I, and/or J(B) can be omitted.
- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1 : Argument J-command
 = 0 : Argument B-command

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

13-2-14 G83 (Deep-hole drilling)

G83 [Xx Yy] Rr Zz Qt_z Ff₀ [Dd₀ Kk₀ Ii₀ Jj₀(Bb₀)]



t_z : Depth of cut per pass
 d_0 : Rapid motion stopping allowance
 k_0 : Distance from R-point to the starting point of cutting feed
 i_0 : Feed override distance
 j_0 : Feed override ratio (%) (b_0)
 f_0 : Feed rate
 f_1 : Feed overridden $f_1 = f_0 \times j_0(b_0)/100$

- The feed rate will remain unchanged if either I or J(B) is omitted.
- X, Y, P, D, K, I, and/or J(B) can be omitted.
If D is omitted or set to 0, the machine will operate according to the value of parameter **F13**.
- The alarm **809 ILLEGAL NUMBER INPUT** will occur if Q is set to 0.
- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1: Argument J-command
 = 0: Argument B-command

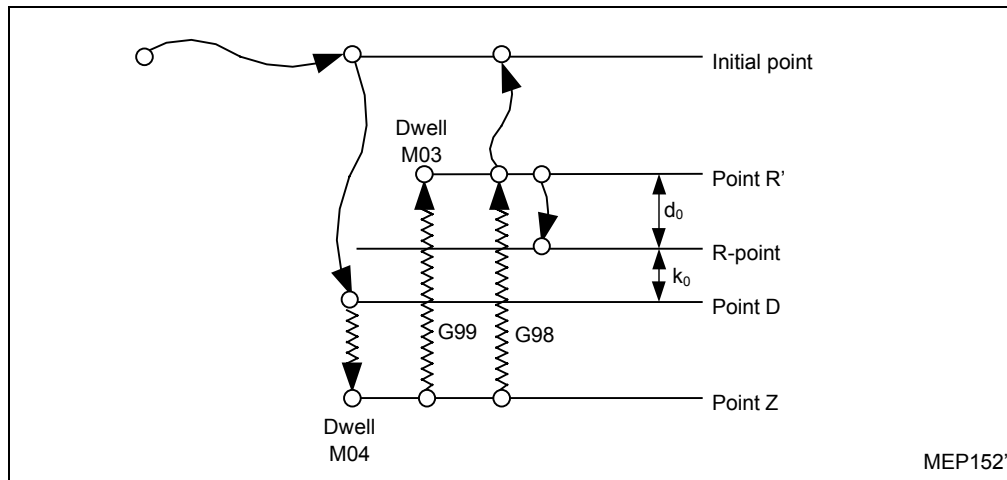
Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

- The feed rate is f_1 only if the starting point of a cutting pass is within the range of i_0 .

Example: In the diagram shown above, during the second cutting operation, since rapid feed positioning point [1] falls outside the range of feed override distance i_0 , feeding does not decelerate and cutting is performed at feed rate f_0 ; during the third cutting operation, since rapid feed positioning point [2] falls within the range of i_0 , feeding decelerates and cutting is performed at feed rate f_1 .

13-2-15 G84 (Tapping)

G84 [Xx Yy] Rr Zz [Pt_c] Ff₀ [Jj₀(Bb₀) Dd₀ Hh₀ Kk₀]



t_c : Dwell (always in time)

f_0 : Feed rate

j_0 : 1...M04 after dwell at hole bottom

(b₀) 2...M04 before dwell at hole bottom

4...M03 after dwell at R-point

d_0 : Distance from R-point
(Tap lifting distance)

h_0 : Flag for synchronous/asynchronous tapping and
the return speed override (%) for synchronous
tapping

$h_0 = 0$ Asynchronous tapping

$h_0 > 0$ Synchronous tapping

k_0 : Distance from R-point

- X, Y, P, J(B), D, H, and/or K can be omitted.

If, however, J(B) is omitted or set to 0, the setting of J(B) will be regarded as 2.

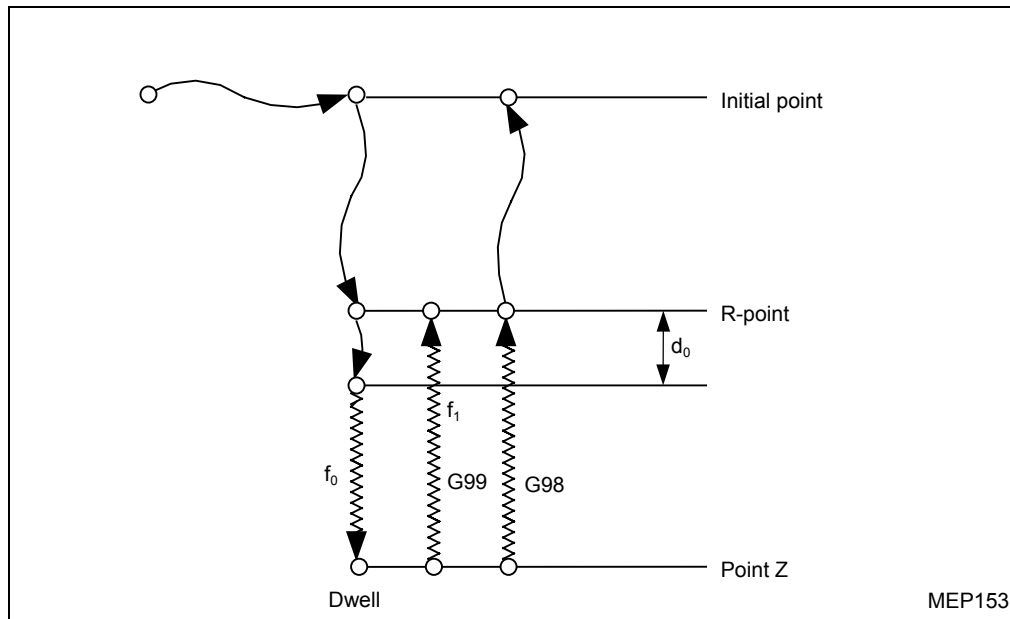
If H is omitted, synchronous tapping method is automatically selected.

- For synchronous tapping, see Subsection 13-2-21.

- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

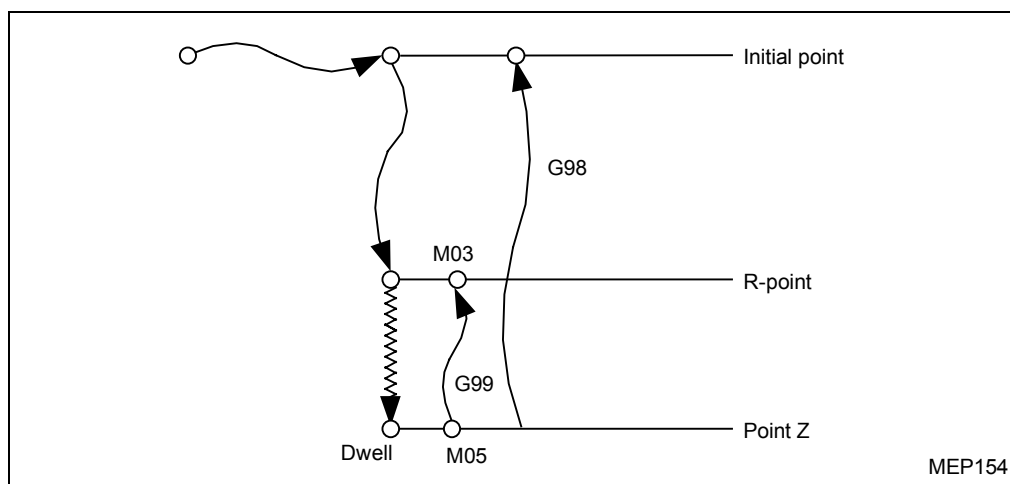
Parameter **F84** bit 1 = 1 : Argument J-command
= 0 : Argument B-command

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

13-2-16 G85 (Reaming)G85 [Xx Yy] Rr Zz [Pt_z] Ff₀ [Ef₁ Dd₀]

t_z : Dwell (in time or No. of revolutions) f_1 : Feed rate 1
 f_0 : Feed rate 0 d_0 : Distance from R-point

- Asynchronous feed is used for f_1 .
 If, however, f_1 is set equal to 0 or is not set, then the tool is fed at the setting of f_0 .
- X, Y, P, E, and/or D can be omitted.

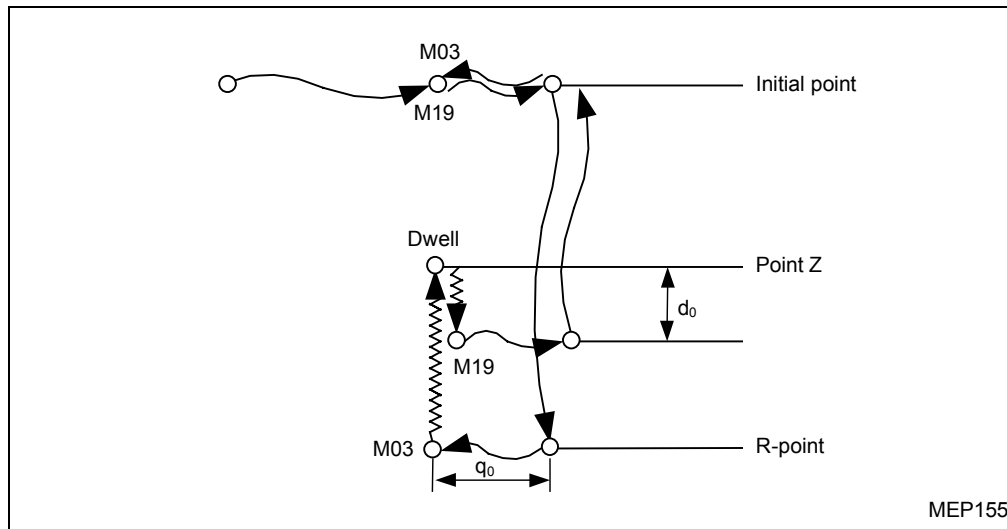
13-2-17 G86 (Boring)G86 [Xx Yy] Rr Zz [Pt_c]

t_c : Dwell (in time or No. of revolutions)

- X, Y, and/or P can be omitted.

13-2-18 G87 (Back boring)

G87 [Xx Yy] Rr Zz [Pt_c Qq₀] Ff₀ [Dd₀ Jj₀(Bb₀)]



t_c : Dwell (in time or No. of revolutions)

q₀ : Amount of relief on the XY-plane
(Direction determined by bits 3 & 4 of I14)

f₀ : Feed rate

d₀ : Distance from point Z

j₀ : 0 or omitted ······ M03 at R-point

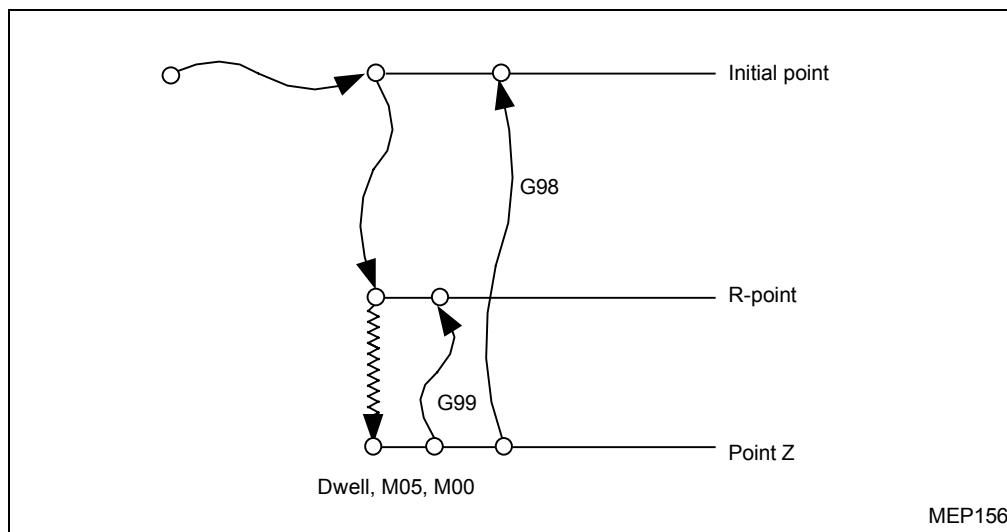
(b₀) Value except 0 ······ M04 at R-point

- X, Y, P, Q, D, and/or J(B) can be omitted.
- Initial-point return is always used for G87 (even if the current modal is of G99).
- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

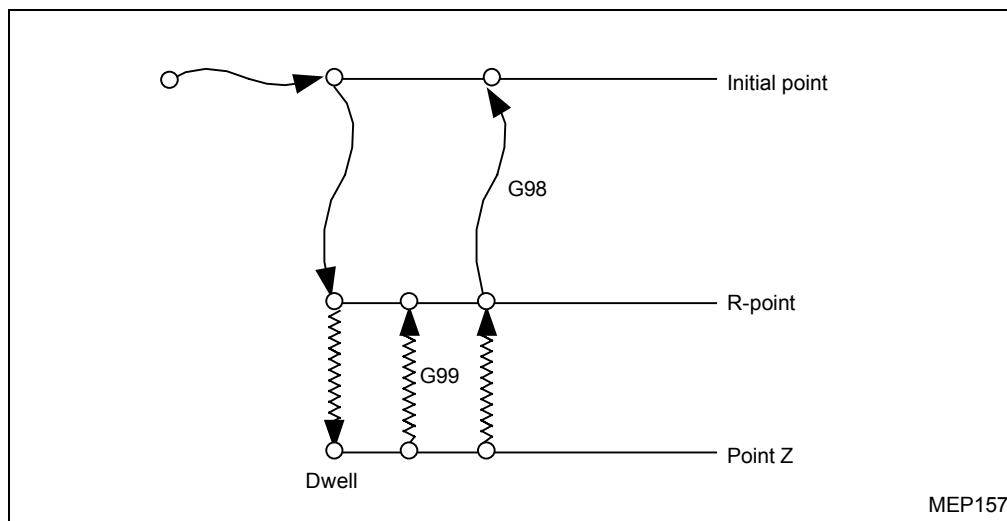
Parameter **F84** bit 1 = 1: Argument J-command
= 0: Argument B-command

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

- In G91 (incremental data input) mode, the direction of hole machining is automatically determined according to the sign of Z data (the sign of data at address R will be ignored).

13-2-19 G88 (Boring)G88 [Xx Yy] Rr Zz [Pt_c]t_c : Dwell (in time or No. of revolutions)

- X, Y, and/or P can be omitted.
- At the hole bottom, M05 and M00 are outputted.

13-2-20 G89 (Boring)G89 [Xx Yy] Rr Zz [Pt_c]t_c : Dwell (in time or No. of revolutions)

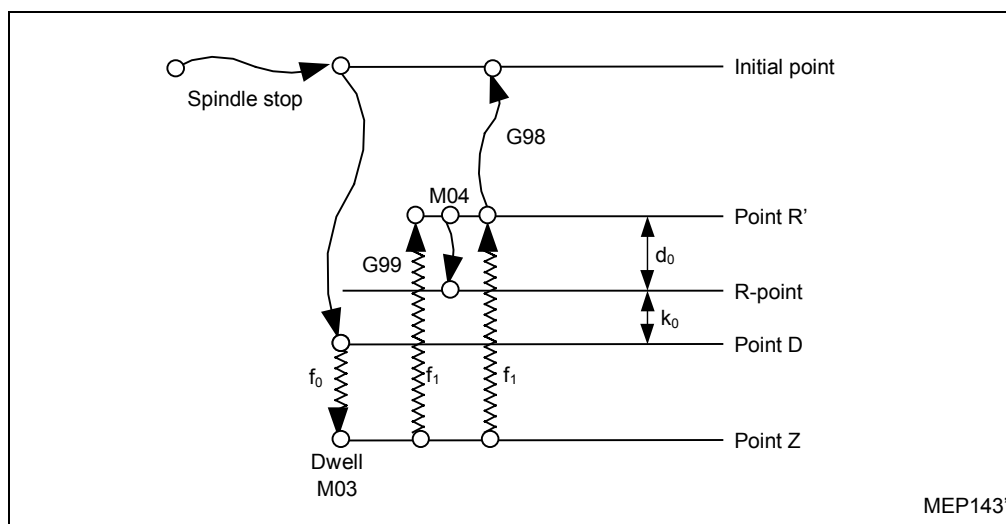
- X, Y, and/or P can be omitted.

13-2-21 Synchronous tapping (Option)

In an EIA/ISO program, synchronous tapping can be selected by additionally setting data at the address H in the tapping cycle block of G74 or G84. Address H is used to select a synchronous/asynchronous tapping and to designate the override of return speed. Special preparatory functions G84.2 and G84.3 are also provided for both types of synchronous tapping.

1. G74 [Reverse tapping]

G74 [Xx Yy] Rr Zz [Pt_c] Ff₀ [Jj₀(Bb₀) Dd₀ Hh₀ Kk₀]



t_c : Dwell (always in time)

f_0 : Feed rate

(Set the pitch for synchronous tapping)

j_0 : 1...M03 after dwell at hole bottom

(b₀) 2...M03 before dwell at hole bottom

4...M04 after dwell at R-point

d_0 : Distance from R-point (Tap lifting distance)

h_0 : Return speed override (%)

$h_0 = 0$ Asynchronous tapping

$h_0 \geq 1$ Synchronous tapping

k_0 : Distance from R-point

- X, Y, P, J(B), D, H, and/or K can be omitted.

If, however, J(B) is omitted or set to 0, the setting of J(B) will be regarded as 2.

If H is omitted, synchronous tapping method is automatically selected.

- H is used to select whether synchronous tapping cycle operation or asynchronous tapping cycle operation is to be performed using a machine capable of synchronous tapping. This code is also used to override the return speed for synchronous tapping cycle operation.

As for a machine not capable of synchronous tapping, explicitly enter 'H0' to select asynchronous tapping method. Otherwise, i.e. omission or specification with an effective value for synchronous tapping method will cause an alarm (**952 NO SYNCHRONIZED TAP OPTION**).

- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1: Argument J-command

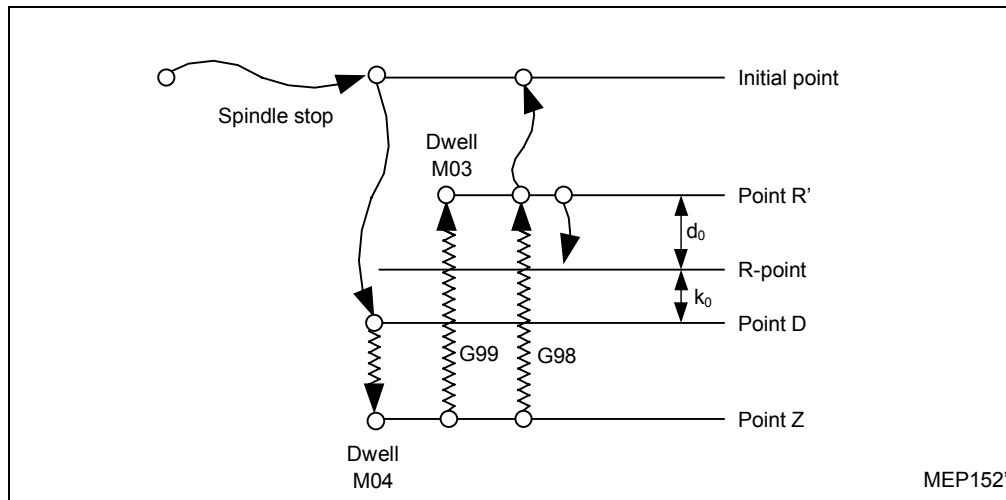
= 0: Argument B-command

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

- During gear selection for tapping, due consideration must be given to ensure the minimum spindle acceleration/deceleration time. Refer to the machine-operating manual.

2. G84 (Normal tapping)

G84 [Xx Yy] Rr Zz [Pt_c] Ff₀ [Jj₀(Bb₀) Dd₀ Hh₀ Kk₀]



t_c : Dwell (always in time)

f₀ : Feed rate

(Set the pitch for synchronous tapping)

j₀ : 1...M04 after dwell at hole bottom

(b₀) 2...M04 before dwell at hole bottom

4...M03 after dwell at R-point

d₀ : Distance from R-point (Tap lifting distance)

h₀ : Return speed override (%)

h₀ = 0 Asynchronous tapping

h₀ ≥ 1 Synchronous tapping

k₀ : Distance from R-point

- X, Y, P, J(B), D, H, and/or K can be omitted.

If, however, J(B) is omitted or set to 0, the setting of J(B) will be regarded as 2.

If H is omitted, synchronous tapping method is automatically selected.

- H is used to select whether synchronous tapping cycle operation or asynchronous tapping cycle operation is to be performed using a machine capable of synchronous tapping. This code is also used to override the return speed for synchronous tapping cycle operation.

As for a machine not capable of synchronous tapping, explicitly enter 'H0' to select asynchronous tapping method. Otherwise, i.e. omission or specification with an effective value for synchronous tapping method will cause an alarm (**952 NO SYNCHRONIZED TAP OPTION**).

- Whether argument J or B is to be used depends on the value that has been set in bit 1 of parameter **F84**.

Parameter **F84** bit 1 = 1: Argument J-command

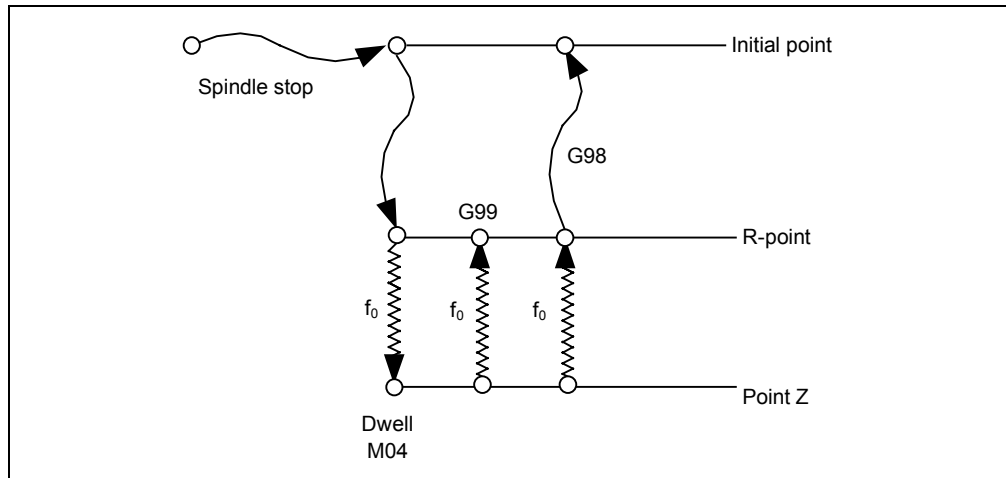
= 0: Argument B-command

Note: For a horizontal machining center, if the value of bit 1 of parameter **F84** is 1 (argument J-command), setting a B-command will cause the table to rotate. Be careful in that case to ensure no interference between the workpiece and the tool.

- During gear selection for tapping, due consideration must be given to ensure the minimum spindle acceleration/deceleration time. Refer to the machine-operating manual.

3. G84.2 (Normal tapping)

G84.2 [Xx Yy] Rr Zz [Pt_c] Ff₀



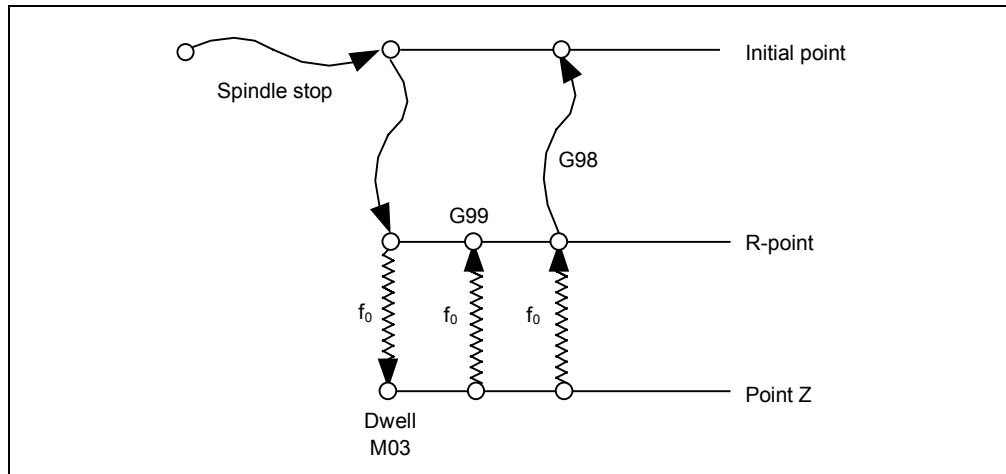
t_c : Dwell (in time) at point Z and upon return to R-point

f₀ : Feed rate (in pitch)

- X, Y, and/or P can be omitted.
- Designation of G84.2 or G84.3 without the corresponding option would cause the alarm No. **952 NO SYNCHRONIZED TAP OPTION**.
- During gear selection for tapping, due consideration must be given to ensure the minimum spindle acceleration/deceleration time. Refer to the machine-operating manual.
- The value of parameter **K90** is always referred to as the return speed override (%).

4. G84.3 (Reverse tapping)

G84.3 [Xx Yy] Rr Zz [Pt_c] Ff₀



t_c : Dwell (in time) at point Z and upon return to R-point

f₀ : Feed rate (in pitch)

- X, Y, and/or P can be omitted.
- Designation of G84.2 or G84.3 without the corresponding option would cause the alarm No. **952 NO SYNCHRONIZED TAP OPTION**.
- During gear selection for tapping, due consideration must be given to ensure the minimum spindle acceleration/deceleration time. Refer to the machine-operating manual.
- The value of parameter **K90** is always referred to as the return speed override (%).

13-3 Suppression of Single-Block Stop for Fixed Cycles

13-3-1 Function description

This function allows point-machining (i.e. hole-machining) operation to be checked efficiently in the mode of single-block operation by reducing the number of start button operations (in other words, the frequency of operation stop).

When this suppression function is valid, the single-block stop is reduced to the following three positions:

1. Completion position of positioning,
2. Completion position of approaching to R-point,
3. Completion position of successive hole-machining operations and return.

Remark 1: Use the following parameter to make this function valid or invalid:

Suppression of single-block stop for fixed cycles:

F87 bit 4 = 1 (Valid) / 0 (Invalid)

Remark 2: This function is available for both EIA/ISO and MAZATROL program types.

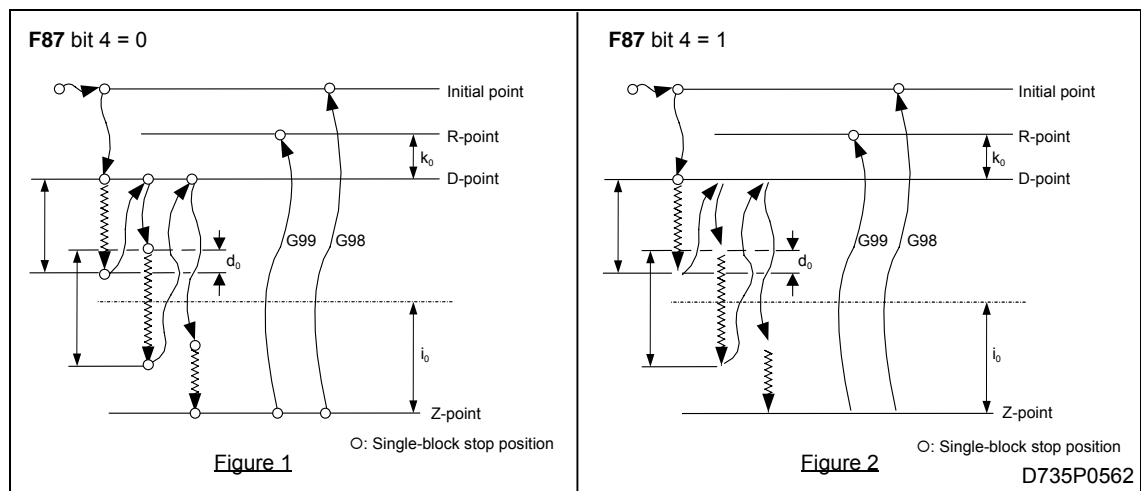
13-3-2 Examples of operation

The changes made by this function to the single-block operation are described below taking the deep-hole drilling cycle (G83) as an example.

G83 [Xx Yy] Rr Zz Ff Qq₀ [Dd₀ Kk₀ Pt_c li₀ Hh₀]

In the deep-hole drilling cycle, the tool is to return repeatedly after drilling through a certain depth in order to avoid an increase in load due to clogging with chips. As shown in Figure 1, there are a number of positions at which the operation stops in the machining cycle.

The number of stopping positions can be reduced as in Figure 2 by the single-block stop suppression.



13-4 Initial Point and R-Point Level Return: G98 and G99

1. Function and purpose

Commands G98 or G99 can be used to select whether the return level of the final sequence during fixed-cycle operation is to be set at R-point or at the initial point of machining.

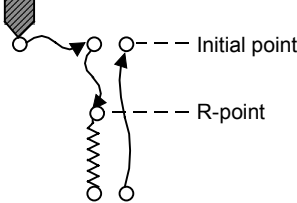
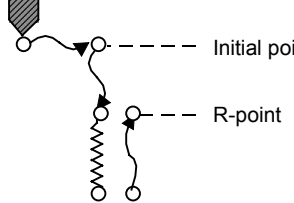
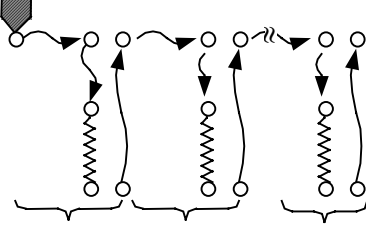
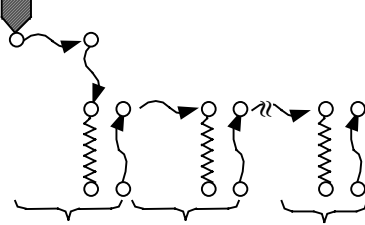
2. Programming format

G98: Initial point level return

G99: R-point level return

3. Detailed description

The following represents the relationship between the G98/G99 mode and repeat times:

Number of holes	Sample program	G98 (At power-on or after cancellation using M02, M30, or RESET key)	G99
Only one	G81 X100. Y100. Z-50. R25. F1000	 <p>Initial point R-point Return to initial point level.</p>	 <p>Initial point R-point Return to R-point level.</p>
Two or more	G81 X100. Y100. Z-50. R25. L5 F1000	 <p>1st hole 2nd hole Last hole Always return to initial point.</p>	 <p>1st hole 2nd hole Last hole MEP158</p>

13-5 Scaling ON/OFF: G51/G50

1. Function and purpose

The shape specified in a machining program can be enlarged or reduced in size using scaling command G51. The range of scaling (enlargement/reduction) factors is from 0.000001 to 99.999999.

Use command G51 to specify a scaling axis, the center of scaling, and a scaling factor.

Use command G50 to specify scaling cancellation.

2. Programming format

G51 Xx Yy Zz Pp Scaling on (specify a scaling axis, the center of scaling (incremental/absolute), and a scaling factor)

G50 Scaling cancel

3. Detailed description

A. Specifying a scaling axis

The scaling mode is set automatically by setting G51. Command G51 does not move any axis; it only specifies a scaling axis, the center of scaling, and a scaling factor.

Scaling becomes valid only for the axis to which the center of scaling has been specified.

Center of scaling

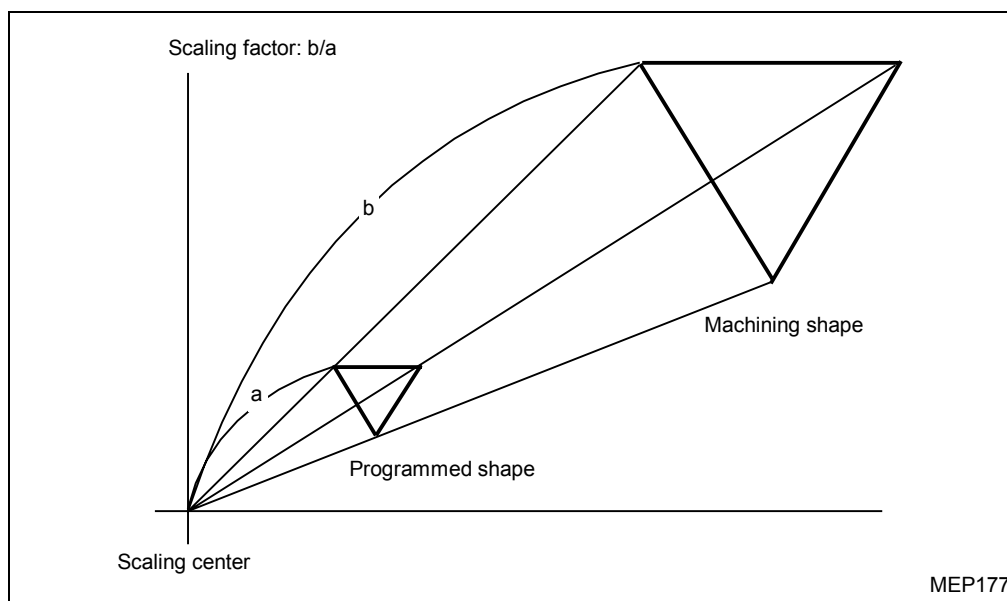
The center of scaling must be specified with the axis address according to the absolute or incremental data command mode (G90 or G91). This also applies even when specifying the current position as the center.

Scaling factor

Use address P to specify a scaling factor.

Minimum unit of specification: 0.000001

Specifiable range of factors: 1 to 99999999 or 0.000001 to 99.999999 (times)
(Although both are valid, the latter with a decimal point must be preceded by G51.)



The scaling factor set in parameter **F20** will be used if you do not specify any scaling factor in the same block as that of G51. The current setting of this parameter will be used if it is updated during the scaling mode. That is, the parameter setting existing when G51 is set is valid. Data will be calculated at a scaling factor of 1 if neither the program nor the parameter has a specified scaling factor.

Alarms occur in the following cases:

- If scaling is specified for a machine not capable of scaling (Alarm **872 G51 OPTION NOT FOUND**)
- If a scaling factor exceeding its maximum available value is specified in the same block as that of G51 (Alarm **809 ILLEGAL NUMBER INPUT**) (All scaling factors less than 0.000001 are processed as 1.)

B. Cancellation of scaling

The scaling cancel mode is set automatically by setting G50. Setting this command code offsets any deviation between the program coordinates and the coordinates of the actual machine position. Even for axes that have not been designated in the same block as that of G50, the machine moves through the offset amount specified by scaling.

4. Precautions

1. Scaling does not become valid for tool radius compensation, tool length offsetting, or tool position offsetting. Offsets and other corrections are calculated only for the shape existing after scaling.
2. Scaling is valid only for move commands associated with automatic operation (tape, memory, or MDI); it is not valid for manual movement.
3. After-scaling coordinates are displayed as position data.
4. Scaling is performed on the axis for which the center of scaling is specified by G51. In that case, scaling becomes valid for all move commands associated with automatic operation, as well as for the parameter-set return strokes of G73 and G83 and for the shift strokes of G76 and G87.
5. If only one axis of the plane concerned is selected for scaling, circular interpolation is performed with the single scaling on that axis.
6. Scaling will be cancelled if either M02, M30, or M00 (only when M0 contains reset) is issued during the scaling mode. Scaling is also cancelled by an external reset command or any other reset functions during the reset/initial status.
7. Data P, which specifies a scaling factor, can use a decimal point. The decimal point, however, becomes valid only if scaling command code G51 precedes data P.

```
G51P0.5      0.5 time
P0.5G51      1 time (regarded as P = 0)
P500000G51   0.5 time
G51P500000   0.5 time
```

8. The center of scaling is shifted accordingly if the coordinate system is shifted using commands G92 or G52 during scaling.

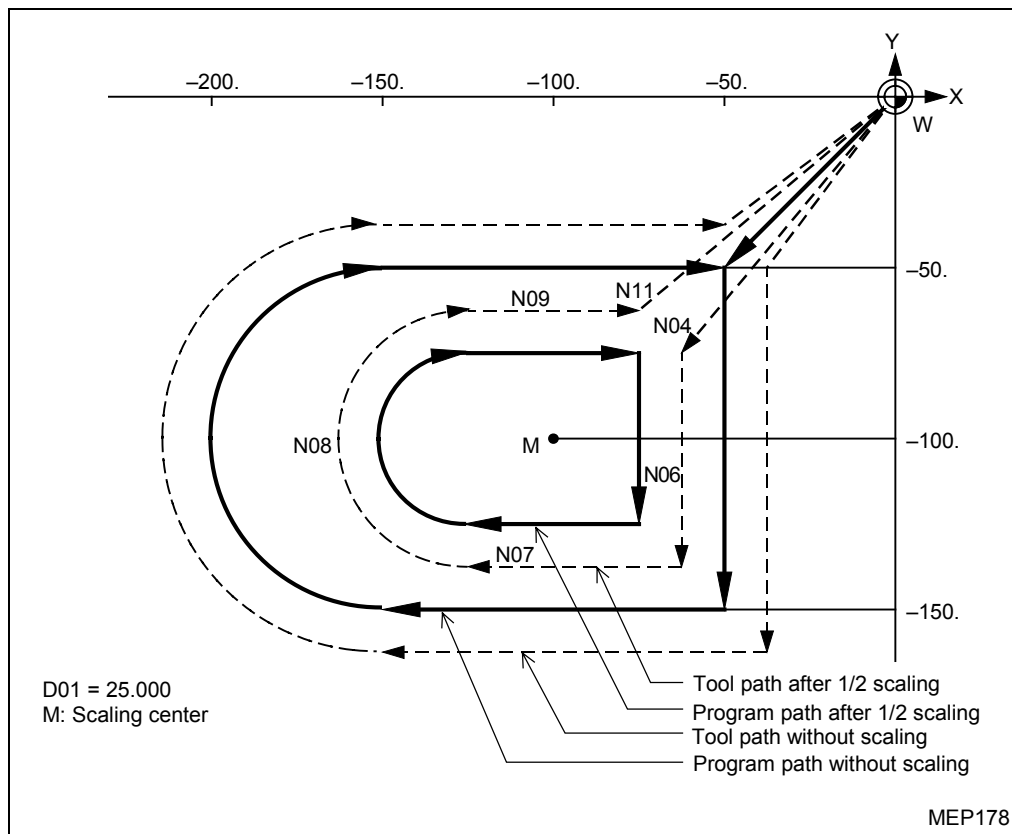
5. Sample programs

1. Basic operation I

```

N01 G92X0Y0Z0
N02 G90G51X-100.Y-100.P0.5
N03 G00G43Z-200.H02
N04 G41X-50.Y-50.D01
N05 G01Z-250.F1000
N06 Y-150.F200
N07 X-150.
N08 G02Y-50.J50.
N09 G01X-50.
N10 G00Z0
N11 G40G50X0Y0
N12 M02

```



2. Basic operation II

```

N01 G92X0Y0
N02 G90G51P0.5 ..... See [1] to [4] below.
N03 G00X-50.Y-50.
N04 G01X-150.F1000
N05 Y-150.
N06 X-50.
N07 Y-50.
N08 G00G50
N09 M02

```

[1] Without scaling

N02 G90G51P0.5

[2] If scaling is to be done for X, Y

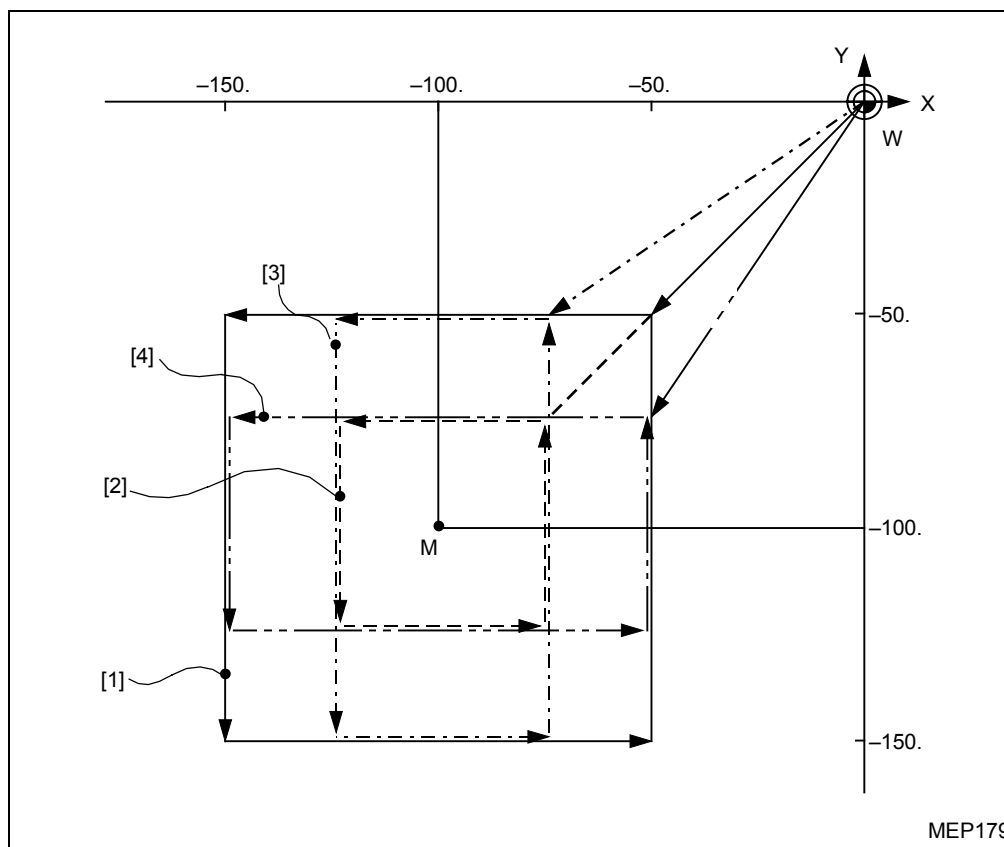
N02 G90G51X-100.Y-100.P0.5

[3] If scaling is to be done for X only

N02 G90G51X-100.P0.5

[4] If scaling is to be done for Y only

N02 G90G51Y-100.P0.5



MEP179

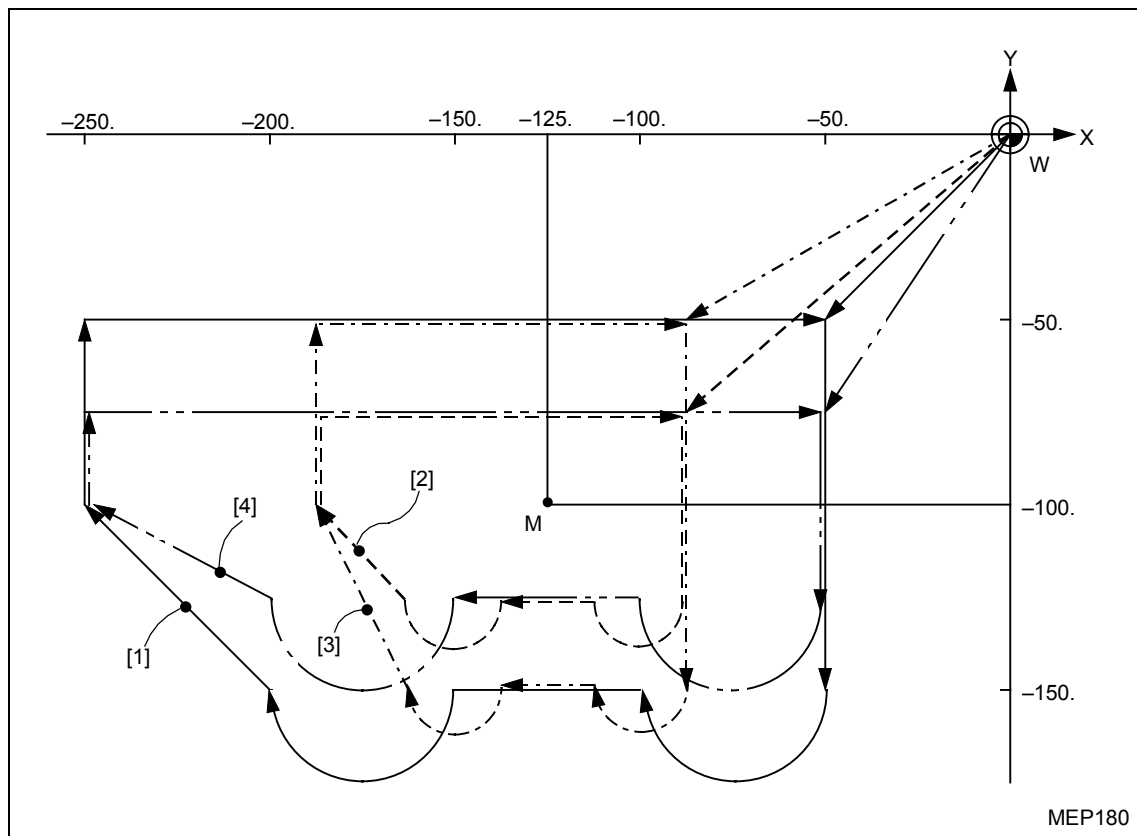
3. Basic operation III

```

N01 G92X0Y0
N02 G90G51P0.5 ..... See [1] to [4] below.
N03 G00X-50.Y-50.
N04 G01Y-150.F1000
N05 G02X-100.I-25.
N06 G01X-150.
N07 G02X-200.I-25.
N08 G01X-250.Y-100.
N09 Y-50.
N10 X-50.
N11 G00G50
N12 M02

```

- [1] Without scaling N02 G90G51P0.5
- [2] If scaling is to be done for X, Y N02 G90G51X-125.Y-100.P0.5
- [3] If scaling is to be done for X only N02 G90G51X-125.P0.5
- [4] If scaling is to be done for Y only N02 G90G51Y-100.P0.5



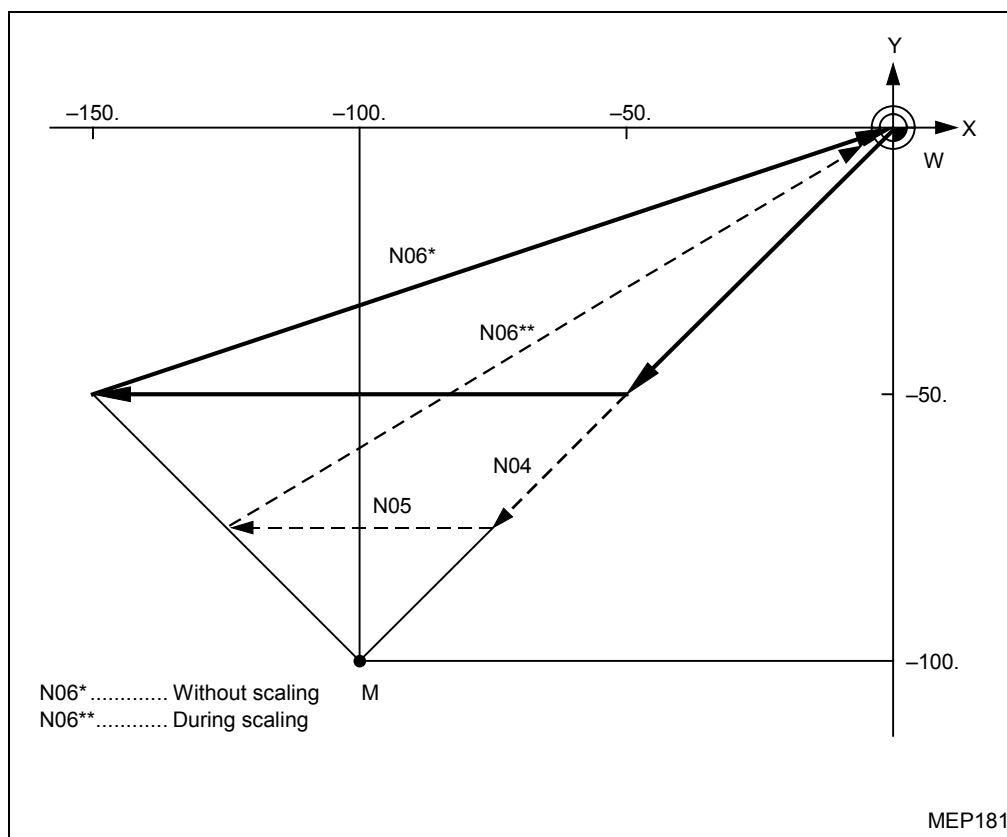
4. Reference-point (zero point) check (G27) during scaling

Setting G27 during scaling cancels the scaling mode after G27 has been executed.

```

N01 G28X0Y0
N02 G92X0Y0
N03 G90G51X-100.Y-100.P0.5
N04 G00X-50.Y-50.
N05 G01X-150.F1000
N06 G27X0Y0
      :
```

If a program is constructed in the manner that the reference point is reached under normal mode, it will also be reached even under scaling mode.

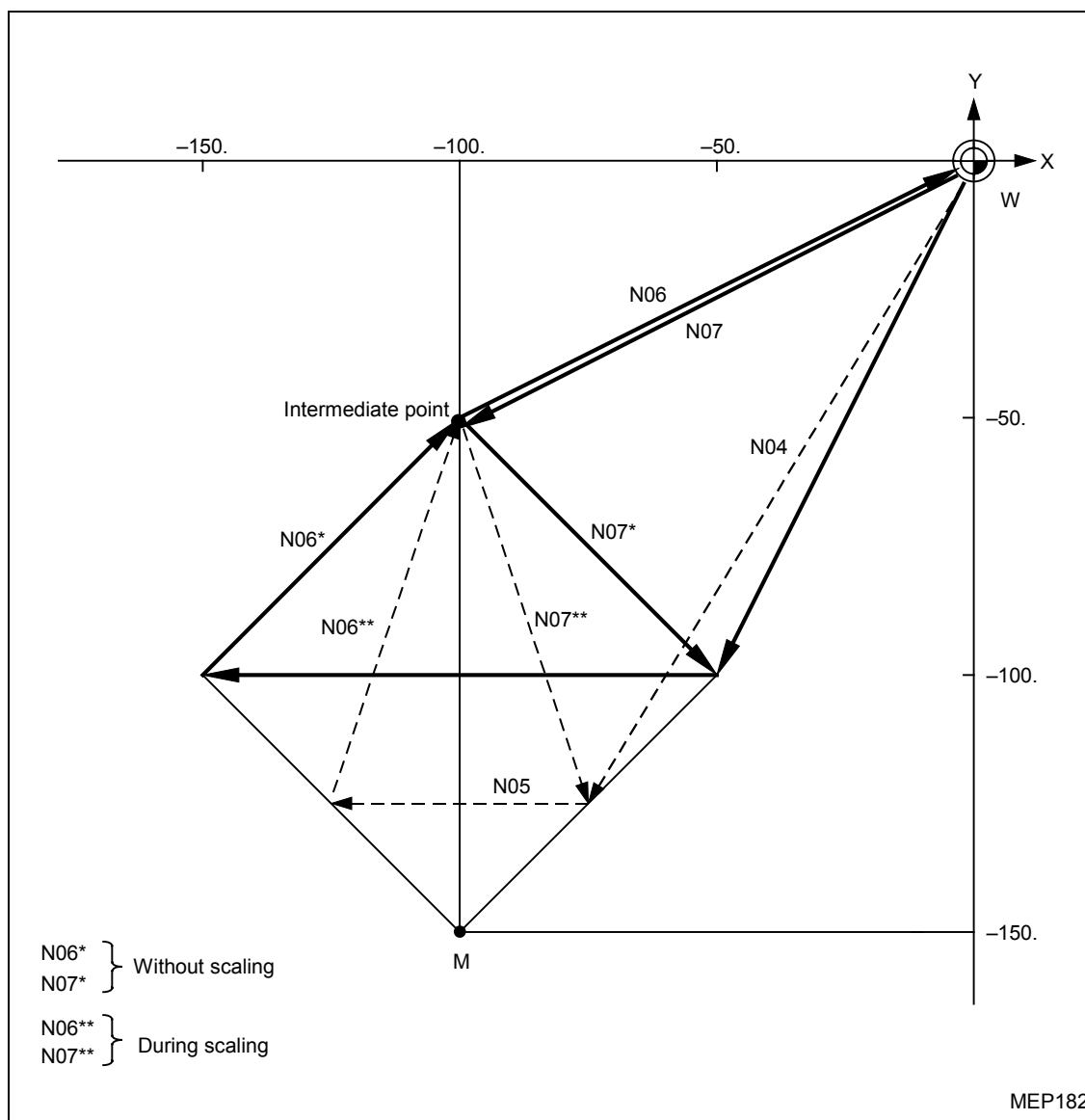


- Reference-point (zero point) return (G28, G29, or G30) during scaling

Setting G28 or G30 during scaling cancels the scaling mode at the middle point and then executes the reference-point (zero point) return command. If the middle point has not been set, the reference-point (zero point) return command is executed with the point where scaling has been cancelled as middle point.

If G29 is set during the scaling mode, scaling will be performed for the entire movement after the middle point.

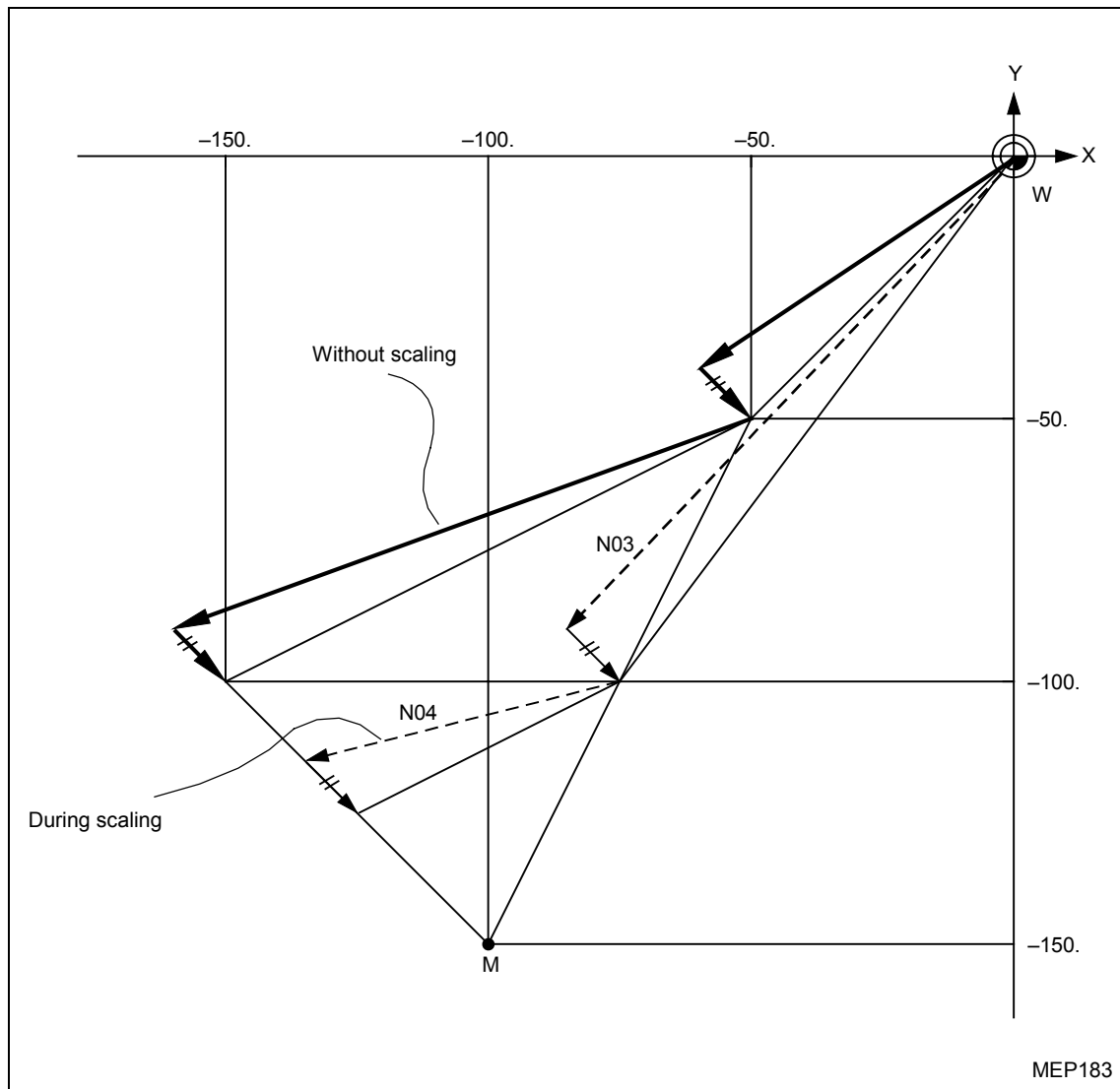
N01	G28X0Y0	
N02	G92X0Y0	
N03	G90G51X-100.Y-150.P500000	
N04	G00X-50.Y-100.	0.5
N05	G01X-150.F1000	
N06	G28X-100.Y-50.	
N07	G29X-50.Y-100.	



6. One-way positioning (G60) during scaling

Setting G60 during the scaling mode executes scaling at the final point of positioning, and thus no scaling is performed for the parameter **I1** of creeping. That is, the amount of creeping remains constant, irrespective of whether scaling is valid.

```
N01 G92X0Y0  
N02 G91G51X-100.Y-150.P0.5  
N03 G60X-50.Y-50.  
N04 G60X-150.Y-100.
```



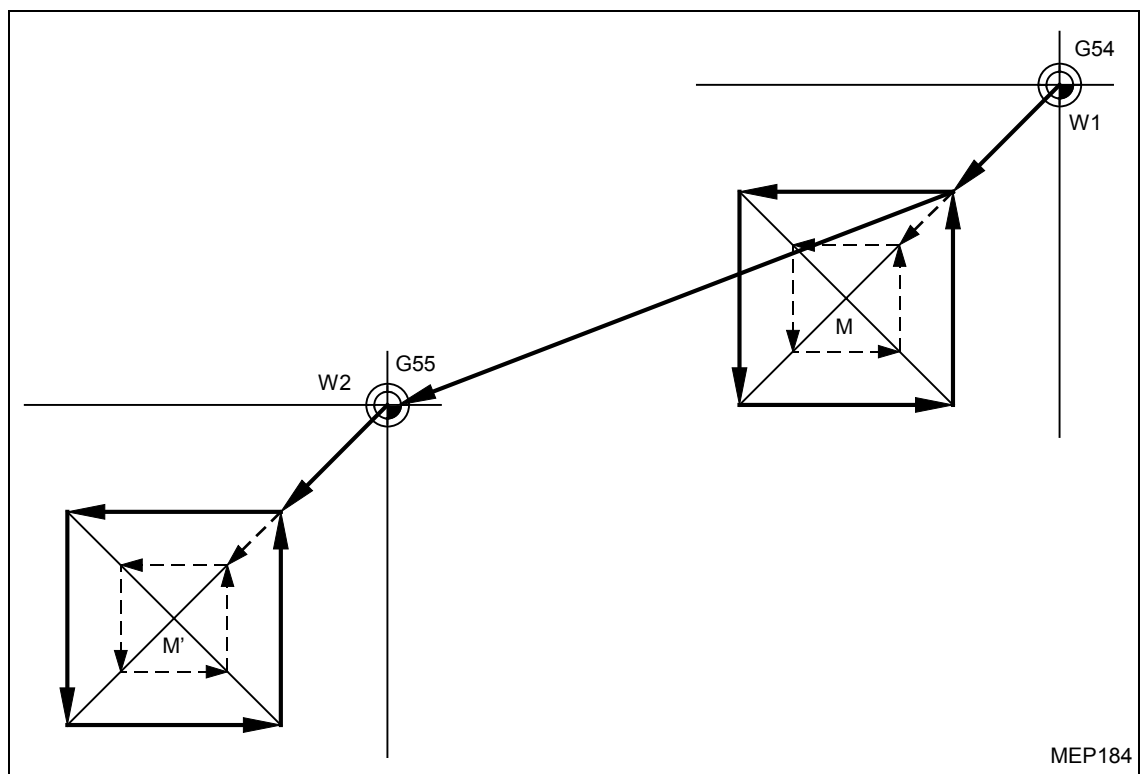
7. Workpiece coordinate system updating during scaling

Updating of the workpiece coordinate system during scaling causes the center of scaling to be shifted according to the difference in offset amount between the new workpiece coordinate system and the old one.

Subprogram

```

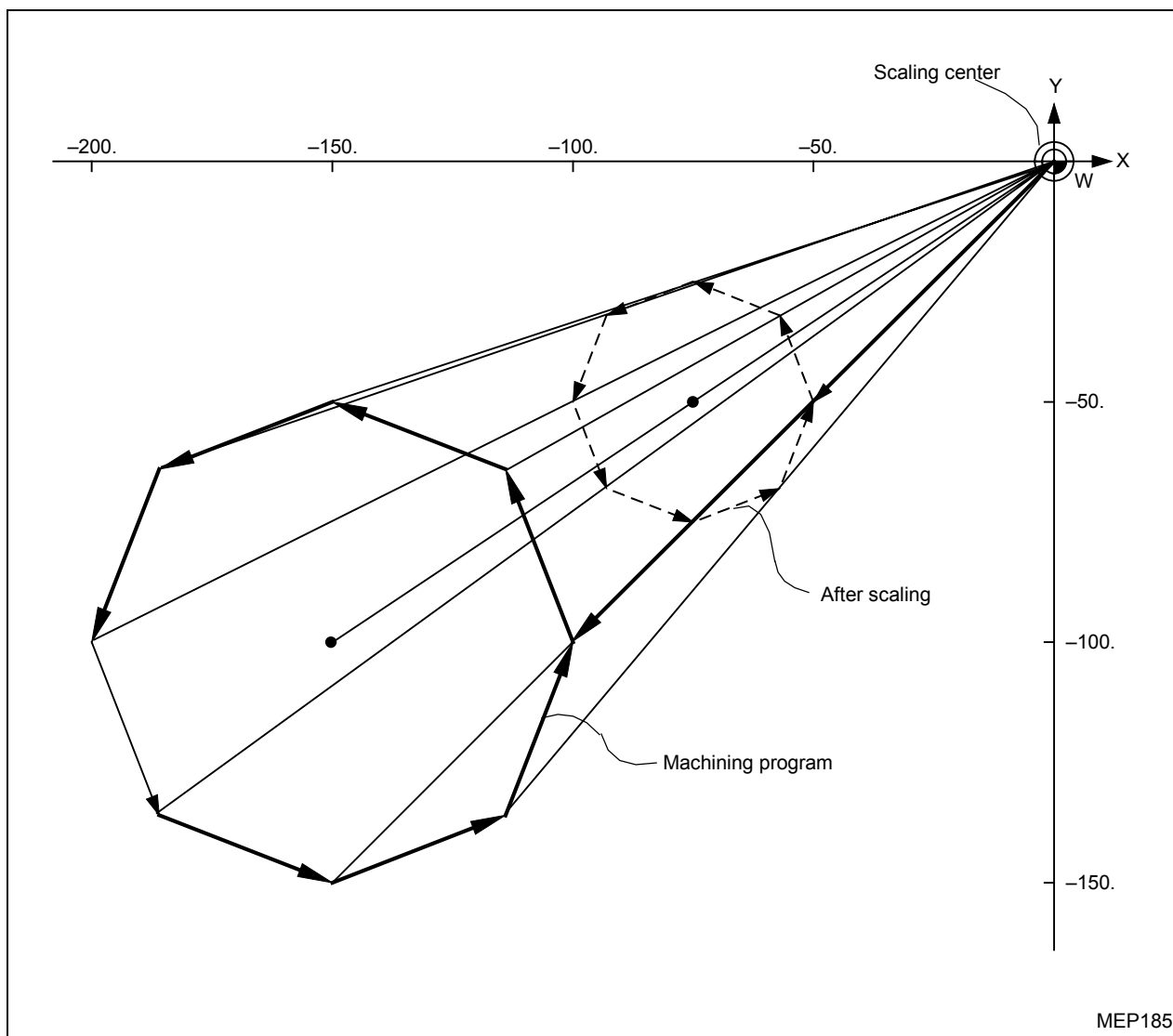
N01 G90G54G00X0Y0      O100
N02 G51X-100.Y-100.P0.5  G00X-50.Y-50.
N03 G65P100             G01X-150.F1000
N04 G90G55G00X0Y0      Y-150.
N05 G65P100             X-50.
                        Y-50.
M99
%
```



8. Figure rotation during scaling

Setting a figure rotate command during scaling executes scaling for both the center and radius of rotation of the figure.

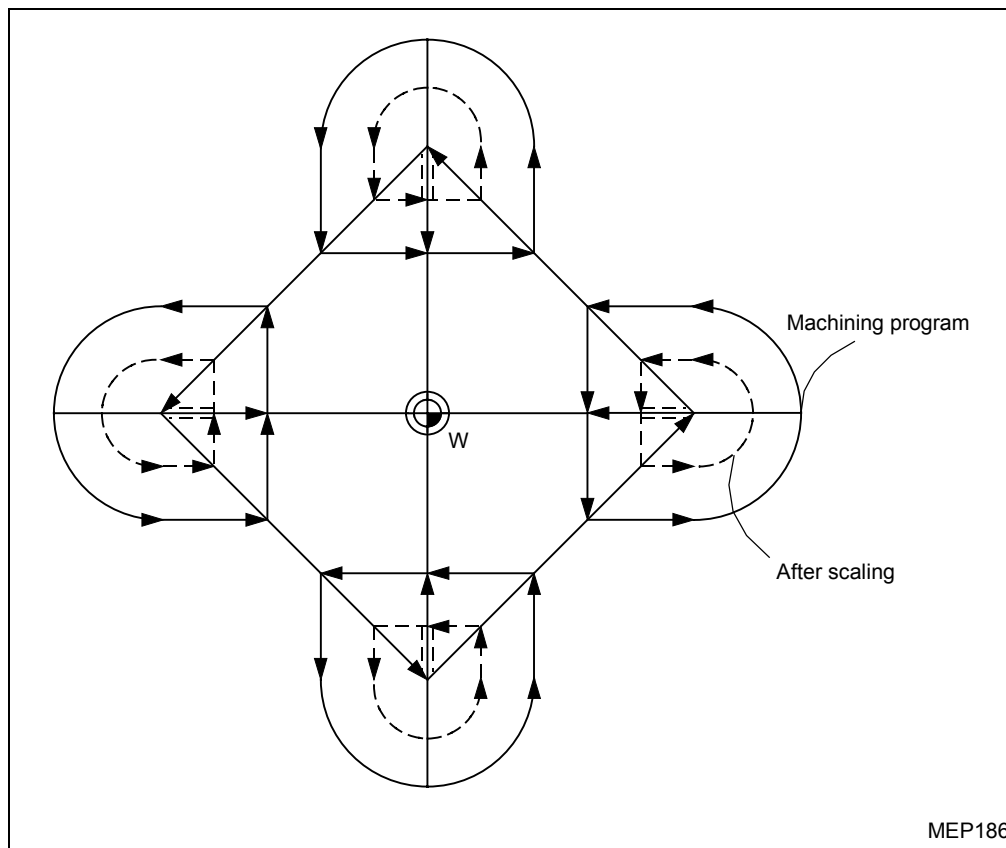
	Subprogram
N01 G92X0Y0	O200
N02 G90G51X0Y0P0.5	G91G01X-14.645Y35.355F1000
N03 G00X-100.Y-100.	M99
N04 M98P200I-50.L8	%



9. Scaling using a figure rotation subprogram

Setting a scaling command in a figure rotation subprogram executes scaling only for the shape predefined in the subprogram. Scaling is not executed for the radius of rotation of the figure.

	Subprogram
G92X0Y0	O300
G90G00X100.	G91G51X0Y0P0.5
M98P300I-100.L4	G00X-40.
G90G00X0Y0	G01Y-40.F1000
M02	X40.
	G03Y80.J40.
	G01X-40.
	Y-40.
	G00G50X40.
	X-100.Y100.
	M99
	%



MEP186

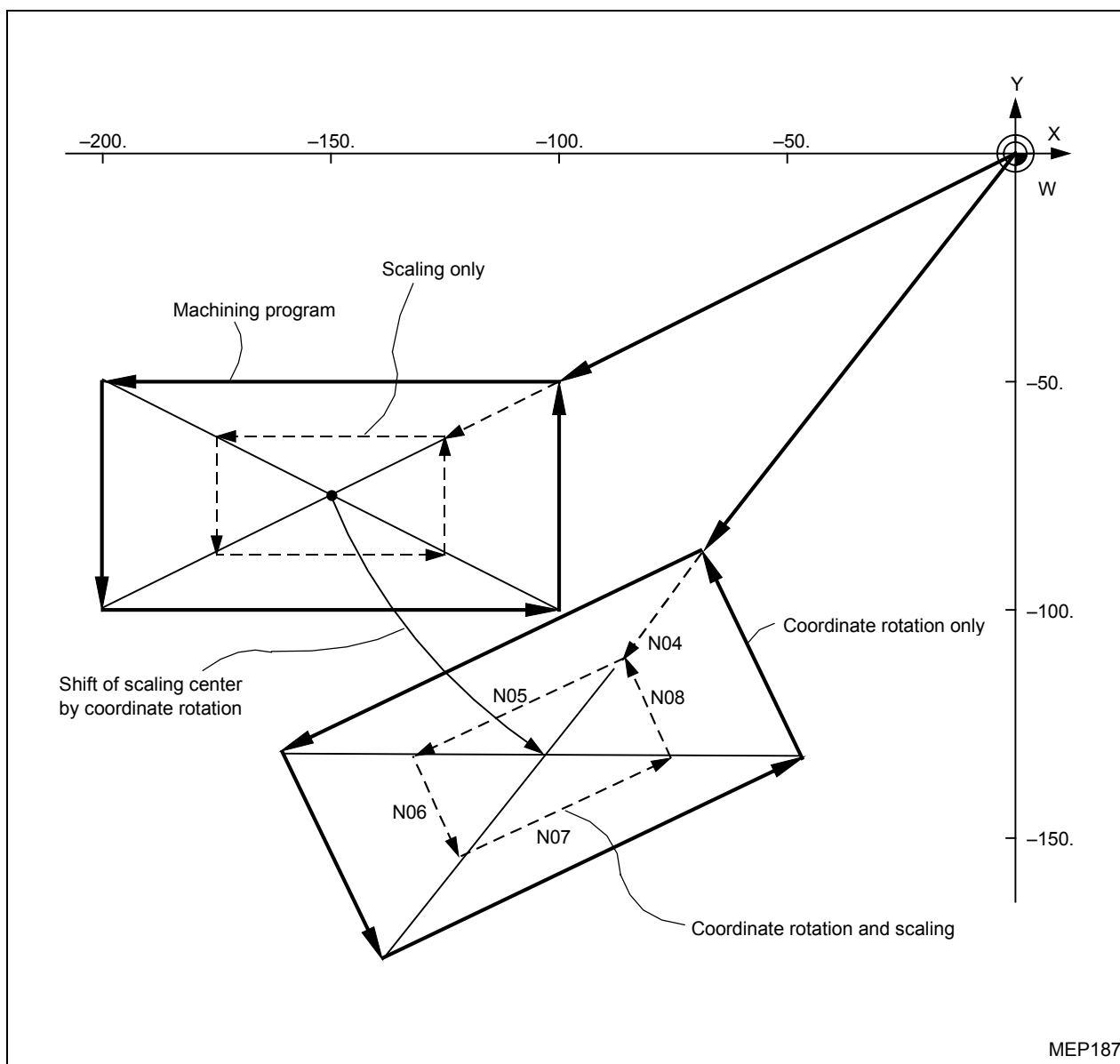
10. Scaling during coordinate rotation

If scaling during coordinate rotation is programmed the center of scaling will rotate and scaling will be performed at that rotated center of scaling.

```

N01 G92X0Y0
N02 M00                                     (Coordinate rotation data setting)
N03 G90G51X-150.Y-75.P0.5
N04 G00X-100.Y-50,
N05 G01X-200.F1000
N06 Y-100.
N07 X-100.
N08 Y-50.
N09 G00G50X0Y0

```



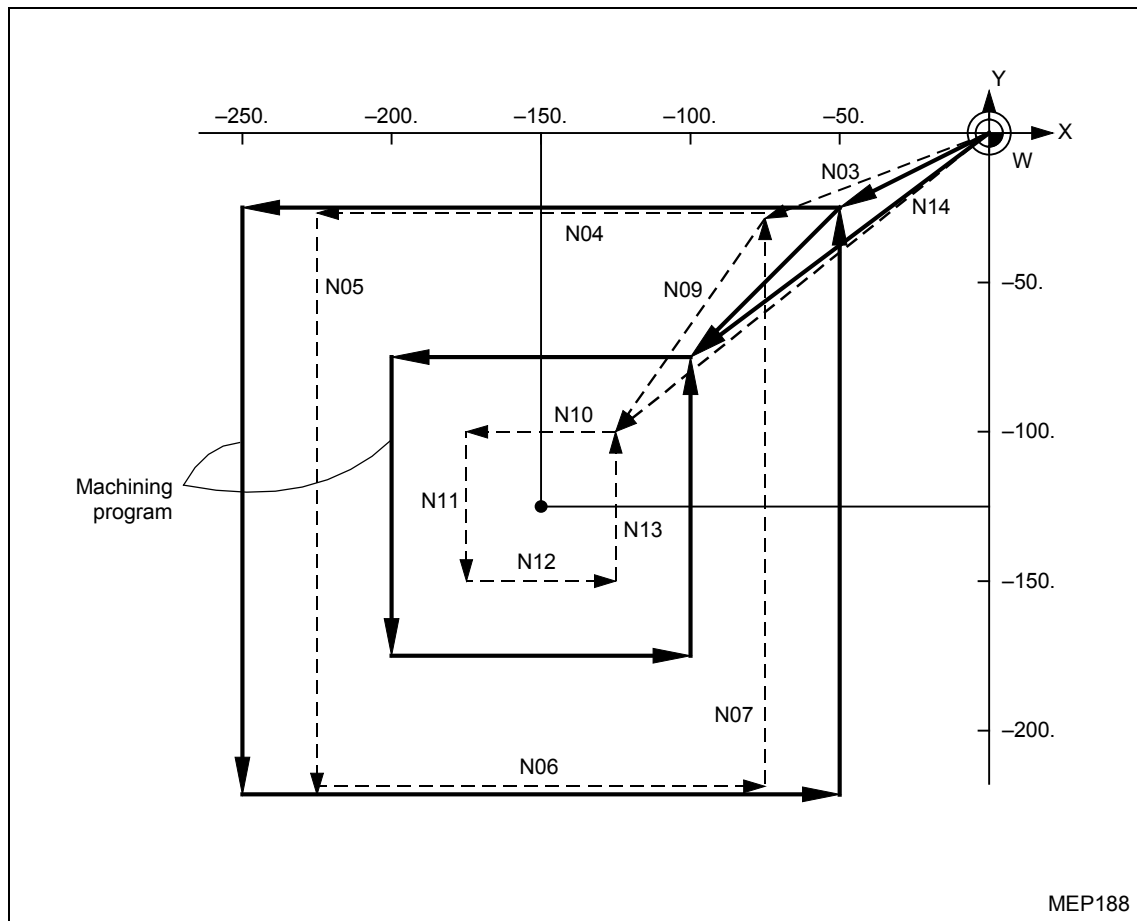
11. Setting G51 during scaling

If command G51 is set during the scaling mode, the axis for which the center of scaling is newly specified will also undergo scaling. The scaling factor specified by the latest G51 command becomes valid in that case.

```

N01 G92X0Y0
N02 G90G51X-150.P0.75      Scaling axis X; P = 0.75
N03 G00X-50.Y-25.
N04 G01X-250.F1000
N05 Y-225.
N06 X-50.
N07 Y-25.
N08 G51Y-125.P0.5          Scaling axes X and Y; P = 0.5
N09 G00X-100.Y-75.
N10 G01X-200.
N11 Y-175.
N12 X-100.
N13 Y-75.
N14 G00G50X0Y0            Cancel

```



MEP188

13-6 Mirror Image ON/OFF: G51.1/G50.1

1. Function and purpose

Mirror image mode can be turned on and off for each axis using G-codes. Higher priority is given to the mirror image setting with the G-codes over setting by any other methods.

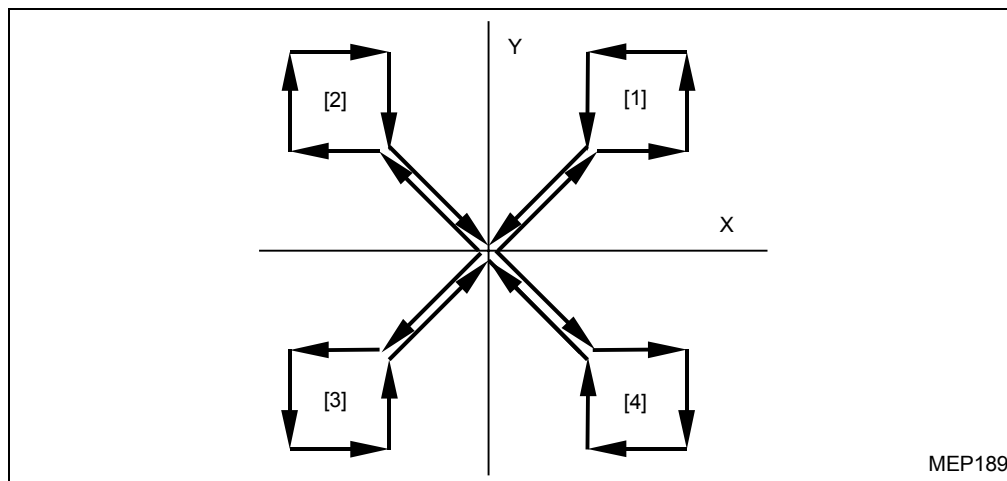
2. Programming format

G51.1 Xx₁ Yy₁ Zz₁ Mirror image On
G50.1 Xx₂ Yy₂ Zz₂ Mirror image Off

3. Detailed description

- Use the address and coordinates in a G51.1 block to specify the mirroring axis and mirroring center (using absolute or incremental data), respectively.
- If the coordinate word is designated in G50.1, then this denotes the axis for which the mirror image is to be cancelled. Coordinate data, even if specified, is ignored in that case.
- After mirror image processing has been performed for only one of the axes forming a plane, the rotational direction and the offset direction become reverse during circular interpolation, tool radius compensation, or coordinate rotation.
- Since the mirror image processing function is valid only for local coordinate systems, the center of mirror image processing moves according to the particular counter preset data or workpiece coordinate offsetting data.

4. Sample programs



(Main program)

G00G90G40G49G80
M98P100
G51.1X0
M98P100
G51.1Y0
M98P100
G50.1X0
M98P100
G50.1Y0
M30

X Y

[1] OFF OFF
[2] ON OFF
[3] ON ON
[4] OFF ON
OFF OFF

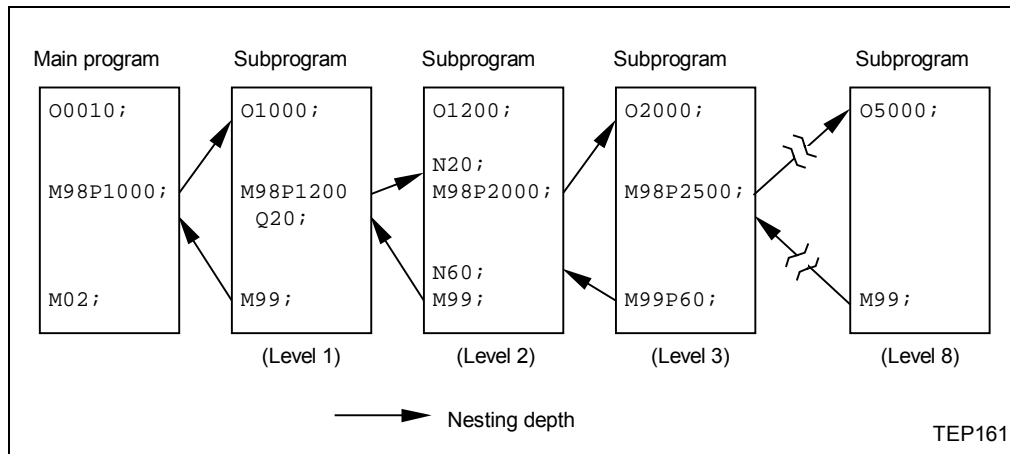
(Subprogram O100)

G91G28X0Y0
G90G00X20.Y20.
G42G01X40.D01F120
Y40.
X20.
Y20.
G40X0Y0
M99

13-7 Subprogram Control: M98, M99

1. Function and purpose

Fixed sequences or repeatedly used programs can be stored in the memory as subprograms which can then be called from the main program when required. M98 serves to call subprograms and M99 serves to return from the subprogram. Furthermore, it is possible to call other subprograms from particular subprograms and the nesting depth can include as many as 8 levels.



The table below shows the functions which can be executed by adding and combining the tape storing and editing functions, subprogram control functions and fixed cycle functions.

(○: available ×: not available)

	Case 1	Case 2	Case 3	Case 4
1. Tape storing and editing	Yes	Yes	Yes	Yes
2. Subprogram control	No	Yes	Yes	No
3. Fixed cycles	No	No	Yes	Yes
Function				
1. Memory operation	○	○	○	○
2. Tape editing (main memory)	○	○	○	○
3. Subprogram call	×	○	○	×
4. Subprogram nesting (*)	×	○	○	×
5. Fixed cycles	×	×	○	○
6. Fixed cycle subprogram editing	×	×	○	○

* The nesting depth can include as many as 8 levels.

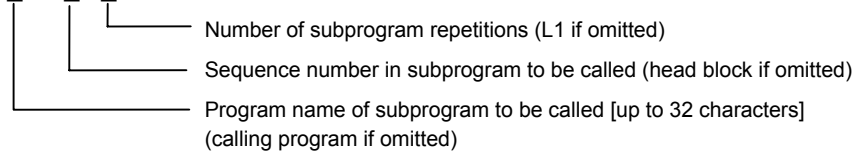
Note: In order that it may function correctly, do not give a command of subprogram control in one block together with any one for the following functions (described in this chapter):

- Hole machining pattern cycle
- Fixed cycle
- Macro call

2. Programming format

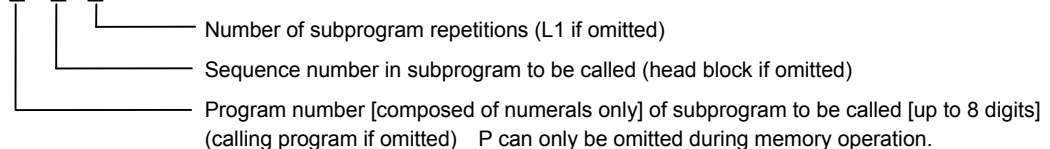
Subprogram call

M98 < > H L ;



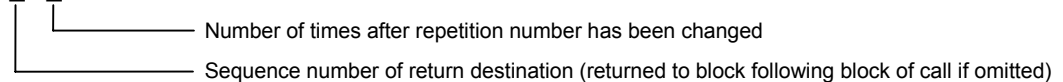
Alternatively,

M98 P H L ;



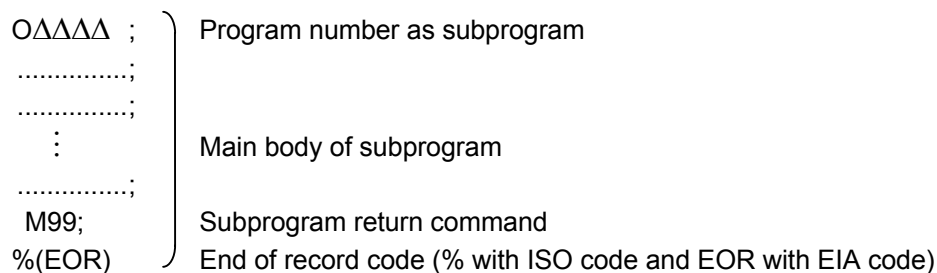
Return to main program from subprogram

M99 P L ;



3. Creating and entering subprograms

Subprograms have the same format as machining programs for normal memory operation except that the subprogram completion instruction M99 (P_ L_) is entered as an independent block at the last block.



The above program is registered by editing operations. For further details, refer to the PROGRAMMING MANUAL (MAZATROL Program).

Only those subprograms numbers ranging from 1 through 9999 designated by the optional specifications can be used. When there are no program numbers on the tape, the setting number for "program input" is used.

Up to 8 nesting levels can be used for calling programs from subprograms, and an alarm (**SUB PROGRAM NESTING EXCEEDED**) occurs if this number is exceeded.

Main programs and subprograms are registered in order in which they were read because no distinction is made between them. This means that main programs and subprograms should not be given the same numbers. (If the same numbers are given, error occurs during entry.)

```

;
OOOOO ;
.....;
:
M99;
%

```

Subprogram A

```

;
OΔΔΔΔ ;
.....;
:
M99;
%

```

Subprogram B

```

;
O□□□□ ;
.....;
:
M99;
%

```

Subprogram C

Note 1: Main programs can be used during memory and tape operation but subprograms must have been entered in the memory.

Note 2: The following commands are not the object of subprogram nesting and can be called even beyond the 8th nesting level.

- Fixed cycles
- Pattern cycles

4. Subprogram execution

M98: Subprogram call command

M99: Subprogram return command

Programming format

M98 <_> H_ L_; or M98 P_ H_ L_;

Where < > : Name of the subprogram to be called (up to 32 characters)

P : Number of the subprogram to be called (up to 8 digits)

H : Any sequence number within the subprogram to be called (up to 5 digits)

L : Number of repetitions from 1 to 9999 with numerical value of four figures; if L is omitted, the subprogram is executed once ; with L0, there is no execution.

For example,

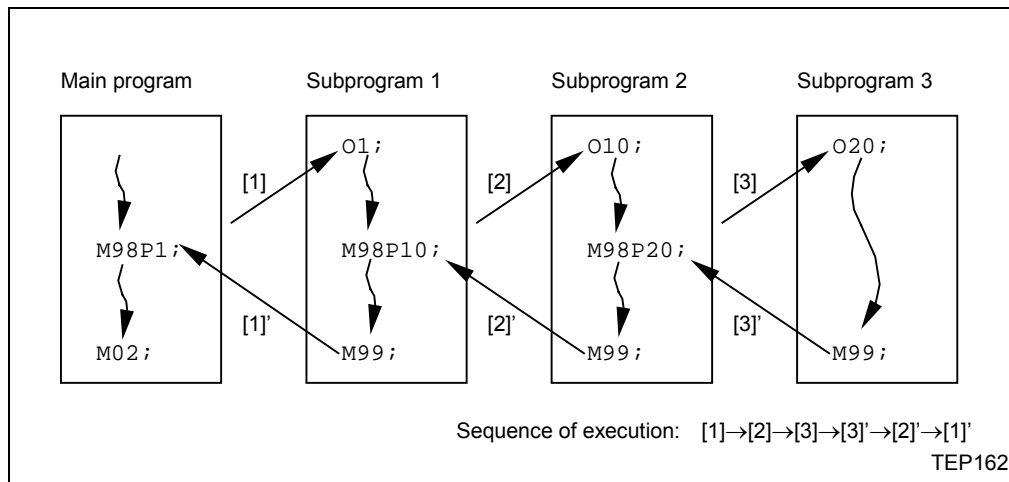
M98 P1 L3; is equivalent to the following :

M98 P1;

M98 P1;

M98 P1;

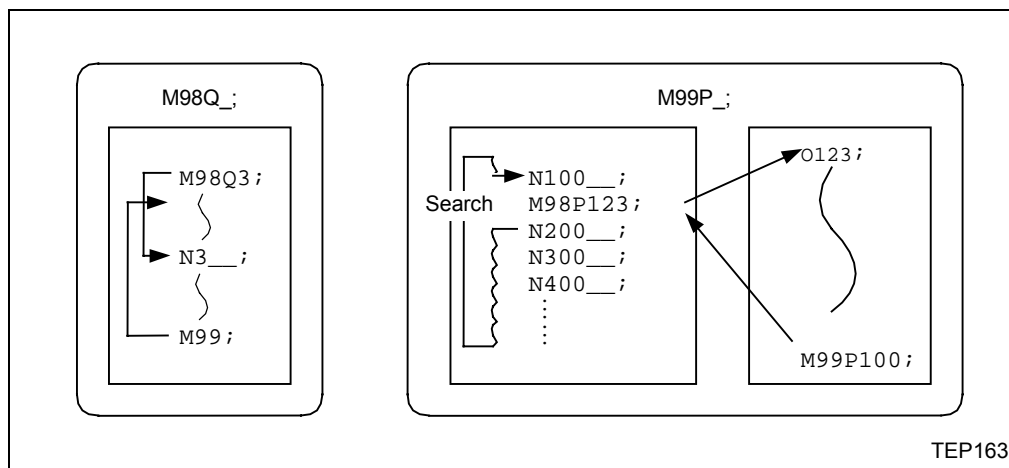
Example 1: When there are 3 subprogram calls (known as 3 nesting levels)



For nesting, the M98 and M99 commands should always be paired off on a 1 : 1 basis [1]' for [1], [2]' for [2], etc.

Modal information is rewritten according to the execution sequence without distinction between main programs and subprograms. This means that after calling a subprogram, attention must be paid to the modal data status when programming.

Example 2: The M98 Q_ ; and M99 P_ ; commands designate the sequence numbers in a program with a call instruction.



Example 3: Main program M98 P2 ;

O1;	}	Subprogram 1
⋮		
M99;		
%		
O2;	}	Subprogram 2
⋮		
N200		
⋮		
M99;		
%		
O3;	}	Subprogram 3
⋮		
N200		
⋮		
M99;		
%		

- When the O2 N200 block is searched with the memory search function, the modal data are updated according to the related data of O2 to N200.
- The same sequence number can be used in different subprograms.
- When the subprogram (No. p_1) is to be repeatedly used, it will be repeatedly executed for I_1 times provided that M98 P p_1 L I_1 ; is programmed.

5. Other precautions

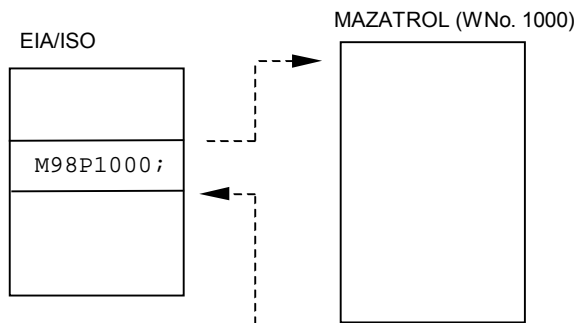
- An alarm occurs when the designated program number (P) is not found.
- Single block stop does not occur in the M98P_ ; and M99 ; block. If any address except O, N, P, Q or L is used, single block stop can be executed. (With X100. M98 P100 ; operation branches to O100 after X100. is executed.)
- When M99 is commanded in the main program, operation returns to the head.
- Operation can branch from tape or PTR operation to a subprogram by M98P_ but the sequence number of the return destination cannot be designated with M99P_ ;. (P_ is ignored.)
- Care should be taken that the search operation will take time when the sequence number is designated by M99P_ ;

6. MAZATROL program call from EIA/ISO program

A. Overview

MAZATROL machining program can be called as a subprogram from the machining program described with EIA/ISO codes.

EIA/ISO → MAZATROL (Program call)



MAZATROL machining program is called from EIA/ISO program, and entire machining program can be used.

Note: When the execution of MAZATROL machining program is completed, the execution is returned again to EIA/ISO program.
It should be noted that the used tool, current position and others are changed though EIA/ISO modal information is not changed.

B. Programming format

M98 <_> L_; or M98 P_ L_;

< > or P:

- Name, or number, of the MAZATROL machining program to be called.
- When omitted, an alarm (**NO DESIGNATED PROGRAM**) will be caused.
- The same alarm will occur when the specified program is not stored.

L:

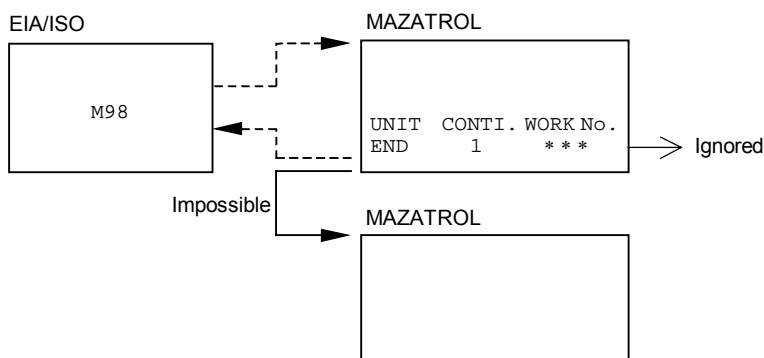
- Number of repetitions of program execution (1 to 9999).
- When omitted or L = 0, the called program will be executed one time (as if L = 1).

C. Detailed description

1. END unit of the MAZATROL program

The END unit of the MAZATROL program to be called must have "1" specified under **CONTI.** for correct return to the EIA/ISO main program.

As for the END unit's items other than **CONTI.**: Even if **WORK No.** is specified, program chain cannot be made with the MAZATROL program called from an EIA/ISO program.



2. MAZATROL program execution

When MAZATROL program is called from an EIA/ISO program, the MAZATROL program is executed like automatic operation of MAZATROL.

MAZATROL program is executed independently of EIA/ISO program which has made the call. In other words, it performs the same machining as MAZATROL program alone is executed. When calling MAZATROL program, always place a tool outside the safety profile beforehand. Failure to do this may cause interference of a workpiece with the tool.

Note: The END unit of the MAZATROL program to be called from an EIA/ISO program must have "1" specified under **CONTI.** for correct return to the EIA/ISO main program. Never use an M99 command for the return.

D. Remarks

1. MDI interruption and macro interruption signal during MAZATROL program execution are ignored.
2. MAZATROL program cannot be restarted halfway.
3. MAZATROL program call in the mode of a fixed cycle results in an alarm.
4. MAZATROL program call in the mode of tool radius compensation results in an alarm.
5. MAZATROL program call is not available in the MDI operation mode (results in an alarm).
6. A MAZATROL program called by M98 cannot be executed but in its entirety (from the head to the end).
7. Commands to addresses other than O, N, P, Q, L and H in a block of M98 for MAZATROL program call will not be processed till completion of the called program.

13-8 End Processing: M02, M30, M998, M999

If the program contains M02, M30, M998, M999 or EOR (%), the block containing one of these codes will be executed as the end of the program in the NC unit. The program end processing will not be commanded by M98 or M99. In end processing, tool life processing, parts count, and work No. search will be executed.

1. M02, M30
Tool life processing only will be executed.
2. M998, M999
Tool life processing, parts count, and work No. search will be executed.

M998(999) <111> Q1;

└─ Specification of execution or non-execution of parts count
(counting updated on **POSITION** display)
0: Parts count non-execution
1: Parts count execution

└─ Name of the program to be executed next

└─ M-code for program chain
M998: Continuous execution after parts count and work No. search
M999: Ending after parts count and work No. search

As shown below, the next program can be designated alternatively with address S if its "name" consists of numerals only.

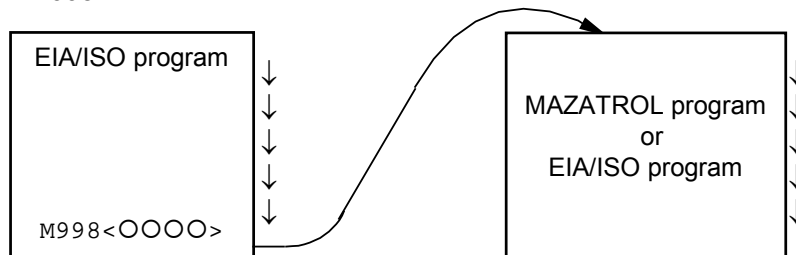
M998(999) S111 Q1;

└─ Specification of execution or non-execution of parts count
(counting updated on **POSITION** display)
0: Parts count non-execution
1: Parts count execution

└─ Number of the program to be executed next

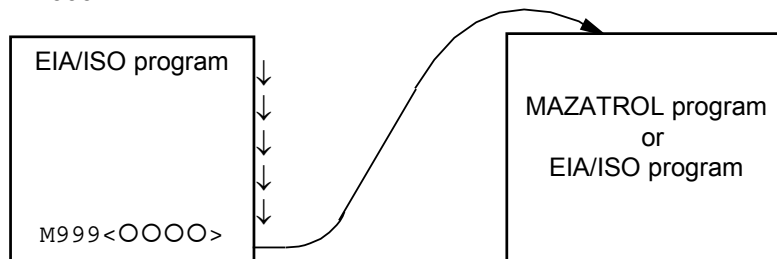
└─ M-code for program chain
M998: Continuous execution after parts count and work No. search
M999: Ending after parts count and work No. search

- M998<0000>



MAZATROL or EIA/ISO program is called from EIA/ISO program and executed as the next program.

- M999<0000>



MAZATROL or EIA/ISO program is only called from EIA/ISO program and the operation is terminated.

Note 1: In order to prevent the machine operation from being inconveniently stopped, the NC will process a block of M998Q1 automatically as that of M999Q1 in case the total number of machined parts should amount to, or exceed, the preset number of parts required.

Note 2: Omission of the designation of the next program, be it by name or number, will result in the current main program being called up as the next one.

13-9 Linear Angle Commands

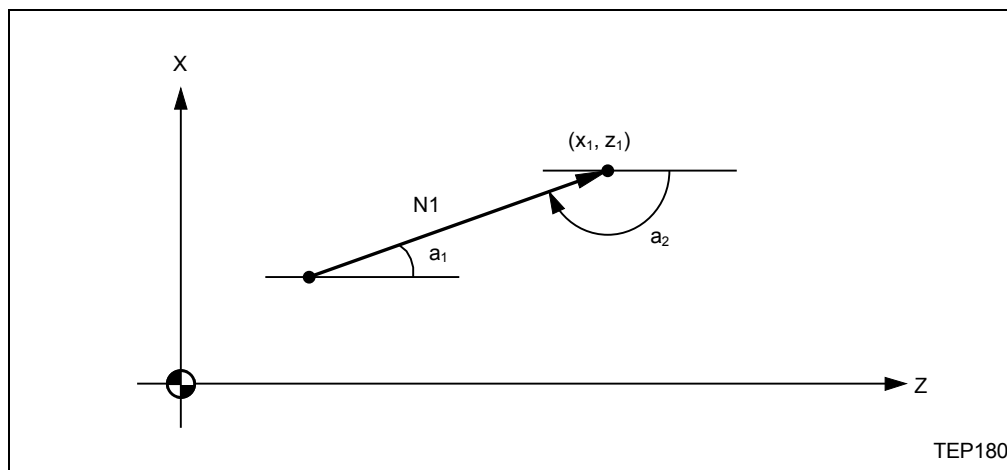
1. Function and purpose

Programming the linear angle and one of the coordinates of the ending point makes the NC unit automatically calculate the coordinates of that ending point.

2. Programming format

N1 G01 Aa₁ Zz₁ (Xx₁) Designate the angle and the coordinates of the X-axis or the Z-axis.

N1 G01 A-a₂ Zz₁ Xx₁



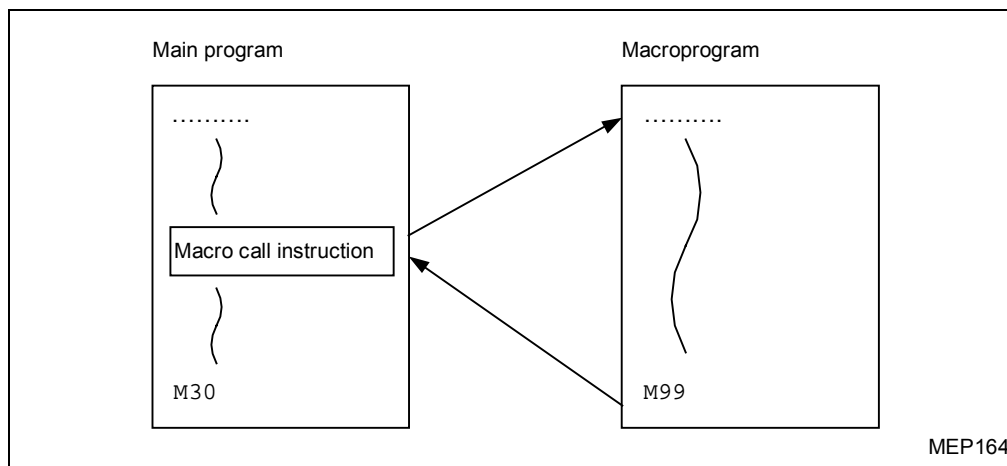
3. Detailed description

1. The angle denotes that relative to the plus (+) direction of the first axis (horizontal axis) on the selected plane.
Assign the sign + for a counterclockwise direction (CCW) or the sign – for a clockwise direction (CW).
2. Set the ending point on one of the two axes of the selected plane.
3. Angle data will be ignored if the coordinates of both axes are set together with angles.
4. If angles alone are set, the command will be handled as a geometric command.
5. For the second block, the angle at either the starting point or the ending point can be specified.
6. The linear angle command function does not work if address A is to be used for an axis name or for the No. 2 auxiliary function.
7. This function is valid only for the G01 command; it is not valid for other interpolation or positioning commands.

13-10 Macro Call Function: G65, G66, G66.1, G67

13-10-1 User macros

Macroprogram call, data calculation, data input to/output from a personal computer, data control, judgment, branching, and various other instructions can be used with variables commands to perform measurements and other operations.



A macroprogram is a subprogram which is created using variables, calculation instructions, control instructions, etc. to have special control features.

These special control features (macroprograms) can be used by calling them from the main program as required. These calls use macro call instructions.

Detailed description

- When command G66 is entered, the designated user macro subprogram will be called every time after execution of the move commands within a block until G67 (cancellation) is entered.
- Command codes G66 and G67 must reside in the same program in pairs.

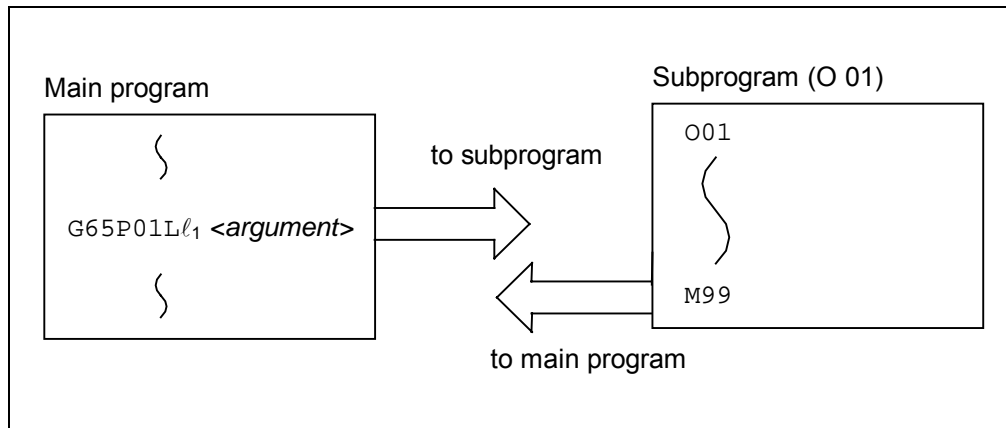
Note: In order that it may function correctly, do not give a command of macro call in one block together with any one of the following commands (described in this chapter):

- Hole machining pattern cycle
- Fixed cycle
- Subprogram control

13-10-2 Macro call instructions

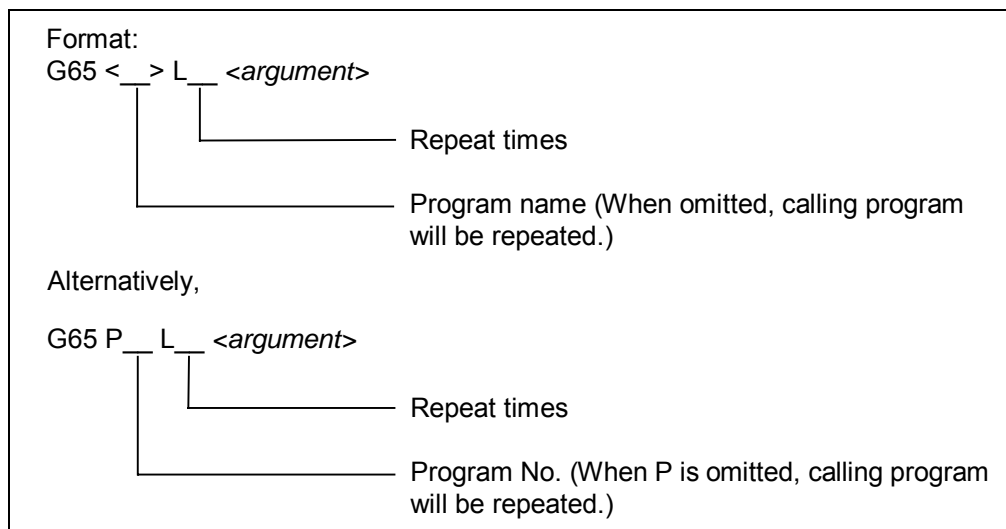
Two types of macro call instructions are provided: single-call instructions used to call only at the designated block, and modal call instructions used to call at each block within a macro call mode. Modal call instructions are further divided into type A and type B.

1. Single call



The designated user macro subprogram ends with M99.

Instruction G65 calls the designated user macro subprogram only once.



<Argument>

When argument is to be delivered to the user macro subprogram as a local variable, designate the required data with the respective addresses. (Argument designation is not available for a user macro subprogram written in MAZATROL language.)

In such a case, the argument can have a sign and a decimal point, irrespective of the address. Arguments can be specified using method I or II, as shown below.

A. Argument specification I

Format: A_B_C_ X_Y_Z_

Detailed description

- An argument can be specified using all addresses, except G, L, N, O, and P.
- Except for I, J, and K, addresses does not need be specified in an alphabetical order.
- Addresses I, J, and K must be specified in an alphabetical order.
I_J_K_ ... Correct
J_I_K_ ... Wrong
- Addresses whose specification is not required can be omitted.
- The relationship between addresses that can be specified using argument specification I, and variables numbers in a user macro unit, is shown in the following table:

Relationship between address and variables number		Call commands and usable addresses	
Address specified using method I	Variable in macro-program	G65, G66	G66.1
A	#1	○	○
B	#2	○	○
C	#3	○	○
D	#7	○	○
E	#8	○	○
F	#9	○	○
G	#10	×	×*
H	#11	○	○
I	#4	○	○
J	#5	○	○
K	#6	○	○
L	#12	×	×*
M	#13	○	○
N	#14	×	×*
O	#15	×	×
P	#16	×	×*
Q	#17	○	○
R	#18	○	○
S	#19	○	○
T	#20	○	○
U	#21	○	○
V	#22	○	○
W	#23	○	○
X	#24	○	○
Y	#25	○	○
Z	#26	○	○

○: Usable ×: Unusable *: Usable in G66.1 modal

B. Argument specification II

Format: A_B_C_I_J_K_I_J_K_.....

Detailed description

- Up to a maximum of 10 sets of arguments that each consist of addresses I, J, and K, as well as A, B, and C, can be specified.
- If identical addresses overlap, specify them in the required order.
- Addresses whose specification is not required can be omitted.
- The relationship between addresses that can be specified using argument specification II, and variables numbers in a user macro unit, is shown in the following table:

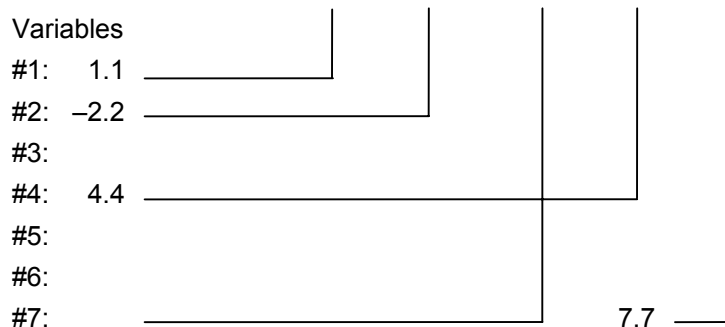
Argument specification II addresses	Variables in macro-programs	Argument specification II addresses	Variables in macro-programs
A	#1	K5	#18
B	#2	I6	#19
C	#3	J6	#20
I1	#4	K6	#21
J1	#5	I7	#22
K1	#6	J7	#23
I2	#7	K7	#24
J2	#8	I8	#25
K2	#9	J8	#26
I3	#10	K8	#27
J3	#11	I9	#28
K3	#12	J9	#29
I4	#13	K9	#30
J4	#14	I10	#31
K4	#15	J10	#32
I5	#16	K10	#33
J5	#17		

Note: In the table above, the numerals 1 through 10 have been added to addresses I, J, and J just to denote the order of arrangement of the designated sets of arguments: these numerals are not included in actual instructions.

C. Combined use of argument specification I and II

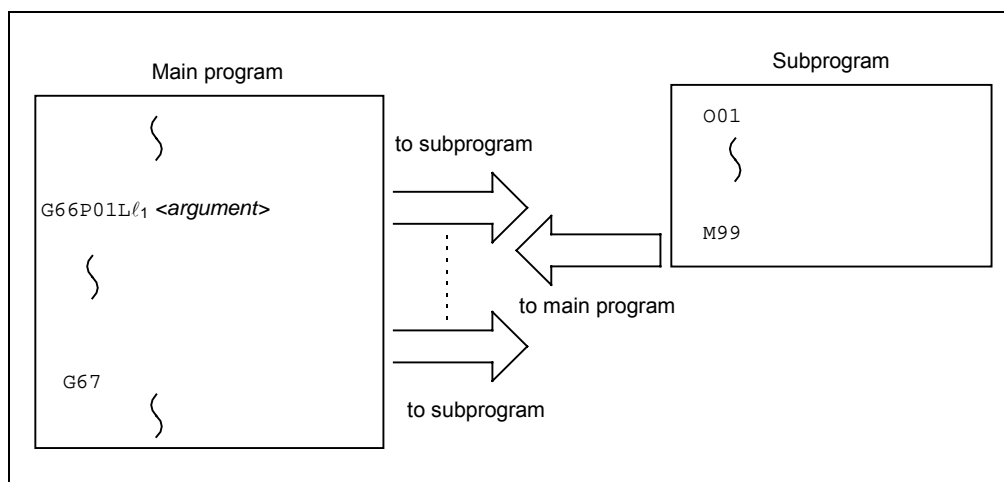
When both method I and method II are used to specify arguments, only the latter of two arguments which have an address corresponding to the same variable will become valid.

Example: Call command G65 A1.1 B-2.2 D3.3 I4.4 I7.7



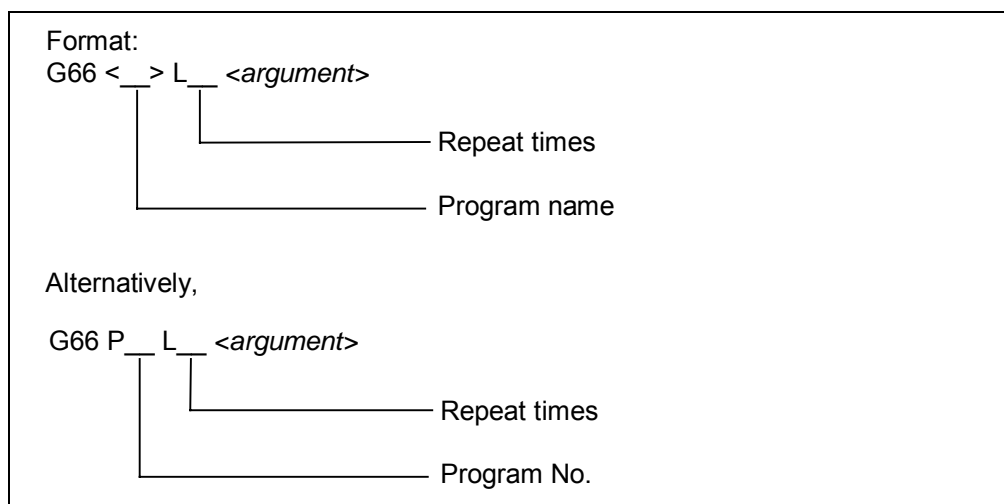
If two arguments (D3.3 and I7.7) are designated for the variable of #7, only the latter argument (I7.7) will be used.

2. Modal call, type A (Move command call)



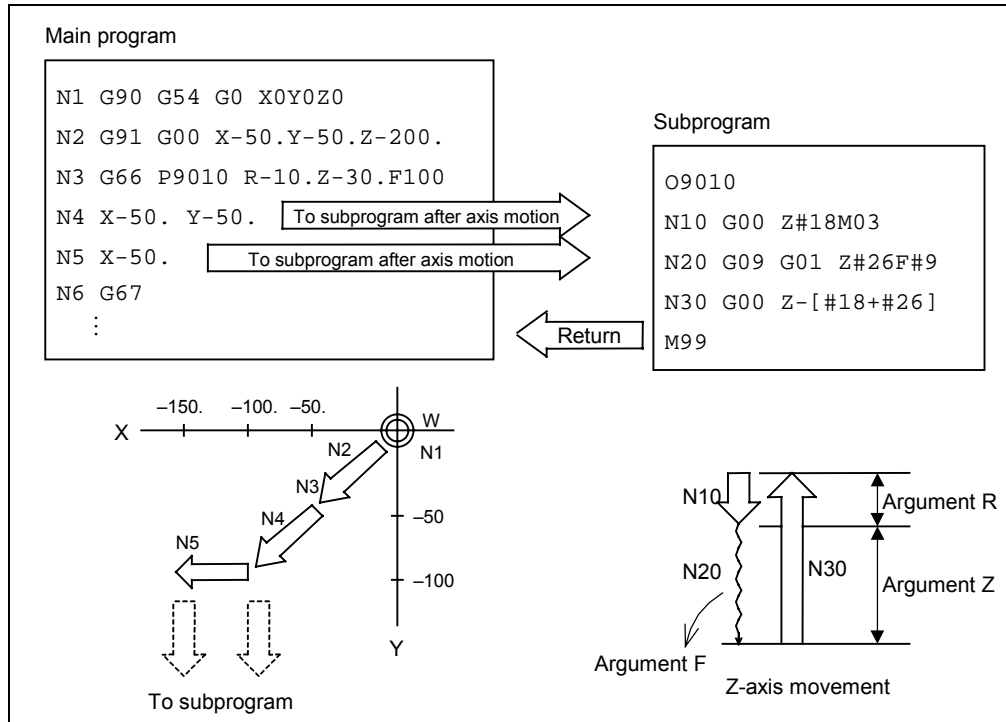
For a block that has a move command code between G66 and G67, the designated user macro subprogram is executed after that move command has been executed. The subprogram is executed an ℓ_1 number of times for the first call, or once for subsequent calls.

For modal call of type A, the methods of specifying <argument> are the same as used for single call.



Detailed description

- When command G66 is entered, the designated user macro subprogram will be called every time after execution of the move commands within a block until command G67 (cancellation) is entered.
- Command codes G66 and G67 must reside in the same program in pairs.
Entry of a G67 command without a G66 command results in an alarm **857 INCORRECT USER MACRO G67 PROG.**

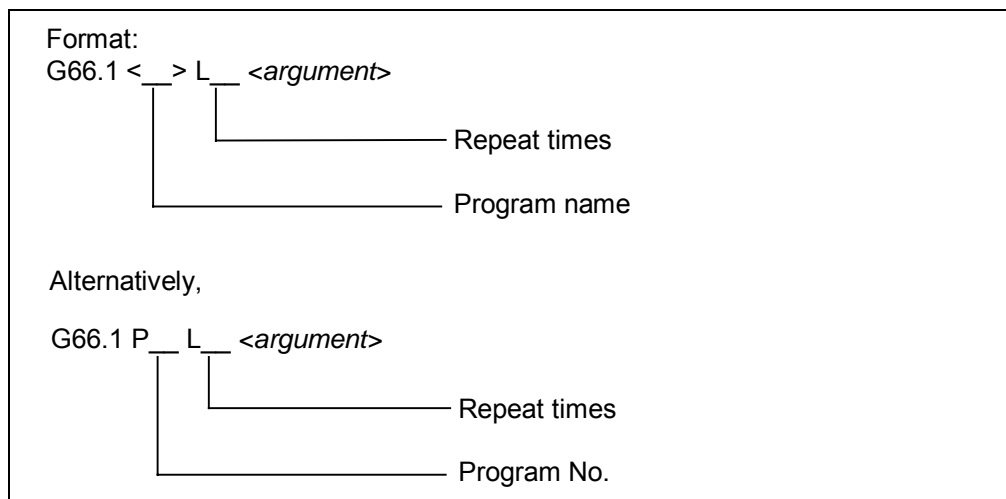
Example: Drilling cycle

Note 1: The designated subprogram is executed after the axis commands in the main program have been executed.

Note 2: No subprograms are executed for the G67 block and its successors.

3. Modal call, type B (Block-to-block call)

The designated user macro subprogram is called unconditionally for each of the command blocks present between G66.1 and G67. Execution of the macro program is repeated as specified with L for the first call, and only once for each of subsequent calls.



Detailed description

- During the G66.1 mode, only the codes O, N, and G in each of the read command blocks are executed. No other codes in those blocks are executed; codes other than O, N, and G are handled as arguments. However, only the last G-code and the N-codes following a code other than O or N become arguments.

- All significant blocks in the G66.1 mode are regarded as preceded by the command G65P_.

For example, the block of

```
N100G01G90X100. Y200. F400R1000
```

in the G66.1P1000 mode is handled as equivalent to

```
N100G65P1000G01G90X100. Y200. F400R1000.
```

Note: Call is executed even for the G66.1 command block of the G66.1 mode, with the relationship between the addresses of the arguments and the variables numbers being the same as for G65 (single call).

- The data range of the G, L, P, and N commands that you can set as new variables using the G66.1 mode is the same as the data range of usual NC commands.
- Sequence number N, modal G-codes, and O are all updated as modal information.

4. G-code macro call

The user macro subprograms of the required program number can be called just by setting G-codes.

Format:

Gxx <argument>

————— G-code which calls macro-subprogram

Detailed description

- The instruction shown above performs the same function as those of the instructions listed below. Which of these listed instructions will apply is determined by the parameter data to be set for each G-code.

```
M98PΔΔΔΔ
```

```
G65PΔΔΔΔ <argument>
```

```
G66PΔΔΔΔ <argument>
```

```
G66.1PΔΔΔΔ <argument>
```

- Use parameters to set the relationship between Gxx (macro call G-code) and PΔΔΔΔ (program number of the macro to be called).
- Of G00 through G255, up to a maximum of 10 command codes can be used with this instruction unless the uses of these codes are clearly predefined by EIA/ISO Standards, such as G00, G01, G02, etc.
- The command code cannot be included in user macro subprograms that have been called using G-codes.

5. Auxiliary command macro call (M-, S-, T-, or B-code macro call)

The user macro subprograms of the required program number can be called just by setting M-, S-, T-, or B-codes.

Format:

Mm (or Ss, Tt and Bb)

————— M (or S, T and B) code which calls macro-subprogram

Detailed description (The following description also applies to S-, T-, and B-codes.)

- The instruction shown above performs the same function as those of the instructions listed below. Which of these listed instructions will apply is determined by the parameter data to be set for each M-code.

M98P $\Delta\Delta\Delta\Delta$

G65P $\Delta\Delta\Delta\Delta$ Mm

G66P $\Delta\Delta\Delta\Delta$ Mm

G66.1P $\Delta\Delta\Delta\Delta$ Mm

- Use parameter to set the relationship between Mm (macro call M-code) and P $\Delta\Delta\Delta\Delta$ (program number of the macro to be called).
Up to a maximum of 10 M-codes, ranging from M00 to M95, can be registered. Do not register the M-codes that are fundamentally required for your machine, nor M0, M1, M2, M30, and M96 through M99.
- If registered auxiliary command codes are set in the user macro subprograms that have been called using M-codes, macro calls will not occur since those special auxiliary command codes will be handled as usual ones (M-, S-, T-, or B-codes).

6. Differences in usage between commands M98, G65, etc.

- Arguments can be designated for G65, but cannot be designated for M98.
- Sequence numbers can be designated for M98, but cannot be designated for G65, G66, or G66.1.
- Command M98 executes a subprogram after M98 block commands other than M, P, H, and L have been executed, whereas G65 just branches the program into a subprogram without doing anything.
- Single-block stop will occur if the block of command M98 has addresses other than O, N, P, H, and L. For G65, however, single-block stop will not occur.
- The level of local variables is fixed for M98, but for G65 does change according to the depth of nesting. (For example, #1s, if present before and after M98, always mean the same, but if present before and after G65, they have different meanings.)
- Command M98 can have up to a maximum of 15 levels of call multiplexity when combined with G65, G66, or G66.1, whereas the maximum available number of levels for command G65 is four when it is combined with G66 or G66.1.

7. Multiplexity of macro call commands

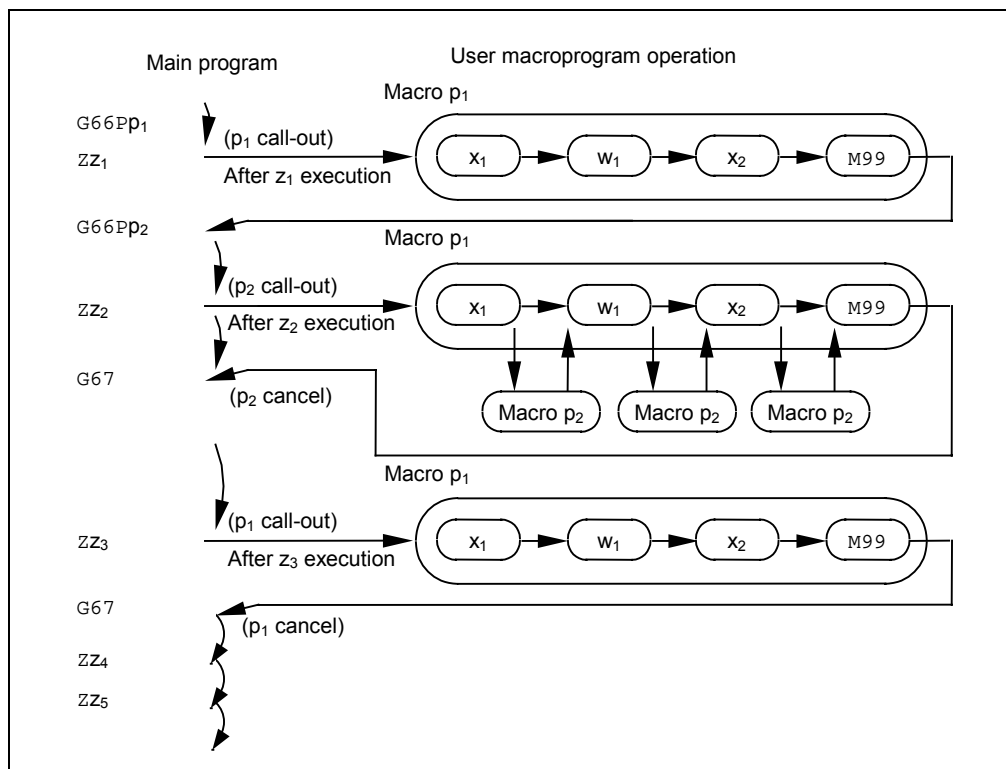
The maximum available number of levels of macro subprogram call is four, whether it is single or modal. Arguments in macro call instructions become valid only within the level of the called macro. Since the multiplexity of macro call is of up to a maximum of four levels, arguments can be included in a program as local variables each time a macro call is made.

Note 1: When a G65, G66, or G66.1 macro call or an auxiliary command macro call is made, nesting will be regarded as single-level and thus the level of local variables will also increase by 1.

Note 2: For modal call of type A, the designated user macro subprogram is called each time a move command is executed. If, however, multiple G66s are present, the next user macro subprogram will be called even for the move commands in the macro each time axis movement is done.

Note 3: User macro subprograms are cancelled in a reverse order to that in which they have been arranged.

Example:



8. User macro call based on interruption

Outline

Prior creation of special user macros for interrupt processing allows the user macros to be executed during automatic operation when a user macro interrupt signal is input. After the user macro has been executed, the program can also be returned to the interrupted program block and then started from this block.

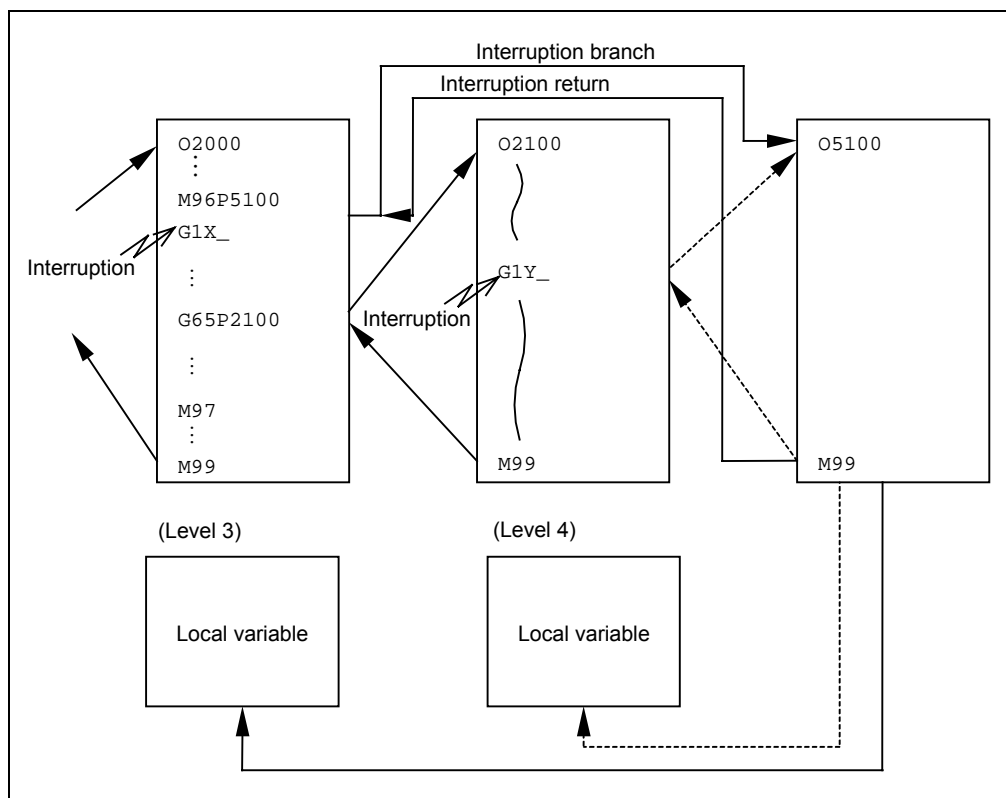
Detailed description

- Format for selecting the user macro branching destination

⋮	
M96<_>L_ (or M96P_L_)	}
⋮	
⋮	
M97 (Branching mode off)	
⋮	

(Branching mode on)
When user macroprogram interruption signal is input during this space, the branch into the specified user macroprogram will be applied.

- User macro interrupts can be processed even when the number of levels of macro call multiplexity during the occurrence of an interrupt is four. The local variables' level of the user macros used for interruption is the same as the level of the user macros existing during the occurrence of an interrupt.



13-10-3 Variables

Of all types of variables available for the NC unit, only local variables, common variables, and part of system variables are retained even after power-off.

1. Multiplexing of variables

Under user macro specifications, variables can have their identifiers (identification numbers) either transformed into variables, which is referred to as multiplexing, or replaced with <expression>.

For <expression>, only one arithmetic expression (for either multiplication, division, addition, or subtraction) can be used.

Example 1: Multiplexing variables

#1=10 #10=20 #20=30	From #1 = 10, #[[#1]] = #[#10] will result.
#5=#[[#1]]	From #10 = 20, #[#10] = #20 will result. Therefore #5 = #20, i.e. #5 = 30 will result.
#1=10 #10=20 #20=30	From #1 = 10, #[[#1]] = #[#10] will result.
#5=1000	From #10 = 20, #[#10] = #20 will result. Therefore #20 = #5, i.e.
#[[#1]]=#5	#20 = 1000 will result.

Example 2: Replacing variables identifiers with <expression>

#10=5	
#[#10+1]=1000	#6 = 1000 will result.
#[#10-1]=-1000	#4 = -1000 will result.
#[#10*3]=100	#15 = 100 will result.
#[#10/2]=100	#2 = -100 will result.

2. Undefined variables

Under user macro specifications, variables remaining unused after power-on or local variables that are not argument-specified by G65, G66, or G66.1 can be used as <empty>. Also, variables can be forcibly made into <empty>.

Variable #0 is always used as <empty> one, and this variable cannot be defined on the left side of the expression.

A. Arithmetic expression

```
#1=#0.....#1 = <empty>
#2=#0+1 .....#2 = 1
#3=1+#0 .....#3 = 1
#4=#0*10 .....#4 = 0
#5=#0+#0 .....#5 = 0
```

Note: Be careful that <empty> is handled the same as 0 during processing of expressions.

<empty> + <empty> = 0

<empty> + <constant> = constant

<constant> + <empty> = constant

B. Applying variables

Application of an undefined variable alone causes even the address to be ignored.

If #1 = <empty>

G0X#1Y1000 is equivalent to G0Y1000, and

G0X[#1+10]Y1000 is equivalent to G0X10Y1000.

C. Conditional expression

Only for EQ and NE, does <empty> differ from 0 in meaning.

If #101 = <empty>		If #101 = 0	
#101EQ#0	<empty> = <empty> holds.	#101EQ#0	0 = <empty> does not hold.
#101NE0	<empty> ≠ 0 holds.	#101NE0	0 ≠ 0 does not hold.
#101GE#0	<empty> ≥ <empty> holds.	#101GE#0	0 ≥ <empty> holds.
#101GT0	<empty> > 0 does not hold.	#101GT0	0 > 0 does not hold.

Hold-conditions and not-hold-conditions list

(For conditional expressions including undefined variables)

Right side Left side	EQ		NE		GT		LT		GE		LE	
	Empty	Constant	Empty	Constant	Empty	Constant	Empty	Constant	Empty	Constant	Empty	Constant
Empty	H			H				H	H		H	
Constant			H		H				H			

H: Holds (The conditional expression holds.)

Blank: The conditional expression does not hold.

13-10-4 Types of variables

1. Common variables (#100 to #199, and #500 to #999)

Common variables refer to the variables to be used in common at any position. The identifiers of common variables which can be used are from #100 to #199, or from #500 to #999.

2. Local variables (#1 to #33)

Local variables refer to variables that can be defined as <argument> when calling a macro subprogram, or those which can be used locally within the main program or a subprogram. There is no relationship between macros. Thus, these variables can be overlapped on each other, but up to a maximum of four levels of overlapping.

G65P_{p₁}L_{l₁} <argument>

where p_1 : Program number

l_1 : Number of repeat times

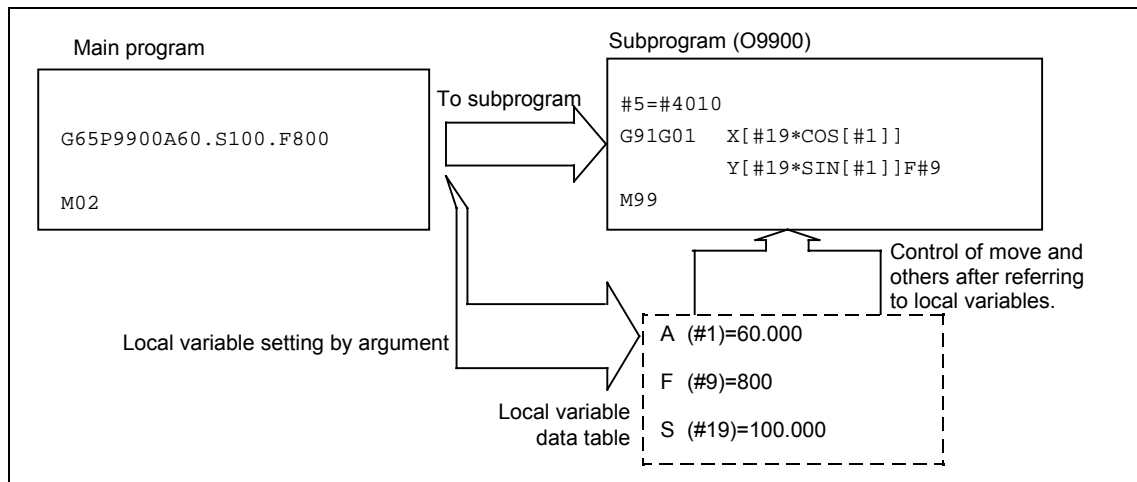
<Argument> must be: Aa₁ Bb₁ Cc₁ ... Zz₁.

The following represents the relationship between the address specified by <argument> and the local variables number used in the user macro unit:

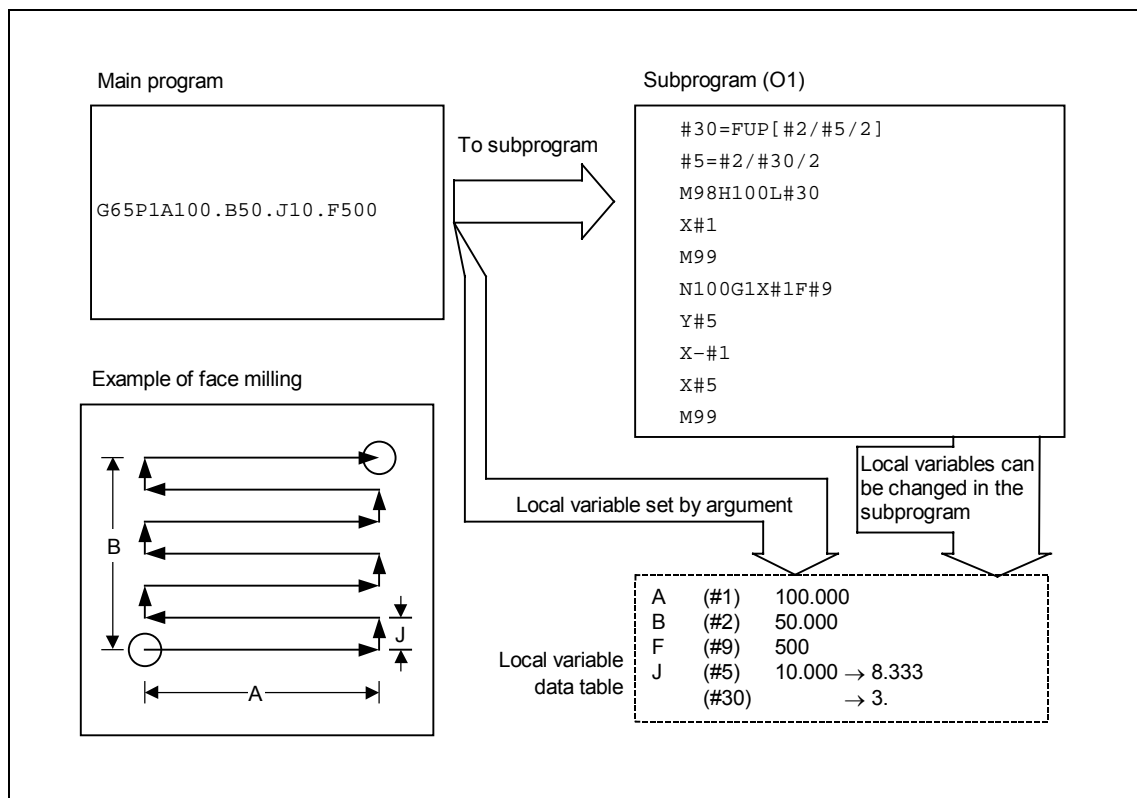
Call commands		Argument address	Local variable	Call commands		Argument address	Local variable
G65 G66	G66.1			G65 G66	G66.1		
○	○	A	#1	○	○	R	#18
○	○	B	#2	○	○	S	#19
○	○	C	#3	○	○	T	#20
○	○	D	#7	○	○	U	#21
○	○	E	#8	○	○	V	#22
○	○	F	#9	○	○	W	#23
×	×	G	#10	○	○	X	#24
○	○	H	#11	○	○	Y	#25
○	○	I	#4	○	○	Z	#26
○	○	J	#5			–	#27
○	○	K	#6			–	#28
×	×	L	#12			–	#29
○	○	M	#13			–	#30
×	×	N	#14			–	#31
×	×	O	#15			–	#32
×	×	P	#16			–	#33
○	○	Q	#17				

Argument addresses marked as × in the table above cannot be used. Only during the G66.1 mode, however, can argument addresses marked with an asterisk (*) in this table be additionally used. Also, the dash sign (–) indicates that no address is crosskeyed to the local variables number.

- Local variables for a subprogram can be defined by specifying <argument> when calling a macro.

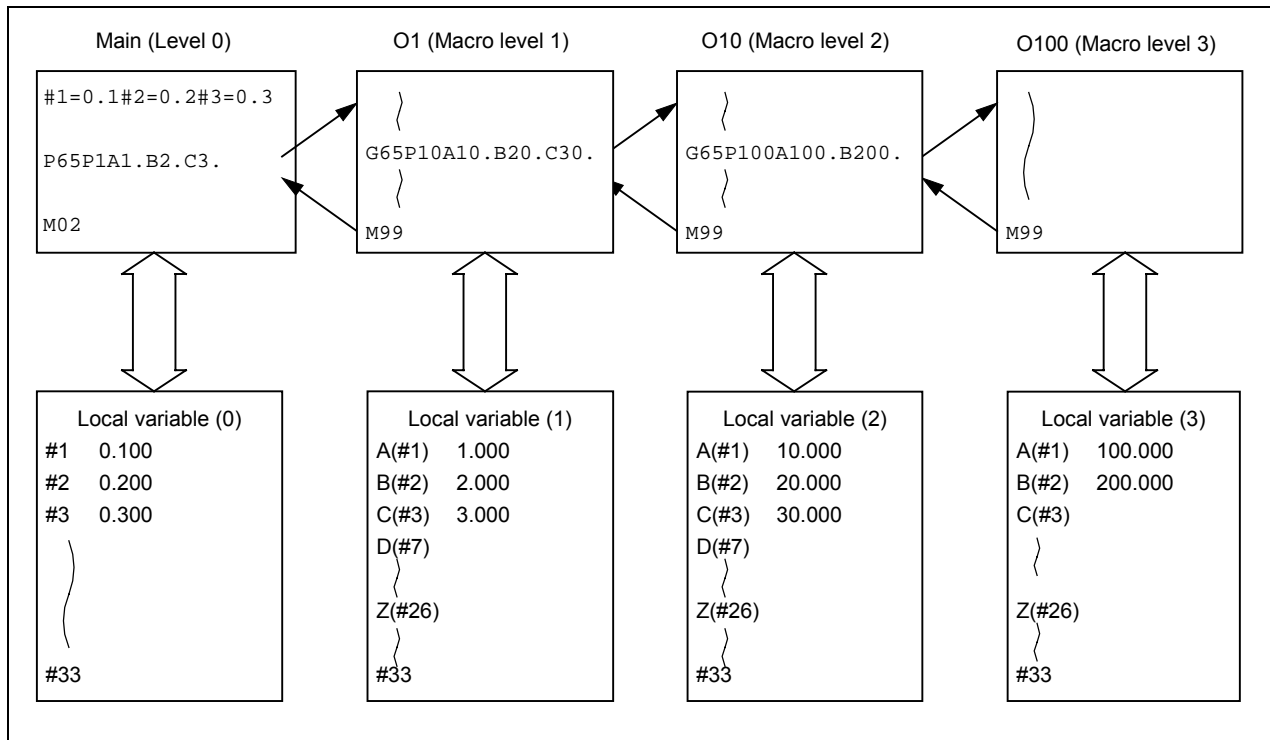


- Within a subprogram, local variables can be freely used.



In the sample program for face-milling that is shown above, although the argument J has initially been programmed as a machining pitch of 10 mm, it has been changed into 8.333 mm to ensure equal-pitched machining. Also, local variable #30 contains the calculated data about the number of times of reciprocal machining.

3. Local variables can be used for each of the four levels of macro call separately. For the main program (macro level 0), separate local variables are also provided. The local variables of level 0, however, cannot be designated with arguments.



How the local variables are currently being used is displayed on the screen.

For further details, refer to the Operating Manual.

3. Macro interface input system variables (#1000 to #1035)

You can check the status of an interface input signal by reading the value of the appropriate variables number (#1000 to #1035).

The read value of the variables number is either 1 (contact closed) or 0 (contact open). You can also check the status of all input signals of the variables from #1000 to #1031 by reading the value of variables number 1032. Variables from #1000 to #1035 can only be read; they cannot be placed on the left side of an arithmetic expression.

System variable	Points	Interface input signal	System variable	Points	Interface input signal
#1000	1	Register R72, bit 0	#1016	1	Register R73, bit 0
#1001	1	Register R72, bit 1	#1017	1	Register R73, bit 1
#1002	1	Register R72, bit 2	#1018	1	Register R73, bit 2
#1003	1	Register R72, bit 3	#1019	1	Register R73, bit 3
#1004	1	Register R72, bit 4	#1020	1	Register R73, bit 4
#1005	1	Register R72, bit 5	#1021	1	Register R73, bit 5
#1006	1	Register R72, bit 6	#1022	1	Register R73, bit 6
#1007	1	Register R72, bit 7	#1023	1	Register R73, bit 7
#1008	1	Register R72, bit 8	#1024	1	Register R73, bit 8
#1009	1	Register R72, bit 9	#1025	1	Register R73, bit 9
#1010	1	Register R72, bit 10	#1026	1	Register R73, bit 10
#1011	1	Register R72, bit 11	#1027	1	Register R73, bit 11
#1012	1	Register R72, bit 12	#1028	1	Register R73, bit 12
#1013	1	Register R72, bit 13	#1029	1	Register R73, bit 13
#1014	1	Register R72, bit 14	#1030	1	Register R73, bit 14
#1015	1	Register R72, bit 15	#1031	1	Register R73, bit 15

System variable	Points	Interface input signal
#1032	32	Register R72 and R73
#1033	32	Register R74 and R75
#1034	32	Register R76 and R77
#1035	32	Register R78 and R79

Note: The following interface input signals are used exclusively in the NC system operation (cannot be used for other purposes).

Interface input signal	Description
Register R72, bit 0	Touch sensor mounted in the spindle
Register R72, bit 4	X- and Y-axis machine lock ON
Register R72, bit 5	M-, S-, T-code lock ON
Register R72, bit 6	Z-axis machine lock ON

4. Macro interface output system variables (#1100 to #1135)

You can send an interface output signal by assigning a value to the appropriate variables number (#1100 to #1135).

All output signals can take either 0 or 1.

You can also send all output signals of the variables from #1100 to #1131 at the same time by assigning a value to variables number 1132. In addition to the data writing for offsetting the #1100 to #1135 output signals, the reading of the output signal status can be done.

System variable	Points	Interface output signal	System variable	Points	Interface output signal
#1100	1	Register R172, bit 0	#1116	1	Register R173, bit 0
#1101	1	Register R172, bit 1	#1117	1	Register R173, bit 1
#1102	1	Register R172, bit 2	#1118	1	Register R173, bit 2
#1103	1	Register R172, bit 3	#1119	1	Register R173, bit 3
#1104	1	Register R172, bit 4	#1120	1	Register R173, bit 4
#1105	1	Register R172, bit 5	#1121	1	Register R173, bit 5
#1106	1	Register R172, bit 6	#1122	1	Register R173, bit 6
#1107	1	Register R172, bit 7	#1123	1	Register R173, bit 7
#1108	1	Register R172, bit 8	#1124	1	Register R173, bit 8
#1109	1	Register R172, bit 9	#1125	1	Register R173, bit 9
#1110	1	Register R172, bit 10	#1126	1	Register R173, bit 10
#1111	1	Register R172, bit 11	#1127	1	Register R173, bit 11
#1112	1	Register R172, bit 12	#1128	1	Register R173, bit 12
#1113	1	Register R172, bit 13	#1129	1	Register R173, bit 13
#1114	1	Register R172, bit 14	#1130	1	Register R173, bit 14
#1115	1	Register R172, bit 15	#1131	1	Register R173, bit 15

System variable	Points	Interface output signal
#1132	32	Register R172 and R173
#1133	32	Register R174 and R175
#1134	32	Register R176 and R177
#1135	32	Register R178 and R179

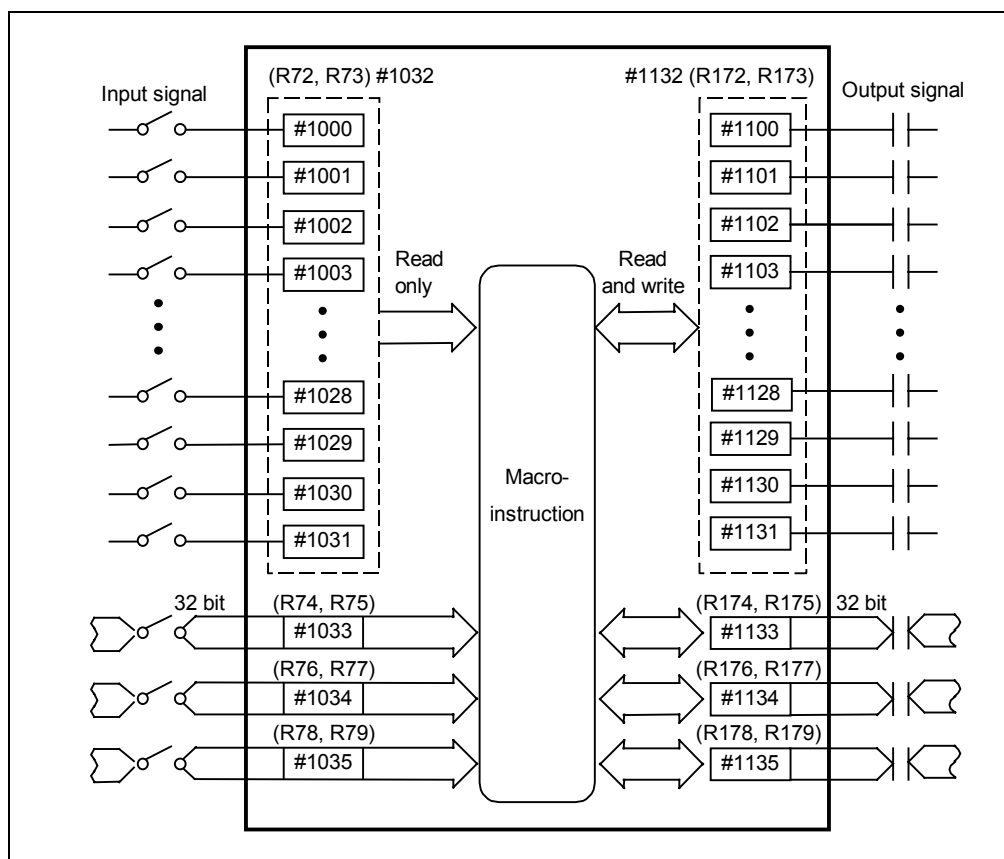
Note 1: Data of the system variables from #1100 to #1135 is saved according to the logical level (1 or 0) of the signal that has been lastly sent. The saved data is cleared by power-on/off automatically.

Note 2: The following applies if a data other than 1 or 0 is assigned to the variables from #1100 to #1131:

<empty> is regarded as equal to 0.

Data other than 0 and <empty> is regarded as equal to 1.

Data less than 0.00000001, however, is regarded as undefined.



5. Tool offset

Standard 128 sets
Optional 512 sets

Range of variable Nos.		Type A	Type B
#10001 - #10000+n	#2001 - #2000+n	○	○ Length Geom. offset
#11001 - #11000+n	#2201 - #2200+n	×	○ Length Wear compensation
#16001 - #16000+n *(#12001 - #12000+n)	#2401 - #2400+n	×	○ Dia. Geom. offset
#17001 - #17000+n *(#13001 - #13000+n)	#2601 - #2600+n	×	○ Dia. Wear compensation

*: The numbers of variables used for tool offset depend upon a parameter:

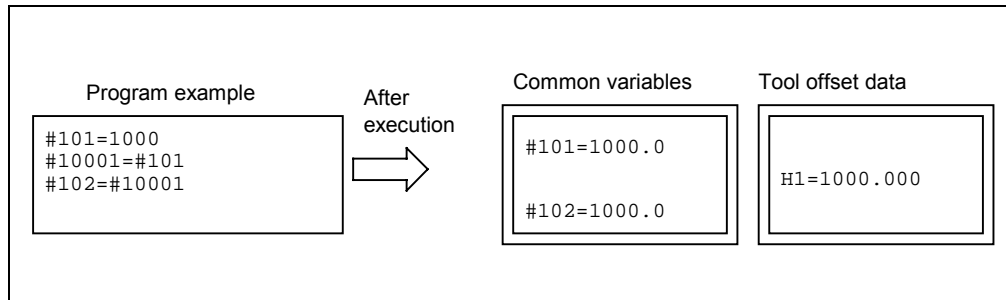
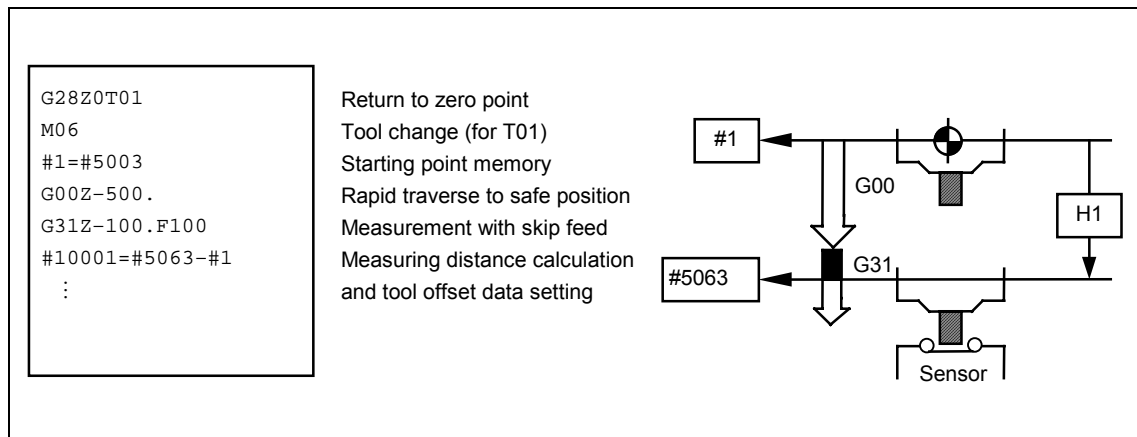
F96 bit 0 = 0: #16001 to #16000+n, and #17001 to #17000+n
= 1: #12001 to #12000+n, and #13001 to #13000+n.

Using variables numbers, you can read tool data or assign data.

Usable variables numbers are of the order of either #10000 or #2000. For the order of #2000, however, only up to 200 sets of tool offsets can be used.

The last three digits of a variables number denote a tool offset number.

As with other variables, tool offset data is to contain the decimal point. The decimal point must therefore be included if you want to set data that has decimal digits.

**Example:** Tool offset data measuring

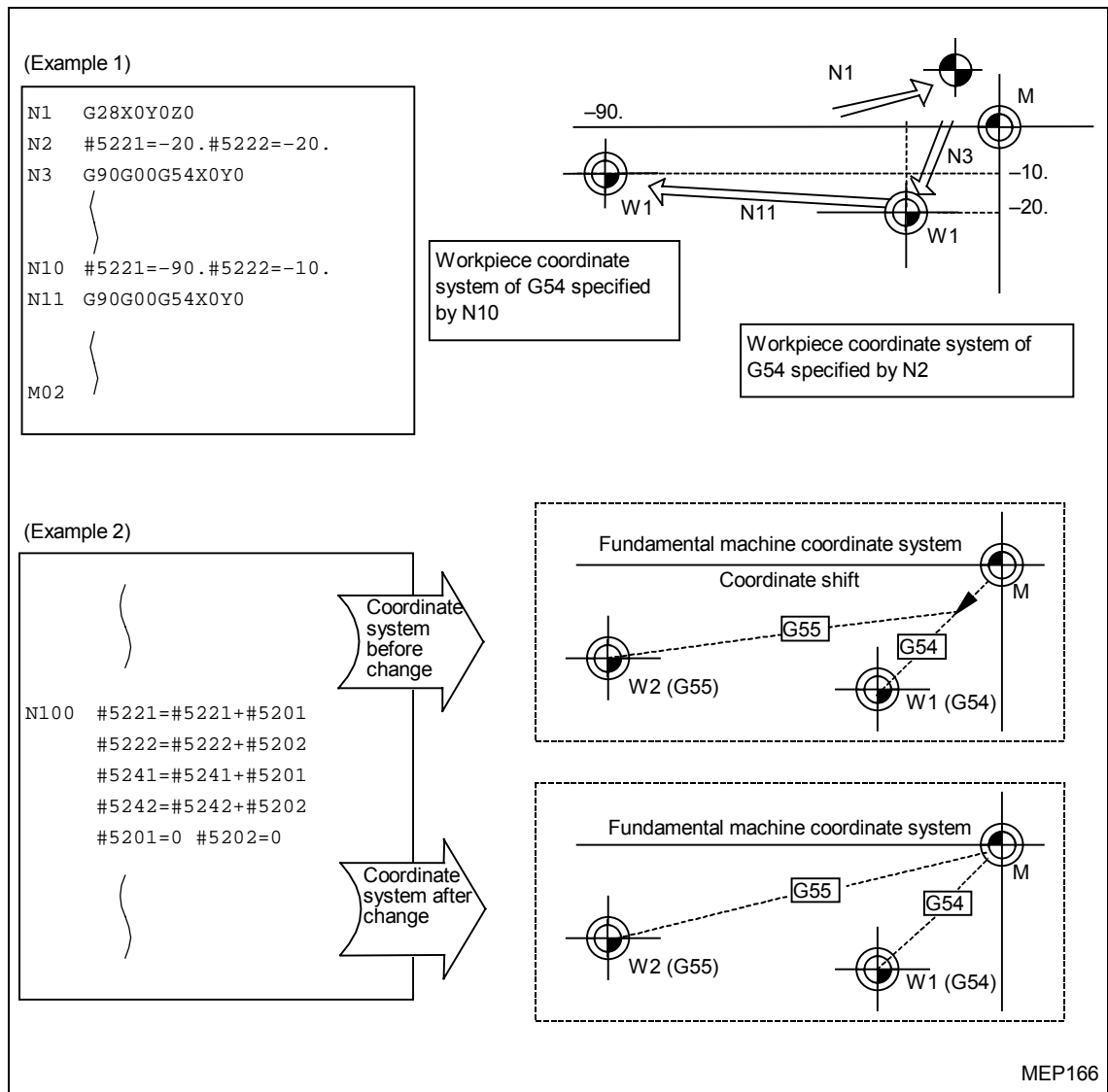
Note: The example shown above does not allow for any skip sensor signal delay. Also, #5003 denotes the position of the starting point of the Z-axis, and #5063 denotes the skip coordinate of the Z-axis, that is, the position at which a skip signal was input during execution of G31.

6. Workpiece coordinate system offset

Using variables numbers from 5201 to 5336, you can read workpiece coordinate system offset data or assign data.

Note: The number of controllable axes depends on the machine specifications.

Axis No. Data name	1st axis	2nd axis	3rd axis		16th axis	Remarks
SHIFT	#5201	#5202	#5203		#5216	An external data input/output optional spec. is required.
G54	#5221	#5222	#5223		#5236	A workpiece coordinate system offset feature is required.
G55	#5241	#5242	#5243		#5256	
G56	#5261	#5262	#5263		#5276	
G57	#5281	#5282	#5283		#5296	
G58	#5301	#5302	#5303		#5316	
G59	#5321	#5322	#5323		#5336	



The example 2 shown above applies only when coordinate shift data is to be added to the offset data of a workpiece coordinate system (G54 or G55) without changing the position of the workpiece coordinate system.

[Additional workpiece coordinate system offset]

Variables numbered 70001 to 75996 can be used to read or assign additional workpiece coordinate system offsetting dimensions. The variable number for the k-th axis origin of the “Pn” coordinate system can be calculated as follows:

$$70000 + (n - 1) \times 20 + k$$

Note: The total number of controllable axes depends on the machine specifications.

Axis No. Data name	1st axis	2nd axis	3rd axis	4th axis	16th axis	Remarks
G54.1P1	#70001	#70002	#70003	#70004	#70016	Only available with the optional function for additional coordinate system offset.
G54.1P2	#70021	#70022	#70023	#70024	#70036	
G54.1P299	#75961	#75962	#75963	#75964	#75976	
G54.1P300	#75981	#75982	#75983	#75984	#75996	

Alternatively, variables numbered 7001 to 7956 can be used to read or assign additional work-piece coordinate system offsetting dimensions. The variable number for the k-th axis origin of the “Pn” coordinate system can be calculated as follows:

$$7000 + (n - 1) \times 20 + k$$

Note: The total number of controllable axes depends on the machine specifications.

Axis No. Data name	1st axis	2nd axis	3rd axis	4th axis		16th axis	Remarks
G54.1P1	#7001	#7002	#7003	#7004		#7016	Only available with the optional function for additional coordinate system offset.
G54.1P2	#7021	#7022	#7023	#7024		#7036	
G54.1P3	#7041	#7042	#7043	#7044		#7056	
G54.1P48	#7941	#7942	#7943	#7944		#7956	

7. NC alarm (#3000)

The NC unit can be forced into an alarm status using variables number 3000.

#3000 = 70 (CALL#PROGRAMMER#TEL#530)
<div style="display: flex; justify-content: space-around; align-items: center;"> <div style="text-align: center;"> <div style="border-bottom: 1px solid black; width: 100px; margin: 0 auto;"></div> <p>Alarm No.</p> </div> <div style="text-align: center;"> <div style="border-bottom: 1px solid black; width: 100px; margin: 0 auto;"></div> <p>Alarm message</p> </div> </div>

The setting range for the alarm No. is from 1 to 6999.

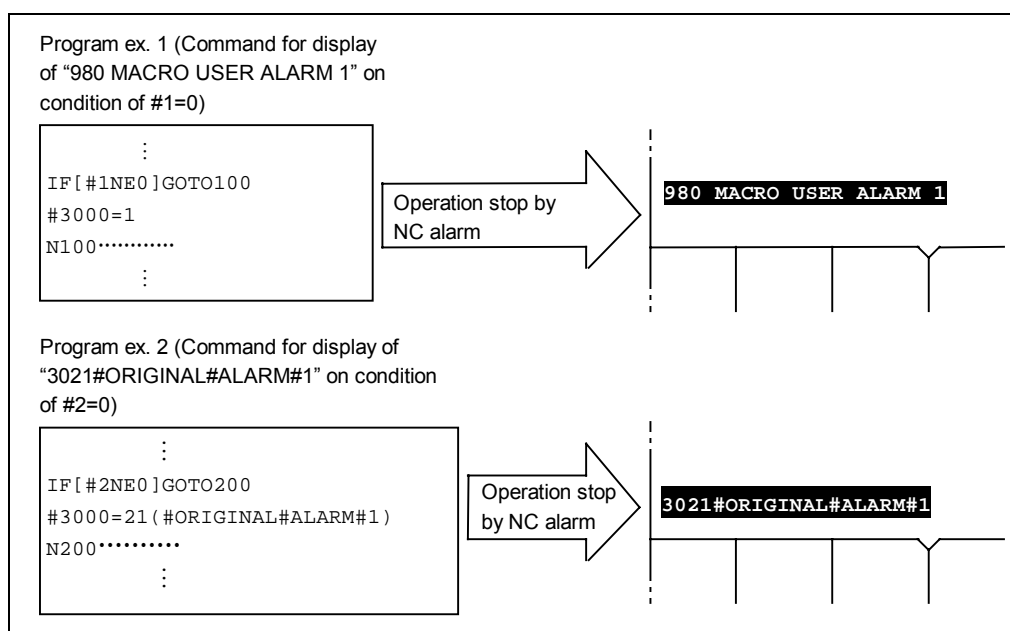
The maximum available length of the alarm message is 31 characters.

Note: The type of alarm message displayed on the screen depends on the designated alarm number, as indicated in the following table.

Designated alarm No.	Displayed alarm No.	Displayed alarm message
1 to 20	[Designated alarm No.] + 979	Message preset for the displayed alarm No. *1
21 to 6999	[Designated alarm No.] + 3000	Designated alarm message as it is *2

*1 Refers to alarm Nos. 980 to 999 whose messages are preset as indicated in Alarm List.

*2 Display of a message as it is set in the macro statement.

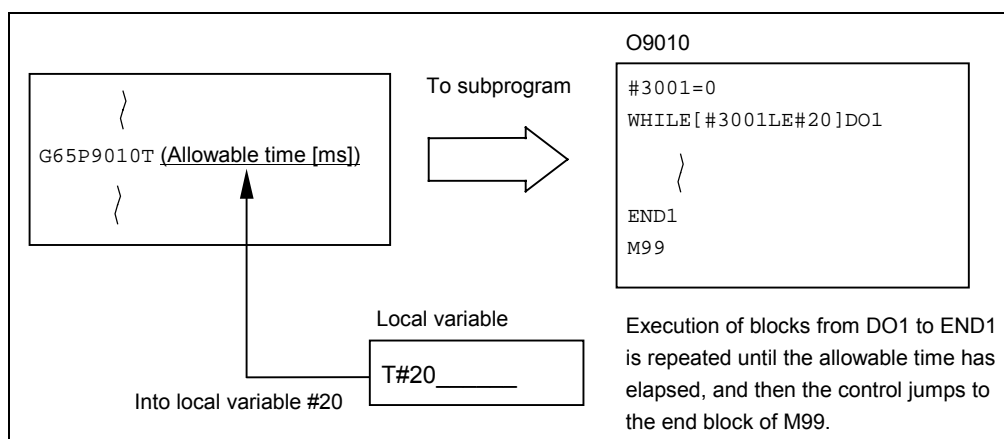


8. Integrated time (#3001, #3002)

Using variables #3001 and #3002, you can read the integrated time existing during automatic operation or assign data.

Type	Variable No.	Unit	Data at power-on	Initialization	Counting
Integrated time 1	3001	ms	Same as at power-off	Data is assigned in variables.	Always during power-on
Integrated time 2	3002				During auto-starting

The integrated time is cleared to 0 after having reached about 2.44×10^{11} ms (about 7.7 years).



9. Validation/invalidation of single-block stop or auxiliary-function finish signal wait (#3003)

Assigning one of the values listed in the table below to variables number 3003 allows single-block stop to be made invalid at subsequent blocks or the program to be advanced to the next block without ever having to wait for the arrival of an auxiliary-function code (M, S, T, or B) execution finish signal (FIN).

#3003	Single block stop	Auxiliary-function completion signal
0	Effective	Wait
1	Ineffective	Wait
2	Effective	No wait
3	Ineffective	No wait

Note: Variable #3003 is cleared to 0 by resetting.

10. Validation/invalidation of feed hold, feed rate override, or G09 (#3004)

Feed hold, feed rate override, or G09 can be made valid or invalid for subsequent blocks by assigning one of the values listed in the table below to variables number 3004.

#3004 Contents (Value)	Bit 0	Bit 1	Bit 2
	Feed hold	Feed rate override	G09 check
0	Effective	Effective	Effective
1	Ineffective	Effective	Effective
2	Effective	Ineffective	Effective
3	Ineffective	Ineffective	Effective
4	Effective	Effective	Ineffective
5	Ineffective	Effective	Ineffective
6	Effective	Ineffective	Ineffective
7	Ineffective	Ineffective	Ineffective

Note 1: Variable #3004 is cleared to 0 by resetting.

Note 2: Each of the listed bits makes the function valid if 0, or invalid if 1.

11. Program stop (#3006)

Use of variables number 3006 allows the program to be stopped after execution of the immediately preceding block.

Format:

#3006 = 1 (CHECK OPERAT)

Character string to be displayed

Additional setting of a character string (in 29 characters at maximum) in parentheses allows the required stop message to be displayed on the monitor.

12. Mirror image (#3007)

The mirror image status of each axis at one particular time can be checked by reading variables number 3007.

Variable #3007 has its each bit crosskeyed to an axis, and these bits indicate that:

If 0, the mirror image is invalid.

If 1, the mirror image is valid.

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
Axis no.											6	5	4	3	2	1

13. G-command modal status

The G-command modal status in a pre-read block can be checked using variables numbers from 4001 to 4027. For variables numbers from #4201 to #4227, the modal status of the block being executed can be checked in a similar manner to that described above.

Variable Nos.		Function
Block pre-read	Block executed	
#4001	#4201	Interpolation mode G0-G3 : 0-3, G2.1 : 2.1, G3.1 : 3.1
#4002	#4202	Plane selection G17 : 17, G18 : 18, G19 : 19
#4003	#4203	Absolute/Incremental programming G90/91 : 90/91
#4004	#4204	Pre-move stroke check G22 : 22, G23 : 23
#4005	#4205	Feed specification G94 : 94, G95 : 95
#4006	#4206	Inch/metric selection G20/21 : 20/21
#4007	#4207	Tool radius compensation G40 : 40, G41 : 41, G42 : 42
#4008	#4208	Tool length offset G43/44 : 43/44, G43.4 : 43.4, G43.5 : 43.5, G49 : 49
#4009	#4209	Fixed cycle G80 : 80, G273/274 : 273/274, G276 : 276, G81-G89 : 81-89
#4010	#4210	Returning level G98 : 98, G99 : 99
#4011	#4211	Scaling OFF/ON G50/51 : 50/51
#4012	#4212	Workpiece coordinate system G54-G59 : 54-59, G54.1 : 54.1
#4013	#4213	Mode of machining (feed) G61-64 : 61-64, G61.1 : 61.1, G61.4 : 61.4
#4014	#4214	Macro modal call G66 : 66, G66.1 : 66.1, G67 : 67
#4015	#4215	Shaping function G40.1 : 40.1, G41.1 : 41.1, G42.1 : 42.1
#4016	#4216	Programmed coordinate conversion ON/OFF G68/69 : 68/69
#4017	#4217	
#4018	#4218	
#4019	#4219	Mirror image by G-codes (5-surf. machining) G17.1-17.9 : 17.1-17.9, G45.1 : 45.1, G49.1 : 49.1, G50.1 : 50.1, G51.1 : 51.1
#4020	#4220	Cross machining control G110 : 110, G110.1 : 110.1, G111 : 111
#4021	#4221	
#4022	#4222	
#4023	#4223	Polygonal machining and Hobbing G50.2 : 50.2, G51.2 : 51.2, G113 : 113, G114.3 : 114.3
#4024	#4224	
#4025	#4225	
#4026	#4226	
#4027	#4227	Dynamic offsetting II G54.2 : 54.2

14. Other modal information

Modal information about factors other than the G-command modal status in a pre-read block can be checked using variables numbers from 4101 to 4132. For variables numbers from #4301 to #4330, the modal information of the block being executed can be checked in a similar manner to that described above.

Variable Nos.		Modal information	Variable Nos.		Modal information
Block pre-read	Block executed		Block pre-read	Block executed	
#4101	#4301		#4113	#4313	Miscellaneous function...M
#4102	#4302	No. 2 miscellaneous function	#4114	#4314	Sequence No. ...N
#4103	#4303		#4115	#4315	Program No. ...O
#4104	#4304		#4116	#4316	
#4105	#4305		#4117	#4317	
#4106	#4306		#4118	#4318	
#4107	#4307	Tool radius comp. data No. ...D	#4119	#4319	Spindle function ...S
#4108	#4308		#4120	#4320	Tool function ...T
#4109	#4309	Rate of feed ...F	#4130	#4330	Addt. workpiece coordinate system G54-G59: 0, G54.1P1-P300: 1-300
#4110	#4310		#4131		Surface type Upper: 0, 0°/180°: 1, 90°/270°: 2
#4111	#4311	Tool length offset data No. ...H			
#4112	#4312		#4132		Machining surface Upper: 5, 0°: 6, 90°: 7, 180°: 8, 270°: 9

Note 1: Use the variable #4315 from the second block of the program. If it is used in the first block, the reading of the program number is not successful.

Note 2: The variables #4115 and #4315 are only effective for programs whose ID-number or name consists of numerals only. During execution of a program whose name contains even one single character other than numerals, the reading in question remains zero (0).

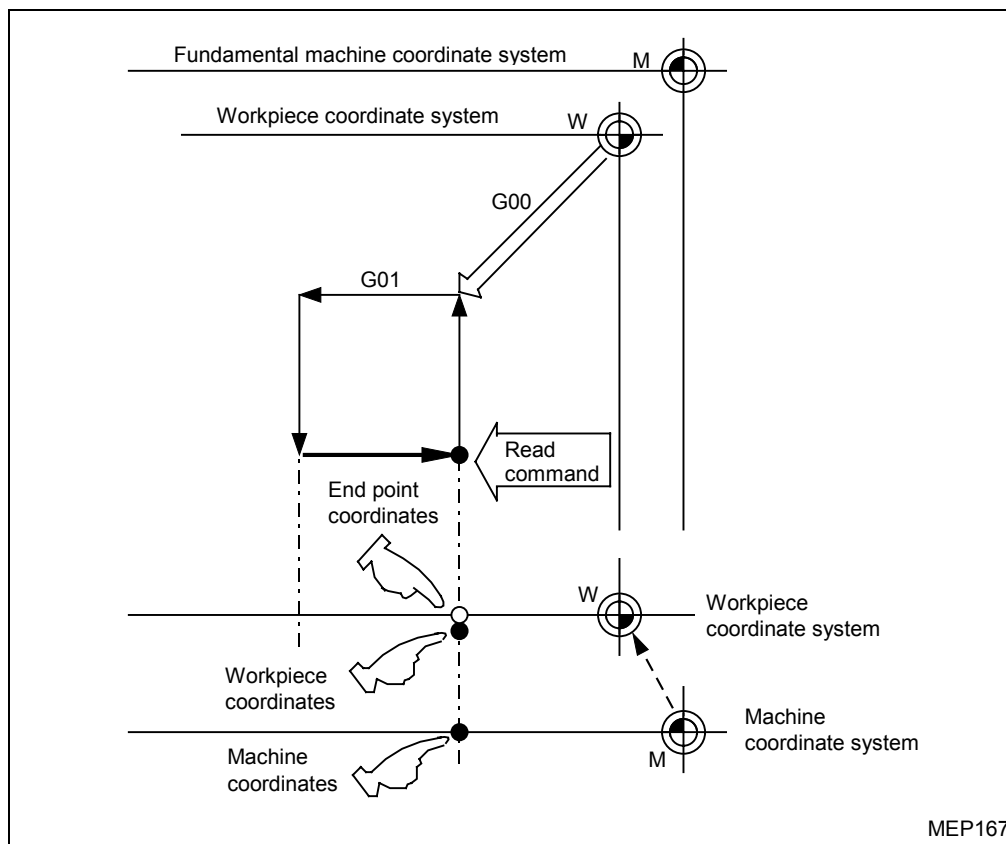
15. Position information

Using variables numbers from #5001 to #5116, you can check the ending-point coordinates of the previous block, machine coordinates, workpiece coordinates, skip coordinates, tool position offset coordinates, and servo deviations.

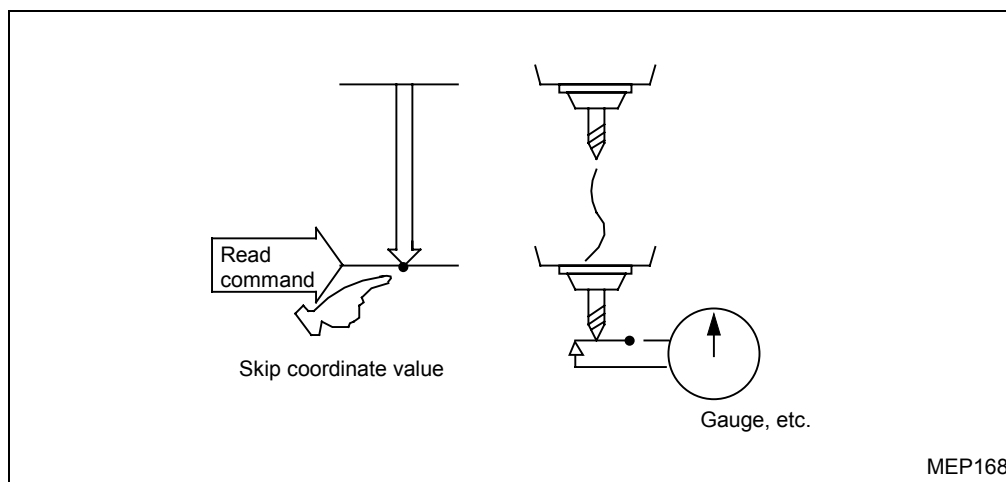
Position information Axis No.	End point coordinates of previous block	Machine coordinate	Workpiece coordinate	Skip coordinate	Tool position offset coordinates	Servo deviation amount
1	#5001	#5021	#5041	#5061	#5081	#5101
2	#5002	#5022	#5042	#5062	#5082	#5102
3	#5003	#5023	#5043	#5063	#5083	#5103
16	#5016	#5036	#5056	#5076	#5096	#5116
Remarks (Reading during move)	Possible	Impossible	Impossible	Possible	Impossible	Possible

Note: The number of controllable axes depends on the machine specifications.

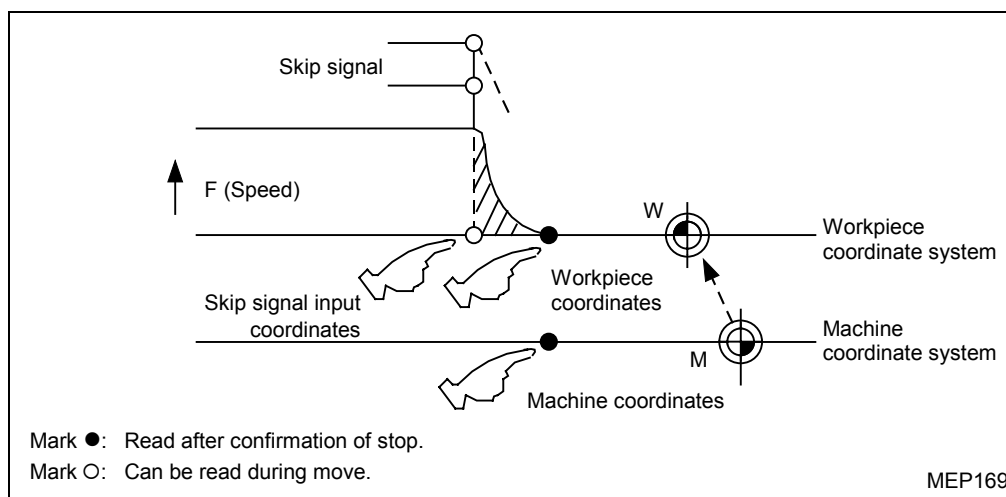
1. The ending-point coordinates and skip coordinates read will be those related to the workpiece coordinate system.
2. Ending-point coordinates, skip coordinates, and servo deviations can be checked even during movement. Machine coordinates, workpiece coordinates, and tool position offset coordinates must be checked only after movement has stopped.



3. Skip coordinates denote the position at which a skip signal has turned on at the block of G31. If the skip signal has not turned on, skip coordinates will denote the corresponding ending-point position.



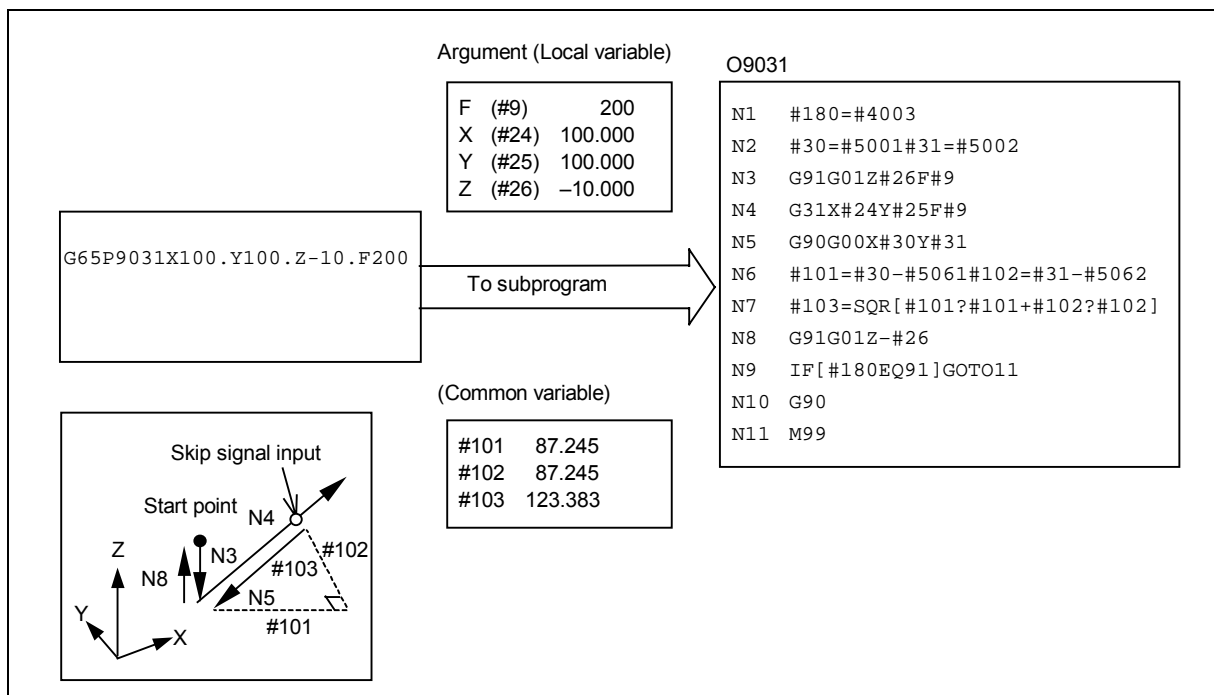
4. The ending-point position denotes the tool tip position which does not allow for any tool offsets, whereas machine coordinates, workpiece coordinates, and skip coordinates denote the tool reference-point position which allows for tool offsets.



The input coordinates of a skip signal denote the position within the workpiece coordinate system. The coordinates stored in variables from #5061 to #5066 are those existing when skip signals were input during movement of the machine. These coordinates can therefore be read at any time after that. See Chapter 15 for further details.

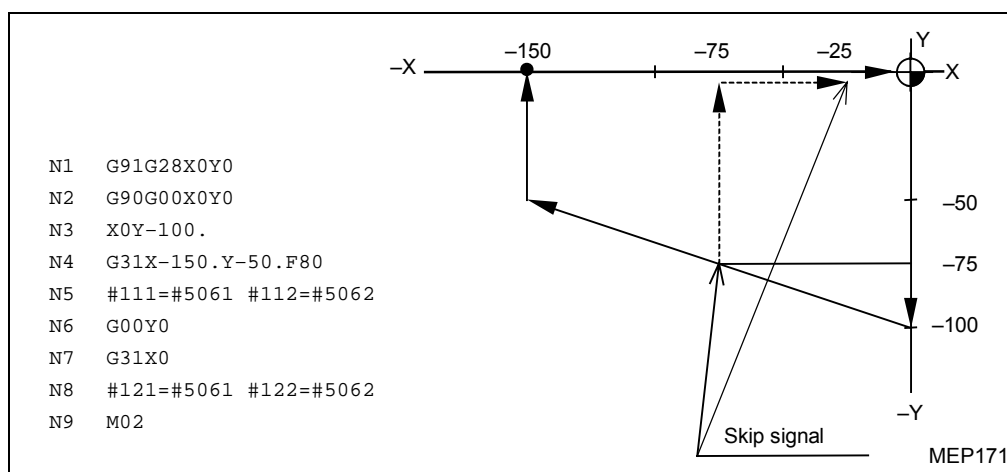
Example 1: Workpiece position measurement:

The following shows an example of measuring the distance from a reference measurement point to the workpiece end:



#101	X-axis measuring amount	N1	Modal data storage of G90/G91
#102	Y-axis measuring amount	N2	X, Y starting point data storage
#103	Measuring line linear amount	N3	Z-axis entry
#5001	X-axis measuring start point	N4	X, Y measuring (Stop at skip input)
#5002	Y-axis measuring start point	N5	Return to X, Y starting point
#5061	X-axis skip input point	N6	X, Y measuring incremental data calculation
#5062	Y-axis skip input point	N7	Measuring line linear amount calculation
		N8	Z-axis escape
		N9, N10	Modal return of G90/G91
		N11	Return from subprogram

Example 2: Skip signal input coordinates reading:



$$\begin{aligned}
 \#111 &= -75. + \varepsilon & \#112 &= -75. + \varepsilon \\
 \#121 &= -25. + \varepsilon & \#122 &= -75. + \varepsilon
 \end{aligned}$$

where ε denotes an error due to response delay. (See Chapter 15 on skip functions for further details.) Variable #122 denotes the skip signal input coordinate of N4 since N7 does not have a Y-command code.

16. TNo. (#51999) and Number of index (#3020) of the spindle tool

Variables numbered 51999 and 3020 can be used to read the tool number, and the number of tool data index, of the tool mounted in the spindle. The number of index can be used, instead of TNo., in reading the tool data of a particular tool with the aid of macro variables.

System variable	Description
#51999	Tool number of the spindle tool
#3020	Number of tool data index of the spindle tool

Note 1: These system variables are read-only variables.

Note 2: During tool path checking operation both variables (#51999 and #3020) store data with mere reference to the "TNo." programmed in a T-code and, therefore, remain zero (0) when the program concerned has no T-codes used.

17. Number of tool data index (#3022 and #3023)

Variables numbered 3022 and 3023 can be used to read the number of tool data index of any desired tool.

Variables	Description
#3022	Designation of the required tool (for writing only). Use the integral and decimal parts respectively for specifying the required tool with its number and suffix. #3022 = ○○○. ΔΔ ○○○: Tool number (TNo.) ΔΔ: Suffix
#3023	Number of index of the specified tool (for reading only). Use this variable to read out the index line number of the tool specified by the variable #3022. The reading in #3023 is zero (0) if there is no corresponding tool registered in the memory.

Example:

TNo.		#3022 setting	Reading in #3023
1	A	1.01	21
2	B	2.02	24
3	C	3.03	40
4	A	4.61	31
5	B	5.62	34
6	C	6.63	35
7	H	7.08	15
8	V	8.22	18
9	Z	9.26	19
:	:	:	:
:	:	:	:
Failure		—	0

Note: The number of tool data index undergoes changes by an exchange of data sets using the [TOOL DATA MOVE] menu function on the **TOOL DATA** display. It is necessary, therefore, to obtain the number of tool data index in order to correctly fetch a tool's information item from the **TOOL DATA** display with the aid of system variables.

18. MAZATROL tool data

Using variables tabulated below, MAZATROL tool data can be read or written, as required.

Tool quantity: 960 (maximum)

The maximum applicable tool quantity depends on the machine specifications.

(n = Number of index of the tool)

Usable variables numbers	MAZATROL tool data
#60001 to #60000 + n	Tool length
#61001 to #61000 + n	Tool diameter
#62001 to #62000 + n	Tool life flag (1: ON, 0: OFF)
#63001 to #63000 + n	Tool damage flag (1: ON, 0: OFF)
#64001 to #64000 + n	Wear compensation X
#65001 to #65000 + n	Wear compensation Y
#66001 to #66000 + n	Wear compensation Z
#67001 to #67000 + n	Group number

Note 1: During tool path check, tool data can be read but cannot be written.

Note 2: Tool life flags (variables numbers of the order of #62000) and tool damage flags (likewise, the order of #63000) can take either 1 or 0 as their logical states (1 for ON, 0 for OFF).

Note 3: Tool life flags (variables numbers of the order of #62000) only refer to tool life management according to the tool's application time.

19. EIA/ISO tool data

Using variables numbers tabulated below, EIA/ISO tool data (tool life management data) can be read or updated, as required.

Tool quantity: 960 (maximum)

The maximum applicable tool quantity depends on the machine specifications.

(n = Number of index of the tool)

System variables	Corresponding data
#40001 to #40000 + n	Length offset data number or offset amount
#41001 to #41000 + n	Radius compensation data number or compensation amount
#42001 to #42000 + n	Tool life flag (1: ON, 0: OFF)
#43001 to #43000 + n	Tool damage flag (1: ON, 0: OFF)
#44001 to #44000 + n	Tool data flag (See the table below.)
#45001 to #45000 + n	Tool operation time (s)
#46001 to #46000 + n	Tool life time (s)

Note 1: During tool path check, tool data can be read but cannot be written.

Note 2: Tool life flags (variables numbers of the order of #42000) and tool damage flags (likewise, the order of #43000) can take either 1 or 0 as their logical states (1 for ON, 0 for OFF).

Note 3: The identification between number and amount of tool length offset or radius compensation is made by referring to the tool data flag.

Tool data flag	bit 0	bit 1	bit 2	bit 3
Length offset data No.	0	0	—	—
Length offset amount	0	1	—	—
Radius comp. data No.	—	—	0	0
Radius comp. amount	—	—	0	1

20. Date and time (Year-month-day and hour-minute-second)

Variables numbered 3011 and 3012 can be used to read date and time data.

Variable Nos.	Description
#3011	Date (Year-month-day)
#3012	Time (Hour-minute-second)

Example: If the date is December 15, 1995 and the time is 16:45:10, data is set as follows in the corresponding system variables:

#3011 = 951215

#3012 = 164510.

21. Total number of machined parts and the number of parts required

Variables numbered 3901 and 3902 can be used to read or assign the total number of machined parts and the number of parts required.

Variable Nos.	Description
#3901	Total number of machined parts
#3902	Number of parts required

Note 1: These variables must be integers from 0 to 9999.

Note 2: Data reading and writing by these variables is surely suppressed during tool path checking.

22. Setting and using variables names

Any variables name can be assigned to each of common variables #500 through #519. The variables name, however, must be of seven alphanumeric or less that begin with a letter of the alphabet.

Format:	
SETVNn	[NAME1, NAME2,]
	Starting number of the variable to be named
	Name of #n (Variables name)
	Name of #n + 1 (Variables name)

Each variables name must be separated using the comma (,).

Detailed description

- Once a variables name has been set, it remains valid even after power-off.
- Variables in a program can be called using the variables names. The variable to be called must, however, be enclosed in brackets ([]).

Example: G01X[#POINT1]
[#TIMES]=25

- Variables names can be checked on the **USER PARAMETER No. 1** display. The names assigned to variables #500 to #519 are displayed at F47 to F66.

Example: Program SETVN500[ABC,EFG]

On the display

```

F46      0
F47  ABC  ← Variables name assigned to #500
F48  EFG  ← Variables name assigned to #501
F49
F50
F51
F52
F53
F54
F55
F56
F57
F58
F59
F60
F61
F62
F63
F64
F65
F66

```

23. Contents of the S12 or S23 parameters (#3200 and #3212 or #3223)

Variables numbered 3200, 3212 and 3223 can be used to read the settings of particular S parameters. Use #3200 beforehand to specify the desired axis.

#3200 setting	Description
1 to 16	Axis number

Example: #3200 = 1;

Designation of the first significant axis setting (normally: X) within the system.

Variable Nos.	S parameter
#3212	S12
#3223	S23

Note 1: #3200 is initialized (to "1") by NC-resetting.

Note 2: It is not necessary to repeat writing into #3200 next time the reading with #3212 or #3223 is to be done for the same axis.

Note 3: Read #3200 as required to check its last setting.

Note 4: The setting for #3200 is not checked for the appropriateness (as to whether the designated axis is preset properly) at the block of #3200, but results in an alarm (**809 ILLEGAL NUMBER INPUT**) being caused at the reading block of #3212 or #3223 when the setting is found inappropriate.

24. Workpiece setup error correction values

Using variables tabulated below, it is possible to read and write the values used for workpiece setup error correction.

	Common	No. 1	No. 2	No. 3	No. 4	No. 5	No. 6	No. 7
Δx	#5801	#5811	#5821	#5831	#5841	#5851	#5861	#5871
Δy	#5802	#5812	#5822	#5832	#5842	#5852	#5862	#5872
Δz	#5803	#5813	#5823	#5833	#5843	#5853	#5863	#5873
Δa	—	#5814	#5824	#5834	#5844	#5854	#5864	#5874
Δb	—	#5815	#5825	#5835	#5845	#5855	#5865	#5875
Δc	—	#5816	#5826	#5836	#5846	#5856	#5866	#5876
Rotat. axis coord. 1	#5807	#5817	#5827	#5837	#5847	#5857	#5867	#5877
Rotat. axis coord. 2	#5808	#5818	#5828	#5838	#5848	#5858	#5868	#5878

Use #5800 to read the number (1 to 7) of the currently selected data set for workpiece setup error correction.

Note: An attempt to overwrite the system variable #5800 will only lead to an alarm (**1821 UNWRITABLE SYSTEM VARIABLE**).

25. Settings in an indexing unit (for VERSATECH machines only)

Using variables tabulated below, it is possible to read the setting items of an indexing unit (INDEX) in a MAZATROL program.

Variables	Description	Unit
#550610	TURN POS X	0.0001 mm or 0.00001 in
#550612	TURN POS Y	0.0001 mm or 0.00001 in
#550614	TURN POS Z	0.0001 mm or 0.00001 in
#550616	TURN POS W	0.0001 mm or 0.00001 in
#550618	ANGLE B	0.0001°
#550620	ANGLE C	0.0001°
#550698	W-POS	0.0001 mm or 0.00001 in

Note: These system variables are read-only variables.

13-10-5 Arithmetic operation commands

Various operations can be carried out between variables using the following format.

`#i = <expression>`

where <expression> must consist of a constant(s), a variable(s), a function(s), or an operator(s).

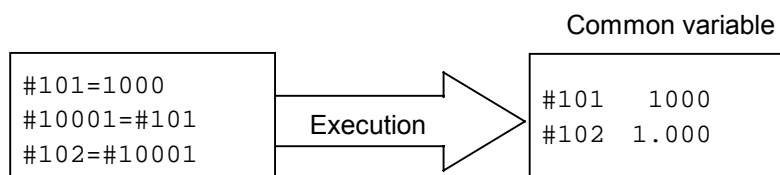
In the table given below, constants can be used instead of #j and/or #k.

[1] Definition/replacement of variables	#i=#j	Definition/replacement
[2] Additional-type operations	#i=#j+#k	Addition
	#i=#j-#k	Subtraction
	#i=#jOR#k	Logical addition (For each of 32 bits)
	#i=#jXOR#k	Exclusive OR (For each of 32 bits)
[3] Multiplicative-type operations	#i=#j*#k	Multiplication
	#i=#j/#k	Division
	#i=#jMOD#k	Surplus
	#i=#jAND#k	Logical product (For each of 32 bits)
[4] Functions	#i=SIN[#k]	Sine
	#i=COS[#k]	Cosine
	#i=TAN[#k]	Tangent (tanq is used as sinq/cosq.)
	#i=ATAN[#j]	Arc-tangent (Either ATAN or ATN can be used.)
	#i=ACOS[#j]	Arc-cosine
	#i=SQRT[#k]	Square root (Either SQRT or SQR is available.)
	#i=ABS[#k]	Absolute value
	#i=BIN[#k]	BINARY conversion from BCD
	#i=BCD[#k]	BCD conversion from BINARY
	#i=ROUND[#k]	Rounding to the nearest whole number (Either ROUND or RND is available.)
	#i=FIX[#k]	Cutting away any decimal digits
	#i=FUP[#k]	Counting any decimal digits as 1s
	#i=LN[#k]	Natural logarithm
	#i=EXP[#k]	Exponent with the base of e (= 2.718 ...)

Note 1: In principle, data without a decimal point is handled as data that has a decimal point.
(Example: 1 = 1.000)

Note 2: Offsets from variable #10001, workpiece coordinate system offsets from variable #5201, and other data become data that has a decimal point. If data without a decimal point is defined using these variables numbers, therefore, a decimal point will also be assigned to the data.

Example:

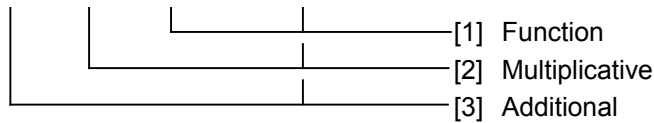


Note 3: The <expression> after a function must be enclosed in brackets ([]).

1. Operation priority

Higher priority is given to functions, multiplicative operations, and additive operations, in that order.

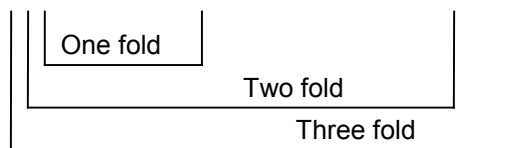
```
#101=#111+#112*SIN[#113]
```



2. Specifying an operational priority level

The part to which the first level of operation priority is to be given can be enclosed in brackets ([]). Up to five sets of brackets, including those of functions, can be used for one expression.

```
#101=SQRT[[[#111-#112]*SIN[#113]+#114]*#15]
```



3. Examples of operation instructions

[1] Main program and argument specification	G65 P100 A10 B20. #101=100.000 #102=200.000	#1 10.000 #2 20.000 #101 100.000 #102 200.000
[2] Definition, replacement =	#1=1000 #2=1000. #3=#101 #4=#102 #5=#5081	#1 1000.000 #2 1000.000 #3 100.000 #4 200.000 #5 -10.000
		Data of common variables Offset amount
[3] Addition, subtraction + -	#11=#1+1000 #12=#2-50. #13=#101+#1 #14=#5081-3. #15=#5081+#102	#11 2000.000 #12 950.000 #13 1100.000 #14 -13.000 #15 190.000
[4] Logical addition OR	#3=100 #4=#3OR14	#3 = 01100100 14 = 00001110 #4 = 01101110 = 110
[5] Exclusive OR XOR	#3=100 #4=#3XOR14	#3 = 01100100 14 = 00001110 #4 = 01101010 = 106
[6] Multiplication, Division * /	#21=100*100 #22=100.*100 #23=100*100. #24=100.*100. #25=100/100 #26=100./100 #27=100/100. #28=100./100. #29=#5081*#101 #30=#5081/#102	#21 10000.000 #22 10000.000 #23 10000.000 #24 10000.000 #25 1.000 #26 1.000 #27 1.000 #28 1.000 #29 -1000.000 #30 -0.050
[7] Surplus MOD	#31=#19MOD#20	#19 = 48 #20 9 = 5 surplus 3
[8] Logical product AND	#9=100 #10=#9AND15	#9 = 01100100 15 = 00001111 #10 = 00000100 = 4

[9] Sine SIN	#501=SIN[60] #502=SIN[60.] #503=1000*SIN[60] #504=1000*SIN[60.] #505=1000.*SIN[60] #506=1000.*SIN[60.] Note: SIN[60] is equal to SIN[60.].	#501 0.866 #502 0.866 #503 866.025 #504 866.025 #505 866.025 #506 866.025
[10] Cosine COS	#541=COS[45] #542=COS[45.] #543=1000*COS[45] #544=1000*COS[45.] #545=1000.*COS[45] #546=1000.*COS[45.] Note: COS[45] is equal to COS[45.].	#541 0.707 #542 0.707 #543 707.107 #544 707.107 #545 707.107 #546 707.107
[11] Tangent TAN	#551=TAN[60] #552=TAN[60.] #553=1000*TAN[60] #554=1000*TAN[60.] #555=1000.*TAN[60] #556=1000.*TAN[60.] Note: TAN[60] is equal to TAN[60.].	#551 1.732 #552 1.732 #553 1732.051 #554 1732.051 #555 1732.051 #556 1732.051
[12] Arc-tangent ATAN	#561=ATAN[173205/1000000] #562=ATAN[173.205/100.] #563=ATAN[1.732]	#561 60.000 #562 60.000 #563 59.999
[13] Arc-cosine ACOS	#521=ACOS[100000/141421] #522=ACOS[100./141.421] #523=ACOS[1000/1414.213] #524=ACOS[10./14.142] #525=ACOS[0.707]	#521 45.000 #522 45.000 #523 45.000 #524 44.999 #525 45.009
[14] Square root SQRT	#571=SQRT[1000] #572=SQRT[1000.] #573=SQRT[10.*10.+20.*20.] #574=SQRT[#14*#14+#15*#15] Note: For enhanced accuracy, perform operations within [] as far as possible.	#571 31.623 #572 31.623 #573 22.361 #574 190.444
[15] Absolute value ABS	#576=-1000 #577=ABS[#576] #3=70. #4=-50. #580= ABS[#4-#3]	#576 -1000.000 #577 1000.000 #580 120.000
[16] BIN, BCD	#1=100 #11=BIN[#1] #12=BCD[#1]	#11 64 #12 256
[17] Rounding into the nearest whole number ROUND	#21=ROUND[14/3] #22=ROUND[14./3] #23=ROUND[14/3.] #24=ROUND[14./3.] #25=ROUND[-14/3] #26=ROUND[-14./3] #27=ROUND[-14/3.] #28=ROUND[-14./3.]	#21 5 #22 5 #23 5 #24 5 #25 -5 #26 -5 #27 -5 #28 -5
[18] Cutting away any decimal digits FIX	#21=FIX[14/3] #22=FIX[14./3] #23=FIX[14/3.] #24=FIX[14./3.] #25=FIX[-14/3] #26=FIX[-14./3] #27=FIX[-14/3.] #28=FIX[-14./3.]	#21 4.000 #22 4.000 #23 4.000 #24 4.000 #25 -4.000 #26 -4.000 #27 -4.000 #28 -4.000

[19] Counting any decimal digits as 1s FUP	#21=FUP[14/3]	#21	5.000
	#22=FUP[14./3]	#22	5.000
	#23=FUP[14/3.]	#23	5.000
	#24=FUP[14./3.]	#24	5.000
	#25=FUP[-14/3]	#25	-5.000
	#26=FUP[-14./3]	#26	-5.000
	#27=FUP[-14/3.]	#27	-5.000
	#28=FUP[-14./3.]	#28	-5.000
[20] Natural logarithm LN	#101=LN[5]	#101	1.609
	#102=LN[0.5]	#102	-0.693
	#103=LN[-5]	Alarm 860 CALCULATION IMPOSSIBLE	
[21] Exponent EXP	#104=EXP[2]	#104	7.389
	#105=EXP[1]	#105	2.718
	#106=EXP[-2]	#106	0.135

4. Operation accuracy

The errors listed in the table below are generated by one arithmetic operation, and the error rate increases each time an operation is performed.

Operation format	Mean error	Max. error	Kind of error
a = b + c a = b - c	2.33×10^{-10}	5.32×10^{-10}	Min. $\left \frac{\varepsilon}{c} \right $, $\left \frac{\varepsilon}{b} \right $
a = b • c	1.55×10^{-10}	4.66×10^{-10}	Relative error $\left \frac{\varepsilon}{a} \right $
a = b/c	4.66×10^{-10}	1.86×10^{-9}	
a = \sqrt{b}	1.24×10^{-9}	3.73×10^{-9}	
a = sin b a = cos b	5.0×10^{-9}	1.0×10^{-8}	Absolute error $\left \varepsilon \right $ degree
a = $\tan^{-1} b/c$	1.8×10^{-6}	3.6×10^{-6}	

Note: The function TAN (Tangent) is calculated as SIN/COS (Sine/Cosine).

5. Notes on deterioration of accuracy

A. Addition/subtraction

As for additional-type operations, if an absolute value is subtracted from the other, the relative error cannot be reduced below 10^{-8} .

For example, when the true values (such values, by the way, cannot be substituted directly) of #10 and #20 are as follows:

#10 = 2345678988888.888

#20 = 2345678901234.567

then #10 - #20 = 87654.321 would not result from calculation of #10 - #20. This is because, since the effective number of digits of the variable is eight (decimal), the approximate values of #10 and #20 are:

#10 = 2345679000000.000

#20 = 2345678900000.000

More strictly, internal binary values slightly differ from these values. Actually therefore, a significant error results as follows:

#10 - #20 = 100000.000.

B. Logical relationship

As for EQ, NE, GT, LT, GE and LE, the processing is executed in a similar manner to addition and subtraction, so be careful to errors. For example, to judge whether #10 is equal to #20 of the above example, the conditional expression

IF [#10EQ#20]

is not appropriate due to the errors. In such a case, therefore, give a macro-instruction as shown below to allow for an acceptable tolerance in the judgement on the equality of two values.

IF [ABS[#10 – #20] LT200000]

C. Trigonometric functions

For trigonometric functions, although the absolute error is guaranteed, the relative error is not below 10^{-8} . Be careful, therefore, when carrying out multiplication, or division after trigonometric function operations.

13-10-6 Control commands

The flow of a program can be controlled using IF ~ GOTO ~ and WHILE DO ~ commands.

1. Branching

Format: IF [conditional expression] GOTO n

where n is a sequence number in the same program.

The branching will occur to the block headed by sequence number 'n' if the condition holds, or if the condition does not hold, the next block will be executed.

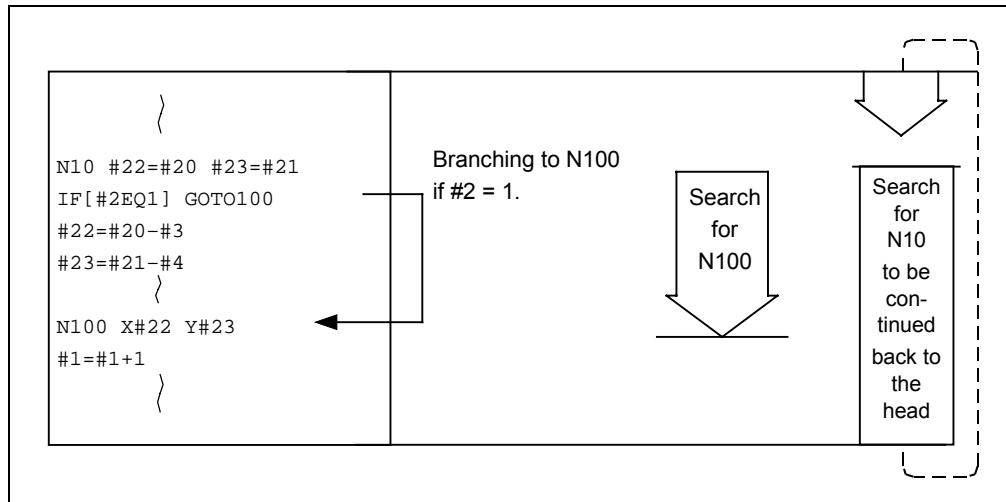
An independent setting of GOTO statement without IF [conditional expression] will perform unconditional branching to the specified block.

The [conditional expression] consists of the following six types:

#i EQ #j	=	(#i is equal to #j.)
#i NE #j	≠	(#i is not equal to #j.)
#i GT #j	>	(#i is larger than #j.)
#i LT #j	<	(#i is smaller than #j.)
#i GE #j	≥	(#i is equal to #j, or larger than #j.)
#i LE #j	≤	(#i is equal to #j, or smaller than #j.)

For GOTO n, n must be a sequence number within the same program. If the sequence number does not exist in that program, an alarm **843 DESIGNATED SNo. NOT FOUND** will occur. An expression or a variable can be used instead of #i, #j, or n.

Sequence number designation Nn must be set at the beginning of the destination block. Otherwise, an alarm **843 DESIGNATED SNo. NOT FOUND** will result. If, however, the block begins with "/" and Nn follows, the program can be branched into that sequence number.



Note: During search for the branching destination sequence number, if the area from the block after “IF ...” to the program end (% code) is searched (forward search) in vain, then the area from the head down to the block before “IF ...” will be searched next (backward search). It will therefore take more time to execute backward search (searching in the opposite direction to the flow of the program) than to execute forward search.

2. Looping

Format:

```

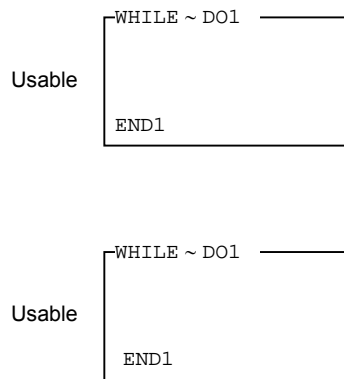
    WHILE [Condition expression] DOm (m = 1, 2, 3 ... 127)
    {
    ENDm
  
```

The area from the next block to the ENDm block loops while the conditional expression holds. If the conditional expression does not hold, control will be transferred to the block after ENDm. In the format shown above, DOm can precede WHILE.

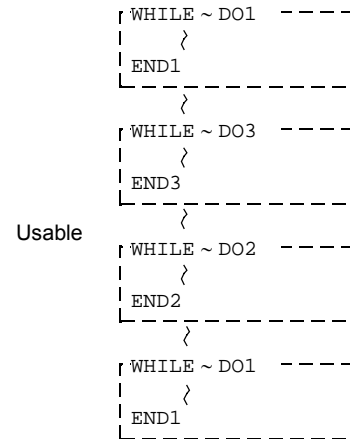
You must always use WHILE [conditional expression] DOm and ENDm in pairs. If you omit WHILE [conditional expression], the area from DOm to ENDm will endlessly loop. In DOm, m (1 to 127) identifies the number of looping. (DO1, DO2, DO3, and so on up to DO127)

The maximum available number of degrees of multiplicity is 27.

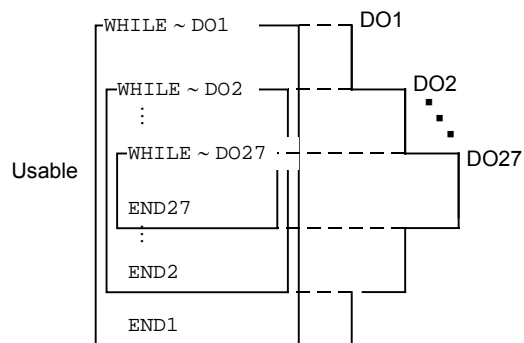
[1] Same identifying No. can be used repeatedly.



[2] The identifying No. of WHILE ~ DOm is arbitrary.

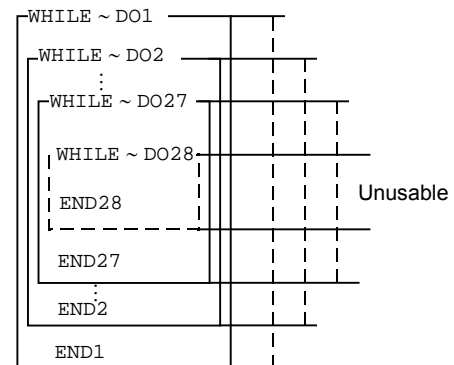


[3] Up to 27 levels of WHILE ~ DOm can be used.
m can be 1 to 127, independent of the depth of nesting.

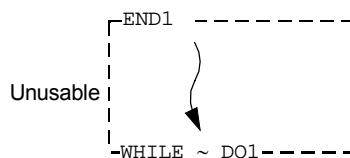


Note: For nesting, m once used cannot be used again.

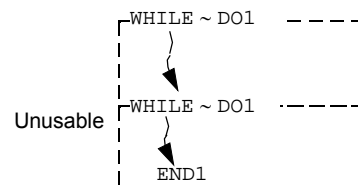
[4] The total number of levels of WHILE ~ DOm must not exceed 27.



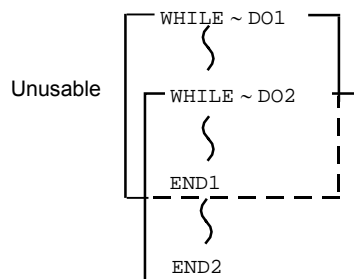
[5] WHILE ~ DOm must precede ENDm.



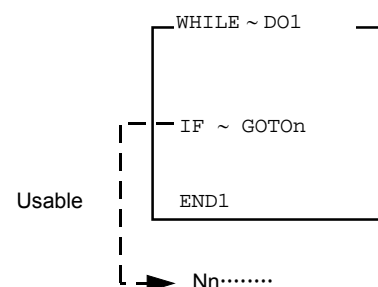
[6] WHILE ~ DOm must correspond to ENDm one-to-one in the same program.



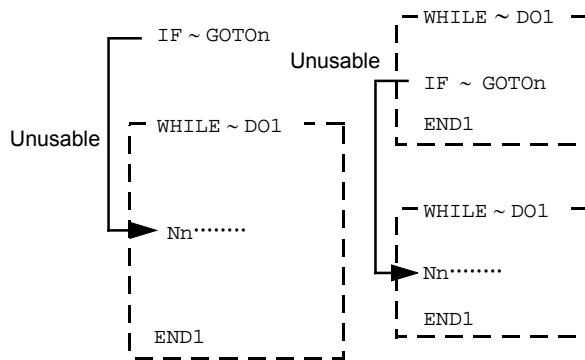
[7] WHILE ~ DOm must not overlap.



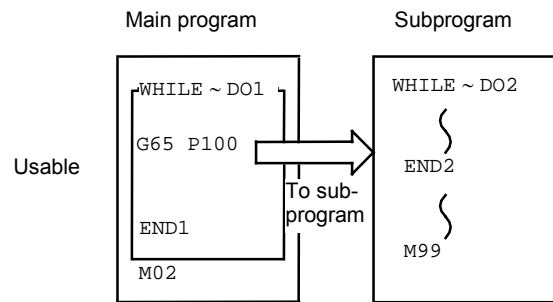
[8] Outward branching from the range of WHILE ~ DOm is possible.



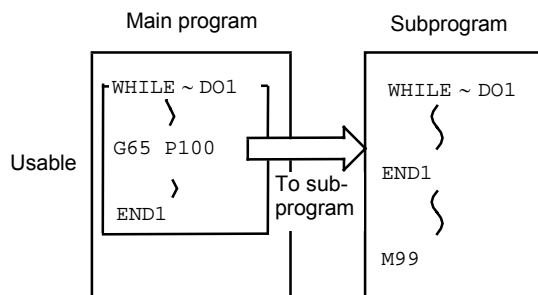
[9] Branching into WHILE ~ DOm is not allowed.



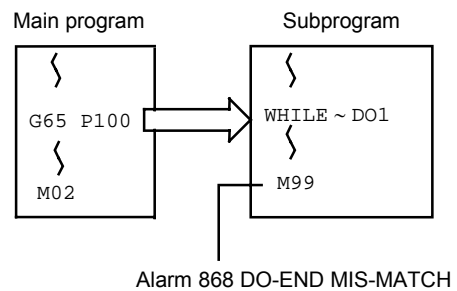
[10] Subprogram can be called using M98, G65, G66, etc. from the midway of WHILE ~ DOm.



[11] The looping can be independently programmed in a subprogram which is called using G65/G66 from the midway of WHILE ~ DOm. Up to 27 levels of nesting for both programs can be done.



[12] If WHILE and END are not included in pairs in subprogram (including macro subprogram), an alarm will result at M99.



13-10-7 External output commands (Output via RS-232C)

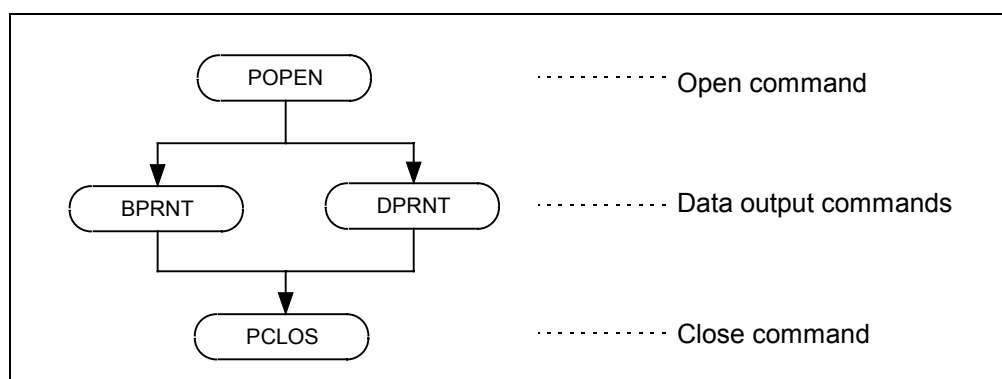
1. Overview

In addition to standard user macros, the types of macros listed below are provided as external output commands. These external output macros can be used to output character data or the numerical data in variables to an external unit via an RS-232C interface. The data are outputted in a data length of 7 bits with an even-parity bit added.

A. Types and functions of external output macros

POPEN	Setup processing for data output
PCLOS	Termination processing of data output
BPRNT	Printout of character data or binary printout of variable data
DPRNT	Printout of character data or numerical printout of variable data on a digit-by-digit basis

B. Programming order



2. Open command POPEN

Programming format: POPEN

Detailed description

- The command code POPEN must be included before a series of data output command codes.
- The control code for DC2 and the percentage code % are output from the NC unit to an external output unit.
- Once POPEN has been set, it will remain valid until PCLOS is set.

3. Close command PCLOS

Programming format: PCLOS

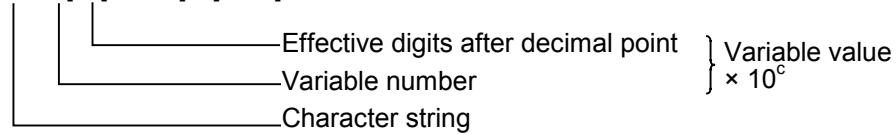
Detailed description

- The command code PCLOS must be included after all data output command codes.
- The control code for DC4 and the percentage code % are outputted from the NC unit to an external output unit.
- This command must be used together with POPEN. This command code must be included only after POPEN.
- This command must be set at the end of the program even after data output has been aborted using, for example, the NC reset switch.

4. Data output command BPRNT

Programming format:

BPRNT[*ℓ*1#*v*1[*c*1]#2#*v*2[*c*2].....]



Detailed description

- The command BPRNT can be used to output characters or to output variable data in binary form.
- The designated character string is outputted directly in the ISO coded format. Alphanumerics (A to Z, and 0 to 9) and/or special characters (+, -, *, /) can be used. Of these characters, only the asterisk (*) is outputted as a space code.
- Since all variables are saved as those having a decimal point, the necessary number of decimal digits must be enclosed in brackets ([]).

All variables are handled as data of four bytes (32 bits), and each byte is outputted as binary data in the order of the most significant byte first. Minus data is processed as the complement for that data.

Example 1: If three digits are specified for 12.3456, then
 $[12.346 \times 10^3] = 12346$ (0000303A)
 will be outputted as binary data.

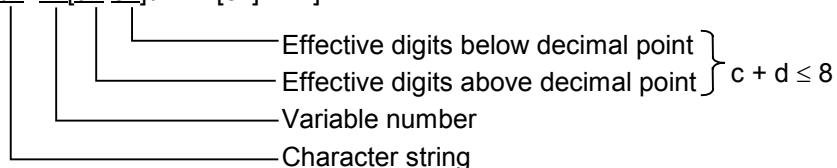
Example 2: If no digits are specified for -100.0, then
 -100 (FFFFFF9C)
 will be outputted as binary data.

- After the specified data has been outputted, the EOB (End Of Block) code is outputted in the format of the appropriate ISO code.
- Variables containing <empty> are interpreted as 0s.

5. Data output command DPRNT

Programming format:

DPRNT[*ℓ*1#*v*1[*d*1 *c*1]#2#*v*2[*c*2].....]



Detailed description

- Output of character data or decimal output of variable data is performed in the format of ISO codes.
- The designated character string is outputted directly in the ISO coded format. Alphanumerics (A to Z, and 0 to 9) and/or special characters (+, -, *, /) can be used. Of these characters, only the asterisk (*) is outputted as a space code.
- Of the data contained in a variable, the necessary number of digits above the decimal point and that of digits below the decimal point must each be enclosed in brackets ([]). The variable data will then have its total specified number of digits, including the decimal point, outputted in the ISO coded format in the order of the most significant digit first. No trailing zeros will be left out in that case.

13-10-8 External output command (Output onto the hard disk)

1. Overview

External output macros can also be used to output data in text file format into the predetermined directory on the hard disk.

2. Related parameters

- **DPR14**: Selection of an output destination port
Set **DPR14** to "4" (Output onto the hard disk) under OTHER on the **DATA I/O PARAMETER** display.
- **DPR15**: Number of lines in feed section
Set the required number of lines to be fed.
- **DPR8**: Output file size
Use this parameter to specify the maximum permissible output file size.
Maximum permissible file size: $\text{DPR8} \times 100$ (KB)
A command for outputting a greater file will cause a corresponding alarm.
However, the file size limit is 100 KB if the value in **DPR8** is 0.

Note: Output of a file of smaller size than the limit, however, may not be possible due to a shortage of available area on the hard disk.

The **DATA I/O PARAMETER** display can be selected by pressing the **[DATA I/O PARAM.]** menu key on the **DATA I/O** display.

See the Parameter List for details of the parameters.

3. Output file

The text file will be automatically outputted with a particular file name into the predetermined directory.

Output directory: c:\MC_sdg\print\

Output file name: print.txt

(A file of this name will be automatically created, if required, or the text data will be added to the current contents of the file.)

File contents:

Given below on the right is an example of text file contents created by the execution of the program shown on the left under the particular parameter settings.

[Program]	[Output example]
G28XYZ	print.txt
POPEN	%
DPRNT[OOOOOOOOOOOO]	OOOOOOOOOOOO
DPRNT[XXXXXXXXXXXXXX]	XXXXXXXXXXXXXX
DPRNT[IIIIIIIIIIIII]	IIIIIIIIIIII
PCLOS	%
G0X100.Y100.Z100.	
M30	

[Parameter]
DPR14: 4
DPR15: No setting

4. Related alarms

The alarm given for text file output is described below.

No.	Message	Argument 1		Argument 2	Argument 3
887	TAPE I/O ERROR	-100	File open error	0	0
		-111	File write error	0	0
		-112	File size too great	0	0

13-10-9 Precautions

Use of user macro commands allows a machining program to be created by combining arithmetic operation, judgment, branching, or other macro commands with conventional NC commands such as move commands, M-, S-, T-commands, etc. The statement defined by these macro commands and that of conventional NC commands are taken as a macro statement and an NC execute statement, respectively. The treatment of a macro statement has no direct relations with machine control. Its treatment as short as possible is effective for shortening machining time.

Parallel processing of the NC execute statement and the macro statement becomes possible according to the setting of bit 6 of parameter **F93**.

(It becomes possible to process all macro statements in batch form by setting the parameter bit to OFF when machining the workpiece, or to execute the macro statements block-by-block by setting the parameter bit to ON when checking the program. Therefore, set the parameter bit according to your requirements.)

Sample program

```
N1 G91G28X0Y0Z0
```

```
N2 G92X0Y0Z0
```

```
N3 G00X-100.Y-100.
```

```
N4 #101=100.*COS[210.]
```

```
N5 #102=100.*SIN[210.]
```

```
N6 G01X#101Y#102F800
```

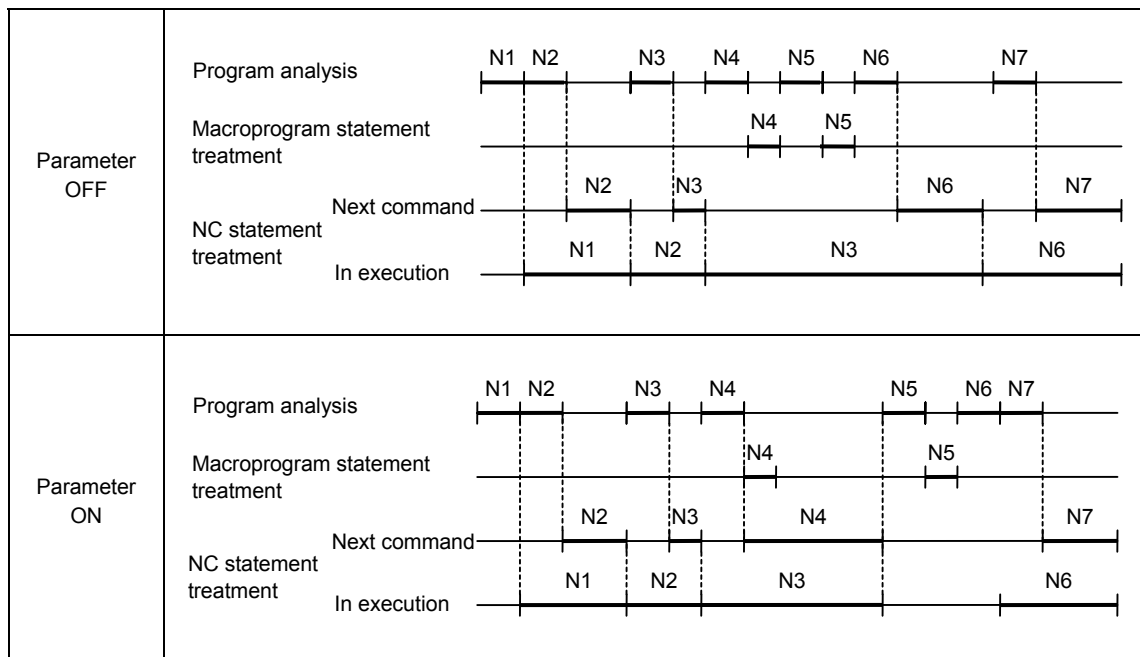
← Macro statements

A macro statement refers to a statement that consists of the following blocks:

- Arithmetic operation command block (compassing the equal sign =)
- Control command block (compassing GOTO, DO ~ END, etc.)
- Macro call command block (compassing macro call or cancellation G-code commands G65, G66, G66.1, or G67)

An NC execute statement refers to a non-macro statement.

The flow of processing of these two types of statements is shown below.



Machining program data is displayed as follows:

Parameter OFF	(In execution) N3 G00X-100.Y-100. (Next command) N6 G01X#101Y#102F800	N4, N5 and N6 are treated in parallel with NC execution sentence of N3, and N6 is displayed as next command because it is NC execution sentence. When N4, N5, and N6 are analyzed during NC execution sentence of N3, machine control continues.
Parameter ON	(In execution) N3 G00X-100.Y-100. (Next command) N4 #101=100.*COS[210.]	N4 is treated in parallel with the control of NC execution sentence of N3, and is displayed as next command. After N3 is completed, N5 and N6 are analyzed so the machine control is forced to wait by the analyzing time of N5 and N6 before N6 can be executed.

13-10-10 Specific examples of programming using user macros

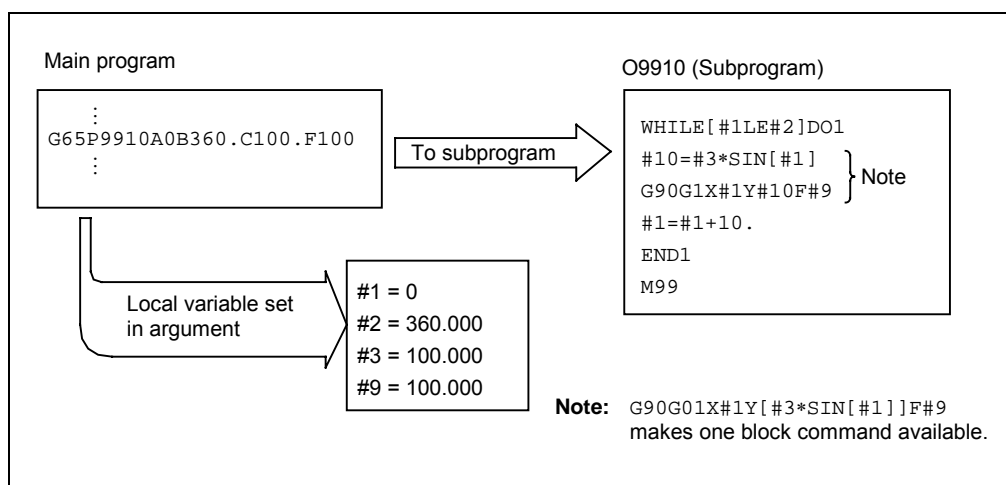
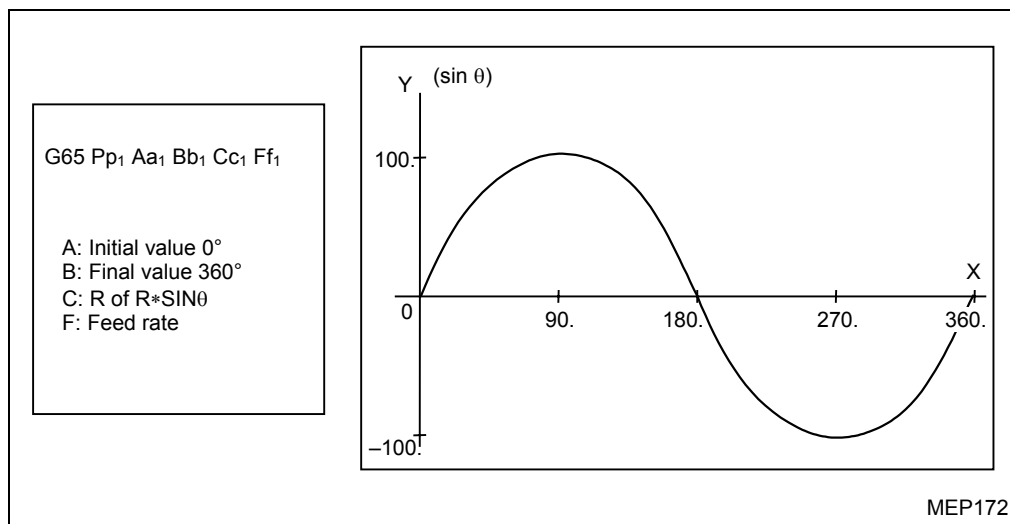
The following three examples of programming are shown here:

Example 1: SIN curve

Example 2: Bolt-hole circle

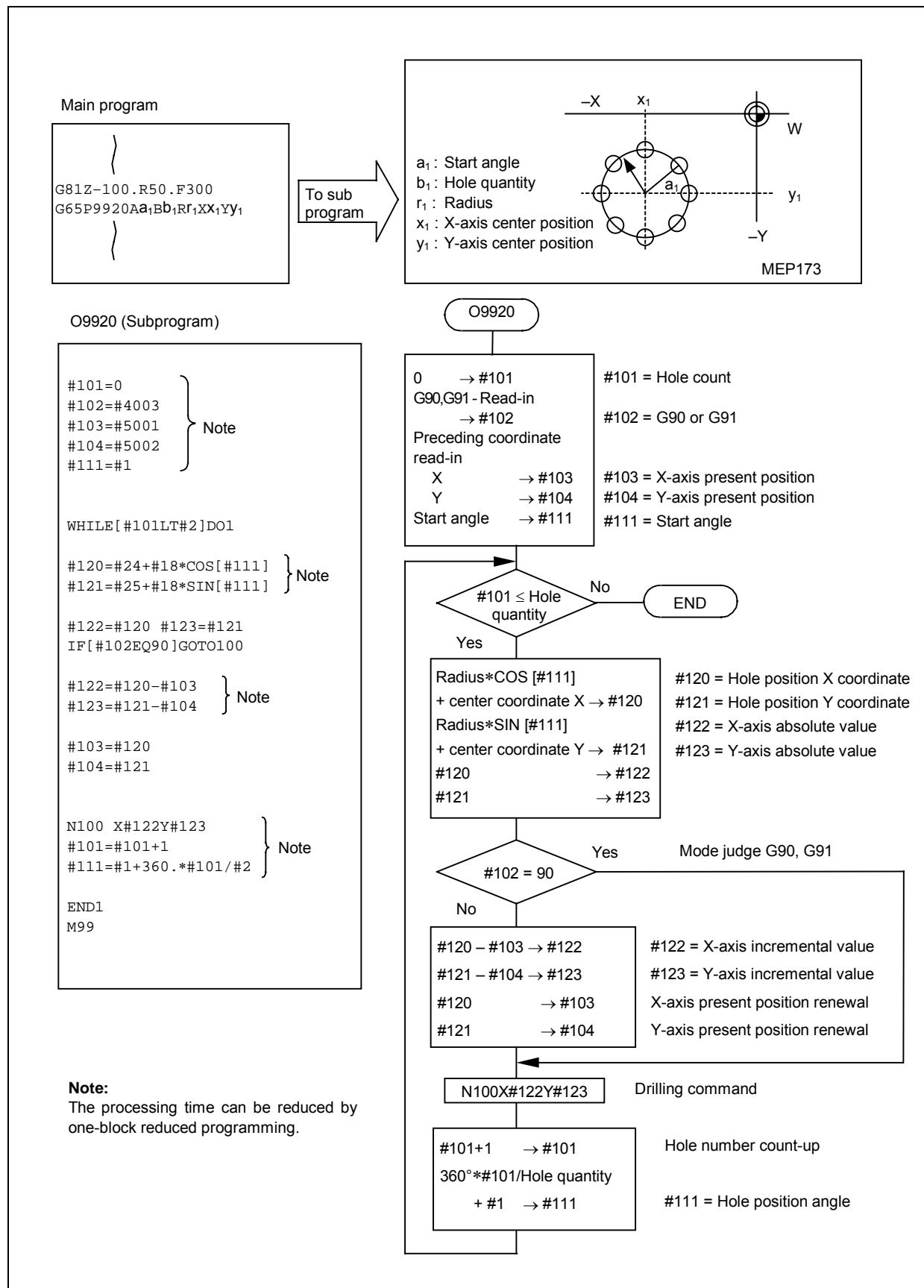
Example 3: Grid

Example 1: SIN curve



Example 2: Bolt-hole circle

After hole data has been defined using fixed-cycle machining commands G72 to G89, hole positions are to be designated using macro commands.

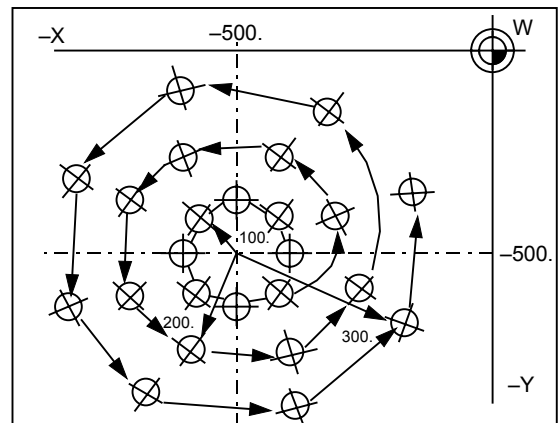


```

G28X Y Z
T1 M06
G90 G43 Z100. H01
G54 G00 X0 Y0
G81 Z-100. R3. F100 M03
G65 P9920X-500. Y-500. A0 B8R100.
G65 P9920X-500. Y-500. A30. B8R200.
G65 P9920X-500. Y-500. A60. B8R300.

```

To subprogram



MEP174

Example 3: Grid

After hole data has been defined using fixed-cycle machining commands G72 to G89, hole positions are to be designated using macro call commands.

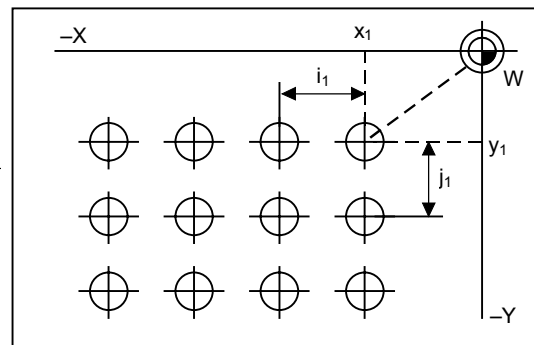
```

G81 Zz1 Rr1 Ff1
G65Pp1 Xx1 Yy1 Ii1 Jj1 Aa1 Bb1

```

X : X-axis hole position
Y : Y-axis hole position
I : X-axis distance
J : Y-axis distance
A : X direction hole quantity
B : Y direction hole quantity

Subprogram is shown on the next page.



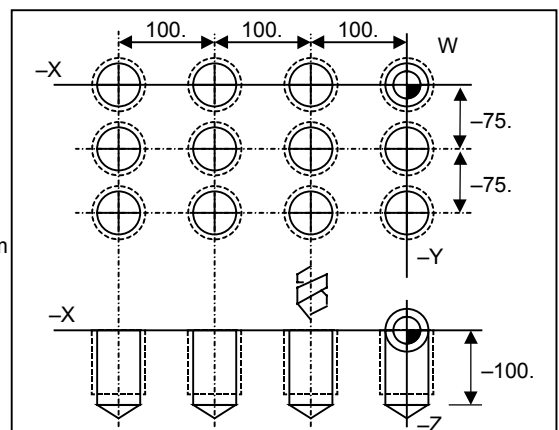
MEP175

```

G28X Y Z
T1 M06
G90 G43 Z100. H01
G54 G00 X0 Y0
G81 Z-100. R3. F100M03
G65P9930 X0 Y0 I100. J-75. A5B3

```

To subprogram



MEP176

O9930 (Subprogram)

```
#101=#24
#102=#25
#104=#10
#105=#1
#106=#2-1
#110=0
#111=0
#112=0
```

Note

```
N2 #113=0
#103=#9
```

Note

```
WHILE[#105GT0]DO1
#101=#101+#113
#105=#105-1
X#101Y#102
```

Note

```
IF[#112EQ1]GOTO10
IF[#111NE1]GOTO10
```

Note

```
#103=0-#103
#112=1
```

```
N10 #113=#103
END1
```

```
N100 #106=#106-1
#112=0
#110=#110+1
```

Note

```
IF[#106LT0]GOTO200
```

```
#105=#1
#102=#102+#104
#111=#110
```

Note

```
#111=#111AND1
```

```
GOTO2
```

```
N200 M99
```

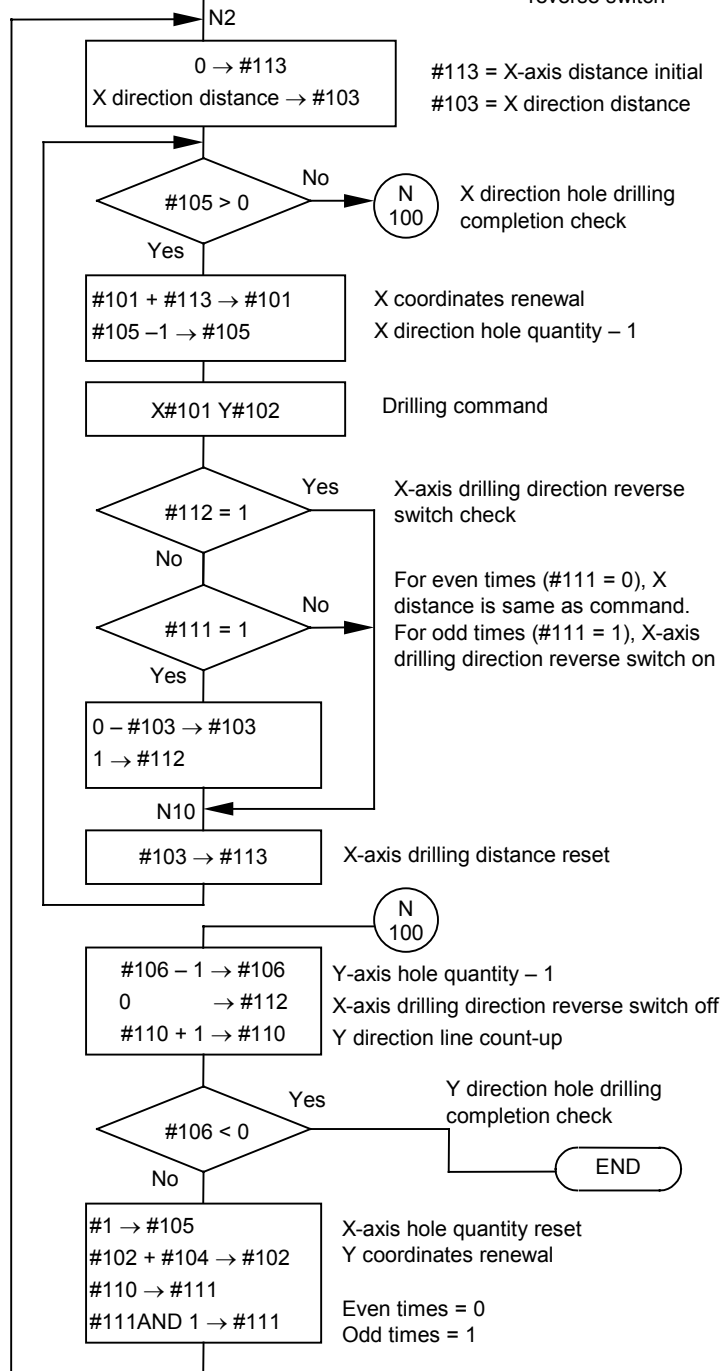
Note:

The processing time can be reduced by one-block reduced programming.

O9930

```
Start point X coordinate → #101
Start point Y coordinate → #102
Y-axis direction distance → #104
X-axis hole quantity → #105
Y-axis hole quantity -1 → #106
0 → #110
0 → #111
0 → #112
```

```
#101 = X-axis start point
#102 = Y-axis start point
#104 = Y direction distance
#105 = X-axis hole quantity
#106 = Y-axis hole quantity - 1
#110 = Y direction line count
#111 = Even/odd times judge
#112 = X-axis drilling direction
reverse switch
```



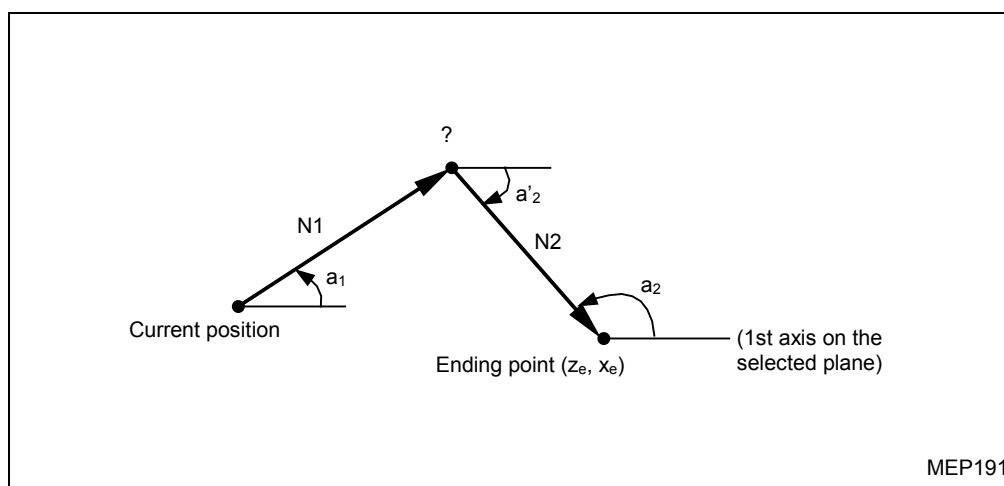
13-11 Geometric Commds (Option)

1. Function and purpose

Even if it is difficult to find the crossing point of two lines using linear interpolation commands, setting the slope of the first line and the absolute coordinates of the ending point of the second line and its slope will allow the NC unit to calculate the coordinates of the ending point of the first line and thus to control move commands.

2. Programming format

G18	Specify the intended plane using G17, G18, or G19.
N1 G01 Aa ₁ Ff ₁	Specify the angle and speed for the first block.
N2 Xx _e Zz _e Aa ₂ (a' ₂) Ff ₂	Specify the absolute coordinates of the ending point of the next block, angles, and a speed.

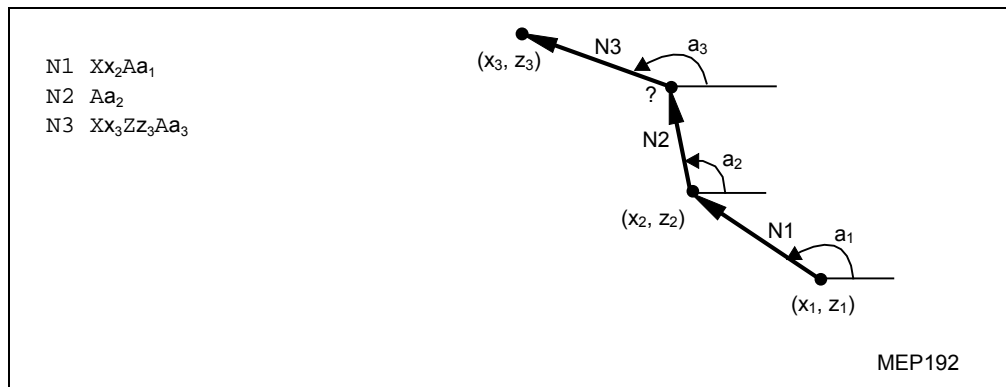


3. Detailed description

- The slope of a line denotes an angle relative to the plus (+) direction of the first axis (horizontal axis) on the selected plane. Assign the sign + for a counterclockwise direction (CCW), or the sign – for a clockwise direction (CW).
- The range of the slope a must be $-360.000^\circ \leq a \leq +360.000^\circ$.
- For the second block, the slope at either the starting point or the ending point can be set. The NC unit will identify whether the specified slope is for the starting point or for the ending point.
- The coordinates of the ending point of the second block must be specified using absolute data. Otherwise an alarm will result.
- Any speed can be specified for each block.
- An alarm will result if the angle of the crossing point of the two lines is 1 degree or less.
- An alarm will result if the preselected plane for the first block is changed over at the second block.
- The geometric command function does not work if address A is to be used for an axis name or for the No. 2 auxiliary function.
- Single-block stop can be used at the ending point of the first block.
- An alarm will result if the first block or the second block is not linear.

4. Correlationships to other functions

Geometric command can be set following a linear angle command.



13-12 Corner Chamfering and Corner Rounding Commands

Automatic chamfering (or rounding) at any angle becomes possible by adding characters “,C_” or “,R_” at the end of the first block of the two blocks that use only lines to generate corners.

13-12-1 Corner chamfering (,C_)

1. Function and purpose

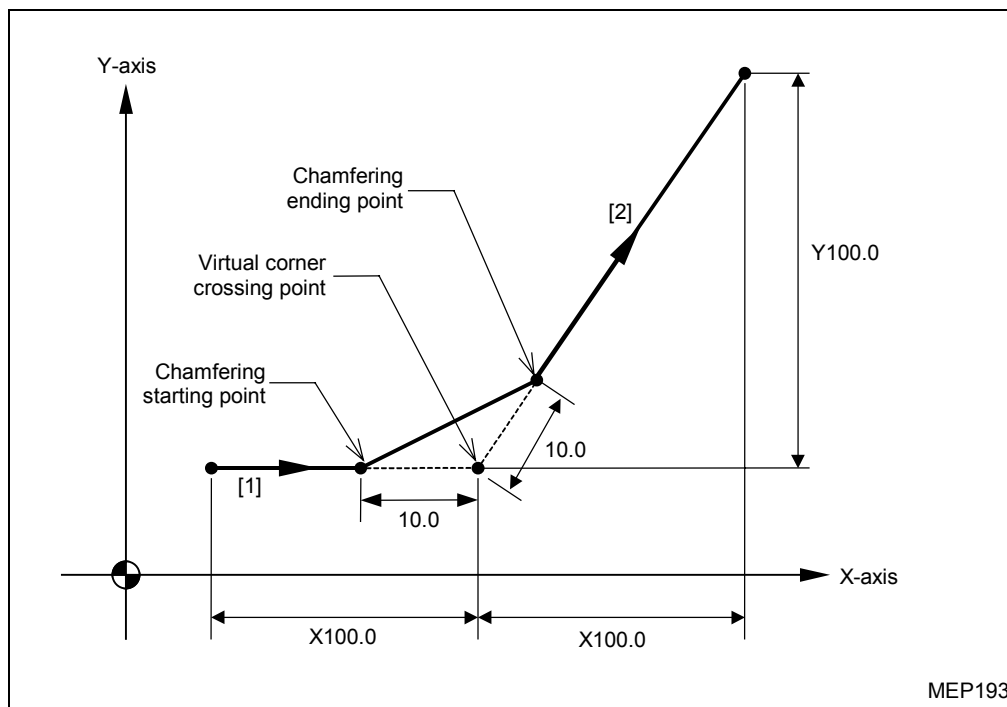
The sections before and after virtual corners, for which it is assumed that no chamfering is to take place, are chamfered over a distance which is specified in C_.

2. Programming format

N100 G01 X_Y_,C	}	Chamfering will be executed at the crossing point of N100 and N200.
N200 G01 X_Y_		
	└─┬─┘	Distance from the virtual corner to the starting or ending point of chamfering

3. Sample program

[1] G91G01X100.,C10.
[2] X100.Y100.



4. Detailed description

- The starting point of the next block of corner chamfering is a virtual corner crossing point.
- If character C is not preceded by a comma (,), that command will be regarded as a C-code command.
- If one block contains both “,C_” and “,R_”, only the last issued command will become valid.
- Tool offset data is calculated for the shape existing after completion of corner chamfering.

- Scaling, if already set, also works for the amount of corner chamfering.
- Alarm **912 NO MOTION COMMAND AFTER R/C** results if the command in the block immediately succeeding that of the corner-chamfering command is not a linear-machining command.
- Alarm **913 INCORRECT R/C COMMAND** results if the distance of movement, specified in the corner-chamfering command block, is less than the amount of chamfering.
- Alarm **914 INCORRECT COMMAND AFTER R/C** results if the distance of movement, specified in the block immediately succeeding the corner chamfering command block, is less than the amount of chamfering.
- Alarm **911 CORNER R/C OPTION NOT FOUND** results if the command in the block immediately succeeding that of the corner chamfering command is an arc-machining command.

13-12-2 Corner rounding (,R_)

1. Function and purpose

The virtual corners to be used when it is assumed that no corner rounding is to take place for the corners that each consist of only lines, are rounded using the arcs whose radii are specified using R_.

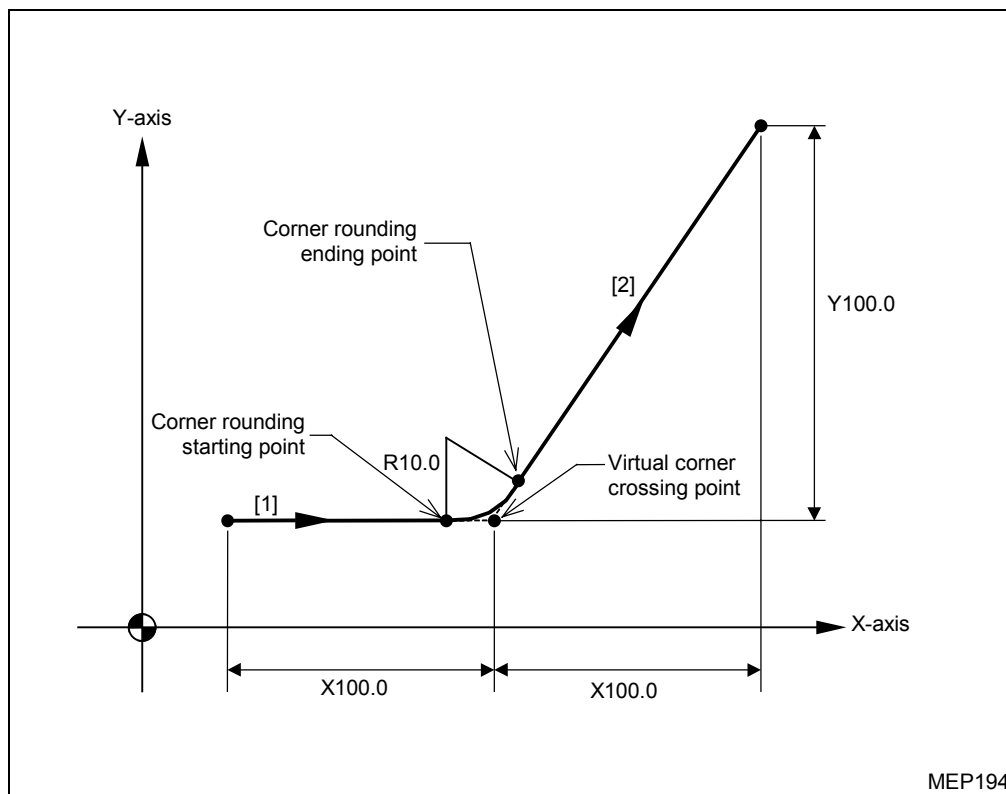
2. Programming format

```
N100 G01 X_Y_,R_
N200 G01 X_Y_
```

} Rounding will be executed at the crossing point of N100 and N200.
 — Radius of the arc for corner rounding

3. Sample program

```
[1] G91G01X100.,R10.
[2] X100.Y100.
```



4. Detailed description

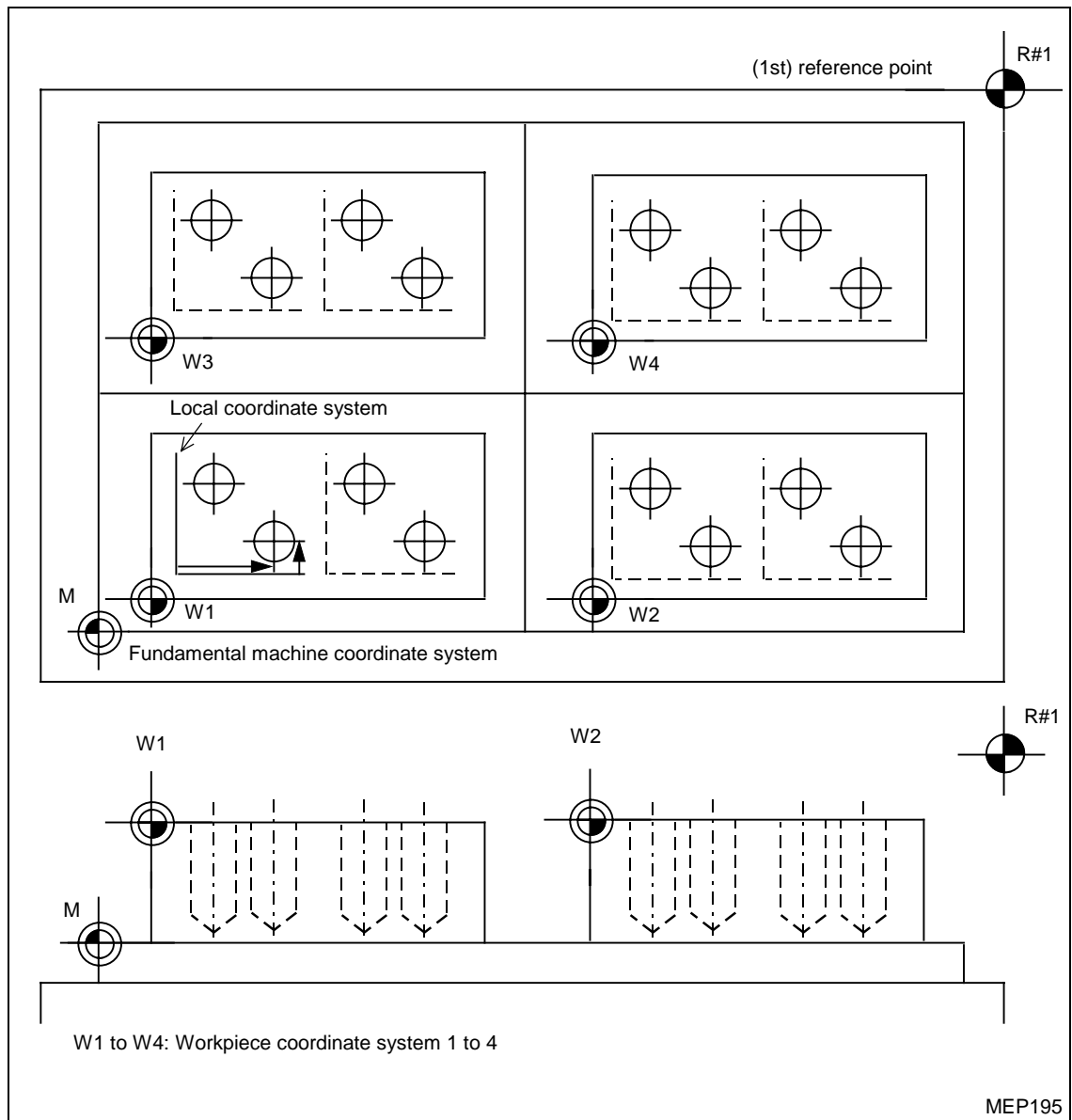
- The starting point of the next block of corner rounding is a virtual corner crossing point.
- If character R is not preceded by a comma (,), that command will be regarded as an R-code command.
- If one block contains both “,C_” and “,R_”, only the last issued command will become valid.
- Tool offset data is calculated for the shape existing after completion of corner rounding.
- Scaling, if already set, also works for the amount of corner rounding.
- Alarm **912 NO MOTION COMMAND AFTER R/C** results if the command in the block immediately succeeding that of the corner-rounding command is not a linear-machining command.
- Alarm **913 INCORRECT R/C COMMAND** results if the distance of movement, specified in the corner-rounding command block, is less than the amount of rounding.
- Alarm **914 INCORRECT COMMAND AFTER R/C** results if the distance of movement, specified in the block immediately succeeding the corner-rounding command block, is less than the amount of rounding.
- Alarm **911 CORNER R/C OPTION NOT FOUND** results if the command in the block immediately succeeding that of the corner-rounding command is an arc-machining command.

14 COORDINATE SYSTEM SETTING FUNCTIONS

14-1 Fundamental Machine Coordinate System, Workpiece Coordinate Systems, and Local Coordinate Systems

The fundamental machine coordinate system, which is a fixed coordinate system for the machine, is used to designate the tool change position, stroke end position, etc. that are predetermined for the machine.

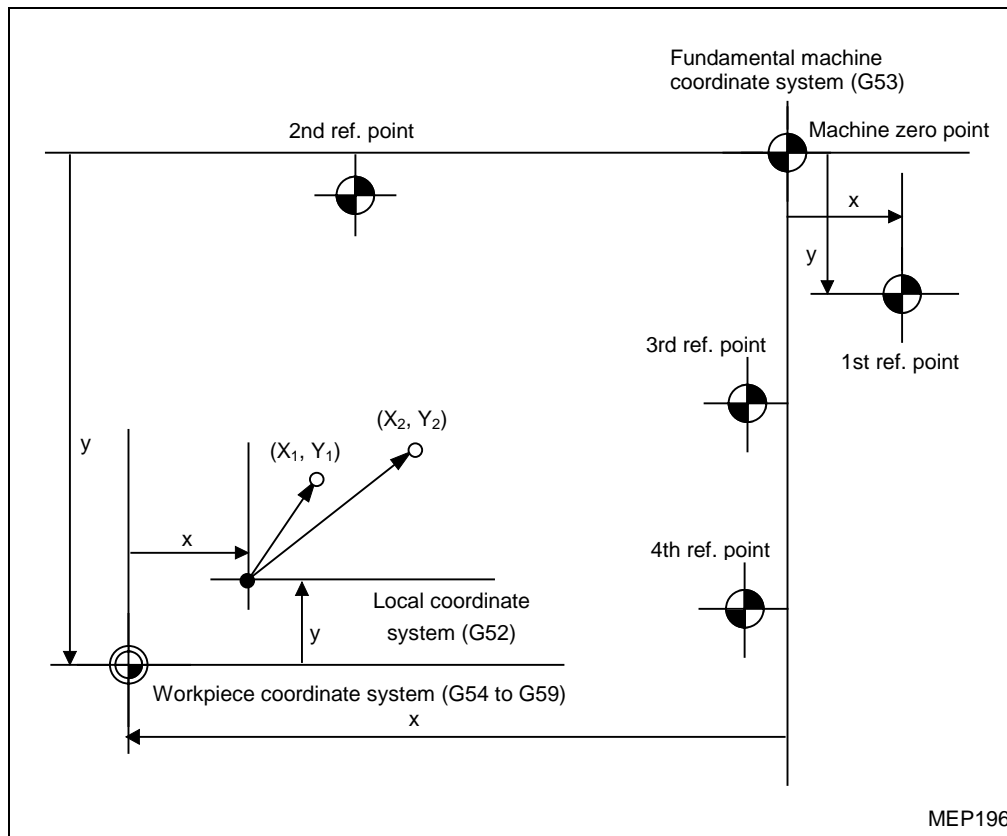
Workpiece coordinate systems are the coordinate systems to be used by a programmer when creating programs. These coordinate systems have their origins set at reference points on workpieces. Local coordinate systems are the coordinate systems to be set on the workpiece coordinate systems to facilitate the creating of partial-machining programs.



14-2 Machine Zero Point and Second, Third, and Fourth Reference Points

The machine zero point refers to a position that acts as a reference point for the fundamental machine coordinate system. That is, the machine zero point is a point specific to the machine, determined by return to the reference point (zero point).

The second, third, and fourth reference points (zero point) are the points whose positions in the fundamental machine coordinate system are parameter-preset.



14-3 Fundamental Machine Coordinate System Selection: G53

1. Function and purpose

The fundamental machine coordinate system is used to designate the tool change position, stroke end position, etc. that are predetermined for the machine.

Command G53 and its succeeding coordinate words move the tool at the rate of rapid traverse to the designated position on the fundamental machine coordinate system.

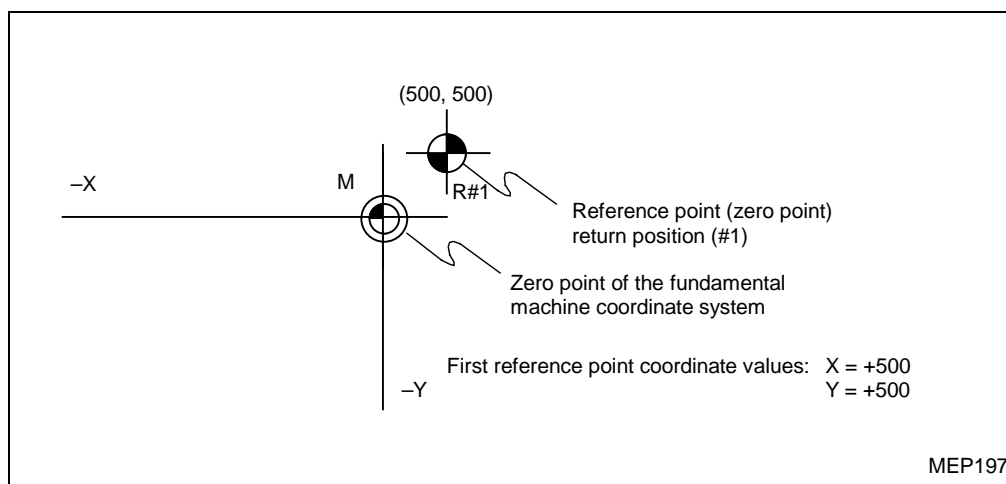
2. Programming format

Selection of the fundamental machine coordinate system:

(G90) G53 Xx Yy Zz $\alpha\alpha$ (α : Additional axis)

3. Detailed description

- When power is turned on, the fundamental machine coordinate system will be set according to the reference point (zero point) return position which is determined by a manual or automatic reference point (zero point) return command.
- The fundamental machine coordinate system is not updated by command G92.
- Command G53 is valid only for designated blocks.
- During the incremental data command mode (G91), command G53 moves the tool on the selected coordinate system according to incremental data.
- Setting of command G53 does not cancel the tool radius compensation amount of the designated axis.
- The coordinate data of the first reference point represents the distance from point 0 on the fundamental machine coordinate system to the reference point (zero point) return position.



14-4 Coordinate System Setting: G92

1. Function and purpose

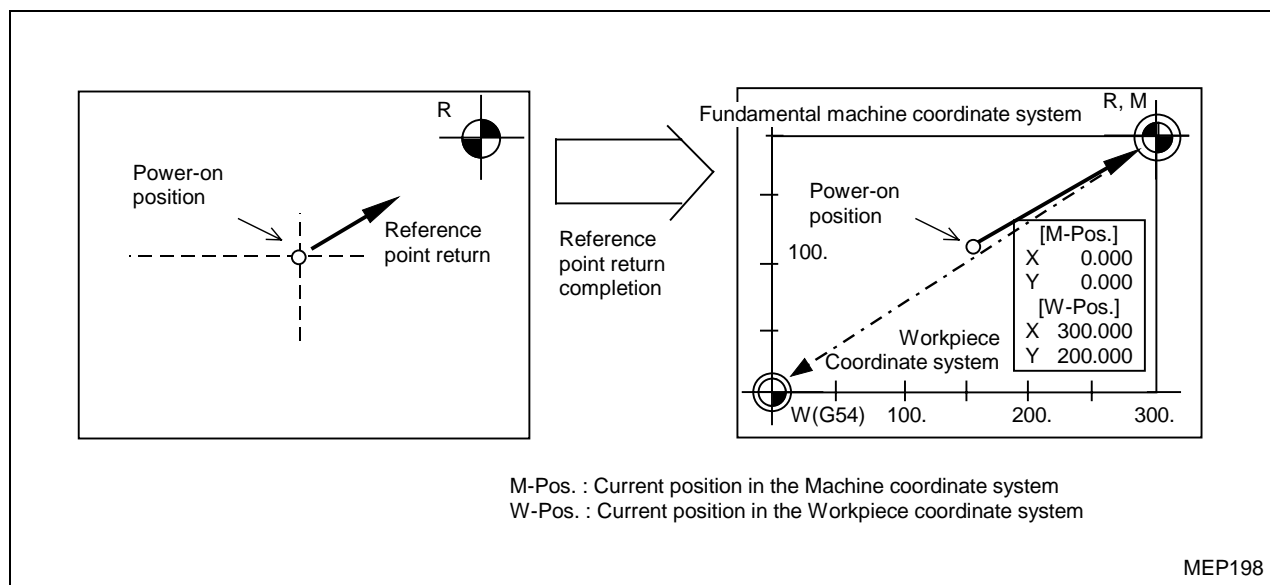
Setting of command G92 allows absolute coordinate system and current position display data to be newly preset exactly as programmed, without ever having to move the machine.

2. Programming format

G92 $Xx_1 Yy_1 Zz_1 \alpha\alpha_1$ (α : Additional axis)

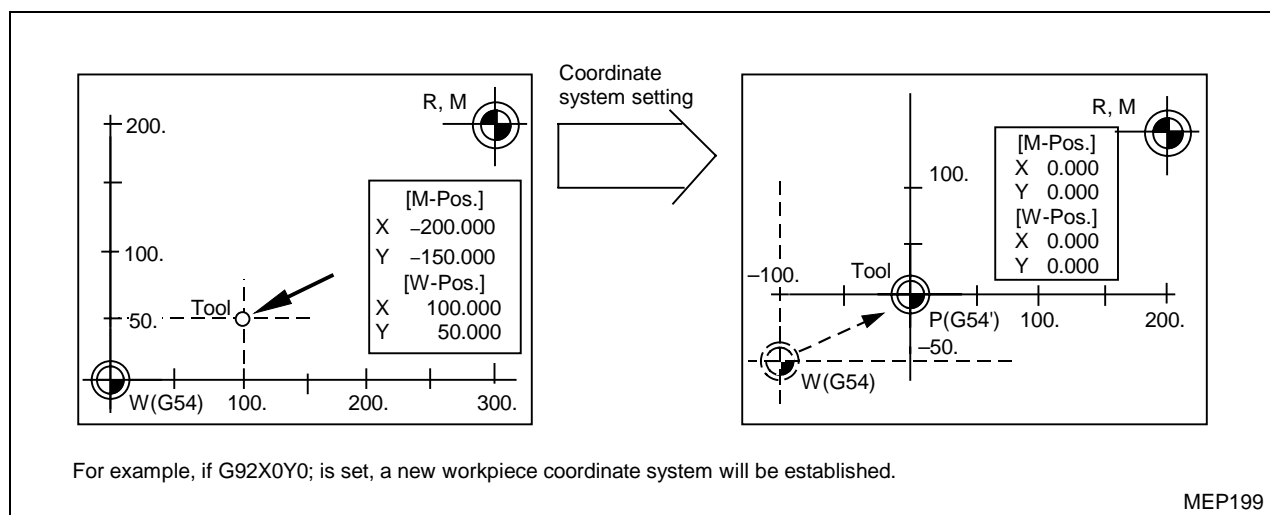
3. Detailed description

- After power-on, return to the reference point is initially performed using watchdogs. Coordinate systems are set automatically on completion of return.



Fundamental machine and workpiece coordinate systems are established at predetermined positions.

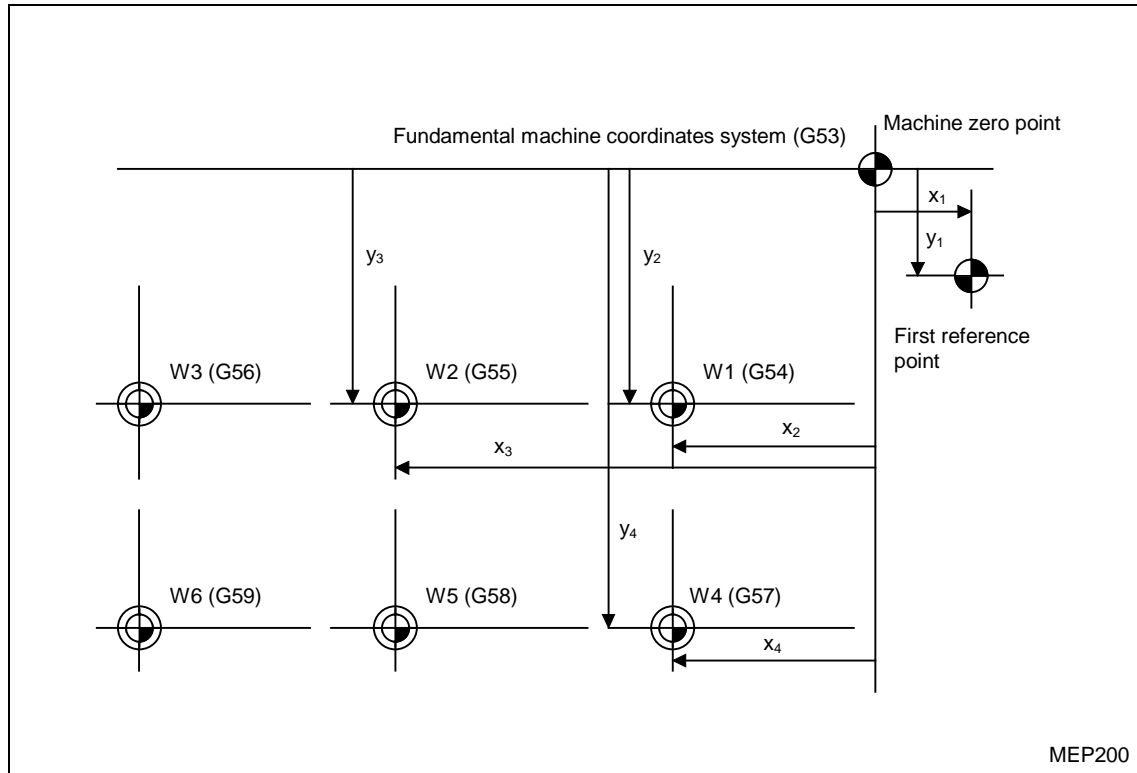
- Setting of command G92 allows absolute (workpiece) coordinate system and current position display data to be newly preset exactly as programmed, without ever having to move the machine.



14-5 Automatic Coordinate System Setting

The automatic coordinate system setting function allows various coordinate systems to be generated according to parameters previously set on the operation panel when the first manual reference point return or watchdog-based reference point return is completed.

Use the automatically set coordinate systems when creating machining programs.



- This function generates the following coordinate systems:

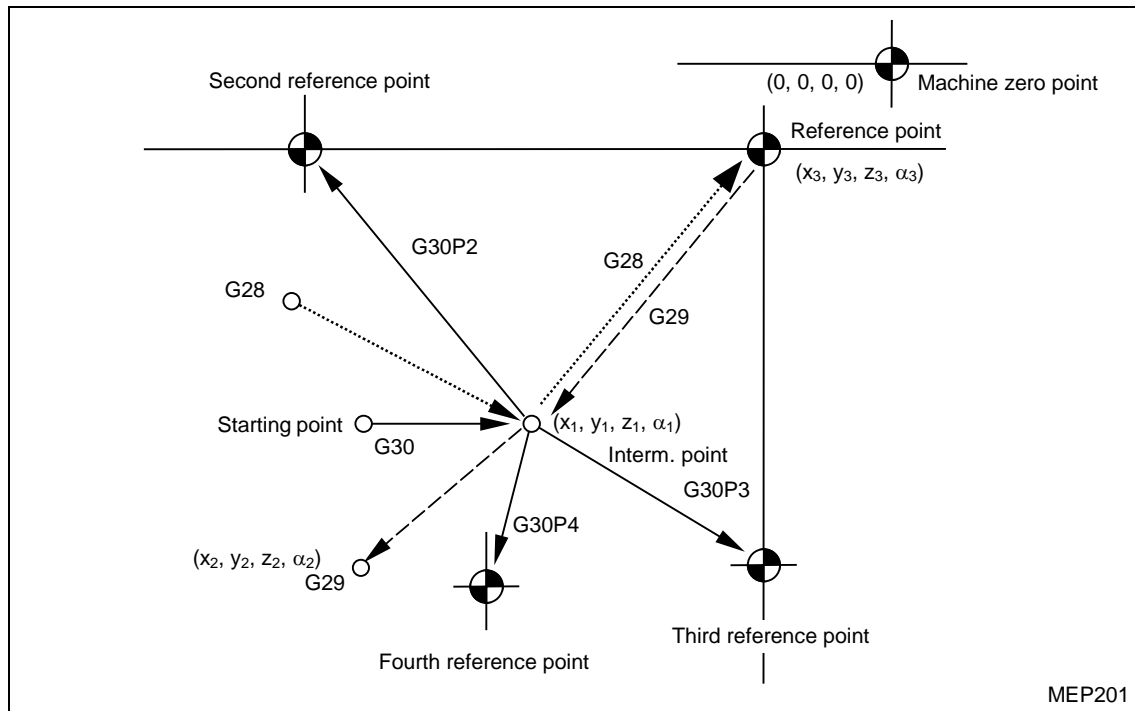
- 1) Fundamental machine coordinate system (G53)
- 2) Workpiece coordinate systems (G54 through G59)

- Parameters for the NC unit must be the distance data from the origin of the fundamental machine coordinate system. You must therefore determine where on the fundamental machine coordinate system the first reference point is to be set, and then set the zero point of the workpiece coordinate system.

14-6 Reference Point Return: G28, G29

1. Function and purpose

- G28 command first performs a G00 (rapid) positioning to the specified intermediate position along the specified axes, and then a returning to the first reference point at the rapid feed rate independently along each specified axis.
- G29 command first performs a returning to the intermediate point of the last G28 or G30 command at the rapid feed rate independently along each specified axis, and then a G00 (rapid) positioning to the specified position.



2. Programming format

- G28 $Xx_1 Yy_1 Zz_1 \alpha\alpha_1$ (α : Additional axis) [Automatic reference point return]
- G29 $Xx_2 Yy_2 Zz_2 \alpha\alpha_2$ (α : Additional axis) [Start point return]

3. Detailed description

1. Command G28 is equivalent to the following commands:

$$\left[\begin{array}{l} G00 Xx_1 Yy_1 Zz_1 \alpha\alpha_1 \\ G00 Xx_3 Yy_3 Zz_3 \alpha\alpha_3 \end{array} \right]$$

where x_3, y_3, z_3 and α_3 denote the coordinates of the reference point, that are parameter-set as the distance from the zero point of the fundamental machine coordinate system.

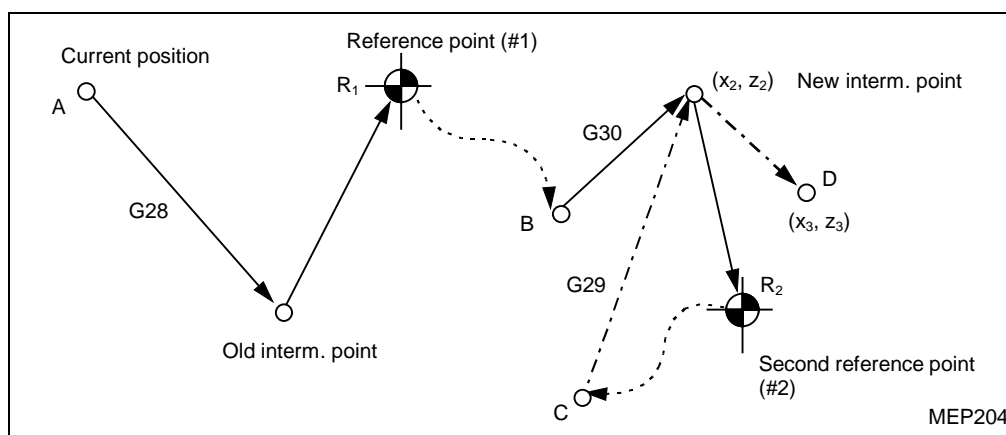
2. Axes that have not been returned to the reference point in manual mode after power-on are returned using the watchdog method. In that case, the direction of return is regarded as the same as the sign-designated direction. The direction of return will not be checked if it is of the straight-return type. For the second time onward, the axes are returned at high speed to the reference point that was stored into the memory by execution of the first return command the direction is not checked at this time, either.

3. When reference point (zero point) return is completed, a return complete output signal will be output and "#1" will be displayed in the field of the axis name.
4. Command G29 is equivalent to the following commands:

$\begin{array}{l} \text{G00 } Xx_1 \ Yy_1 \ Zz_1 \ \alpha\alpha_1 \\ \text{G00 } Xx_2 \ Yy_2 \ Zz_2 \ \alpha\alpha_2 \end{array}$	Independent rapid feed on each axis (not the same as for G0).
---	--
- where x_1 , y_1 , z_1 and α_1 are the coordinates of the middle point specified by the last G28 or G30 command.
5. An alarm will result if G29 is executed without any preceding G28 (automatic reference point return command) after turning-on.
6. Under machine locked status or Z-axis cancelled status, any movements of the Z-axis up to the middle point are ignored and only subsequent positioning is performed.
7. The coordinates of the intermediate point (x_1 , y , z_1 , α_1) must be given according to the type of dimensional data input (G90 or G91).
8. G29 command can refer to both G28 and G30, and the positioning along the specified axes is performed through the intermediate point of the last G28 or G30 command.
9. The tool offsetting, if left valid, is cancelled during execution of the return command and the offset data is also cleared.

4. Sample programs

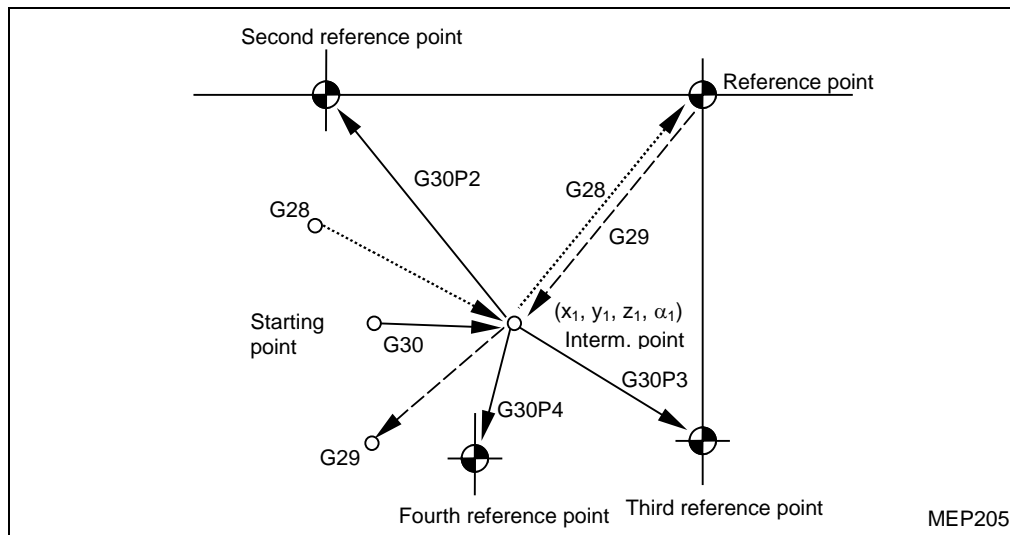
Example: $G28Xx_1Zz_1$ From point A to reference point
 $G30Xx_2Zz_2$ From point B to second reference point
 $G29Xx_3Zz_3$ From point C to point D



14-7 Second, Third, or Fourth Reference Point Return: G30

1. Function and purpose

The returning to the second, third, or fourth reference point can be programmed by setting "G30 P2 (P3, P4)".

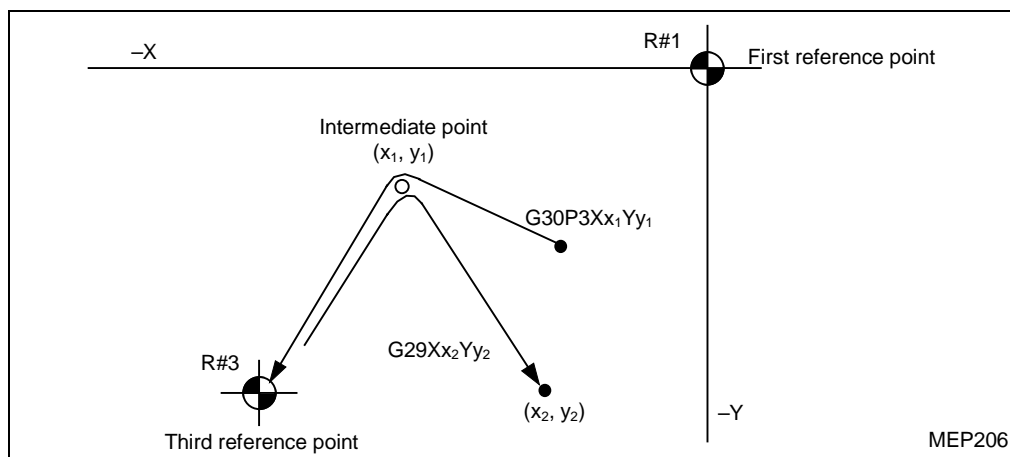


2. Programming format

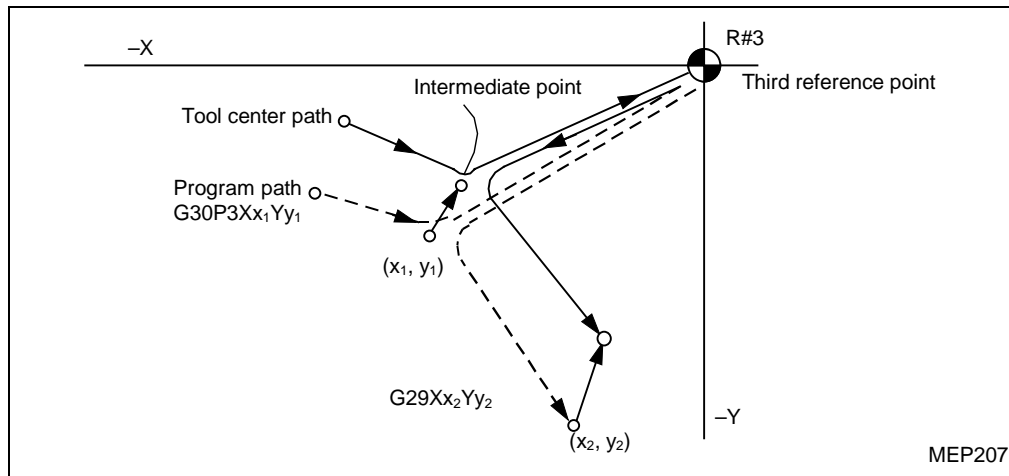
G30 P2 (P3, P4) Xx₁ Yy₁ Zz₁ αα₁ (α: Additional axis)

3. Detailed description

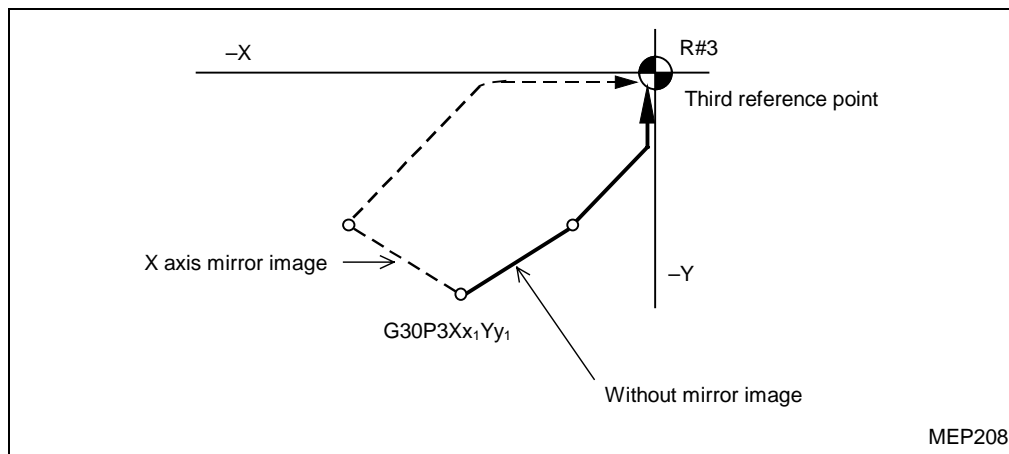
1. Use address P to specify the number of the required reference point (P2, P3 or P4). Return to the second reference point is automatically selected if P-command is omitted or zero, one, five or a greater integer is set at address P.
2. Return to the second, third or fourth reference point is performed through the specified intermediate point like the return to the first reference point.
3. The coordinates of the second, third, or fourth reference point represent the positions specific to the machine. Display machine parameters **M5**, **M6** and **M7** on the screen to check the coordinates concerned.
4. A command of G29 after return to the second, third or fourth reference point is carried out through the intermediate point of the last command for return to the reference point.



5. Tool radius compensation function is temporarily cancelled during return to a reference point in the offsetting plane and made valid again for a return (by G29) from there. The cancellation and resumption of offsetting dimensions occur during movement from the intermediate point to the reference point and vice versa.



6. Upon second, third, or fourth reference point return, the tool length offset data for the return axis is cancelled automatically.
7. For the second, third, or fourth reference point return under machine locked status, movement from the middle point to the reference point is skipped. The next block is executed after the designated axis has arrived at the intermediate point.
8. For the second, third, or fourth reference point return in the mirror image mode, the mirror image is valid for movement from the starting point to the intermediate point and the axis moves in an opposite direction to the designated one. For movement from the intermediate point to the reference point, however, the mirror image becomes invalid and thus the axis moves to the reference point.

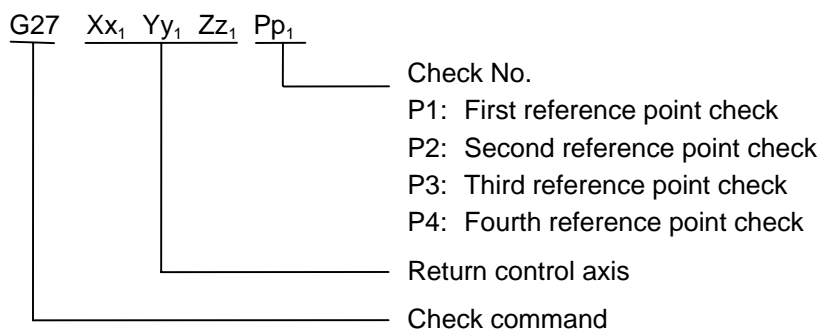


14-8 Reference Point Check Command: G27

1. Function and purpose

G27 command performs a rapid positioning to the specified position, checks the final position for agreement with the specified reference point and, upon confirmation of the agreement, outputs a reference point return complete signal to the machine side as is the case with G28. For a machining program, therefore, of which the starting and the ending positions are the reference point, this function is useful for checking whether the returning, i.e. the program itself, has been correctly executed to the end.

2. Programming format



3. Detailed description

- The first reference point check will occur if the P code is omitted.
- The number of axes for which reference point check can be done at the same time depends on the number of simultaneously controllable axes.
- An alarm will result if the designated reference point is not the final position on completion of this command.

14-9 Workpiece Coordinate System Setting and Selection: (G92) G54 to G59

1. Function and purpose

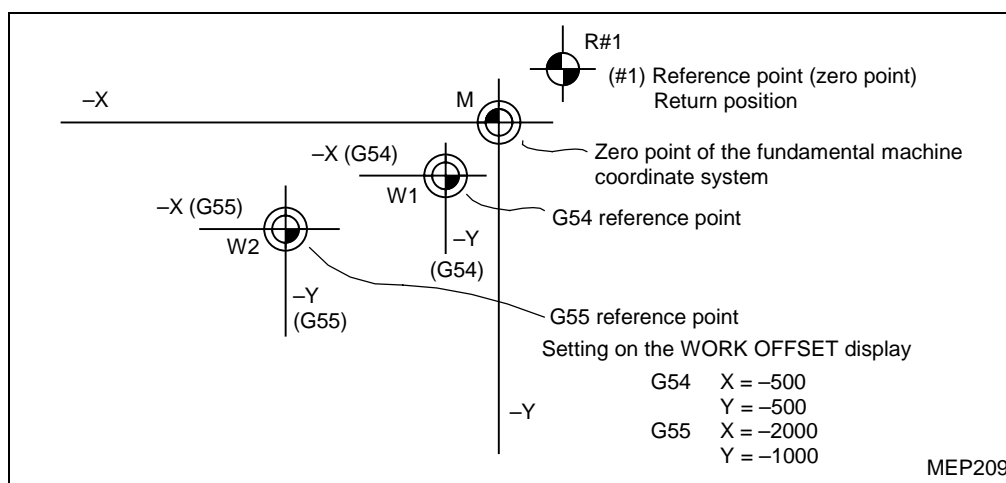
- Workpiece coordinate systems are set to facilitate creation of a workpiece machining program having its origin set at a machining reference point for the intended workpiece. Commands G54 through G59 move the designated axis to the designated position on the workpiece coordinate system corresponding to the command code number. These six types of commands generate respective workpiece coordinate systems.
- Command G92 can be used to modify the current workpiece coordinate system in order that the current tool position be indicated by the specified coordinates. (The current tool position refers to a position that has incorporated tool radius compensation, tool length offset, and tool position offset data.)
Command G92 also generates a virtual machine coordinate system according to which the current tool position is indicated by the specified coordinates. (The current tool position refers to a position that has incorporated tool radius compensation, tool length offset, and tool position offset data.)

2. Programming format

- Selecting a workpiece coordinate system (G54 to G59)
(G90) G54 $X_{x_1} Y_{y_1} Z_{z_1} \alpha \alpha_1$ (α : Additional axis)
- Setting the workpiece coordinate system (G54 to G59)
(G54) G92 $X_{x_1} Y_{y_1} Z_{z_1} \alpha \alpha_1$ (α : Additional axis)

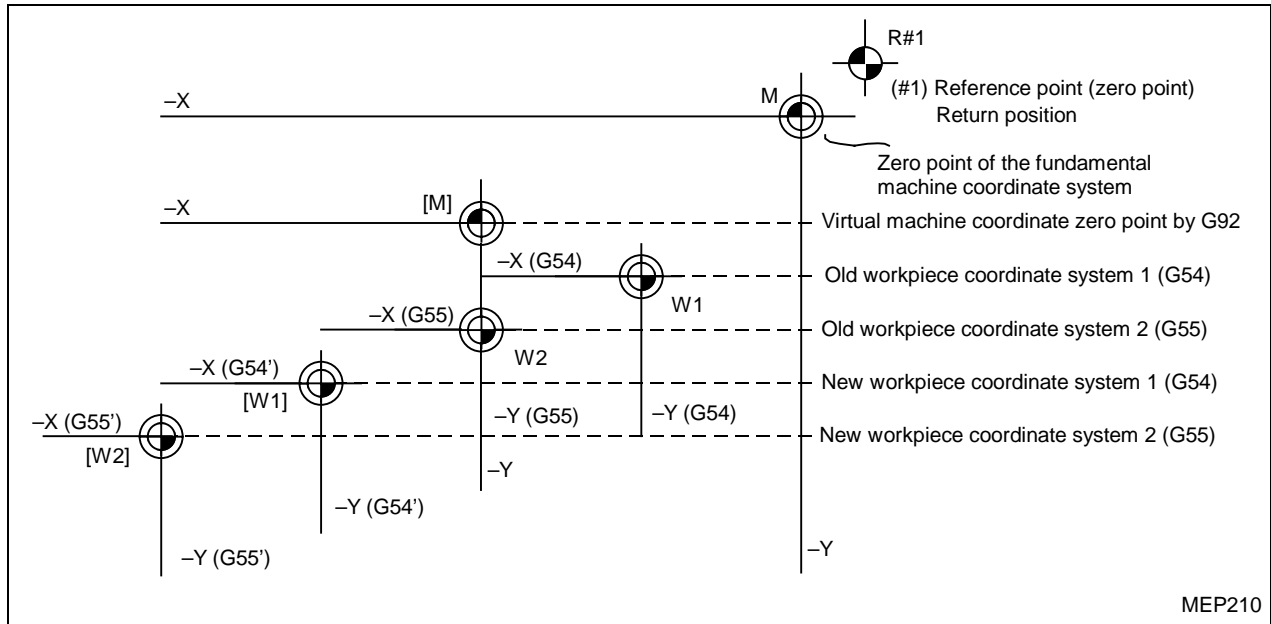
3. Detailed description

1. Tool radius compensation data for the designated axis is not cancelled by the selection of a new workpiece coordinate system using commands G54 to G59.
2. The coordinate system corresponding to G54 will always be selected upon turning-on.
3. Commands G54 through G59 are modal commands (of group 12).
4. Command G92 only moves the workpiece coordinate system.
5. The workpiece origins (reference points) of G54 to G59 must be externally preset with reference to the fundamental machine coordinate system.



6. Settings of workpiece origin can be updated repeatedly (manually or by programming "G10 L2 Pp₁ Xx₁ Yy₁ Zz₁").

7. New workpiece coordinate system 1 (G54) can be generated by setting G92 during the mode of G54. At the same time, other workpiece coordinate systems 2 through 6 (G55 through G59) are shifted parallel to the new workpiece coordinate system 1.



A virtual machine coordinate system is also formed in accordance with shifting of workpiece coordinate system by G92.

After power has been turned on, the virtual machine coordinate system will coincide with the fundamental machine coordinate system by the first automatic (G28) or manual reference point return.

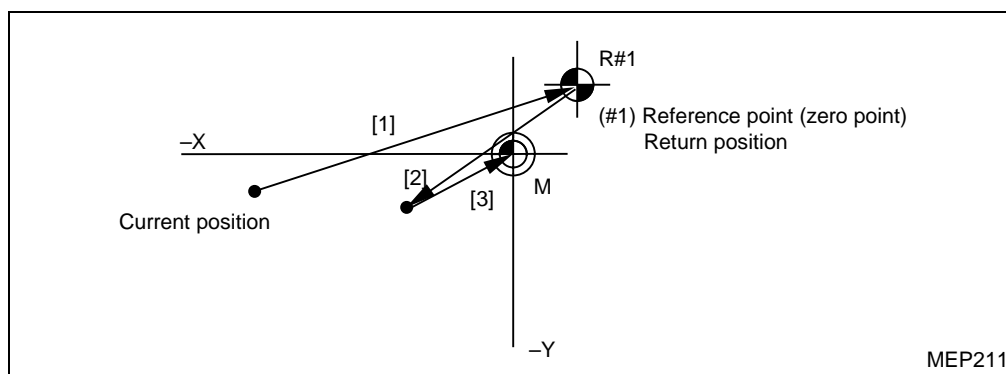
When a virtual machine coordinate system is generated, new workpiece coordinate systems will be set by using the preset origins with reference to the virtual machine coordinate system.

8. After power has been turned on, the fundamental machine coordinate system and workpiece coordinate systems will be set automatically according to the presets when the first automatic (G28) or manual reference point return is completed.

4. Sample programs

Example 1:

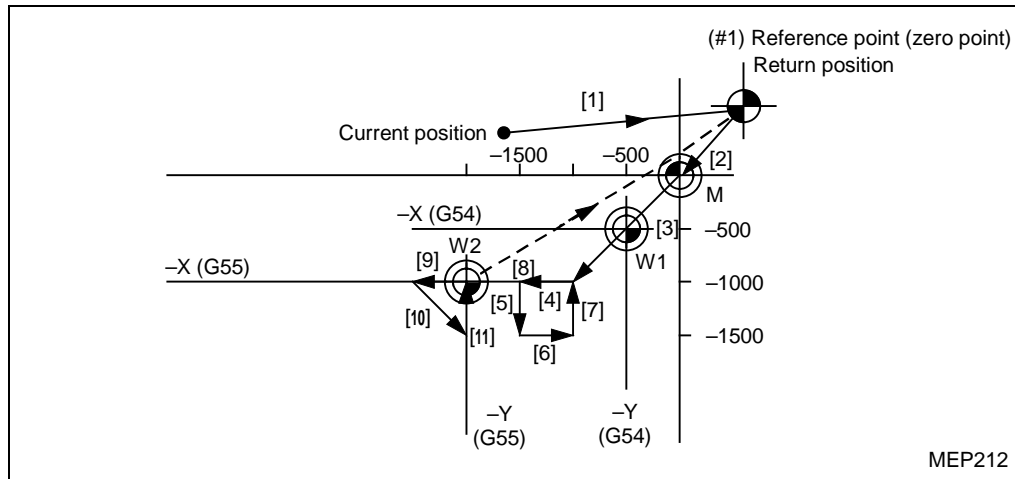
```
[1] G28X0Y0
[2] G53X-100.Y-50.
[3] G53X0Y0
```



If the coordinate data of the first reference point is 0 (zero), the zero point (origin) of the fundamental machine coordinate system and the reference point return position (#1) will agree.

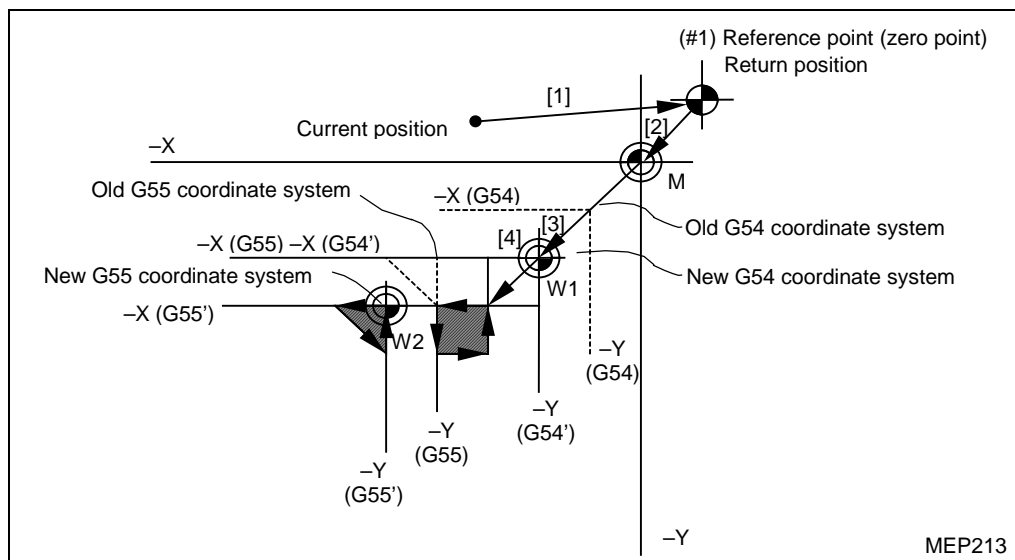
Example 2:

[1] G28X0Y0	[7] Y+500.
[2] G90G00G53X0Y0	[8] G90G00G55X0Y0
[3] G54X-500.Y-500.	[9] G01X-500.F200
[4] G01G91X-500.F100	[10] X0Y-500.
[5] Y-500.	[11] G90G28X0Y0
[6] X+500.	



Example 3: Same machining as is given in **Example 2** after shifting through $(-500, -500)$ the workpiece coordinate system of G54 (provided that [3] through [10] in Example 2 have been registered in subprogram O1111):

[1] G28X0Y0	
[2] G90G00G53X0Y0	(Not required if #1 ref. pt. = machine origin)
[3] G54X-500.Y-500.	Shift amount of workpiece coordinate system
[4] G92X0Y0	New workpiece coordinate system setting
[5] M98P1111	



Note: If blocks [3] through [5] are used repeatedly, reference point return command G28 must be set at the end of the program since the workpiece coordinate systems will shift each time those blocks are executed.

Example 4: If one and the same machining is to be done for each of six identical workpieces placed on the coordinate systems of G54 through G59:

A. Setting of workpiece origins

Workpiece	1	X = -100.000	Y = -100.000	G54
	2	X = -100.000	Y = -500.000	G55
	3	X = -500.000	Y = -100.000	G56
	4	X = -500.000	Y = -500.000	G57
	5	X = -900.000	Y = -100.000	G58
	6	X = -900.000	Y = -500.000	G59

B. Machining program (Subprogram)

```

O100
N1G90G00G43X-50.Y-50.Z-100.H10      } Positioning
N2G01X-200.F50
      Y-200.
      X-50.
      Y-50.
N3G28X0Y0Z0
T□□M06
N4G98G81X-125. Y-75.Z-150.R-95.F40    } Drilling 1
      X-175. Y-125.                    } Drilling 2
      X-125. Y-175.                    } Drilling 3
      X-75.  Y-125.                    } Drilling 4

G80
N5G28X0Y0Z0
      }

N6G98G84X-125. Y-5.Z-150.R-95.F40    } Tapping 1
      X-175. Y-125.                    } Tapping 2
      X-125. Y-175.                    } Tapping 3
      X-75.  Y-125.                    } Tapping 4

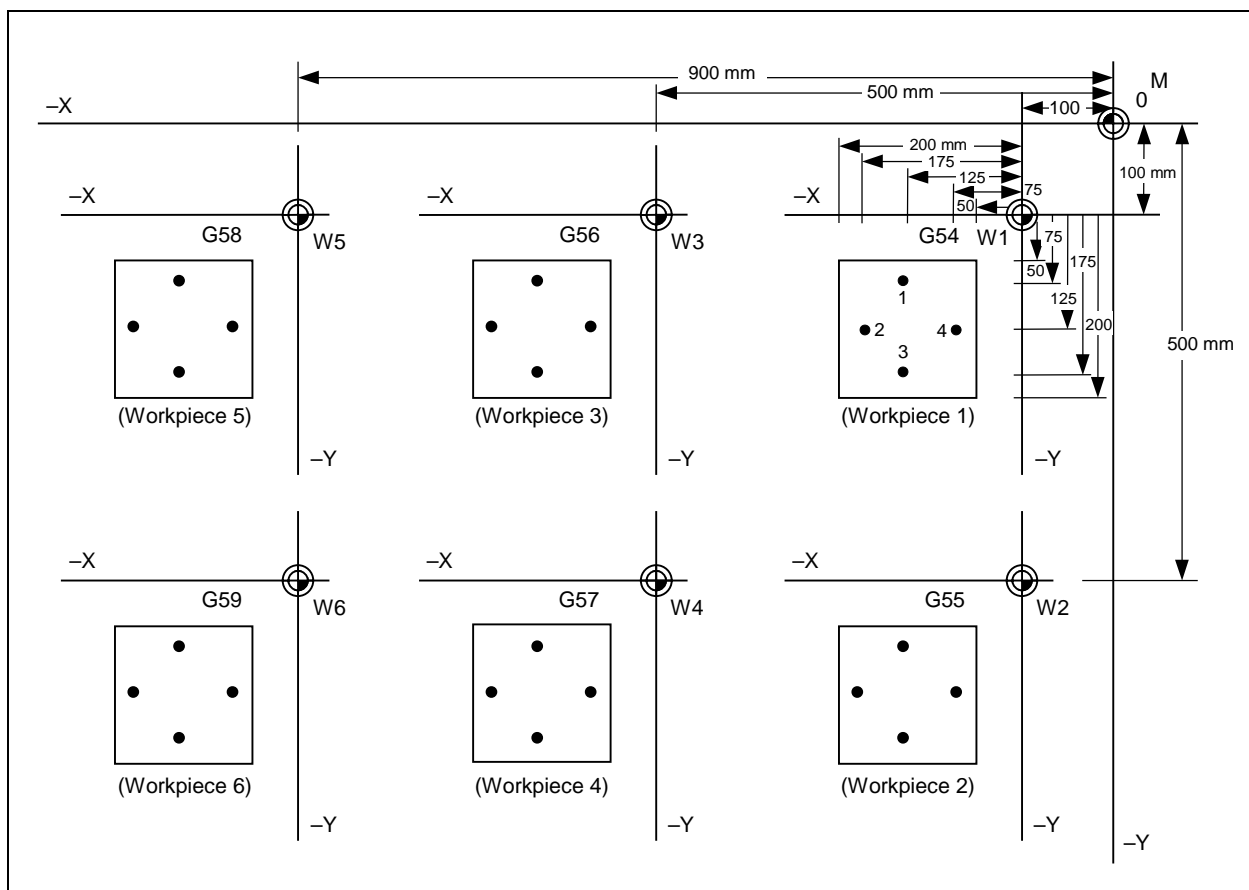
G80
M99

```

C. Positioning program (Main program)

```

G28X0Y0Z0      } At power-on
N1 G90G54M98P100
N2   G55M98P100
N3   G57M98P100
N4   G56M98P100
N5   G58M98P100
N6   G59M98P100
N7 G28X0Y0Z0
N8 M02
%
```



14-10 Additional Workpiece Coordinate System Setting and Selection: G54.1

1. Function and purpose

In addition to the six standard systems G54 to G59, up to 300 sets of workpiece origin data can be used to facilitate program creation.

Note 1: Local coordinate system setting is not available in G54.1 mode.

Note 2: Setting a G52-command during G54.1 mode will cause the alarm **949 NO G52 IN G54.1 MODE**.

2. Programming format

A. Selection of a workpiece coordinate system

G54.1 Pn (n = 1 to 300)

Example: G54.1P48 Selection of P48 system

Note: Omission of P and setting of "P0" function the same as "P1". Setting a value other than integers from 0 to 300 at address P causes the alarm **809 ILLEGAL NUMBER INPUT**.

B. Movement in a workpiece coordinate system

G54.1Pn (n = 1 to 300)

G90 Xx Yy Zz

Example: G54.1P1 Selection of P1 system
G90X0Y0Z0 Movement to P1-system origin (0, 0, 0)

C. Setting of workpiece origin data

G10 L20 Pn Xx Yy Zz (n = 1 to 300)

Example: G90G10L20P30X-255.Y-50. Data at addresses X and Y are set as data of P30-system origin.

G91G10L20P30X-3.Y-5. Data at addresses X and Y are added to data of P30-system origin.

3. Detailed description

A. Remarks on omission of P and/or L

G10 L20 Pn Xx Yy Zz When n = 1 to 300: Correct setting of data for Pn-system origin
Otherwise: Alarm **809 ILLEGAL NUMBER INPUT**

G10 L20 Xx Yy Zz Correct setting of workpiece origin data for the current system, except for G54- to G59-system (in which case: Alarm **807 ILLEGAL FORMAT**)

G10 Pn Xx Yy Zz Correct setting of workpiece origin data for the current system

G10 Xx Yy Zz Correct setting of workpiece origin data for the current system

B. Precautions for programming

- Do not set together in a block of G54.1 or L20 any G-code that can refer to address P.

Such G-codes are for example:

G04 Pp Dwell
 G30 Pp Reference-point return
 G72 to G89 Fixed cycle
 G65 Pp, M98 Pp Subprogram call

- Setting the G54.1-command without the option will cause the alarm **948 NO G54.1 OPTION**.
- Setting "G10 L20" without the option will cause the alarm **903 ILLEGAL G10 L NUMBER**.
- Local coordinate system setting is not available in G54.1 mode. Setting a G52-command during G54.1 mode will cause the alarm **949 NO G52 IN G54.1 MODE**.

C. Related system variables

The origin data of additional workpiece coordinate systems are assigned to system variables as listed-up in the following table.

Note: The total number of controllable axes depends on the machine specifications.

	1st axis	to	16th axis
P1	#70001	to	#70016
P2	#70021	to	#70036
	:		
	:		
	:		
P299	#75961	to	#75976
P300	#75981	to	#75996

or

	1st axis	to	16th axis
P1	#7001	to	#7016
P2	#7021	to	#7036
	:		
	:		
	:		
P47	#7921	to	#7936
P48	#7941	to	#7956

The variable number for the k-th axis origin of the "Pn" coordinate system can be calculated as follows:

$$70000 + 20(n - 1) + k$$

(n = 1 to 300, k = 1 to 16)

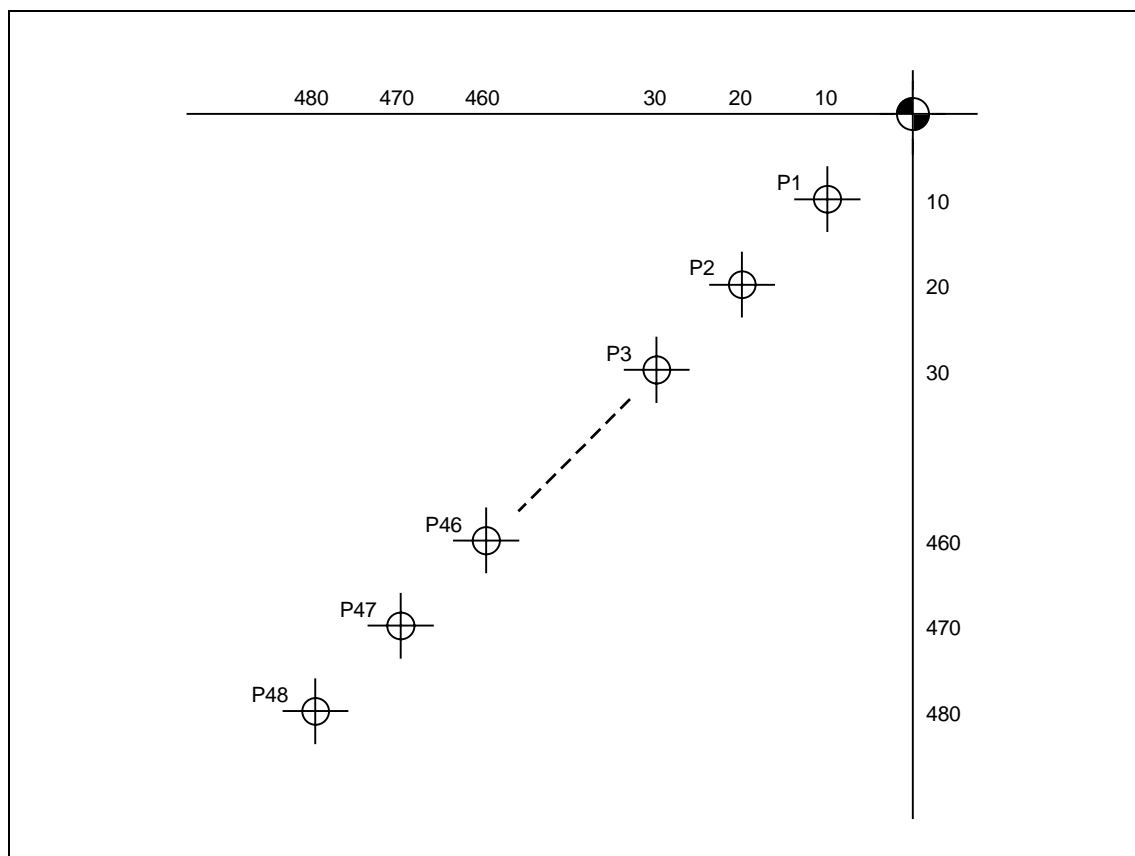
or

$$7000 + 20(n - 1) + k$$

(n = 1 to 48, k = 1 to 16)

4. Sample programs

1. Consecutive setting of origin data for all the 48 sets of additional workpiece coordinate system



Setting in format "G10L20PpXxYyZz"

O100

#100=1

#101=10.

WHILE[#100LT49]DO1

G90G10L20P#100X#101Y#101

#100=#100+1

#101=#101+10.

END1

M30

%

P-No. initial.

Origin setting

P-No. count-up

Setting in assignment of variables

O200

G90

#100=7001

#101=10.

#102=1

WHILE[#102LT49]DO1

#103=0

WHILE[#103LT2]DO2

#[#100]=#101

#100=#100+1

#103=#103+1

END2

#100=#100+19

#101=#101+10.

#102=#102+1

END1

M30

%

Sys.-var.-No. initial.

Counter initial.

Counter initial.

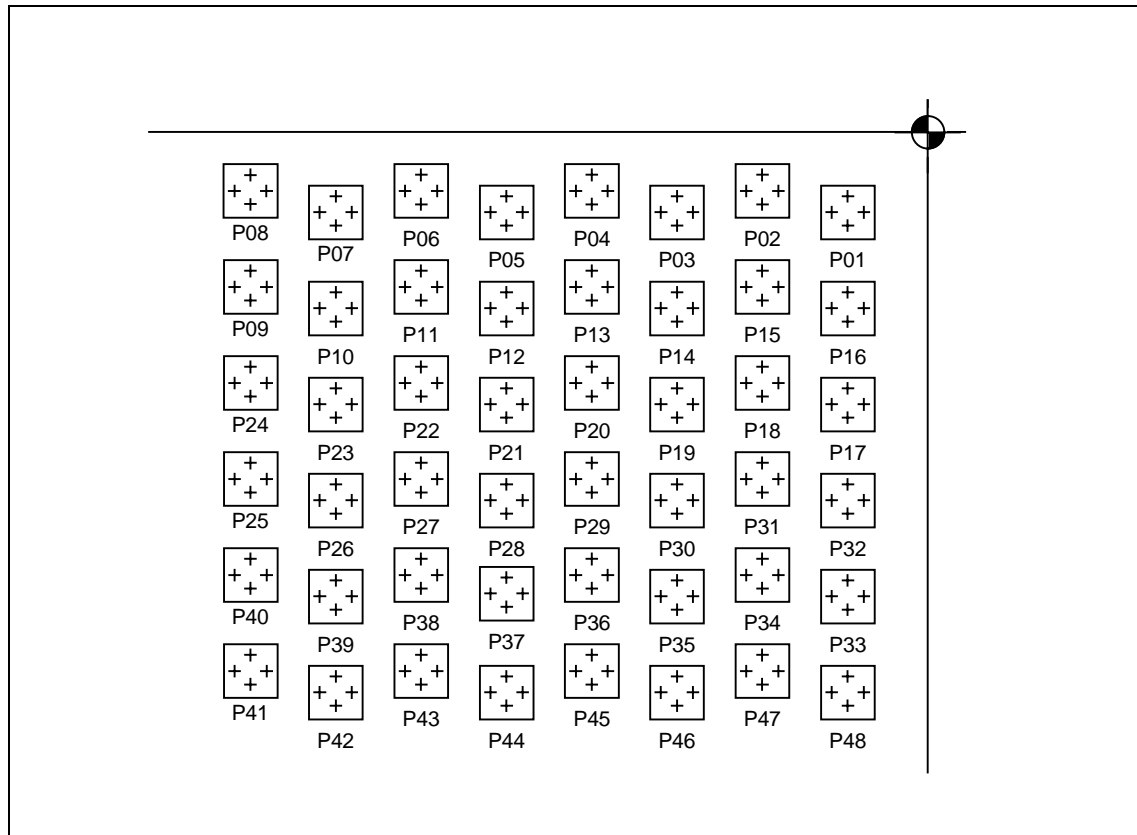
Sys.-var. Setting

Sys.-var.-No. count-up

Counter count-up

2. Consecutive application of all the 48 sets of additional workpiece coordinate system

Provided that preparatory setting of origin data in P1 to P48 is completed in accordance with the 48 workpieces fixed on the table in the arrangement shown in the figure below:



O1000 (Main prg.)

```
G28XYZ
#100=1
G90
WHILE[#100LT49]DO1
G54.1P#100
M98P1001
#100=#100+1
END1
G28Z
G28XY
M02
%
```

Reference-pt. return
P-No. initial.
Absolute data input
Repeat while P-No.<49
Wpc. coordn. sys. Setting
Subprogram call
P-No. count-up

Reset to reference-pt.

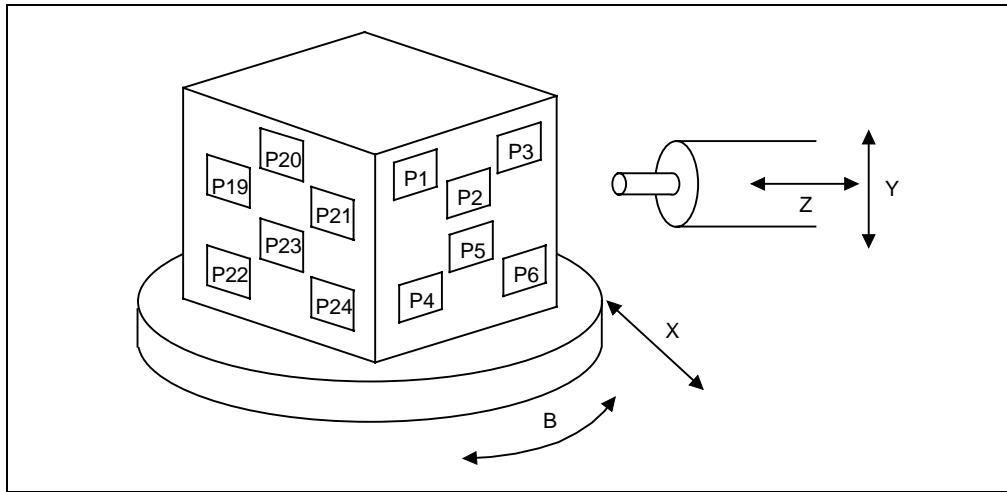
O1001 (Subprg.)

```
G43X-10.Y-0.Z-100.H10
G01X-0.
Y-30.
X-10.
Y-10.
G00G40Z10.
G98G81X-20.Y-15.Z-150.R5.F40
X-25.Y-20.
X-20.Y-25.
X-15.Y-20.
G80
M99
%
```

Profile
Drilling

3. Application of additional systems via transmission into G54 to G59

Provided that preparatory setting of origin data in P1 to P24 is completed in accordance with the 24 sections of a workpiece fixed on the rotary table as shown in the figure below:



O2000 (Main prg.)

```
G28 XYZB      Reference-pt. return
G90           Absolute data input
G00 B0        Table indexed f. 1. surf.
G65 P2001A1   Origin-data loading
M98 P2002     Drilling-subprg. call
G00 B90.      Table indexed f. 2. surf.
G65 P2001A7   Table indexed f. 3. surf.
M98 P2002     Table indexed f. 4. surf.
G00 B180.
G65 P2001A13
M98 P2002
G00 B270.
G65 P2001A19
M98 P2002
G28 XYB       Reset to reference-pt.
M02
%
```

O2001 (Origin-data transference)

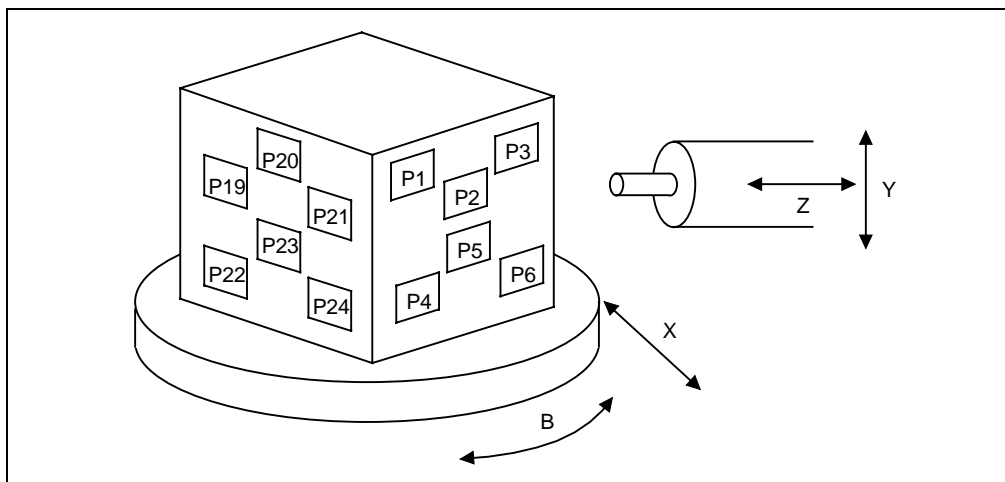
```
#2=5221      Init.-var.-No. of 1. G-sys.
#3=[#1-1].20+7001 Init.-var.-No. of 1. P-sys.
#5=0         Sys.-qty-counter cleared
WHILE[#5LT6]DO1
#6=#2       Check f. sys.-qty.
#7=#3       1.-ax.-var.-No. of receiver
#4=0        1.-ax.-var.-No. of transm.
WHILE[#4LT6]DO2
#[#6]=#[#7] Axis-qty-counter cleared
#6=#6+1     Check f. axis-qty.
#7=#7+1     Transmission of var.-data
#4=#4+1     Axis-qty count-up f. recvr.
END2         Axis-qty count-up f. transm.
#2=#2+20    Axis-qty counter up
#3=#3+20
#5=#5+1
END1
M99
%
```

O2002 (Drilling subprg.)

```
G54 M98 H100      Drilling in G54-system
G55 M98 H100      Drilling in G55-system
G56 M98 H100      Drilling in G56-system
G57 M98 H100      Drilling in G57-system
G58 M98 H100      Drilling in G58-system
G59 M98 H100      Drilling in G59-system
G28 Z0
M99
N100G98G81X-0.Y-15.Z-50.R5.F40 Fixed cycle for drilling
X-25. Y-20.
X-20. Y-25.
X-15. Y-20.
G80
G28Z
M99
%
```

4. Simplified version of **Example 3** program in application of "G54.1 Pp"

Provided that preparatory setting of origin data in P1 to P24 is completed in accordance with the 24 sections of a workpiece fixed on the rotary table as shown in the figure below:



O3000	
G28 XYZB	Reference-point return
G90	Selection of absolut data input
G00 B0	Indexing of table for 1st surface
G65 P3001A1	
G00 B90.	Indexing of table for 2nd surface
G65 P3001A7	
G00 B180.	Indexing of table for 3rd surface
G65 P3001A13	
G00 B270.	Indexing of table for 4th surface
G65 P3001A19	
G28 XYB	Reset to reference-point
M30	
%	
O3001	
#100=#1	Initialization of P-No.
#101=0	Initialization of counter
WHILE[#101LT6]DO1	
G54.1P#100	Setting of additional workpiece coordinate system
M98H100	Call for drilling subroutine
#100=#100+1	P-No. count-up
#101=#101+1	Checking counter count-up
END1	
G28Z0	
M99	
N100G98G81X-20.Y-15.Z-150.R5.F40	Fixed cycle for drilling
X-25. Y-20.	
X-20. Y-25.	
X-15. Y-20.	
G80	
G28Z	
M99	
%	

14-11 Local Coordinate System Setting: G52

1. Function and purpose

A local coordinate system in which the designated position becomes the program origin can be set on the current workpiece coordinate system by setting command code G52.

Command code G52 can also be used instead of G92 to update any positional shift between the machining program origin and the workpiece origin.

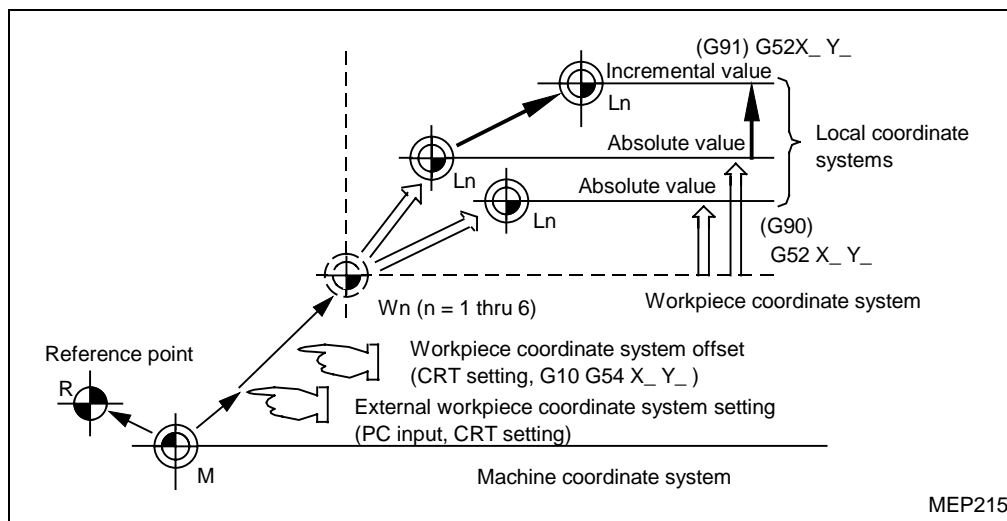
2. Programming format

G52 $X_{\alpha_1} Y_{\alpha_1} Z_{\alpha_1} \alpha_{\alpha_1}$ (α : Additional axis)

3. Detailed description

Command G52, which causes no machine movement, is valid until a new G52 is issued. Command G52 is therefore convenient for using a new coordinate system without changing the origin of the workpiece coordinate system.

The local coordinate system offset data is cleared by execution of either an automatic reference-point return operation, or a manual reference-point (zero point) return operation using the watchdog method following power-on.



Coordinates given in the absolute data input mode (G90) refer to the local coordinate system.

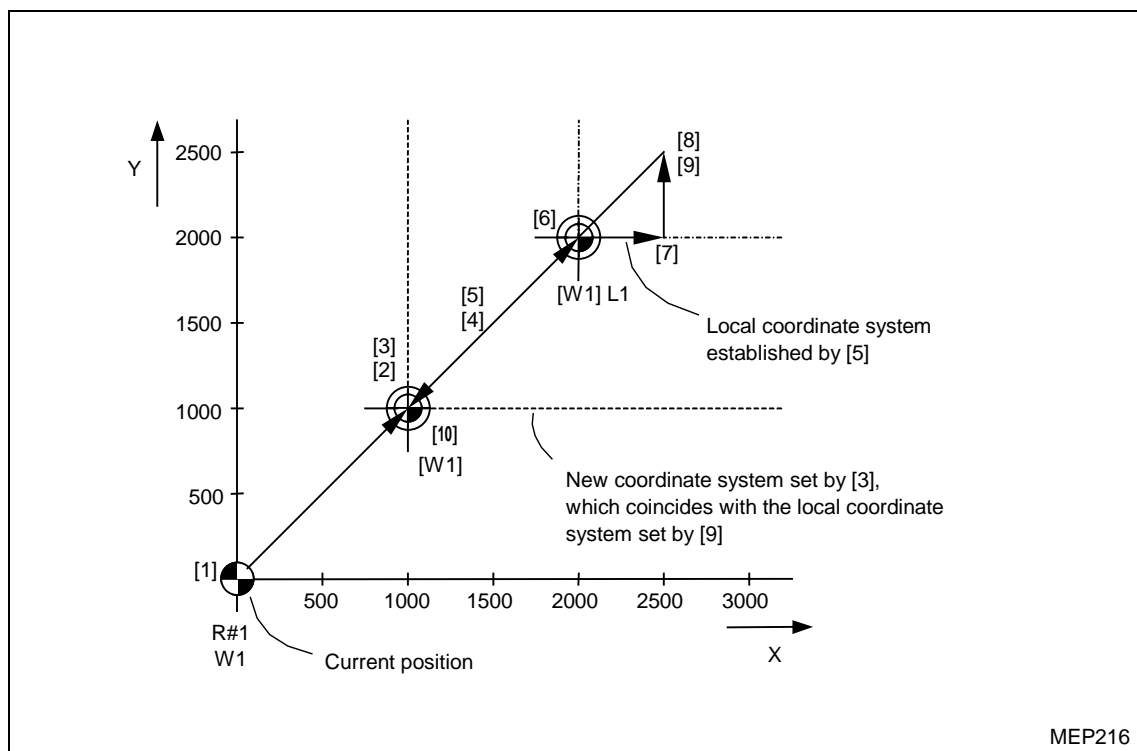
4. Sample programs

1. Setting local coordinates in the absolute data input mode (local coordinate system offset data is not integrated)

```

[1] G28X0Y0
[2] G00G90X1.Y1.
[3] G92X0Y0
[4] G00X500.Y500
[5] G52X1.Y1.
[6] G00X0Y0
[7] G01X500.F100
[8] Y500.
[9] G52X0Y0
[10] G00X0Y0

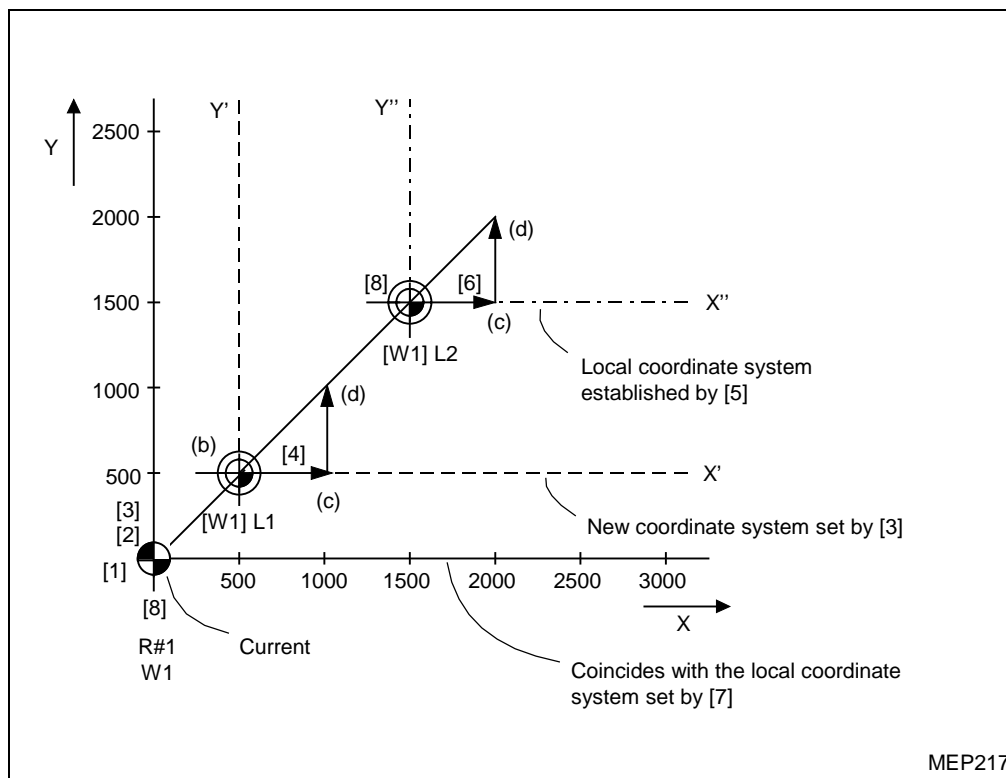
```



The local coordinate system is established by [5] and cancelled by [9], and coincides with the coordinate system of [3].

2. Setting local coordinates in the incremental data input mode (local coordinate system offset data is integrated)

[1] G28X0Y0	(a) O100
[2] G92X0Y0	(b) G90G00X0Y0
[3] G91G52X500.Y500.	(c) G01X500.
[4] M98P100	(d) Y500.
[5] G52X1.Y1.	(e) G91
[6] M98P100	(f) M99
[7] G52X-1.5Y-1.5	
[8] G00G90X0Y0	
⋮	
M02	

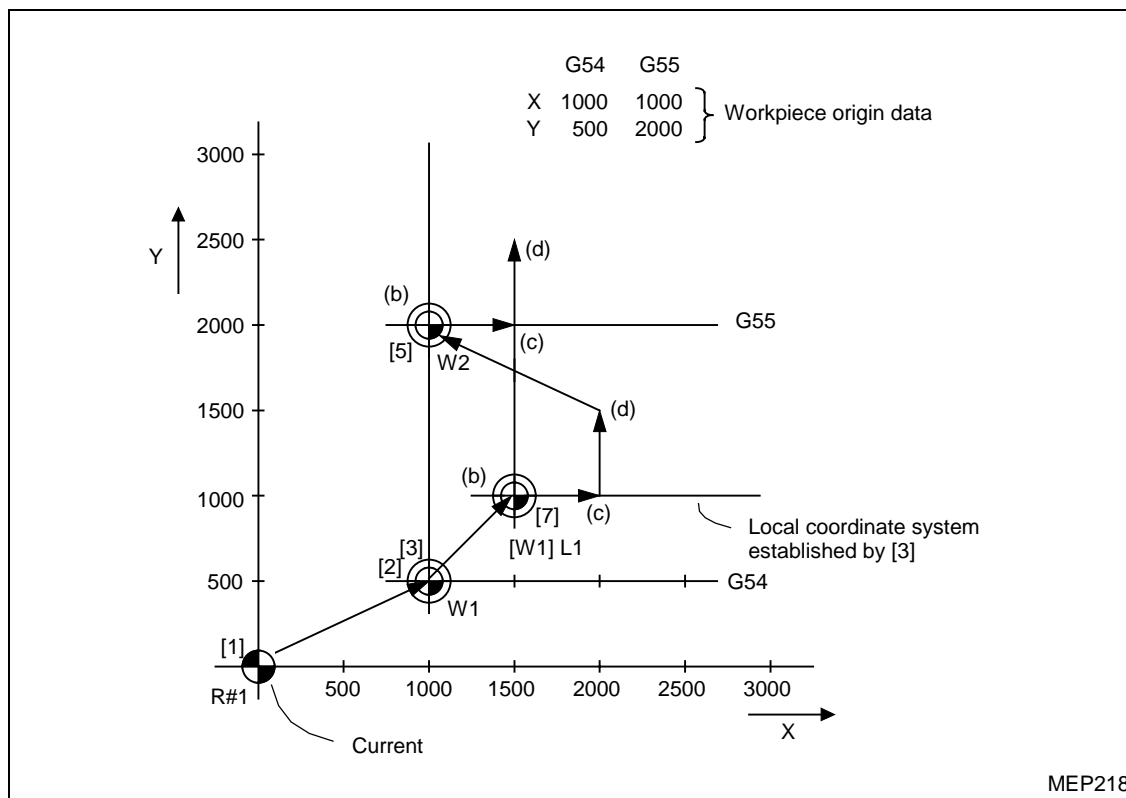


Block [3] generates local coordinate system X'-Y' at the position of (500, 500) in the X-Y coordinate system.

Block [5] generates local coordinate system X''-Y'' at the position of (1000, 1000) in the X'-Y' coordinate system.

Block [7] generates a local coordinate system at the position (-1500, -1500) in the X''-Y'' coordinate system. That is, the local coordinate system and the X-Y coordinate system coincide, which means that the former has been cancelled.

[1]	G28X0Y0	(a)	O200
[2]	G00G90G54X0Y0	(b)	G00X0Y0
[3]	G52X500.Y500.	(c)	G01X500.F100
[4]	M98P200	(d)	Y500.
[5]	G00G90G55X0Y0	(e)	M99
[6]	M98P200		%
[7]	G00G90G54X0Y0		
	:		
	M02		



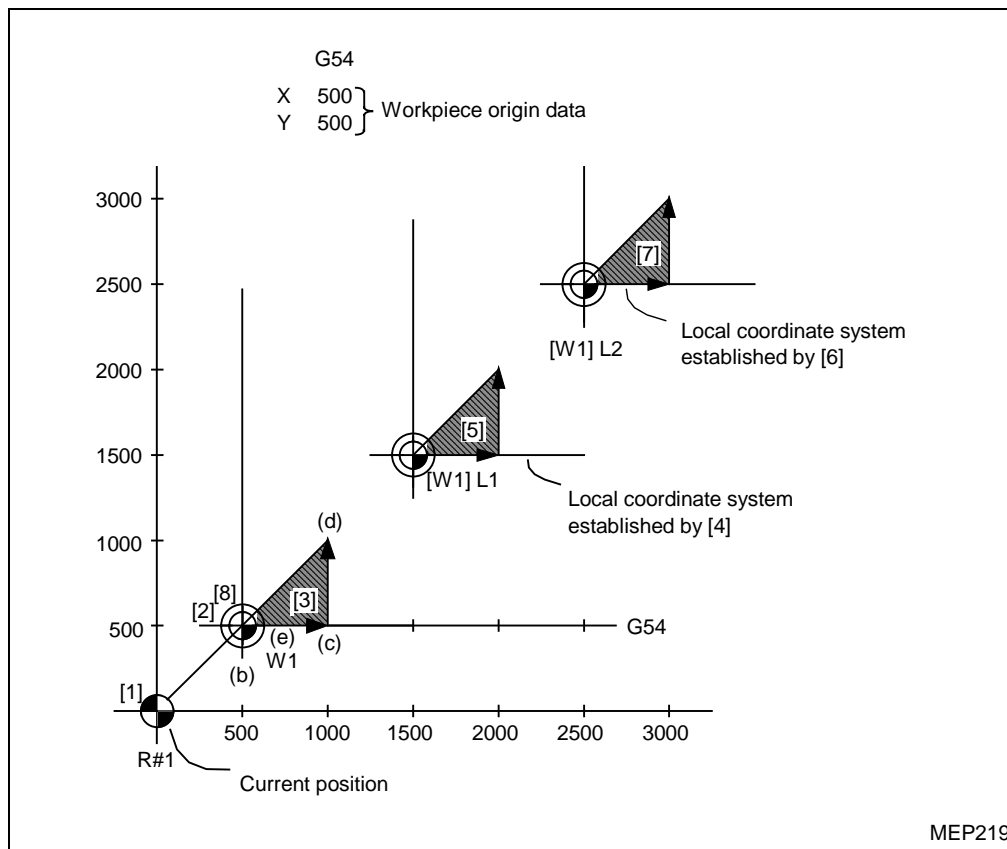
Block [7] performs a movement to the local coordinate system reference point (zero point) of G54.

The local coordinate system can be cancelled using this format:

G90 G54 G52 X0 Y0

4. Combined use of G54 workpiece coordinate system and multiple local coordinate systems

[1] G28X0Y0	(a) O300
[2] G00G90G54X0Y0	(b) G00X0Y0
[3] M98P300	(c) G01X500.F100
[4] G52X1.Y1.	(d) Y500.
[5] M98P300	(e) X0Y0
[6] G52X2.Y2.	(f) M99
[7] M98P300	%
[8] G52X0Y0	
⋮	
M02	



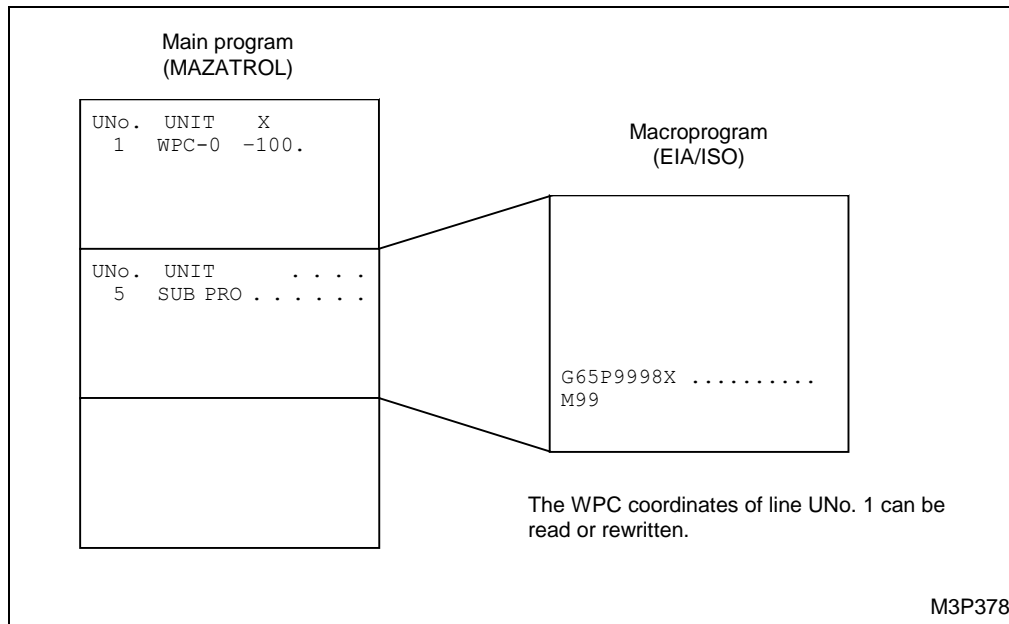
Block [4] generates a local coordinate system at the position of (1000, 1000) in the G54 coordinate system.

Block [6] generates a local coordinate system at the position of (2000, 2000) in the G54 coordinate system.

Block [8] makes the local coordinate system coincide with the G54 coordinate system.

14-12 Reading/Writing of MAZATROL Program Basic Coordinates

The basic coordinates of a MAZATROL program can be read or rewritten by calling a user macroprogram in the subprogram unit of the MAZATROL program.



Note 1: For updating the basic coordinates, do not forget to select the **[MEASURE MACRO]** menu function in entering a subprogram unit for the macroprogram concerned (WNo. 9998); otherwise the new coordinates may not always be used in time for the execution of the machining unit immediately following the subprogram unit.

Note 2: On the other hand, do not select the **[MEASURE MACRO]** menu function unless the above macroprogram is to be used.

14-12-1 Calling a macroprogram (for data writing)

To rewrite the basic coordinates data, call the specific user macroprogram from a subprogram unit of the MAZATROL program (macroprogram call is not required for data reading).

Refer to the section of subprogram unit in the PROGRAMMING MANUAL (MAZATROL Program) for details on data setting for subprogram call.

14-12-2 Data reading

System variables can be used to read the MAZATROL basic coordinates that are effective during macroprogram execution.

System variables for MAZATROL basic coordinates (WPC)

Variables number	Function	Variables number	Function
#5341	WPC-X	#5344	WPC-A
#5342	WPC-Y	#5345	WPC-C
#5343	WPC-Z	#5347	WPC-th

14-12-3 Rewriting

Same as reading, you are to use system variables when rewriting the basic coordinates.

The basic coordinates, however, cannot be rewritten just by entering the data into #5341 through #5347. It is therefore necessary to create a macroprogram in the format shown below.

Note: It is not possible to rewrite, using the macroprogram, the **R. T** (Rotary Table) setting, a special item in the basic coordinate unit (WPC) which is used in reference to the table selection function optionally enabled.

1. Macroprogramming format



The body of the macroprogram
to be created by the user

G65 P9998 X_Y_Z_D_B_C_

M99

- At the end of macroprogramming, call the rewriting macroprogram (WNo. 9998). At this time, assign new coordinates as arguments. The relationship between each argument and axis is as follows:

X: WPC-X Y: WPC-Y Z: WPC-Z

D: WPC-th B: WPC-A C: WPC-C

- Only the coordinates assigned with the respective argument are rewritten. The argument data is handled as data having the decimal point.

2. Rewriting macroprogram

The rewriting macroprogram (WNo. 9998) is shown below.

O9998	N30	#50467=#50467OR-65536
IF[#50600EQ0]GOTO60	IF[#7EQ#0]GOTO40	#50499=#50499OR1
IF[#24EQ#0]GOTO10	#5347=#7	N60
#5341=#24	#50441=#7	M99
#50449=#24	#50467=#50467OR512	%
#50467=#50467OR32	N40	
N10	IF[#2EQ#0]GOTO45	
IF[#25EQ#0]GOTO20	#5344=#2	
#5342=#25	#50443=#2	
#50447=#25	#50467=#50467OR256	
#50467=#50467OR64	N45	
N20	IF[#3EQ#0]GOTO50	
IF[#26EQ#0]GOTO30	#5345=#3	
#5343=#26	#58161=#3	
#50445=#26	#50467=#50467OR2048	
#50467=#50467OR128	N50	

Note: An alarm will occur when executing the macroprogram if no basic coordinates of MAZATROL program are currently validated.

14-13 Workpiece Coordinate System Rotation

1. Function and purpose

The function refers to rotating the workpiece coordinate system around the position of the specified machine coordinates. The machining program can be rotated in its entirety as required for the actual inclination of the workpiece.

2. Programming format

(G17) G92.5 Xx Yy Rr X-Y plane

(G18) G92.5 Zz Xx Rr Z-X plane

(G19) G92.5 Yy Zz Rr Y-Z plane

or

(G17) G92.5 Xx Yy li Jj X-Y plane

(G18) G92.5 Zz Xx Kk li Z-X plane

(G19) G92.5 Yy Zz Jj Kk Y-Z plane

x, y, z : Coordinates of the rotational center.

The position along the two axes of the previously selected X-Y, Z-X, or Y-Z plane must be designated in machine coordinates.

The designation for an axis not corresponding to the plane will be ignored.

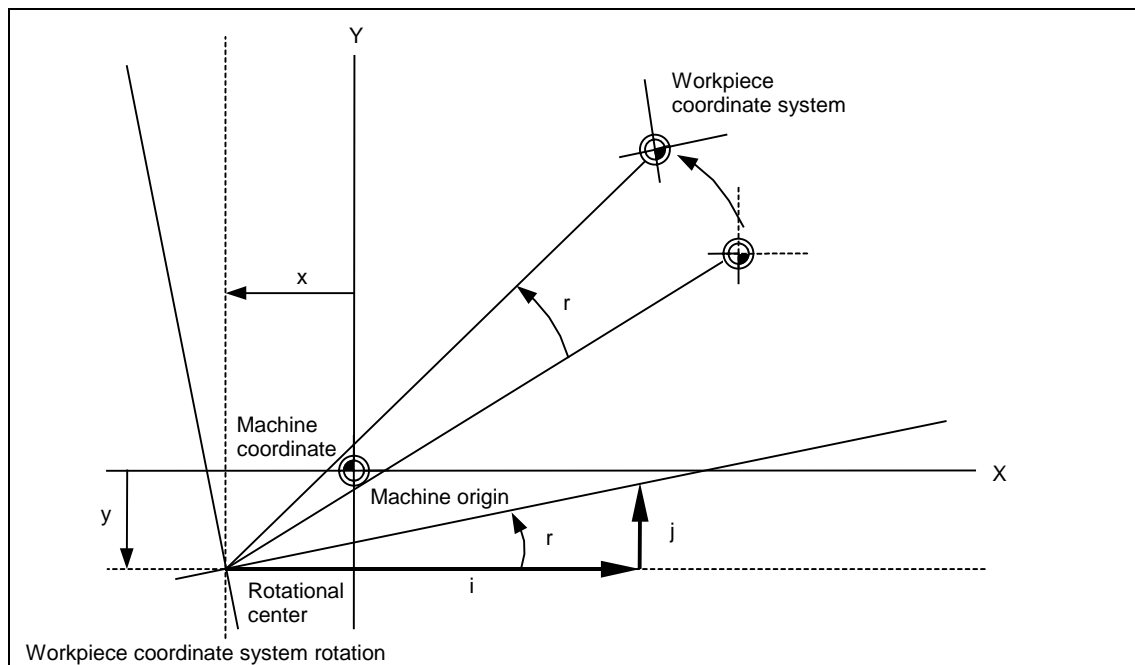
R : Angle of rotation.

Designate the rotational angle for the coordinate system. A positive value refers to a counterclockwise rotation.

i, j, k : Axial component vectors.

The angle for coordinate system rotation can also be designated in axial component vectors corresponding to the previously selected plane.

The designation for an axis not corresponding to the plane will be ignored.



Range and unit for angle data setting

Setting method		Setting range	Setting unit
Axial component vectors (i, j, k)	Metric system	0 to ±99999.999	0.001 mm
	Inch system	0 to ±9999.9999	0.0001 in
Angle of rotation (r)	Metric system	0 to ±99999.999°	0.001°
	Inch system		

3. Detailed description

1. Irrespective of the actual mode for incremental or absolute data input, the values at addresses X, Y, Z, or I, J, K as well as R are always referred to the machine coordinate system.
2. Two methods are available to designate a rotational angle:
 - (a) Designation in rotational angle (r), or
 - (b) Designation in axial component vectors (i, j, k).
3. If angle data are entered using both methods (a) and (b) above, the rotational angle (at address R) will govern.
4. If, during rotation of the workpiece coordinate system, a rotational angle of zero degrees is designated (by setting G92.5 R0, for example), the coordinate system rotation will be cancelled, irrespective of the data input mode of G90 (absolute) or G91 (incremental). The next move command will then be executed for the ending point in the original (not rotated) workpiece coordinate system (refer to Article 1 in Item 5. Precautions).
5. The rotational center coordinates will be retained and automatically applied for a succeeding rotation command without data designation at addresses X, Y, and/or Z.

Example:

N1 G17	Selection of the X-Y plane
N2 G92.5X100.Y100.R45.	Rotation of the workpiece coordinate system through 45 deg around the point of (X, Y) = (100, 100)
⋮	
N3 G92.5R0	Cancellation of the workpiece coordinate system rotation
⋮	
N4 G17G92.5R90.	Rotation of the workpiece coordinate system through 90 deg around the center last programmed (X100, Y100)
⋮	
%	

6. Omission of addresses R and I, J, K is regarded as a rotational angle designation of zero degrees.

Example: "G92.5 X0. Y0." is equivalent to "G92.5 X0. Y0. R0".

7. Alarm No. **809 ILLEGAL NUMBER INPUT** will be displayed if the specified axial component vectors (i, j, k) or rotational angle (r) oversteps the effective setting range.
8. Plane selection (by codes G17, G18, and G19) need not be included in the block of G92.5, if the rotation shall be performed on the currently active plane.
9. The designation for an axis not corresponding to the selected plane will be ignored. The designations at addresses Z and K in a block of G92.5, for example, will be ignored in the mode of G17 (X-Y plane).

Example: The second block shown below rotates the workpiece coordinate system through 63.435°, calculation from $\tan^{-1}(2/1)$, around the point of (X, Y) = (10, 20) on the X-Y plane, and the values at Z and K are ignored for the rotation.

```
G17
G92.5X10.Y20.Z30.I1.J2.K3.
```

Even the ignored axial values at X, Y, and Z in a G92.5 block are retained as well as the values actually used (see Article 5 above) and, for example, if the G92.5 block shown above is followed by

```
G19
G92.5J2.K3.
```

then the workpiece coordinate system will be rotated around the point of (Y, Z) = (20, 30) through 56.301°, calculation from $\tan^{-1}(3/2)$, on the Y-Z plane (G19).

4. Examples of operation

1. Rotation around the machine origin.

N1 G28X0Y0

N2 G17

N3 G90

N4 G55

N5 G92.5X0Y0R90. (or G92.5X0Y0I0J1.)

N6 G0X0Y0

N7 G1X100.F1000.

N8 Y200.

N9 X0.

N10 Y0.

N11 M30

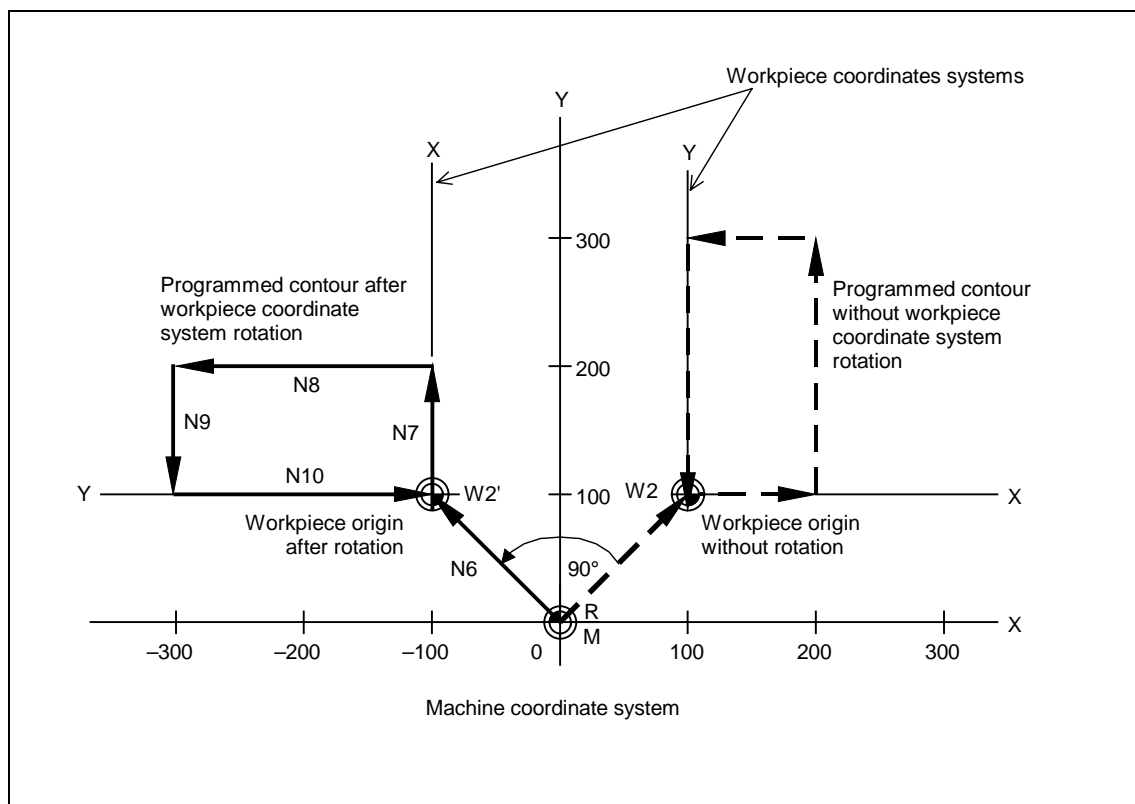
%

G55 (Work Offset)

X100.

Y100.

Rotation around the machine origin
through 90 deg



- The block of G92.5 under N5 rotates the workpiece coordinate system through 90 degrees around the origin of the machine coordinate system. For N6 onward, the machine operates according to the rotated workpiece coordinate system.

- The above example of the vector setting method for the same 90-deg rotation is based on the following calculation:

$$\theta = \tan^{-1} (J/I) = \tan^{-1} (1/0) = 90^\circ.$$

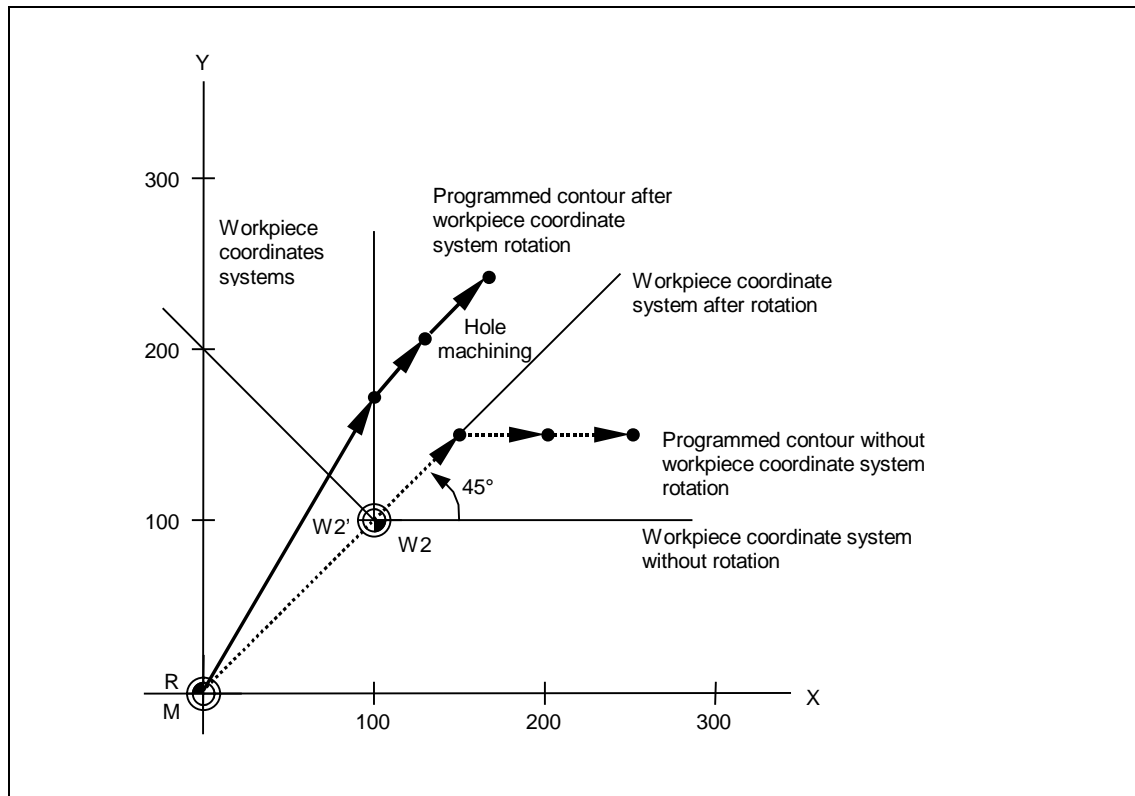
2. Rotation around the workpiece origin.

```

N1 G28X0Y0Z0
N2 G17
N3 G55
N4 G90
N5 G92.5X100.Y100.R45.....
N6 G81X50.Y50.Z-25.R-5.F500
N7 X100.
N8 X150.
N9 M30
%
```

G55 (Work Offset)
X100.
Y100.

Rotation through 45 deg around the point of machine coordinates X=100 and Y=100 (that is, the origin of the G55 workpiece coordinate system).



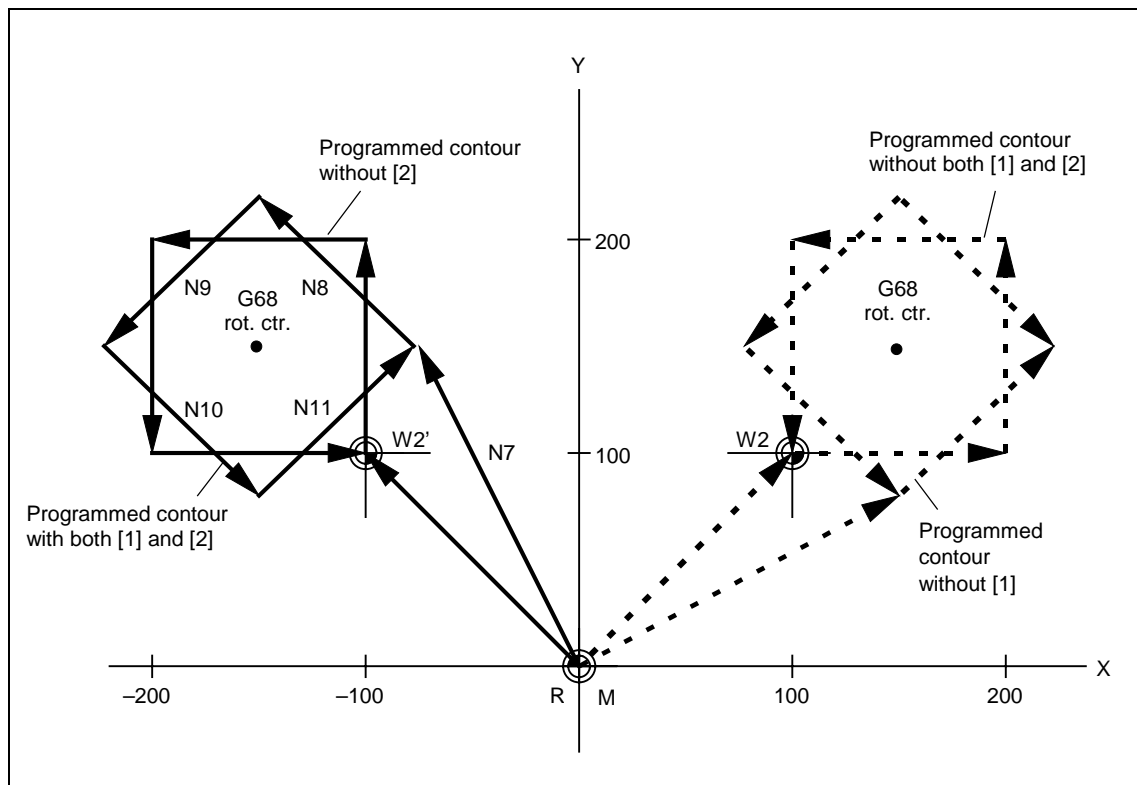
- The block of G92.5 under N5 rotates the workpiece coordinate system around its own origin through 45 degrees. For N6 onward, the machine operates according to the rotated workpiece coordinate system.
- Set the rotational center on the workpiece origin, as shown in this example, to rotate the current workpiece coordinate system around its own origin.

3. Programmed coordinate rotation (G68) in the mode of G92.5

```

N1 G28X0Y0
N2 G17
N3 G55
N4 G90
N5 G92.5X0Y0R90. .... [1]
N6 G68X50.Y50.R45. .... [2]
N7 G0X0Y0
N8 G1X100.F500
N9 Y100.
N10 X0
N11 Y0
N12 M30
%
```

G55 (Work Offset)
X100.
Y100.



In a combined use with G92.5, the center of programmed coordinate rotation by G68 will be a position which corresponds with the workpiece coordinate system rotation designated by the G92.5 command.

It will not affect operation even if the order of the program blocks marked [1] and [2] above is reversed.

4. Figure rotation (M98) in the mode of G92.5

N1 G28X0Y0

N2 G17

N3 G55

N4 G90

N5 G92.5X0Y0R90.

N6 G0X0Y0

N7 M98H10I-50.J50.L4

N8 M30

N9

N10 G1X100.Y50.F500

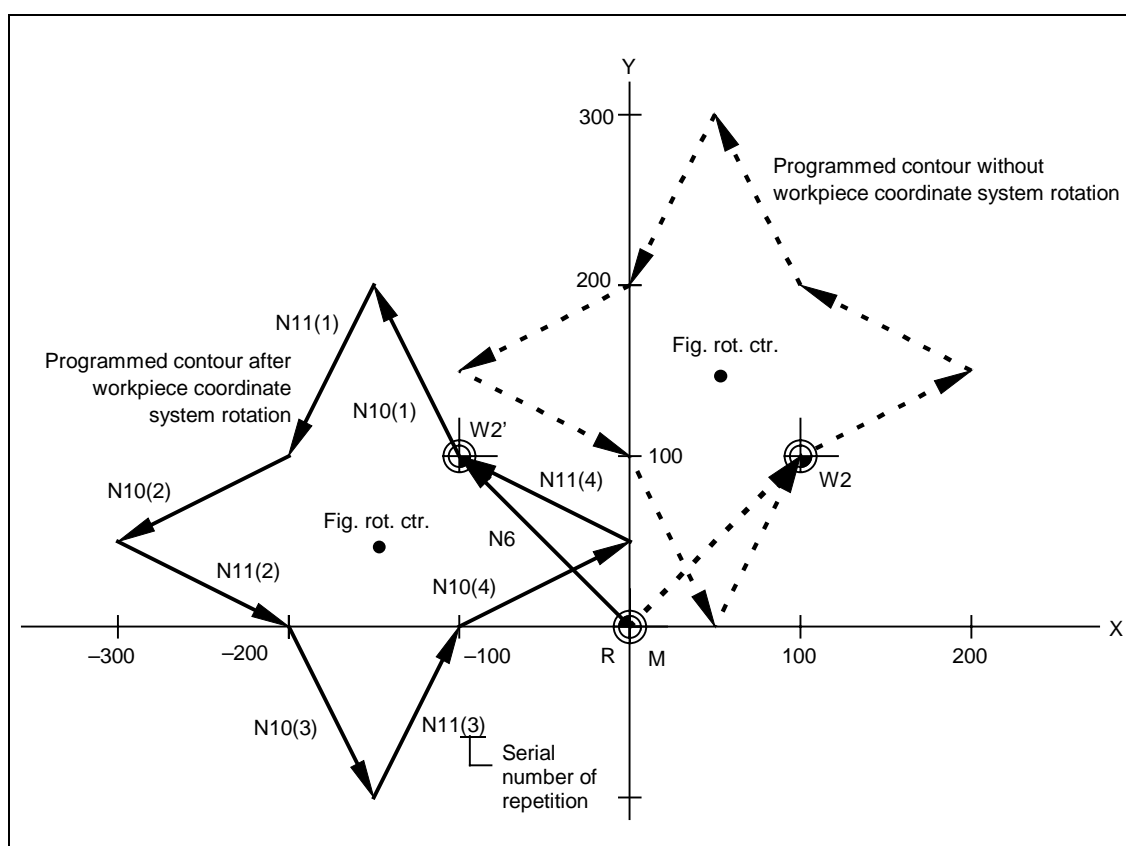
N11 X0Y100.

N12 M99

%

G55 (Work Offset)
X100.
Y100.

Rotation through 90 deg around the origin of the machine coordinate system



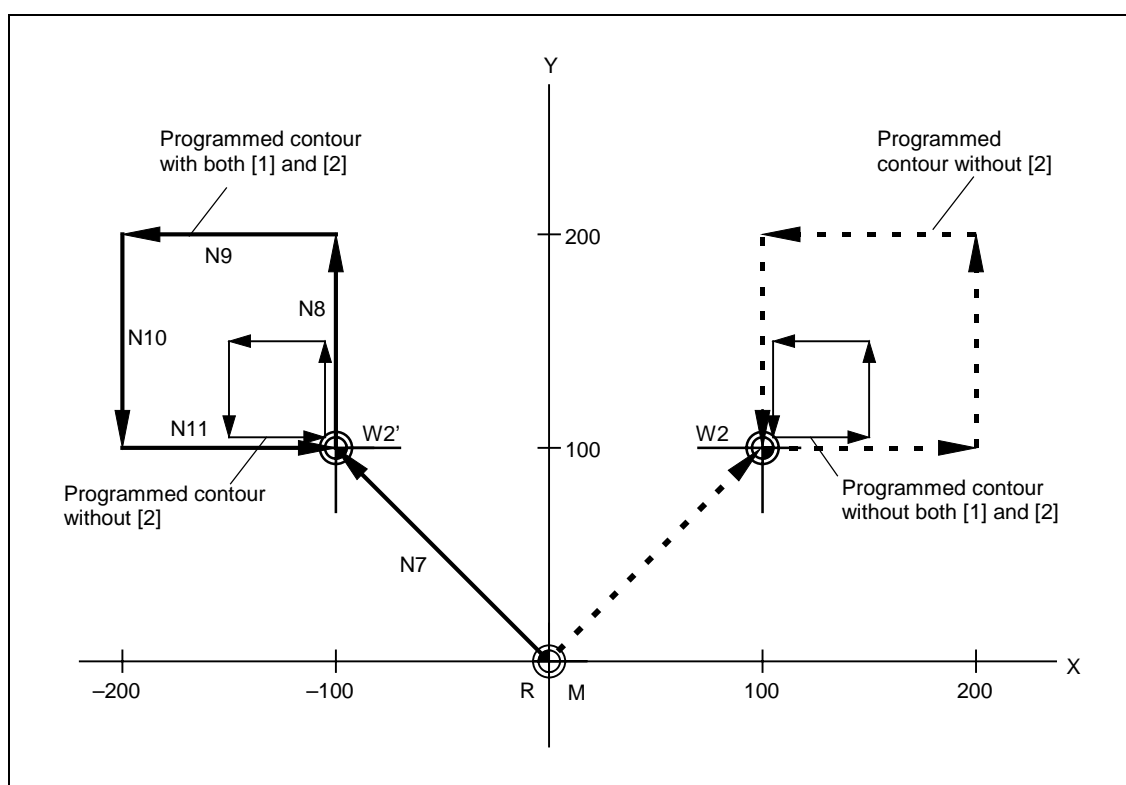
In a combined use with G92.5, the center of figure rotation by M98 will be a position which corresponds with the workpiece coordinate system rotation designated by the G92.5 command.

5. Scaling (G51) in the mode of G92.5

```

N1 G28X0Y0
N2 G17
N3 G55
N4 G90
N5 G92.5X0Y0R90. .... [1]
N6 G51X0Y0P2. .... [2]
N7 G0X0Y0
N8 G1X50.F500
N9 Y50.
N10 X0
N11 Y0
N12 M30
%
```

G55 (Work Offset)
X100.
Y100.



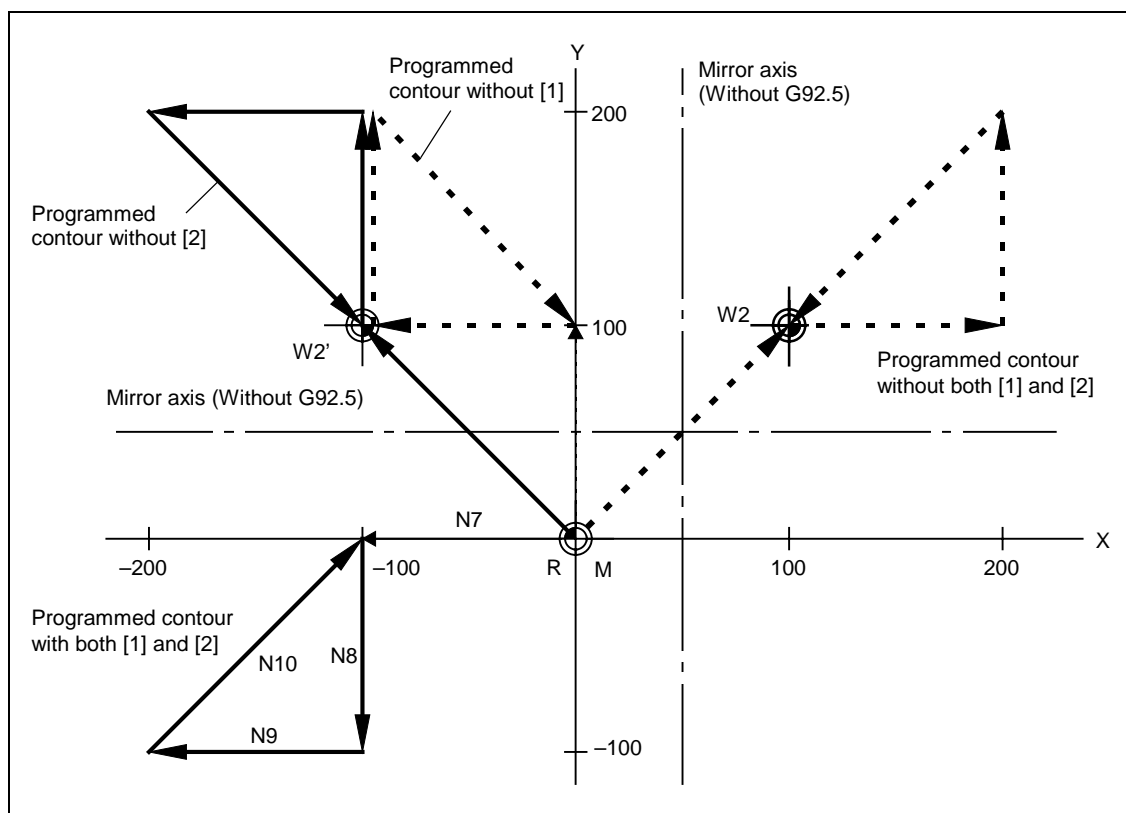
In a combined use with G92.5, the scaling center will be a position which corresponds with the workpiece coordinate system rotation designated by the G92.5 command.

6. Mirror image in the mode of G92.5

a) G-code mirror image

N1	G28X0Y0	
N2	G17	
N3	G55	
N4	G90	
N5	G92.5X0Y0R90.	[1]
N6	G51.1X-50.	[2]
N7	G0X0Y0	
N8	G1X100.F500	
N9	Y100.	
N10	X0Y0	
N11	M30	
%		

G55 (Work Offset)
X100.
Y100.

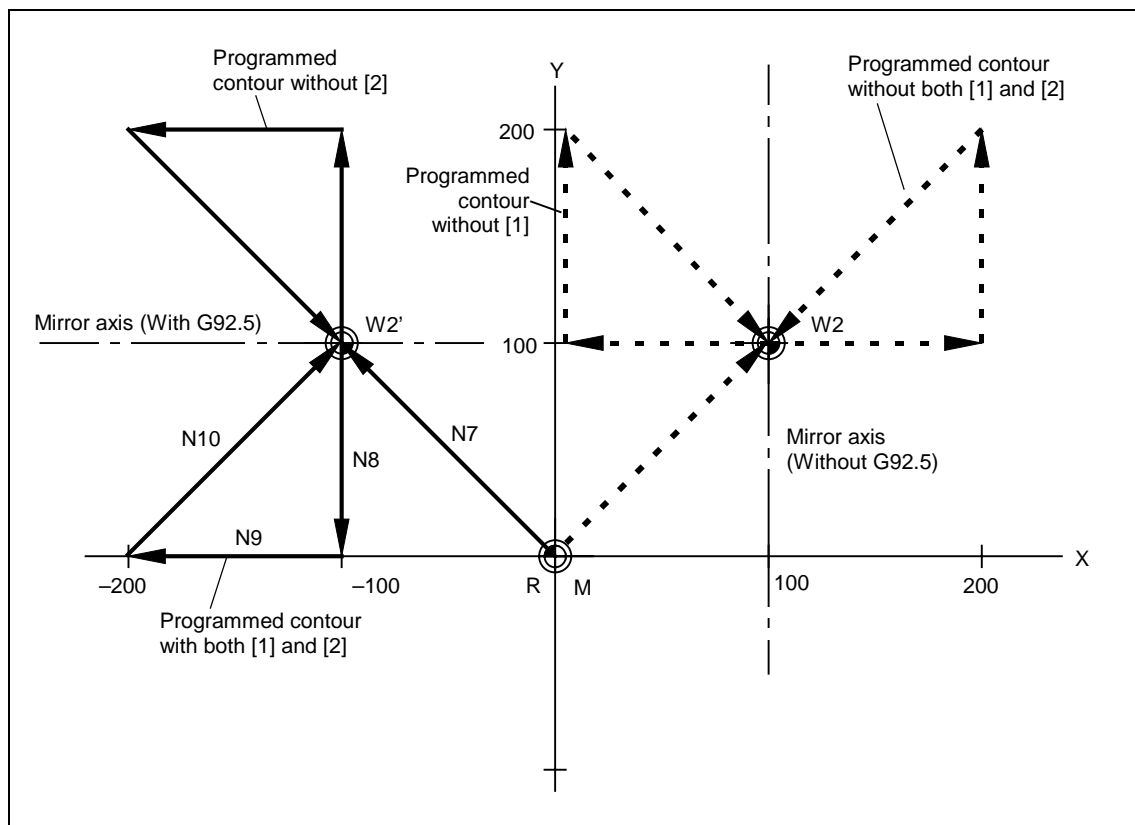


b) M-code mirror image

```

N1  G28X0Y0
N2  G17
N3  G55
N4  G90
N5  G92.5X0Y0R90. .... [1]
N6  M91. .... [2]
N7  G0X0Y0
N8  G1X100.F500
N9  Y100.
N10 X0Y0
N11 M30
%
```

G55 (Work Offset)
X100.
Y100.



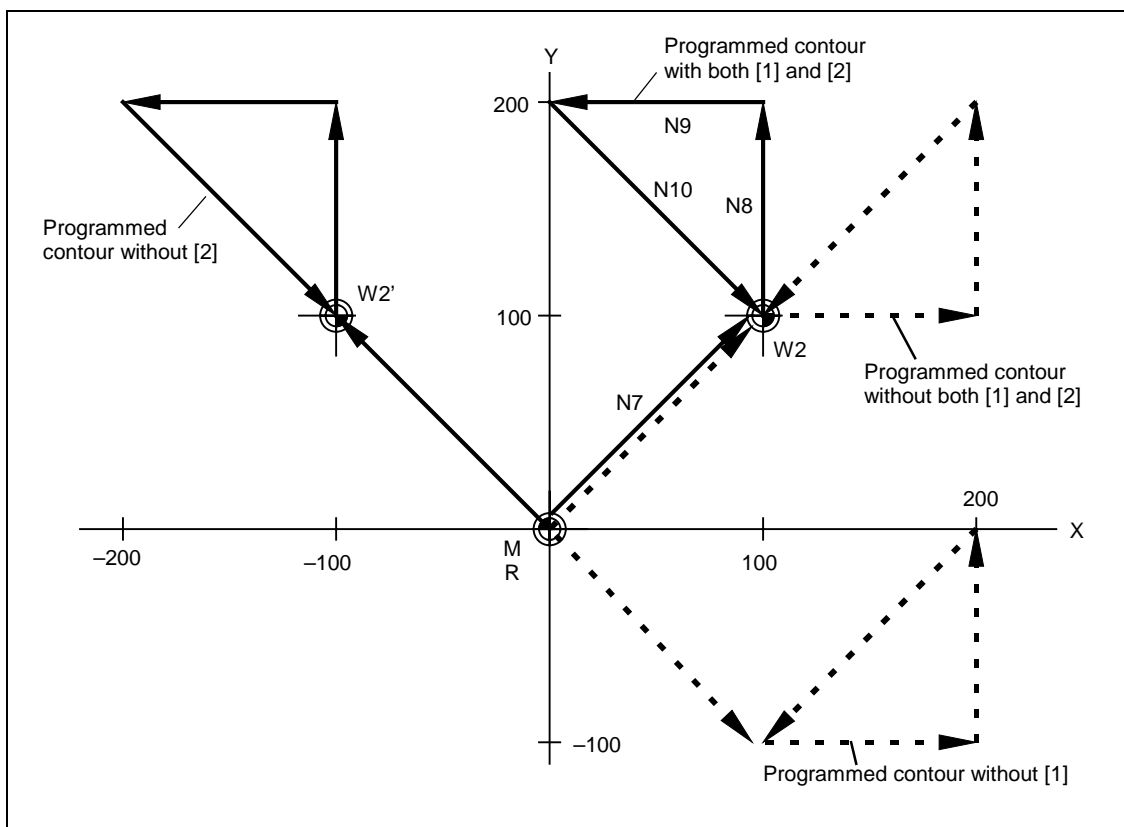
In a combined use with G92.5, the axis of symmetry for G-code or M-code mirror image will be set in accordance with the workpiece coordinate system rotation designated by the G92.5 command.

7. Coordinate system setting (G92) in the mode of G92.5

```

N1  G28X0Y0
N2  G17
N3  G55
N4  G90
N5  G92.5X0Y0R90..... [1]
N6  G92X-100.Y100..... [2]
N7  G0X0Y0
N8  G1X100.F500
N9  Y100.
N10 X0Y0
N11 M30
%
```

G55 (Work Offset)
X100.
Y100.



Coordinate system setting by a G92 block after G92.5 will be performed in reference to the coordinate system rotation designated by the G92.5 command.

5. Precautions

1. If, during rotation of the workpiece coordinate system, a rotational angle of zero degrees is designated (by setting G92.5 R0, for example), the coordinate system rotation will be cancelled, irrespective of the data input mode of G90 (absolute) or G91 (incremental). The next move command will then be executed for the ending point in the original (not rotated) workpiece coordinate system.

Example 1: For incremental data input

```

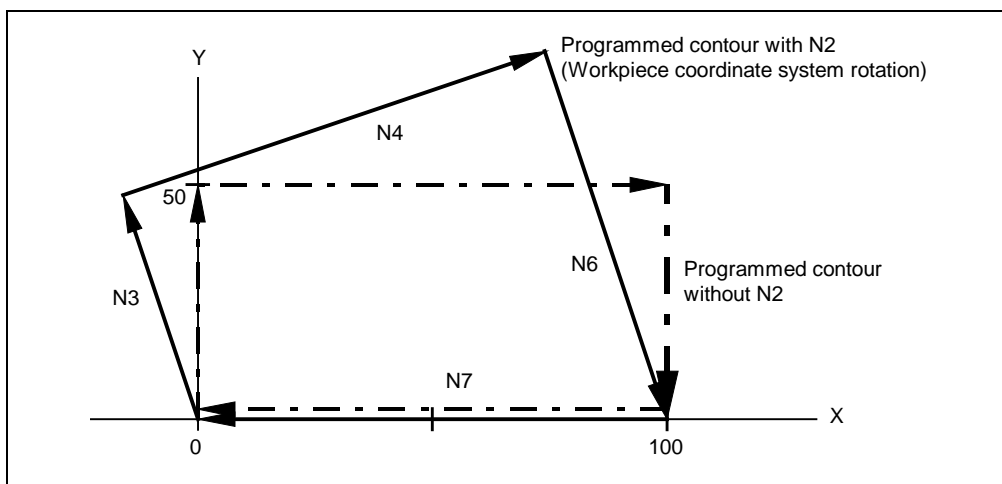
N1 G28X0Y0
N2 G17G92.5X0Y0R20.
N3 G91G01Y50.F1000.
N4 X100.
N5 G92.5R0..... Command for 0-deg rotation
N6 Y-50. .... Motion to (X100, Y0)
N7 X-100.
N8 M30
%
```

Example 2: For absolute data input

```

N1 G28X0Y0
N2 G17G92.5X0Y0R20.
N3 G90G01Y50.F1000.
N4 X100.
N5 G92.5R0..... Command for 0-deg rotation
N6 Y0..... Motion to (X100, Y0)
N7 X0
N8 M30
%
```

Programmed contour for **Examples 1 and 2** above



2. Use a linear motion command (with G00 or G01) for the first movement to be executed after G92.5 command.

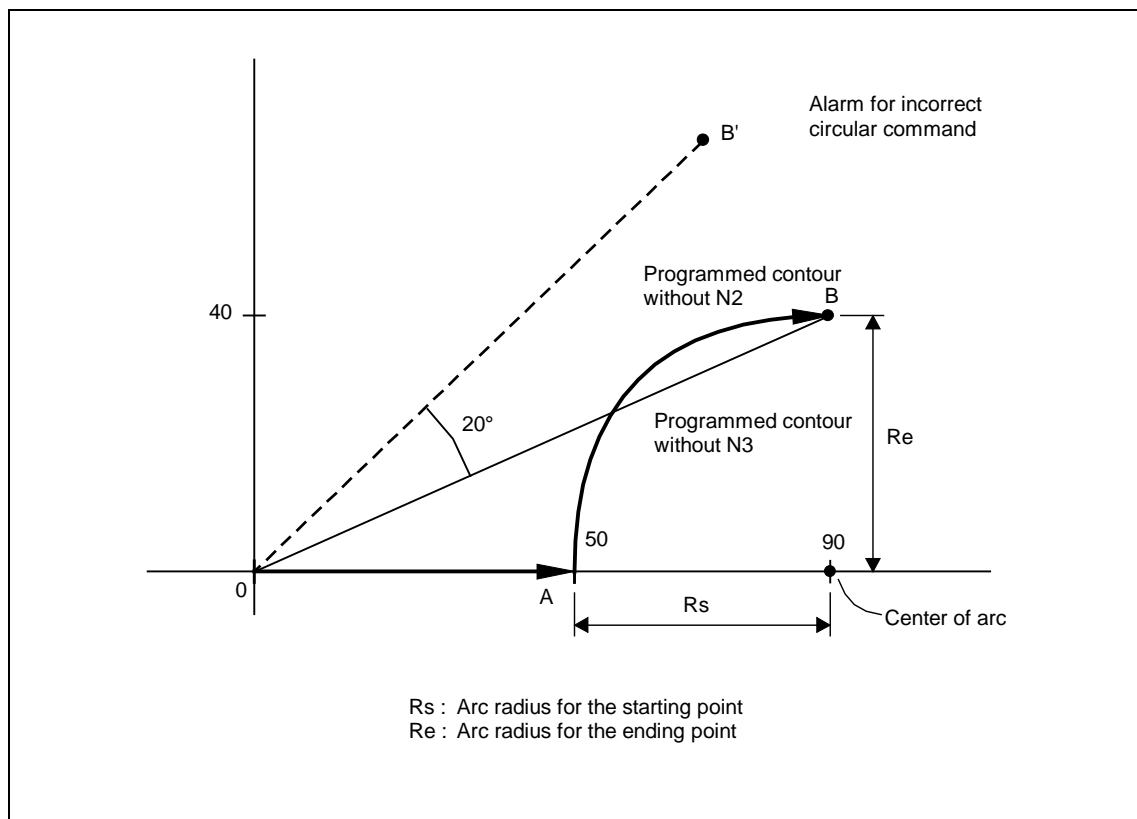
Circular interpolation in such a case, as shown below, would have to take place from the current position A, which refers to the original workpiece coordinate system, to the ending point B' to which the point B should be shifted in accordance with the rotation. As a result, the radii of the starting and ending points would differ too significantly and the alarm **No. 817 INCORRECT ARC DATA** would be caused.

Example:

```

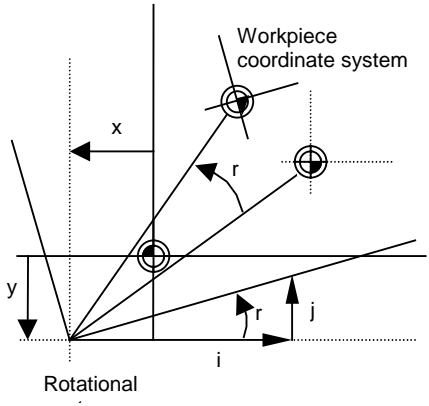
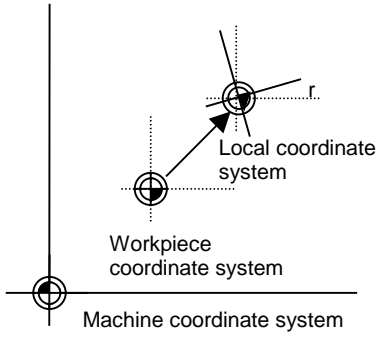
N1  G28X0Y0
N2  G91G01X50.F1000.
N3  G17G92.5X0Y0R20.
N4  G02X40.Y40.I40.
N5  M30
%
```

Circular interpolation as the first motion after G92.5



3. Set a G92.5 command in the mode of G40.
4. The machine will operate on the rotated coordinate system for an MDI interruption during the mode of G92.5.
5. For a manual interruption during the mode of G92.5 using the JOG or handle feed mode, the machine will operate independently of the coordinate system rotation.

6. Differences between workpiece coordinate system rotation and programmed coordinate rotation.

Function name		Workpiece coordinate system rotation	Programmed coordinate rotation
System to be rotated		Workpiece coordinate system	Local coordinate system
Programming format		(G17) G92.5 Xx Yy Rr (G18) G92.5 Yy Zz Rr (Angle) (G19) G92.5 Zz Xx Rr or (G17) G92.5 Xx Yy Ii Jj (G18) G92.5 Yy Zz Jj Kk (Vector comp.) (G19) G92.5 Zz Xx Kk Ii	(G17) G68 Xx Yy Rr (G18) G68 Yy Zz Rr (G19) G68 Zz Xx Rr
Operation			
Rotational center coordinates		Designation at addresses X, Y, Z	Designation at addresses X, Y, Z
Angle of rotation		Designation at R (angle) or at I, J, K (vector components)	Designation at R (angle)
Information on center and angle of rotation cleared?	Power-off → on	Retained	Cleared
	M02/M30	Retained	Cleared
	Reset key	Retained	Cleared
	Resumption of readiness after emergency stop	Retained	Cleared

Note: Resetting or M02/M30 cancels the G92.5 mode itself, while the information on the rotational center, etc., at related addresses is retained as indicated above.

14-14 Three-Dimensional Coordinate Conversion: G68

1. Function and purpose

The three-dimensional (3D) coordinate conversion mode makes it possible for a new coordinate system to be defined by rotating the X-axis, Y-axis, or Z-axis of the currently valid workpiece coordinate system and moving the workpiece origin in parallel to that axis. Thus, definition of any plane on a space allows creation of a program assuming that the plane is an X-Y plane.

2. Programming format

G68 [Xx0 Yy0 Zz0] Ii Jj Kk Rr 3D coordinate conversion mode on

G69 3D coordinate conversion mode off

Where x_0, y_0, z_0 : Rotational center coordinates (Absolute)

i, j, k: Axis of rotation (1: Valid, 0: Invalid)

i: X-axis

j: Y-axis

k: Z-axis

r: Angle and direction of rotation of the coordinate system (the clockwise rotational direction when the positive side of the rotational axis is seen from the center of rotation, is taken as a plus direction)

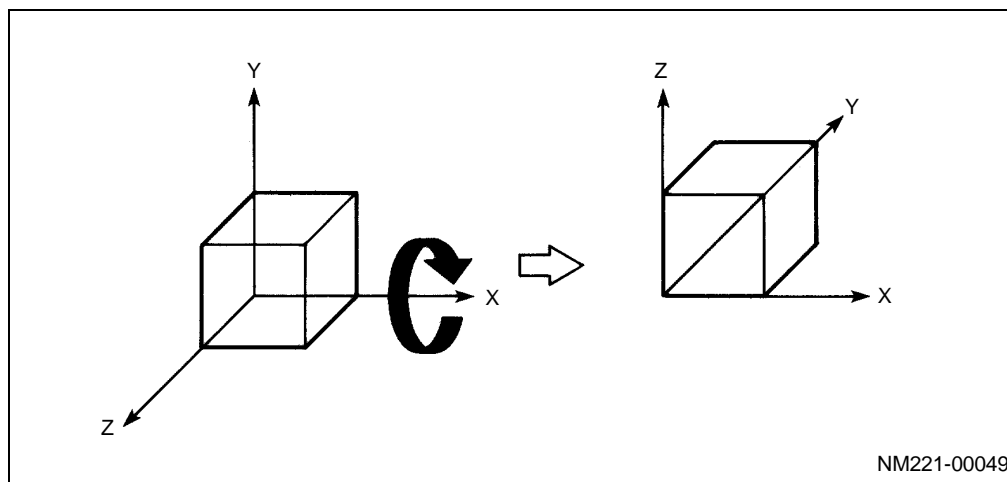
Example:

G68X0Y0Z0 I1J0K0 R-90.

Rotates the coordinate system counterclockwise through 90°.

Sets the X-axis as the axis of rotation of the coordinate system.

Sets the workpiece origin as the center of rotation.



<Precautions>

- If X, Y, or Z is omitted, the workpiece origin in the currently valid workpiece coordinate system will become rotational center coordinates.
- The characters I, J, and K must all be set. Omission of even one of these three characters results in alarm **807 ILLEGAL FORMAT**. Omission of all the three characters makes program coordinate rotation valid, instead of 3D coordinate conversion.

G68X0Y0Z0I1K0R-45. Format error

G68X0Y0Z0R-45. Program coordinate rotation ON

- Setting of 0 in all the arguments of I, J, and K also results in alarm **807 ILLEGAL FORMAT**.
- Setting of 1 in the arguments of two or more of the three characters (I, J, and K) also results in alarm **807 ILLEGAL FORMAT**.

G68X0Y0Z0I1J1K0R-90. Format error

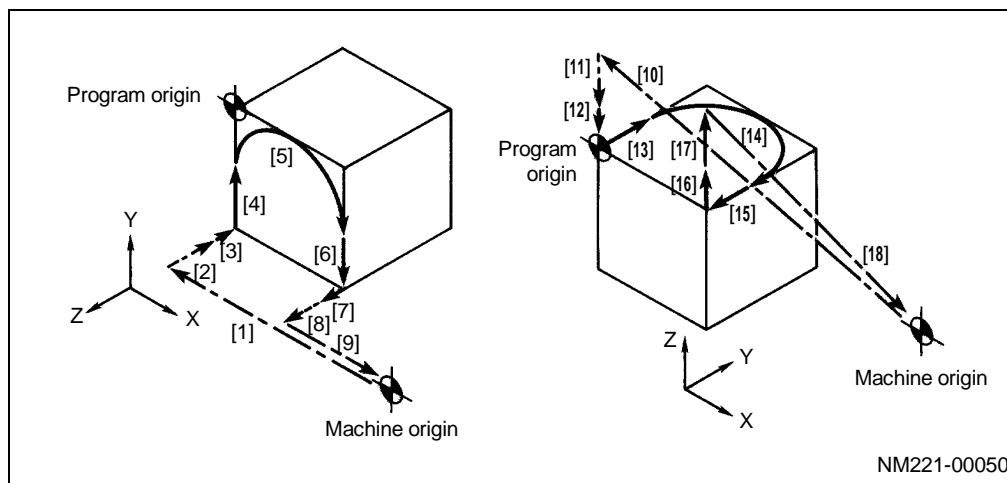
- Setting of command G68 in the absence of a 3D coordinate conversion option results in alarm **942 NO 3-D CONVERSION OPTION**.

- Setting of an illegal G-code during the 3D coordinate conversion mode results in alarm **943 CONVERTING IN 3-D CORDINATES**.

See Paragraph 5. Relationship to other functions, for the listing of illegal G-codes.

3. Sample program

N01	G90G00G40G49G80		G68I1J0K0R-90.	
	G54X0Y-100. [1]	G00X0Y0 [10]
	G43Z50.H01 [2]	G43Z50.H01 [11]
	G01Z-1.F1200 [3]	G01Z-1. [12]
	Y-50. [4]	Y50. [13]
	G02X100.R50. [5]	G02X100.R50. [14]
	G01Y-100. [6]	G01Y0 [15]
	G01Z50. [7]	G01Z50. [16]
	G91G28Z0 [8]	G69	
	G28X0Y0 [9]	G91G28Z0 [17]
N02	G90G00G40G49G80		G28X0Y0 [18]
	G54G00A90.		M30	

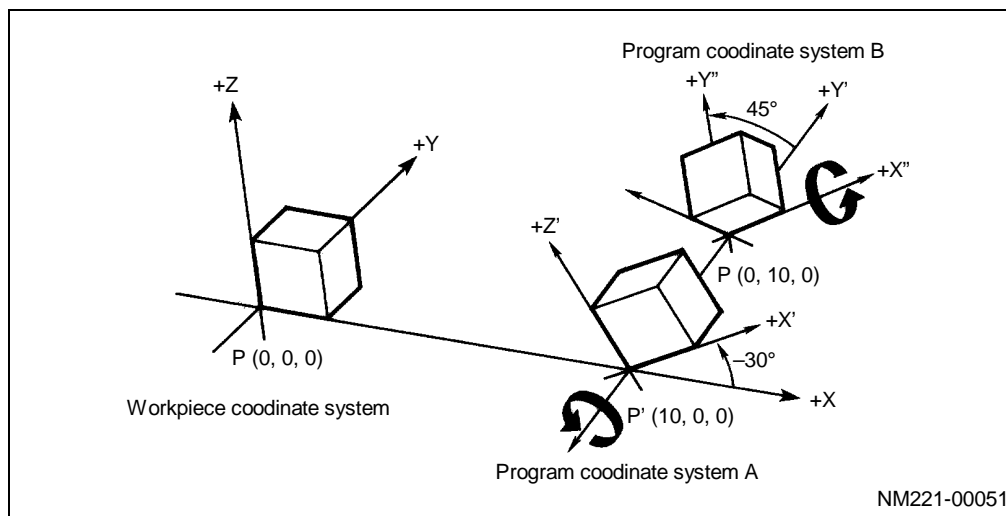


4. 3D coordinate conversion and a program coordinate system

Use of command G68 allows the coordinates of the rotational center in the currently valid coordinate system to be shifted to the designated position (X, Y, Z) and the corresponding graphics to be rotated through the designated angle of rotation (R) in the designated direction of rotational center (I, J, K). Thus, a new program coordinate system can be set.

Example: [1] G90 G54 X0 Y0
 [2] G68 X10.Y0 Z0 I0 J1 K0 R-30.
 [3] G68 X0 Y10.Z0 I1 J0 K0 R45.
 [4] G69

- [1] Set a workpiece coordinate system using workpiece offset command G54.
- [2] The program origin in the workpiece coordinate system that was set during step [1] above shifts to the position of (x, y, z) =(10, 0, 0) and the graphics rotates through 30 degrees in a clockwise direction around the Y-axis. Thus, new program coordinate system A is set.
- [3] The program origin in the workpiece coordinate system that was set during step [2] above shifts to the position of (x, y, z) = (0, 10, 0) and the graphics rotates through 45 degrees in a counter clockwise direction around the X-axis. Thus, new program coordinate system B is set.
- [4] Both the program coordinate systems that were set during steps [2] and [3] above are cancelled and the coordinate system that was set during step [1] above becomes valid once again.



5. Relationship to other functions

1. The table below indicates the compatibility of each G-code with the 3D coordinate conversion function in the two columns on the right.

[1] Is the G-code available in the mode of 3D coordinate conversion?

[2] Is the 3D coordinate conversion mode selectable under the condition of the G-code?

G-code	Group	Function	[1]	[2]
00	01	Positioning	Yes	Yes
01	01	Linear interpolation	Yes	Yes
01.1	01	Threading with C-axis	Yes	No
02	01	Circular interpolation CW	Yes (*1)	No
03	01	Circular interpolation CCW	Yes (*1)	No
02.1	00	Spiral interpolation CW	No	No
03.1	00	Spiral interpolation CCW	No	No
04	00	Dwell	Yes (*4)	
05	00	High-speed machining	No	No
06.1	01	Spline interpolation	No	No
06.2	01	NURBS interpolation	No	No
07	00	Virtual-axis interpolation	No	No
07.1	00	Cylindrical interpolation	Yes	No
09	00	Exact-stop check	Yes	
10	00	Programmed parameter input	Yes (*4)	No
11	00	Programmed parameter input cancel	Yes (*4)	
17	02	Plane selection X-Y	Yes	Yes
18	02	Plane selection Z-X	Yes	Yes
19	02	Plane selection Y-Z	Yes	Yes
20	06	Inch command	No	Yes
21	06	Metric command	No	Yes
22	04	Pre-move stroke check ON	No	No
23	04	Pre-move stroke check OFF	No	Yes
27	00	Reference-point check	No	
28	00	Reference-point return	Yes (*2)	
29	00	Starting-point return	Yes	
30	00	No. 2 through 4 reference-point return	Yes (*2)	
31	00	Skip	Yes	
31.1	00	Multi-step skip 1	Yes	
31.2	00	Multi-step skip 2	Yes	
31.3	00	Multi-step skip 3	Yes	
32	01	Constant lead threading	No	No
33	01	Constant lead threading	No	No
34	01	Variable lead threading	No	No
37	00	Automatic tool-length measurement	No	
38	00	Vector selection for tool radius compensation	No	No
39	00	Corner arc for tool radius compensation	No	No
40	07	Tool radius compensation OFF	Yes	Yes
40.1	15	Shaping cancel	Yes	No
41	07	Tool radius compensation (to the left)	Yes	No
41.1	15	Shapint to the left	Yes	No
42	07	Tool radius compensation (to the right)	Yes	No
42.1	15	Shaping to the right	Yes	No

G-code	Group	Function	[1]	[2]
43	08	Tool-length offset (+)	Yes	No
44	08	Tool-length offset (–)	Yes	No
45	00	Tool-position offset, extension	Yes	No
45.1	19	Attachment compensation	Yes	Yes
46	00	Tool-position offset, reduction	Yes	No
47	00	Tool-position offset, double extension	Yes	No
48	00	Tool-position offset, double reduction	Yes	No
49	08	Tool-length offset cancel	Yes	Yes
49.1	19	Attachment compensation cancel	Yes	Yes
50	11	Scaling cancel	No	Yes
51	11	Scaling on	No	No
50.1	19	G-command mirror image cancel	Yes	Yes
50.2	23	Polygonal machining cancel	Yes	Yes
51.1	19	G-command mirror image on	Yes	No
51.2	23	Polygonal machining	Yes	Yes
52	00	Local coordinate system setting	Yes	Yes
53	00	Machine coordinate system selection	Yes (*5)	
54	12	Workpiece coordinate system 1 selection	No	Yes
54.1	12	Additional workpiece coordinate system selection	No	Yes
55	12	Workpiece coordinate system 2 selection	No	Yes
56	12	Workpiece coordinate system 3 selection	No	Yes
57	12	Workpiece coordinate system 4 selection	No	Yes
58	12	Workpiece coordinate system 5 selection	No	Yes
59	12	Workpiece coordinate system 6 selection	No	Yes
60	00	One-way positioning	Yes	
61	13	Exact-stop check mode	Yes	Yes
61.1	13	Geometry compensation mode	Yes	Yes
61.4	13	Inverse model control	Yes	Yes
62	13	Automatic corner override	Yes	Yes
63	13	Tapping mode	Yes	Yes
64	13	Cutting mode	Yes	Yes
65	00	User macro simple call	Yes	
66	14	User macro modal call A	Yes	No
66.1	14	User macro modal call B	Yes	No
67	14	User macro modal call cancel	Yes	Yes
68	16	Program coordinates rotation	No	No
		3D coordinate conversion	Yes	Yes
69	16	Cancellation of coordinates rotation/3D conversion	Yes (*2)	Yes
71.1	09	Fixed cycle (chamfering cutter 1)	Yes	No
72.1	09	Fixed cycle (chamfering cutter 2)	Yes	No
73	09	Fixed cycle (high-speed deep-hole drilling)	Yes	No
74	09	Fixed cycle (reverse tapping)	Yes	No
75	09	Fixed cycle (boring)	Yes	No
76	09	Fixed cycle (boring)	Yes	No
77	09	Fixed cycle (back facing)	Yes	No
78	09	Fixed cycle (boring)	Yes	No
79	09	Fixed cycle (boring)	Yes	No
80	09	Fixed cycle cancel	Yes	Yes
81	09	Fixed cycle (drill/spot drill)	Yes	No
82	09	Fixed cycle (drill)	Yes	No

G-code	Group	Function	[1]	[2]
82.2	09	Fixed cycle (drill)	Yes	No
83	09	Fixed cycle (deep hole drill)	Yes	No
84	09	Fixed cycle (tapping)	Yes	No
84.2	09	Fixed cycle (synchronous tapping)	Yes	No
84.3	09	Fixed cycle (synchronous reverse tapping)	Yes	No
85	09	Fixed cycle (reaming)	Yes	No
86	09	Fixed cycle (boring)	Yes	No
87	09	Fixed cycle (back boring)	Yes	No
88	09	Fixed cycle (boring)	Yes	No
89	09	Fixed cycle (boring)	Yes	No
90	03	Absolute data input	Yes	Yes
91	03	Incremental data input	Yes	Yes
92	00	Machine coordinate system setting	No	
92.5	00	Workpiece coordinate system rotation	Yes	
93	05	Inverse time feed	Yes	Yes
94	05	Asynchronous feed (feed per minute)	Yes (*4)	Yes
95	05	Synchronous feed (feed per revolution)	Yes (*4)	Yes
98	10	Initial level return in fixed cycle	Yes	Yes
99	10	R-point level return in fixed cycle	Yes	Yes

*1: Setting of helical interpolation results in an alarm.

*2: Only the intermediate point has its coordinates converted.

*3: Setting of program coordinate rotation results in an alarm.

*4: The preparatory function is executed independently of the coordinate conversion.

*5: The operation is always performed in the original coordinate system (free from any conversion).

- Setting of a G-code other than G17, G18, and G19, in the block containing G68 or G69 results in alarm **807 ILLEGAL FORMAT**.
- G28 and G30 commands permit a return to the reference point through the 3D converted intermediate point, whereas a G53 command is performed without any conversion.
- Codes G41 (Radius comp. to the left), G42 (Radius comp. to the right), G51.1 (G-command mirror image on), and Fixed cycle codes must be present with their cancellation codes between G68 and G69 so that a nested relationship is established. Moreover, give a motion command with G90 (absolute data input) in the next block to that of the G68 command.

Example:

```

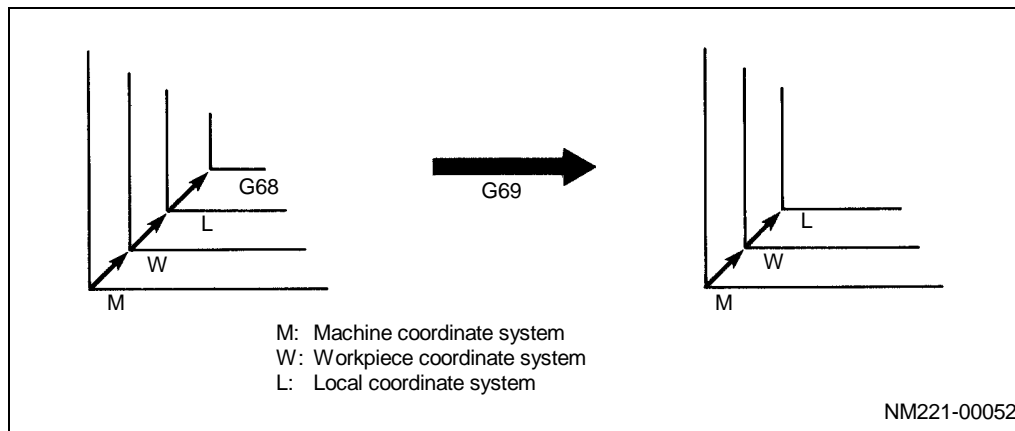
G68X50.Y100.Z150.I1J0K0R-90.
G90G00X0Y0Z0
G41G01X10.F1000
      :
      :
G40
G69

```

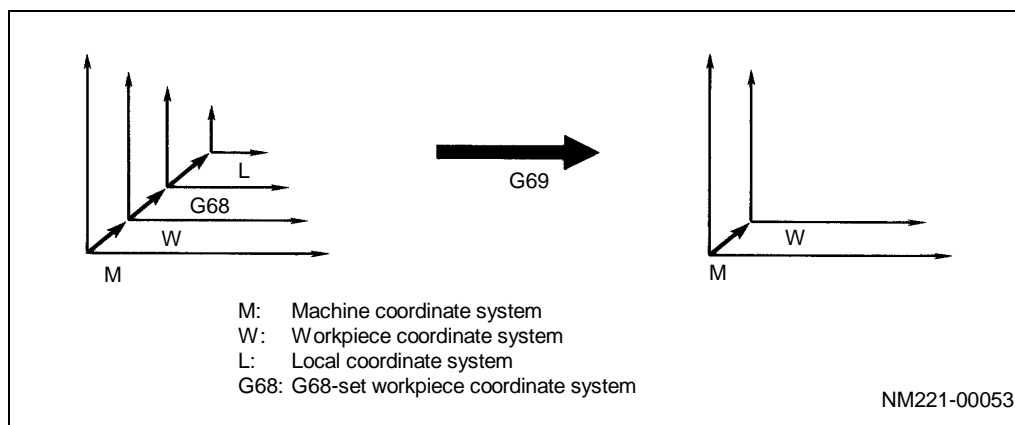
- Tool-length, -radius and -position offset is first processed, and the 3D conversion is conducted on the offset contour.
- Mirror image can only be applied by G51.1 (the related M-codes are not available). The processing order is: mirror image first, then 3D conversion.

7. In mode G68, code G52 is handled as follows:

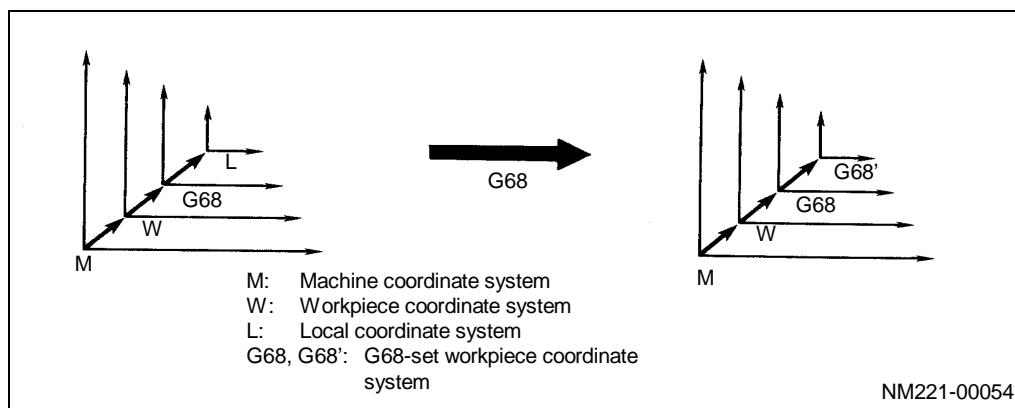
- If code G68 is set in the coordinates that have been set using G52, that local coordinate system will be overridden with a new workpiece coordinate system by G68. Setting of G69 under that status will make the original local coordinate system valid once again.



- If code G52 is present in the data set using mode G68, a local coordinate system can be set on the workpiece coordinate system that has been set using G68. Setting of G69 under that status will cancel the local coordinate system and the G68-set workpiece coordinate system and thus the original coordinate system existing before G68 was set will become valid once again.



- If code G52 is present in the data set using mode G68, a local coordinate system can be set that starts from the workpiece coordinate system to be set using G68. Setting of G68 under that status will cancel the local coordinate system and then set a G68 program coordinate system.



-
8. Setting of G68 during figure rotation results in alarm **850 G68 AND M98 COMMANDS SAME BLOCK**.
 9. Three-dimensional coordinate conversion is not valid for the axis of rotation.
 10. The G68 mode does not accept any external workpiece origin setting commands.
 11. Cancel the 3D coordinate conversion mode temporarily if a MAZATROL program is to be called up as a subprogram.

- NOTE -

15 MEASUREMENT SUPPORT FUNCTIONS

Measurement by an EIA/ISO program is basically the same as that by a MAZATROL program. Information given by a MAZATROL program may be executed by the following preparatory function.

G31: Skip function

15-1 Skip Function: G31

15-1-1 Function description

1. Overview

During linear interpolation by G31, when an external skip signal is inputted, the feed will stop, all remaining commands will be cancelled and then the program will skip to the next block.

2. Programming format

G31 Xx Yy Zz Ff ;

x, y, z : The coordinates of respective axes. These coordinates are designated using absolute or incremental data.

F : Feed rate (mm/min)

This command starts linear interpolation, and when an external skip signal is inputted, the feed will stop, the remaining distance of movement will be cancelled and the next block will be executed.

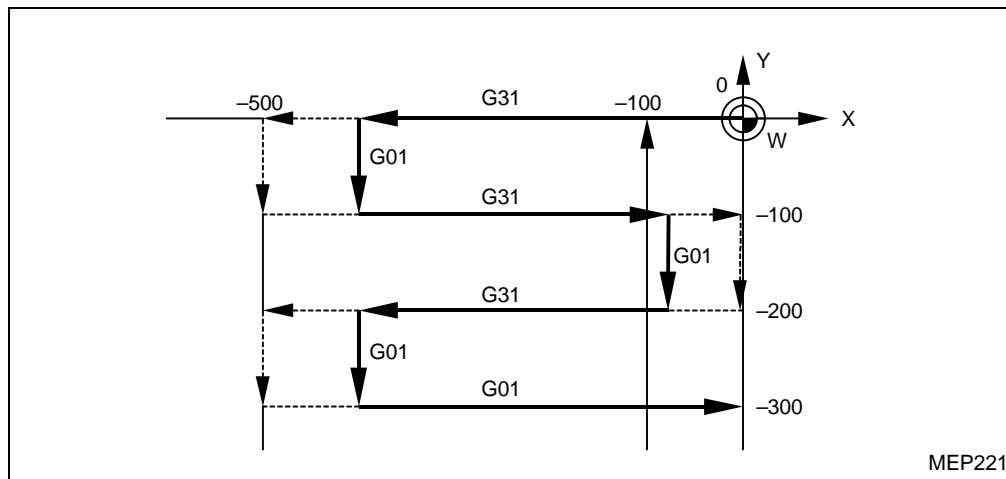
3. Detailed description

1. An asynchronous feed rate commanded previously will be used as feed rate. If an asynchronous feed command is not made previously and if Ff is not commanded, the alarm **SKIP SPEED ZERO** will be caused. F-modal command data will not be updated by the F-command given in the G31 block.
2. Automatic acceleration/deceleration is not applied to command block G31.
3. If feed rate is specified per minute, override, dry run and automatic acceleration/deceleration will not be allowed. They will be effective when feed rate is specified per revolution.
4. Command G31 is unmodal, and thus set it each time.
5. The execution of command G31 will immediately terminate if a skip signal is inputted at the beginning.
Also, if a skip signal is not inputted until the end of command block G31, execution of this command will terminate on completion of execution of move commands.
6. Setting this command code during tool radius compensation results in an alarm.
7. Under a machine lock status the skip signals will be valid.

4. Execution of G31

```

G90 G00 X-100. Y0
      G31 X-500. F100
      G01 Y-100.
      G31 X0
            Y-200.
      G31 X-500.
            Y-300.
      X0
  
```



15-2 Skip Coordinate Reading

The coordinates of the positions where skip signals are input will be stored into system variables #5061 (first axis) through #5076 (sixteenth axis). These coordinates can be called using user macros.

```

:
G90 G00 X-100.
G31 X-200. F60 — Skip command
#101=#5061 ——— Skip signal input coordinate value (in the workpiece coordinates
                  system) is stored into variable #101.
:
  
```

15-3 Amount of Coasting in the Execution of a G31 Block

The amount of coasting of the machine from the time a skip signal is inputted during G31 command to the time the machine stops differs according to the G31-defined feed rate or the F command data contained in G31.

Accurate machine stop with a minimum amount of coasting is possible because of a short time from the beginning of response to a skip signal to the stop with deceleration.

The amount of coasting is calculated as follows:

$$\delta_0 = \frac{F}{60} \times T_p + \frac{F}{60} (t_1 \pm t_2) = \underbrace{\frac{F}{60} \times (T_p + t_1)}_{\delta_1} \pm \underbrace{\frac{F}{60} \times t_2}_{\delta_2}$$

δ_0 : Amount of coasting (mm)

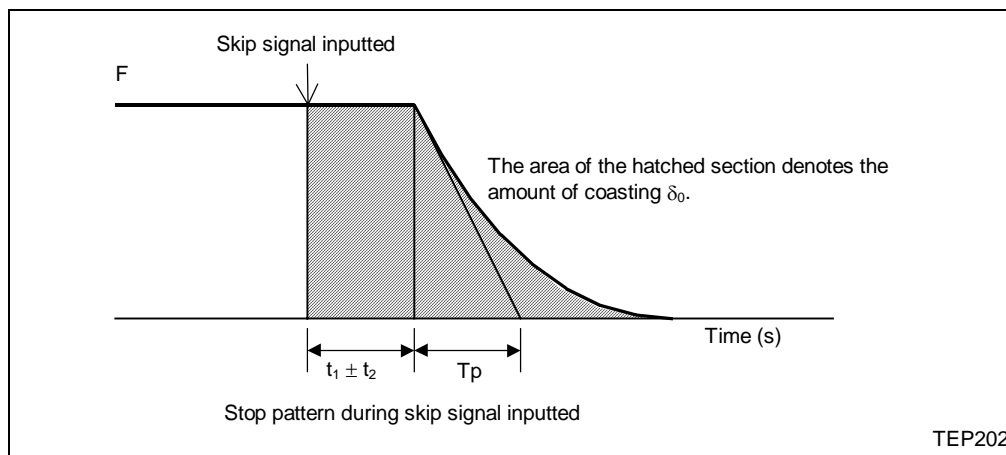
F : G31 skip rate (mm/min)

T_p : Position loop time constant (s) = (Position loop gain)⁻¹

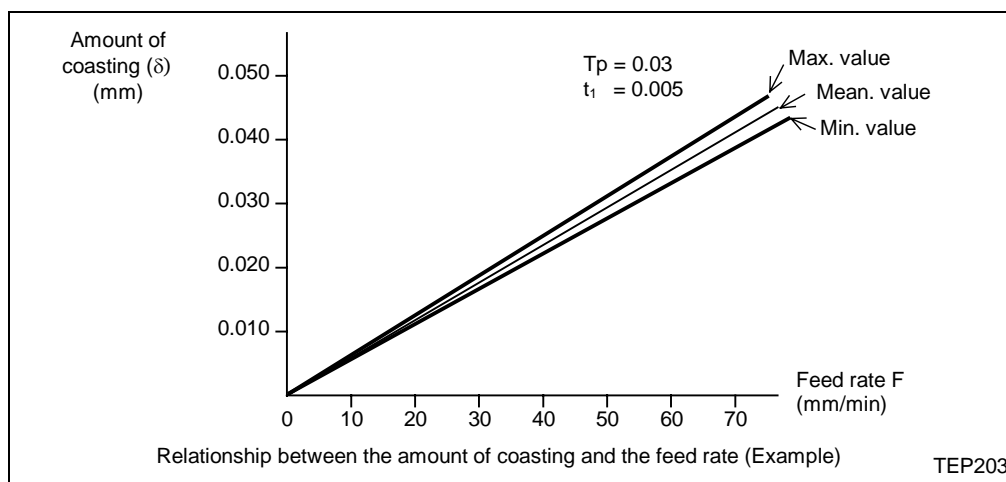
t_1 : Response delay time (s) = (The time from skip signal detection until arrival at NC through PC)

t_2 : Response error time = 0.001 (s)

When using command G31 for measurement purposes, measured data δ_1 can be corrected. Such corrections, however, cannot be performed for δ_2 .



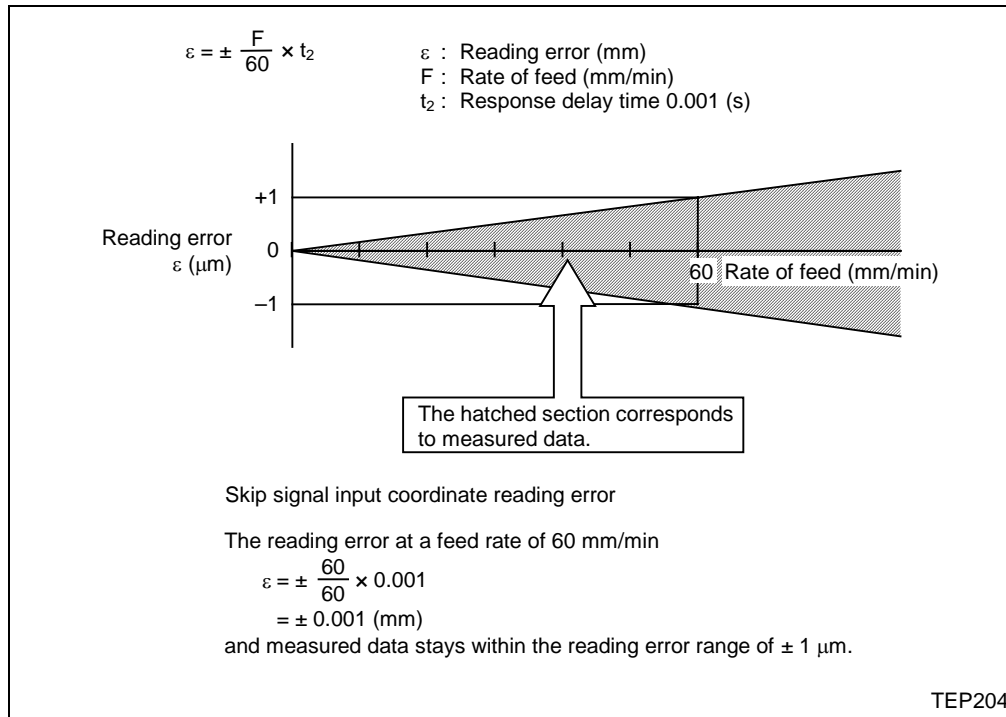
The diagram shown below represents the relationship between the feed rate and the amount of coasting that will be established if T_p is set equal to 30 ms and, t_1 to 5 ms.



15-4 Skip Coordinate Reading Error

1. Reading the skip signal input coordinates

Skip signal input coordinate data does not include the amounts of coasting defined by position loop time constant T_p and cutting feed time constant T_s . Thus skip signal input coordinates can be checked by reading within the error range shown in the diagram below the workpiece coordinates existing when skip signals were inputted. The amount of coasting that is defined by response delay time t_1 , however, must be corrected to prevent a measurement error from occurring.



2. Reading coordinates other than those of skip signal inputs

Coordinate data that has been read includes an amount of coasting. If, therefore, you are to check the coordinate data existing when skip signals were inputted, perform corrections as directed above. If, however, the particular amount of coasting defined by response delay time t_2 cannot be calculated, then a measurement error will occur.

15-5 Multi-Step Skip: G31.1, G31.2, G31.3, G04

1. Function and purpose

Conditional skipping becomes possible by previously setting a combination of skip signals that are to be input. Skipping occurs in the same manner as that done with G31.

The skip function can be designated using commands G31.1, G31.2, G31.3, or G04. The relationship between each of these G commands and the type of skip signal can be set by the parameters **K69** to **K73**.

2. Programming format

G31.1 Xx Yy Zz αα Ff (Same as for G31.2 or G31.3 Ff is not required for G04)

Feed rate (mm/min)

Axis address and target coordinate data

Using this programming format, you can execute linear interpolation in the same manner as that done using command G31.

During linear interpolation, the machine will stop when the previously set skip signal input conditions are satisfied, and then all remaining commands will be cancelled and the next block will be executed.

3. Detailed description

- For feed rates set by the parameter **K42** to **K44**, the following relationship holds:
 G31.1 G31.1 skip feed rate
 G31.2 G31.2 skip feed rate
 G31.3 G31.3 skip feed rate
- The program will skip when the skip signal input conditions appropriate for each of these G commands are satisfied.
- Except for items other than 1 and 2 above, the description of command code G31 also applies.

4. Parameter setting

- The feed rate appropriate for each of the G31.1, G31.2, and G31.3 command codes can be set by the parameters **K42** to **K44**.
- The skip conditions appropriate for each of the G31.1, G31.2, G31.3 and G04 command codes are to be set by parameters **K69** to **K73**. (The skip conditions refer to the logical sum of previously set skip signals).

A parameter setting of 7 is equivalent to G31.

Parameter setting value	Valid skip signals		
	1	2	3
1	○		
2		○	
3	○	○	
4			○
5	○		○
6		○	○
7	○	○	○

5. Machine action

1. Use of the multi-step skip function allows the following type of machine action control, and hence, reduction of the measurement time with improved measurement accuracy.
If parameter settings are as shown below.

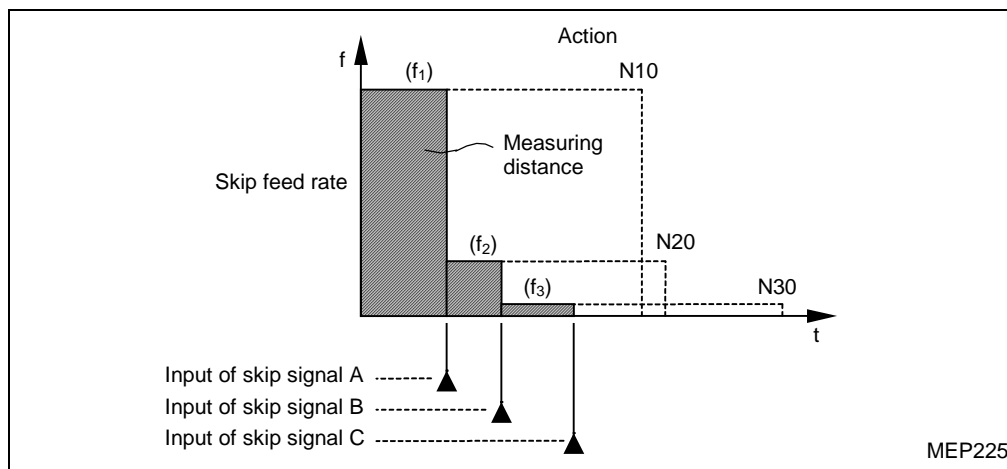
Skip condition	Skip feed rate
G31.1 = 7	20.0 mm/min (f_1)
G31.2 = 3	5.0 mm/min (f_2)
G31.3 = 1	1.0 mm/min (f_3)

Sample program

```

N10 G31.1X200.0
N20 G31.2X40.0
N30 G31.3X1.0

```



Note: During the machine action shown above, if input of skip signal 1 precedes that of skip signal 2, the remaining distance of N20 will be skipped and N30 will also be ignored.

2. If the skip signal corresponding to the conditions previously set is input during dwell (command G04), the remaining time of dwell will be cancelled and the next block will be executed.

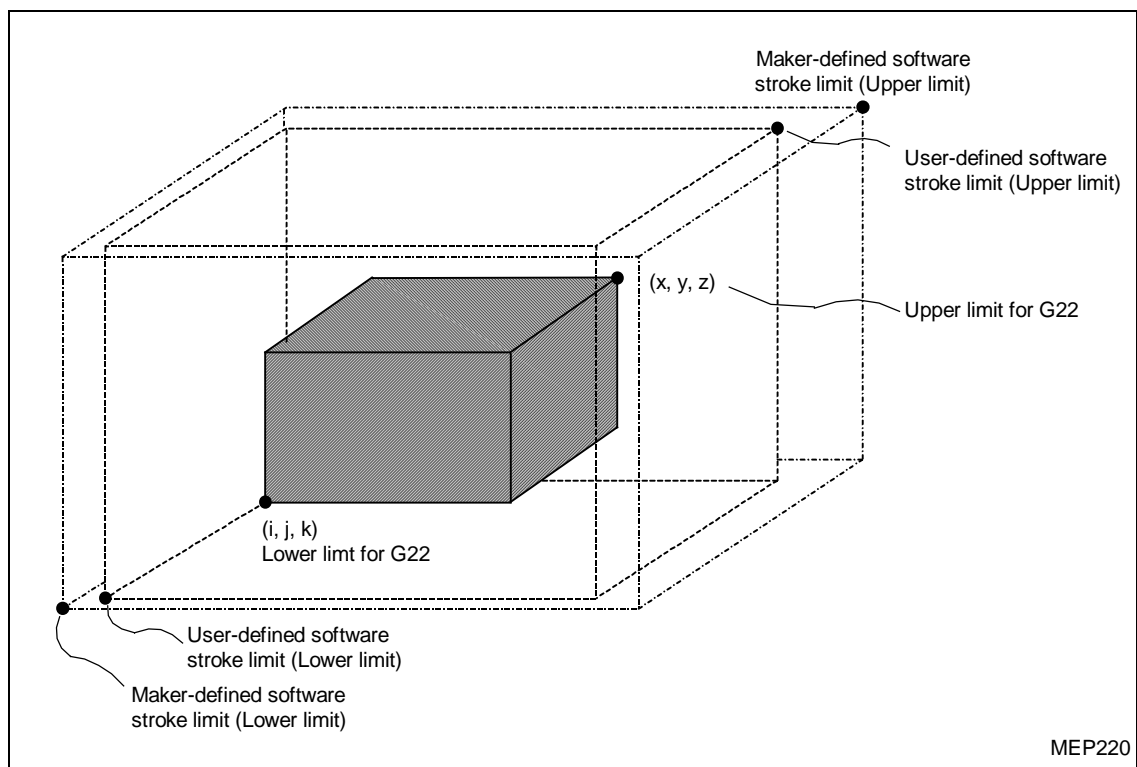
16 PROTECTIVE FUNCTIONS

16-1 Pre-move Stroke Check ON/OFF: G22/G23

1. Function and purpose

While the user-defined software stroke limit check generates an outside machining prohibit area, the pre-move stroke check function sets an inside machining prohibit area (shaded section in the diagram below).

An alarm will occur beforehand for an axis movement command whose execution would cause the tool to touch, or move in or through, the prohibit area.



2. Programming format

G22 $\underbrace{X_Y_Z}_{\text{Lower limit specification}} \underbrace{I_J_K}_{\text{Upper limit specification}}$ (Inside machining prohibit area specification)

G23 (Cancellation)

3. Detailed description

- Both upper-limit and lower-limit values must be specified with machine coordinates.
- Use X, Y, Z to set the upper limit of the prohibit area, and I, J, K to set the lower limit. If the value of X, Y, Z is smaller than that of I, J, K, then the former and the latter will be used as the lower-limit and the upper-limit, respectively.

3. No stroke limit checks will be performed if the upper- and lower-limit values that are assigned to one and the same axis are identical.

G22X200.Y250.Z100.I200.J-200.K0



The X-axis does not undergo the stroke check.

4. Give a G23 command to cancel the pre-move stroke limit check function.
5. A block of G23 X_Y_Z_ will cause the axis motion command X_Y_Z_ to be executed in the current mode of axis movement after cancellation of pre-move stroke limit check.

Note: Before setting G22, move the tool to a position outside the prohibit area.

17 THREADING: G33 (OPTION)

17-1 Equal-Lead Threading

1. Function and purpose

Setting a G33 command in the program enables equal-lead threading under tool feed control synchronized with spindle rotation.

Also, multiple-thread screws can be machined by specifying the starting angle of threading. A d'ANDREA tool is required for fully automatic threading.

2. Programming format

A. Standard lead threading

G33 Zz Ff Qq

z: Threading direction axis address and the thread length

f: Lead in the direction of the long axis (the axis to be moved through the longest distance among all axes)

q: Threading start shift angle (0 to 360 degrees)

(The starting angle of threading becomes 0 degrees if omitted.)

B. Precise lead threading

G33 Zz Ee Qq

z: Threading direction axis address and the thread length

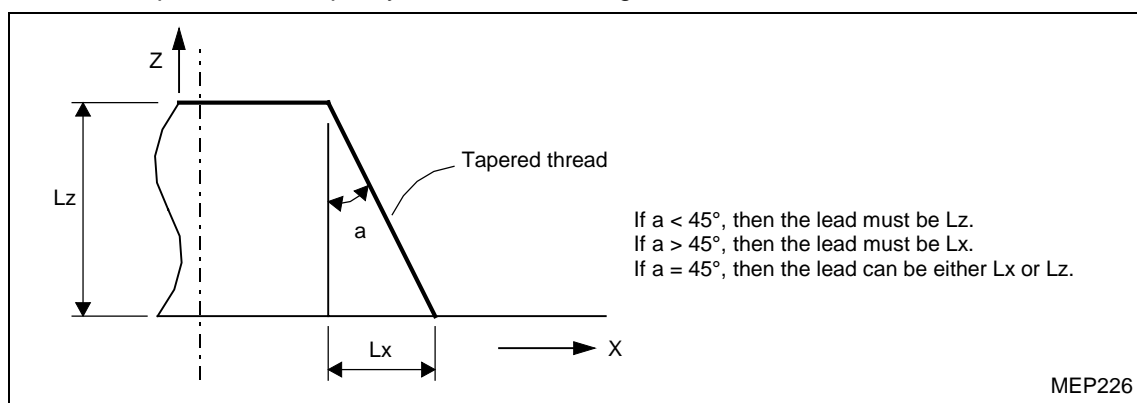
e: Lead in the direction of the long axis (the axis to be moved through the longest distance among all axes)

q: Threading start shift angle (0 to 360 degrees)

(The starting angle of threading becomes 0 degrees if omitted.)

3. Detailed description

- For a tapered screw, specify the lead in the long-axis direction.



The setting ranges of leads F and E are as follows:

Input unit	F-command lead data range (6-digit)	E-command lead data range (8-digit)
Metric	0.001 to 99.9999 mm/rev	0.00002 to 99.999999 mm/rev
Inch	0.0001 to 9.99999 in/rev	0.000002 to 9.9999999 in/rev

Note: Alarm **134 SPINDLE ROTATION EXCEEDED** will be displayed if the feed rate, after being converted into a feed rate per minute, is in excess of the maximum cutting feed rate.

2. The E-command data will also be used as number-of-threads command data for threading in inches. Whether the command data is to be used only as precise lead data or number-of-threads data can be selected using bit 7 of parameter **F91**.
3. Keep the spindle speed constant during the entire machining cycle from rough cutting to finish cutting.
4. During threading, feed hold does not work. Pressing the feed hold button during threading brings the program to a block stop at the end of the block which immediately succeeds the block under the control of the G33 mode (that is, the block at which the threading operation has ended).
5. For tapered screws, since machining cannot be stopped in the middle of threading, the cutting feed rate after being converted may exceed the maximum cutting feed according to the particular command data.
To prevent this from occurring, therefore, the command lead data must be set according to the maximum feed rate obtained after conversion, not for the starting point of threading.
6. Usually, the leads at the beginning of threading and at the end of threading become incorrect because of a delay in the operation of the servo system. A thread length must therefore be specified that has a probable, incorrect lead length added to the required thread length.
7. The spindle speed undergoes the following restriction:

$$1 \leq R \leq \frac{\text{Maximum feed rate}}{\text{Thread lead}}$$

where R must be smaller than or equal to the maximum allowable encoder revolutions (min^{-1}), and

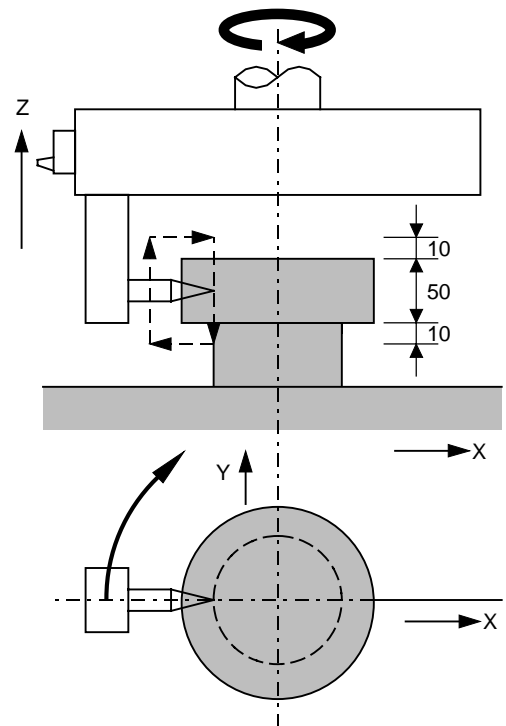
R:	Spindle speed (min^{-1})
Thread lead:	mm or inches
Maximum feed rate:	mm/min or inches/min

8. The threading start shift angle must be specified using an integer from 0 to 360.
9. The cutting feed override value is fixed at 100%.

4. Sample program

```

N110 G90G0X-200.Y-200.S50M3
N111 Z110.
N112 G33Z40.F6.0
N113 M19
N114 G0X-210.
N115 Z110.M0
N116 X-200.
      M3
N117 G04X5.0:
N118 G33Z40.
    
```



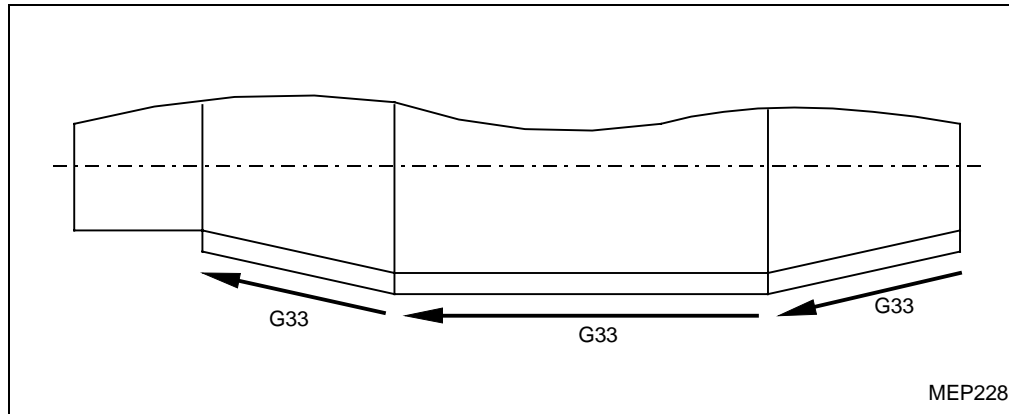
MEP227

<Operation description>

- | | |
|------------|--|
| N110, N111 | The spindle center is positioned at the workpiece center and the spindle rotates forward. |
| N112 | The first threading operation is performed.
Tread lead = 6.0 mm |
| N113 | Spindle orientation based on an M19 command is performed. |
| N114 | The tool moves away in the X-axis direction. |
| N115 | The tool returns to a position above the workpiece, and an M00 command stops the program. Adjust the tool as required. |
| N116 | Preparations for the second threading operation are made. |
| N117 | Set a dwell time, as required, to stabilize the spindle speed. |
| N118 | The second threading operation is performed. |

17-2 Continuous Threading

Continuous threading becomes possible by setting threading command codes in succession in the program. Thus, special threads that change in lead and/or shape during threading can be machined. A D'ANDREA tool is required for continuous threading.



17-3 Inch Threading

1. Function and purpose

Including in a G33-command format the number of threads per inch in the long-axis direction enables tool feed to be controlled in synchronization with spindle rotation, and thus enables equal-lead straight threading and tapered threading.

2. Programming format

G33 Zz Ee Qq

- z: Threading direction axis address and the thread length
- e: Number of threads per inch in the direction of the long axis (the axis to be moved through the longest distance among all axes)
(A decimal point can be included.)
- q: Threading start shift angle (0 to 360 degrees)

3. Detailed description

1. The number of threads per inch must be that existing in the long-axis direction.
2. E-command data will also be used as command data for precise lead threading. Whether the command data is to be used only as number-of-threads command data or precise lead command data can be selected using bit 7 of parameter **F91**.
3. E-command data value must be in the setting range for the command.

4. Sample program

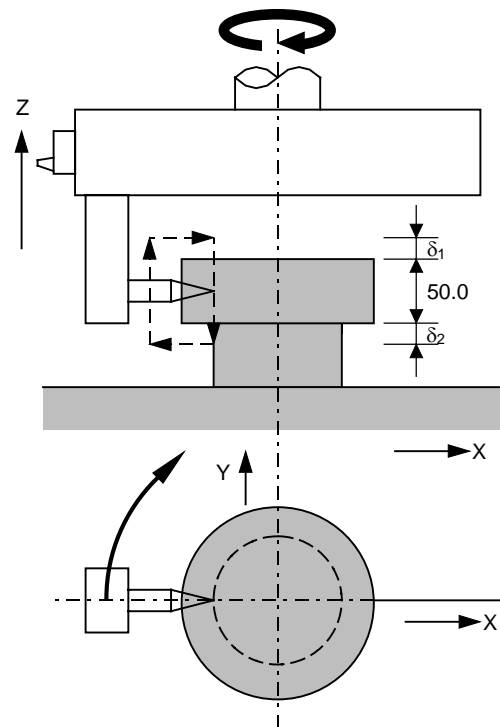
Thread lead3 thread leads/inch (= 3.46666...)

$\delta_1 = 10 \text{ mm}$ $\delta_2 = 10 \text{ mm}$

If programmed in millimeters:

```

N210 G90G0X-200.Y-200.S50M3
N211 Z110.
N212 G91G33Z-70.E3.0      (First threading)
N213 M19
N214 G90G0X-210.
N215 Z110.M0
N216 X-200.
      M3
N217 G04X2.0
N218 G91G33Z-70.          (Second threading)
    
```



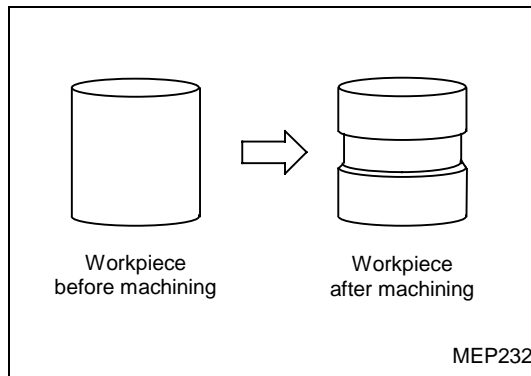
MEP227

- NOTE -

18 DYNAMIC OFFSETTING: M173, M174 (OPTION)

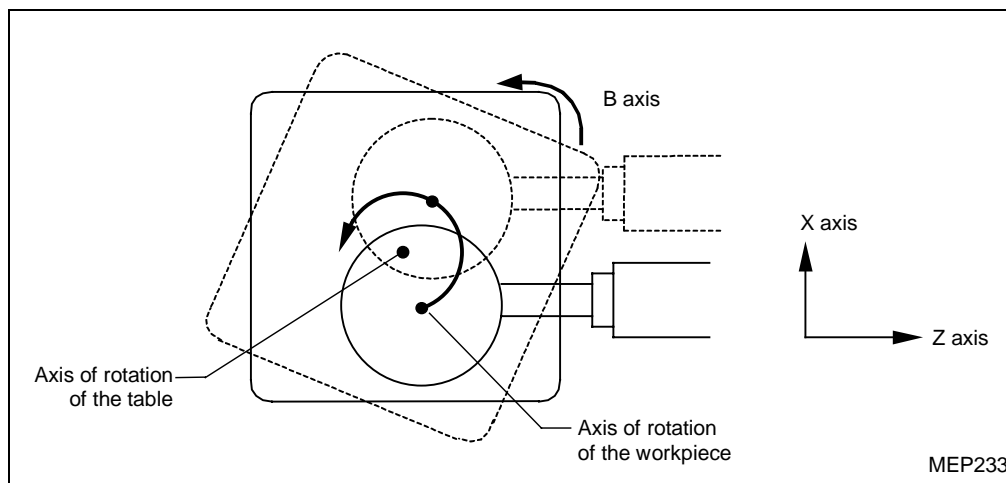
1. Function and purpose

Machining with rotation of the rotary table (B-axis) requires in principle the axis of rotation of the workpiece to be completely aligned with the axis of rotation of the table.



In practice, however, this is very difficult to implement for the reasons of fixture design, unless a very precise fixture is used.

Dynamic offsetting is a function that internally compensates continuous deviation due to the misalignment in question. As a result, machining program can easily be prepared on the assumption that the alignment is ideal.



M173 Dynamic offsetting ON
 G01 B360. F500
 M174 Dynamic offsetting OFF

2. Detailed description

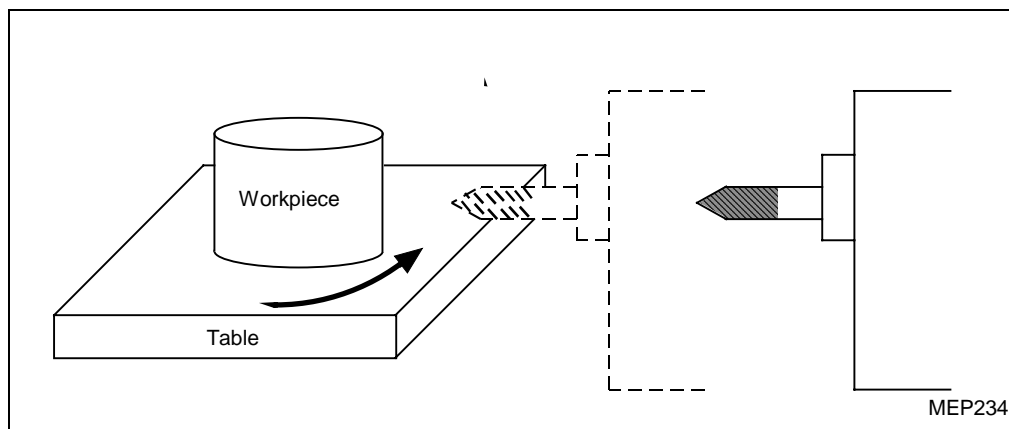
- Automatic limitation by the software does not occur even if dynamic offsetting may cause the stroke limit to be overstepped.
- Reduce to 3 mm or less the eccentricity of an actually mounted workpiece from the axis of rotation of the table; otherwise the alarm **137 DYNAMIC COMPENSATION EXCEEDED** is caused.

Automatic operation is executed on the assumption that the origin of the workpiece coordinates lies on the axis of rotation of the workpiece. Manual operation uses the data in parameter **I11** (as described later).

- The workpiece origin must be set on the axis of rotation of a workpiece when it is to be machined using dynamic offsetting.
- Dynamic offsetting is not effective in the 3-dimensional coordinates conversion mode (G68).
- The related parameters are as follows:

Address	Name	Setting	Description
S5	Axis of table rotation	Unit: 0.001 mm	Set the machine coordinates of the axis of the table rotation for the controlled axes concerned.
I11	Axis of work-piece rotation	Range: ±99999999	Set the machine coordinates of the axis of the workpiece rotation, existing at a table angle of 0 degrees, for the controlled axes concerned. (This parameter is valid only for manual operation.)

- Dynamic offsetting is provided for the type of machining that can in principle be achieved by turning the workpiece only with the tool in a fixed position.



3. Sample program

```

G55 .....The workpiece axis is assumed to go through the origin of the G55 system.
G0X_Y_Z_ .....Approach
M173 .....Dynamic offsetting ON
G1 Z_F_      Start of cutting
  B_F_ .....B-axis rotation
  Z_F_ .....Relief on the Z-axis
M174 .....Dynamic offsetting OFF
M30 .....End of machining

```

19 HIGH-SPEED SMOOTHING CONTROL FUNCTION (OPTION)

The high-speed smoothing control function allows rapid and highly accurate execution of an EIA/ISO program which is prepared to approximate free-curved surfaces with very small lines. Compared with the conventional high-speed machining mode, this function allows machining almost free of cut-surface defects and stripes.

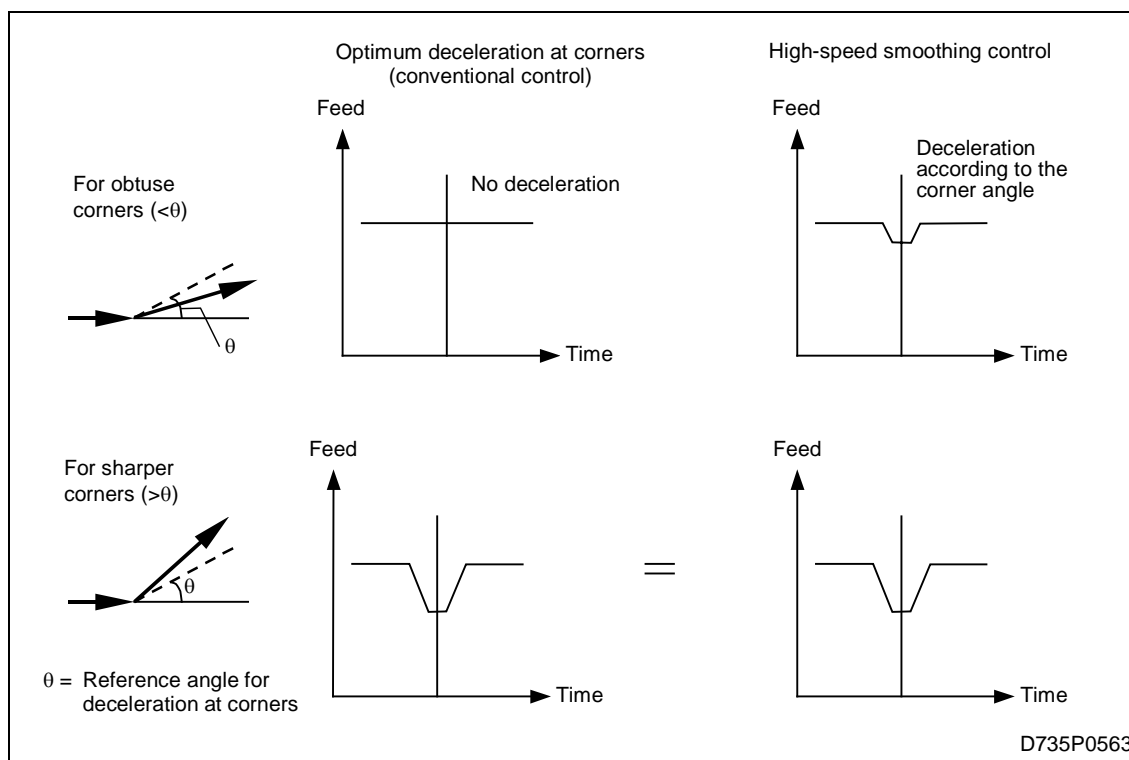
The application of speed control based on a block-to-block angle, such as corner deceleration, in the conventional high-speed machining mode may cause acceleration and deceleration to be repeated in too formal a response to minute steps or to errors. As a result, scratch-like traces or stripes may be left on a cut surface.

By judging the machining shape or contour from the specified continuous lines as well as from the angle between two blocks, the high-speed smoothing control conducts the optimum speed control not affected too significantly by very slight steps or undulations. Consequently, a cut surface with less scratch-like traces or stripes can be obtained.

Some of the outstanding features of high-speed smoothing control are listed below.

- Effective for machining a die of a smooth shape using a microsegment program.
- Speed control is insusceptible to the effects of any errors contained in the tool path.
- If adjacent paths are similar in terms of geometrical accuracy, acceleration and deceleration patterns will also be similar.
- Even at the sections where corner deceleration is not necessary with respect to the angle, speed will be clamped if the estimated acceleration is great.

This function is valid during high-speed mode with the geometry compensation being selected.



19-1 Programming Format

```
G61.1 (G61.2);
```

```
G5P2; .....High-speed machining mode (with smoothing control) ON
```

- Either 0 or 2 is to be set with address P (P0 or P2).
- No other addresses than P and N must be set in the same block with G05.
- Use the following parameter to make the high-speed smoothing control valid.
 - F3** bit 0 = 1: High-speed smoothing control valid
 - 0: High-speed smoothing control invalid (Only high-speed machining valid)
- Give G61.1 (Shape correction ON) or G61.2 (Modal spline interpolation) before G05P2 to use the high-speed smoothing control.

19-2 Commands Available in the High-Speed Smoothing Control Mode

As is the case with the high-speed machining mode feature (as described in Chapter 22), only axis motion commands with the corresponding preparatory functions (G-codes) and feed functions (F-codes), designation of sequence number, codes for Control Out and Control In (parentheses), as well as M/S/T functions are available in the high-speed smoothing control. Setting data of any other type will result in an alarm (**807 ILLEGAL FORMAT**).

1. G-codes

The available preparatory functions are G00, G01, G02, G03, G17, G18, G19, G93 and G94.

The circular interpolation (helical interpolation unavailable) can be programmed with R (radius designation) as well as with I and J (center designation), and executed always (independently of the setting in bit 2 of the **F96** parameter) with the control for a uniform feed. Do not give, however, any command for helical or spiral interpolation; otherwise an alarm occurs (**807 ILLEGAL FORMAT**).

Moreover, the type of feed function can be selected even in the middle of the high-speed smoothing control mode between G93 (Inverse time feed) and G94 (Asynchronous feed). However, synchronous feed (Feed per revolution; G95) is not available.

With the exception of group 1, the modal G-functions will be saved during, and restored upon cancellation of, the high-speed smoothing control mode.

2. Axis motion commands

The three linear axes (X, Y, Z) can be specified. Absolute data input as well as incremental data input is applicable, indeed, but the former input mode requires the validation of bit 5 of the **F84** parameter.

F84 bit 5 : Type of position data input in the high-speed machining mode:

- 1: The input mode (G90/G91) before selection of the high-speed machining mode
- 0: Always incremental data input

3. Feed functions

The rate of feed can be specified with address F.

4. Sequence number

Sequence number can be specified with address N. This number, however, is skipped as a meaningless code during reading.

5. Control Out and Control In

Use round brackets "(" and ")" as required to insert a comment. See Section 3-1 for more information.

It should be noted here, however, that useless deceleration may be caused by a move-free block. Move-free are, for instance, an empty block (null contents), one beginning and ending with "(" and ")", a G91-block with a moving distance of zero (0) as well as a G90-block for the very same position.

6. M/S/T functions

Codes (M, S, and T) of miscellaneous, spindle, and tool functions can be used in general as required. Use of these functions, however, may cause useless deceleration.

Use of the following functions in particular can only cause an alarm (**807 ILLEGAL FORMAT**):

- No. 2 miscellaneous functions (8-digit A-, B- or C-codes).
- Special M-codes for interrupting by, and calling for, a macroprogram (M96, M97, M98, M99).

19-3 Additional Functions in the High-Speed Smoothing Control Mode

During high-speed smoothing control, fairing, although basically not required, can be made valid, as with normal high-speed machining mode.

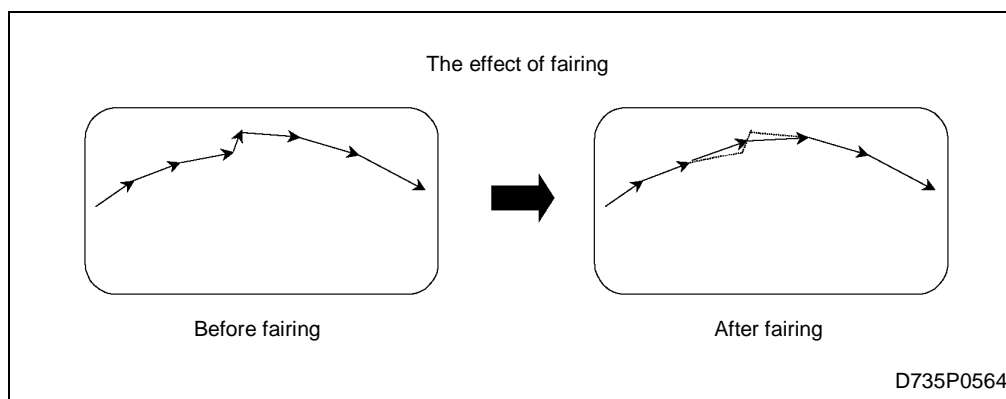
1. Fairing function

If, in a series of linear paths, a protruding section exists in the CAM-created microsegment machining program, this protruding path can be removed and the preceding and following paths connected smoothly by setting parameter **F96** bit 1 to "1".

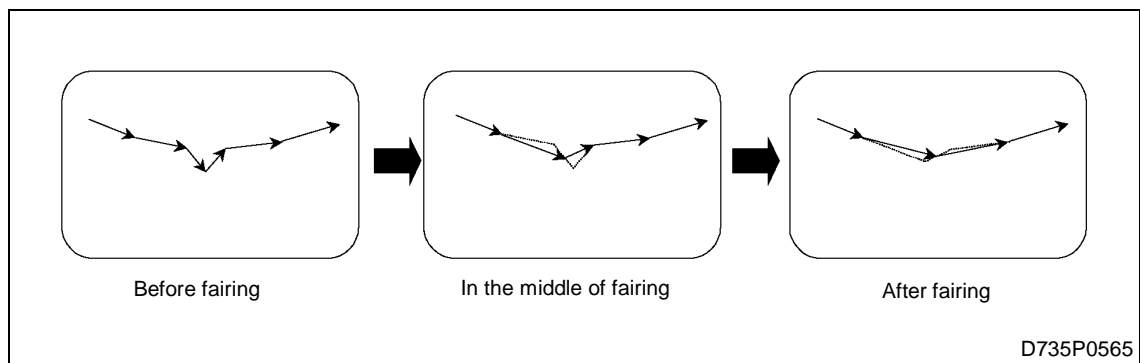
F96 bit 1: Fairing function for the microsegment machining program

- 1: Fairing for a protruding path
- 0: No fairing

F103: Maximum length of a block to be removed for fairing



Fairing is also valid for a succession of protruding paths as shown below:



19-4 Related Parameters

The parameter settings related to this function are as follows:

F3 = 0 : High-speed smoothing control invalid

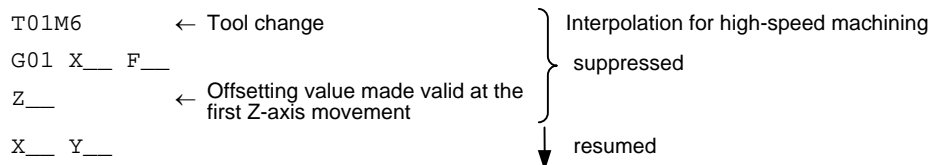
1 : High-speed smoothing control valid (No deceleration at very slightly stepped sections)

3 : High-speed smoothing control valid (Deceleration at all stepped sections)

Under the default setting (1), deceleration does not occur at very slight steps of 5 microns or less since such steps are processed as errors in the programmed path. For a machining program which requires all the described contours in it to be respected as they are, set this parameter to 3 to obtain precise feed control for the as-programmed shape.

19-5 Restrictions

1. The modal functions for tool radius compensation, mirror image, scaling, coordinate system rotation, virtual axis interpolation, three-dimensional radius compensation and shaping function should have been cancelled beforehand to give a G05 P2 command. Otherwise, an alarm may be caused or the modal function unexpectedly cancelled.
2. The high-speed smoothing control mode should be selected and cancelled with the tool sufficiently cleared from the workpiece since the selection and cancellation always cause a deceleration of feed motions.
3. Fairing function cannot be executed in the mode of single-block operation.
4. The high-speed smoothing control is not valid for rotational axes.
5. Fairing function cannot be executed in the mode of tool tip point control, tool radius compensation for five-axis machining, workpiece setup error correction, or inclined-plane machining.
6. In the mode of high-speed machining a block of M/S/T function causes the flow of processing for fairing to be interrupted temporarily.
7. In the mode of high-speed machining a block of T**M6 for tool change causes the execution speed and the fairing function to be lowered and suppressed, respectively, until a length-offsetting value is made valid for the new tool, which does not occur implicitly if the length values of the MAZATROL tool data are not to be used in executing EIA/ISO programs (**F93** bit 3 = 0). In this case, therefore, it is necessary to cancel the high-speed machining mode temporarily so as to give explicitly the required G43-command for tool length offset.



19-6 Related Alarms

The alarms related to this function are as follows:

Alarm No.	Alarm message	Cause	Remedy
169	ILLEGAL OPER. HIGH SMOOTHING CTR	In the mode of high-speed smoothing control an unavailable operation (e. g. manual interruption) was attempted.	Manual interruption cannot be performed in the mode of high-speed smoothing control.
807	ILLEGAL FORMAT	An unavailable command code is given in the G5P2 mode.	Check the machining program and make corrections as required.
809	ILLEGAL NUMBER INPUT	The number of digits of the entered numerical data is too large.	Check the machining program and make corrections as required.

- NOTE -

20 FUNCTION FOR SELECTING THE CUTTING CONDITIONS


1. Function and purpose

The workpiece can be machined under the desired cutting conditions by specifying one of ten accuracy levels (from **FAST** to **ACCURATE**). The accuracy level is to be specified either by an M-code in the machining program or from the **CUTTING LEVEL SELECT** window.

2. Selecting an accuracy level

A. Use of the M-codes

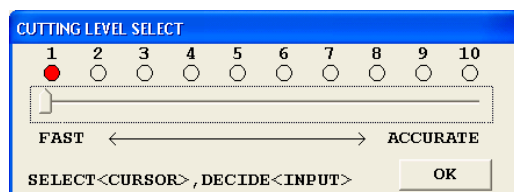
Use the following miscellaneous functions (M-codes) to select the desired one from among the 10 accuracy levels (Level 1 for the highest speed and Level 10 for the highest accuracy):

M-code		Highest speed
M821	Accuracy level 1	
M822	Accuracy level 2	
M823	Accuracy level 3	
M824	Accuracy level 4	
M825	Accuracy level 5	
M826	Accuracy level 6	
M827	Accuracy level 7	
M828	Accuracy level 8	
M829	Accuracy level 9	
M830	Accuracy level 10	Highest accuracy

B. Programming example

```
G00G40G80G90G94G98
G91G00G28Z0.
G28X0.Y0.
T1T2M6
G00G90G54X182.15Y20.974S180M3
G43H1Z100.M8
Z5.
M825 ← Selection of Accuracy level 5.
G01Z-9.F400.
G03X170.15Y0.189R24.F180.
G01Y-0.189
G02X152.793Y-20.144R20.15
G01X152.186Y-20.229
X151.573Y-20.315
X150.96Y-20.4
:
:
```

Remark: See the corresponding section in the Operating Manual for the details of the **CUTTING LEVEL SELECT** window.



Note: Since accuracy level adjustment is a special function for die machining, this function can be used only for machines capable of utilizing the intended purpose of the function. The **CUTTING LEVEL SELECT** window is not displayed for other machines.

21 TORNADO TAPPING (G130)

1. Function and purpose

Tornado tapping cycle is provided to machine a tapped hole by one axial cutting motion with the aid of a special tool. While usual tapping cycles require multiple tools to be used in sequence, use of this cycle function spares tool change time as well as repetitive cutting motion in order to enhance the machining efficiency.

This cycle function is only available on machines equipped with the Y-axis control facility.

Note: Tornado tapping function requires the following parameter settings for macro-call G-codes:

J37 = 100009401 (Fixed value for the number of the macroprogram to be called for tornado tapping)

J38 = 130 (Fixed value for the number of the G-code to be used for macro call)

J39 = 2 (Fixed value for the type of macro call)

2. Programming format

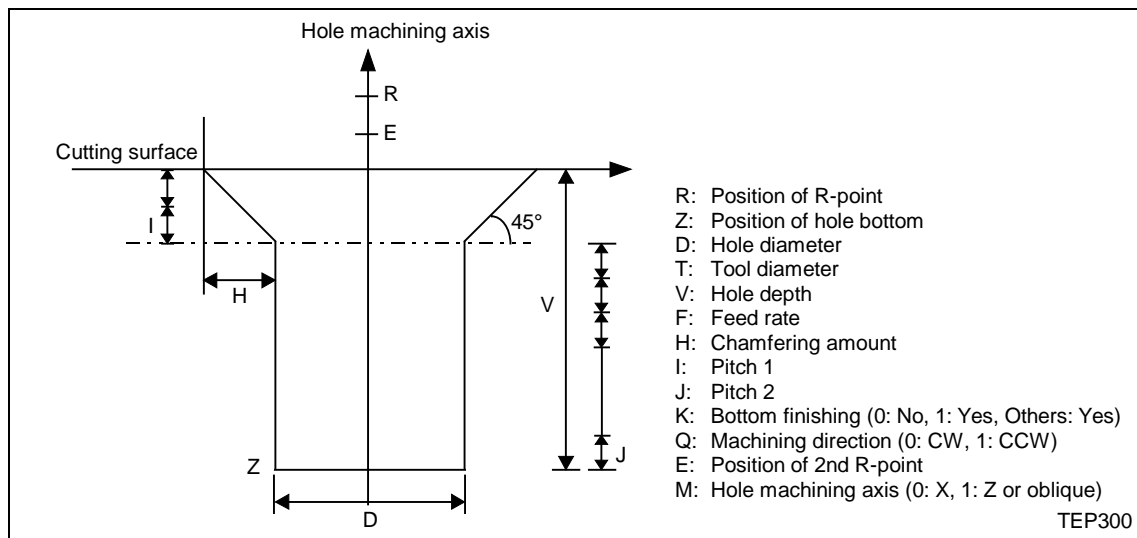
The following format refers to hole machining on the face [or O. D. surface].

G17 [or G19];

G130 R_Z_D_T_V_F_H_I_J_K_Q_E_M1 [or M0];

X [or Z] _Y_ ; (Setting of hole position)

G67 ;



- The chamfering angle is fixed at 45°.
- Arguments D (hole diameter) and T (tool diameter) must satisfy the following condition:

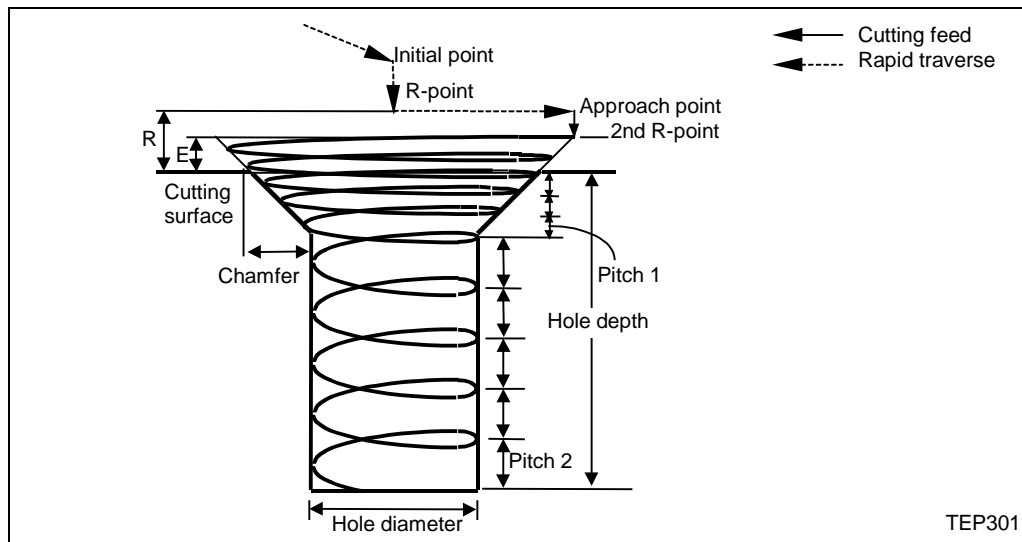
$$D \geq T \geq D/2.$$
- Argument K is used to select whether finishing is to be (K1) or not to be (K0) executed on the bottom of the hole.
- Set the hole position separately from the macro-call G-code (G130).
- As is the case with usual fixed cycles, actual machining with axial movement can only be executed for a block containing the hole position data.
- Do not fail to set the code G67 as required to cancel the modal call.

3. Description of movement

A. Hole machining

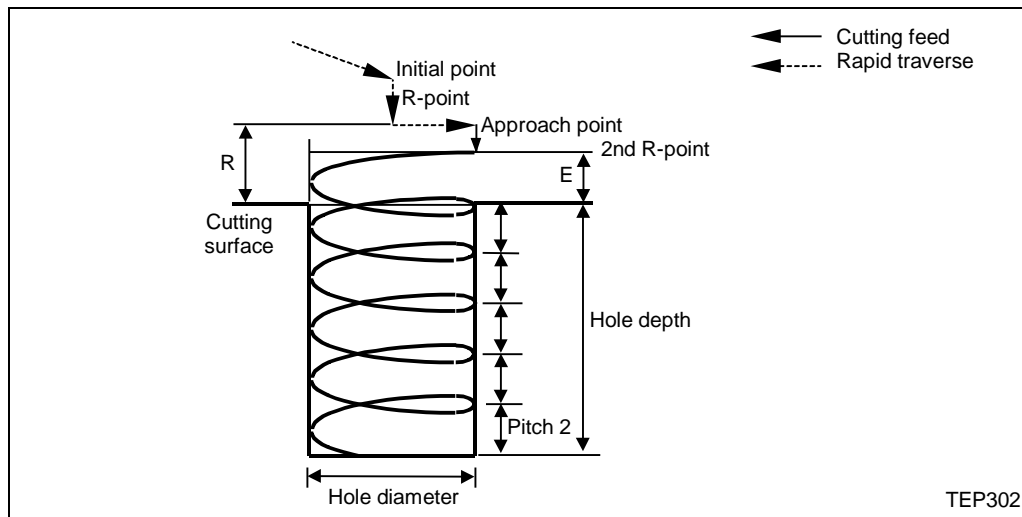
1. With chamfering

After moving from the current position to the R-point on the hole axis and then approaching to a point on the 2nd R-point level, chamfering is performed by a spiral-helical interpolation first, and then cylindrical machining is carried out to the bottom by a circular-helical interpolation.



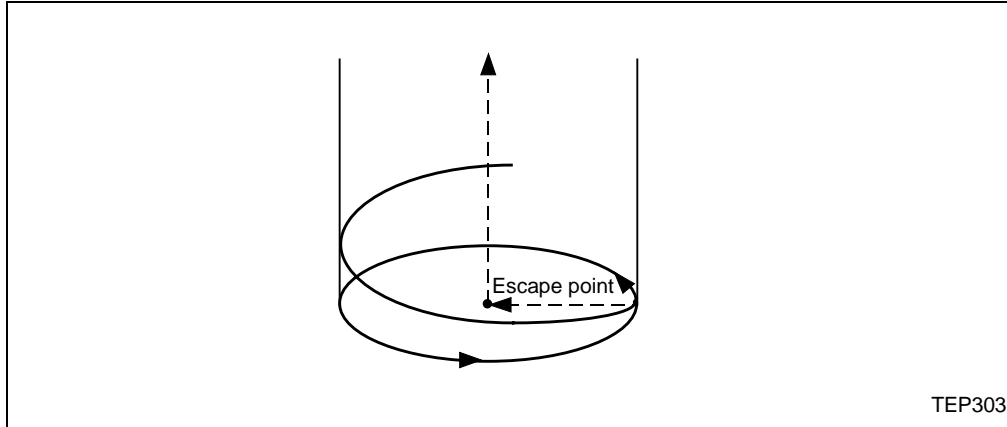
2. Without chamfering

After moving from the current position to the R-point on the hole axis and then approaching through the hole radius and to a point on the 2nd R-point level, cylindrical machining is carried out from the top to the bottom by a circular-helical interpolation.

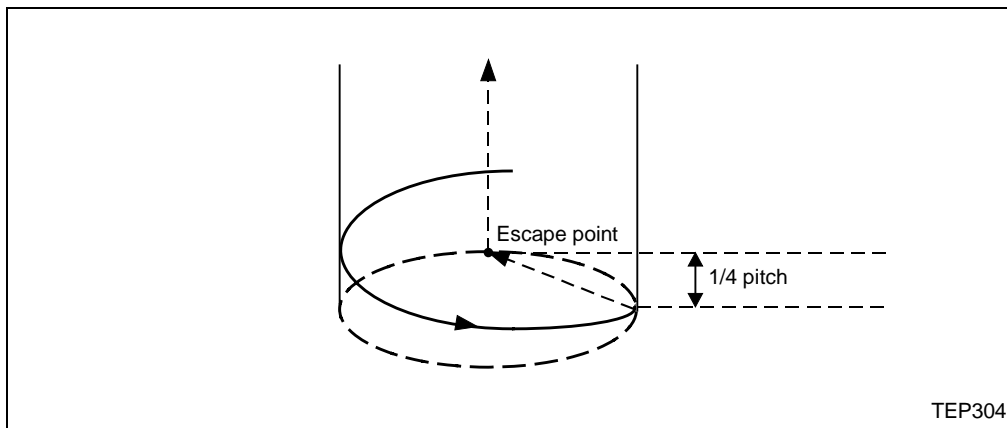


B. Movement on the bottom**1. With bottom finishing**

After cutting down to the bottom of the hole by helical interpolation, the tool performs a circular interpolation for full circle, and then escapes radially to the axis of the hole before returning in the axial direction to the initial point or R-point at the rapid traverse.

**2. Without bottom finishing**

After cutting down to the bottom of the hole by helical interpolation, the tool escapes radially to the axis of the hole while axially returning through quarter the pitch, and then returns in the axial direction to the initial point or R-point at the rapid traverse.



- NOTE -

22 HIGH-SPEED MACHINING MODE FEATURE (OPTION)

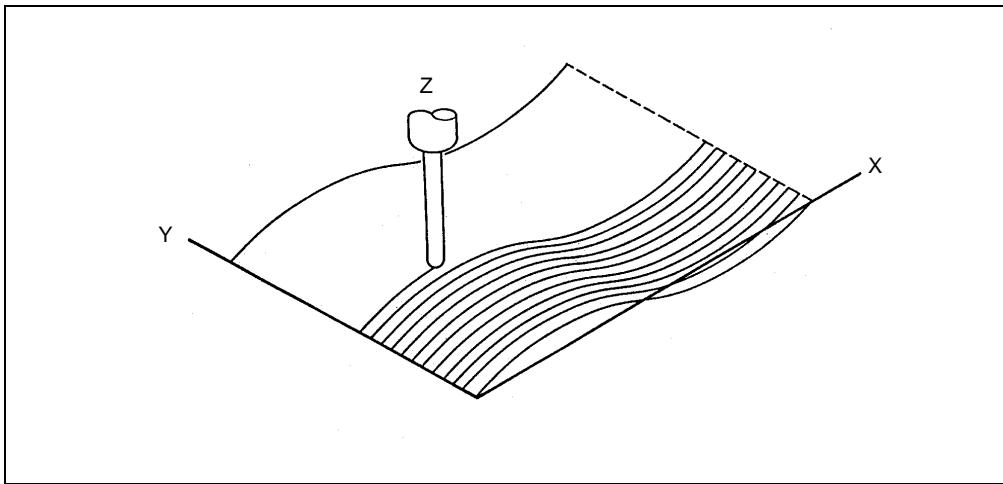
The high-speed machining mode feature allows high-speed execution of programs used for the machining of free-curved surfaces that have been approximated using very small lines.

In high-speed machining mode, microsegment machining capabilities improve by several times, compared with conventional capabilities. This allows the same machining program to be executed at several times the original feed rate, and thus the machining time to be reduced significantly.

Conversely, a machining program that has been approximated using lines of several fractions of the original segment length, can also be executed at the same feed rate, so more accurate machining is possible.

Combined use of the high-speed machining mode and the shape correction function allows more accurate machining to be implemented.

If, moreover, a protruding section exists in the microsegment machining program, smooth interpolation can be conducted automatically by removing this illegal path.



High-speed machining is available in the automatic operation modes: Memory, HD (Hard Disk), IC card and Ethernet.

Even in the high-speed machining mode can be applied various operational functions: override functions, cutting feed rate limit function, single-block operation function, dry run function, graphic trace function and geometry compensation function.

The microsegment machining capability in the high-speed machining mode is as follows:

Operation mode	Max. speed	Conditions required
Memory operation	135 m/min (5315 IPM)	None
HD operation	67 m/min (2638 IPM)	With the POSITION display selected on the screen (Note 2)
Ethernet operation	135 m/min (5315 IPM)	Avoid unusual key operations (Note 3)
IC card operation	135 m/min (5315 IPM)	None

The microsegment machining capability is restricted further by the functions used in, or applied to, the program as shown below:

Preparatory functions			Fairing function	
			Not applied	Applied
G01	Linear interpolation only (Note 1)		135 m/min (5315 IPM)	84 m/min (3307 IPM)
G02/G03	Circular interpolation included (Note 1)		33 m/min (1299 IPM)	
G06.1	Fine-spline interpolation included		101 m/min (3976 IPM)	50 m/min (1969 IPM)
G54.4	Workpiece setup error correction (Note 1)		67.2 m/min (2646 IPM)	
G68.2	Inclined-plane machining (Note 1)		67.2 m/min (2646 IPM)	
G41.x/G42.x	Tool radius compensation for five-axis machining to the left/right (Note 1)		33.6 m/min (1323 IPM)	
G43.4	Tool tip point control (Note 1)		67.2 m/min (2646 IPM)	

Note 1: The microsegment machining capabilities shown above apply in the case where 3-axis and 5-axis simultaneous motion commands consist of up to 32 and 52 characters per block, respectively, for a segment length of 1 mm. Exceeding the limit of block size may deteriorate the machining capabilities in question. Moreover, the lowest value of the maximum available rate of feed applies in combined use of multiple functions.

Note 2: If the **POSITION** display should be changed to any other display during operation, program reading from the hard disk may be aborted to damage the surface to be machined.

Note 3: If unusual operations, such as holding down any cursor/page key or a mouse button, are performed, program reading from the network may be aborted to damage the surface to be machined.

Note 4: Before executing a microsegment machining program for hard disk operation or Ethernet operation, terminate the commercially available software if it is being used.

Note 5: Since optimum corner deceleration occurs during the shape correction mode, the machining time may be longer than in other modes.

22-1 Programming Format

G5 P2 High-speed machining mode ON
G5 P0 High-speed machining mode OFF

Note 1: Both commands must be given in a single-command block.

Note 2: Do not use both commands in the MDI operation mode; otherwise an alarm (807 **ILLEGAL FORMAT**) occurs.

22-2 Commands Available in the High-Speed Machining Mode

Only axis motion commands with the corresponding preparatory functions (G-codes) and feed functions (F-codes), designation of sequence number, codes for Control Out and Control In (parentheses), as well as M/S/T functions are available in the high-speed machining mode. Setting data of any other type will result in an alarm (**807 ILLEGAL FORMAT**).

1. G-codes

The available preparatory functions are G00, G01, G02, G03, G17, G18, G19, G93 and G94.

The circular interpolation (helical interpolation unavailable) can be programmed with R (radius designation) as well as with I and J (center designation). If the machining program includes circular commands, however, set bit 2 of the **F96** parameter to one (1).

F96 bit 2: Type of control for circular commands in the high-speed machining mode:

- 0: Control for the specified speed (with acceleration/deceleration)
- 1: Control for a uniform feed

2. Axis motion commands

Absolute data input as well as incremental data input is applicable, indeed, but the former input mode requires the validation of bit 5 of the **F84** parameter.

F84 bit 5: Type of position data input in the high-speed machining mode:

- 0: Always incremental data input
- 1: According to the input mode before selection of the high-speed machining mode

3. Feed functions

Feed rate can be specified with address F.

4. Sequence number

Sequence number can be specified with address N. This number, however, is skipped as a meaningless code during reading.

5. Control Out and Control In

Use round brackets "(" and ")" as required to insert a comment. See Section 3-1 for more information.

It should be noted here, however, that useless deceleration may be caused by a move-free block. Move-free are, for instance, an empty block (null contents), one beginning and ending with "(" and ")", a G91-block with a moving distance of zero (0) as well as a G90-block for the very same position.

6. M/S/T functions

Codes (M, S, and T) of miscellaneous, spindle, and tool functions can be used in general as required. Use of these functions, however, may cause useless deceleration.

Use of the following functions in particular can only cause an alarm (**807 ILLEGAL FORMAT**):

- No. 2 miscellaneous functions (8-digit A-, B- or C-codes).
- Special M-codes for interrupting by, and calling for, a macroprogram (M96, M97, M98, M99).

7. Sample program

```

G28 X0 Y0 Z0
G90 G0X-100.Y-100.
G43 Z-5.H03
G01 F3000
G05 P2 _____ High-speed machining mode ON
X0.1
X0.1 Y0.001
X0.1 Y0.002
      ⋮
X0.1 F200
G05 P0 _____ High-speed machining mode OFF
G49 Z0
M02

```

When **F84** bit 5 = 0:
Incremental motion under G01

When **F84** bit 5 = 1:
Absolute motion under G01

Note 1: Either 0 or 2 is to be set with address P (P0 or P2). Setting any other value will result in an alarm (**807 ILLEGAL FORMAT**).

Note 2: No other addresses than P and N must be set in the same block with G05.

Note 3: A decimal point must not be appended to address P.

Note 4: The maximum permissible length of one block is 30 characters.

22-3 Additional Functions in the High-Speed Machining Mode

1. Fairing function

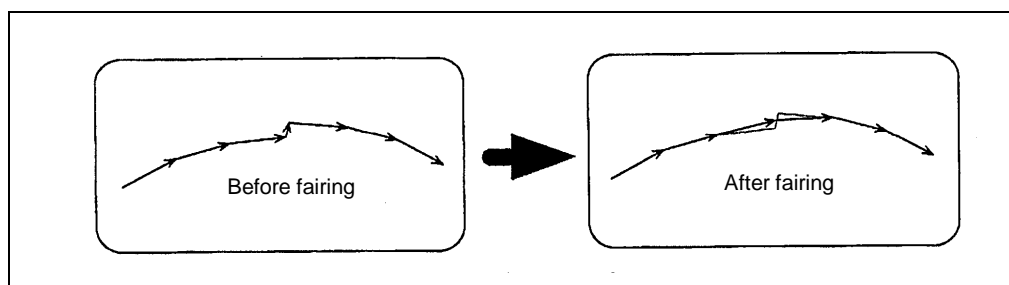
If, in a series of linear paths, a protruding section exists in the CAM-created microsegment machining program, this protruding path can be removed and the preceding and following paths connected smoothly by setting parameter **F96** bit 1 to "1".

F96 bit 1: Fairing function for the microsegment machining program

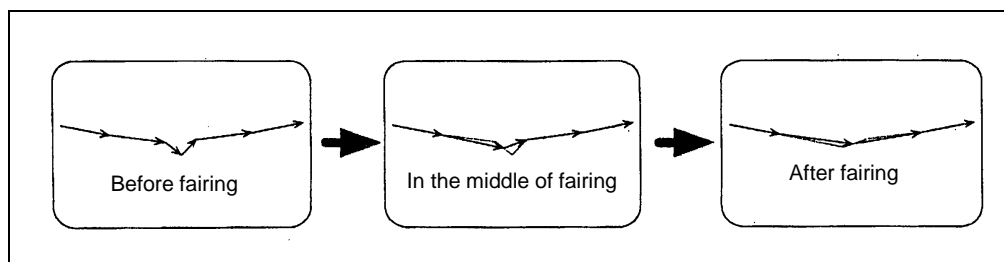
0: No fairing

1: Fairing for a protruding path

F103: Maximum length of a block to be removed for fairing



Fairing is also valid for a succession of protruding paths as shown below:

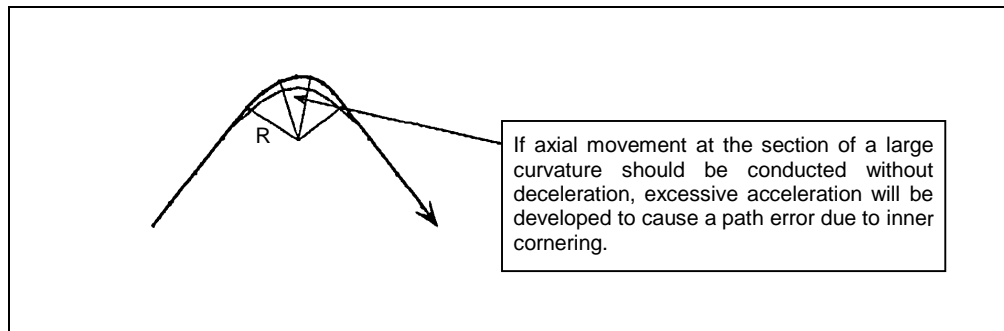


2. Cutting feed limiting speed

In shape correction mode, the minimum of the cutting feed limiting speeds of the movable axes is set as the cutting feed limiting speed in the high-speed machining mode. Setting parameter **F96** bit 5 to "1", however, allows the curvature of every curved section to be judged for limiting the speed so as not to exceed the maximum available acceleration.

F96 bit 5: Type of cutting feed limiting speed for the high-speed machining mode

- 0: Minimum of the cutting feed limiting speeds of the movable axes
- 1: Limiting speed based on the radius of curvature



3. Deceleration at corners in the high-speed machining mode

In shape correction mode, automatic deceleration at corners of significantly large angle is provided in general to ensure that the acceleration developed during cornering shall fall within the predetermined tolerance.

A micro-length block between relatively longer blocks intersecting each other in a large angle in CAM-created microsegment machining programs, in particular, may cause the cornering speed to mismatch the surroundings and thus affect surface quality.

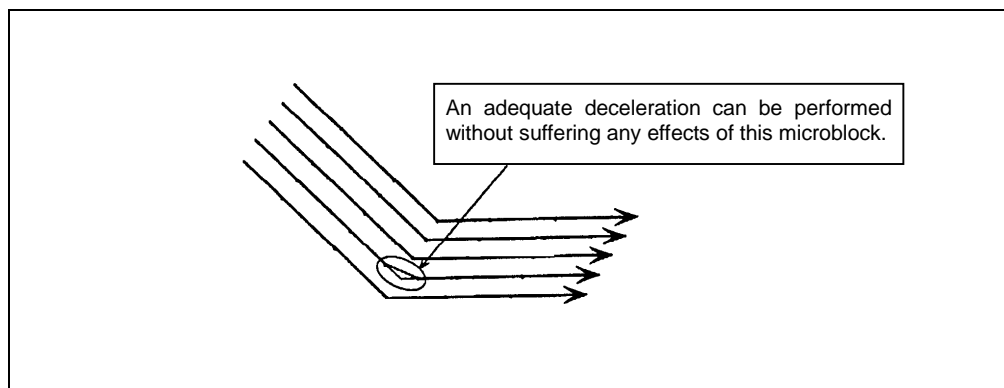
Setting parameter **F96** bit 4 to "1" will now allow corner judgment and deceleration without suffering any effects of such a microblock.

To use this function, however, the high-accuracy control option is required in addition to the optional high-speed machining function.

F96 bit 4: Type of corner judgment in the high-speed machining mode

- 0: Always judging from the angle between adjacent blocks
- 1: Judging after removing any microblock (if present between large-angle blocks)

F107: Reference length for microblock judgement



22-4 Restrictions

1. The modal functions other than that of G-code group 01 will be saved during, and restored upon cancellation of, the high-speed machining mode, indeed; but the modal functions for tool radius compensation, mirror image, scaling, coordinate system rotation, virtual axis interpolation and three-dimensional radius compensation should have been cancelled beforehand to give a G05 P2 command. Otherwise, an alarm may be caused or the modal function unexpectedly cancelled.

Example:

Main program

G28 X0 Y0 Z0

G90 G92 X0 Y0 Z100.

G00 X-100.Y-100.

G43 Z-10.H001 Movement under the conditions of G90, G00 and G43

M98 H001

G49 Z0

Movement under the conditions of G90 and G01

G28 X0 Y0 Z0

M02

Subprogram (O001)

N001 F3000

G05 P2

High-speed machining mode ON

G01 X0.1

X-0.1 Y-0.001

X-0.1 Y-0.002

⋮

X0.1

G05 P0

High-speed machining mode OFF

M99

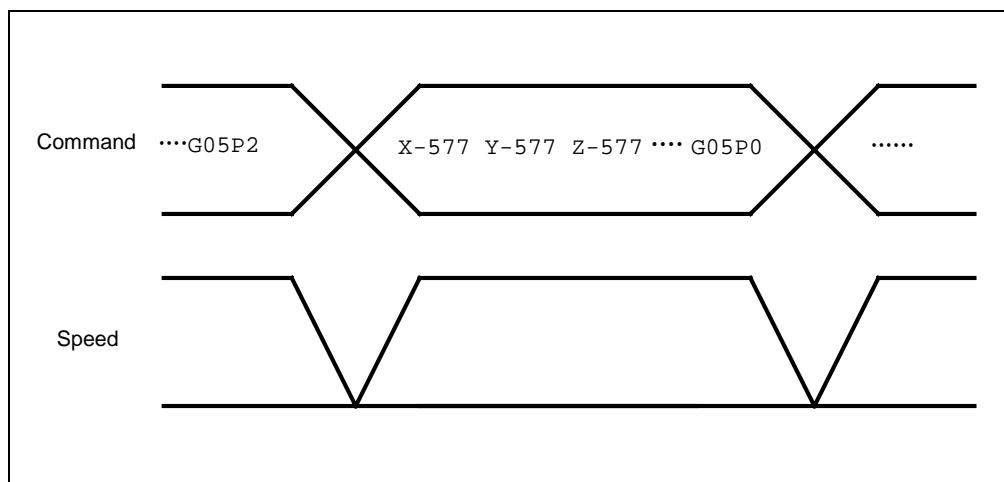
When **F84** bit 5 = 0:

Incremental motion under G01

When **F84** bit 5 = 1:

Absolute motion under G01

2. In the high-speed machining mode there may occur a delay in display response since priority is always given to the processing for the automatic operation.
3. The high-speed machining mode should be selected and cancelled by using commands of G05 P2 and G05 P0, respectively, with the tool sufficiently cleared from the workpiece since the selection and cancellation always cause a deceleration of feed motions as shown below:



4. Fairing function cannot be executed in the mode of single-block operation.
5. In the mode of high-speed machining a block of M/S/T function causes the flow of processing for fairing to be interrupted temporarily.
6. In the mode of high-speed machining a block of T**M6 for tool change causes the execution speed and the fairing function to be lowered and suppressed, respectively, until a length-offsetting value is made valid for the new tool, which does not occur implicitly if the length values of the MAZATROL tool data are not to be used in executing EIA/ISO programs (**F93** bit 3 = 0). In this case, therefore, it is necessary to cancel the high-speed machining mode temporarily so as to give explicitly the required G43-command for tool length offset.

T01M6 ← Tool change
 G01 X__ F__
 Z__ ← Offsetting value made valid at the
 first Z-axis movement
 X__ Y__

} Interpolation for high-speed machining
 } suppressed
 ↓ resumed

7. Restrictions on programming and machine operation are listed in the following table:

O: Valid, -: Invalid, err: Error

Specification		Standard mode	High-speed mode (Designation in the mode)
Classification	Subclassification		
Control axes	Maximum controllable axis quantity	14	14
	Effective controllable axis quantity	14	7
	Simultaneously controllable axis quantity	5	5
	Axis name	○	○ (○)
	CT axis	○	○ (○)
Units of control	Unit of input	ABC	ABC
	Unit of programming	○	○
	Unit-of-programming × 10	○	○
Input formats	Tape code	ISO/EIA	ISO/EIA
	Label skip	○	- (-)
	ISO/EIA automatic identification	○	○ (○)
	Parity H	○	○ (○)
	Parity V	○	○ (○)
	Tape format	○	Refer to the programming format.
	Program number	○	○ (err)
	Sequence number	○	○ (○)
	Control IN/OUT	○	○ (○)
	Optimal block skip	○	○ (err)
Buffers	Tape input buffer	○	○ (○)
	Pre-read buffer	○	○ (○)
Position commands	Absolute/incremental data input	○	○ (err)
	Inch/metric selection	○	○ (err)
	Decimal point input	○	○ (○)

O: Valid, -: Invalid, err: Error

Specification		Standard mode	High-speed mode (Designation in the mode)
Classification	Subclassification		
Interpolation functions	Positioning	O	O (O)
	One-way positioning	O	- (err)
	Linear interpolation	O	O (O)
	Circular interpolation	O	O (O)
	Helical cutting	O	- (err)
	Spiral interpolation	O	- (err)
	Virtual-axis interpolation	O	- (err)
	Threading	O	- (err)
	Plane selection	O	O (O)
	Fine-Spline interpolation	O	O (err)
	NURBS interpolation	O	- (err)
Feed functions	Rapid feed rate	O	O (O)
	Cutting feed rate	O	O (O)
	Synchronous feed	O	O (err)
	Automatic acceleration/deceleration	O	O (O)
	Linear acceleration/deceleration before cutting interpolation	O	O (err)
	Cutting feed rate limitation	Limitation in cutting direction	Minimum limiting speed of feed axes/ According to curvature
	Rapid feed override	O	O (O)
	No. 1 cutting feed override	O	O (O)
	No. 2 cutting feed override	O	O (O)
	Exact-stop mode	O	- (err)
	Cutting mode	O	O (err)
	Tapping mode	O	- (err)
	Automatic corner override	O	-
	Error detection	O	O (O)
	Override cancellation	O	O
Dwell	Dwell in time	O	- (err)
	Dwell in number of revolutions	O	- (err)
Miscellaneous function	M-command	O	O (O)
	Optional stop	O	- (O)
	No. 2 miscellaneous functions	O	O (err)
Spindle functions	S-command	O	O (O)
Tool functions	T-command	O	O (O)
	Tool operation time integration	O	O (O)
	Spare-tool selection	O	O (-)
Tool offset functions	Tool-length offset	O	O (err)
	Tool-position offset	O	- (err)
	Tool radius compensation	O	- (err)
	3D tool radius compensation	O	- (err)
	Tool-offset memory	O	O (O)
	Number of tool offset data sets	O	O (O)
	Programmed tool-offset input	O	- (err)
	Tool-offset number auto selection	O	O (err)

O: Valid, -: Invalid, err: Error

Specification		Standard mode	High-speed mode (Designation in the mode)
Classification	Subclassification		
Program auxiliary functions	Fixed cycle for drilling	O	- (err)
	Pattern cycle	-	- (-)
	Subprogram control	O	O (err)
	Variable command	O	- (err)
	Figure rotation	O	- (err)
	Coordinate rotation	O	- (err)
	User macro	O	O (err)
	User macro interruption	O	O (err)
	Scaling	O	- (err)
	Mirror image	O	- (err)
	Programmable mirror image	O	- (err)
	Geometric function	O	- (err)
	Programmed parameter setting	O	err (err)
Coordinate system setting	Watchdog-based reference-point return	O	O (-)
	Memory-based reference-point return	O	O (-)
	Automatic reference-point return	O	- (err)
	#2/#3/#4 reference-point return	O	- (err)
	Reference-point check	O	- (err)
	Machine coordinate system offset	O	- (err)
	Workpiece coordinate system offset	O	- (err)
	Local coordinate system offset	O	- (err)
	Coordinate system setting	O	- (err)
	Coordinate system rotation setting	O	- (err)
	Program restart	O	O (err)
	Absolute data detection	O	O (O)
Machine error correction	Backlash correction	O	O (O)
	Lost-motion correction	O	O (O)
	Memory-based relative position correction	O	O (O)
	Machine coordinate system correction	O	O (O)
Protection functions	Emergency stop	O	O (O)
	Stroke end	O	O (O)
	Software limit	O	O (O)
	Programmed software limit	O	- (err)
	Interlock	O	O (O)
	External deceleration	O	O (O)
	Data protection	O	O (O)

O: Valid, -: Invalid, err: Error

Specification		Standard mode	High-speed mode (Designation in the mode)	
Classification	Subclassification			
Operation modes	Tape operation	O	O	(-)
	Memory operation	O	O	(-)
	MDI operation	O	O	(O)
	Jog feed	O	-	(O)
	Incremental feed	O	-	(O)
	Handle feed	O	-	(O)
	Manual rapid feed	O	-	(O)
	Handle interruption	O	O	(O)
	Auto/manual simultaneous	O	O	(O)
	HD operation	O	O	(-)
	IC card operation	O	O	(-)
	Ethernet operation	O	O	(-)
External control signals	Automatic-operation start	O	O	(O)
	Automatic-operation halt	O	O	(O)
	Single-block stop	O	O	(O)
	NC reset	O	-	(O)
	External reset	O	-	(O)
	All-axis machine lock	O	O	(O)
	Axis-by-axis machine lock	O	O	(O)
	Dry run	O	O	(O)
	Miscellaneous-function lock	O	O	(O)
	Manual-absolute selection	O	O	(-)
Status output signals	Control-unit ready	O	O	(O)
	Servo-unit ready	O	O	(O)
	Auto-run mode	O	O	(O)
	Auto-run in progress	O	O	(O)
	Auto-run halted	O	O	(O)
	Cutting feed in progress	O	O	(O)
	Tapping in progress	O	-	(-)
	Threading in progress	O	-	(-)
	Axis selected	O	O	(O)
	Axis-movement direction	O	O	(O)
	Rapid feed in progress	O	O	(O)
	Rewind	O	O	(O)
	NC alarm	O	O	(O)
	Reset	O	O	(O)
	Movement-command completed	O	O	(O)
Measurement aid functions	Manual tool-length measurement	O	-	(-)
	Automatic tool-length measurement	O	-	(err)
	Skip	O	-	(err)
	Multi-step skip	O	-	(err)
	Manual skip	O	-	(err)
Axis control functions	Servo off	O	O	(O)
	Follow-up	O	O	(O)
	Control-axis removal	O	O	(O)

O: Valid, -: Invalid, err: Error

Specification		Standard mode	High-speed mode (Designation in the mode)
Classification	Subclassification		
Data input/output	External data input I/F	O	O (O)
	External data output I/F	O	O (O)
	External data input/output	O	O (O)
Setting/display functions	Setting/Display unit	O	O (O)
	Settings display	O	O (O)
	Search	O	O (err)
	Check-and-stop	O	- (-)
	MDI	O	O (O)
	Program restart	O	O (err)
	Machining-time calculation	O	O (O)
	PC opening	O	O (O)
	Program-status display	O	O (O)
	Integrated-time display	O	O (O)
	Graphics display	O	O (O)
Program creation	Multi-step skip	O	- (err)
	Graphics check	O	O (O)
Self-diagnostics	Alarm display	O	O (O)
	Operation-error display	O	O (O)
	Servo-error display	O	O (O)
	Operation-stop-cause display	O	O (O)
	Servo monitor display	O	O (O)
	NC-PC I/O signal display	O	O (O)
	DIO display	O	O (O)
	Keyboard-operation record	O	O (O)

8. The table below enumerates the modes in which high-speed machining mode is selectable.

Function		Code	Function	Code
Positioning		G00	Selection of additional workpiece coordinate system	G54.1
Linear interpolation		G01		
Circular interpolation (CW)		G02	Workpiece setup error correction	G54.4
Circular interpolation (CCW)		G03	Selection of workpiece coordinate system 2	G55
Spiral interpolation (CW)		G02.1	Selection of workpiece coordinate system 3	G56
Spiral interpolation (CCW)		G03.1	Selection of workpiece coordinate system 4	G57
Spline interpolation		G06.1	Selection of workpiece coordinate system 5	G58
Polar coordinate interpolation OFF		G13.1	Selection of workpiece coordinate system 6	G59
Polar coordinate input OFF		G15	Exact stop mode	G61
Selection of XY-plane		G17	Geometry compensation	G61.1
Selection of ZX-plane		G18	Cutting mode	G64
Selection of YZ-plane		G19	User macro modal call OFF	G67
Five-surface machining OFF		G17.9	Inclined-plane machining	G68.2
Inch data input		G20	3-D coordinate conversion ON	G68
Metric data input		G21	3-D coordinate conversion OFF	G69
Pre-move stroke check OFF		G23	Fixed cycle OFF	G80
Tool radius compensation OFF		G40	Absolute data input	G90
Tool radius compensation for five-axis machining	(to the left)	G41.5	Incremental data input	G91
	(to the right)	G42.5	Inverse time feed	G93
Tool length offset (+)		G43	Feed per minute (asynchronous)	G94
Tool length offset (–)		G44	Constant surface speed control OFF	G97
Tool tip point control, type 1/2		G43.4/G43.5	Initial point level return in hole-machining fixed cycles	G98
Tool position offset OFF		G49		
Head offset for five-surface machining OFF		G49.1	R-point level return in hole-machining fixed cycles	G99
Scaling OFF		G50		
Mirror image OFF		G50.1	Radius data input for X-axis ON	G10.9X0
Selection of workpiece coordinate system 1		G54	Radius data input for X-axis OFF	G10.9X1

23 AUTOMATIC TOOL LENGTH MEASUREMENT: G37 (OPTION)

1. Function and purpose

When the tool for which command data has been assigned moves to a programmed measurement position, the NC system will measure and calculate any differential data between the coordinates at that time and those of the programmed measurement position. Data thus obtained will become offset data for that tool.

Also, if offsetting has already been performed for the tool, the current offset data will be further offset, provided that after movement of that tool under an offset status to the required measurement position, the measurements and calculations of any differential coordinates show some data to be further offset.

At this time, further offsetting will occur for the tool offset data if only one type of offset data exists, or for the tool wear offset data if two types of offset data exist (tool length offsets and tool wear offsets).

Note: This preparatory function cannot be used for VERSATECH machines.

2. Programming format

G37 Z_ (X_, Y_) R_ D_ F_

X, Y, Z: Address of the measurement axis and the coordinate of the measurement position

R: Distance from the starting point of movement at a measurement feed rate, to the measurement position

D: The area where the tool is to stop moving

F: Measurement feed rate

If R, D, or F is omitted, respective parameter values will become valid.

3. Description of parameters

Parameter	Description
F42	R-code command. Deceleration area
F43	D-code command. Measurement area
F44	F-code command. Measurement feed rate f
F72	Conditions for skipping based on G37 (EIA/ISO)

(See the separate Parameter List/Alarm List/M-code List for further details.)

4. Example of execution

If $H01 = 0$

T01T00M06

G90G00G43Z0H01

G37Z-600.R200.D150.F300

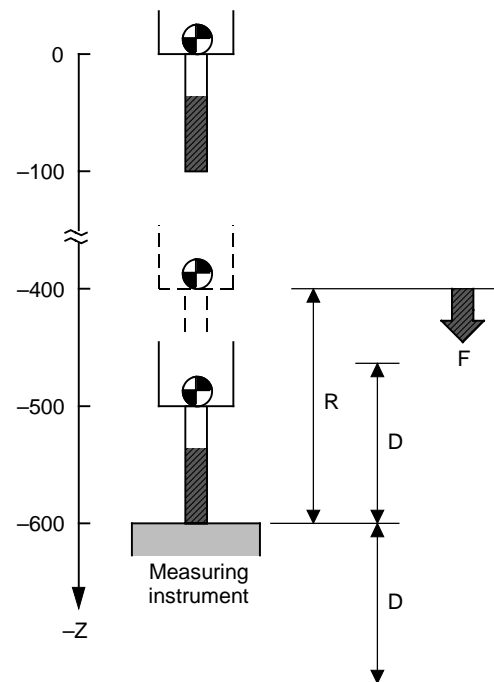
Coordinate to reach the measurement position

= -500.01

$-500.01 - (-600) = 99.99$

$0 + 99.99 = 99.99$

Thus, $H01 = 99.99$



MEP229

If $H01 = 100$

T01T00M06

G90G00G43Z-200.H01

G37Z-600.F300

Coordinate to reach the measurement position

= -600.01

$-600.01 - (-600) = -0.01$

$100 + (-0.01) = 99.99$

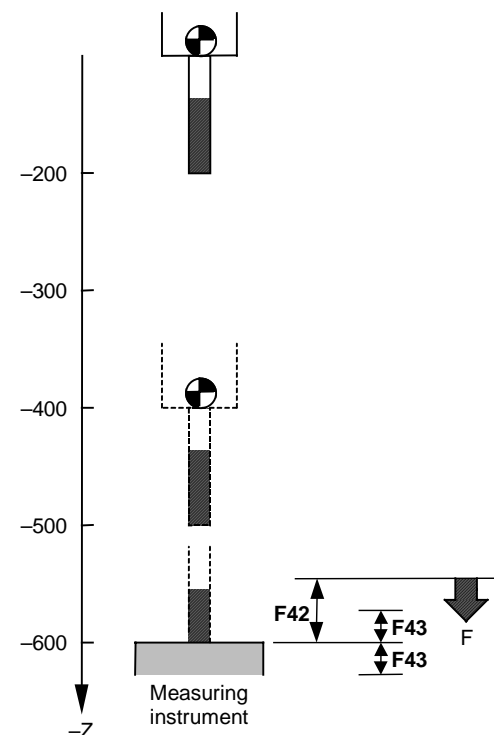
Thus, $H01 = 99.99$

<Supplement>

When the program shown above is executed, parameter **F42** and **F43** are set as follows:

F42 (R-code command) : 25000 (25 mm)

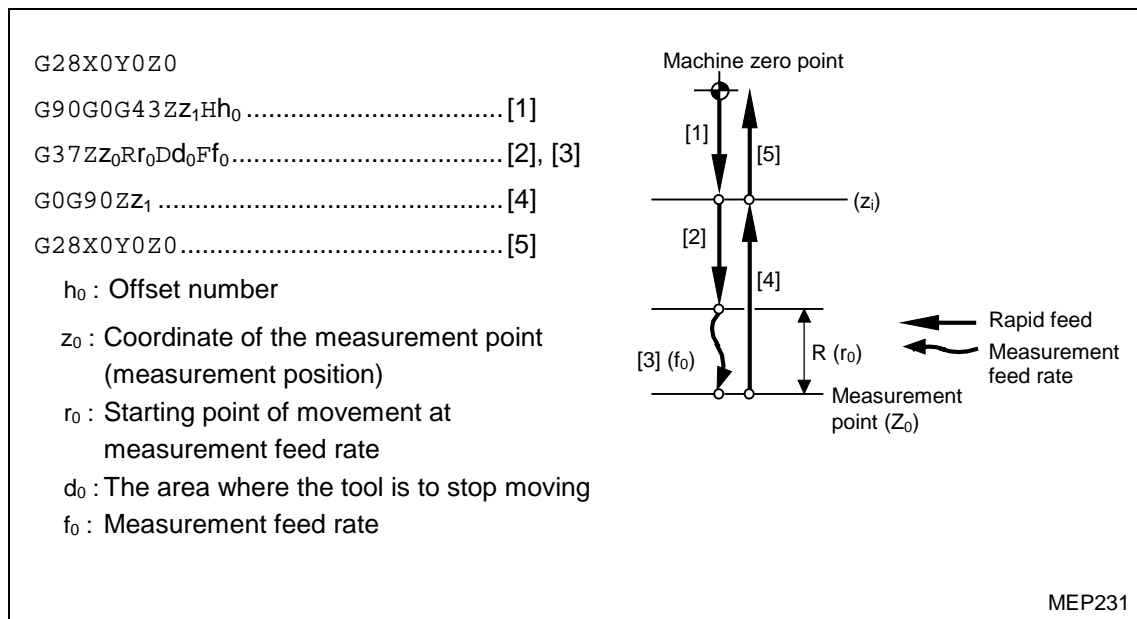
F43 (D-code command) : 2000 (2 mm)



MEP230

5. Detailed description

1. Machine action based on command G37



2. Sensor signals (Measurement Position Reached) also act as skip signals.
3. If the F-code value is 0, the feed rate becomes 1 mm/min.
4. Update offset data becomes valid from the Z-axis (measurement axis) command codes that succeed the block of G37.
5. The delay and dispersion in processing of sensor signals, except for the PLC side, is from 0 to 0.2 ms for the NC side alone. Accordingly, the following measurement error may occur:

Maximum measurement error [mm]

$$= \text{Measurement feed rate [mm/min]} \times \frac{1}{60} \times \frac{0.2 \text{ [ms]}}{1000}$$

6. When a sensor signal is detected, although the coordinates of the machine position at that time will be read, the machine will stop only after overrunning through the distance equivalent to a servo droop.

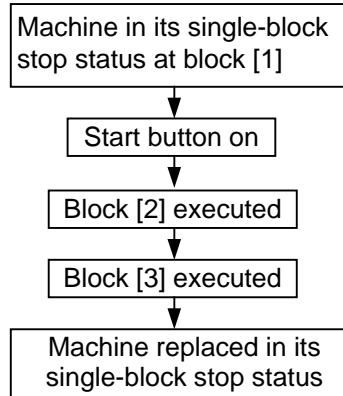
Maximum amount of overrun [mm]

$$= \text{Measurement feed rate [mm/min]} \times \frac{1}{60} \times \frac{30.3 \text{ [ms]}}{1000}$$

30.3 [ms] if the position loop gain is 33.

7. If command G37 is executed in the single-block operation mode, the machine will come to a single block stop after execution of the block that immediately succeeds the G37-containing block.

Example: G0G90G43Z-200.H01 [1]
 G37Z-600.R25.D2.F10 [2]
 G0G90Z-200. [3]



6. Precautions

- Alarm **889 G37 OPTION NOT FOUND** will result if G37 is set for a machine that does not have a mounted option for automatic tool length measurement.
- Alarm **923 ILLEGAL COMMAND G37 AXIS** will result if the block of G37 does not contain axis data or contains data of two or more axes.
- Alarm **924 G37, H COMMANDS SAME BLOCK** will result if an H code exists in the block of G37.
- Alarm **925 H CODE REQUIRED** will result if G43 H_ does not exist before the block of G37.
- Alarm **926 ILLEGAL G37 SIGNAL** will result if input sensor signals occur outside a predetermined allowable measurement range or if a sensor signal is not detected on arrival of the tool at the ending point of movement.
- If a manual interruption operation has been carried out during movement of the tool at a measurement feed rate, the program must be restarted only after returning that tool to the position existing when the interruption operation was carried out.
- Set G37 data or parameter data so that the following condition is satisfied:

$$\text{Measurement point} - \text{Starting point} > \begin{matrix} \text{R-code value} \\ \text{or parameter r} \end{matrix} > \begin{matrix} \text{D-code value} \\ \text{or parameter d} \end{matrix}$$
- If the R-code value, the D-code value and parameter d, mentioned in Item G above, are all 0s, the program will come to a normal end only when the designated measurement point and the sensor signal detection point agree. Alarm **926 ILLEGAL G37 SIGNAL** will result in all other cases.
- If the R-code value, the D-code value, parameter r, and parameter d, mentioned in Item G above, are all 0s, alarm **926 ILLEGAL G37 SIGNAL** will result after the tool has been positioned at the designated measurement point, irrespective of whether a sensor signal is detected.
- Set G37 (automatic tool length measurement code) together with G43 H_ (offset number assignment code).

G43 H_
 G37 Z_R_D_F_

11. If the offset data is tool offsets of type A, then automatic correction of tool data occurs, or if the offset data is tool offsets of type B, then automatic correction of tool wear offsetting data occurs.

Example: The **TOOL OFFSET** displays in both cases after offsetting of H1 = 100

	TOOL OFFSET (Type A)				TOOL OFFSET (Type B)		
Before measurement					TOOL LENGTH		
	No.	OFFSET	No.	OFFSET	No.	GEOMETRY	WEAR
	1	100	17	0	1	100	0
	2	0	18	0	2	0	0
	3	0	19	0	3	0	0
After measurement					TOOL LENGTH		
	No.	OFFSET	No.	OFFSET	No.	GEOMETRY	WEAR
	1	110	17	0	1	100	10
	2	0	18	0	2	0	0
	3	0	19	0	3	0	0

12. The distance from the machine zero point to the measurement point (skip sensor) is preset in register R2392 or R2393. Use this value as reference to set a coordinate using Z-, X-, or Y-code command.
13. When this function is used for tool offsets of type B, the correct data will not be displayed if the wear offset value exceeds 100.
14. When executing this function in the presence of offset data, set the value of a D code to 2mm or less to prevent damaging the measuring instrument.
15. When executing this function in the absence of offset data (offset data = 0), set the values of an R code and a D code to those larger than the tool length of the tool to be measured. Also, in that case, before executing this function, make sure that the skip sensor in the measuring instrument correctly operates.

- NOTE -

24 DYNAMIC OFFSETTING II: G54.2P0, G54.2P1 - G54.2P8 (OPTION)

1. Function and purpose

When a workpiece fixed on the turntable is to be machined with the rotation of the table, mismatching between the workpiece reference position (program origin) and the origin of workpiece coordinates (center of rotation of the table) leads to an error in machining contour. Provided that the vector of a particular deviation from the center of rotation to the workpiece reference position is given as a "reference", the "Dynamic Offsetting II" function will calculate for each command of rotation the deviation vector for the designated angular motion in order to control the linear axes for an adequate movement to the ending point as programmed with respect to the ideal workpiece origin, and thus to prevent the above-mentioned faulty machining from occurring.

2. Programming format

G54.2 Pn;

n: Dynamic offset number (1 to 8)

Give a "G54.2 P0" command (n = 0) to cancel the dynamic offsetting function.

Cancellation is the initial state of the function (upon turning-on).

3. Definitions of terms

A. Deviation vector

The vector of a deviation from the center of rotation of the table (Wo: presupposed position of the workpiece origin) to the actual origin of coordinates of the workpiece mounted on the table.

B. Dynamic offset

The offsetting vector (= deviation vector; whose direction depends upon the angular position of the table) for the ending point of each block containing a command of rotation.

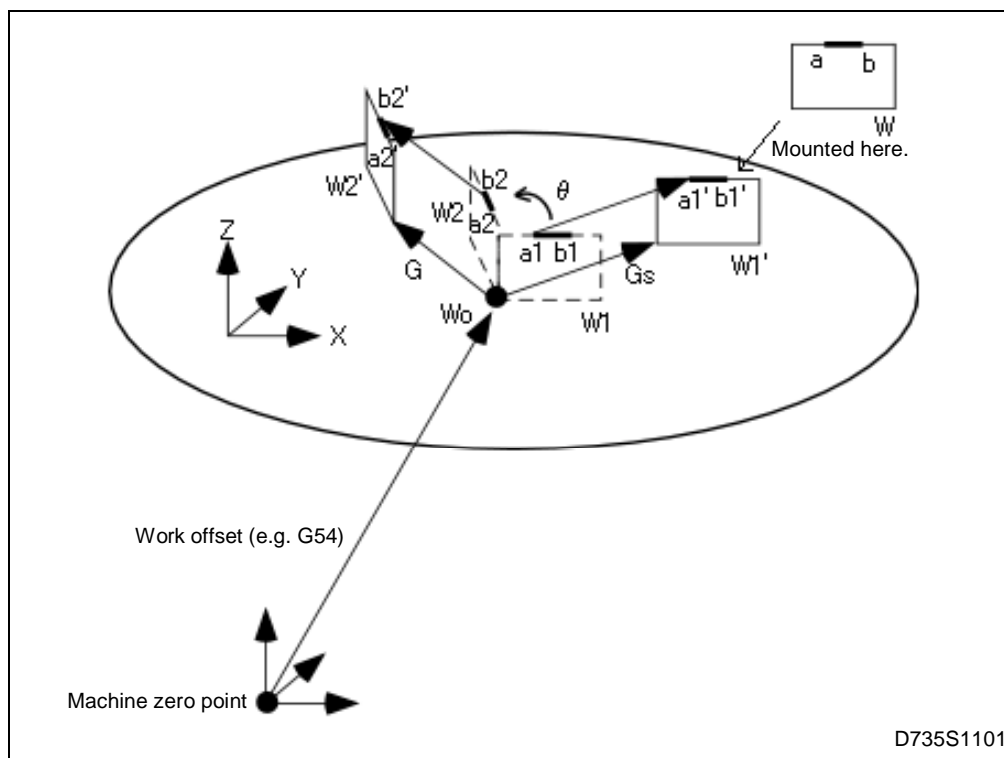
C. Reference dynamic offset

A particular deviation vector entered as the reference for the calculation of dynamic offsets. Consists of the vector proper (measured and entered in three-axis component vectors) and the positions (in machine coordinates) of the rotational and tilting axis for the measurement.

4. Operation description

A. Operation by a command of rotation in the G54.2 mode

In the G54.2 mode (modal group 23), which is selected by a “G54.2Pn” command, deviation vector (to be used in a vector addition for offsetting) is re-calculated for each command of table rotation beforehand in order to create an adequate tool path for the block’s ending point as programmed with respect to the ideal workpiece origin.



[Legend]

- W1: The ideal workpiece mounting position (the workpiece origin set on to the center of rotation of the table)
- W1': The actual workpiece mounting position (vector G_s denotes the deviation from the ideal position)
- W2': The position of the actual workpiece W1' after a table rotation by θ
- W2: The position of the ideally mounted workpiece W1 after a table rotation by θ
- W_o: The origin of workpiece coordinates (given by a corresponding preparatory function, such as G54)
- G_s: The reference deviation vector (to be registered in the NC unit as a reference dynamic offset.)
- G: The deviation vector for the rotation of the rotational axis by θ
- a (a1, a1'): The starting point of the G1 (linear interpolation) microsegment command
- b (b1, b2'): The ending point of the G1 (linear interpolation) microsegment command

With the measurement results of the reference dynamic offset (G_s) registered for workpiece W fixed on the turntable, the selection (activation) of the G54.2 mode causes the tool to be shifted by the deviation vector G_s from the current position, point **a1** for example, to point **a1'** (if bit 0 of the **F87** parameter described later is set to “0”).

A succeeding command of “G1b1” (b1 = designation of a point with X-, Y-, and Z-coordinates) feeds the tool from **a1'** to **b1'** in the G1 mode (linearly). If, however, simultaneous motion of the rotational axis is designated in the same block, “G1b1C θ ” for example, the tool is also fed linearly from the current position **a1'** to the offset position **b2'** which is obtained by adding the deviation vector G internally calculated for the θ rotation to point **b2**, the ending point on the ideally mounted workpiece.

B. On-reset operation

It depends on the setting of parameter **F95** bit 7 whether or not dynamic offsetting is canceled by resetting.

F95 bit 7 = 0: The dynamic offset is canceled and the G54.2 mode is also canceled.

= 1: The existing dynamic offset is held along with the G54.2 mode. When the automatic operation is started again after resetting, the dynamic offsetting mode is active from the beginning of the program.

Note: When the dynamic offset is canceled by resetting, the tool will not move on the path corresponding to the canceled vector (even if bit 0 of the **F87** parameter described later is set to "0").

C. Operation by the selection of the G54.2 mode

When a G54.2Pn command is given, the deviation vector for the current position of the rotational axis is calculated and an offsetting movement is carried out on the linear axes by their respective components of the computed vector (dynamic offset). If an axis motion command is given in the same block, the deviation vector for the ending point of that block is calculated and the corresponding motion is performed from the current point to the dynamically offset ending point.

D. Operation by the cancellation of the G54.2 mode

The cancellation command (G54.2P0) moves the tool by a vector reverse to the current dynamic offset. It depends, however, on the setting of parameter **F168** bit 0 whether or not an independent cancellation command (G54.2P0) causes the above-mentioned movement.

F168 bit 0 = 0: G54.2P0 causes canceling motions on the linear axes concerned.

= 1: G54.2P0 does not cause any motions on the linear axes.

If an axis motion command is given in the same block, the corresponding motion is performed from the current point to the ending point as designated with workpiece coordinates (a movement including the cancellation of the dynamic offsetting).

The axis motion occurs according to the current modal function concerned (of G-code group 1).

E. Manual interruption in the G54.2 mode

The deviation vector does not change if automatic operation is stopped in the G54.2 mode (by single-block stop, etc.) and then a movement on the rotational axis carried out in manual mode. The re-calculation of the deviation vector for dynamic offsetting will not occur until a rotational axis motion command or another G54.2 command is given after setting the MDI or automatic operation mode.

5. Input and output of the reference dynamic offset**A. Setting the reference dynamic offset by G10**

G10 L21 Pn Xx Yy $\alpha\alpha$;

Use this format of programmed parameter input. Argument P (n) denotes a dynamic offset number (1 to 8).

According to the data input mode, absolute (G90) or incremental (G91), the designated axis value overwrites, or is added to, the current one.

B. Reading/writing the reference dynamic offset with system variables

System variable number = $5500 + 20 \times n + m$

n: Dynamic offset number (1 to 8)

m: Axis number (1 to 6)

Use system variable #5510 to read the selected dynamic offset number (1 to 8).

C. Reading the machine coordinates of the center of table rotation with system variables

#50700: X-coordinate of the center of table rotation (Machine parameter **S5 X**)

#50705: Y-coordinate of the center of table rotation (Machine parameter **S5 Y**)

#50701: Z-coordinate of the center of table rotation (Machine parameter **S5 Z**)

6. Other detailed precautions

1. When the related parameters and reference dynamic offset are modified in the G54.2 mode, the modifications will become valid for the next G54.2Pn command onward.
2. The following describes how some specific commands are executed in the G54.2 mode.
 - (a) Machine coordinate system selection (G53)
A G53 command temporarily suppresses the dynamic offset and the axis motion is performed to the ending point as designated in machine coordinates. The deviation vector is not re-calculated even when a value for the rotational axis is specified. The dynamic offsetting function will not be recovered until a motion command is given with workpiece coordinates.
 - (b) Workpiece coordinate system change (G54 to G59, G54.1, G92, G52)
Even when the workpiece coordinate system is changed in the G54.2 mode, the reference dynamic offset is not re-calculated and dynamic offsets are calculated according to the existing reference dynamic offset. The axis motion is carried out to the position obtained by adding the deviation vector to the ending point specified in the new workpiece coordinate system.
 - (c) Commands related to zero point return (G27, G28, G29, G30, G30.n)
The dynamic offsetting function is temporarily canceled for the path from the intermediate point to the reference point and recovered for the movement from there to a position specified in the workpiece coordinate system. (Similar to the processing of the commands related to zero point return in the tool length offset mode)
3. When the work offset data (workpiece origin) being used is modified by a G10 command in the G54.2 mode, the new work offset data will be valid for the next block onward.
4. As for the tool motion caused by a change only in the deviation vector, it is executed in the current mode of G-code group 1 and at the current rate of feed. If, however, the mode concerned is other than that of G0 or G1, e.g. a mode of circular interpolation (G2, G3, etc.), the tool is temporarily moved in the mode of linear interpolation (G1).
5. The type of the control axis for the turntable must be specified as "rotational". The dynamic offsetting function II cannot be used for the C-axis specified as "linear type".
6. The polar coordinate interpolation with the rotational axis cannot be executed properly in the G54.2 mode.
7. The following function commands cannot be executed in the G54.2 mode:

- Restarting the program	- Figure rotation (M98)
- Mirror image (by G51.1 or control signal)	- Coordinates rotation (G68)
- Scaling (G51)	- G61.1, G61.2, G5P0, G5P2
8. The workpiece coordinates read with system variables include dynamic offsets.

9. The component vectors of the current dynamic offset can be read using system variables #5121 (X-axis), #5122 (Y-axis) and #5123 (Z-axis).

7. Related alarms

936 OPTION NOT FOUND

The dynamic offset II option is not installed.

959 WORKPIECE COORDINATE ERROR

The origin of workpiece coordinates does not match the center of rotation of the turntable.

807 ILLEGAL FORMAT

Argument P is missing in the block of G54.2.

An incompatible G-code is used in the G54.2 mode or G54.2 is given in the mode of an incompatible G-code.

809 ILLEGAL NUMBER INPUT

The value of P in the block of G54.2 is not proper.

8. Related parameters

A. Rotational axis configuration

Specify the type of rotational axis configuration of the machine to be operated.

L81 = 0: Makes the dynamic offsetting function invalid.

= 1: Two rotational axes (C-axis on A-axis)

= 2: One rotational axis (A-axis)

= 3: One rotational axis (C-axis)

= 4: One rotational axis (B-axis)

Specify "1" for the VARIAXIS series, and "4" for the FH/PFH series with an NC rotary table.

B. Dynamic offset type

Specify whether or not the tool is to be offset by each change only in the deviation vector.

F87 bit 0 = 0: Offset (the indication of both workpiece and machine coordinates changes.)

= 1: Not offset (no change in the position indication at all)

Normally set this parameter to "0".

C. Center of table rotation

Specify the center of rotation of the table in machine coordinates. These parameters are also used in the VARIAXIS control for MAZATROL programs.

The preset values refer to the factory adjustment at Mazak.

S5 X, Y Center of rotation of the turntable (Machine coordinates)

S12 Y, Z Axis of rotation of the tilting table (Machine coordinates)

S11 Z Distance (length) from the tilting axis to the turntable surface
(The turntable center must be in the direction of -Z from the tilting axis.)

Note: When **L81** = 2, 3, or 4, the **S11** and **S12** settings are not required.

D. Workpiece origin mismatch check

The origin of the selected workpiece coordinate system must correspond to the center of table rotation in order that the dynamic offsetting may effectively function. The following parameter is provided to check the condition in question for each G54.2 command.

F87 bit 1 = 0: The mismatch check is conducted.

= 1: The mismatch check is not conducted.

Normally set this parameter to "0".

9. Mechanical requirements

The dynamic offsetting function requires the following conditions to be satisfied:

1. The machine is equipped with a table of either two-axis rotational control (construction of a turntable on the tilting axis) or of a single rotational axis control (turntable or tilting table). The tilting and rotational axis must refer to rotating around the X- and Z-axis, respectively. Moreover, the construction must not be of the tilting axis mounted on the turntable.
2. The workpiece coordinate origin corresponds to the center of table rotation, and the X-, Y-, and Z-axes of workpiece coordinates are in parallel with, and the same direction as, the corresponding axes of machine coordinates.
3. The requirements for machining with table rotation: The machining contour is described using a workpiece coordinate system fixed in parallel with the machine coordinate system (not rotated with the table rotation) and microsegment command blocks of G1.

10. Operation description using a sample program

The following describes the operation using a sample program (created for explanation only).

A. Settings on the related displays

WORK OFFSET (G54) X = -315.0, Y = -315.0, Z = 0.0, A = 0.0, C = 0.0

DYNAMIC OFFSET (P1) X = -1.0, Y = 0.0, Z = 0.0, A = 0.0, C = 90.0

Parameters **L81** = 1 (Rotational axis configuration: Two rotational axes; C-axis on A-axis)

F87 bit 0 = 0 (Dynamic offset type: Offset)

S5 X = -315000

S5 Y = -315000

B. Sample program (for explanation of operation)

```

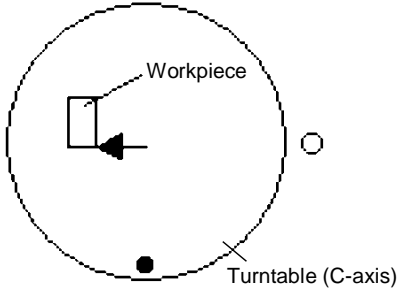
N1 G91 G28 X0 Y0 Z0 A0 C0
N2 G54
N3 G90 G00 X0 Y0 Z0 A0 C0
N4 G54.2P1
N5 G01 C180.0 F1000
N6 G01 X10.0
N7 G03 X0 Y10.0 R10.0
N8 G01 C240.0

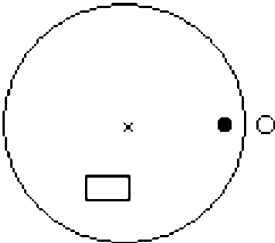
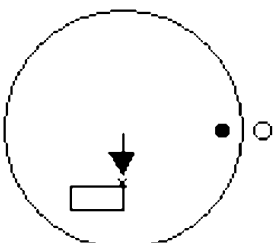
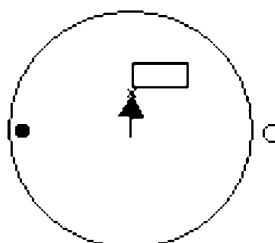
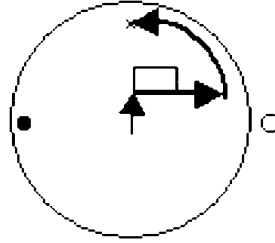
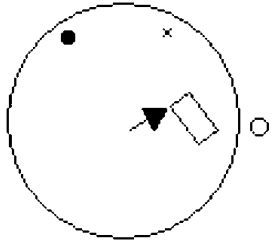
```

C. Position indication and dynamic offset for each line of the program

N-No.	POSITION (workpiece coordinates)					MACHINE (machine coordinates)					Dynamic offset		
	X	Y	Z	A	C	X	Y	Z	A	C	X	Y	Z
N1	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
N2	315.000	315.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
N3	0.000	0.000	0.000	0.000	0.000	-315.000	-315.000	0.000	0.000	0.000	0.000	0.000	0.000
N4	0.000	-1.000	0.000	0.000	0.000	0.000	-316.000	0.000	0.000	0.000	0.000	-1.000	0.000
N5	0.000	1.000	0.000	0.000	180.000	0.000	-314.000	0.000	0.000	180.000	0.000	1.000	0.000
N6	10.000	1.000	0.000	0.000	180.000	-305.000	-314.000	0.000	0.000	180.000	0.000	1.000	0.000
N7	0.000	11.000	0.000	0.000	180.000	-315.000	-304.000	0.000	0.000	180.000	0.000	1.000	0.000
N8	0.866	10.500	0.000	0.000	240.000	-314.134	-325.500	0.000	0.000	240.000	0.866	0.500	0.000

D. Illustration of the sample program

Measurement of the reference dynamic offset	
 <p>Workpiece</p> <p>Turntable (C-axis)</p>	<p>Let the position where the ● mark on the table is aligned with the fixed position marked with O be the zero point of the C-axis.</p> <p>The reference dynamic offset (arrow) = $(-1, 0, 0)$ was measured with the table positioned at $C = 90.0$, as shown on the left.</p>

N-No.	N3	N4	N5
Illustration	<p>1</p> 	<p>2</p> 	<p>3</p> 
N-No.	N6, N7	N8	N5
Illustration	<p>4</p> 	<p>5</p> 	

<Explanation>

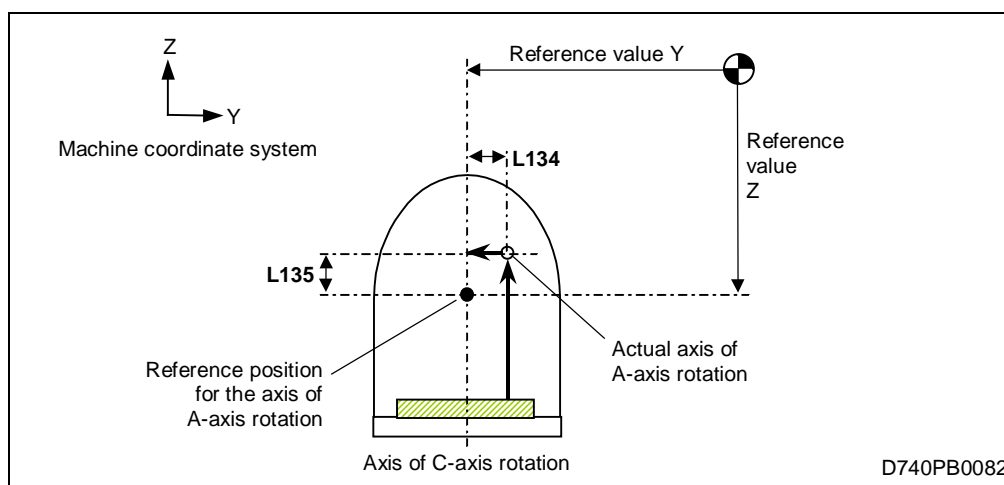
1. N3 turns the table on the C-axis to ● ($C = 0$) and positions the tool tip to the x point ($X, Y, Z = 0, 0, 0$).
2. N4 causes the tool tip to be shifted by the dynamic offset (arrow) for an angular position of $C = 0$ to the x point ($X, Y, Z = 0, -1, 0$).
3. N5 turns the table on the C-axis to ● ($C = 180$) and causes the tool tip to be shifted by linear interpolation to the x point ($X, Y, Z = 0, 1, 0$) determined by the dynamic offset (arrow) for an angular position of $C = 180$.
4. N6 and N7 interpolate the linear and circular paths to the x point.
5. N8 turns the table on the C-axis to ● and causes the tool tip to be shifted by linear interpolation to the x point.

- NOTE -

25 COMPENSATION FOR DEVIATION OF THE AXIS OF ROTATION OF THE TILTING TABLE

1. Outline

On machines with a tilting table (the axis of whose rotation is named A-axis) the position of the reference point for programming slightly deviates from the actual axis of table tilting and so moves with rotation on the A-axis, which in turn requires correction on the programmed point. The compensation function in question uses parameters **L134** and **L135**, which denote the deviation existing with both A- and C-axis positions being 0°, to conduct the correction by shifting on the orthogonal axes. There is no need any more, therefore, to use a macroprogram which is created for the compensation in question with the aid of the values α and β (nameplate ratings for each machine).



2. Precautions

- Do not use superfluously a macroprogram created for the compensation in question with the aid of the values α and β . Otherwise an inexpedient correction will be added.
- Set the postprocessor for creating EIA/ISO programs, if used, so as not to include in the program created the compensation in question with the aid of the values α and β . Otherwise an inexpedient correction will be added.

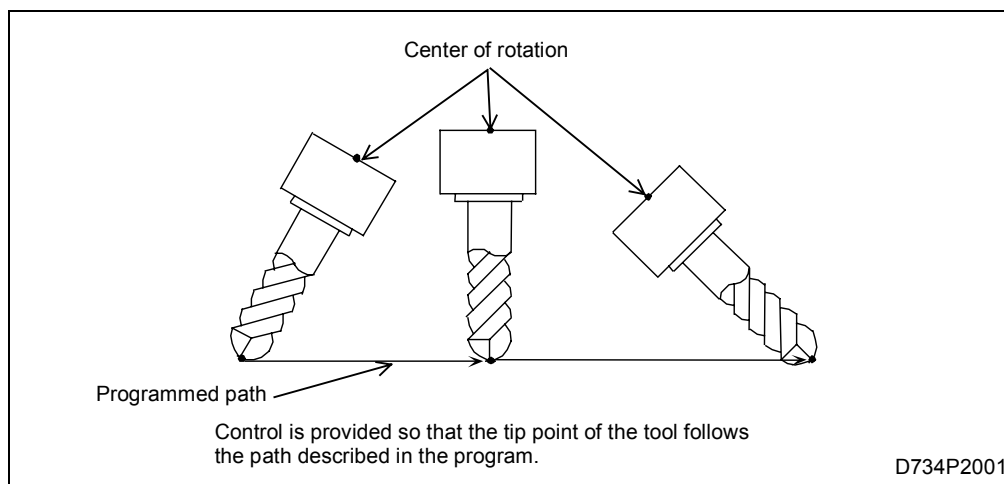
- NOTE -

26 FIVE-AXIS MACHINING FUNCTION

26-1 Tool Tip Point Control for Five-Axis Machining (Option)

26-1-1 Function outline

The tool-tip point control function provides simultaneous axis control for moving the tool (including its attitude) on a five-axis control machine in order that the tool tip point may describe such a path, at such a rate of feed, as programmed in terms of the relative position of the tool to the workpiece.



- This function is valid only for five-axis control machines.
- If the required option is not added, giving a command of tool tip point control will result in an alarm.

Remark: Three types of five-axis control machines

Tool rotating type	Table rotating type	Mixed type
<p>Tool's first rotational axis</p> <p>Tool's second rotational axis</p> <p>Tool</p> <p>Workpiece</p> <p>Table</p>	<p>Tool</p> <p>Table</p> <p>Workpiece</p> <p>Table's first rotational axis</p> <p>Table's second rotational axis</p>	<p>Tool's rotational axis</p> <p>Tool</p> <p>Workpiece</p> <p>Table</p> <p>Table's rotational axis</p>

D740PB0034

- * The VARIAXIS machines belong to the table rotating type.
- * The VERSATECH machines belong to the tool rotating type.

26-1-2 Detailed description

1. Programming coordinate system

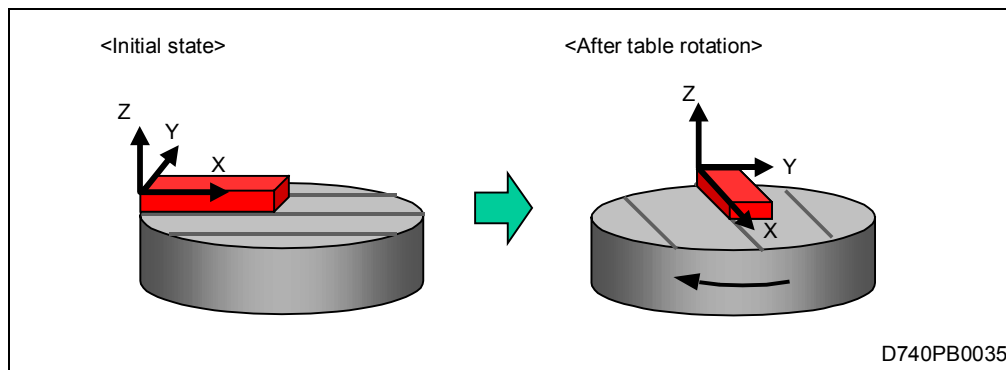
In the mode of tool tip point control, where an axis control is provided for the tip point to describe the programmed path, specify in a sequence of motion blocks the ending positions of the tool tip point in the peculiar coordinate system for programming. Two types of coordinate system are available (according to the parameter setting concerned) for describing the motion of the tool tip point: a table coordinate system (system fixed to the table) and a workpiece coordinate system.

Irrespective of which coordinate system is selected for programming, the relative motion of the tool tip point to the workpiece can be achieved as programmed in a sequence of linear interpolation (or circular interpolation, if available).

A. Programming with a table coordinate system

With the parameter **F85** bit 2 being set to "0", selecting the tool tip point control mode includes establishment of a programming coordinate system by fixing the current workpiece coordinate system to the table. The table coordinate system will rotate as the table rotates. It will not change in position, however, as the direction of the tool axis changes. Subsequent X-, Y- and Z-axis motion commands will be executed with respect to the table coordinate system.

The initial state of the table coordinate system refers to the current table position, or is to be specified by a table rotation command given in the block of G43.4 or G43.5.

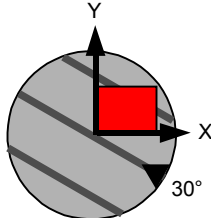
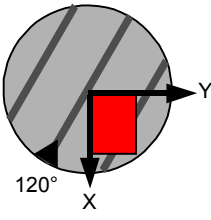
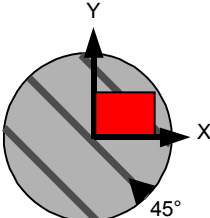
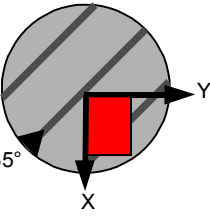


A workpiece coordinate system fixed to the table.

The table coordinate system will rotate as the table rotates. It will not rotate, however, as the tool rotates on its rotational axis. The tool path described on the basis of this coordinate system refers to the relative motion of the tool to the workpiece.

<Description of the table coordinate system>

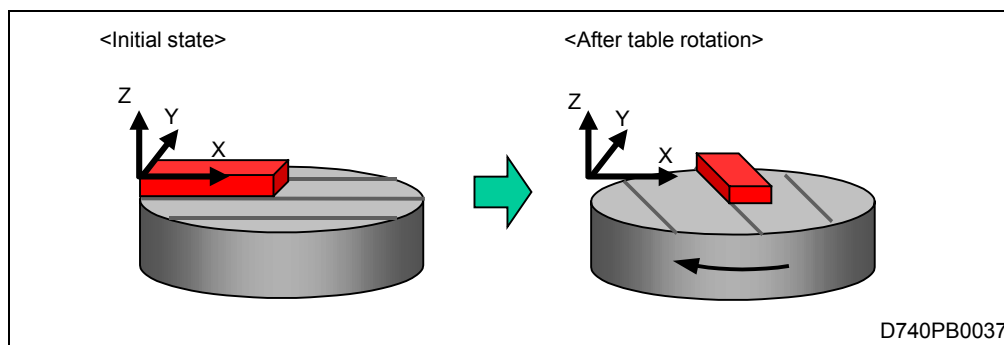
The angular position of the table at the fixing of the workpiece coordinate system depends on the setting of the parameter concerned as follows:

Reference angular position	Starting position (F86 bit 6 = 0)	0° position (F86 bit 6 = 1)
Method of fixing the workpiece coordinate system to the table	The workpiece coordinate system is fixed to the table in its angular position at the start of tool tip point control. That is, the initial state of the table coordinate system refers to the current table position, or is to be specified by a table rotation command given in the block of G43.4 or G43.5.	The workpiece coordinate system is fixed to the table in its angular position of 0°.
<p>Example: Workpiece origin setting for the C-axis = 45°</p>	<p>C-axis origin = 45°</p> <p>Reference position</p> <p>C-15. G43.4Hh</p>  <p>30°</p> <p>Table rotation</p> <p>Positioning</p> <p>C = 90.</p>  <p>120°</p> <p>Fixing the workpiece coordinate system to the table occurs at the start of tool tip point control.</p>	<p>C-axis origin = 45°</p> <p>Reference position</p> <p>C = 0</p>  <p>45°</p> <p>Table rotation</p> <p>Positioning</p> <p>C = 90.</p>  <p>135°</p> <p>Fixing the workpiece coordinate system to the table occurs with the table in a C-axis position of 0° in the workpiece coordinate system.</p> <p>D740PB0036</p>
Feature	Since the table coordinate system depends on the angular position at the start of tool tip point control, correctly perform initial positioning on the rotational axis concerned; otherwise the relative position of the tool tip point to the workpiece may deviate from the expected path.	Since the table coordinate system is independent of the angular position at the start of tool tip point control, the relative position of the tool tip point to the workpiece cannot deviate from the expected path due to the initial angular positioning.

B. Programming with a workpiece coordinate system

With the parameter **F85** bit 2 being set to "1", the current workpiece coordinate system is to be used for describing the movement of the tool tip position. The coordinate system in this case will not rotate as the table rotates.

Subsequent X-, Y- and Z-axis commands will be of linear motion with respect to the table (workpiece). Specify the ending positions along the orthogonal axes always taking into consideration the particular angle of the table rotation.



A workpiece coordinate system established by offsetting the machine coordinate system in accordance with the workpiece origin settings.

This coordinate system is stationary on the machine and does not rotate, therefore, with any motion on the rotational axis of the table or the tool.

2. Programming format

Two types of programming formats are available for tool tip point control: type 1 for programming only a tool length offset, and type 2 for programming a tool length offset and the direction (attitude) of the tool axis.

A. Tool tip point control ON

<Type 1>

G43.4 (Xx Yy Zz Aa Bb Cc) Hh Tool tip point control type 1 ON

<Type 2>

G43.5 (Xx Yy Zz) Ii Jj Kk Hh Tool tip point control type 2 ON

x, y, z : Orthogonal coordinate axis motion command

a, b, c : Rotational axis motion command

i, j, k : Direction of tool axis (Position vector from tip point to center of rotation of the tool)

h : Tool length offset number

B. Tool tip point control OFF (cancellation)

G49 Tool tip point control OFF

Other G-codes in group 8

G43/G44 (Tool length offset in the plus/minus direction)

C. Notes

1. The startup axis movement is executed in the currently active mode of movement. However, giving a G43.4 or G43.5 command under a mode other than of linear movement (G00 or G01) will cause an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**).
2. Do not give a command that reverses the direction of the tool with respect to the workpiece. Otherwise an alarm will be caused (**973 ILLEGAL TOOL AXIS VECTOR**).

3. Do not use an axis address other than those for the particular axes specified in the parameters concerned (3 linear and 2 rotational ones). Otherwise an alarm will be caused (**974 TOOL TIP PT CTRL FORMAT ERROR**).
4. Do not specify the attitude of the tool axis using I, J, and K in the mode of tool tip point control type 1 (G43.4). Otherwise an alarm will be caused (**974 TOOL TIP PT CTRL FORMAT ERROR**).
5. Do not give any rotational axis motion command in the mode of tool tip point control type 2 (G43.5). Otherwise an alarm will be caused (**974 TOOL TIP PT CTRL FORMAT ERROR**).
6. Do not give a G43.5 command (for tool tip point control type 2) without the table coordinate system being appropriately selected for programming. Otherwise an alarm will be caused (**971 CANNOT USE TOOL TIP PT CONTROL**).
7. An argument of zero (0) for vector components (I, J, K) can be omitted in the mode of tool tip point control type 2 (G43.5). The position vector of the preceding block will be kept intact if all the three components are omitted.
8. Parameter **F36** bit 6 determines the number of effective decimal digits in the tool axis vector's components (I, J, K) for tool tip point control type 2.

- **F36** bit 6 = 0:

4 and 5 decimal digits are effective for the metric and inch system, respectively.

Example: For a block of **G43.5I0.12345678J0.12345678K0.12345678H**:

By rounding off the fifth decimal digit, 0.1235 is used as arguments I, J, and K for the metric system.

By rounding off the seventh decimal digit, 0.12346 is used as arguments I, J, and K for the inch system.

- **F36** bit 6 = 1:

7 decimal digits are effective, irrespective of the system of units.

Example: For a block of **G43.5I0.12345678J0.12345678K0.12345678H**:

By rounding off the eighth decimal digit, 0.1234568 is used as arguments I, J, and K.

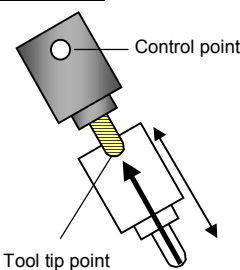
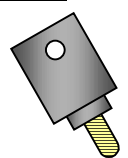
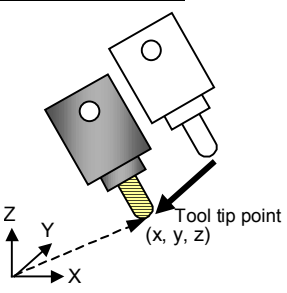
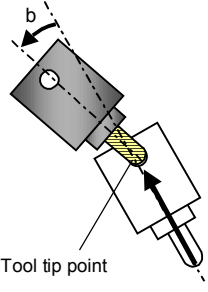
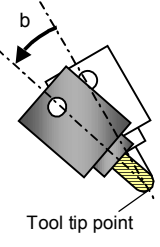
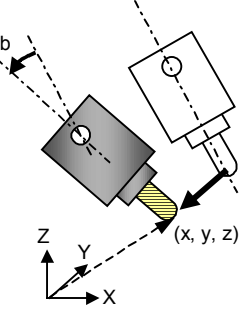
<Number of effective digits in vector components I, J, and K>

F36 bit 6	System of units	Minimum value	Maximum value
0	Metric system	-99999.9999	99999.9999
	Inch system	-9999.99999	9999.99999
1	Metric system	-99.9999999	99.9999999
	Inch system	-99.9999999	99.9999999

9. It depends upon the setting of a parameter (**F114** bit 1) whether or not the cancellation causes an axis motion (in the currently active mode in G-code group 1: at rapid traverse [G00] or cutting feed [G01]) of canceling the tool length offset. Do not give a cancellation command in a mode of circular interpolation. Otherwise an alarm will be caused (**971 CANNOT USE TOOL TIP PT CONTROL**).
10. The cancellation command G49 must be given with no other instruction codes. Giving the G49 code with an axis motion command will cause an alarm (**974 TOOL TIP PT CTRL FORMAT ERROR**).
11. In the mode of tool tip point control, the angular movement on a rotational axis of rotational type always occur on the shortest path, and an angular distance of just 180° is covered in particular by rotation in the positive direction.

3. Startup

- The startup axis movement is executed with the tool tip point control being active (by an interpolation with reference to the table coordinate system).
- As explained in the table below, the startup operation depends upon whether or not motion commands for the orthogonal or rotational axes are given in the same block as G43.4 or G43.5. (The figures in the following table refer to a case where G43.4 is used for a machine equipped with the tool's rotational axis named B. This example applies in principle to all other cases: use of G43.5 on a machine of different configuration, etc.)

Commands for rotational axis motion or tool-axis direction	Motion commands for orthogonal axes		
	not given (*1)		given (*1)
	F162 bit 0 = 0 (Motion by the offset amount)	F162 bit 0 = 1 (No motion by the offset amount)	
not given	G43 . 4Hh  <p>A shifting motion occurs for the tip point to be at the current control point, i.e. in the tool-axis direction through the length offset distance.</p>	G43 . 4Hh  <p>No motions occur at all. (A motion by the offset amount does not occur.)</p>	G43 . 4XxYyZzHh  <p>A motion occurs for the tip point to be at a point of the specified coordinates, inclusive of offset amount.</p>
given (*2)	G43 . 4BbHh  <p>A linear and rotational motion occurs for the tip point to be at the current control point, as in the programming coordinate system (*2), through the length offset distance.</p>	G43 . 4BbHh  <p>A rotational motion occurs with automatic linear axis movements in order not to move the position of the tool tip point, as in the programming coordinate system (*2).</p>	G43 . 4XxYyZzBbHh  <p>A linear and rotational motion occurs for the tip point to be at a point of the specified coordinates, as in the programming coordinate system (*2), inclusive of offset amount.</p>

D740PB0038

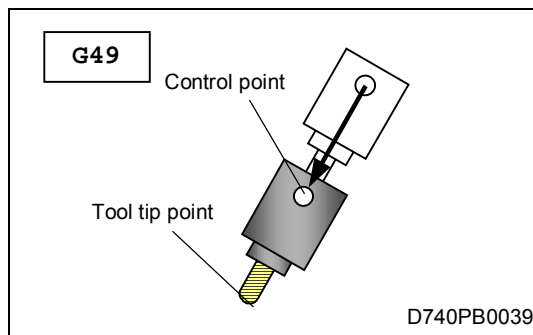
- (*1) "Given" is the orthogonal-axis motion command when the G43.4 or G43.5 block contains even only one command for an orthogonal axis.
- (*2) If there is also a motion command for the table's rotational axis given with the table coordinate system being selected for programming, then that particular programming coordinate system (the table coordinate system) will move accordingly. In such a case the final position of the tip point's (eventual) motion denotes a position relative to the table in the specified angle.

4. Cancellation

A parameter is provided for the selection of whether or not the cancellation includes the corresponding axis movement.

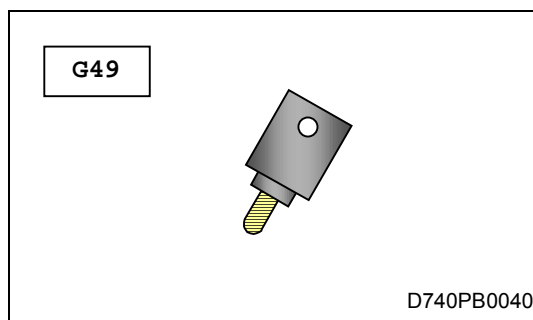
- Cancellation with axis movement (**F114** bit 1 = 0)

The tool tip point control mode is cancelled with a shifting motion in the direction of the tool axis for canceling the particular amount of length offset.



- Cancellation without axis movement (**F114** bit 1 = 1)

The tool tip point control mode is cancelled without any axis motions being caused by the cancellation block.



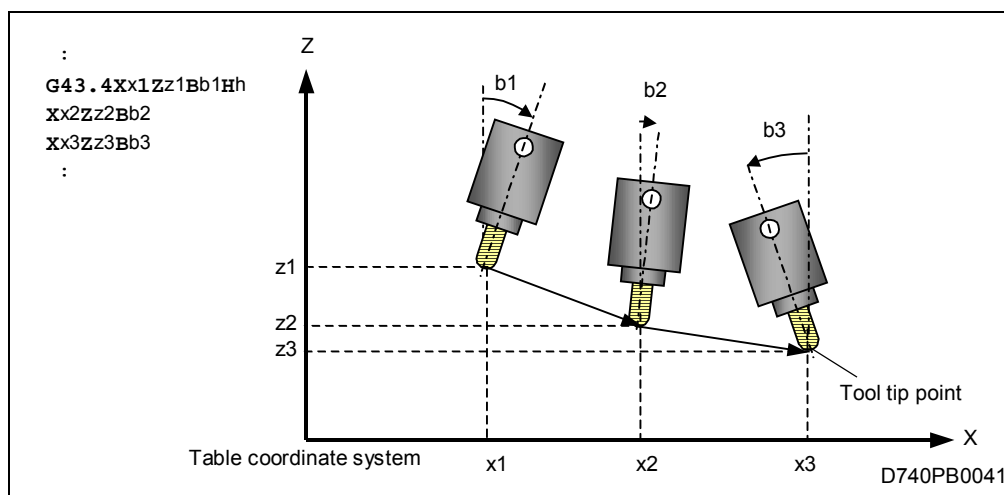
Note: Giving the G49 code with an axis motion command will cause an alarm (**974 TOOL TIP PT CTRL FORMAT ERROR**).

5. Operation in the mode of tool tip point control

A. Tool tip point control type 1 (G43.4)

<Orthogonal and rotational axis motion commands given in one block>

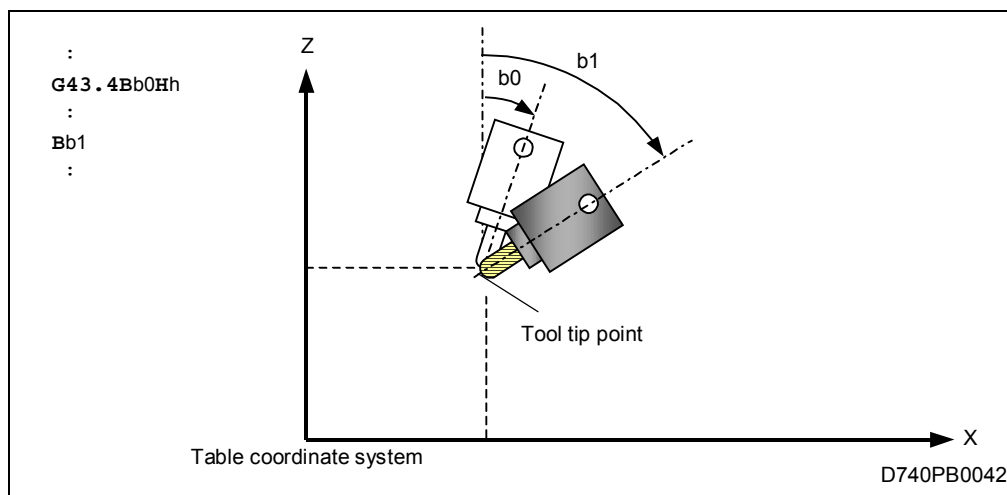
- The motion is controlled in order that the tool tip point may describe the programmed path.



- The points on the tool path programmed using the workpiece coordinate system will be traced by the tool after internal conversion of coordinates for the table coordinate system.

<An independent rotational axis motion command>

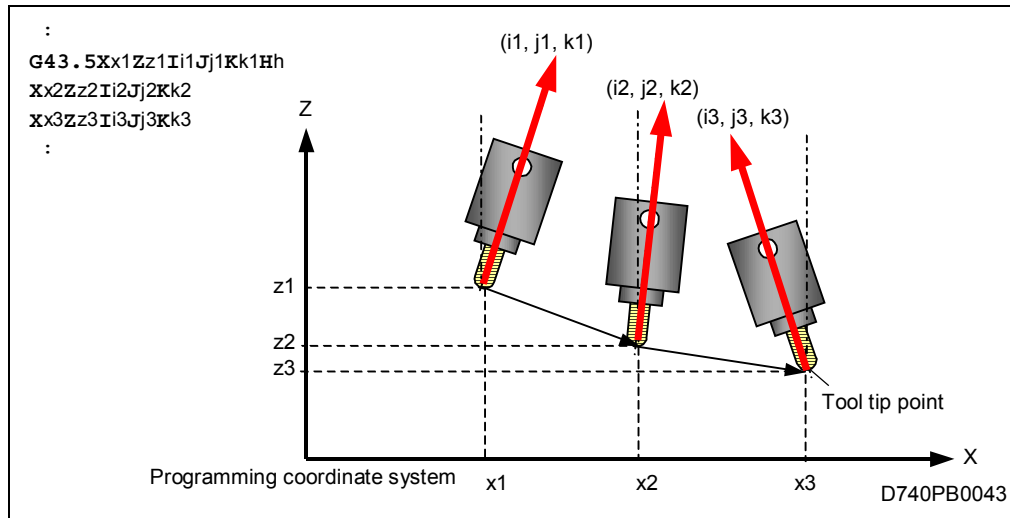
- The explicit command of rotation is executed together with automatic orthogonal axis movements in order not to move the position of the tool tip point, as in the table coordinate system (a position relative to the workpiece).



B. Tool tip point control type 2 (G43.5)

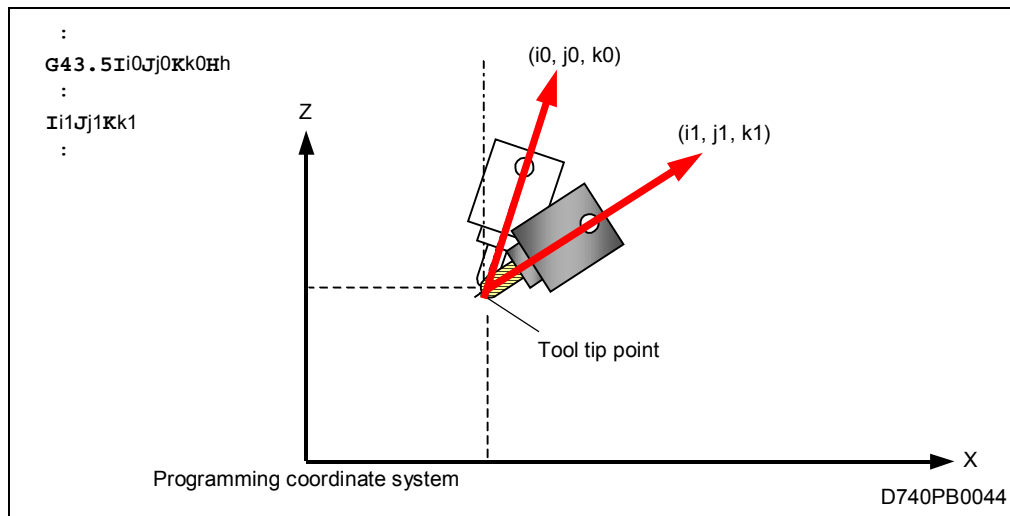
<Orthogonal axis motion and position vector commands given in one block>

The motion is controlled in order that the tool tip point may describe the programmed path.



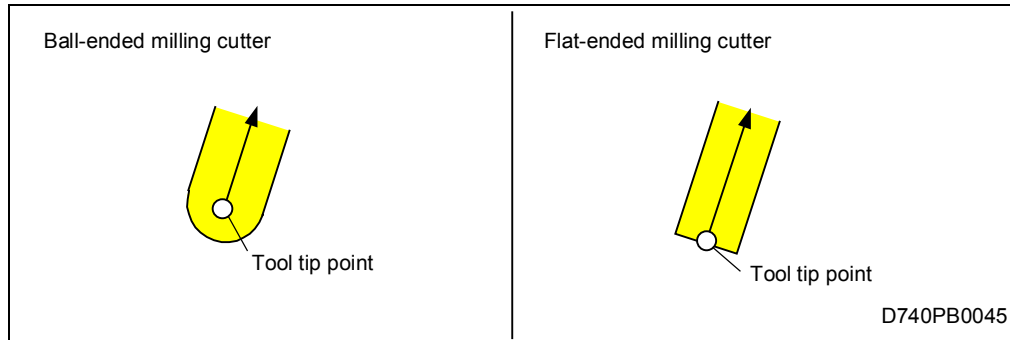
<An independent position vector command>

The rotation command included in the position vector is executed together with automatic orthogonal axis movements in order not to move the position of the tool tip point.

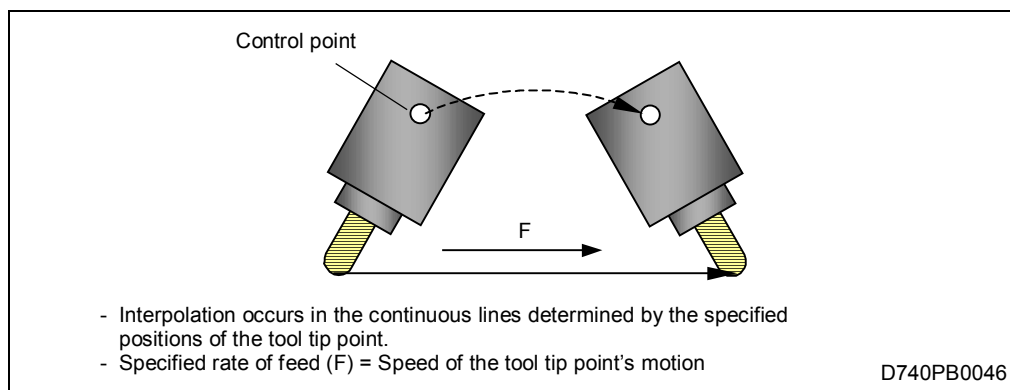


6. Positions, and Rate of feed, to be programmed in the mode of tool tip point control

Describe in the program the movement of the center of the tool tip.



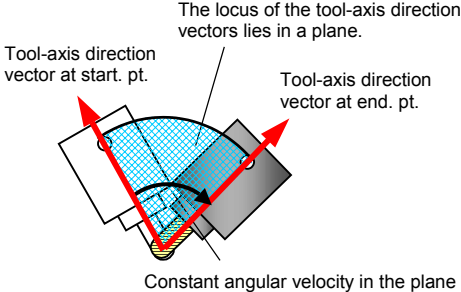
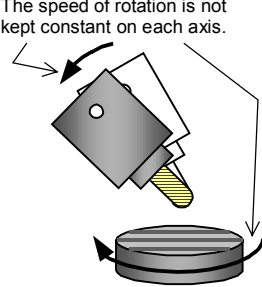
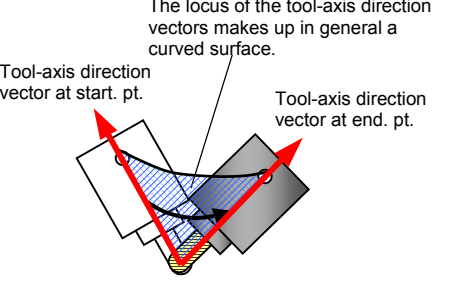
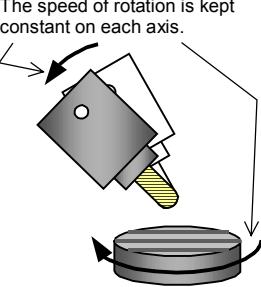
The feed is controlled so that the center of the tool tip may move at the specified rate.



7. Interpolation methods for rotational axes

A parameter is provided for the selection between two interpolation methods for rotational axes: Uniaxial rotation interpolation or Joint interpolation.

(In parentheses, the tool's attitude varies from the same one at the start in different ways to the same one at the end of an interpolation block, indeed, but in both methods the locus of the tool tip points is the same in reference to the workpiece.)

Method	Uniaxial rotation interpolation	Joint interpolation
Parameter	F85 bit 3 = 0	F85 bit 3 = 1
Operation	<p>A simultaneous control is conducted on the rotational axes in order that the tool-axis direction vector may move at a constant angular velocity in a plane which is determined by the tool-axis direction vectors at the starting and ending points.</p> <p><Tool axis direction relative to the workpiece></p>  <p><Machine action></p> 	<p>A linear interpolation with a constant rotational speed is applied to each rotational axis (as is normally the case with G01).</p> <p><Tool axis direction relative to the workpiece></p>  <p><Machine action></p> 
Feature	<ul style="list-style-type: none"> - There arises no unexpected collision or faulty machining during execution of an interpolation block since the tool attitude relative to the workpiece is always kept in a plane which is determined by the tool-axis direction vectors at the starting and ending points. - In order that the tool attitude relative to the workpiece may always be kept in one and the same plane, however, an intense motion on a particular controlled axis may be caused at the beginning, the end, or in the middle, of a block. 	<p>D740PB0047</p> <ul style="list-style-type: none"> - During execution of an interpolation block the tool axis deviates from the plane including the tool-axis direction vectors at the starting and ending points. The extent of the deviation depends upon the configuration of the machine's controlled axes. This must be taken into consideration in programming so as to prevent collision and faulty machining. - As the motion speed on each axis is kept constant, machine operation progresses smoothly and the machining time can be reduced to some extent in general. Consequently, this method of joint interpolation should be selected unless special care must be taken about collision and faulty machining.

<Notes>

In order to move the tool-axis direction vector in one and the same plane, the method of uniaxial rotation interpolation may sometimes cause abrupt and discontinuous axis motions, as illustrated below with an example. The method of joint interpolation should in general be selected unless, for operational safety or other reasons, it is desired to keep the change in tool's attitude always in the plane determined by the attitude vectors at the starting and ending points.

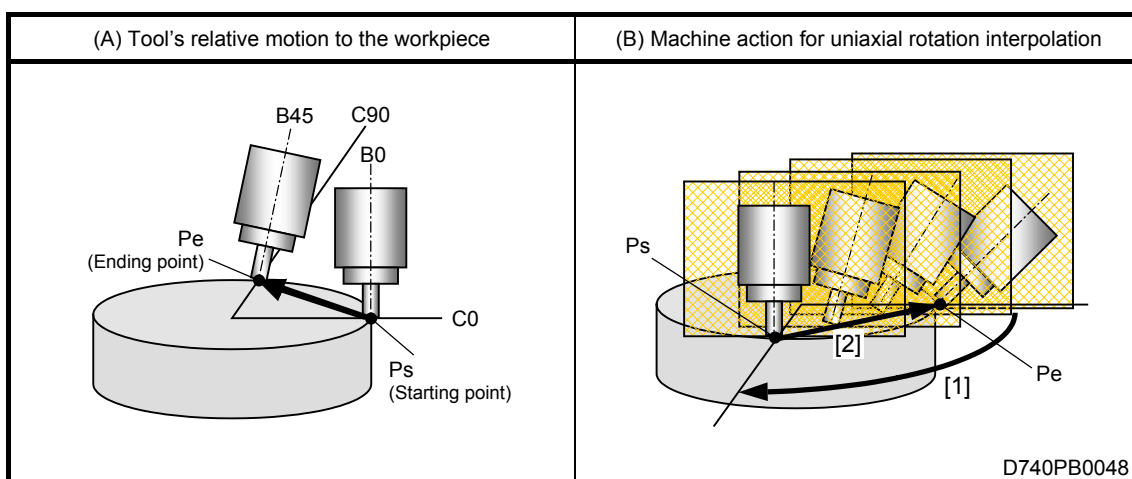
Program example

```
N1 B0.C0.
N2 G43.4H1
N3 B45.C90.
```

Figure **A** below shows the relative position of the tool to the workpiece for block N3.

Let us now take as an example the execution of block N3 by uniaxial rotation interpolation. In order that the tool's attitude vector may move in the plane including the tool-axis direction vectors at the starting and ending points, actual machine operation will be as shown in Figure **B**. That is: first, in order to fit the B-axis rotation plane to that of the motion of the tool's attitude vector

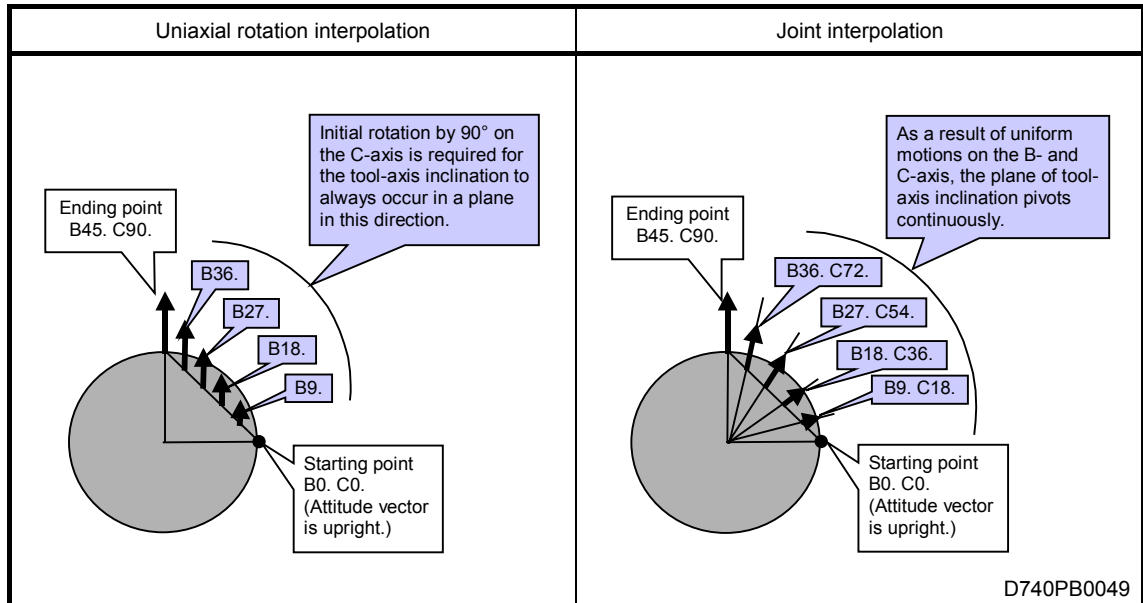
- a rotation by 90° occurs on the C-axis and, in synchronization with it, a movement of the tool tip point is performed to follow the starting point ([1] in Figure **B**),
- then the tool tip point is fed to the ending point on the workpiece with a synchronized tool axis rotation by 45° , as programmed, on the B-axis ([2] in Figure **B**).



As illustrated above, the uniaxial rotation interpolation may sometimes cause discontinuous machine actions, while the joint interpolation is executed in continuous and smooth motions from the start to the end of the block.

- Graphic explanation with tool's attitude vectors in the top view of the machining surface.

The interpolation of tool's attitude vector occurs in synchronization with the tip point's movement. The starting point, and the length, of the vector in the figure below refer to the position of the tool tip point, and the attitude (inclination) of the tool axis, respectively. (The shorter the vector, the uprigher [nearer to a position of $B = 0$] the tool.)



8. Selection for determining the angle on the rotational axis

A. Tool tip point control type 1 (G43.4)

As for tool tip point control type 1, movement on the rotational axis is carried out exactly as programmed (independently of the selected “type of passage through singular point” described later).

Because of the specified tool attitude being unobtainable, an alarm will be caused (**973 ILLEGAL TOOL AXIS VECTOR**) when the following three conditions are all met:

1. The method of uniaxial rotation interpolation is selected,
2. The position on the primary rotational axis differs in algebraic sign between the starting and ending points, and
3. A singular point (a state where the tool axis is parallel to the axis of rotation of the secondary rotational axis) is not passed as the starting point, nor on the way to the ending point.
(This applies when, for instance, a motion command for the secondary rotational axis is included in the same block, with condition 2 above being satisfied.)

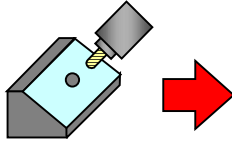
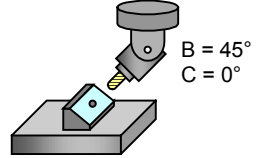
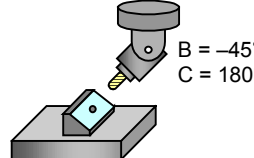
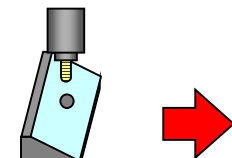
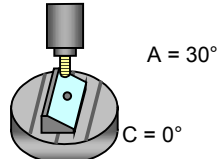
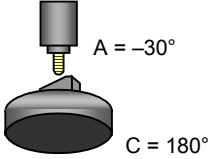
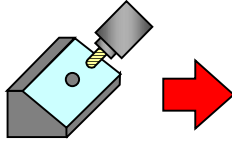
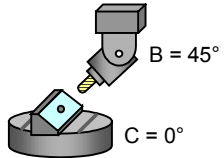
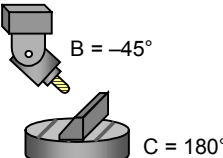
Remark: Definition of primary and secondary rotational axes

Tool rotating type	Table rotating type	Mixed type
<p>Secondary rotational axis</p> <p>Primary rotational axis</p> <p>Tool</p> <p>Workpiece</p> <p>Table</p>	<p>Tool</p> <p>Table</p> <p>Workpiece</p> <p>Primary rotational axis</p> <p>Secondary rotational axis</p>	<p>Primary rotational axis</p> <p>Tool</p> <p>Workpiece</p> <p>Table</p> <p>Secondary rotational axis</p> <p>D740PB0034</p>

B. Tool tip point control type 2 (G43.5)

<At the startup of tool tip point control>

As for tool tip point control type 2, a command for the tool-axis direction with I, J, and K can in general be executed by using either of the two pairs of angles on the rotational axes concerned.

	Programmed command	Pairs of angles on the rotational axes for obtaining the specified tool's attitude relative to the workpiece	
For tool rotating type	 I0.707J0K0.707	Solution with a positive angle of the B-axis  $B = 45^\circ$ $C = 0^\circ$	Solution with a negative angle of the B-axis  $B = -45^\circ$ $C = 180^\circ$
For table rotating type	 I0J0.866K0.5	Solution with a positive angle of the A-axis  $A = 30^\circ$ $C = 0^\circ$	Solution with a negative angle of the A-axis  $A = -30^\circ$ $C = 180^\circ$
For mixed type	 I0.707J0K0.707	Solution with a positive angle of the B-axis  $B = 45^\circ$ $C = 0^\circ$	Solution with a negative angle of the B-axis  $B = -45^\circ$ $C = 180^\circ$

D740PB0091

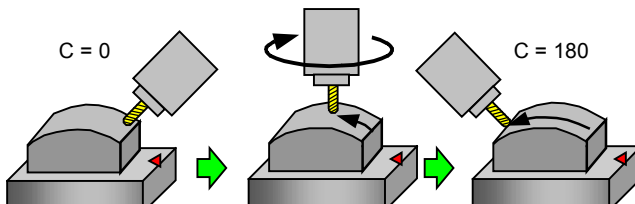
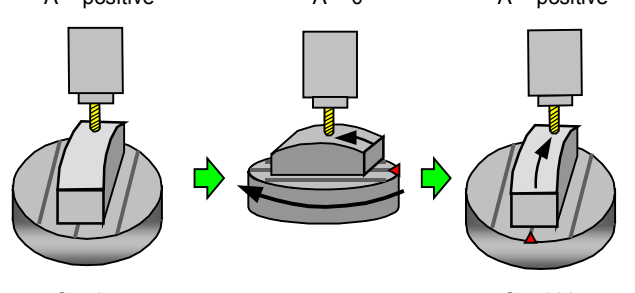
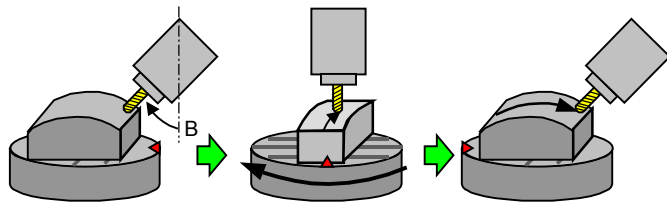
The appropriate one of the two solutions is internally selected by the following process:

- [1] Decision for the solution whose angle of the primary rotational axis has the same sign as its initial position (position at the startup of the tool tip point control by a G43.5 command),
- [2] If not decided in step [1] above (i.e. when the initial position on the primary rotational axis is equal to 0), then decision for the solution whose angle of the primary rotational axis has the same sign as the wider side of its axis stroke,
- [3] If not yet decided in step [2] above (i.e. when the positive and negative sides of the stroke on the primary rotational axis are equal to each other), then final decision for the solution whose angle of the primary rotational axis has a negative sign.

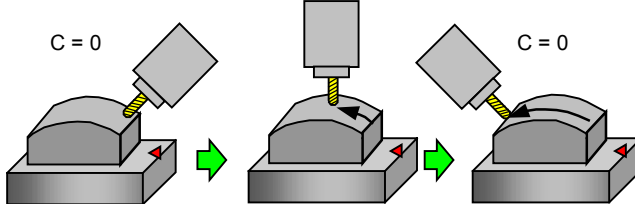
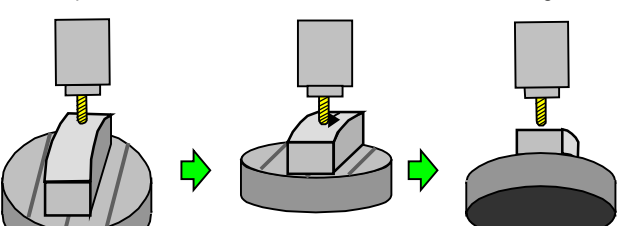
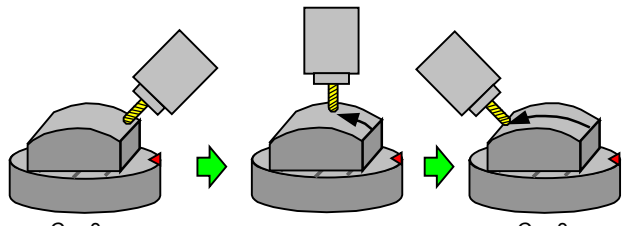
<In the mode of tool tip point control (on passage through singular point)>

In the mode of tool tip point control type 2 there are two types available for the passage through a singular point (a state where the tool axis is parallel to the axis of rotation of the secondary rotational axis), which are especially distinctive with regard to the angle of the rotational axis concerned at the ending point. Use parameter **F162** bit 1 to select the desired type.

- Type 1 of passage through singular point

	Operation	Selected is a simultaneous control of rotational axes which results in the ending point's angle of the primary rotational axis obtaining the same sign as its initial position (position at the startup by a G43.5 command).
F162 bit 1 = 1	Example	<p>The angle of the primary rotational axis (B- or A-axis in this example) has always the same sign (including 0).</p> <p>- For tool rotating type</p> <p>B = positive B = 0 B = positive</p> <p>C = 0 C = 180</p>  <p>- For table rotating type</p> <p>A = positive A = 0 A = positive</p> <p>C = 0 C = 180</p>  <p>- For mixed type</p> <p>B = positive B = 0 B = positive</p> <p>C = 0 C = 180</p>  <p>D740PB0092</p>

- Type 2 of passage through singular point

	Operation	<p>With respect to the sign of the ending point's angle of the primary rotational axis the appropriate method of axis control is internally selected by the following process:</p> <ol style="list-style-type: none"> [1] Decision for a method which does not require an angular position beyond the permissible motion range of rotational axis, [2] If not decided in step [1], then decision for a method which requires smaller distance of angular motion on the secondary rotational axis, (Short-cut method for secondary rotational axis) [3] If not decided in step [2], then decision for a method which requires smaller distance of angular motion on the primary rotational axis, (Short-cut method for primary rotational axis) [4] If not yet decided in step [3], then final decision for a method which results in the ending point's angle of the primary rotational axis obtaining the same sign as its initial position (position at the startup by a G43.5 command).
F162 bit 1 = 0	Example	<p>The ending point's angle of the primary rotational axis is determined in principle so that smaller distance of angular motion is required for the secondary rotational axis (C-axis in this example).</p> <p>- For tool rotating type</p> <p style="text-align: center;">B = positive B = 0 B = negative</p>  <p style="text-align: center;">C = 0 C = 0 C = 0</p> <p>- For table rotating type</p> <p style="text-align: center;">A = positive A = 0 A = negative</p>  <p style="text-align: center;">C = 0 C = 0 C = 0</p> <p>- For mixed type</p> <p style="text-align: center;">B = positive B = 0 B = negative</p>  <p style="text-align: center;">C = 0 C = 0 C = 0</p> <p style="text-align: right;">D740PB0093</p>

26-1-3 Relationship to other functions

1. Commands usable in the same block with G43.4/G43.5

Function	Code	Function	Code
Positioning	G00	Incremental data input	G91
Linear interpolation	G01	Feed function	F
Absolute data input	G90		

Giving any other command than those enumerated above in the same block as G43.4/G43.5 will lead to an alarm (**972 ILLEGAL CMD TOOL TIP PT CTRL**).

2. Relationship to other commands

A. Commands available in the mode of tool tip point control

Function	Code	Function	Code
Positioning	G00	Scaling OFF	G50
Linear interpolation	G01	Exact stop mode	G61
Circular interpolation	G02/G03 (*1) (*5) (*6)	Geometry compensation	G61.1 (*3)
		Cutting mode	G64
Dwell	G04	Macro call	G65
High-speed machining mode	G05 (*4)	Absolute data input	G90
Exact-stop	G09	Incremental data input	G91
Plane selection	G17/G18/G19	Inverse time feed	G93
Tool radius compensation OFF	G40	Feed per minute	G94
Tool radius compensation for five-axis machining (to the left)	G41.2 G41.4 G41.5	M, S, T, B output to opposite system	G112 (*2)
		Subprogram call/End of subprogram	M98/M99
		Feed function	F
Tool radius compensation for five-axis machining (to the right)	G42.2 G42.4 G42.5	M, S, T, B function	MSTB (*2)
		Local variables, Common variables, Operation commands (arithmetic operations, trigonometric functions, square root, etc), Control commands (IF ~ GOTO ~, WHILE ~ DO ~)	Macro instructions
Tool length offset (+/-)	G43/G44		
Tool position offset OFF	G49		

*1 A command for helical/spiral interpolation will lead to an alarm (**972 ILLEGAL CMD TOOL TIP PT CTRL**).

*2 Use of a tool function (T-code) in the mode of tool tip point control will lead to an alarm (**972 ILLEGAL CMD TOOL TIP PT CTRL**).

*3 Giving a G61.1 command with the geometry compensation being invalid for rotational axes will lead to an alarm (**972 ILLEGAL CMD TOOL TIP PT CTRL**).

*4 The fairing function for high-speed machining cannot be made effective. The related parameter (**F96** bit 1) is of no effect during tool tip point control.

*5 With the table coordinate system being selected for programming, giving a circular interpolation command will lead to an alarm.

*6 In combined use with the function for inclined-plane machining or workpiece setup error correction, giving a circular interpolation command will lead to an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**).

Giving any other command than those enumerated above in the mode of tool tip point control will lead to an alarm (**972 ILLEGAL CMD TOOL TIP PT CTRL**).

B. Modes in which tool tip point control is selectable

Function	Code	Function	Code
Positioning	G00	Cutting mode	G64
Linear interpolation	G01	Macro call	G65
High-speed machining mode OFF	G05P0	Modal macro call OFF	G67
Radius data input for X-axis ON	G10.9X0	3-D coordinate conversion ON	G68 (*2)
Programmed data setting OFF	G11	Inclined-plane machining	G68.2
Plane selection	G17/G18/G19		G68.3
Inch data input	G20		G68.4
Metric data input	G21	3-D coordinate conversion OFF	G69
Pre-move stroke check OFF	G23	Fixed cycle OFF	G80
Tool radius compensation OFF	G40	Absolute data input	G90
Control for normal-line direction OFF	G40.1	Incremental data input	G91
Tool length offset (+/-)	G43/G44	Inverse time feed	G93
Tool position offset OFF	G49	Feed per minute	G94
Head offset for five-surface machining OFF	G49.1	Constant surface speed control OFF	G97
Scaling OFF	G50	Initial point level return in hole-machining fixed cycles	G98
Mirror image OFF	G50.1		
Polygonal machining mode OFF	G50.2	R-point level return in hole-machining fixed cycles	G99
Selection of workpiece coordinate system, additional workpiece coordinate system	G54-59, G54.1	Single program multi-process control	G109
Workpiece setup error correction	G54.4	Cross machining control OFF	G111
Exact stop mode	G61	Hob milling mode OFF	G113
Geometry compensation	G61.1 (*1)		

*1 Giving a G43.4/G43.5 command under G61.1 with the geometry compensation being invalid for rotational axes will lead to an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**).

*2 A G43.4/G43.5 command can be given under G68 only when parameter **F168** bit 4 = 1 (Replacing the 3-D coordinate conversion command [G68] with inclined-plane machining command [G68.2]).

Giving a G43.4/G43.5 command in a mode other than those enumerated above will lead to an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**).

3. On the use of the MAZATROL tool data

The data settings of the **TOOL DATA** display (prepared for the execution of MAZATROL programs) can also be used for the tool tip point control. The table below indicates those usage patterns [1] to [4] of the externally stored tool offset data items which are applied to the tool tip point control according to the settings of the relevant parameters (**F93** bit 3 and **F94** bit 7).

Table 26-1 Tool offset data items used according to the parameter settings

Pattern	Data items used (Display and Data item names)		Parameter		Programming method
			F94 bit 7	F93 bit 3	
[1]	TOOL OFFSET	Offset data items	0	0	G43.4/G43.5 with H-code
[2]	TOOL DATA	LENGTH	1	1	T-code
		LENGTH + LENG. No. LENGTH + LENG. CO.			T-code + H-code
[3]	TOOL DATA	LENG. No. LENG. CO.	1	0	G43.4/G43.5 with H-code
[4]	TOOL OFFSET + TOOL DATA	Offset data items + LENGTH	0	1	G43.4/G43.5 with H-code + T-code

4. Change between Tip control and Length offset

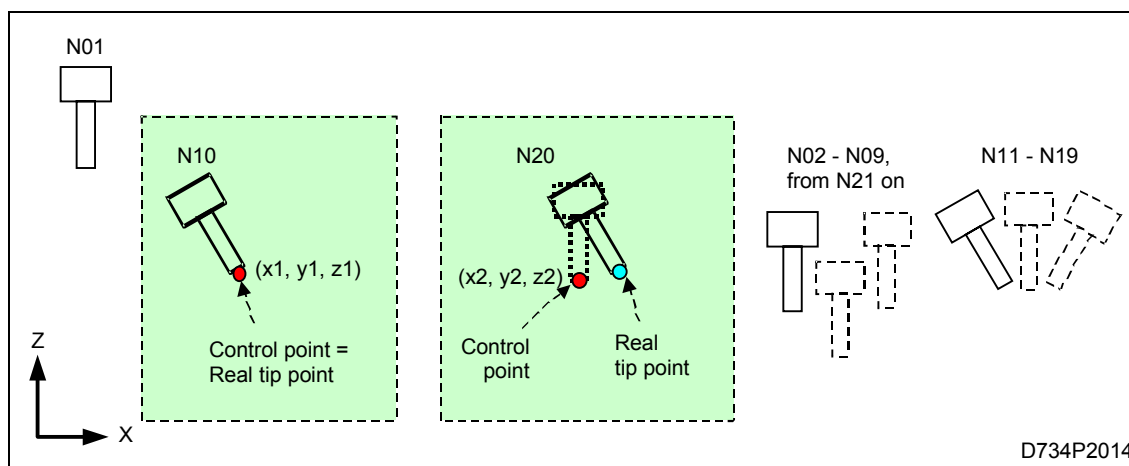
A. Immediate change between G43.4/G43.5 and G43/G44 (without using G49)

An immediate change of one mode for the other with a motion command added causes the “control point” to be moved to the specified position. The control point here refers to the real tool tip point in the case of G43.4 and G43.5 or, for G43 and G44, to an imaginary tip point which corresponds to a B-axis angle of 0°.

An independent change (without axis motion commands) does not include any change at all in axis positions, indeed, but causes the POSITION values (indicated with respect to the workpiece coordinate system) to be changed because of the above-mentioned difference in the meaning of the control point unless the current B-axis position is 0° (see the description under B below).

Shown below is a programming example with an explanatory figure for the use of the MAZATROL tool data (stored on the **TOOL DATA** display).

N01	T01 T02 M6	ATC includes selection of length offset function.	(Note 1)
N02	G01 X_Y_Z_F_		
...		Machining with length offset function (G43)	
...			
N10	G43.4 Xx1 Yy1 Zz1 Bb1	Tool tip point control ON	(Note 2)
N11	G01 X_Y_Z_B_C_		
...		Machining with tip point control (G43.4)	
...			
N20	G43 Xx2 Yy2 Zz2	Tool length offset ON	(Note 2)
N21	G01 X_Y_Z_B0		
...		Machining with length offset function (G43)	
...			



D734P2014

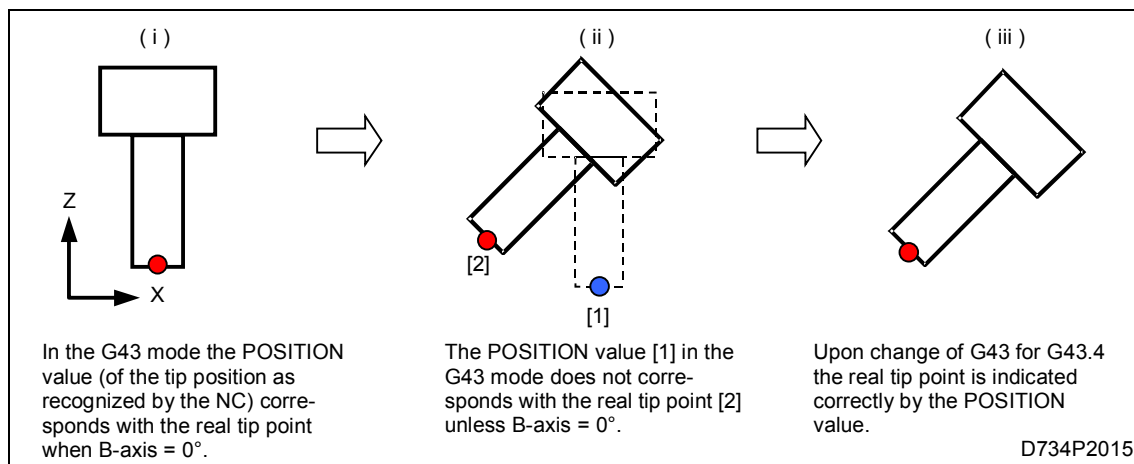
Note 1: When **F93** bit 3 = 1, the length offset function is automatically made active by each ATC operation according to the new tool's LENGTH value on the **TOOL DATA** display.

Note 2: Add an H-code as required to use as the offset amount the sum of the LENGTH value and another related setting (see Table 26-1 for details).

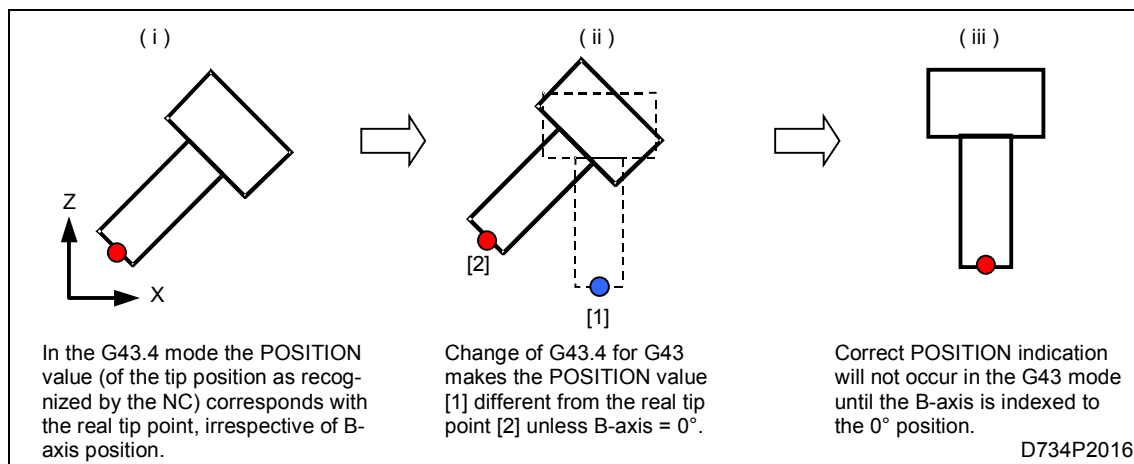
B. POSITION counting displayed on the screen

An immediate (without using G49) and independent (without axis motion commands) change between G43.4/G43.5 and G43/G44 brings about no actual machine motion (axis movements), but causes the POSITION values on the display to be changed unless the current B-axis position is 0°.

1. Change of G43/G44 for G43.4/G43.5



2. Change of G43.4/G43.5 for G43/G44

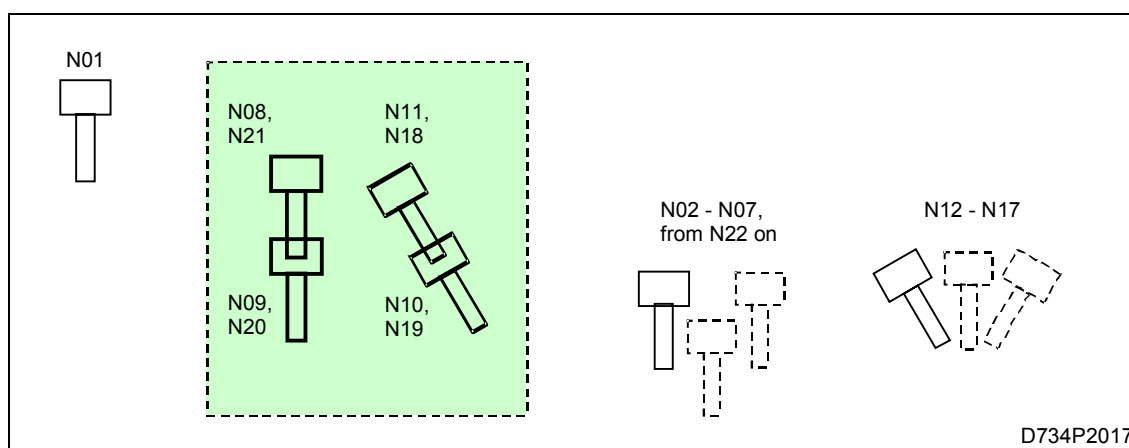


C. Change between G43.4/G43.5 and G43/G44 by means of G49

The execution of the cancellation command G49 includes axis movements for canceling the offset amount in general (see Note 4 below). Add, therefore, a motion command as required for a safety position before the G49 command is given as a means of mode change.

Shown below is a programming example with an explanatory figure for the use of the MAZATROL tool data (stored on the **TOOL DATA** display).

N01	T01 T02 M6	ATC includes selection of length offset function.	(Note 1)
N02	G01 X_Y_Z_F_		
...		Machining with length offset function (G43)	
...			
N08	G0 Xx1 Yy1 Zz1	Positioning to a safety position for canceling the length offset function.	
N09	G49	Canceling the length offset function in the safety position.	
N10	G0 Xx2 Yy2 Zz2 Bb2 Cc2	Positioning to a safety position for selecting the tip point control.	
N11	G43.4	Selecting the tip point control in the safety position.	(Note 1, 2)
N12	G01 X_Y_Z_B_C_		
...		Machining with tip point control (G43.4)	
...			
N18	G0 Xx3 Yy3 Zz3 Bb3 Cc3	Positioning to a safety position for canceling the tip point control.	
N19	G49	Canceling the tip point control in the safety position.	(Note 4)
N20	G0 Xx4 Yy4 Zz4 Bb4 Cc4	Positioning to a safety position for selecting the length offset function.	
N21	G43	Selecting the length offset function in the safety position.	(Note 1, 3)
N22	G01 X_Y_Z_B0		
...		Machining with length offset function (G43)	
...			



D734P2017

Note 1: When **F93** bit 3 = 1, the length offset function is automatically made active by each ATC operation according to the new tool's LENGTH value on the **TOOL DATA** display.

Note 2: Add an H-code as required to use as the offset amount the sum of the LENGTH value and another related setting (see Table 26-1 for details).

Note 3: Irrespective of the related parameters (**F94** bit 7 and **F93** bit 3), the execution of a G49 command always clears the currently used offset amount. Do not fail, therefore, to give a G43 command or a tool change command (T_T_M6) again as required to replace the tip point control when the automatic selection of the length offset function is used (with **F93** bit 3 = 1).

Note 4: Axis movements for canceling the offset amount do not occur by the execution of an independent command of G49 unless **F114** bit 1 = 0.

5. Prefiltering for rotational axes

The tool tip point control may sometimes require a series of fluctuating angular movements which, in turn, necessitates frequent alternation of acceleration and deceleration, resulting in the surface quality being deteriorated. The function of prefiltering for rotational axes applies a moving average to the amounts of serial angular movements to smooth fluctuations of tool attitude.

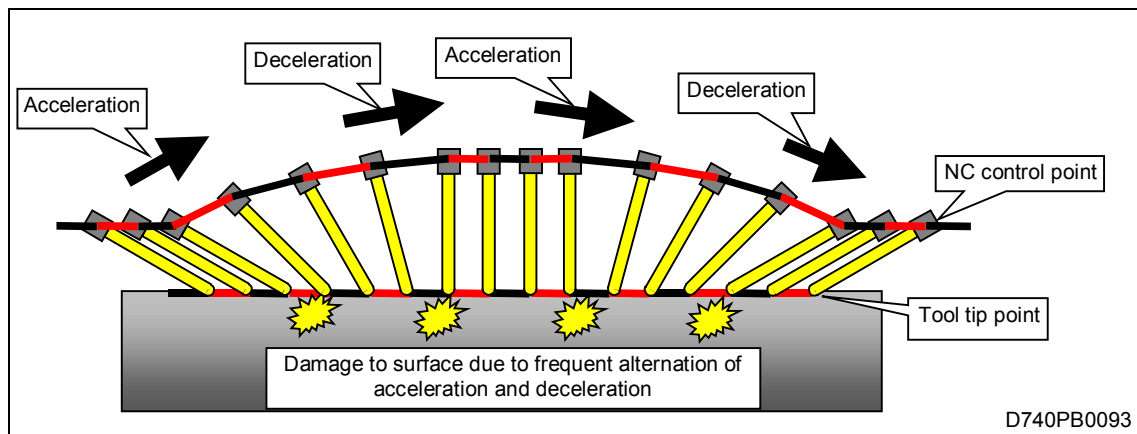
The time constant used in prefiltering for rotational axes can be set in parameter **L125**. The higher the time constant is, the more smoothly the tool attitude is changed. The prefiltering in question cannot be made effective at all when the time constant (**L125**) is set to zero (0).

The prefiltering function works with its time constant (**L125**) on condition that

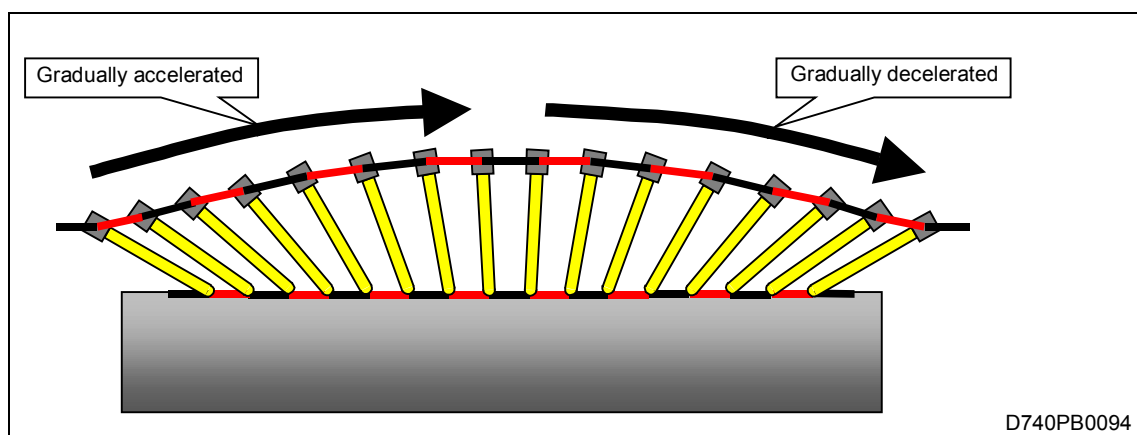
- high-speed smooth interpolation is valid (G5P2, G61.1, **F3** = 1),
- **F36** bit 7 is set to 1 (Prefiltering for rotational axes is made valid),
- tool tip point control is ON (under G43.4 or G43.5), and
- the mode of cutting feed, or interpolation, is ON.

* The setting in **L125** is exclusively used in prefiltering for rotational axes.

- Without prefiltering



- With prefiltering



26-1-4 Restrictions

1. The selection code of tool tip point control (G43.4/G43.5) must not be given together with any other G-code.
2. Calculation of the machining time
The machining time cannot be calculated accurately for a program containing tool tip point control commands.
3. Tracing
Tracing in the mode of tool tip point control is displayed with reference to the machine coordinate system.
4. Tool path check
Tool path check function cannot be applied to a program containing tool tip point control commands. (The programmed data can only be checked as if tool tip point control were throughout cancelled.)
5. Restart
As for restarting operation, do not specify a block within the G43.4/G43.5 mode (nor a block of cancellation [G49]) as a restarting position. Otherwise an alarm will be caused (**956 RESTART OPERATION NOT ALLOWED**).
6. Resetting
The mode of tool tip point control is cancelled by resetting.
7. Corner chamfering/rounding
Do not use corner chamfering or rounding commands in the mode of tool tip point control. Otherwise an alarm will be caused (**972 ILLEGAL CMD TOOL TIP PT CTRL**).
8. Mirror image function (activated by a code or external switch)
Mirror image function is not available at all in the mode of tool tip point control.
9. Macro interruption and MDI interruption
Macro interruption as well as MDI interruption is not available in the mode of tool tip point control.
Giving a G43.4/G43.5 command with macro interruption being active will lead to an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**), while an attempt to perform MDI interruption in the mode of tool tip point control will only cause another alarm (**167 ILLEGAL OPER TOOL TIP PT CTRL**).
10. Display of actual feed rate
The feed rate displayed is the final resultant rate of feed, not the feed rate at the tool tip with respect to the workpiece.
11. Manual interruption
As a general precaution to be taken for normal machine operation, resetting is required after any manual interruption (resumption of operation is prohibited).
12. The relevant items of digital information on the **POSITION** display denote the following in the mode of tool tip point control:

POSITION:	Position of the tool tip point in the workpiece coordinate system
MACHINE:	Position of the control point in the machine coordinate system
BUFFER:	Moving distance of the control point in the next block
REMAIN:	Remaining distance of movement of the control point in the current block
13. Tool function
Use of a tool function (T-code) in the mode of tool tip point control will lead to an alarm (**972 ILLEGAL CMD TOOL TIP PT CTRL**).

14. Calling a MAZATROL program as a subprogram
Do not specify a MAZATROL program as a subprogram to be called up in the mode of tool tip point control. Otherwise an alarm will be caused (**972 ILLEGAL CMD TOOL TIP PT CTRL**).
15. Circular interpolation
Circular interpolation can only be applied when the current tool tip point control is of type 1 (G43.4) with workpiece coordinate system being selected for programming. In other cases giving a command for circular interpolation will cause an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**). Moreover, even thus available circular interpolation mode cannot accept any commands for rotational axes, and such an inadmissible command will result in the alarm **972 ILLEGAL CMD TOOL TIP PT CTRL**.
16. MAZATROL tool data
When the MAZATROL tool data (data prepared on the **TOOL DATA** display) are made valid for executing EIA/ISO programs, milling tools should generally be used in the mode of tool tip point control. Tool length offset for tool tip point control will be executed for turning tools with their LENGTH B and NOSE-R data being ignored.
17. High-speed machining mode
The fairing function for high-speed machining cannot be made effective. The related parameter (**F96** bit 1) is of no effect during tool tip point control.
18. Positioning commands (under G0) in the mode of tool tip point control are always executed in interpolation-type paths even if **F91** bit 6 is set to 1.
19. Prefiltering for rotational axes
When the function of prefiltering for rotational axes is made valid, the maximum permissible acceleration for rotational axes is increased according to the time constant concerned (as set in **L125**).
20. The acceleration and deceleration for positioning commands (under G0) in the mode of tool tip point control occurs according to the parameters **L74** (rate of cutting feed for pre-interpolational acceleration/deceleration control) and **L75** (time constant for pre-interpolational cutting feed) when they are given under G61.1 (geometry compensation) or with the (optional) fixed gradient control for G0 being selected.

26-1-5 Related parameters

1. Offset for the axis of rotation of the rotational axis (for table rotating type)

A. Offset for the axis of rotation of the first rotational axis

Parameter	Contents	Setting range	Setting unit	Remarks
K122	Axis-of-rotation position on the horizontal axis (X)	0 - 99999999	0.0001 mm	
K123	Axis-of-rotation position on the vertical axis (Y)	0 - 99999999	0.0001 mm	
K124	Axis-of-rotation position on the height axis (Z)	0 - 99999999	0.0001 mm	

B. Offset for the axis of rotation of the second rotational axis

Parameter	Contents	Setting range	Setting unit	Remarks
K126	Axis-of-rotation position on the horizontal axis (X)	0 - 99999999	0.0001 mm	To be omitted
K127	Axis-of-rotation position on the vertical axis (Y)	0 - 99999999	0.0001 mm	To be omitted
K128	Axis-of-rotation position on the height axis (Z)	0 - 99999999	0.0001 mm	To be omitted

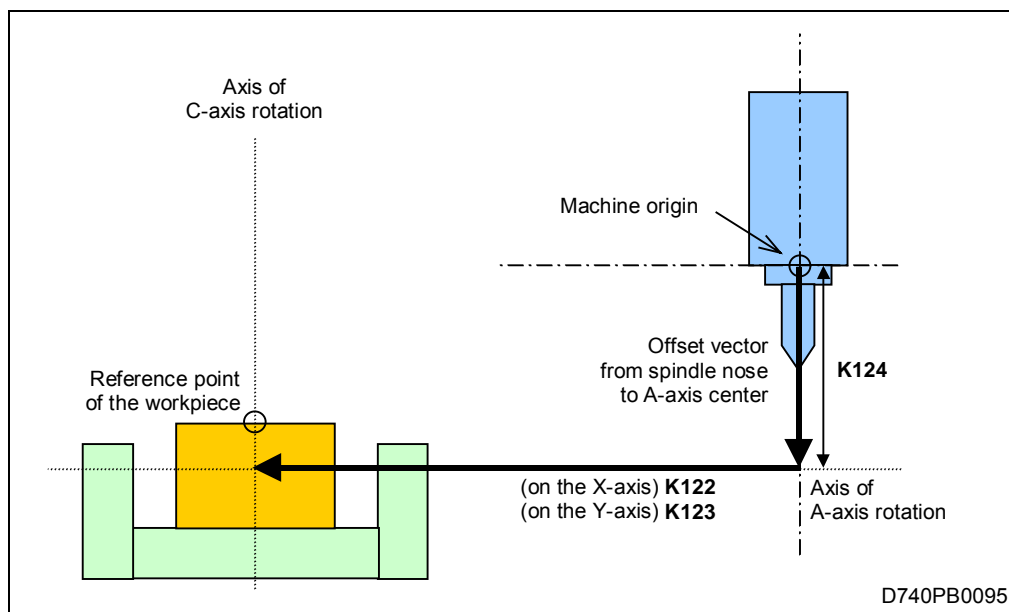


Fig. 26-1 For table rotating type

2. Offset for the axis of rotation of the rotational axis (for tool rotating type)

A. Offset for the axis of rotation of the first rotational axis

Parameter	Contents	Setting range	Setting unit	Remarks
K122	Axis-of-rotation position on the horizontal axis (X)	0 - 99999999	0.0001 mm	To be omitted
K123	Axis-of-rotation position on the vertical axis (Y)	0 - 99999999	0.0001 mm	To be omitted
K124	Axis-of-rotation position on the height axis (Z)	0 - 99999999	0.0001 mm	To be omitted

B. Offset for the axis of rotation of the second rotational axis

Parameter	Contents	Setting range	Setting unit	Remarks
K126	Axis-of-rotation position on the horizontal axis (X)	0 - 99999999	0.0001 mm	To be omitted
K127	Axis-of-rotation position on the vertical axis (Y)	0 - 99999999	0.0001 mm	To be omitted
K128	Axis-of-rotation position on the height axis (Z)	0 - 99999999	0.0001 mm	

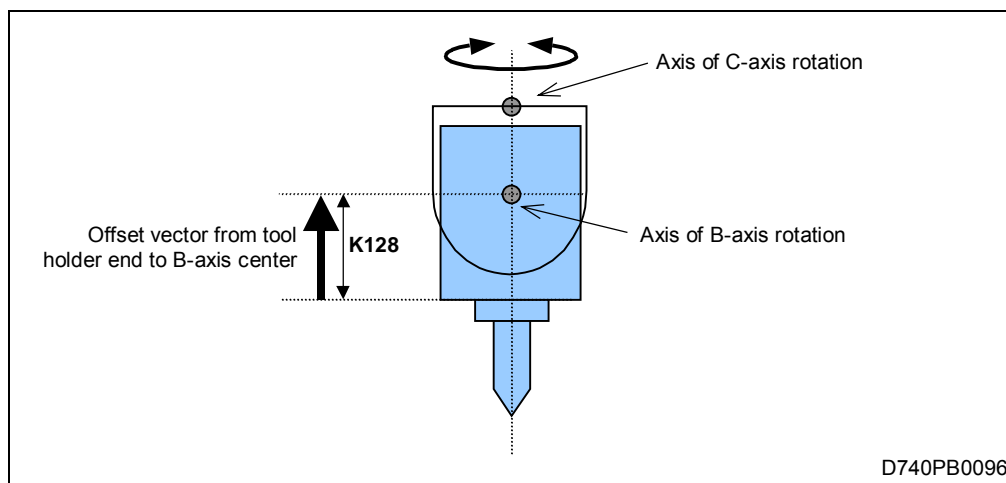


Fig. 26-2 For tool rotating type

3. Selection of programming coordinate system

Parameter	Setting range
F85 bit 2	0: Selection of the table coordinate system 1: Selection of the workpiece coordinate system

4. Selection of interpolation method for tool tip point control

Parameter	Setting range
F85 bit 3	0: Uniaxial rotation interpolation 1: Joint interpolation

5. Selection of override scheme for tool tip point control

Parameter	Setting range
F86 bit 2	Overriding the rapid traverse in the mode of tool tip point control is applied 0: to the speed of motion of the tool tip point 1: to the speed limit for the axis control's reference point
F86 bit 5	Overriding the cutting feed in the mode of tool tip point control is applied 0: to the speed of motion of the tool tip point 1: to the speed limit for the axis control's reference point

6. Selection of reference position on the rotational axis for tool tip point control

Parameter	Setting range
F86 bit 6	0: Angular position at the start of tool tip point control 1: Position of 0°

7. Selection of canceling operation by G49 in the mode of tool tip point control

Parameter	Setting range
F114 bit 1	0: Cancellation with axis movement for clearing the length offset amount 1: Cancellation without axis movement

8. Selection of operation for starting up the tool tip point control without motion commands

Parameter	Setting range
F162 bit 0	0: Startup with axis movement by the length offset amount 1: Startup without axis movement

9. Selection of type of passage through singular point for tool tip point control

Parameter	Setting range
F162 bit 1	0: Selected is a simultaneous control of rotational axes which requires, for passing through the singular point, smaller distance of angular motion on the secondary rotational axis. (The motion on the primary rotational axis may use both positive and negative sides of the stroke.) 1: Selected is a simultaneous control of rotational axes which does not require, throughout one block, any motion on the primary rotational axis into the opposite side of its stroke to that of its initial position (position at the startup).

10. Limit of feed rate for the mode of tool tip point control

Set the maximum admissible rate of cutting feed for the mode of tool tip point control.

The cutting feed during tool tip point control will be limited by parameter **M3** (general limit of feed rate) if it is lower than the setting of parameter **S22**.

Note: Set zero (0) in **S22** to make the parameter invalid. If both parameters **S22** and **M3** are set to zero (0), then parameter **M1** (for rapid traverse) serves as the speed limit in question.

Parameter	Setting range	Setting unit	
S22	0 - 200000	Linear axis	1 mm/min
		Rotational axis	1 deg/min

11. Criterion for the neighborhood of singular point

Parameter	Setting range	Setting unit
K110	0 - 360	1 deg

12. Effective decimal digits in arguments I, J, and K for G43.5 (tool tip point control type 2)

Parameter	Setting range
F36 bit 6	Number of effective decimal digits in arguments I, J, K for tool tip point control type 2 0: 4 or 5 for the metric or inch system 1: 7 (irrespective of the system of units)

13. Function of prefiltering for rotational axes valid/invalid

Parameter	Setting range
F36 bit 7	Function of prefiltering for rotational axes 0: Invalid 1: Valid

14. Time constant for prefiltering for rotational axes

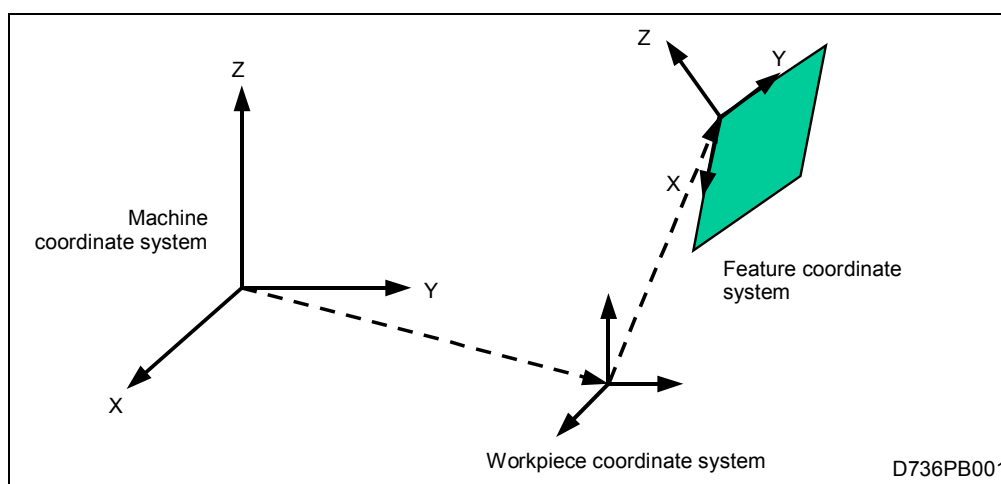
Parameter	Setting range	Setting unit
L125	0 - 200	1 ms

Note: The prefiltering cannot be made effective at all when **L125** is set to zero (0).

26-2 Inclined-Plane Machining: G68.2, G68.3, G68.4, G53.1 (Option)

The inclined-plane machining function makes it possible to define a new coordinate system (referred to as a feature coordinate system) by translating as well as rotating the current workpiece coordinate system around the X-, Y- and Z-axis. Using this function, therefore, an arbitrarily inclined plane can be defined in a space and the desired machining contour can be programmed easily as if the plane concerned were an ordinary XY-plane.

In addition, there is a function provided to adjust the direction of tool axis to the positive direction of the Z-axis of a feature coordinate system. Since the feature coordinate system is then redefined according to the change in tool-axis direction, a machining program can be prepared without consideration of the inclination of the coordinate system or the tool axis.



26-2-1 Function description

1. Command codes for inclined-plane machining

A. Inclined-plane machining ON (startup)

The feature coordinate system is to be set, or defined, by the following methods at the beginning of a program section for inclined-plane machining.

Setting method	Programming format
Using Eulerian angles	G68.2 [P0]
Using roll, pitch, and yaw angles	G68.2 P1
Using three points in the plane	G68.2 P2
Using two vectors	G68.2 P3
Using projection angles	G68.2 P4
Using tool-axis direction	G68.3

- Argument P can be omitted if it is 0 (for the use of Eulerian angles).
- If any number other than 0 to 4 is entered with P in a G68.2 block, an alarm will be caused (**1809 TILTED PLANE CMD FORMAT ERROR**).
- Argument P or Q with a decimal point in a G68.2 block will be processed by simply deleting the decimal point and decimal digits.
- Be sure to enter the G68.2 or G68.3 command independently. If a block of G68.2 or G68.3 should contain any other G-codes or motion commands, an alarm is caused (**1809 TILTED PLANE CMD FORMAT ERROR**).

B. Tool-axis direction control

The G53.1 command causes motions on the rotational axes concerned so as to make parallel to, and of the same direction with, each other the direction of the tool axis (from the tip along the perpendicular to the tool-swiveling axis) and the positive direction of the Z-axis of the current feature coordinate system.

G53.1 Pp

G53.1: Tool-axis direction control

P: Selection of a solution for the axis of rotation. (See the description under 3.)

- 0: Processed as "P = 1" or "P = 2", depending on the construction of the machine.
Works the same as "P = 1" and "P = 2", respectively, for machines of tool rotating type and table rotating type.
- 1: Solution with a positive angle of rotation on the C-axis (for VARIAXIS),
Solution with a positive angle of rotation on the B-axis (for VERSATECH).
- 2: Solution with a negative angle of rotation on the C-axis (for VARIAXIS),
Solution with a negative angle of rotation on the B-axis (for VERSATECH).

1. Enter the G53.1 command in the G68.2 mode.
2. Be sure to enter the G53.1 command independently. If a block of G53.1 should contain any other G-codes or motion commands, an alarm is caused (**1808 CANNOT USE G53.1**).
3. The motion caused by a G53.1 block is executed in the currently active mode of motion.
4. Argument P can be omitted if it is zero (0). If any number other than 0, 1 or 2 is entered with P, an alarm will be caused (**809 ILLEGAL NUMBER INPUT**).
5. Use of any other addresses than P will lead to an alarm (**1808 CANNOT USE G53.1**).

C. Inclined-plane machining OFF (cancellation)

G69 Inclined-plane machining OFF

1. Be sure to enter the G69 command independently. If a block of G69 should contain any other G-codes or motion commands, an alarm is caused (**1809 TILTED PLANE CMD FORMAT ERROR**).
2. Do not enter the cancellation command (G69) in the mode of circular interpolation or fixed cycles; otherwise an alarm is caused (**1807 CANNOT USE G68.2**).

2. Various methods for selecting the inclined-plane machining mode

2-1 Setting with Eulerian angles

A. Programming format

G68.2 [P0] Xx Yy Zz I α J β K γ

G68.2 [P0]: Inclined-plane machining ON (Setting with Eulerian angles)

x, y, z: Coordinates of the origin of feature coordinate system. To be specified with its absolute values in the currently active workpiece coordinate system.

α , β , γ : Eulerian angles (to describe the orientation of the feature coordinate system).
Setting range: -360° to 360°

1. The designation of X, Y, or Z can be omitted when the argument is zero (0: no shift along the particular axis). Set zero for X, Y, and Z if no translation of the origin is required.
2. The designation of I, J, or K can be omitted when the argument is zero (0: no rotation around the axis concerned).

3. Use of any other addresses than P, X, Y, Z, I, J, and K will lead to an alarm (**1809 TILTED PLANE CMD FORMAT ERROR**).

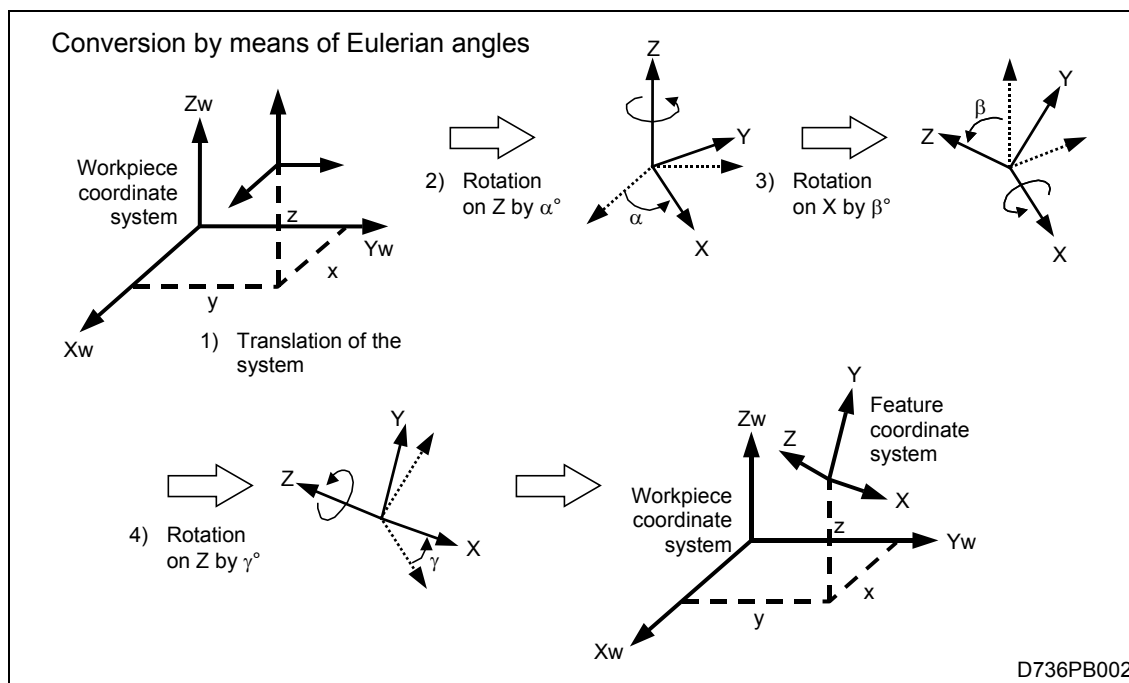
B. Setting a feature coordinate system

Enter a block of G68.2 [P0] (with arguments for translating and rotating the current workpiece coordinate system) to set a feature coordinate system. Use Eulerian angles to specify the rotation.

G68.2 Xx Yy Zz I α J β K γ sets a feature coordinate system as follows:

- 1) Point (x, y, z) in the current workpiece coordinate system is set as a new origin.
- 2) The translated coordinate system is then rotated around the Z-axis by an angle of α° .
- 3) The turned coordinate system is then rotated around the X-axis by an angle of β° .
- 4) Finally, the coordinate system is rotated around the Z-axis by an angle of γ° .

A counterclockwise rotation when viewed from the positive side of the axis of rotation to the origin is to be specified with a positive value (in degrees). The figure below shows the relationship between a workpiece coordinate system and its feature coordinate system.



2-2 Setting with roll, pitch, and yaw angles

A. Programming format

G68.2 P1 Qq Xx Yy Zz I α J β K γ

G68.2 P1: Inclined-plane machining ON (Setting with roll, pitch, and yaw angles)

x, y, z: Coordinates of the origin of feature coordinate system. To be specified with its absolute values in the currently active workpiece coordinate system.

q: Order of rotations

q	Axis of the first rotation	Axis of the second rotation	Axis of the third rotation
123	X-axis	Y-axis	Z-axis
132	X-axis	Z-axis	Y-axis
213	Y-axis	X-axis	Z-axis
231	Y-axis	Z-axis	X-axis
312	Z-axis	X-axis	Y-axis
321	Z-axis	Y-axis	X-axis

α : Angle of rotation around the X-axis [Roll angle].

β : Angle of rotation around the Y-axis [Pitch angle].

γ : Angle of rotation around the Z-axis [Yaw angle].

(Setting range: -360° to 360°)

1. The designation of X, Y, or Z can be omitted when the argument is zero (0: no shift along the particular axis). Set zero for X, Y, and Z if no translation of the origin is required.
2. The designation of I, J, or K can be omitted when the argument is zero (0: no rotation around the axis concerned).
3. Use of any other addresses than P, Q, X, Y, Z, I, J, and K will lead to an alarm (**1809 TILTED PLANE CMD FORMAT ERROR**).
4. Argument Q can be omitted if Q123 is required.
5. The designation of Q with any other value than enumerated above will lead to an alarm (**1809 TILTED PLANE CMD FORMAT ERROR**).

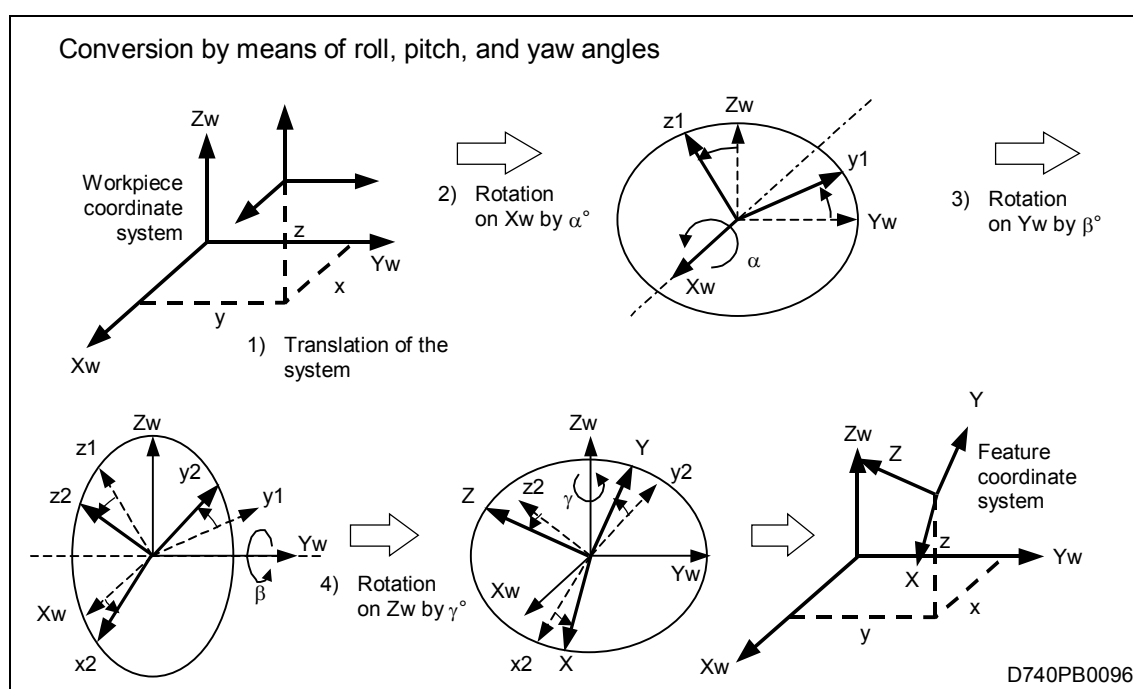
B. Setting a feature coordinate system

Enter a block of G68.2 P1 (with arguments for translating and rotating the current workpiece coordinate system) to set a feature coordinate system. Use roll, pitch, and yaw angles to specify the rotation.

G68.2 P1 Q123 Xx Yy Zz I α J β K γ sets, for example, a feature coordinate system as follows:

- 1) Point (x, y, z) in the current workpiece coordinate system is set as a new origin.
- 2) The translated coordinate system is then rotated around the Xw-axis by an angle of α° .
- 3) The turned coordinate system is then rotated around the Yw-axis by an angle of β° .
- 4) Finally, the coordinate system is rotated around the Zw-axis by an angle of γ° .

A counterclockwise rotation when viewed from the positive side of the axis of rotation to the origin is to be specified with a positive value (in degrees). The figure below shows the relationship between a workpiece coordinate system and its feature coordinate system.



2-3 Setting with three points in the plane

A. Programming format

[G68.2 P2 Q0 Xx0 Yy0 Zz0 R α]

G68.2 P2 Q1 Xx1 Yy1 Zz1

G68.2 P2 Q2 Xx2 Yy2 Zz2

G68.2 P2 Q3 Xx3 Yy3 Zz3

G68.2 P2: Inclined-plane machining ON (Setting with three points in the plane)

Q: Point designation (for point 1 to 3, or for shifting amounts).

x0, y0, z0: Amounts of shift from point 1 to the definite origin of feature coordinate system.

To be specified with their incremental values from the origin of the feature coordinate system to be established.

α : Angle of rotation of the feature coordinate system around the Z-axis.

Setting range: -360° to 360°

x1, y1, z1: Coordinates of point 1 (origin of feature coordinate system). To be specified with its absolute values in the currently active workpiece coordinate system.

x2, y2, z2: Coordinates of point 2 (a point on the feature coordinate system's X-axis [on the positive side]). To be specified with its absolute values in the currently active workpiece coordinate system.

x3, y3, z3: Coordinates of point 3 for plane definition. To be specified with its absolute values in the currently active workpiece coordinate system.

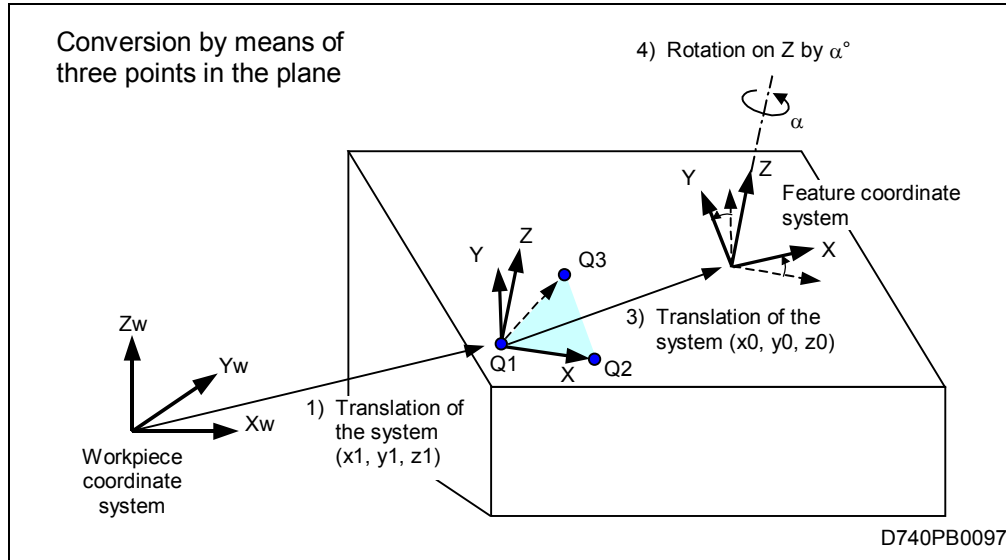
1. Argument Q can be omitted if Q0 (for designation of shifting amounts) is required.
2. The designation of X, Y, or Z can be omitted when the argument is zero.
3. The designation of R can be omitted when the argument is zero (0: no rotation).
4. Use of any other addresses than P, Q, X, Y, Z, and R will lead to an alarm (**1809 TILTED PLANE CMD FORMAT ERROR**).
5. Alarm **1809 TILTED PLANE CMD FORMAT ERROR** occurs in the following cases:
 - A series of blocks from G68.2 P2 Q0 to Q3 is interrupted by any other block,
 - Any of the blocks from G68.2 P2 Q1 to Q3 is wanting,
 - Any of the blocks from G68.2 P2 Q0 to Q3 is overlapping,
 - Argument Q is of any other value than 0 to 3,
 - Argument R is entered in multiple blocks.
6. Alarm **1810 TILTED PLANE CANNOT BE DEFINED** occurs in the following cases:
 - Any pair of the three points 1 to 3 denotes one and the same point,
 - All the three points 1 to 3 lie on one and the same line,
 - The distance between any one of the three points and the line through the other points should be less than 0.1 mm.

B. Setting a feature coordinate system

G68.2 P2 blocks set a feature coordinate system as follows:

- 1) Point (x1, y1, z1) in the current workpiece coordinate system is set as a new origin.
- 2) The feature coordinate system's X-, Y-, and Z-axis are determined as follows:
 The X-axis direction is set to the direction from point 1 (Q1) to point 2 (Q2).
 The Z-axis direction is then determined by the cross product $(Q2 - Q1) \times (Q3 - Q1)$.
 Finally, the Y-axis is set in accordance with the rule of right-handed system.

- 3) The feature coordinate system is then translated by the shifting amounts (x_0 , y_0 , z_0), if specified in a Q0 block. The desired new position of the origin must be specified with incremental values from the origin of the feature coordinate system to be established.
- 4) Finally, the feature coordinate system is rotated around the Z-axis by an angle of α° , if specified as argument R in a Q0 block.

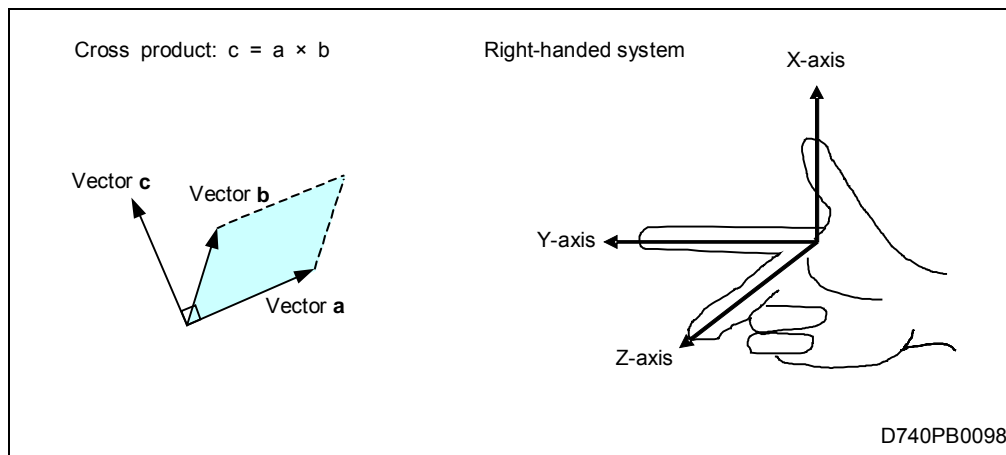


Remark 1: Cross product

The cross product of two ordered vectors **a** and **b** (denoted by $\mathbf{a} \times \mathbf{b}$) is another vector **c** which is perpendicular to the plane containing the given two vectors, with a direction determined in accordance with the rule of right-handed system, and has a magnitude equal to the parallelogram that the vectors span.

Remark 2: Right-handed system

Right-handed system refers to one of the methods of defining the three-dimensional axes of rectangular coordinates. The orientation of the X-, Y-, and Z-axis of a right-handed coordinate system corresponds to the directions pointed by the right hand's thumb, index, and middle fingers, in that order, at right angles to one another (with the thumb and the index finger lying in the same plane with the palm).



2-4 Setting with two vectors

A. Programming format

G68.2 P3 Q1 Xx Yy Zz Iix Jjx Kkx

G68.2 P3 Q2 Iiz Jjz Kkz

G68.2 P3: Inclined-plane machining ON (Setting with two vectors)

Q: Vector designation (for the X- or Z-axis direction).

x, y, z: Coordinates of the origin of feature coordinate system. To be specified with its absolute values in the currently active workpiece coordinate system.

ix, jx, kx: Components of the vector representing the X-axis direction of the feature coordinate system. To be specified (in dimensionless numbers) with respect to the currently active workpiece coordinate system.

Setting range: as specified for the orthogonal axes.

iz, jz, kz: Components of the vector representing the Z-axis direction of the feature coordinate system. To be specified (in dimensionless numbers) with respect to the currently active workpiece coordinate system.

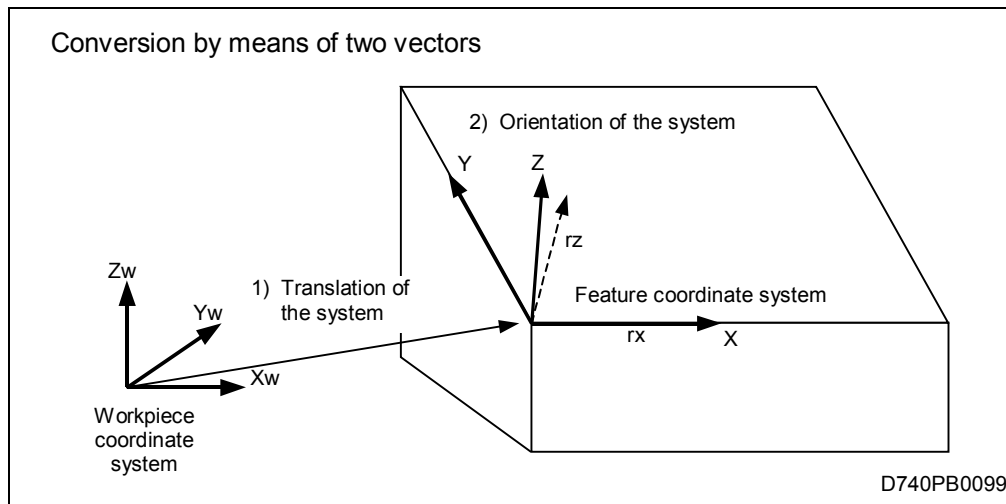
Setting range: as specified for the orthogonal axes.

1. The designation of X, Y, or Z can be omitted when the argument is zero (0: no shift along the particular axis). Set zero for X, Y, and Z if no translation of the origin is required.
2. The designation of I, J, or K can be omitted when the argument is zero.
3. Use of any other addresses than P, Q, I, J, and K will lead to an alarm (**1809 TILTED PLANE CMD FORMAT ERROR**).
X, Y, and Z can be additionally used in a block of G68.2P3Q1.
4. Alarm **1809 TILTED PLANE CMD FORMAT ERROR** occurs in the following cases:
 - The successive blocks G68.2 P3 Q1 and Q2 are interrupted by any other block,
 - Either of the blocks G68.2 P3 Q1 and Q2 is wanting,
 - Either of the blocks G68.2 P3 Q1 and Q2 is overlapping,
 - Argument Q is of any other value than 1 and 2,
 - Argument Q is omitted.
5. Alarm **1810 TILTED PLANE CANNOT BE DEFINED** occurs in the following cases:
 - All the values ix, jx, kx amount to zero (0),
 - All the values iz, jz, kz amount to zero (0),
 - The angle that is formed by the vectors specified for the feature coordinate system's X- and Z-axis deviates by 5° or more from the perpendicularity.

B. Setting a feature coordinate system

G68.2 P3 blocks set a feature coordinate system as follows:

- 1) Point (x, y, z) in the current workpiece coordinate system is set as a new origin.
- 2) The feature coordinate system's X-, Y-, and Z-axis are determined as follows:
 The X-axis direction is set to the direction of the vector $rx = (ix, jx, kx)$.
 The Y-axis direction is then determined by the cross product $(iz, jz, kz) \times (ix, jx, kx)$.
 Finally, the Z-axis is set in accordance with the rule of right-handed system.



2-5 Setting with projection angles

A. Programming format

G68.2 P4 Xx Yy Zz Iα Jβ Kγ

G68.2 P4: Inclined-plane machining ON (Setting with projection angles)

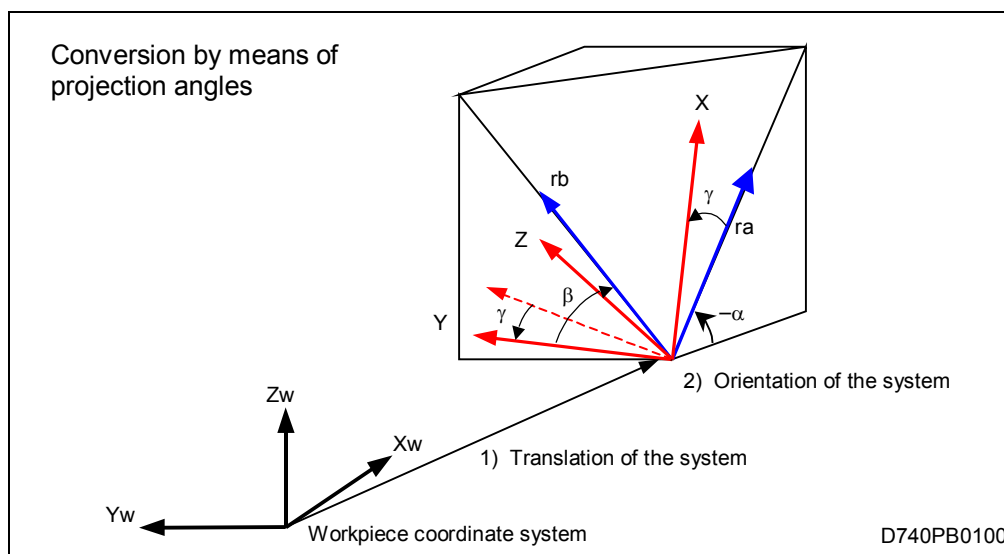
- x, y, z: Coordinates of the origin of feature coordinate system. To be specified with its absolute values in the currently active workpiece coordinate system.
- α: Angle of rotating the X-axis around the Y-axis of the current workpiece coordinate system.
- β: Angle of rotating the Y-axis around the X-axis of the current workpiece coordinate system.
- γ: Angle of rotation around the Z-axis of the feature coordinate system.
- (Angle setting range: -360° to 360°)

1. The designation of X, Y, or Z can be omitted when the argument is zero (0: no shift along the particular axis). Set zero for X, Y, and Z if no translation of the origin is required.
2. The designation of I, J, or K can be omitted when the argument is zero (0: no rotation around the axis concerned).
3. Use of any other addresses than P, X, Y, Z, I, J, and K will lead to an alarm (**1809 TILTED PLANE CMD FORMAT ERROR**).
4. Alarm **1810 TILTED PLANE CANNOT BE DEFINED** occurs if the angle that is formed by the X-axis rotated around the Y-axis by $-\alpha^\circ$, on the one side, and the Y-axis rotated around the X-axis by β° , on the other side, should be 1° or less.

B. Setting a feature coordinate system

A block of G68.2 P4 sets a feature coordinate system as follows:

- 1) Point (x, y, z) in the current workpiece coordinate system is set as a new origin.
- 2) The feature coordinate system's X-, Y-, and Z-axis are determined as follows:
 Assume that the direction of the X-axis rotated by $-\alpha^\circ$ around the Y-axis of the current workpiece coordinate system be **ra**, and that the direction of the Y-axis rotated by β° around the X-axis of the current workpiece coordinate system be **rb**.
 The direction of the feature coordinate system's Z-axis is determined by the cross product **ra** \times **rb**.
 The X-axis direction is then set to the direction of **ra** resulting from a rotation around the Z-axis by γ° .
 Finally, the Y-axis is set in accordance with the rule of right-handed system.



2-6 Setting with tool-axis direction

A. Programming format

G68.3 Xx Yy Zz Rα

G68.3: Inclined-plane machining ON (Setting with tool-axis direction)

x, y, z: Coordinates of the origin of feature coordinate system. To be specified with its absolute values in the currently active workpiece coordinate system.

α: Angle of rotation around the Z-axis of the feature coordinate system.
 Setting range: -360° to 360°

1. The designation of X, Y, or Z can be omitted when the argument is zero (0: no shift along the particular axis). Set zero for X, Y, and Z if no translation of the origin is required.
2. The designation of R can be omitted when the argument is zero (0: no rotation around the axis concerned).
3. Use of any other addresses than X, Y, Z, and R will lead to an alarm (**1809 TILTED PLANE CMD FORMAT ERROR**).

B. Setting a feature coordinate system

A block of G68.3 sets a feature coordinate system as follows:

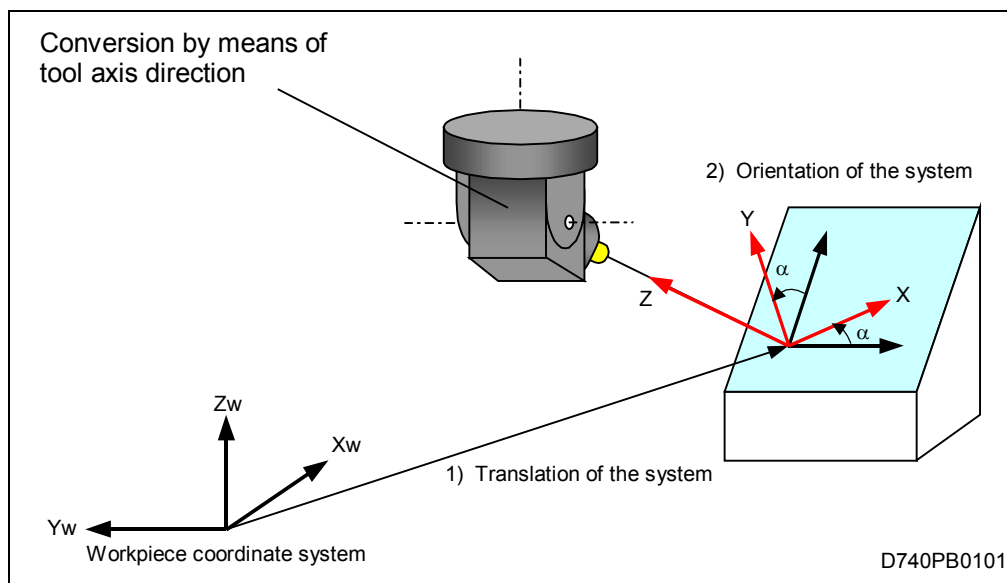
- 1) Point (x, y, z) in the current workpiece coordinate system is set as a new origin.
- 2) The feature coordinate system's X-, Y-, and Z-axis are determined as follows:

The feature coordinate system's Z-axis is set to be parallel with the current direction of the tool axis.

The X-axis is inclined, with reference to the workpiece coordinate system, so as to correspond with the angular positions of the tool. (The X-axis is, therefore, orientated in parallel with the original one if the machine coordinates of the tool on its rotational axes are 0° .)

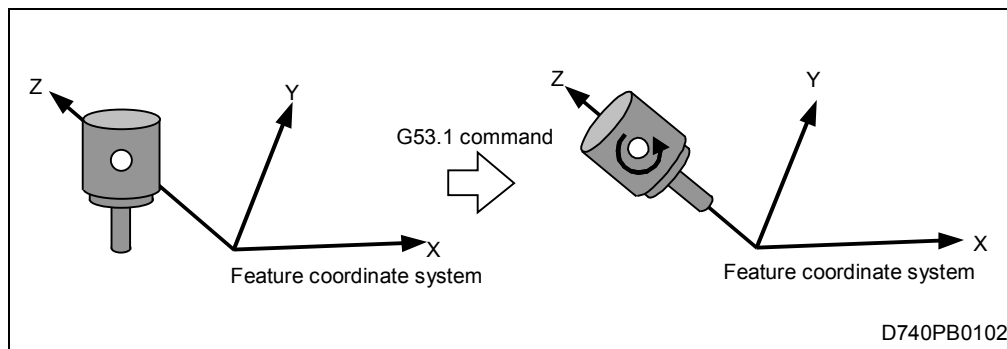
The Y-axis is inclined, with reference to the workpiece coordinate system, so as to correspond with the angular positions of the tool. (The Y-axis is, therefore, orientated in parallel with the original one if the machine coordinates of the tool on its rotational axes are 0° .)

Finally, the feature coordinate system is rotated around the Z-axis by an angle of α° .

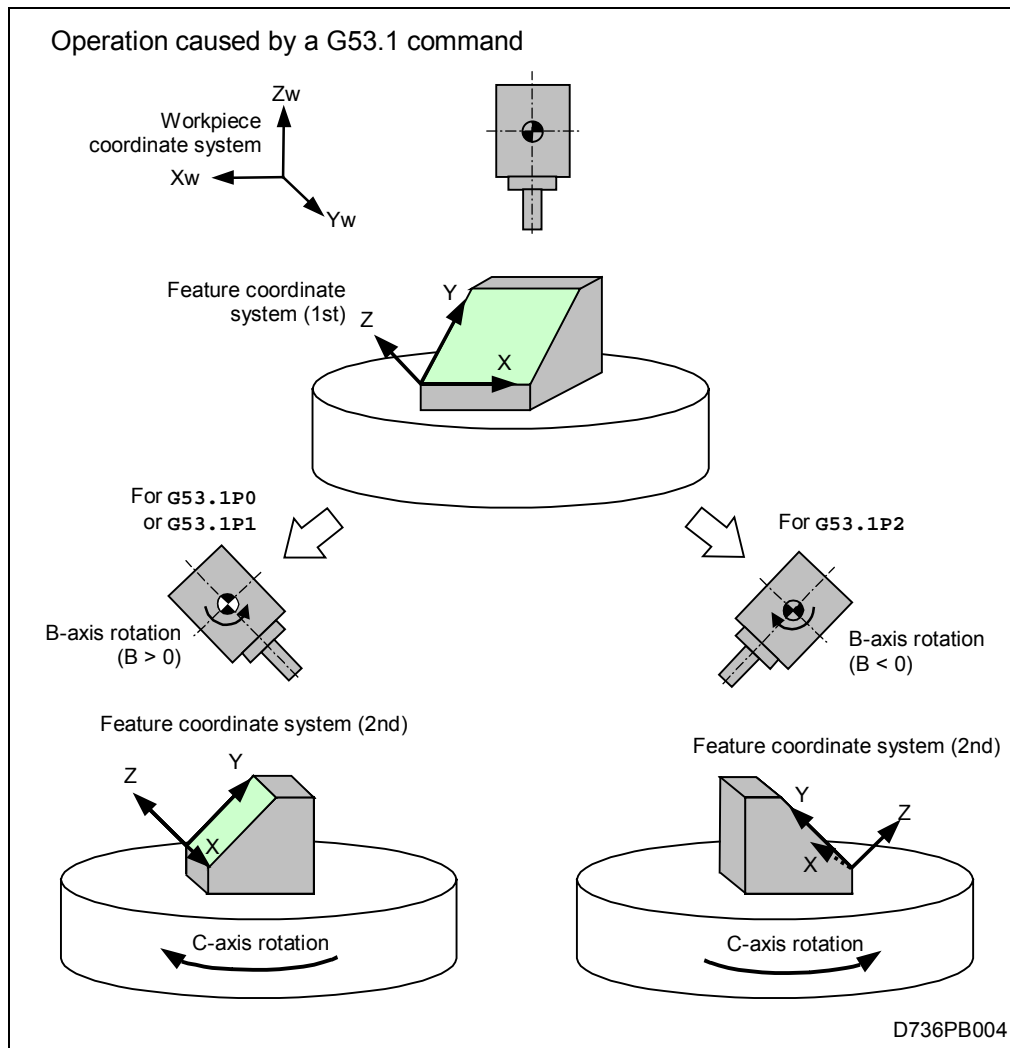


3. Tool-axis direction control

The G53.1 command causes motions on the rotational axes concerned so as to make parallel to, and of the same direction with, each other the direction of the tool axis (from the tip along the perpendicular to the tool-swiveling axis) and the positive direction of the Z-axis of a feature coordinate system. Although no movements occur on the X-, Y- and Z-axis, the resultant rotation of the table makes changes to the feature coordinate system in some cases, and the positional indication with respect to the currently valid coordinate system changes accordingly for the orthogonal axes.



As shown in the figure below, the G53.1 command causes a rotation on the C-axis so as to make the Z-axis of the (1st) feature coordinate system parallel to the XZ-plane of the workpiece coordinate system and, on the other hand, a rotation on the B-axis so as to make the direction of the tool axis and the positive direction of the Z-axis of the changed (2nd) feature coordinate system parallel to, and of the same direction with, each other. (No movements occur on the X-, Y- and Z-axis.) The positional indication on the display with respect to the currently valid coordinate system changes according to the 2nd feature coordinate system. The speed of the rotation caused by a G53.1 command depends on the currently active mode of motion (G00/G01).



For angles that are calculated for G53.1, there are usually two solutions: solution by a positive, or a negative angle of rotation on the B-axis. Use argument P for the G53.1 command to specify which solution to select.

1. G53.1P1 for the solution by a positive angle of rotation on the B-axis. (Left figure above)
2. G53.1P2 for the solution by a negative angle of rotation on the B-axis. (Right figure above)

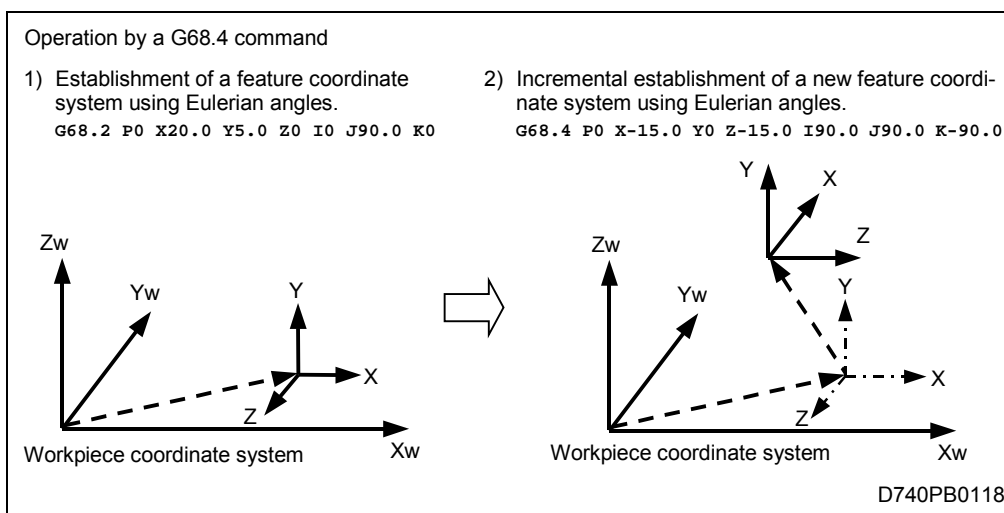
Omission of argument P (or P0) selects a positive angle of rotation (as with case 1 above).

4. Incremental coordinate system establishment for inclined-plane machining

Use G68.4, in the same manner as for G68.2, to establish a new coordinate system on the basis of the currently valid feature coordinate system (defined with G68.2 and G68.3).

Setting method	Programming format
Using Eulerian angles	G68.4 [P0]
Using roll, pitch, and yaw angles	G68.4 P1
Using three points in the plane	G68.4 P2
Using two vectors	G68.4 P3
Using projection angles	G68.4 P4

- Argument P can be omitted if it is 0 (for the use of Eulerian angles).
- If any number other than 0 to 4 is entered with P in a G68.4 block, an alarm will be caused (**1809 TILTED PLANE CMD FORMAT ERROR**).
- Argument P or Q with a decimal point in a G68.4 block will be processed by simply deleting the decimal point and decimal digits.
- Be sure to enter the G68.4 command independently. If a block of G68.4 should contain any other G-codes or motion commands, an alarm is caused (**1809 TILTED PLANE CMD FORMAT ERROR**).
- Giving a G68.4 command without any feature coordinate system (defined with G68.2, G68.3 and G68.4) being valid will lead to an alarm (**1807 CANNOT USE G68.2**).
- It is possible to give another G68.4 command with respect to a feature coordinate system established by the preceding G68.4 command.
- Use G69 to cancel the mode of inclined-plane machining and to restore the original work-piece coordinate system.



5. Operation description

A. Operation in the mode of inclined-plane machining

The G68.2 command causes a feature coordinate system to be set as described above, and the positional indication on the display with respect to the currently valid coordinate system to change accordingly (without any actual movements on the machine).

Axis motion commands in the G68.2 mode are processed in general with respect to the feature coordinate system.

B. Tool-axis direction control

The G53.1 command causes actual motions on the rotational axes concerned so as to adjust the direction of tool axis to the positive direction of the Z-axis of a feature coordinate system. No motions, however, will occur on the orthogonal axes (X, Y and Z). The speed of the rotation depends on the currently active mode of motion (G00/G01).

Note: Depending on the particular attitude of the feature coordinate system, the G53.1 command may cause a considerable amount of motion on the rotational axis concerned. To prevent interference, therefore, move the tool well away from the table before entering the command.

C. Cancellation of the mode of inclined-plane machining

The G69 command cancels the mode of inclined-plane machining. The feature coordinate system will be replaced by the original workpiece coordinate system with reference to which the G68.2 command was given, and the positional indication on the display with respect to the currently valid coordinate system will be changed accordingly (without any actual movements on the machine).

Resetting the NC-unit includes cancellation of the mode of inclined-plane machining.

D. Example of programming

- Shown below is a program for machining the same shape on each inclined surface of a hexagonal prism. Blocks N1 to N6 in the main program define a feature coordinate system for each surface, and the tool path for machining is described in a subprogram (WNo. 100). The workpiece origin is assumed to be set at the center of the top surface of the hexagonal prism.

WNo. 10

```

N1  G68.2 X86.602 Y50. Z0. I-90. J-45. K0.
      M98 P100
      G69
      G0 X300. Y0. Z200. B0. C0.

N2  G68.2 X86.602 Y-50. Z0. I-150. J-45. K0.
      M98 P100
      G69
      G0 X300. Y0. Z200. B0. C0.

N3  G68.2 X0. Y-100. Z0. I-210. J-45. K0.
      M98 P100
      G69
      G0 X300. Y0. Z200. B0. C0.

N4  G68.2 X-86.602 Y-50. I-270. J-45. K0.
      M98 P100
      G69
      G0 X300. Y0. Z200. B0. C0.

N5  G68.2 X-86.602 Y50. I-330. J-45. K0.
      M98 P100
      G69
      G0 X300. Y0. Z200. B0. C0.

N6  G68.2 X0. Y100. I-30. J-45. K0.
      M98 P100
      G69
      G0 X300. Y0. Z200. B0. C0.

M30

```

For machining on
surface [1]

For machining on
surface [2]

For machining on
surface [3]

For machining on
surface [4]

For machining on
surface [5]

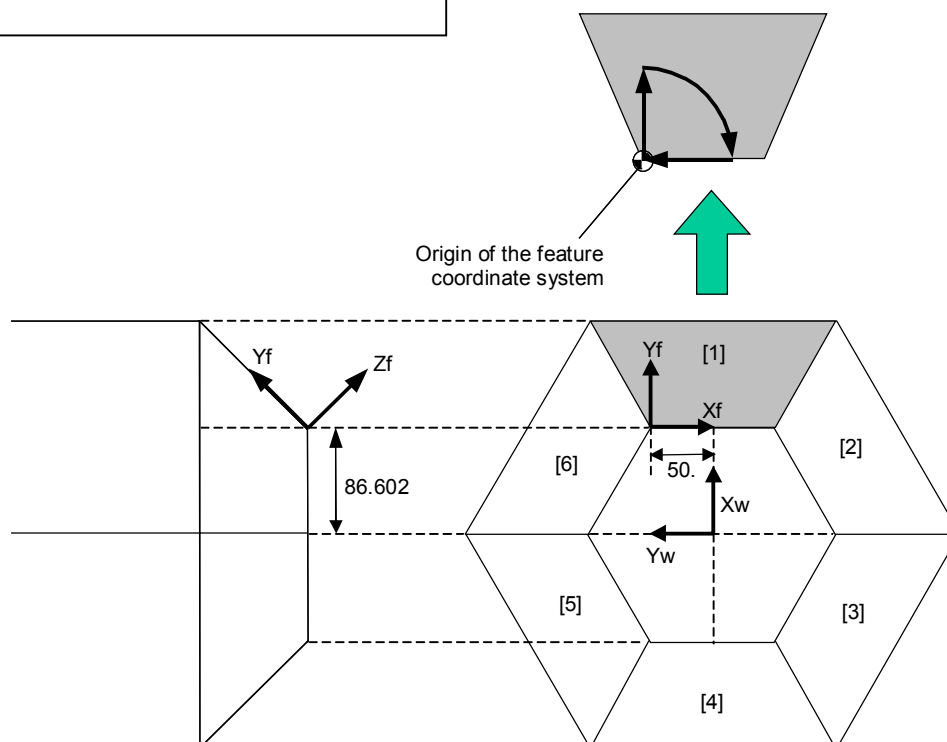
For machining on
surface [6]

WNo. 100

```

G53.1
G90 G0 X0. Y0. Z0.
G1 Y20. F1000
G2 X20. Y0. R20. F1000
G1 X0. F1000
M99

```



D736PB005

2. Shown below is a program for machining on an inclined surface cut on a cube.
Six examples of program section are given for defining a feature coordinate system for the inclined plane, and the tool path for machining is described in a subprogram (WNo. 100).

WNo. 10

```
G28XYZBC
G54X0Y0Z0
M200
```

Section for setting a feature
coordinate system.
See 1 to 6 below.

```
M98P100
G69
M30
```

WNo. 100

```
G53.1
G90 G0 X0.Y0.Z0.B0.C0.
G0 X0Y0Z0
G1 Y50. F1000
G2 X50. Y0. R50. F1000
G1 X0. F1000
M99
```

1. Setting with Eulerian angles

```
G68.2 X33.3333 Y33.3333 Z66.6667 I-45 J54.7356 K0
```

2. Setting with roll, pitch, and yaw angles

```
G68.2 P1 Q321 X33.3333 Y33.3333 Z66.6667 I45 J-35.2644 K-30
```

3. Setting with three points in the plane

```
G68.2 P2 Q0 X70.7107 Y40.8248 Z0 R-60
G68.2 P2 Q1 X0 Y0 Z0
G68.2 P2 Q2 X100 Y0 Z100
G68.2 P2 Q3 X0 Y100 Z100
```

4. Setting with two vectors

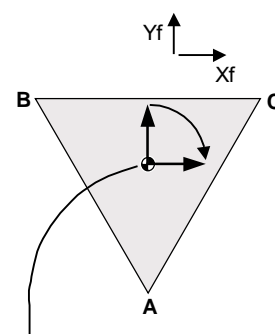
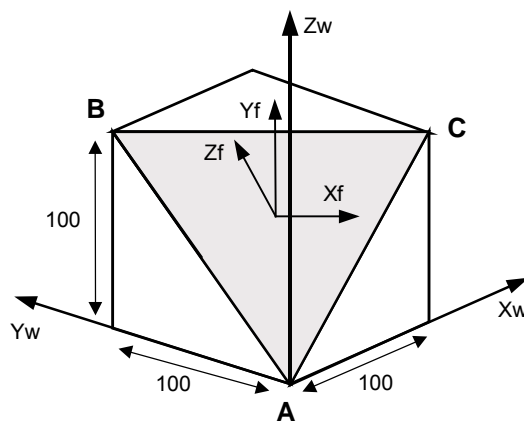
```
G68.2 P3 Q1 X33.3333 Y33.3333 Z66.6667 I100 J-100
G68.2 P3 Q2 I-100 J-100 K100
```

5. Setting with projection angles

```
G68.2 P4 X33.3333 Y33.3333 Z66.6667 I45 J45 K-60
```

6. Setting with tool-axis direction

```
B54.7356 C-135.
G68.3 X33.3333 Y33.3333 Z66.6667 R90
```



Origin of the feature coordinate system
= Centroid of the inclined surface
(x_0, y_0, z_0) = (33.3333, 33.3333, 66.6667)

D740PB0103

26-2-2 Relationship to other functions

1. Relationship to other commands

A. Commands available in the mode of inclined-plane machining

Function	Code	Function	Code
Positioning	G00	Selection of machine coordinate system	G53
Linear interpolation	G01	Tool-axis direction control	G53.1
Circular interpolation (CW)	G02 (*1) (*3)	Exact stop mode	G61
Circular interpolation (CCW)	G03 (*1) (*3)	Geometry compensation	G61.1
Dwell	G04	Cutting mode	G64
High-speed machining mode	G05	User macro single call	G65
Exact-stop	G09	User macro modal call A	G66
Selection of XY-plane	G17	User macro modal call B	G66.1
Selection of ZX-plane	G18	User macro modal call OFF	G67
Selection of YZ-plane	G19	3-D coordinate conversion ON	G68 (*4)
Return to zero point	G28/G30	Incremental coordinate system establishment for inclined-plane machining	G68.4
Skip function	G31 (*5)	3-D coordinate conversion OFF	G69
Tool radius compensation OFF	G40	Hole-machining fixed cycles (G82.2 excluded)	G71.1 - G89
Tool radius compensation (to the left)	G41	Absolute/Incremental data input	G90/G91
Tool radius compensation (to the right)	G42	Inverse time feed	G93
Tool radius compensation for five-axis machining (to the left)	G41.2	Feed per minute (asynchronous)	G94
	G41.4	Feed per revolution (synchronous)	G95
	G41.5		
Tool radius compensation for five-axis machining (to the right)	G42.2	Initial point level return in hole-machining fixed cycles	G98
	G42.4		
	G42.5	R-point level return in hole-machining fixed cycles	G99
Tool length offset (+)	G43	Single program multi-process control	G109
Tool tip point control, type 1	G43.4	M, S, T, B output to opposite system	G112
Tool tip point control, type 2	G43.5	Drilling and tapping fixed cycles	G283 - G289
Tool length offset (-)	G44	Subprogram call/End of subprogram	M98/M99
Tool position offset	G45, G46, G47, G48	Feed function	F
Tool position offset OFF	G49	M, S, T, B function	MSTB (*2)
Mirror image OFF	G50.1	Local variables, Common variables, Operation commands, Control commands	Macro instructions
Mirror image ON	G51.1		

*1 A command for helical/spiral interpolation will lead to an alarm (**1806 ILLEGAL CMD TILTED PLANE CMD**).

*2 Use of a tool function (T-code) in the mode of inclined-plane machining will lead to an alarm (**1806 ILLEGAL CMD TILTED PLANE CMD**).

*3 In combined use with the function for tool tip point control, giving a circular interpolation command will lead to an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**).

*4 A G68 command can be given in the mode of inclined-plane machining only when parameter **F168** bit 4 = 1 (Replacing the 3-D coordinate conversion command [G68] with inclined-plane machining command [G68.2]).

*5 Available only for machines of tool rotating type.

Giving any other command than those enumerated above in the mode of inclined-plane machining will lead to an alarm (**1806 ILLEGAL CMD TILTED PLANE CMD**).

B. Modes in which inclined-plane machining is selectable (with G68.2 or G68.3)

Function	Code	Function	Code
Positioning	G00	Selection of workpiece coordinate system 4	G57
Linear interpolation	G01	Selection of workpiece coordinate system 5	G58
High-speed machining mode OFF	G05P0	Selection of workpiece coordinate system 6	G59
Radius data input for X-axis ON	G10.9X0	Exact stop mode	G61
Selection of XY-plane	G17	Geometry compensation	G61.1
Selection of ZX-plane	G18	Cutting mode	G64
Selection of YZ-plane	G19	User macro single call	G65
Inch data input	G20	User macro modal call OFF	G67
Metric data input	G21	3-D coordinate conversion OFF	G69
Pre-move stroke check OFF	G23	Fixed cycle OFF	G80
Tool radius compensation OFF	G40	Absolute/Incremental data input	G90/G91
Control for normal-line direction OFF	G40.1	Inverse time feed	G93
Tool length offset (+)	G43	Feed per minute (asynchronous)	G94
Tool length offset (–)	G44	Feed per revolution (synchronous)	G95
Tool position offset OFF	G49	Constant surface speed control OFF	G97
Scaling OFF	G50	Initial point level return in hole-machining fixed cycles	G98
Mirror image OFF	G50.1		
Polygonal machining mode OFF	G50.2	R-point level return in hole-machining fixed cycles	G99
Selection of workpiece coordinate system 1	G54	Single program multi-process control	G109
Selection of additional workpiece coordinate system	G54.1	Cross machining control OFF	G111
		Hob milling mode OFF	G113
Workpiece setup error correction	G54.4	Superposition control OFF	G127
Selection of workpiece coordinate system 2	G55		
Selection of workpiece coordinate system 3	G56		

Giving a G68.2 or G68.3 command in a mode other than those enumerated above will lead to an alarm (**1807 CANNOT USE G68.2**).

C. Modes in which incremental coordinate system establishment for inclined-plane machining is possible (with G68.4)

Function	Code	Function	Code
Positioning	G00	Selection of workpiece coordinate system 6	G59
Linear interpolation	G01	Exact stop mode	G61
Radius data input for X-axis ON	G10.9X0	Geometry compensation	G61.1
Selection of XY-plane	G17	Cutting mode	G64
Selection of ZX-plane	G18	User macro single call	G65
Selection of YZ-plane	G19	User macro modal call OFF	G67
Inch data input	G20	3-D coordinate conversion ON	G68 (*1)
Metric data input	G21	Inclined-plane machining	G68.2
Pre-move stroke check OFF	G23		G68.3
Tool radius compensation OFF	G40		G68.4
Control for normal-line direction OFF	G40.1	3-D coordinate conversion OFF	G69
Tool length offset (+)	G43	Fixed cycle OFF	G80
Tool length offset (-)	G44	Absolute/Incremental data input	G90/G91
Tool position offset OFF	G49	Inverse time feed	G93
Scaling OFF	G50	Feed per minute (asynchronous)	G94
Mirror image OFF	G50.1	Feed per revolution (synchronous)	G95
Polygonal machining mode OFF	G50.2	Constant surface speed control OFF	G97
Selection of workpiece coordinate system 1	G54	Initial point level return in hole-machining fixed cycles	G98
Selection of additional workpiece coordinate system	G54.1		
Workpiece setup error correction	G54.4	R-point level return in hole-machining fixed cycles	G99
Selection of workpiece coordinate system 2	G55	Single program multi-process control	G109
Selection of workpiece coordinate system 3	G56	Cross machining control OFF	G111
Selection of workpiece coordinate system 4	G57	Hob milling mode OFF	G113
Selection of workpiece coordinate system 5	G58	Superposition control OFF	G127

*1 A G68.4 command can be given under G68 only when parameter **F168** bit 4 = 1 (Replacing the 3-D coordinate conversion command [G68] with inclined-plane machining command [G68.2]).

Giving a G68.4 command in a mode other than those enumerated above will lead to an alarm (**1807 CANNOT USE G68.2**).

D. Modes in which tool-axis direction control (G53.1) is selectable

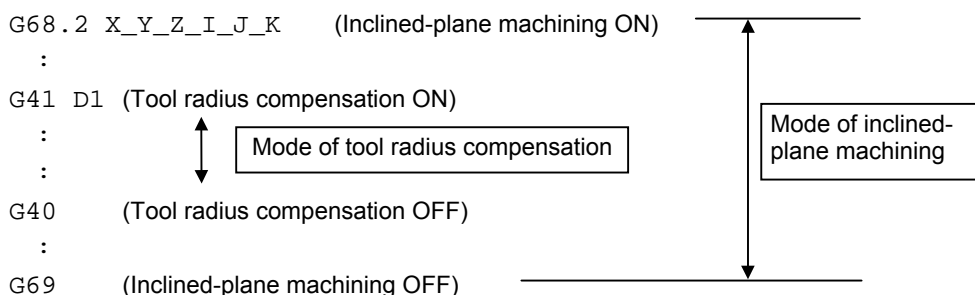
Function	Code	Function	Code
Positioning	G00	Selection of workpiece coordinate system 5	G58
Linear interpolation	G01	Selection of workpiece coordinate system 6	G59
High-speed machining mode OFF	G05P0	Exact stop mode	G61
Radius data input for X-axis ON	G10.9X0	Geometry compensation	G61.1
Selection of XY-plane	G17	Cutting mode	G64
Selection of ZX-plane	G18	User macro single call	G65
Selection of YZ-plane	G19	User macro modal call OFF	G67
Inch data input	G20	3-D coordinate conversion ON	G68 (*1)
Metric data input	G21	Inclined-plane machining	G68.2
Pre-move stroke check OFF	G23		G68.3
Tool radius compensation OFF	G40		G68.4
Control for normal-line direction OFF	G40.1	Fixed cycle OFF	G80
Tool length offset (+)	G43	Absolute/Incremental data input	G90/G91
Tool length offset (-)	G44	Inverse time feed	G93
Tool length offset OFF	G49	Feed per minute (asynchronous)	G94
Scaling OFF	G50	Feed per revolution (synchronous)	G95
Mirror image OFF	G50.1	Constant surface speed control OFF	G97
Polygonal machining mode OFF	G50.2	Initial point level return in hole-machining fixed cycles	G98
Selection of workpiece coordinate system 1	G54		
Selection of additional workpiece coordinate system	G54.1	R-point level return in hole-machining fixed cycles	G99
Dynamic offsetting II OFF	G54.2P0	Single program multi-process control	G109
Workpiece setup error correction	G54.4	Cross machining control OFF	G111
Selection of workpiece coordinate system 2	G55	Hob milling mode OFF	G113
Selection of workpiece coordinate system 3	G56	Superposition control OFF	G127
Selection of workpiece coordinate system 4	G57		

*1 A G53.1 command can be given under G68 only when parameter **F168** bit 4 = 1 (Replacing the 3-D coordinate conversion command [G68] with inclined-plane machining command [G68.2]).

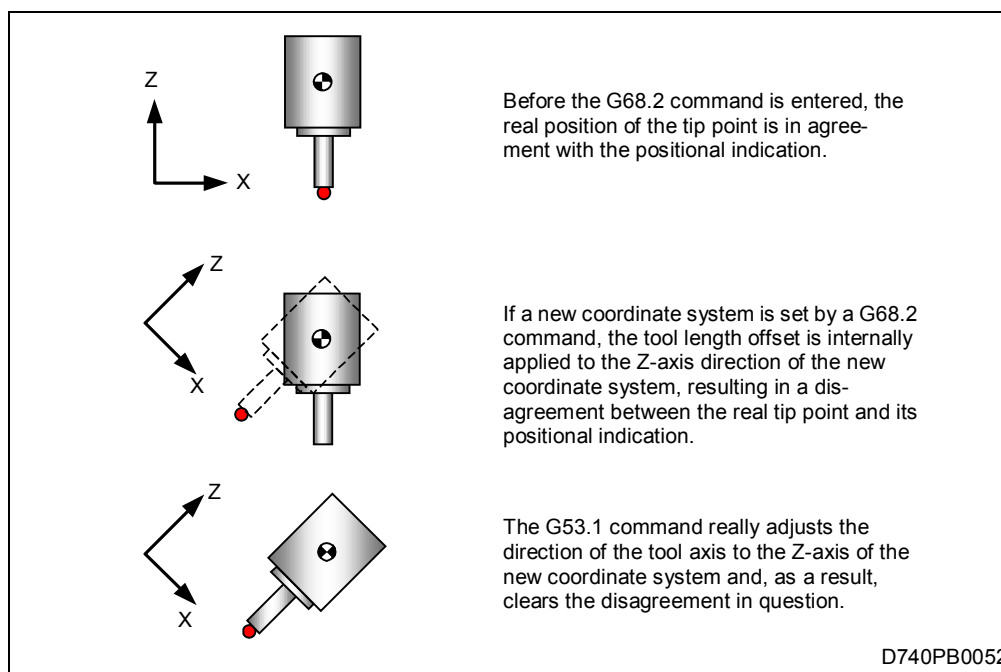
Giving a G53.1 command in a mode other than those enumerated above will lead to an alarm (1808 CANNOT USE G53.1).

26-2-3 Restrictions

1. Move the tool to a safe position before entering the G53.1 command in order to prevent interference from being caused by the resulting motions on the rotational axes.
2. In the mode of inclined-plane machining, system variables #5001 to #5116 for reading positional information refer to the feature coordinate system, while system variables #5021 to #5036 always denote the current position with respect to the machine coordinate system.
3. Resetting in the mode of inclined-plane machining cancels the mode and makes the G69 code modal in the group concerned.
4. Tool radius compensation (for five-axis machining as well), mirroring by G-code, fixed cycle, and tool tip point control must be selected and cancelled within the mode of inclined-plane machining.



5. Selection of inclined-plane machining mode (by G68.2) with the tool length offset function being active will result in a disagreement between the real tip point and its positional indication with respect to the current coordinate system. The disagreement will be cleared, as shown below, by executing the G53.1 command to adjust the tool-axis direction to the Z-axis of the feature coordinate system.



6. Manual interruption is always performed on the basis of the machine coordinate system (without any coordinate conversion). After manual interruption of inclined-plane machining (by or without using the TPS function) do not resume automatic operation without returning the machine components concerned to the original positions on the controlled axes; otherwise an alarm will be caused (**185 ILLEGAL OPER IN G68.2 MODE**).
During interruption of inclined-plane machining, an attempt to perform manual movements on a rotational axis, or to use the manual pulse handle will only lead to an alarm (**185 ILLEGAL OPER IN G68.2 MODE**).
7. Macro interruption as well as MDI interruption is not available in the mode of inclined-plane machining.
Giving a G68.2/G68.3 command with macro interruption being active will lead to an alarm (**1807 CANNOT USE G68.2**), while an attempt to perform MDI interruption in the mode of inclined-plane machining will only cause another alarm (**185 ILLEGAL OPER IN G68.2 MODE**).
8. Do not give any tool change command in the mode of inclined-plane machining; otherwise an alarm will be caused (**1806 ILLEGAL CMD TILTED PLANE CMD**).
9. Tool path check can only be performed on the basis of the original workpiece coordinate system (without coordinate conversion taken into consideration).
10. Tracing in the mode of inclined-plane machining is displayed with reference to the machine coordinate system.
11. Do not use corner chamfering or rounding commands in the mode of inclined-plane machining; otherwise an alarm will be caused (**1806 ILLEGAL CMD TILTED PLANE CMD**).
12. Do not forget to take into consideration the restrictions imposed on tool tip point control, workpiece setup error correction, and tool radius compensation for five-axis machining in combined use with the inclined-plane machining function.
13. As a matter of course, restart operation started by the **[RESTART 2 NONMODAL]** menu function cannot in general be executed otherwise than with the inclined-plane machining mode being cancelled.
14. According to the settings in parameters **SU153** (bits 1, 2, and 3) and **F143** (bit 6), the relevant items of digital information on the **POSITION** display refer to the following types of coordinate system in the mode of inclined-plane machining function:

	Bit in question OFF (0)	Bit in question ON (1)
POSITION (SU153 bit 3)	Feature coordinate system	
MACHINE	Machine coordinate system	
BUFFER (SU153 bit 1)	Workpiece coordinate system	Feature coordinate system
REMAIN (SU153 bit 2)	Workpiece coordinate system	Feature coordinate system
WPC (F143 bit 6)	Workpiece coordinate system	Feature coordinate system

15. Do not give a command for positioning to an arbitrarily definable point, or for external tool measurement (for tool breakage detection) in the mode of inclined-plane machining; otherwise an alarm will be caused (**1806 ILLEGAL CMD TILTED PLANE CMD**).
16. Do not specify a MAZATROL program as a subprogram to be called up in the mode of inclined-plane machining; otherwise an alarm will be caused (**1806 ILLEGAL CMD TILTED PLANE CMD**).
17. Positioning commands (under G0) in the mode of inclined-plane machining are always executed in interpolation-type paths.

18. On machines of table rotating type, a G68.3 block will always establish a feature coordinate system with the Z-axis being parallel to that of the current workpiece coordinate system. Arguments X, Y, Z, and R (for translating and rotating the coordinate system), however, can be specified in the G68.3 block in such a case as well.
19. In the mode of inclined-plane machining, another selection of inclined-plane machining mode will only lead to an alarm (**1806 ILLEGAL CMD TILTED PLANE CMD**).
20. Work simulation on the **VIRTUAL MACHINING** display is done without reference to the feature coordinate system and, therefore, does not represent the operation appropriately.
21. Reference angular position for defining a feature coordinate system
Use parameter **F144** bit 1 to select the reference angular position for setting the angle of rotation in a command (with G68.2) for the definition of a feature coordinate system.

Reference position	Current angular position of the table F144 bit 1 = 0	0° position of the table F144 bit 1 = 1
Description	A feature coordinate system is established with reference to the current position of the table on the rotational axis concerned.	A feature coordinate system is established with reference to the 0° position of the table on the rotational axis concerned.
Remarks	Do not fail to set the table to the angular position appropriate for programming before selecting the mode of inclined-plane machining.	The feature coordinate system can be defined with reference to the 0° position of the table, irrespective of its current position.

- *1 The parameter in question (**F144** bit 1) only applies to G68.2 (out of all relation to G68.3 and G68.4).
- *2 The parameter in question (**F144** bit 1) is of no effect on machines of tool rotating type.
22. In the mode of inclined-plane machining, give motion commands for linear axes with reference to the feature coordinate system and those for rotational axes with reference to the workpiece coordinate system (i.e. machine coordinate system).
23. The acceleration and deceleration for positioning commands (under G0) in the mode of inclined-plane machining occurs according to the parameters **L74** (rate of cutting feed for pre-interpolational acceleration/deceleration control) and **L75** (time constant for pre-interpolational cutting feed) when they are given under G61.1 (geometry compensation) or with the (optional) fixed gradient control for G0 being selected.

26-3 Tool Radius Compensation for Five-Axis Machining (Option)

26-3-1 Function outline

The function described in this section refers to three-dimensional tool radius compensation on a five-axis control machine (equipped with two controlled axes of rotational type) by computing the offset vector in a plane perpendicular to the tool axis.

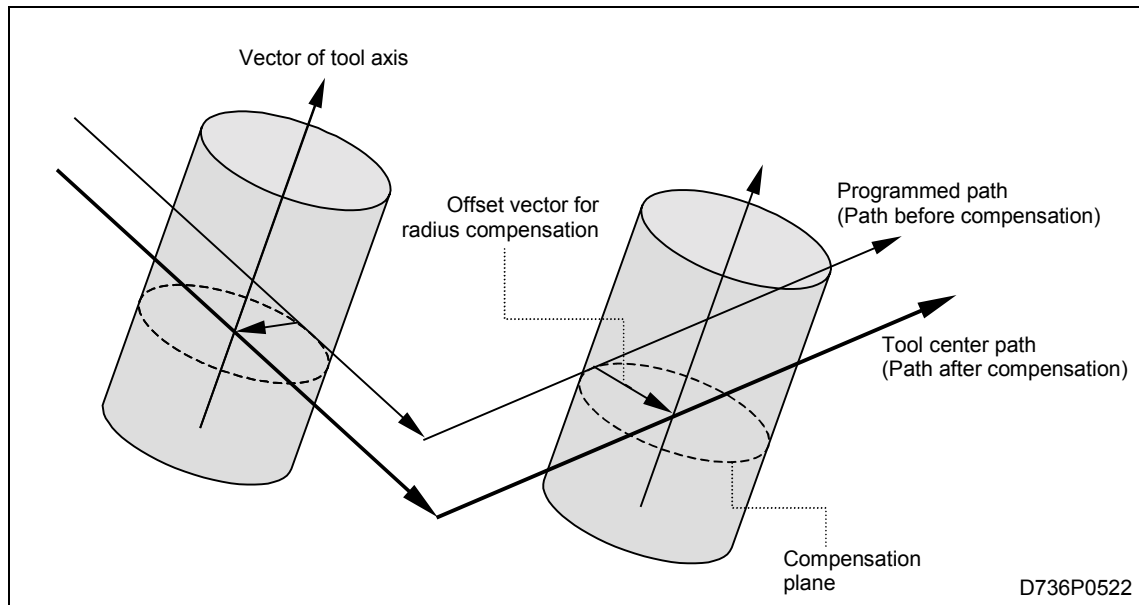


Fig. 26-3 Tool radius compensation for five-axis machining

26-3-2 Function description

The offsetting function in question conducts tool-path control for tool radius compensation in a plane (called 'compensation plane') perpendicular to the tool axis whose direction is determined by the motion command of the rotational axis concerned. This subsection gives operational particulars proper to this special function, with the compensation plane explained in 26-3-4.

Refer to Section 12-4 for a detailed description of the general tool radius compensation.

1. Programming format

A. Tool radius compensation for five-axis machining ON

<For tool rotating type>

```
G41.2 (X_ Y_ Z_ A_ B_ C_ D_);
G42.2 (X_ Y_ Z_ A_ B_ C_ D_);
```

G41.2 : Tool radius compensation (to the left)

G42.2 : Tool radius compensation (to the right)

XYZABC: Axis motion commands

D : Tool offset data No. for radius compensation

- * Do not use G41.4 and G42.4 (provided for "table rotating type") or G41.5 and G42.5 (for "mixed type") for five-axis control machines of tool rotating type; otherwise an alarm will occur (970 TOOL TIP CTRL PARAMETER ERROR).

<For table rotating type>

G41.4 (G41.2) (X_ Y_ Z_ A_ B_ C_ D_);

G42.4 (G42.2) (X_ Y_ Z_ A_ B_ C_ D_);

G41.4 : Tool radius compensation (to the left)

G42.4 : Tool radius compensation (to the right)

- * Use G41.4 and G42.4 for five-axis control machines of table rotating type, for which G41.2 and G42.2 (provided for “tool rotating type”) may be used as well for the same purpose. Do not use G41.5 and G42.5 (provided for “mixed type”), however, for machines of table rotating type; otherwise an alarm will occur (**970 TOOL TIP CTRL PARAMETER ERROR**).

<For mixed type>

G41.5 (G41.2) (X_ Y_ Z_ A_ B_ C_ D_);

G42.5 (G42.2) (X_ Y_ Z_ A_ B_ C_ D_);

G41.5 : Tool radius compensation (to the left)

G42.5 : Tool radius compensation (to the right)

- * Use G41.5 and G42.5 for five-axis control machines of mixed type, for which G41.2 and G42.2 (provided for “tool rotating type”) may be used as well for the same purpose. Do not use G41.4 and G42.4 (provided for “table rotating type”), however, for machines of mixed type; otherwise an alarm will occur (**970 TOOL TIP CTRL PARAMETER ERROR**).

B. Tool radius compensation for five-axis machining OFF (cancellation)

G40 (X_ Y_ Z_ A_ B_ C_);

G40 : Cancellation of tool radius compensation

2. Offset data items used for tool radius compensation

The data settings of the **TOOL DATA** display (prepared for the execution of MAZATROL programs) can also be used in tool radius compensation for five-axis machining. The table below indicates those usage patterns of the externally stored tool offset data items which are applied to the tool radius compensation according to the settings of the relevant parameters (**F92** bit 7 and **F94** bit 7).

Parameter		Data in the TOOL DATA display		Data in the TOOL OFFSET display
F92 bit 7	F94 bit 7	ACT-φ	ACT-φ CO./No.	
0	0	×	×	○
0	1	×	○	×
1	0	○	×	○
1	1	○	○	×

○: Used for tool radius compensation.

×: Not used.

F92 bit 7: **ACT-φ/NOSE-R** in the **TOOL DATA** display for an EIA/ISO program

0: Invalid

1: Valid

F94 bit 7: Offset or compensation values to be used for an EIA/ISO program

0: Values in the **TOOL OFFSET** display

1: Values provided for EIA/ISO programs in the **TOOL DATA** display

26-3-3 Operation of tool radius compensation for five-axis machining

1. Startup of the tool radius compensation

The G41.2/G42.2, G41.4/G42.4, or G41.5/G42.5 code given in the cancellation mode turns on the mode of tool radius compensation for five-axis machining, and describes such an initial offset path to the selection block's ending point as includes compensation in the plane perpendicular to the tool axis in that position. The selection between type A and type B for startup operation in the compensation plane is to be done by setting parameter **F92** bit 4, as is the case with the general mode of tool radius compensation. See the section on the tool radius compensation for more information.

Take care to give the startup code under the appropriate conditions of G-codes (see the table concerned in Subsection 26-3-5); otherwise an alarm will be caused (**962 CANNOT USE 5X RADIUS COMP.**).

2. Operation in the mode of tool radius compensation

The tool radius compensation for five-axis machining only applies to commands of positioning (G00) and linear interpolation (G01). Take care not to use G-codes unavailable in the mode (see the table concerned in Subsection 26-3-5); otherwise an alarm will be caused (**961 ILLEGAL COMMAND 5X RADIUS COMP.**).

As for motion blocks automatically interpolated for turning a corner, the direction of the tool axis at the ending point of the first one of the two blocks concerned (as specified in the last B-axis command) is kept intact, along with the rate of feed and other modal information items, up to the stop point for the single-block operation.

3. Cancellation of the tool radius compensation

The mode of tool radius compensation for five-axis machining is cancelled when one of the following conditions is satisfied:

1. The cancellation command concerned (G40) is executed, or
2. The NC is reset.

The selection between type A and type B for cancellation can be done by setting parameter **F92** bit 4, as for startup operation.

26-3-4 Method of computing the offset vector

The offset vector is internally computed as follows.

1. Conversion for the table coordinate system

The programmed motion commands are converted for a tool path with respect to the table coordinate system. Table coordinate system is a coordinate system which is fixed to the table (or workpiece) and will rotate as the table rotates. The tool path described on the basis of this coordinate system refers to the relative motion of the tool to the workpiece.

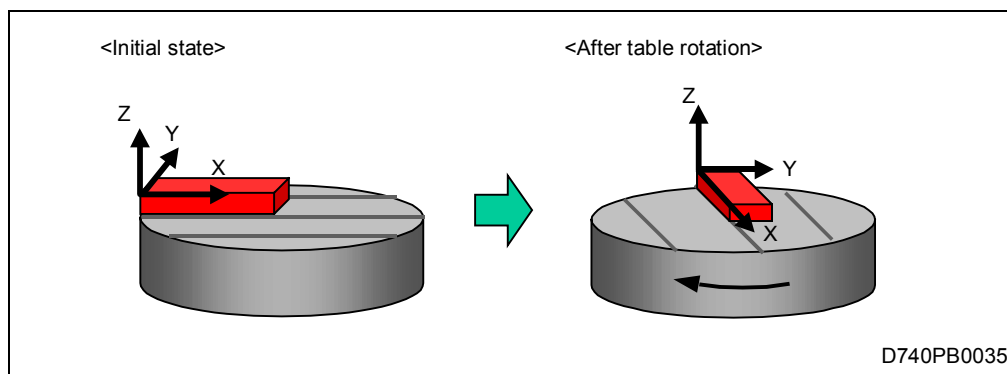


Fig. 26-4 Table coordinate system

2. Conversion of points into the compensation plane

The obtained tool path in the table coordinate system is then converted by orthogonal projection onto the compensation plane (a plane cutting the tool axis perpendicularly in a point for compensation).

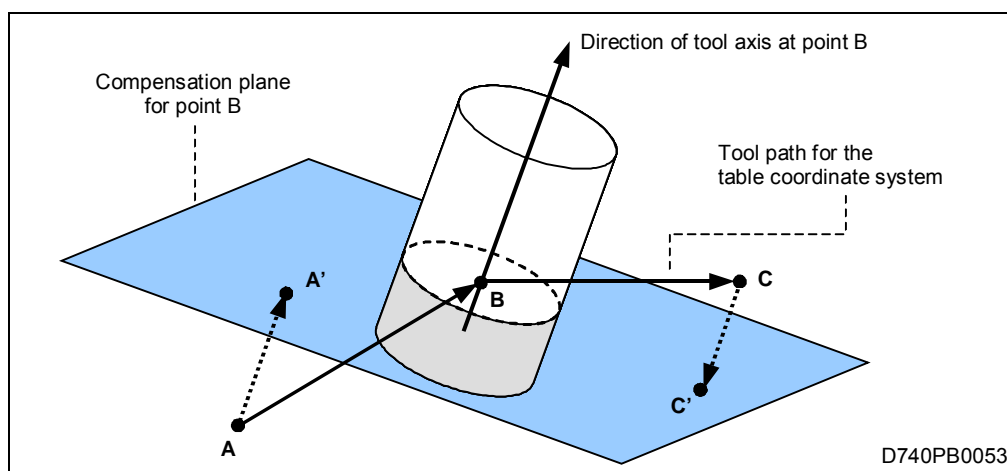


Fig. 26-5 Conversion of points into the compensation plane

The NC finally conducts tool radius compensation on the path projected into the compensation plane to calculate the offset vector at the particular point.

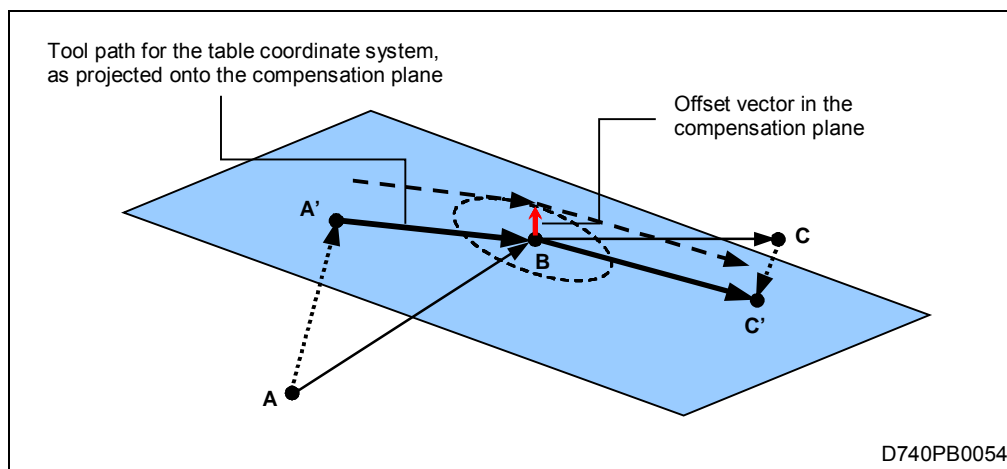


Fig. 26-6 Compensation in the compensation plane

26-3-5 Relationship to other functions

1. Commands usable in the same block with the mode selection code

Function	Code	Function	Code
Positioning	G00	Incremental data input	G91
Linear interpolation	G01	Feed function	F
Absolute data input	G90		

Giving any other command than those enumerated above in the same block as the mode selection code will lead to an alarm (**961 ILLEGAL COMMAND 5X RADIUS COMP.**).

2. Relationship to other commands

A. Commands available in the mode of tool radius compensation for five-axis machining

Function	Code	Function	Code
Positioning	G00	Geometry compensation	G61.1
Linear interpolation	G01	Cutting mode	G64
Dwell	G04	Macro call	G65
High-speed machining mode	G05 (*1)	Absolute data input	G90
Exact-stop	G09	Incremental data input	G91
Pre-move stroke check ON	G22	Inverse time feed	G93
Pre-move stroke check OFF	G23	Feed per minute	G94
Tool radius compensation OFF	G40	Feed per revolution	G95
Tool radius compensation for five-axis machining (to the left)	G41.2	Constant surface speed control OFF	G97
	G41.4	M, S, T, B output to opposite system	G112 (*2)
	G41.5	Subprogram call/End of subprogram	M98/M99
Tool radius compensation for five-axis machining (to the right)	G42.2	M, S, T, B function	MSTB (*2)
	G42.4	Local variables, Common variables, Operation commands (arithmetic operations, trigonometric functions, square root, etc), Control commands (IF ~ GOTO ~, WHILE ~ DO ~)	Macro instructions
	G42.5		
Tool length offset (+/-)	G43/G44		
Tool length offset OFF	G49		
Exact stop mode	G61		

*1 Giving a command for high-speed machining will lead to an alarm (**807 ILLEGAL FORMAT**) under a combined use of tool radius compensation for five-axis machining with tool tip point control or tool length offset (+/-). It is acceptable, however, under a combined use with tool length offset in tool-axis direction (G43.1).

*2 Use of a tool function (T-code) in the mode of tool radius compensation for five-axis machining will lead to an alarm (**961 ILLEGAL COMMAND 5X RADIUS COMP.**).

Giving any other command than those enumerated above in the mode of tool radius compensation for five-axis machining will lead to an alarm (**961 ILLEGAL COMMAND 5X RADIUS COMP.**).

B. Modes in which tool radius compensation for five-axis machining is selectable

Function	Code	Function	Code
Positioning	G00	Workpiece setup error correction	G54.4
Linear interpolation	G01	Exact stop mode	G61
Radius data input for X-axis ON	G10.9	Geometry compensation	G61.1
Plane selection	G17, G18, G19	Cutting mode	G64
		Modal user macro call OFF	G67
Inch data input	G20	3-D coordinate conversion ON	G68 (*1)
Metric data input	G21	Inclined-plane machining	G68.2
Pre-move stroke check ON	G22		G68.3
Pre-move stroke check OFF	G23		G68.4
Tool radius compensation OFF	G40	3-D coordinate conversion OFF	G69
Tool radius compensation for five-axis machining (to the left)	G41.2	Fixed cycle OFF	G80
	G41.4	Absolute data input	G90
	G41.5	Incremental data input	G91
Tool radius compensation for five-axis machining (to the right)	G42.2	Inverse time feed	G93
	G42.4	Feed per minute	G94
	G42.5	Feed per revolution	G95
Tool length offset (+/-)	G43/G44	Constant surface speed control OFF	G97
Tool length offset in tool-axis direction	G43.1	Initial point level return in hole-machining fixed cycles	G98
Tool tip point control	G43.4/G43.5		
Tool position offset OFF	G49	R-point level return in hole-machining fixed cycles	G99
Scaling OFF	G50		
Mirror image OFF	G50.1	Single program multi-process control	G109
Polygonal machining mode OFF	G50.2	Cross machining control OFF	G111
Selection of workpiece coordinate system/ additional workpiece coordinate system	G54-59, G54.1	Hob milling mode OFF	G113
		Superposition control OFF	G127

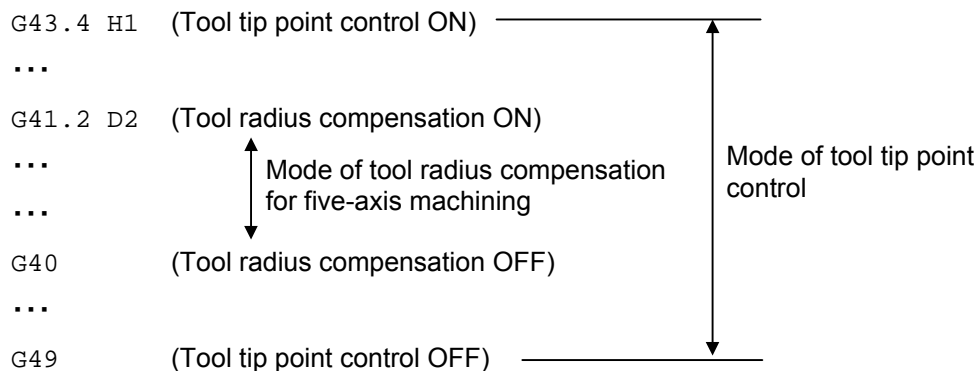
*1 A command for tool radius compensation for five-axis machining can be given under G68 only when parameter **F168** bit 4 = 1 (Replacing the 3-D coordinate conversion command [G68] with inclined-plane machining command [G68.2]).

Giving a command for tool radius compensation for five-axis machining in a mode other than those enumerated above will lead to an alarm (**962 CANNOT USE 5X RADIUS COMP.**).

26-3-6 Restrictions

1. The calculated path of tool radius compensation cannot be checked for interference, irrespective of the setting of the parameter concerned (**F92** bit 5: Checking to avoid interference ON/OFF).
2. Do not use the radius compensation codes G38 (to set an offset vector) and G39 (to interpolate a circular arc at a corner) in the mode of radius compensation for five-axis machining. Otherwise an alarm will be caused (**961 ILLEGAL COMMAND 5X RADIUS COMP.**).
3. Do not use corner chamfering or rounding commands in the mode of radius compensation for five-axis machining. Otherwise an alarm will be caused (**961 ILLEGAL COMMAND 5X RADIUS COMP.**).
4. The tool change command, if required, must always be given after canceling the mode of radius compensation for five-axis machining. Otherwise an alarm will be caused (**961 ILLEGAL COMMAND 5X RADIUS COMP.**).
5. Do not apply manual interruption in general, MDI interruption, and interruption by the manual pulse handle in particular in the mode of radius compensation for five-axis machining. Otherwise an alarm will be caused (**168 ILLEGAL OPER 5X RADIUS COMP.**).
6. Take the following precautions for compound use with the tool tip point control:
 - The tool radius compensation for five-axis machining must be turned on and off within the mode of tool tip point control.

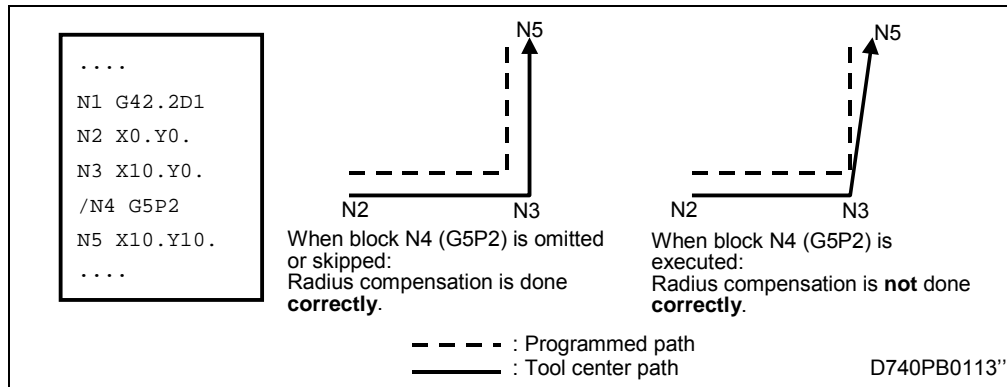
<Programming example>



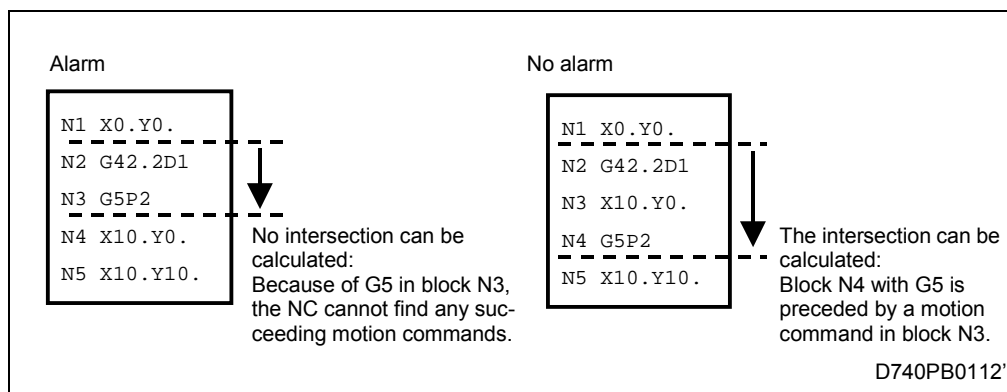
7. Restarting operation is possible even from a block within the mode of radius compensation for five-axis machining unless the block in question is also in the mode of tool tip point control. (Related alarm: **956 RESTART OPERATION NOT ALLOWED**)
8. As is the case in general with tool radius compensation, do not interrupt the flow of cutting feed by a command which prohibits pre-reading for tool radius compensation.

The command codes in question are as follows.

Function	Code
High-speed machining mode	G05
Programmed stop	M0

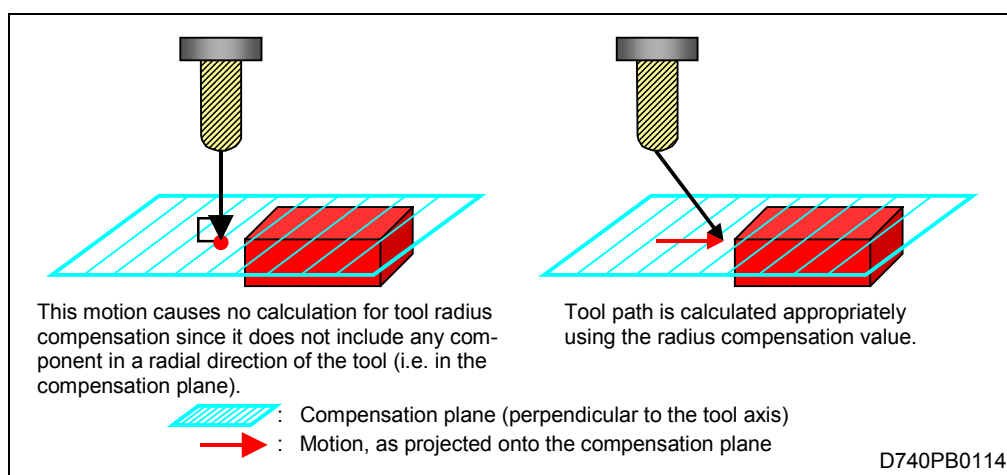


9. The alarm **836 NO INTERSECTION** will be caused when there is no motion command given between G41.2/G42.2 and a command which prohibits pre-reading (as shown under 8 above.)



10. Restrictions imposed on a combined use with high-speed machining.

- The radius compensation value may not be used appropriately in calculation of tool path for a motion merely in the direction of the tool axis (i.e. perpendicular to the compensation plane). For approaching the workpiece at the start of cutting, therefore, do not fail to describe a movement in an oblique direction with respect to the tool axis.

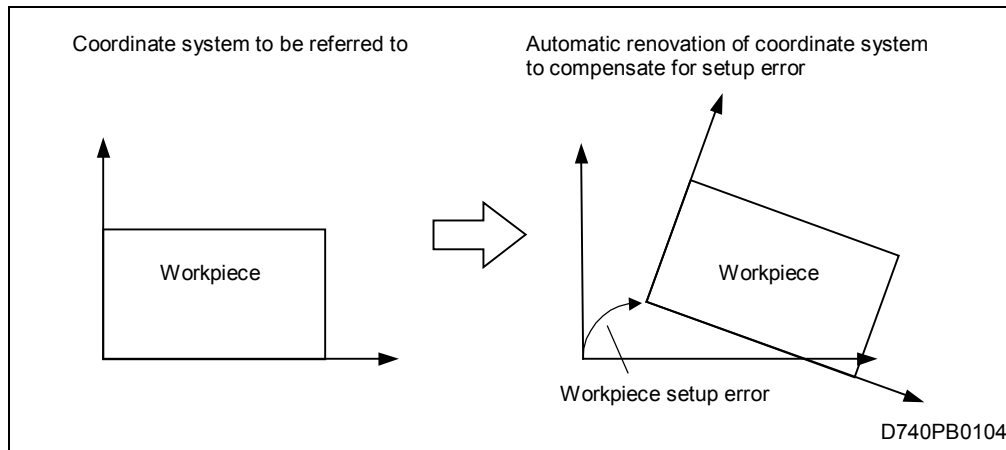


- The fairing function for high-speed machining cannot be made effective. The related parameter (**F96** bit 1) is of no effect in the mode of tool radius compensation for five-axis machining.

26-4 Workpiece Setup Error Correction: G54.4P0, G54.4P1 to G54.4P7 (Option)

26-4-1 Function outline

This function is provided to compensate for the error in setting a workpiece easily, without having to rewrite the program section for describing the machining contour, by modifying the workpiece coordinate system according to the error in question.



26-4-2 Function description

1. Programming format

G54.4 Pn;

n: Number of the data set for workpiece setup error correction (1 to 7)

Enter zero with P (G54.4 P0) to cancel the correction function.

- Be sure to enter the G54.4 command (for selection as well as for cancellation) independently. If a block of G54.4 should contain any other commands, an alarm is caused (**1815 CANNOT USE G54.4**).
- Omission of argument P will lead to an alarm (**807 ILLEGAL FORMAT**).
Alarm **809 ILLEGAL NUMBER INPUT** occurs if argument P is of any other value than 0 to 7.

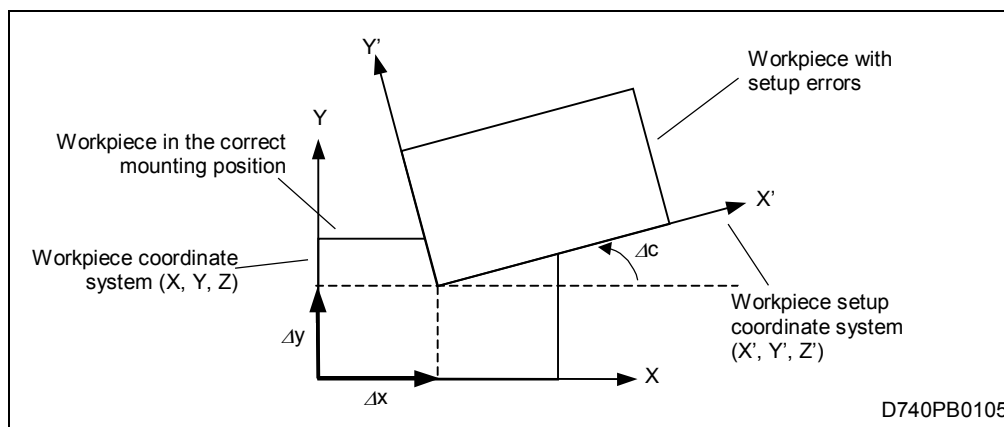
2. Definition of workpiece setup coordinate system

The workpiece setup coordinate system is defined by using the following three types of externally provided data on the setup error.

1. Linear errors on the orthogonal axes X, Y, and Z (Δx , Δy , Δz)
Specify the origin of the workpiece setup coordinate system with its absolute values in the current workpiece coordinate system.
2. Angular errors in rotation around the orthogonal axes (Δa , Δb , Δc)
Specify those angles (Δa , Δb , and Δc) of rotations around the axes X, Y, and Z of the current workpiece coordinate system, in that order, by which the same orientation as that of the workpiece setup coordinate system can be obtained.
Positive values of angle refer to the rotation by which a right-handed screw moves in the positive direction of the third orthogonal axis.

3. Coordinate of the axis of table rotation during error measurement

Set the machine coordinate of the axis of table rotation during measurement of errors Δx , Δy , Δz and Δa , Δb , Δc . No setting is required for machines without the controlled axis of table rotation, while setting for two axes is necessary for five-axis control machines of table rotating type.



3. Setting workpiece setup errors

Seven data sets (No. 1 to No. 7) of workpiece setup errors can be set or prepared for the designation by argument P in a G54.4 block.

It is possible, moreover, to prepare a special set of data to be added commonly to each of the numbered data sets. Values for the linear errors (Δx , Δy , Δz) and the table's angular position during measurement can be set as common data, but not for the angular errors (Δa , Δb , Δc).

The number of available data items for the table's angular position and their names depend upon the specifications of the machine in question.

A. WORK POSITION ERROR COMP. display

Use the **WORK POSITION ERROR COMP.** display to prepare the data for workpiece setup error correction. See the relevant section in PART 3 of the Operating Manual.

Common data		No. 1 to No. 7		
Δx	5.	Δx	15.	Set linear errors on the orthogonal axes X, Y, and Z.
Δy	0.	Δy	3.	
Δz	10.	Δz	0.	
A	0.	Δa	0.	Set angular errors in rotation around the orthogonal axes. No common data can be provided for this type of data items.
C	-45.	Δb	0.	
		Δc	45.	
		A	0.	Set the coordinate of the axis of table rotation during error measurement.
		C	90.	

B. System variables

Using variables tabulated below, it is possible to read and write the values used for workpiece setup error correction.

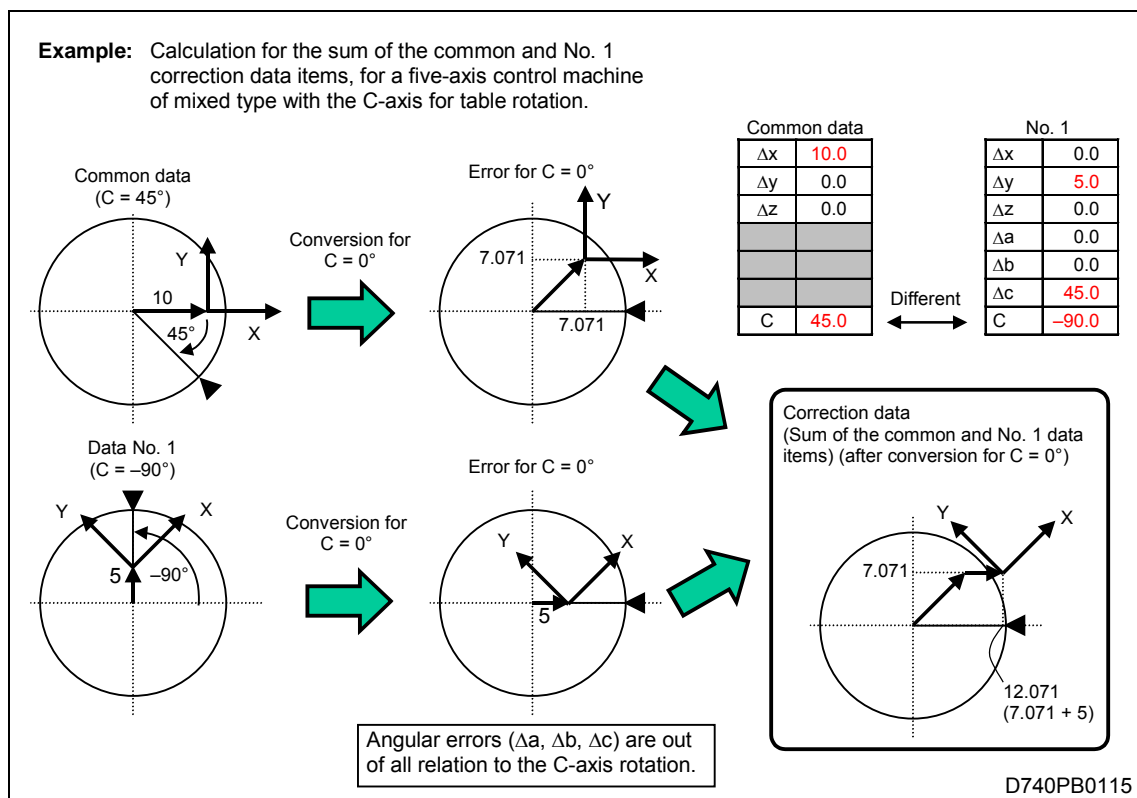
	Common	No. 1	No. 2	No. 3	No. 4	No. 5	No. 6	No. 7
Δx	#5801	#5811	#5821	#5831	#5841	#5851	#5861	#5871
Δy	#5802	#5812	#5822	#5832	#5842	#5852	#5862	#5872
Δz	#5803	#5813	#5823	#5833	#5843	#5853	#5863	#5873
Δa	—	#5814	#5824	#5834	#5844	#5854	#5864	#5874
Δb	—	#5815	#5825	#5835	#5845	#5855	#5865	#5875
Δc	—	#5816	#5826	#5836	#5846	#5856	#5866	#5876
Rotat. axis coord. 1	#5807	#5817	#5827	#5837	#5847	#5857	#5867	#5877
Rotat. axis coord. 2	#5808	#5818	#5828	#5838	#5848	#5858	#5868	#5878

Use #5800 to read the number (1 to 7) of the currently selected data set for workpiece setup error correction.

Note: An attempt to overwrite the system variable #5800 will only lead to an alarm (1821 UNWRITABLE SYSTEM VARIABLE).

C. Sum of correction data items

The sum of the common and particular correction data items for orthogonal axes is calculated as appropriate after conversion for one and the same position on the rotational axis concerned.



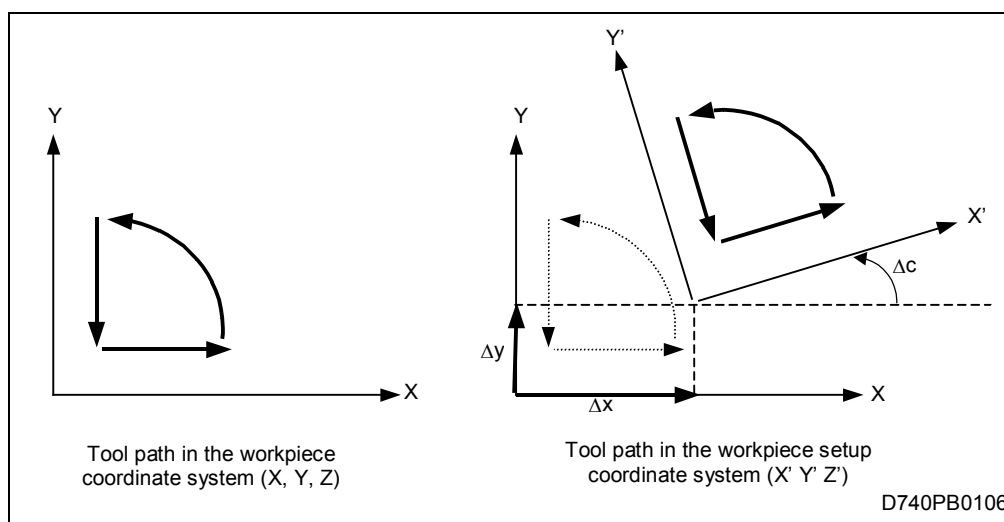
4. Operation description

A. Operation under the modal condition of workpiece setup error correction

A block of G54.4 Pn ($n = 1$ to 7) selects the function of workpiece setup error correction, and causes a workpiece setup coordinate system to be established according to the data set of No. n as well as to the current positions on the rotational axes concerned, with the positional indication (POSITION values) on the display with respect to the currently valid coordinate system being changed accordingly.

Axis motion commands in the G54.4 Pn mode are processed in general with respect to the workpiece setup coordinate system.

A block of G54.4 P0 cancels the function of workpiece setup error correction. The workpiece setup coordinate system will be replaced by the original workpiece coordinate system with reference to which the G54.4 Pn command was given, and the POSITION values on the display will be changed accordingly.



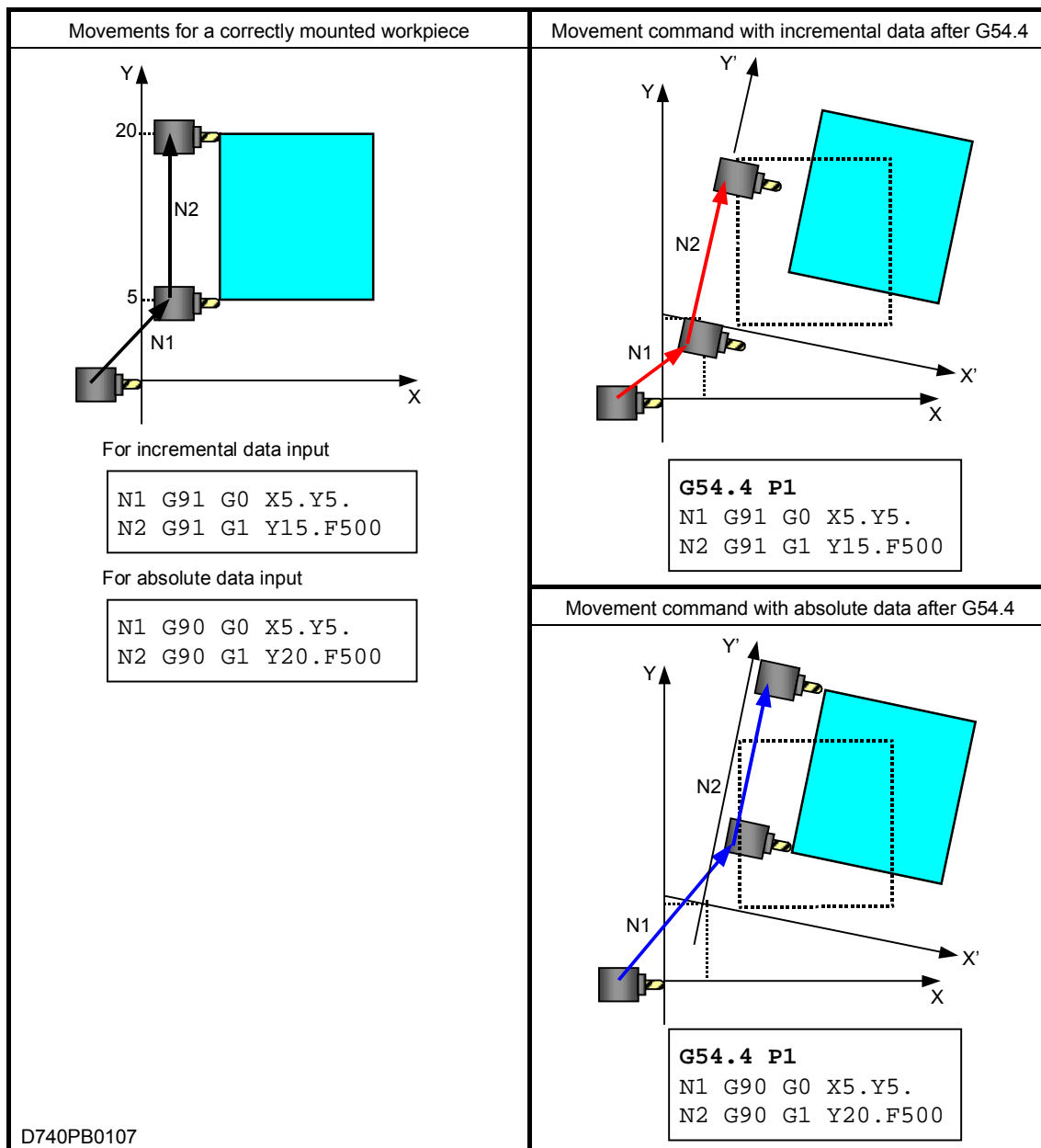
Under the modal condition of workpiece setup error correction (G54.4), the NC unit controls the tool path by including the program-externally provided data of workpiece setup errors.

Note 1: Resetting the NC-unit includes cancellation of the mode of workpiece setup error correction (G54.4).

Note 2: Selection and cancellation of the mode of workpiece setup error correction (with G54.4) causes the POSITION values on the display to be changed as mentioned above, indeed, but no actual movements occur on the machine.

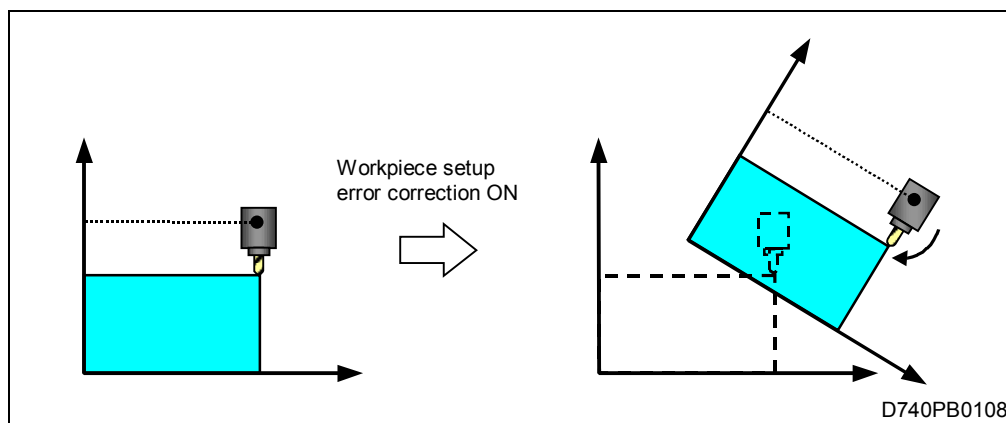
B. Precautions for the selection and cancellation with G54.4

A block of G54.4 does not cause any actual movements on the machine. Use the method of absolute data input, therefore, for the first block of axis movement after the selection of the mode of workpiece setup error correction (with G54.4). Use of incremental data could not move the tool to the expected position. This also applies to the cancellation, and the first motion command after G54.4P0 must be given with absolute data.



C. Correction of the tool-axis direction

In addition to the translating correction in the XYZ space, an angular correction can be achieved on five-axis control machines, according to the inclination on the rotational axis, in order to set the tool-axis direction (tool attitude) normal to the surface to be machined.



The correction of tool attitude in question can be done in general by using either of the two possible pairs of angles on the rotational axes concerned.

Example: On machines of mixed type (with the B- and C-axis for rotating the tool axis and the table, respectively)

Pairs of angular positions for tool attitude correction			
With a positive B-axis angle		With a negative B-axis angle	
<p>B = 30° C = 0°</p>		<p>B = -30° C = 180°</p>	
POSITION	MACHINE	POSITION	MACHINE
B 0.	B 30.	B 0.	B -30.
C 0.	C 0.	C 0.	C 180.

The tool attitude correction conducted can be examined in the positional indication under MACHINE (given with respect to the machine coordinate system), while the POSITION values denote the programmed position.

D740PB0116

The selection between the two pairs is done as follows.

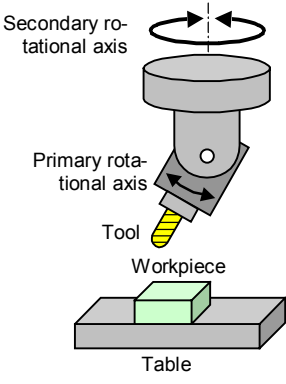
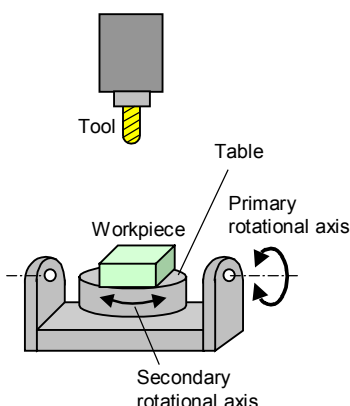
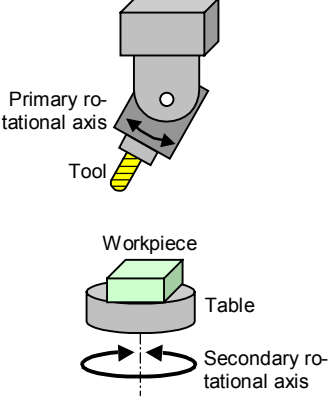
- For the selection block with G54.4
Selected is the pair for a method which requires smaller distance of angular motion on the primary rotational axis.

2. For all the other blocks

The selection is done according to the setting of parameter **F162** bit 1 (Type of passage through singular point for tool tip point control).

	Type 1	Type 2
Parameter	F162 bit 1 = 1	F162 bit 1 = 0
Operation	<p>Selected is the pair for a method which results in the angle of the primary rotational axis obtaining the same sign as its initial position (position at the startup by a G54.4 command).</p> <p>* When the initial position on the primary rotational axis is equal to 0, then decision is made for the pair whose angle of the primary rotational axis has the same sign as the wider side of its axis stroke.</p>	<p>Selected is the pair for a method which requires smaller distance of angular motion on the secondary rotational axis.</p>

Remark: Definition of primary and secondary rotational axes

Tool rotating type	Table rotating type	Mixed type
 <p>Secondary rotational axis</p> <p>Primary rotational axis</p> <p>Tool</p> <p>Workpiece</p> <p>Table</p>	 <p>Tool</p> <p>Table</p> <p>Workpiece</p> <p>Primary rotational axis</p> <p>Secondary rotational axis</p>	 <p>Primary rotational axis</p> <p>Tool</p> <p>Workpiece</p> <p>Table</p> <p>Secondary rotational axis</p> <p>D740PB0034</p>

26-4-3 Relationship to other functions

1. Relationship to other commands

A. Commands available under condition of workpiece setup error correction

Function	Code	Function	Code
Positioning	G00	Geometry compensation	G61.1
Linear interpolation	G01	Cutting mode	G64
Circular interpolation	G02/G03 (*3)	User macro call	G65/G66 G66.1/G67
Dwell	G04	3-D coordinate conversion ON	G68 (*2)
High-speed machining mode	G05	Inclined-plane machining	G68.2 G68.3 G68.4
Exact-stop	G09	3-D coordinate conversion OFF	G69
Programmed data setting ON/OFF	G10/G11	Hole-machining fixed cycles	G71.1 - G89
Plane selection	G17/G18/G19	Absolute data input	G90
Return to zero point	G28/G30	Incremental data input	G91
Skip function	G31 (*4)	Feed per minute (asynchronous)	G94
Tool radius compensation OFF	G40	Feed per revolution (synchronous)	G95
Tool radius compensation (to the left)	G41	Constant surface speed control OFF	G97
Tool radius compensation (to the right)	G42	Single program multi-process control	G109
Tool radius compensation for five-axis machining (to the left)	G41.2	M, S, T, B output to opposite system	G112
	G41.4	Drilling and tapping fixed cycles	G283 - G289
	G41.5	Subprogram call/End of subprogram	M98/M99
Tool radius compensation for five-axis machining (to the right)	G42.2	Feed function	F
	G42.4	M, S, T, B function	MSTB (*1)
	G42.5	Local variables, Common variables, Operation commands (arithmetic operations, trigonometric functions, square root, etc), Control commands (IF ~ GOTO ~, WHILE ~ DO ~)	Macro instructions
Tool length offset	G43/G44		
Tool length offset in tool-axis direction	G43.1		
Tool tip point control, type 1/2	G43.4/G43.5		
Tool position offset OFF	G49		
Selection of machine coordinate system	G53		
Tool-axis direction control	G53.1		

*1 Use of a tool function (T-code) under condition of workpiece setup error correction will lead to an alarm (**1814 ILLEGAL CMD IN G54.4 MODE**).

*2 A G68 command can be given under condition of workpiece setup error correction only when parameter **F168** bit 4 = 1 (Replacing the 3-D coordinate conversion command [G68] with inclined-plane machining command [G68.2]).

*3 In combined use with the function for tool tip point control, giving a circular interpolation command will lead to an alarm (**971 CANNOT USE TOOL TIP PT CONTROL**).

*4 Available only for machines of tool rotating type.

Giving any other command than those enumerated above in the mode of workpiece setup error correction will lead to an alarm (**1814 ILLEGAL CMD IN G54.4 MODE**).

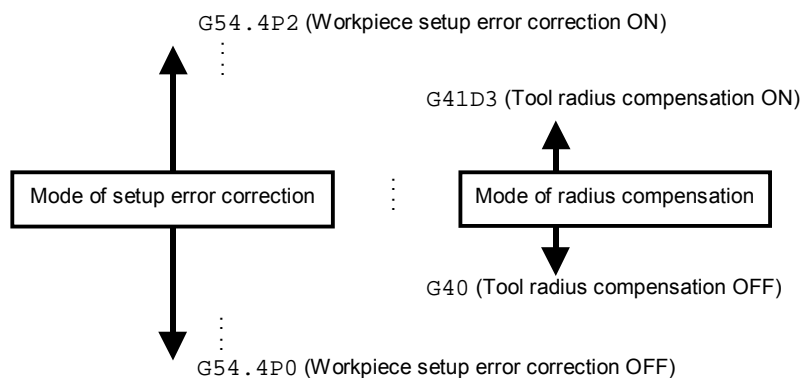
B. Modes in which workpiece setup error correction is selectable

Function	Code	Function	Code
Positioning	G00	Cutting mode	G64
Linear interpolation	G01	Modal user macro call OFF	G67
High-speed machining mode OFF	G05P0	3-D coordinate conversion OFF	G69
Radius data input for X-axis ON	G10.9X0	Fixed cycle OFF	G80
Plane selection	G17/G18/G19	Absolute data input	G90
Inch/Metric data input	G20/G21	Incremental data input	G91
Pre-move stroke check OFF	G23	Inverse time feed	G93
Tool radius compensation OFF	G40	Feed per minute	G94
Shaping OFF	G40.1	Feed per revolution	G95
Tool length offset (+/-)	G43/G44	Constant surface speed control OFF	G97
Tool length offset OFF	G49	Initial point level return in hole-machining fixed cycles	G98
Scaling OFF	G50	R-point level return in hole-machining fixed cycles	G99
Mirror image OFF	G50.1	Single program multi-process control	G109
Polygonal machining mode OFF	G50.2	Cross machining control OFF	G111
Local coordinate system setting	G52	Hob milling mode OFF	G113
Selection of workpiece coordinate system/ additional workpiece coordinate system	G54-59/G54.1	Superposition control OFF	G127
Dynamic offsetting II OFF	G54.2P0		
Geometry compensation	G61.1		

Selecting the function of workpiece setup error correction in a mode other than those enumerated above will lead to an alarm (**1815 CANNOT USE G54.4**).

26-4-4 Restrictions

1. Be sure to enter the G54.4 command (for selection as well as for cancellation) independently. If a block of G54.4 should contain any other commands, an alarm is caused (**1815 CANNOT USE G54.4**).
2. In the mode of workpiece setup error correction, system variables #5001 to #5116 for reading positional information refer to the workpiece setup coordinate system, while system variables #5021 to #5036 always denote the current position with respect to the machine coordinate system.
3. Resetting the NC-unit includes cancellation of the mode of workpiece setup error correction.
4. Tool radius compensation, tool length offset, tool tip point control, mirroring by G-code, scaling, inclined-plane machining, and fixed cycle must be selected and cancelled within the mode of workpiece setup error correction (between G54.4P_ and G54.4P0), as shown in the example below for the use of tool radius compensation.



5. Do not give any tool change command in the mode of workpiece setup error correction; otherwise an alarm will be caused (**1814 ILLEGAL CMD IN G54.4 MODE**).
6. Manual interruption is always performed on the basis of the machine coordinate system (without any coordinate conversion). After manual interruption of workpiece setup error correction (by or without using the TPS function) do not resume automatic operation without returning the machine components concerned to the original positions on the controlled axes; otherwise an alarm will be caused (**184 ILLEGAL OPER IN G54.4 MODE**).
7. During interruption of workpiece setup error correction, an attempt to perform manual movements on a rotational axis, or to use the manual pulse handle will only lead to an alarm (**184 ILLEGAL OPER IN G54.4 MODE**).
8. Tool path check on the **TOOL PATH CHECK** display can only be performed on the basis of the original workpiece coordinate system (without coordinate conversion taken into consideration).
9. In the simulation on the **VIRTUAL MACHINING** display, the setting for workpiece setup error correction in the program is taken into account in representing the movements of machine components, indeed, but not in determining the display position of the material model.
10. Tracing in the mode of workpiece setup error correction is displayed with reference to the machine coordinate system.
11. Do not use corner chamfering or rounding commands in the mode of workpiece setup error correction; otherwise an alarm will be caused (**1814 ILLEGAL CMD IN G54.4 MODE**).
12. Restarting operation from a block under the modal condition of workpiece setup error correction begins with a movement to the accordingly corrected position and to the position without correction, respectively, in the case of using the **[RESTART]** and the **[RESTART 2 NONMODAL]** menu function.
13. Do not specify a MAZATROL program as a subprogram to be called up in the mode of workpiece setup error correction; otherwise an alarm will be caused (**1814 ILLEGAL CMD IN G54.4 MODE**).
14. Do not enter a block of motion command which requires an angular movement through more than 180°; otherwise an alarm will be caused (**1820 ILLEGAL COMMD IN G54.4**).
15. According to the settings in parameter **SU153**, the relevant items of digital information on the **POSITION** display refer to the following types of coordinate system in the mode of workpiece setup error correction:

	Bit in question OFF (0)	Bit in question ON (1)
POSITION (SU153 bit 3)	Workpiece setup coordinate system	
MACHINE	Machine coordinate system	
BUFFER (SU153 bit 1)	Workpiece coordinate system	Workpiece setup coordinate system
REMAIN (SU153 bit 2)	Workpiece coordinate system	Workpiece setup coordinate system

16. Do not give a command for positioning to an arbitrarily definable point, or for external tool measurement (for tool breakage detection) in the mode of workpiece setup error correction; otherwise an alarm will be caused (**1814 ILLEGAL CMD IN G54.4 MODE**).
17. Positioning commands (under G0) in the mode of workpiece setup error correction are always executed in interpolation-type paths.

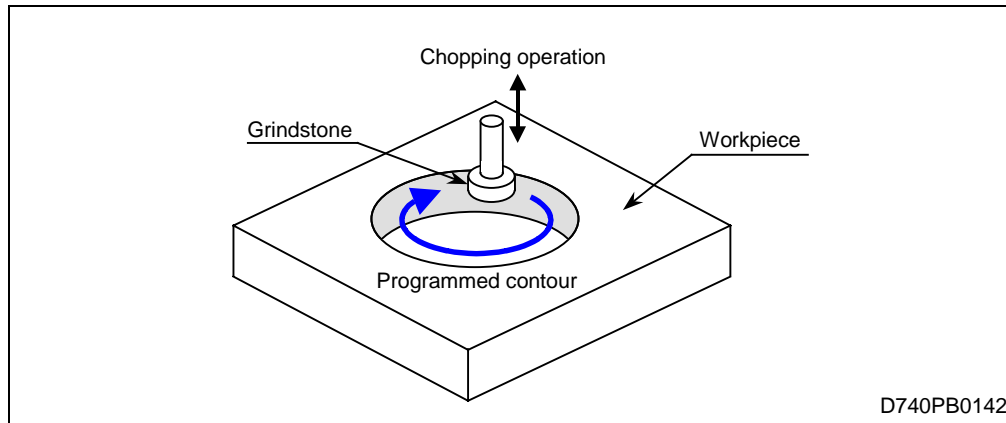
18. The acceleration and deceleration for positioning commands (under G0) in the mode of workpiece setup error correction occurs according to the parameters **L74** (rate of cutting feed for pre-interpolational acceleration/deceleration control) and **L75** (time constant for pre-interpolational cutting feed) when they are given under G61.1 (geometry compensation) or with the (optional) fixed gradient control for G0 being selected.

- NOTE -

27 CHOPPING FUNCTION (OPTION)

1. Function outline

Chopping here refers to independent high-speed reciprocation of a grindstone on an axis (to be called chopping axis) perpendicular to the plane of contour description, for the purpose of grinding the side surface of a workpiece. The axial reciprocation occurs simultaneously with the radial feed along the programmed contour.



Remark 1: This function can be used only in an EIA/ISO program.

Remark 2: The operation mode can be changed in general even during chopping. It is possible to start program execution as well as to perform manual interruption without having to cancel the chopping mode. An attempt, however, to select the zero point return mode, or to apply manual interruption to the chopping axis, will only lead to an alarm (**1133 ILLEGAL OPERATION IN CHOPPING** or **1132 ILLEGAL CHOPPING CONDITION**).

Remark 3: Resetting will immediately cause a return to the reference position on the chopping axis and terminate the chopping function. For more information see the description given later under 3-A.

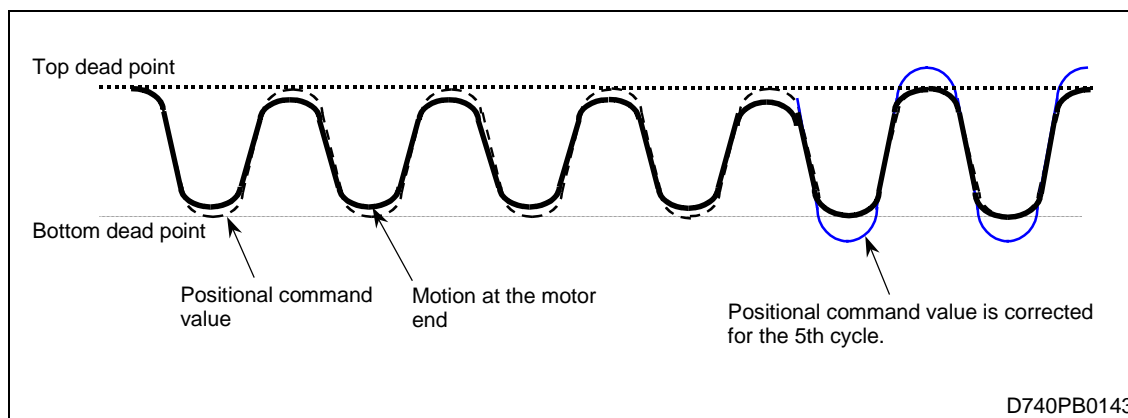
Remark 4: It is possible to change the chopping conditions (reference position, top dead point, bottom dead point, and rate of feed) even in the chopping mode. For more information see the description given later under 3-B.

As the high-speed reciprocation for chopping in general occurs insufficiently within the range between the commanded top and bottom dead points because of a tracking delay of the servo system, among others, there are two compensation methods provided for the command values in question: "updating compensation method" in which the compensation amount is constantly updated with respect to the difference between the command position and the feedback position at the motor end, and "fixed compensation method" in which a fixed amount as obtained through a trial operation is used for the compensation.

A. Updating compensation method

The compensation amount is updated at constant intervals according to the actual operation of the machine.

The compensation amount is calculated once every 4 cycles of chopping operation, and added to the positional command value for the succeeding cycles.

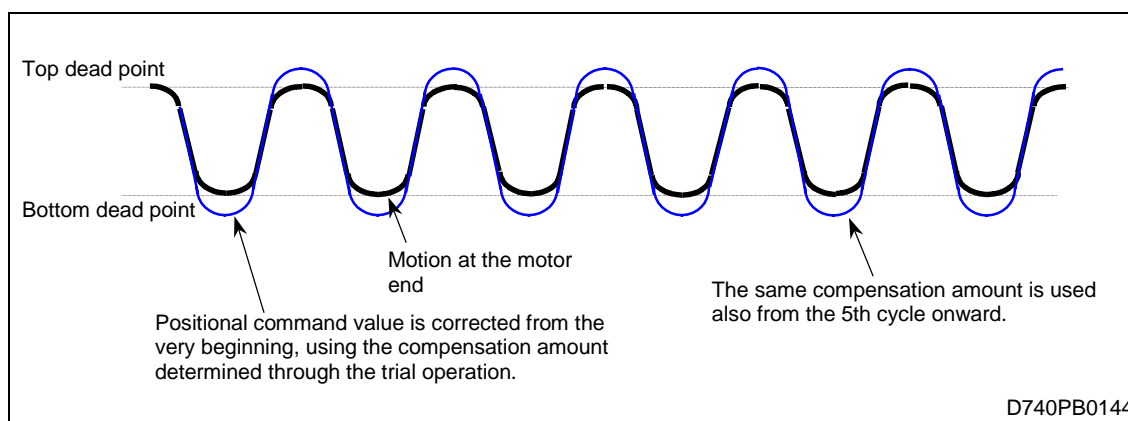


If the error of the feedback position from the command position has been reduced, as a result of the updating compensation, below the allowable level for chopping commands (as set in parameter **SU137**), then the compensation amount will be no more updated.

B. Fixed compensation method

A trial operation is carried out in advance to determine, and record, the particular compensation data to be used for the succeeding chopping operation. The recorded compensation data can be checked on the screen as described later under 6.

In actual machining the chopping operation is carried out, from the first downward movement to the bottom dead point, using the recorded compensation data.



The fixed compensation method consists of two modes of operation: recording mode and playback mode. In recording mode, the chopping axis, reference position, top and bottom dead points, rate of chopping feed and compensation amount are recorded through a trial operation. In playback mode, chopping occurs using the recorded data for the particular chopping operation.

This method uses the compensation amount obtained in recording mode without actualization.

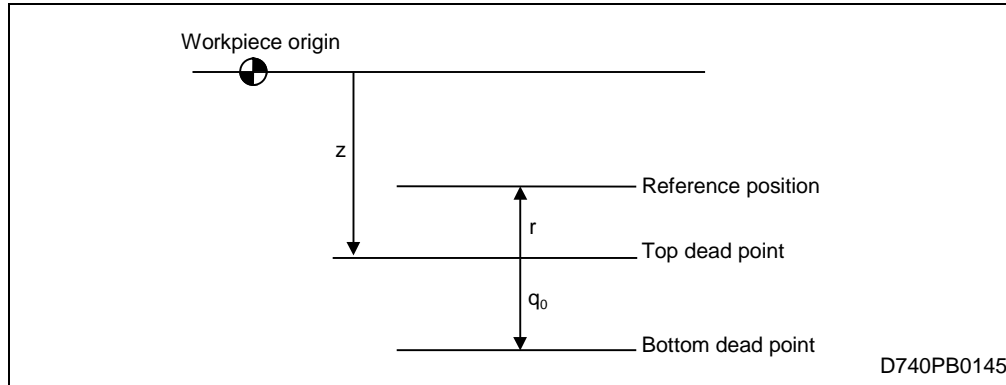
The uniformity in surface finish may be affected in the updating compensation method due to the change in reciprocating width especially through the first compensation. Choose the fixed compensation method, therefore, if accurate reciprocation between the top and bottom dead points is required from the very first cycle of chopping operation.

2. Programming format

There are two types of programming formats available for chopping operation: using either G-codes or M-codes.

A. Programming with G-codes

G81 .1 Zz Qq₀ Rr Ff₀ [Ll₀ Pp₀] Start of chopping
G80 End of chopping



z : Top dead point position (Replace address Z as required to accord with the actual chopping axis.)

q₀ : Distance to the bottom dead point (an incremental value with respect to the top dead point)

r : Distance between top dead point and reference position

f₀ : Rate of chopping feed on the chopping axis

l₀ : Compensation method

0: Updating compensation method

1: Fixed compensation method, Recording mode

2: Fixed compensation method, Playback mode

p₀ : Number (0 to 7) of compensation data, to be used in recording and playback modes of fixed compensation method

- Argument F works independently of, and does not overwrite, the modal feed function (F value) for normal cutting.
- Argument L can be omitted if it is zero (L0).
- If argument L is to be set to zero (L0), omit argument P (ignored even if entered).
- If argument L is to be set to two (L2), omit arguments Z, Q, R, and F (ignored even if entered).
- With argument L set to zero or one (L0 or L1), do not omit arguments Z, Q, R, and F; otherwise an alarm will be caused (**401 ILLEGAL FORMAT**).
- With argument L set to one or two (L1 or L2), do not omit argument P; otherwise an alarm will be caused (**401 ILLEGAL FORMAT**).

Programming example (for the use of Updating compensation method)

⋮	
G81.1 Z-20.Q-30.R10.F3000.	Start of chopping, independent reciprocation on the Z-axis.
G91 G01 X-100.F100.	} Chopping in progress.
Y-100.	
X100.	
Y100.	
⋮	
G91 G00 X10.Y10.	
G80	End of chopping, Z-axis positioning to the reference position.
⋮	

Programming example (for the use of Fixed compensation method)

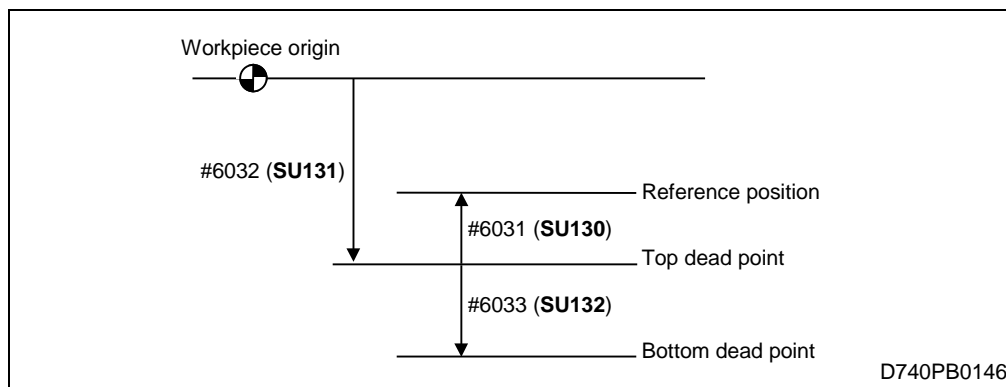
⋮	
G81.1 Z-20.Q-30.R10.F3000.L1 P0	Trial operation for recording compensation data for chopping.
G81.1 L2 P0	Start of chopping in playback mode, using data obtained by the execution of the preceding block.
G91 G01 X-100.F100.	} Chopping in progress.
Y-100.	
X100.	
Y100.	
⋮	
G91 G00 X10.Y10.	
G80	End of chopping, Z-axis positioning to the reference position.
⋮	

B. Programming with M-codes

- M544 Start of chopping
 M545 End of chopping
 M546 Recording of compensation data for chopping

- Use M544 and M545 to start and end the chopping operation, respectively, in updating compensation method or in the playback mode of fixed compensation method.
- Use M546 to execute a trial operation in the recording mode of fixed compensation method.
- System variables are provided to specify the required conditions for chopping – chopping axis, reference position, top and bottom dead points, and rate of chopping feed.
- M544 starts the operation in the playback mode of fixed compensation method even under the condition for recording mode of that method (#6035 = 1).
- M546 executes the operation for the recording mode of fixed compensation method even under the condition for playback mode of that method (#6035 = 2).

Sys. variable	Contents	Setting unit	Parameter
#6030	Chopping axis (1 to 16)		SU26
#6031	Reference position (incremental from top dead point)	0.0001 mm or 0.00001 in	SU130
#6032	Top dead point (workpiece coordinate [absolute])	0.0001 mm or 0.00001 in	SU131
#6033	Bottom dead point (incremental from top dead point)	0.0001 mm or 0.00001 in	SU132
#6034	Rate of chopping feed	0.0001 mm/min or 0.00001 in/min	SU133
#6035	Compensation method (0 to 2) 0: Updating compensation method 1: Fixed compensation method, Recording mode 2: Fixed compensation method, Playback mode		
#6036	Number (0 to 7) of compensation data, to be used for fixed compensation method		



- Note 1:** The M-code for recording the chopping compensation data will not finish till completion of internal recording.
- Note 2:** A chopping start M-code will be executed under the conditions stored in the relevant parameters – chopping axis, reference position, top and bottom dead points, and rate of chopping feed – if the corresponding system variables have not yet been specified after switching on.
- Note 3:** Changing the system variables (for chopping axis, reference position, top and bottom dead points, and rate of chopping feed) has no influence on the parameters concerned.
- Note 4:** The system variables (for chopping axis, reference position, top and bottom dead points, and rate of chopping feed) are initialized to the corresponding parameter settings at the end of the program (M30) or by resetting.

Programming example (for the use of Updating compensation method)

#6030=3	Specifying the chopping axis.
#6031=10.0000	Specifying the reference position using an incremental value from top dead point.
#6032=-20.0000	Specifying the top dead point using an absolute workpiece coordinate.
#6033=-30.0000	Specifying the bottom dead point using an incremental value from top dead point.
#6034=3000.	Specifying the rate of feed on the chopping axis.
#6035=0	Selecting the updating compensation method.
⋮	
M544	Start of chopping in updating compensation method.
G91 G01 X100.	} Chopping in progress.
Y100.	
X-100	
Y-100.	
⋮	
G91 X10.Y10.	
M545	End of chopping, Z-axis positioning to the reference position.
⋮	

Programming example (for the use of Fixed compensation method)

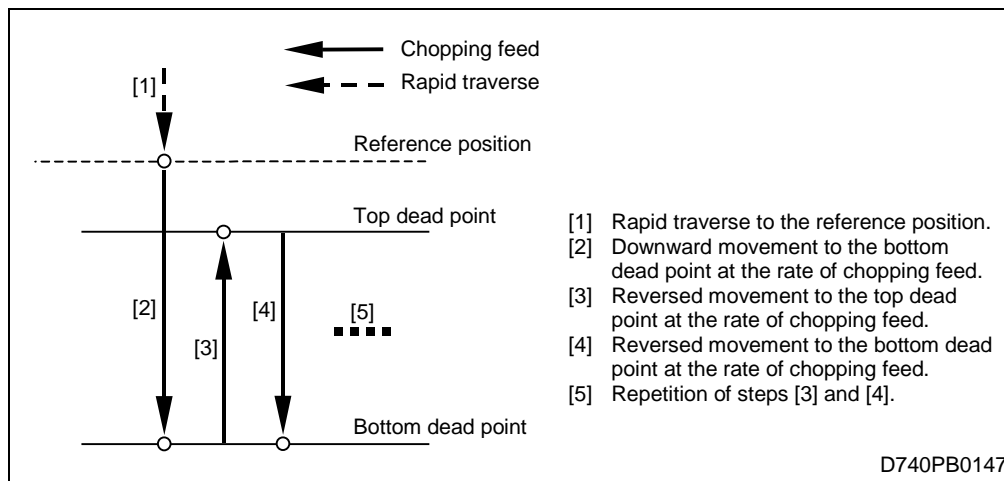
#6030=3	Specifying the chopping axis.
#6031=10.0000	Specifying the reference position using an incremental value from top dead point.
#6032=-20.0000	Specifying the top dead point using an absolute workpiece coordinate.
#6033=-30.0000	Specifying the bottom dead point using an incremental value from top dead point.
#6034=3000.	Specifying the rate of feed on the chopping axis.
#6035=1	Selecting the recording mode of fixed compensation method.
#6036=0	Selecting the number of the compensation data for fixed compensation method.
⋮	
M546	Execution of recording operation. Finished upon completion of internal recording.
M544	Start of chopping in the playback mode of fixed compensation method.
G91 G01 X100.	} Chopping in progress.
Y100.	
X-100	
Y-100.	
⋮	
G91 X10.Y10.	
M545	End of chopping, Z-axis positioning to the reference position.
⋮	

3. Detailed description

A. Start and stop of the chopping operation

Start of the chopping operation

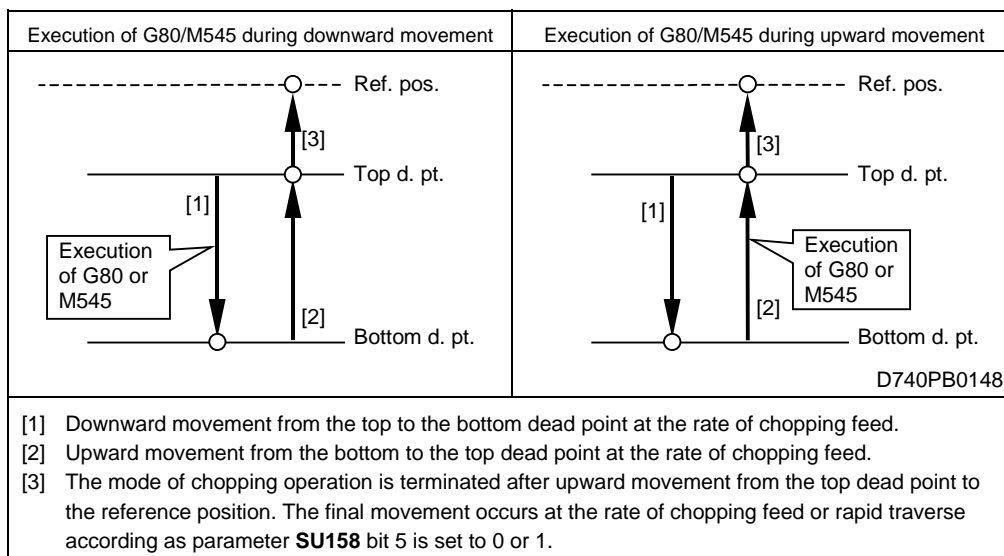
Chopping operation begins with rapid traverse to the reference position, followed by a movement to the bottom dead point at the rate of chopping feed and then proceeds with reciprocation between the top and bottom dead points.



Stop of the chopping operation

Depending on the cause, chopping operation ends, or is brought to a stop, as follows.

- Stop by G80 or M545



- Stop by resetting

Resetting during downward movement	Resetting during upward movement
<p>[1] Immediate stop of the downward movement by resetting.</p> <p>[2] Reversed movement up to the top dead point at the rate of chopping feed.</p> <p>[3] The mode of chopping operation is terminated after upward movement from the top dead point to the reference position. The final movement occurs at the rate of chopping feed or rapid traverse according as parameter SU158 bit 5 is set to 0 or 1.</p>	<p>[1] Downward movement from the top to the bottom dead point at the rate of chopping feed.</p> <p>[2] Upward movement from the bottom to the top dead point at the rate of chopping feed.</p> <p>[3] The mode of chopping operation is terminated after upward movement from the top dead point to the reference position. The final movement occurs at the rate of chopping feed or rapid traverse according as parameter SU158 bit 5 is set to 0 or 1.</p>

D740PB0149

- Stop in the event of an emergency stop

Emergency stop during downward movement	Emergency stop during upward movement
Immediate stop.	Immediate stop.

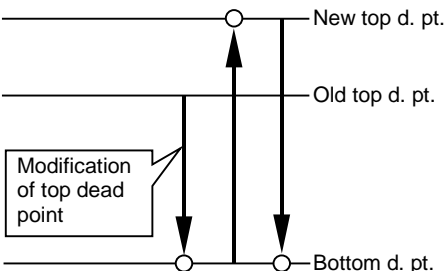
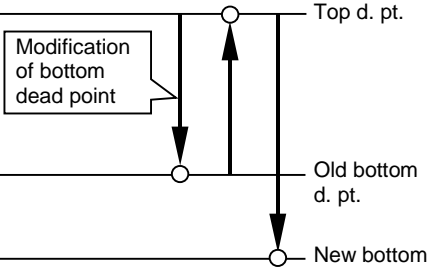
D740PB0150

B. Modifying the top or bottom dead point in the mode of chopping operation

The top or bottom dead point can be modified without having to cancel temporarily the chopping mode. Use the same programming format as for the initial setting to specify the desired position.

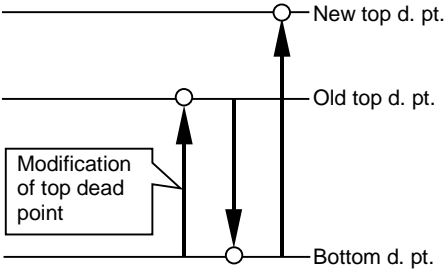
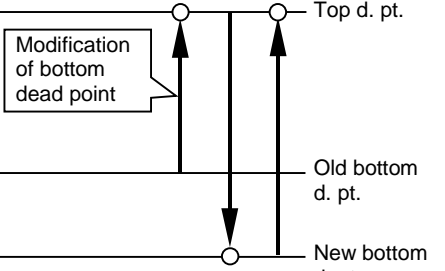
The new position will come into effect for the first movement toward the dead point in question after the modification.

- Modification during downward movement from the top to the bottom dead point

Modification of top dead point	Modification of bottom dead point
 <p>New top d. pt.</p> <p>Old top d. pt.</p> <p>Bottom d. pt.</p> <p>Modification of top dead point</p>	 <p>Top d. pt.</p> <p>Old bottom d. pt.</p> <p>New bottom d. pt.</p> <p>Modification of bottom dead point</p>
After arrival at the bottom dead point, reversed movement to the new top dead point.	After arrival at the old bottom dead point and the top dead point in succession, reversed movement to the new bottom dead point.

D740PB0151

- Modification during upward movement from the bottom to the top dead point

Modification of top dead point	Modification of bottom dead point
 <p>New top d. pt.</p> <p>Old top d. pt.</p> <p>Bottom d. pt.</p> <p>Modification of top dead point</p>	 <p>Top d. pt.</p> <p>Old bottom d. pt.</p> <p>New bottom d. pt.</p> <p>Modification of bottom dead point</p>
After arrival at the old top dead point and the bottom dead point in succession, reversed movement to the new top dead point.	After arrival at the top dead point, reversed movement to the new bottom dead point.

D740PB0152

4. Relationship to other functions

A. Commands available in the mode of chopping

Function	Code	Function	Code
Positioning	G00	Absolute/Incremental data input	G90/G91
Linear interpolation	G01	User macro call	G65/G66/G66.1/G67 (*3)
Circular interpolation	G02/G03 (*1)	Feed function	F
Spiral interpolation	G02.1/G03.1 (*1)	Miscellaneous function	M (*4)
Chopping OFF	G80	Axis movement commands	With X, Y, Z, etc. (*5)
Chopping ON	G81.1 (*2)		

*1 A command for helical interpolation will lead to an alarm (**1832 ILLEGAL COMMAND IN CHOPPING**).

*2 Enter in the same block the arguments as required to specify, or modify, the chopping conditions. Note that an attempt to change the chopping axis in the chopping mode will only lead to an alarm (**1132 ILLEGAL CHOPPING CONDITION**).

*3 An attempt to call up a MAZATROL program will lead to an alarm (**1832 ILLEGAL COMMAND IN CHOPPING**).

*4 Do not use an M-code the execution of which requires an axis movement; otherwise an alarm will be caused (**1132 ILLEGAL CHOPPING CONDITION**).

*5 Movement commands can be given not only for linear axes but also for rotational ones using absolute as well as incremental data. Do not give, however, an explicit movement command for the chopping axis; otherwise an alarm will be caused (**1832 ILLEGAL COMMAND IN CHOPPING**).

Giving any other command than those enumerated above in the mode of chopping will lead to an alarm (**1832 ILLEGAL COMMAND IN CHOPPING**).

B. Modes in which a chopping command can be given

Function	Code	Function	Code
Positioning	G00	Mirror image OFF	G50.1
Linear interpolation	G01	Polygonal machining mode OFF	G50.2
Circular interpolation	G02/G03	Selection of workpiece coordinate systems, or additional workpiece coordinate systems	G54 to G59, or G54.1
Spiral interpolation	G02.1/G03.1		
High-speed machining mode OFF	G05P0	Exact stop mode	G61
Radius data input for X-axis ON	G10.9X0	Cutting mode	G64
Radius data input for X-axis OFF	G10.9X1	User macro call	G65/G66/G66.1/G67
Polar coordinate interpolation OFF	G13.1	3-D coordinate conversion OFF	G69
Polar coordinate input OFF	G15	Fixed cycle OFF	G80
Plane selection	G17/G18/G19	Absolute/Incremental data input	G90/G91
Inch/Metric data input	G20/G21	Inverse time feed	G93
Pre-move stroke check OFF	G23	Feed per minute (asynchronous)	G94
Tool radius compensation OFF	G40	Constant surface speed control OFF	G97
Tool radius compensation	G41/G42	Return to Initial point level/ R-point level	G98/G99
Tool length offset (+/-)	G43/G44	Cross machining control OFF	G111
Tool position offset OFF	G49	Hob milling mode OFF	G113
Scaling OFF	G50		

Giving a chopping command in a mode other than those enumerated above will lead to an alarm (**1831 ILLEGAL CHOPPING CONDITION**).

5. Restrictions

- Reciprocation for chopping can occur only on a linear axis. A rotational axis cannot be selected as the chopping axis.
- Without the chopping function being terminated, an attempt to give a movement command for, or to apply manual interruption to, the chopping axis will only lead to an alarm (**1132 ILLEGAL CHOPPING CONDITION** or **1133 ILLEGAL OPERATION IN CHOPPING**).
- The interference check conducted by the INTELLIGENT SAFETY SHIELD function cannot take into account the motion on the chopping axis in the chopping mode.
- The tool path drawn on the **TOOL PATH CHECK** display cannot include the motion on the chopping axis in the chopping mode. The tool's cutting point remains in a plane which cuts the chopping axis perpendicularly in a point of the position at the start of the chopping mode and, accordingly, the digital indication with respect to the workpiece or the machine origin does not change at all for the position on the chopping axis. The machining time, finally, is estimated with the chopping operation being ignored.
- The motion shown on the **VIRTUAL MACHINING** display cannot include the motion on the chopping axis in the chopping mode. The tool's cutting point remains in a plane which cuts the chopping axis perpendicularly in a point of the position at the start of the chopping mode and, accordingly, the digital indication with respect to the workpiece or the machine origin does not change at all for the position on the chopping axis.
- The **TRACE** display really shows the motion on the chopping axis in the chopping mode visually as well as digitally in the positional indication with respect to the workpiece or the machine origin.
- Do not specify a block within the chopping mode as the restarting position; otherwise an alarm will be caused (**956 RESTART OPERATION NOT ALLOWED**).
- Without the chopping function being terminated, an attempt to execute a MAZATROL program will only lead to an alarm (**1832 ILLEGAL COMMAND IN CHOPPING**).
- Do not enter more than three M-codes in a block of G81.1 command; otherwise an alarm will be caused (**807 ILLEGAL FORMAT**).
- The digital indication of the resultant speed (**FEED**) on the **POSITION** display does not incorporate the motion on the chopping axis.

6. Supplement

The chopping conditions under which the recording occurs can be checked in the **CHOPPING RECORD** window, which can be opened on the **PROGRAM** display for an EIA/ISO program as well as on the **EIA MONITOR** display by selecting the **[Chopping Record]** option under **[Window]** on the menu bar.

Item	Description
P	Number of compensation data
CHOPPING AXIS	Name of the axis on which reciprocation for chopping occurs
REFERENCE	Reference position for chopping Upper row: Programmed value (Lower row: Recorded machine coordinate)
TOP DEAD CNT.	Top dead point for chopping Upper row: Programmed value (Lower row: Recorded machine coordinate)
BOTTOM DEAD CNT.	Bottom dead point for chopping Upper row: Programmed value (Lower row: Recorded machine coordinate)
FEED VEL.	Rate of chopping feed

- NOTE -

28 EIA/ISO PROGRAM DISPLAY

This chapter describes general procedures for and notes on constructing an EIA/ISO program newly, and then editing functions.

28-1 Procedures for Constructing an EIA/ISO Program

- (1) Press the display selector key.
- (2) Press the **[PROGRAM]** menu key.
- The **PROGRAM** display will be selected.


- (3) Press the **[WORK No.]** menu key.
- **[WORK No.]** is highlighted and the **WORK No. SELECT** window appears.


Remark: Refer to the Operating Manual for the **WORK No. SELECT** window.

- (4) Enter the new work number of a program to be constructed.
 - Specifying a work number of a program registered already in NC unit allows the program to be displayed on the screen. Therefore, constructing a new program requires specifying a work number which has not been used. The conditions how work numbers are used are displayed on the **WORK No. SELECT** window.
- (5) Press the **[EIA/ISO PROGRAM]** menu key.
 - Press the **[PROGRAM EDIT]** menu key instead of **[EIA/ISO PROGRAM]** if a work number of the program already registered has been set in Step (4).



- (6) Enter the required programming data.

Set data using alphabetic keys, numeric keys and INPUT key .

 - When INPUT key  is pressed, the cursor is moved to the top of the next line, and then the data of the next block can be entered.
- (7) Press the **[PROGRAM COMPLETE]** menu key to end the editing.

28-2 Editing Function of EIA/ISO PROGRAM Display

28-2-1 Outline

Establishing a constructing mode on the **PROGRAM (EIA/ISO)** display allows the following menu to be displayed as an initial one.

PROGRAM COMPLETE	SEARCH	COPY	ALTER	ERASE	MOVE	FIND & REPLACE	CHANGE PROGRAM	MACRO INPUT	MACRO VARIABLE
	[1]	[2]	[3]	[4]	[5]	[6]			

Terms [1] to [6] represent functions related to the program editing. Use of the functions permits the following operations:

- Inserting and altering data at any position
Data can be inserted and altered at any position on the display.
- Erasing the data
Data displayed on the display can be erased.
- Searching for the data
Data can be searched in the following four ways.
 - 1) Searching for the top line of the program
 - 2) Searching for the bottom line of the program
 - 3) Searching for any required line of the program
 - 4) Searching for any character string
- Copying the data
Other EIA/ISO programs registered in the NC unit can be copied into the selected program, or any data character string in the selected program can be copied into a given position of the selecting program or a new EIA/ISO program.
- Moving the data
Any data character string can be moved to a given position of the selecting program or a new EIA/ISO program.
- Replacing the data
Any data character string can be replaced by another character string.

28-2-2 Operation procedure

The procedure for each operation is described below.

(Given that EIA/ISO program, in which several lines of data are already provided, is selected, and editing mode is established, and also that ALTER menu item is not highlighted in the operations 3 and onward.)

1. Inserting the data

- (1) Press the **[ALTER]** menu key as required to obtain the display status ALTER.
 - When **ALTER** is displayed, press the menu key to cancel the highlighting.
- (2) Move the cursor to the position where data must be inserted.
 - The cursor can be moved to any direction (vertically and horizontally).
- (3) Enter the required data.
 - ➔ Data is inserted in sequence into the position where the cursor is placed.
 - ➔ Data previously set behind the cursor position are moved behind the inserted data.

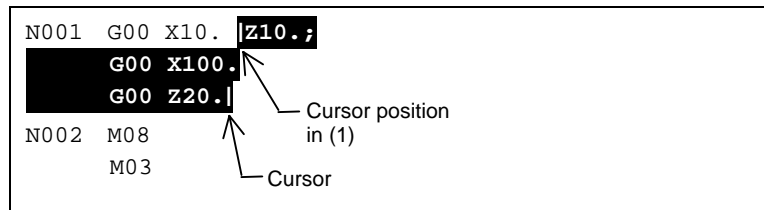
2. Altering the data

- (1) Press **[ALTER]** menu key to display **ALTER**.
 - When ALTER is displayed, press the menu key to highlight the menu item.
- (2) Move the cursor to the position where data must be altered.
 - The cursor can be moved to any direction (vertically and horizontally).
- (3) Enter the required data.
 - ➔ Data is altered in sequence from the position where the cursor is placed.
 - ➔ The character previously set at the cursor position is replaced in sequence by the new data.

3. Erasing the data

- (1) Move the cursor to the head of the character string to be erased.
- (2) Press the **[ERASE]** menu key.
 - ➔ The character at the cursor position is highlighted and the **[ERASE]** menu item is also highlighted.
- (3) Move the cursor to the position next to the end of the character string to be erased.
 - ➔ The portion from the head of the character string specified in (1) to the position before the cursor is highlighted, which indicates that the highlighted portion provides the object of erasure.

Example:



The diagram shows a CNC program display with the following text:

```

N001 G00 X10. Z10.;
      G00 X100.
      G00 Z20.
N002 M08
      M03
  
```

The line `G00 X100.` is highlighted. A cursor is positioned at the end of the line `G00 Z20.`. An arrow points to the cursor with the label "Cursor position in (1)". Another arrow points to the cursor with the label "Cursor".

- (4) Press the input key.
 - ➔ The character string highlighted in (3) is erased.

Example: (Continued)



The diagram shows the result of erasing the highlighted line. The text is now:

```

N001 G00 X10.
N002 M08
      M03
  
```

4. Searching for the data

A. Searching for the top line of the program

- (1) Press the **[SEARCH]** menu key.
- (2) Press the **[PROG HEAD]** menu key.
 - ➔ The cursor moves to the top line.

B. Searching for the bottom line of the program

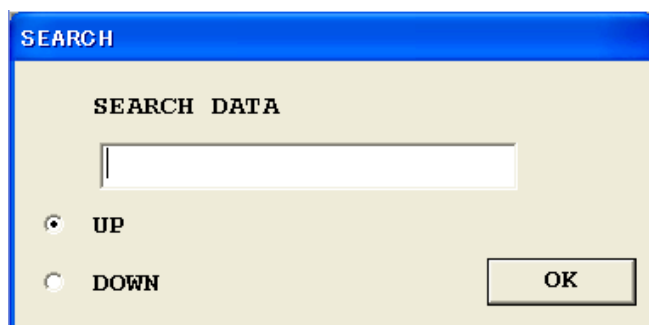
- (1) Press the **[SEARCH]** menu key.
- (2) Press the **[PROG END]** menu key.
 - ➔ The cursor moves to the bottom line.

C. Searching for any required line of the program

- (1) Press the **[SEARCH]** menu key.
- (2) Press the **[SEARCH LINE No.]** menu key.
→ **[SEARCH LINE No.]** is highlighted.
- (3) Set the line No. of the line to be searched for.
- Enter the line No. with numeric keys, and press the input key.
→ The cursor moves to the specified line.

D. Searching for any character string

- (1) Press the **[SEARCH]** menu key.
- (2) Press the **[SEARCH FORWARD]** menu key or **[SEARCH BACKWARD]** menu key.
→ **[SEARCH FORWARD]** or **[SEARCH BACKWARD]** is highlighted.



To search for a character string in the area before the cursor position, press the **[SEARCH FORWARD]** menu key, and for the area after the cursor position, press **[SEARCH BACKWARD]** menu key.

- (3) Set the character string to be searched for and press the input key.
→ The cursor moves to the head of the character string which has been found first.
Press the data cancellation key (CANCEL) to stop halfway the searching operation, whose running state is indicated by the message **CNC BUSY** on the display.

Remark: Pressing the input key in sequence allows the cursor to move to the character string which has been found next.

5. Copying the data

A. Copying a program

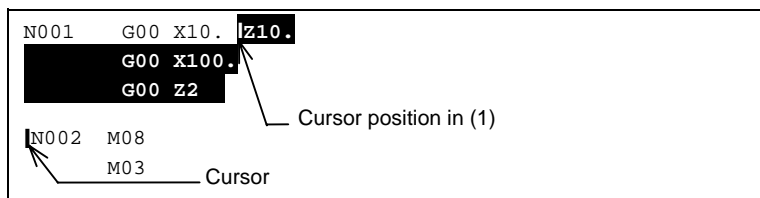
- (1) Move the cursor to the position where the program is to be copied.
- (2) Press the **[COPY]** menu key.
- (3) Press the **[PROGRAM COPY]** menu key.
→ **[PROGRAM COPY]** is highlighted and the **WORK No. SELECT** window appears.
- (4) Set the work number of the program to be copied and press the input key.
→ The program is inserted into the cursor position.
Note: MAZATROL programs cannot be copied.

B. Copying any character string into the selected program

- (1) Move the cursor to the head of the character string to be copied.
- (2) Press the **[COPY]** menu key.

- (3) Press the **[LINE(S) COPY]** menu key.
- ➔ The character at the cursor position is highlighted and the **[LINE(S) COPY]** menu item is also highlighted.
- (4) Move the cursor to the position next to the end of the character string to be copied.
- ➔ The portion from the head of the character string specified in (1) to the position before the cursor is highlighted, which indicates that the highlighted portion provides the object of copying.

Example:



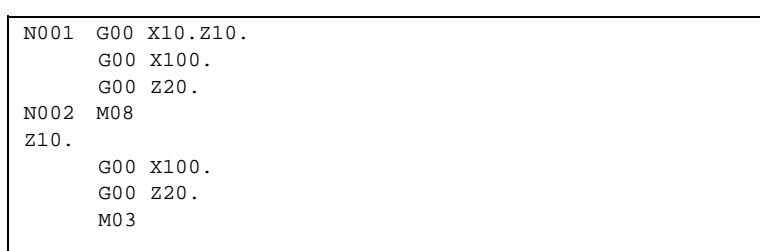
- (5) Press the input key.
- ➔ The area being highlighted is established as the object to be copied.
- (6) Move the cursor to the position where the character string is to be copied.
- ➔ The cursor only moves, and the area highlighted does not change.

Example:



- (7) Press the input key.
- ➔ The character string highlighted is copied at the cursor position.

Example:



C. Copying any character string into a new program

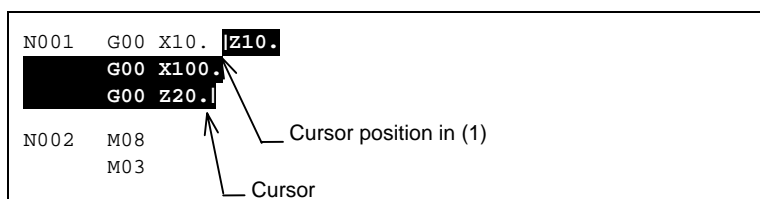
- (6) First, carry out Steps (1) to (5) of **B**. Set the workpiece number of a new program where the character string is to be copied and press the input key.
- ➔ The character string is copied in the new program, and the area highlighted is returned to normal display.

Remark: Pressing the **[PROGRAM FILE]** menu key allows the **WORK No. SELECT** window to be displayed.

6. Moving the data

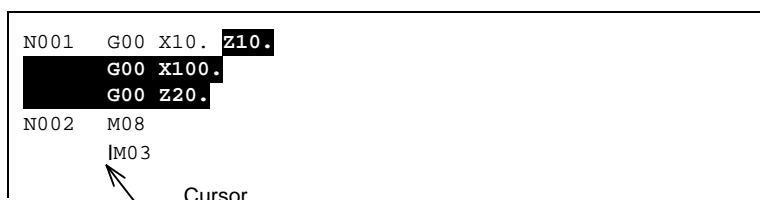
A. Moving the selected program to any position

- (1) Move the cursor to the head of the character string to be moved.
- (2) Press the **[MOVE]** menu key.
 - ➔ The character at the cursor position is highlighted and the **[MOVE]** menu item is also highlighted.
- (3) Move the cursor to the position next to the end of the character string to be moved.
 - ➔ The portion from the head of the character string specified in (1) to the position before the cursor is highlighted, which indicates that the highlighted portion provides the object of moving.



- (4) Press the input key.
 - ➔ The area being highlighted is established as the object to be moved.
- (5) Move the cursor to the position where the character string is to be moved.
 - ➔ The cursor only moves, and the area highlighted does not change.

Example: (Continued)



- (6) Press the input key.
 - ➔ The character string highlighted is moved to the cursor position.

Example: (Continued)



B. Movement to a new program

- (5) First, carry out Steps (1) to (4) of **A**. Set the work number of a new program where the character string is to be moved and press the input key.
 - ➔ The character string is moved to the new program.

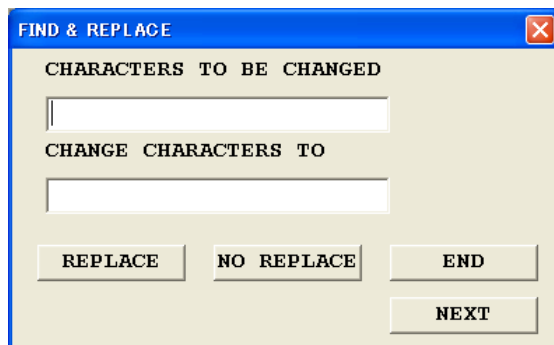
Remark: Pressing the **[PROGRAM FILE]** menu key allows the **WORK No. SELECT** window to be displayed.

7. Replacing the data

- (1) Move the cursor to the starting position of data replacement.
 - Replacement is made downward from the cursor position. To make replacement throughout the program, therefore, move the cursor to the first character of the top line.

- (2) Press the **[FIND & REPLACE]** menu key.

➔ **[FIND & REPLACE]** is highlighted.



- (3) Set the character string before replacement.

Enter the character string to be replaced using alphanumeric keys, and press the tab key



- (4) Set the new character string after replacement using alphanumeric keys, and press the input key.

➔ The cursor moves to the head of the character string before replacement that has been found first after the cursor position specified in (1).

- (5) Press the **[REPLACE]** menu key.

➔ The character string before replacement at the cursor position is replaced by the character string after replacement, and the cursor moves to the head of the next character string before replacement. Pressing the **[REPLACE]** menu key in sequence allows the character string before replacement to be replaced in order of being found.

When replacing the special character string at the cursor position is not required, press the **[NO REPLACE]** menu key in place of **[REPLACE]** menu key.

Remark 1: To stop the replacement, press the **[END]** menu key.

Remark 2: To replace all the character strings in the program, press the **[NEXT]** menu key.

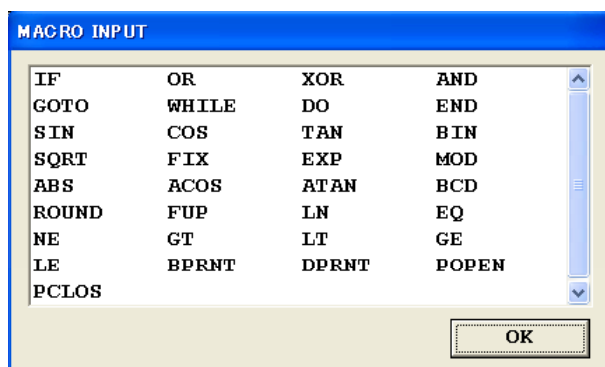
Remark 3: Press the data cancellation key (CANCEL) to stop halfway the total replacement by the NEXT menu function, whose running state is indicated by the message **CNC BUSY** on the display.

28-3 Macro-Instruction Input

This function permits entering the macro-instruction word by word for editing the EIA/ISO program efficiently.

- (1) Press the **[MACRO INPUT]** menu key.

➔ The **MACRO INPUT** window appears.



- The character string selected with the cursor is usable.

- (2) Move the cursor to the characters corresponding to the required macro-instruction and press the input key.

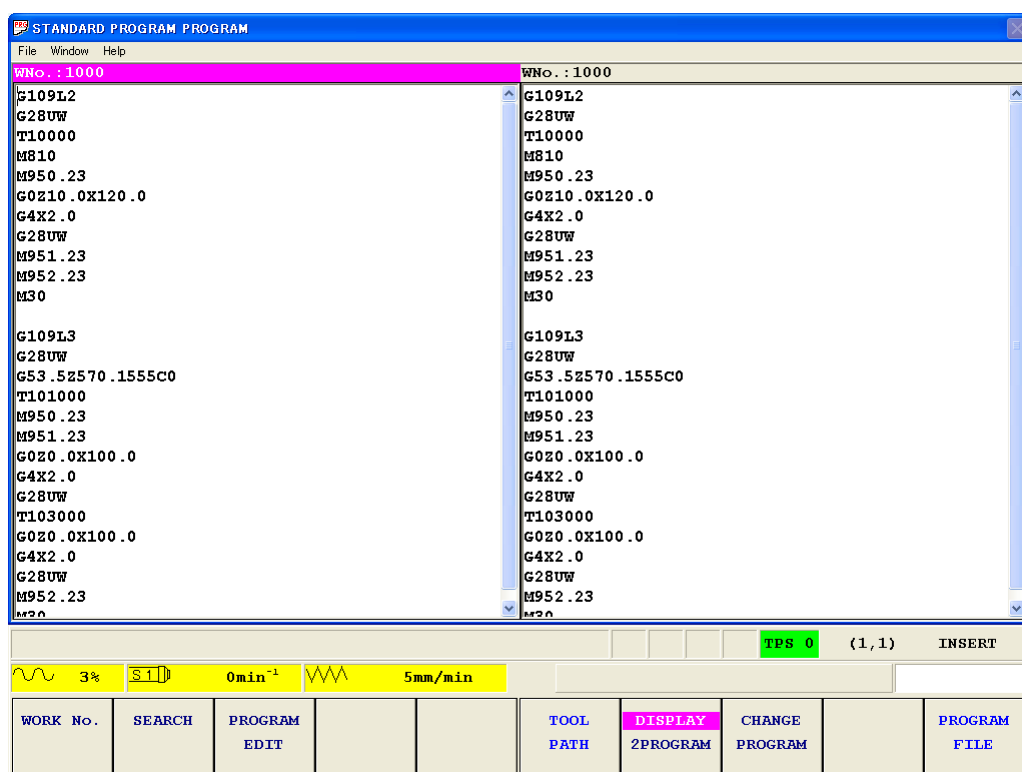
➔ The macro-instruction is entered in the editing zone of the program.

- (3) Press the menu selector key to display the menu for normal data input, and continue program editing.

28-4 Division of Display (Split Screen)

1. Dividing the screen (vertically)

- (1) Temporarily cancel the editing mode, if selected, by pressing the **[PROGRAM COMPLETE]** menu key.
- (2) Press the **[DISPLAY 2 PROGRAM]** menu key.
 - ➔ The display of the menu item is highlighted and the **WORK No. SELECT** window appears.
- (3) Select the work number of the program to be displayed.
 - ➔ The screen is divided into the left and right part. One and the same section of the program is initially displayed in both parts.



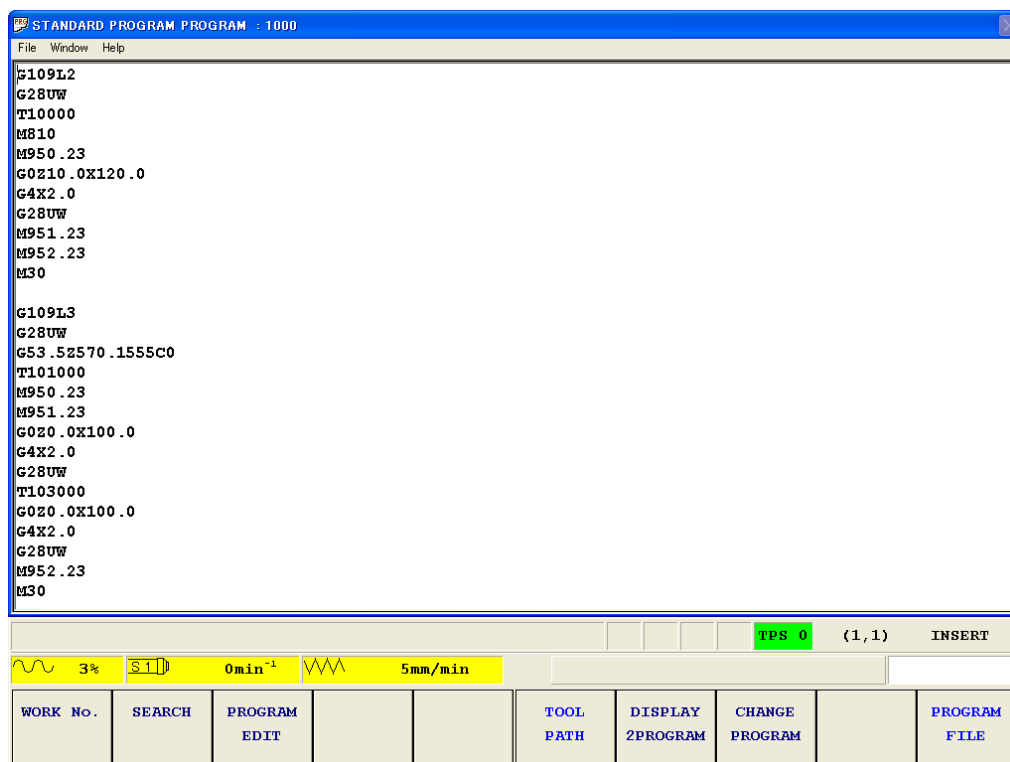
D740PB002'E

- The editing operation can only be carried out in the part the title (WNo.) of which is highlighted.
- The display contents in the other part will remain unchanged even after the editing in the active part. Press the **[CHANGE PROGRAM]** menu key to change the display in the other part according to the editing operation.

2. Cancelling the division

- (1) Temporarily cancel the editing mode, if selected, by pressing the **[PROGRAM COMPLETE]** menu key.
- (2) Press anew the **[DISPLAY 2 PROGRAM]** menu key.

➔ The highlighting of the menu item is cancelled together with the division of the screen.



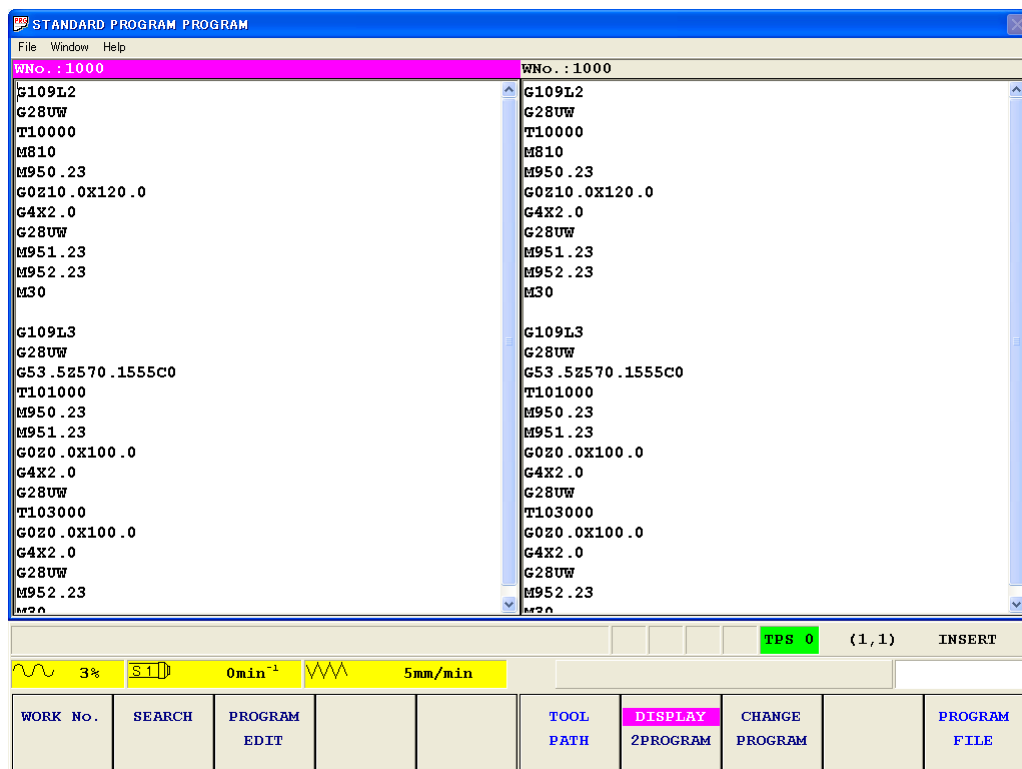
D740PB003'E

3. Changing the active part

The editing is only possible for the part whose title (WNo.) is currently highlighted. The method to change the active part is indicated below.

The data after the editing will not be displayed in the other part (of the same WNo.) unless this changing operation is carried out.

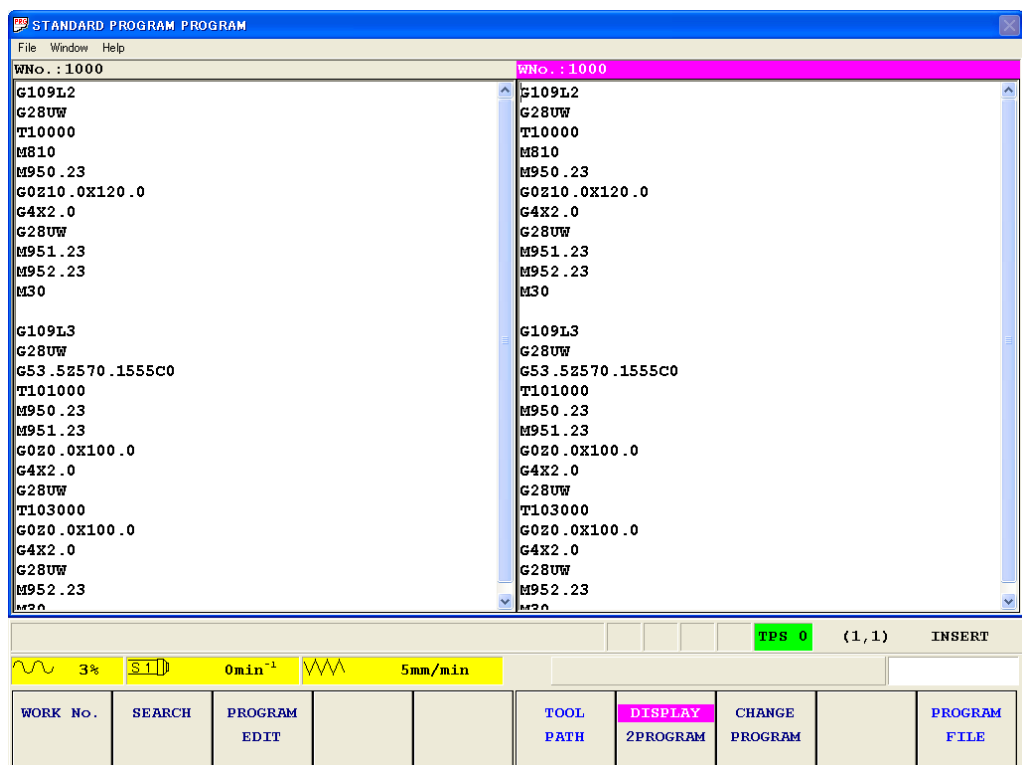
In the example below, the left-hand part is currently active.



D740PB002'E

(1) Press the **[CHANGE PROGRAM]** menu key.

- ➔ The highlighting of the title is transferred from the left-hand to the right-hand part to indicate that the latter has been made active.
- The contents in the right-hand part will have been modified at the same time according to the editing operation performed for the left-hand part (of the same WNo.).



D740PB004'E

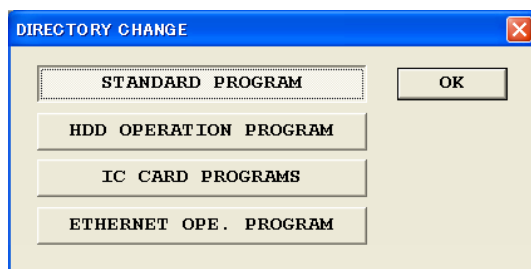
28-5 Editing Programs Stored in External Memory Areas

Follow the procedure below to edit machining programs (to be used for Hard Disk, IC Memory Card, and Ethernet operation) which are created in the EIA/ISO format and stored in external memory areas. The functions for IC Memory Card and Ethernet operation, however, are optional.

- (1) Select [**DIR. CHANGE**] from the initial menu of the **PROGRAM** display for EIA/ISO programs.

WORK No.	SEARCH	PROGRAM EDIT			TOOL PATH	DISPLAY 2PROGRAM	CHANGE PROGRAM	DIR. CHANGE	PROGRAM FILE
----------	--------	-----------------	--	--	--------------	---------------------	-------------------	----------------	-----------------

- ➔ The menu item is highlighted and the **DIRECTORY CHANGE** window appears on the screen.



- The options **IC CARD PROGRAMS** and **ETHERNET OPE. PROGRAM** will only be presented for machines equipped with the corresponding optional functions.
- (2) Use the cursor keys to select the desired storage area.
 - (3) Click the [**OK**] button, or press the INPUT key.
 - ➔ With a memory area other than that of **STANDARD PROGRAM** being selected, the color of the background of the **PROGRAM** display changes to yellow. Follow the same creating and editing procedure, however, as for programs in the **STANDARD PROGRAM** area to prepare a new program, or edit an existing one, for the selected memory area.
 - The area selection made from this window will be maintained till turning off the NC power.
 - The title bar displays the current selection of the memory area.